ENGR 065 Circuit Theory

Laboratory 7

The Operational Amplifier

Objectives

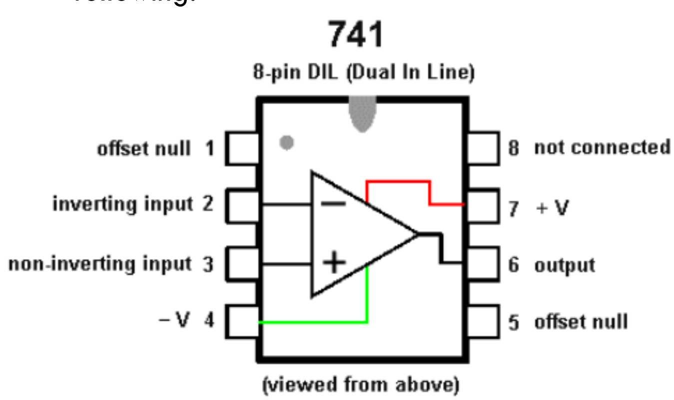
- To learn how to use an operational amplifier (op-amp) construct three different amplifier circuits: an inverting amplifier, a non-inverting amplifier, and a difference amplifier.
- To learn how the circuit structures and component values affect the output of the circuits.
- To learn how to simulate different amplifier circuit with PSPICE

Equipment and components

- A desk computer
- PSPICE software

Preliminary

- The op-amp used in this lab is $\mu A\ 741$, which is one of the cheapest and most popular op-amp. It has open-loop gain and wide operating voltages. The pinout the 741 series is as following.



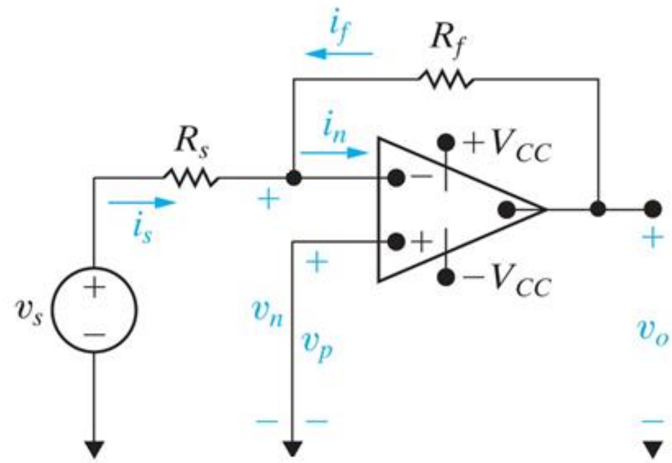
‘

- Carefully read “Lecture 11” to know what the relationship between the input and output of each circuit is and when the circuit is operating in the linear operation range and fill up the tables in this lab instruction before going to lab.
- Review what you have leaned in Lab 8 about how circuits are simulated in PSPICE.

Procedure

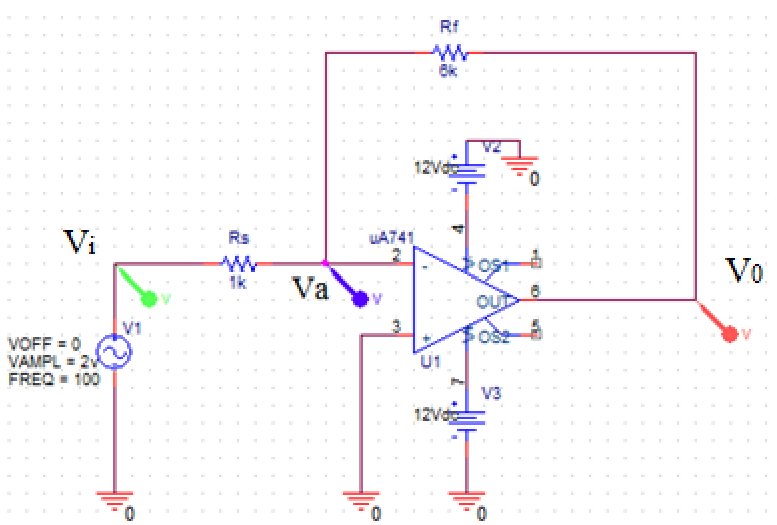
1. The Inverting Amplifier

1. A typical inverting amplifier circuit is shown below.



The input is v_s and the output is v_0. If the op amp is ideal, the output voltage is $v_o = -\frac{R_f}{R_s} v_s$

2. Go to the All Program. Under the "PSpice Student", click the "Capture Student" and open the window "OrCAD Capture". Create a new project "Lab 9 inverting".
3. Click "Place|parts…", add all libraries. Then place " $\mu A\ 741$", and resistors "R_f" and "R_s", "Vdc" and set Vdc =12 V. Place "V_{sin}" and set Voff=0, Vampl=2 V, Freq=100 Hz. Place a couple of "ground 0". Set $R_s = 1\ k\Omega$.
4. Use "wire" to connect all parts. Set voltage measurement points as shown below.
5. Click "New Simulation". Set a name "Lab 9 inverting".
6. Click "Simulation Settings" Under "Analysis Type", select "Time Domain". Under "Options", check "General Settings". Fill in the "Run to time" box with 50ms, "Start saving data after" with 0, and "Maximum step size" with $5\mu s$.
7. Fill up the peak value of each voltage for the different values of R_f.



R_f(kΩ)	0.1	0.5	1	4	6	8	10

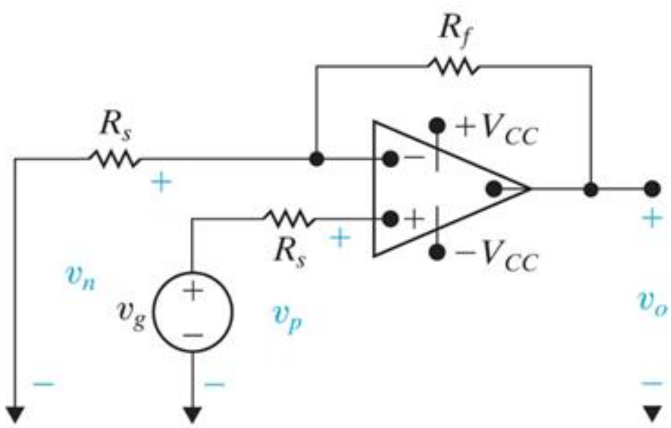
$V_{im}(V)$ (VAMPL)							
V_{am}(V)							
$V_{om}(V)$							
G(Closed-gain)							

What do you observe? Explain them by using what you learned in class. Calculate the currents i_s and i_f based on the voltages in the table above and fill up the table below.

R_f(kΩ)	0.1	0.5	1	4	6	8	10
$i_s(mA)$							
i_f(mA)							

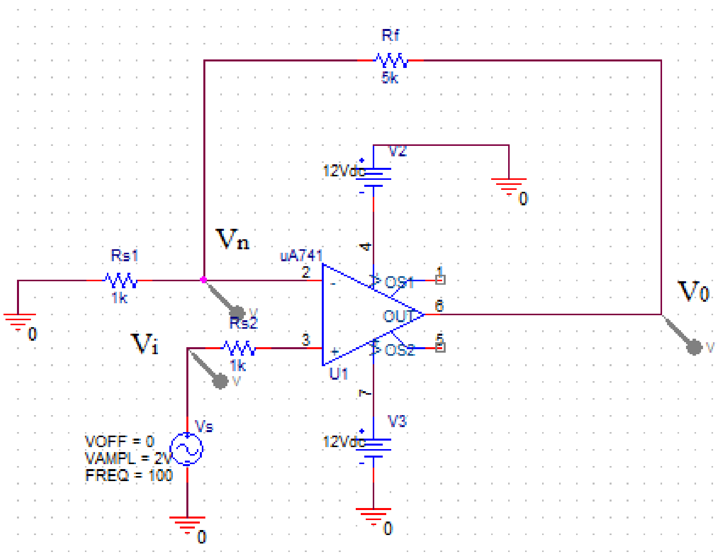
What do you find from the data in the table above?

2. **The Noninverting Amplifier**
8. A typical noninverting amplifier circuit is shown below.



The input is v_s and the output is v_0. If the op amp is ideal, the output voltage is $v_o = (1 + \frac{R_f}{R_s})v_s$

9. Create a new project “Lab 9 noninverting”. Click “Place|parts…”, add all libraries. Then place “ μA 741", right click on the “ μA 741" and then click “Mirror Vertically”. Place resistors "R_f" and "R_s", “Vdc” and set Vdc =12 V. Place “V_{sin}" and set Voff=0, Vampl=2 V, Freq=100 Hz. Place a couple of “ground 0”. Set $R_{s1} = R_{s2} = 1\ k\Omega$.
10. Use “wire” to connect all parts. Set voltage measurement points as shown below.
11. Click “New Simulation”. Set a name “Lab 9 noninverting”.
12. Click “Simulation Settings”. Under “Analysis Type”, select “Time Domain”. Under “Options”, check “General Settings”. Fill in the “Run to time” box with 50ms, “Start saving data after” with 0, and “Maximum step size” with $5\mu s$.
13. Fill up the peak value of each voltage for the different values of R_f.

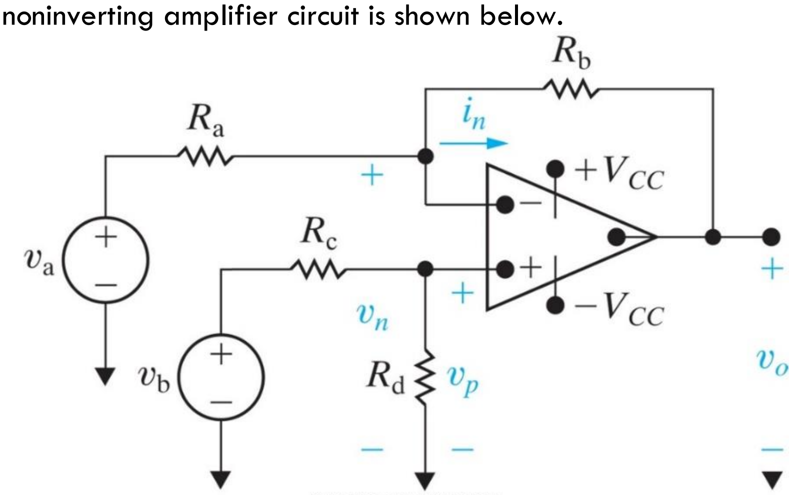


R_f(kΩ)	0.1	0.5	1	4	8	10
$V_{im}(V)$						
$V_{om}(V)$						
V_{nm} (V)						
G(Closed-gain)						

What do you observe? Explain them by using what you learned in class.

3. The Difference Amplifier

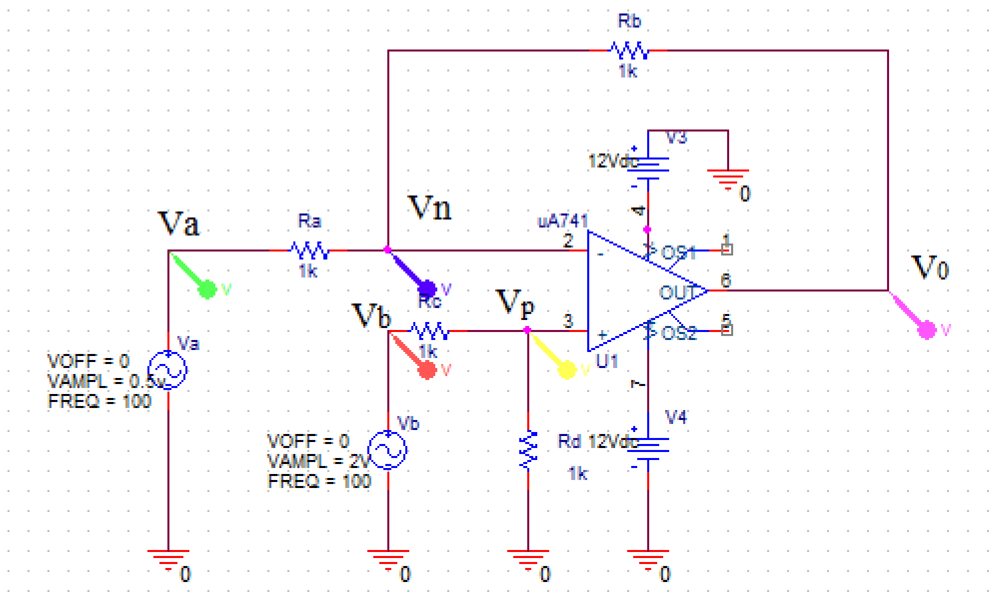
1. A typical noninverting amplifier circuit is shown below.



The inputs are v_a and v_b, the output is v_0. If the op amp is ideal, the output voltage is $v_o = \frac{R_d(R_a+R_b)}{R_a(R_c+R_d)} v_b - \frac{R_b}{R_a} v_a$.

If $\frac{R_a}{R_b} = \frac{R_c}{R_d}$, $v_o = \frac{R_b}{R_a}(v_b - v_a)$, which means the output voltage is proportional to the difference of the two inputs.

2. Create a new project "Lab 9 difference". Click "Place|parts…", add all libraries. Then place " μA 741", right click on the " μA 741" and then click "Mirror Vertically". Place and set resistors "$R_a = 1\ k\Omega$", "$R_b = 1\ k\Omega$", "$R_c = 1\ k\Omega$", and "$R_d = 1\ k\Omega$". Place "Vdc" and set Vdc =12 V. Place "V_{sin}" and set Voff=0, Vampl=2 V, Freq=100 Hz. Place a couple of "ground 0". Set
3. Use "wire" to connect all parts. Set voltage measurement points as shown below.
4. Click "New Simulation". Set a name "Lab 9 difference".
5. Click "Simulation Settings". Under "Analysis Type", select "Time Domain". Under "Options", check "General Settings". Fill in the "Run to time" box with 50ms, "Start saving data after" with 0, and "Maximum step size" with $5\mu s$.
6. Fill up the peak value of each voltage for the different values of V_{bm}.



$V_{am}(V)$	0.5	0.5	0.5	0.5	0.5	0.5
$V_{bm}(V)$	-0.5	0	0.5	1	5	15
$V_{nm}(V)$						
$V_{pm}(V)$						
$V_{om}(V)$						
G(Closed-gain)						

What do you find from the simulation results?

- Summarize your findings of this lab.