KING FAHD UNIVERSITY OF PETROLEUM AND MINERALS
DEPARTMENT OF ELECTRICAL ENGINEERING
Electronic Circuits II – EE303

Experiment # 3
Linear Applications of Operational Amplifier

OBJECTIVE

To investigate the different linear applications of the operational amplifier, for example inverting multiplier, inverting summer, inverting integrator, inverting differentiator and differential amplifier.

PRELAB WORK

Students must perform the following calculations and PSPICE before coming to the lab.

1. For the different configurations shown in Figure 1, perform an approximate hand calculation assuming that the operational amplifier is ideal. In each case sketch the expected output waveform.

2. Using SPICE simulate the different configurations and submit the output waveforms for each case. For the SPICE simulation there are two ways for simulating the operational amplifier. In the first, the op-amp is simulated using the simplified model of Figure 2 which consists of an input resistance and a voltage controlled voltage source. This simplified model for the op-amp is fairly good at this stage. At this point it may be useful to introduce to you the concept of SUBCIRCUIT. In many circuits we may use more than one op-amp. In this case we have to replace each op-amp by its equivalent circuit. This may require a long input file if we have a large number of op-amps in our circuit especially if the op-amp is modeled using more sophisticated models than the one shown in Figure 2. One way to avoid this is to use the concept of SUBCIRCUIT in which the model of the op-amp is written only once and then recalled whenever necessary. The format of the first line of a subcircuit definition is

    .SUBCKT SUBNAME N1 N2 N3 …….
    …………..
    …………..
    .ENDS

    where SUBNAME is the name given to the SUBCIRCUIT and N1, N2, N3, ... are the nodes to which the SUBCIRCUIT will be connected. The .ENDS control line should not be confused with the last control line of the entire input file, .END. In a SPICE input file with three SUBCIRCUITS, there would be three .ENDS control lines (one for each SUBCIRCUIT) and only one .END control line.

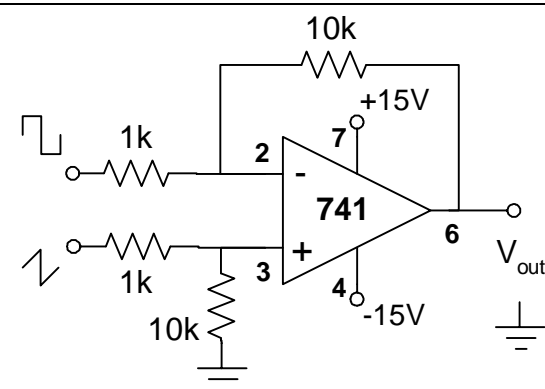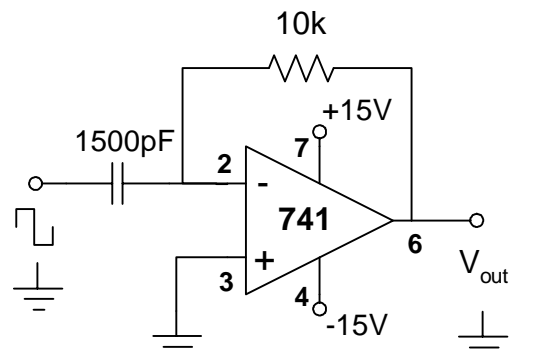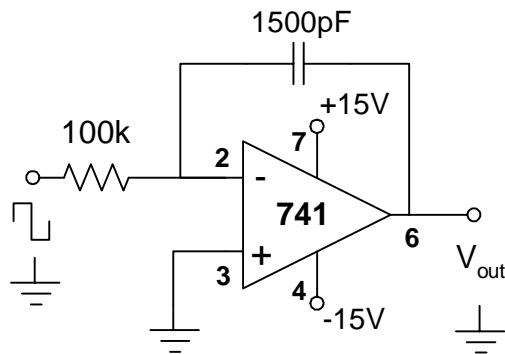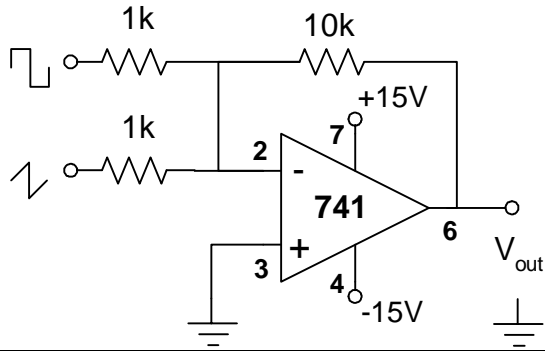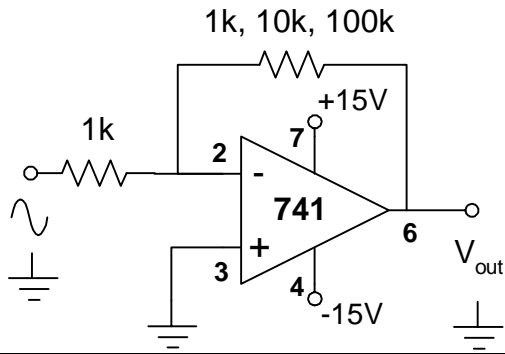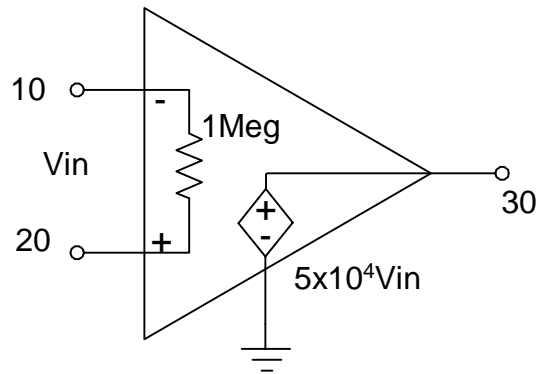| CIRCUIT	Hand Calculation	SPICE Simulation	Experimental Result
1k, 10k, 100k (inverting amplifier with 1k input, ±15V supply, 741 op-amp, 2=−, 3=+, 6=Vout, 7=+15V, 4=−15V)			
1k, 10k (summing amplifier, two 1k inputs, ±15V, 741)			
1500pF, 100k (integrator, ±15V, 741)			
10k, 1500pF (differentiator, ±15V, 741)			
10k, 1k, 1k, 10k (difference amplifier, ±15V, 741)			

Figure 1

Figure 2

Example

```
.SUBCKT OPAMP 20 10 30
RIN 20 10 1MEG
EOUT 30 0 20 10 5E4
.ENDS
```

For the Spice simulations take the input voltages of the order of 1V amplitude and 1kHz frequency. Also you may connect a resistance of 50Ω at the output.
A SUBCIRCUIT may be called as follows:

X……. NA NB NC … SUBNAME

NA, NB, NC, … corresponds to N1, N2, N3, … but are not necessarily the same. For example a call for the subcircuit of the above example can be:

XOP1 2 3 6 OPAMP

where 2 is the positive input of our op-amp named OP1, 3 is negative input and 6 is its output.

You must have your SPICE output file with your hand calculations ready before you come to the lab.

EXPERIMENTAL WORK

1. Construct the circuits shown in Figure 1. Apply the appropriate voltage in each case with amplitudes of the order of 1V and frequencies of the order of 1kHz. In each case measure the output waveform by the oscilloscope and sketch it in the table of Figure 1.

2. Compare your hand calculations, SPICE simulations and experimental results.

3. Comment on your results.