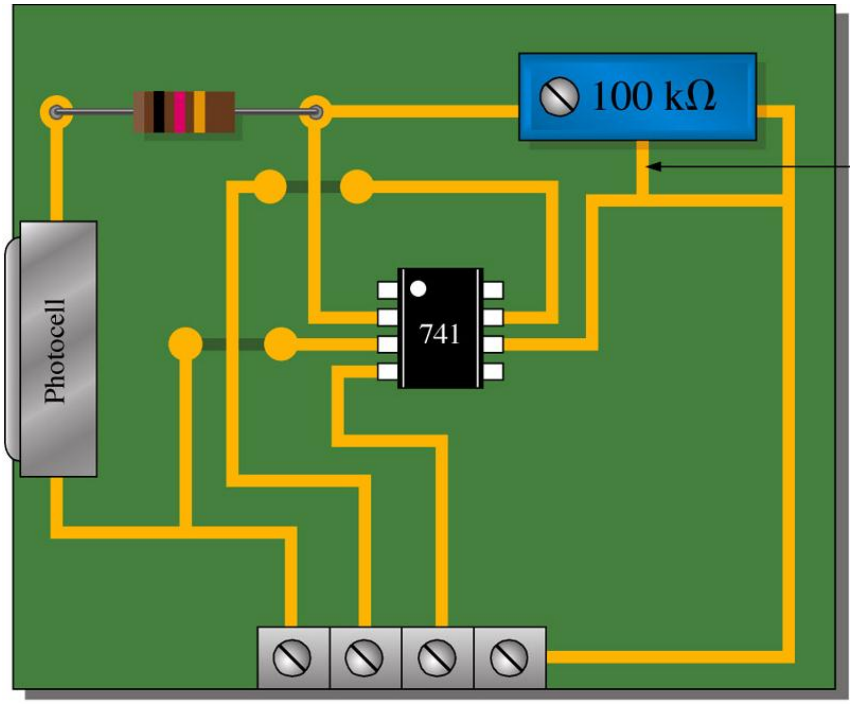
King Fahd University of Petroleum and Minerals

Department of Electrical Engineering

EE 303:
Electronic Circuits II

Laboratory Manual



July 2008

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

Tutorial # 1

Net Listing and Simulation Analysis Using SPICE

OBJECTIVES

1. Describe net listing and simulation analysis using SPICE.
2. Introduce diodes and transistors' models used in SPICE.
3. Explain additional SPICE commands and test new circuits.
4. Test new circuits using default and commercial parameters.

INTRODUCTION

Currently, one of the more widely used general purpose circuit simulation program for industrial and academic computer systems is SPICE. As you know SPICE can be used to simulate circuits containing resistors, capacitors, inductors, mutual inductors, independent and dependent voltage and current sources, and basic semiconductor devices. In EE 203 students were using **Schematic Editor** of SPICE to draw circuits and then run simulation analysis. In fact, drawing circuits and ensuring correct interconnections may be time consuming. Alternatively, circuits can be described in SPICE by specifying their various components and their terminal connections (net listing). A typical SPICE input file format is as follows:

TITLE STATEMENT
CIRCUIT ELEMENTS:
Power Supplies / Signal Sources
Circuit description/Element Descriptions
Model Statement
** Comments*
CONTROL COMMANDS:
Analysis Requests
Output Requests
** Comments*
.END

Notes:

1. The first line must be a title line which is usually reflects the file contents. It cannot be omitted.
2. The last line must be the .END statement.
3. You can insert comment lines starting with "*".

4. You can use upper or lower case letters.
5. The following subsections explain how to describe elements and use the control commands.

CIRCUITS ELEMENTS

1. The general format for describing independent voltage source is

Vname N+ N- [DC value] [AC Magn phase] or [SIN V_0 V_a freq td df phase]

Where
The voltage source must start with letter **V**.
N+ and **N-** are the positive and negative nodes of the source, respectively.
Sources can be assigned values for dc analysis **[DC value]**, ac analysis **[AC magnitude phase]**, or transient analysis **[SIN]**.
The ac phase angle is in degrees.
The parameters of the sin are given in this order: dc offset, amplitude, frequency, delay, damping factor, phase, respectively. In most cases there is no need to specify **td**, **df**, and **phase** but rather leave SPICE use their default values of zeroes.
Other the transient signal generators such as PULSE and PWL are introduced in Experiment 3.

2. An independent current source can be described similarly but using **Iname** to replace **Vname**. But note that current flows from a positive node to the negative node.

3. Various dependent sources are defined when needed in Experiments 3 and 4.

4. Passive elements are described by element statements that specify the type-name and terminal connections as follows:

Rname	**N1**	**N2**	**value**
Cname	**N1**	**N2**	**value**
Lname	**N1**	**N2**	**value**

5. Semiconductor devices such as diodes, MOSFETs and BJTs are described by two statements. In addition to an element statement, a model statement is required. SPICE allows varying degrees of circuit element model complexity. In this tutorial we intend to provide basic default model descriptions and more complex model descriptions. Examples will be used to illustrate the differences between the results obtained using hand calculations, default device models and complex device models. By the end of this tutorial we expect that the student will get an appreciation of the advantages of using SPICE complex models. The following subsections briefly described element and model statements for basic semiconductor devices.

Diode Models

The diode element model is given in Figure 1. The element statement format is given by

Dname NA NC MNAME [AREA]

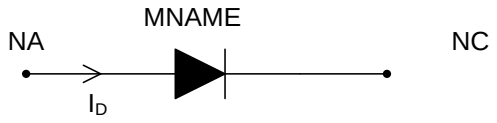


Figure 1: Junction Diode.

The associated model statement is

.MODEL MNAME D [PNAME1=PVAL1 PNAME2=PVAL2…]

The anode of the diode is connected to NA; the cathode to NC. MNAME is an alphanumeric model designation for the device. Detailed model parameters are provided in Table 1.

Bipolar Junction Transistor

The npn and pnp transistor element models are shown in Figure 2. The element statement format is

Qname NC NB NE MNAME [AREA]

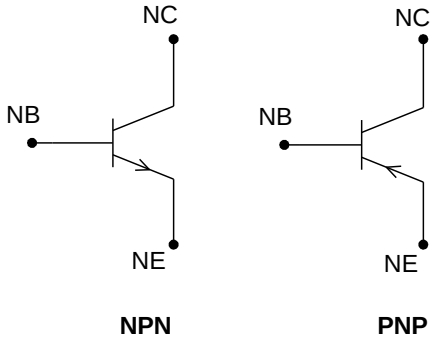


Figure 2: BJT types.

The associated model statements is

.MODEL MNAME npn [PNAME1=PVAL1 PNAME2=PVAL2…]

or

.MODEL MNAME pnp[PNAME1=PVAL1 PNAME2=PVAL2…]

Detailed model parameters are provided in Table 2.

MOS Field Effect Transistor

The n-channel and p-channel MOSFET element models are given in Figure 3. The element statement format is

Mname ND NG NS NB MNAME W=VALW L=VALL

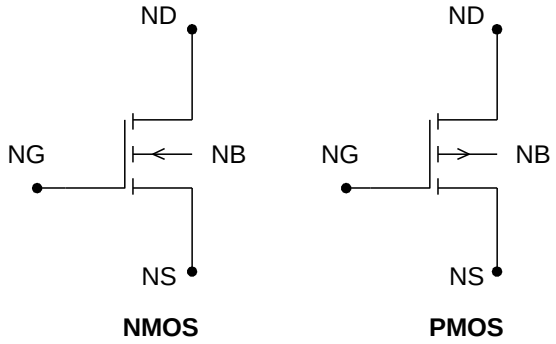


Figure 3: MOSFET types.

The associated model statements is for N-channel

.MODEL MNAME NMOS [PNAME1=PVAL1 PNAME2=PVAL2…]

and for P-channel

.MODEL MNAME PMOS[PNAME1=PVAL1 PNAME2=PVAL2…]

Detailed model parameters are provided in Table 3. Note that the default values for the gate length, VALL, and the gate width, VALW, are 1cm. Obviously, these are not realistic values; however, the model uses the ratio of VALL and VALW rather than the individual values in its calculations.

CONTROL COMMANDS

.OP

The inclusion of the statement .OP makes SPICE perform DC analysis to find the operating point of the circuit.

.AC Sweep-mode NP START STOP

The **.AC** control statement is used to perform ac analysis on a circuit and provide data for frequency response plotting.

Where **Sweep-mode** is one of the keywords that's indicates the frequency variation by decade (DEC), by octave (OCT), or linearly (LIN).

NP is the number of points per sweep-mode;

FSTART is the starting frequency. FSTART cannot be zero.

FSTOP is the final or ending frequency.

.DC SOURCE_NAME START_VAL STOP-VAL INCREMENT_VAL

The **.DC** control statement specifies the values that will be used for dc sweep or dc analysis.

Where **SOURCE_NAME** is the name of an independent voltage or current source**. START_VAL, STOP_VAL** and **INCREMENT_VAL** represent the starting, ending, and increment values of the source, respectively.

.TRAN TSTEP TSTOP
.TRAN makes SPICE perform a time domain transient analysis of the circuit
TSTEP is the time increment used for plotting and/or printing results of the analysis.
TSTOP is the time of the last transient analysis.

.PRINT ANALYSISTYPE OV

The inclusion of the **.PRINT** statement makes SPICE perform a print of a specified output variable resulting from a specified type of analysis.
where **ANALYSISTYPE** is the type of analysis performed from which the output variable OV is obtained.

.PROBE

Probe in SPICE is the graphics postprocessor that calculates and displays results of a simulation. In effect, Probe functions as a "software" oscilloscope, calculator, and spectrum analyzer. Arithmetic operations on output variables are allowed in SPICE Probe. Always include .Probe statement to help in plotting your results.

WRITING AND RUNING THE PROGRAME

- **Create an input file** (source file) **or Circuit description file** for SPICE.

  You can run SPICE by going to **programs→SPICE Student→SPICE AD Student** from the start Menu. Next, we have to create text file that describes our circuit and the simulation protocol. Create a new text file (**File→New→Text File**) with any editor, such as Microsoft editor, Word perfect, Notepad under windows, etc. and immediately save it (**File Save As…)** with the extension .cir (example:circuit1.cir). Now you must open the file (**File→Open**, and change the "Types of Files" to "Circuit Files") before SPICE recognize it as a valid circuit description file.

- **Run the program**

  Once you are in SPICE, pull down the **File** menu at the top of the screen and select *"Open "*. The system prompts you for the name of the file. Type in the file name of the circuit you have created before. As an example: c:\users\Circuit1.cir. Run the simulation (**Simulation→Run**). A window will appear telling you that Spice program is running, or that the simulation has been completed successfully, or that errors were detected. Click on the "OK" button.

- **Look at the output file and print the results**

  The output file always generated by SPICE is the text file that has the file type "OUT". Let's say you submit a data file to SPICE named "CIRCUIT1.CIR", it will create an output file named "CIRCUIT1.OUT". This output file is created even if your run is unsuccessful due to input errors. The cause of failure is reported in the *.OUT file, so this is a good place to start looking when you need to debug your simulation model.

  An easier way of plotting the results is to use (Add about using probe to see results)

  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX
  XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX

GENERAL EXAMPLES

At this stage you are requested to write and run the following three programs, obtain the results from SPICE simulation. Before start writing the program label all nodes with the common node (ground) always has number "0".

1. The input file of the circuit of Figure 4 can be as follows:

```
PWRSUP.CIR
* Sin input with 100V amplitude
*and 50Hz frequency
VAC        10 0 SIN(0 100 50)
D          10 11 MODR
* default model
.MODEL MODR D
RS         11 12 2
CF         12 0 40U
RL         12 0 1K
.TRAN      1M 40M
*Who many cycle will be
plotted?
.PLOT TRAN V(12)
.PROBE V(12)
.END
```

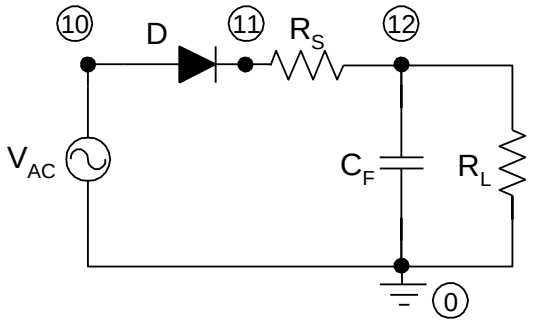


Figure 4: Circuit for example 1.

FREQUNCY RESPONSE EXAMPLES

The main objective of the following two examples is to understand different frequency bands in the amplifier frequency response where a typical frequency response of a capacitively coupled transistor amplifiers is shown in Figure 5. At low frequency band the coupling and bypass capacitors are in effect. Whereas, at high frequency band the transistor internal capacitances are in effect.

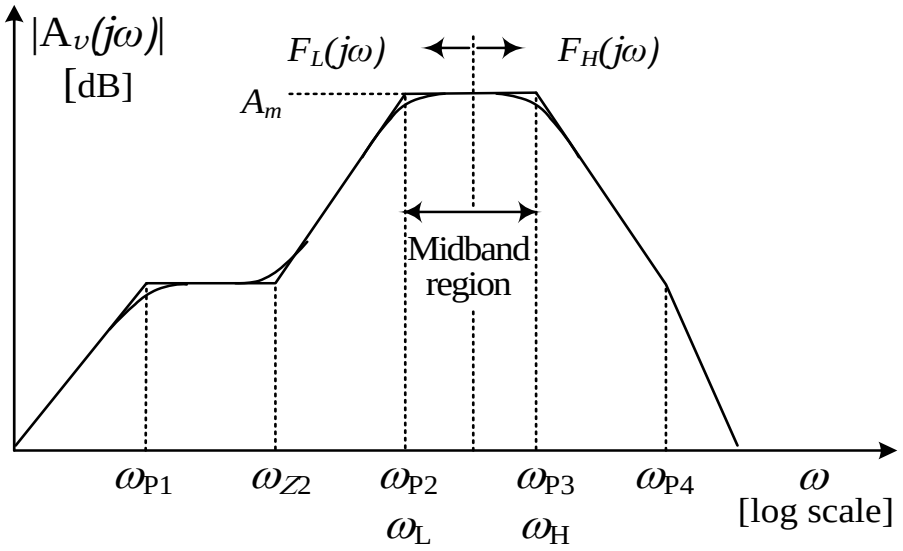


Figure 5: Typical amplifier's frequency response

2. The input file of the circuit of Figure 6 is:

```
MOSFET CS Amplifier
vsig 1 0 ac 1 sin(0 5m 100k)
VDD 6 0 DC 5V
Rsig 1 2 1K
C1 2 3 0.15uF
Mamp 4 3 5 5 M2N4351 W=100U L=100U
.MODEL M2N4351 NMOS (LEVEL=1 +VTO=2.1
KP=1.12M GAMMA=2.6
+ PHI=.75 LAMBDA=2.49M RD=14 RS=14 +
IS=15F PB=.8 MJ=.46
+ CBD=7.95P CBS=9.54P CGSO=11.7N
+ CGDO=9.75N CGBO=16N)
R1 3 0 400K
R2 6 3 100K
RS 5 0 1.3K
Cs 5 0 10uF
RD 6 4 4.3K
C2 4 7 0.15uF
Rl 7 0 100K
.ac dec 100 10 40meg
.print ac v(7)
* Will print the frequency response
.tran 0.01u 20u
.print tran v(7)
*Will print the output wave form at frequency 100kHz
.END
```

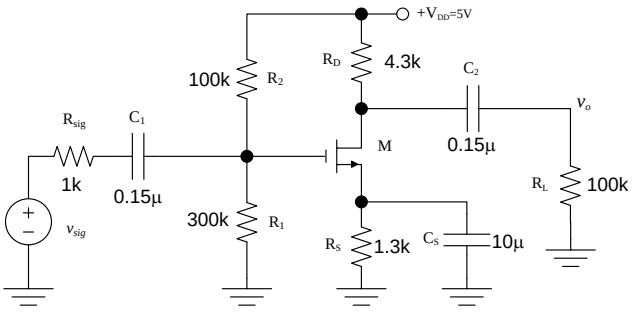


Figure 6: Circuit for example 2.

Requirements:

1. Label the nodes according to the input file.
2. Run the program using default mode first.
3. Find A_M
4. Determine f_H, and f_L where A_M reduces to $0.707A_M$
5. Then, add the given practical model and run the program.
6. Observe the differences.

3. The input file of the circuit of Figure 7 is:

```
BJT CE Amplifier
vsig 1 0 ac 1 sin(0 5m 100k)
VCC 6 0 DC 5V
Rsig 1 2 1K
C1 2 3 1uF
.op
Qamp 4 3 5  Q2N3904
.model Q2N3904 NPN(Is=6.734f +Xti=3 Eg=1.11 +Vaf=74.03 Bf=416.4 Ne=1.259 +Ise=6.734f Ikf=66.78m Xtb=1.5 +Br=.7371 Nc=2 Isc=0 Ikr=0 +Rc=1 Cjc=3.638p Mjc=.3085 +Vjc=.75 Fc=.5 Cje=4.493p +Mje=.2593 Vje=.75 Tr=239.5n +Tf=301.2p Itf=.4 Vtf=4 Xtf=2 +Rb=10)
R1 3 0 400K
R2 6 3 100K
RS 5 0 1.3K
Cs 5 0 10uF
RD 6 4 4.3K
C2 4 7 0.15uF
Rl 7 0 100K
.ac dec 100 10 40meg
.print ac v(7)
.tran 0.01u 20u
.print tran v(7)
.END
```

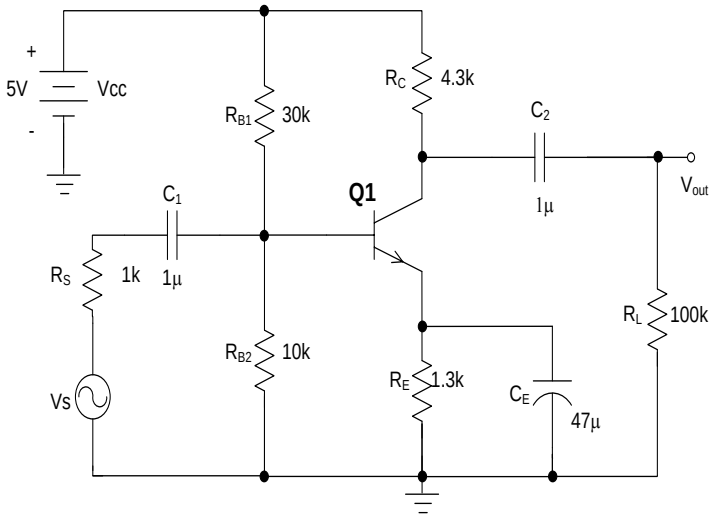


Figure 7: Circuit for example 3.

Requirements:

1. Label the nodes according to the input file.
2. Run the program using default mode first.
3. Find A_M
4. Determine f_H, and f_L where A_M reduces to $0.707A_M$
5. Then, add the given practical model and run the program.
6. Observe the differences.

ASSIGNMENT

Consider the BJT amplifier circuit shown in Figure 8 and perform the following:

1. Using the default parameters of the BJT, write a SPICE program to plot the gain-frequency characteristic. From the SPICE output file, calculate the medium frequency gain and the upper and lower 3dB points.
2. Repeat step 2 using the practical model given below.
3. Comment on your results.

Spice Transistor Model

```
.model Q2N3904        NPN(Is=6.734f Xti=3 Eg=1.11 Vaf=74.03 Bf=416.4 Ne=1.259
+                     Ise=6.734f Ikf=66.78m Xtb=1.5 Br=.7371 Nc=2 Isc=0 Ikr=0 Rc=1
+                     Cjc=3.638p Mjc=.3085 Vjc=.75 Fc=.5 Cje=4.493p Mje=.2593   Vje=.75
+                     Tr=239.5n Tf=301.2p Itf=.4 Vtf=4 Xtf=2 Rb=10)
```

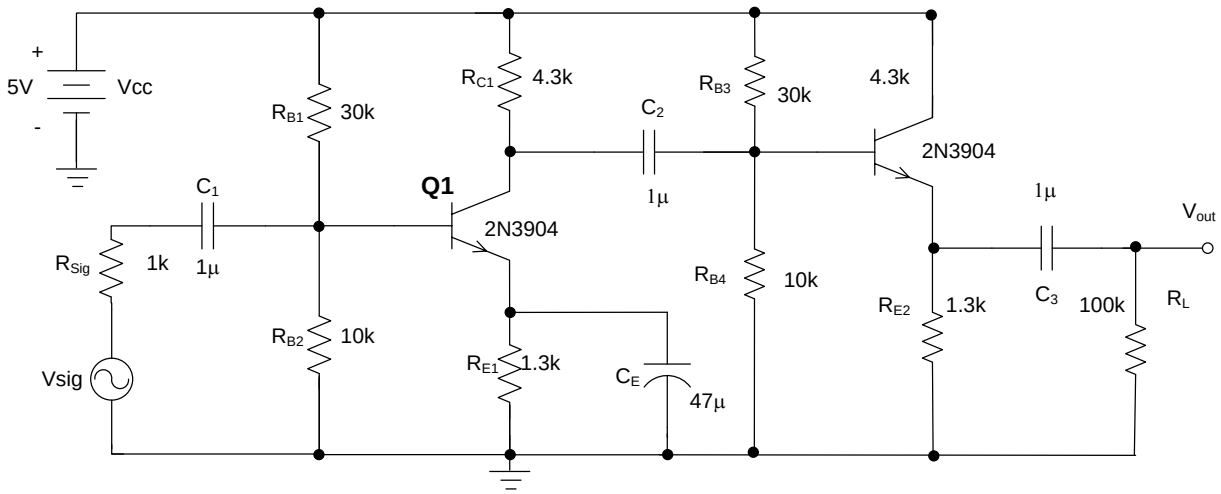


Figure 8: Circuit for assignment.

TABLE 1: DETAILED DIODE MODEL PARAMETERS

Model	Parameters	Default	Units
IS	saturation current	1E-14	A
N	emission coefficient	1	
RS	parasitic resistance	0	ohm
CJO	zero-bias *pn* capacitance	0	farad
VJ	*pn* potential	1	volt
M	*pn* grading coefficient	0.5	
FC	forward-bias depletion capacitance coefficient	0.5	
TT	transit time	0	S
BV	reverse breakdown voltage	infinite	volts
IBV	reverse breakdown current	1E-10	A
EG	bandgap voltage (barrier height)	1.11	eV
XTI	IS temperature exponent	3	
KF	flicker noise coefficient	0	
AF	flicker noise exponent	1	

TABLE 2: DETAILED BJT MODEL PARAMETERS

Model	Parameters	Default	Units
IS	*pn* saturation current	lE-16	A
BF	ideal maximum forward beta	100	
NF	forward current emission coefficient	1	
VAF (VA)	forward Early voltage	infinite	V
IKF (IK)	corner for fwd beta high-cur roll off	infinite	A
ISE (C2)	base-emitter leakage saturation current	0	A
NE	base-emitter leakage emission coefficient	1.5	
BR	ideal maximum reverse beta	1	
NR	reverse current emission coefficient	1	
VAR (VB)	reverse Early voltage	infinite	V
IKR	corner for rev beta hi-cur roll off	infinite	A
ISC (C4)	base-collector leakage saturation current	0	A
NC	base-collector leakage emission coefficient	2.0	
RB	zero-bias (maximum) base resistance	0	ohm
RBM	minimum base resistance	RB	ohm
RE	emitter ohmic resistance	0	ohm
RC	collector ohmic resistance	0	ohm
CJE	base-emitter zero-bias *pn* capacitance	0	F
VJE (PE)	base-emitter built-in potential	0.75	V
MJE (ME)	base-emitter *pn* grading factor	0.33	
CJC	base-collector zero-bias pn capacitance	0	F
VJC (PC)	base-collector built-in potential	0.75	V
MJC (MC)	base-collector *pn* grading factor	0.33	
XCJC	fraction of C_{bc} connected into R_b	1	
CJS (CCS)	collector-substrate zero-bias pn capacitance	0	F
VJS (PS)	collector-substrate built-in potential	0.75	
MJS (MS)	collector-substrate pn grading factor	0	
FC	forward-bias depletion capacitor coefficient	0.5	
TF	ideal forward transit time	0	s
XTF	transit time bias dependence coefficient	0	
VTF	transit time dependency on Vbc	infinite	V
ITF	transit time dependency on Ic	0	A
PTF	excess phase @ 1/ $(2\pi TF)$ Hz	0	°C
TR	ideal reverse transit time	0	s
EG	bandgap voltage (barrier height)	1.11	eV
XTB	forward and reverse beta temp coefficient	0	
XTI(PT)	IS temperature effect exponent	3	
KF	flicker noise coefficient	0	
AF	flicker noise exponent	1	

TABLE 3: DETAILED MOSFET MODEL PARAMETERS

Model	Description	Default	Units
LEVEL	model type(l, 2, or 3)	1	
L	channel length	DEFL	meter
W	channel width	DEFW	meter
LD	lateral diffusion (length)	0	meter
WD	lateral diffusion (width)	0	meter
VTO	zero-bias threshold voltage	0	volt
KP	transconductance	2E-5	A/V^2
GAMMA	bulk threshold parameter	0	$volt^{1/2}$
PHI	surface potential	0.6	volt
LAMBDA	channel-length. modulation (LEVEL 1 or 2)	0	$volt^{-1}$
RD	drain ohmic resistance	0	ohm
RS	source ohmic resistance	0	ohm
RG	gate ohmic resistance	0	ohm
RB	bulk ohmic resistance	0	ohm
RDS	drain-source shunt resistance	infinite	ohms
RSH	drain-source diff. sheet res.	0	ohm/sq.
IS	bulk pn saturation current	1E-14	A
JS	bulk pn sat. current/area	0	A/m^2
PB	bulk pn potential	0.8	volt
CBD	bulk-drain zero-bias pn cap.	0	farad
CBS	bulk-source zero-bias pn cap.	0	farad
CJ	bulk pn zero-bias bot. cap./area	0	F/m^2
CJSW	bulk pn zero-bias perimeter cap./length	0	F/m
MJ	bulk *pn* bottom grading coefficient	0.5	
MJSW	bulk pn sidewall grading coefficient	0.33	
FC	bulk pn forward bias capacitance coefficient	0.5	
CGSO	gate-source overlap capacitance/channel width	0	F/m
CGDO	gate-drain overlap capacitance/channel width	0	F/m
CGBO	gate-bulk overlap capacitance/channel length	0	F/m
NSUB	substrate doping density	0	cm^{-3}
NSS	surface state density	0	cm^{-2}
NFS	fast surface state density	0	cm^{-2}
TOX	oxide thickness	infinite	meter
TPG	gate material type; +1 = opposite of substrate;	+1	
	-1 = same as substrate; 0 = aluminum		
XJ	metallurgical junction depth	0	meter
UO	surface mobility	600	cm^2/Vs
UCRIT	mobility degradation critical field (LEYEL=2)	IE4	V/cm
UEXP	mobility degradation exponent (LEVEL=2)	0	
UTRA	(not used) mobility degradation transverse field coefficient		
VMAX	maximum drift velocity	0	m/s
NEFF	channel charge coefficient (LEVEL=2)	1	
XQC	fraction of channel charge attributed to drain	1	
DELTA	width effect on threshold	0	
THETA	mobility modulation (LEVEL=3)	0	$volt^{-1}$
ETA	static feedback (LEVEL=3)	0	
KAPPA	saturation field factor (LEVEL=3)	0.2	
KF	flicker noise coefficient	0	
AF	flicker noise exponent	1	

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

Experiment # 1

Frequency Response of the Common Source Amplifier

OBJECTIVES

1. To measure the frequency response of common source (CS) amplifier.
2. To determine the useful bandwidth of the CS amplifier by finding the midband frequency gain (A_M), the low and high 3dB corner frequencies.
3. To investigate the effect of load resistance on the frequency response.
4. To study the effect of the bypass capacitor on the frequency response.

BACKGROUND

A typical CS amplifier is shown in Figure 1.

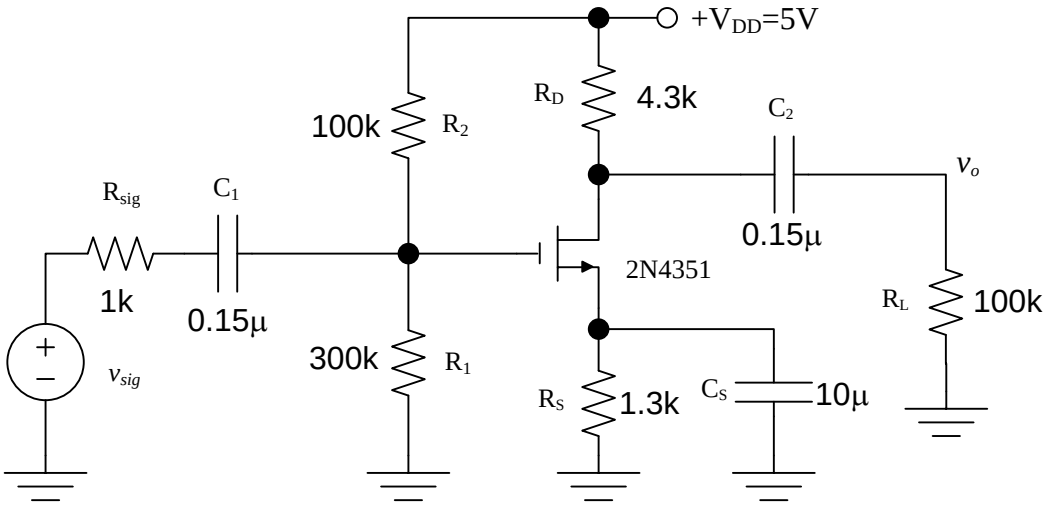


Figure 1: A typical common source amplifier

As you have studied in your lectures small signal ac analysis can be used to shown that:
1. The midband frequency gain (A_M) is given by:

$$A_M = \frac{v_o}{v_{sig}} = -\frac{R_G}{R_{sig}+R_G} g_m R_L' \qquad \textbf{(1)}$$

Where $R_G = R_1 \| R_2$ and $R_L' = R_L \| R_D$

2. The low and high -3dB pole frequencies can be estimated as:

$$\omega_L \cong \sum_{i=1}^{3} \frac{1}{R_{iS} C_i} = \frac{1}{C_1(R_{Sig}+R_G)} + \frac{1}{C_2(R_L+R_D)} + \frac{1}{C_S(R_S \| \frac{1}{g_m})} \qquad \textbf{(2)}$$

$$\omega_H \cong \frac{1}{\sum R_{iS}C_i} = \frac{1}{[C_{GS} + C_{GD}(1 + gmR_L^{'})]R_{sig} // R_G + (C_{GD} + C_{DB})R_D // R_L} \quad \textbf{(3)}$$

3. Also, it can be shown that when Cs is removed the gain will decrease to:

$$\mathrm{A_M} = -\frac{R_g}{R_{sig} + R_g}\frac{g_m R_L^{'}}{1 + g_m R_s} \quad \textbf{(4)}$$

4. The value of ω_L will decrease whereas that of ω_H will increase.

$$\omega_L \cong \frac{1}{C_1(R_S + R_G)} + \frac{1}{C_2(R_L + R_D)} \quad \textbf{(5)}$$

$$\omega_H \cong \frac{1}{C_{GS}(\frac{R_{th} + R_s}{1 + g_m R_s}) + C_{GD}(R_{th} + R_L^{'} + \frac{g_m R_L^{'} R_{th}}{1 + g_m R_s}) + C_{DB}(\frac{R_L^{'} + R_s}{1 + g_m R_s})} \quad \textbf{(6)}$$

In fact, the amplifier bandwidth (BW) is defined as the difference between $f_H = \omega_H/(2\pi)$ and $f_L = \omega_L/(2\pi)$ and since, usually $f_L << f_H$, BW $\approx f_H$. Normally, the amplifier is designed so that its bandwidth coincides with the spectrum of the signals that it is required to amplify. Otherwise, signal distortion will occur.

Finally, a figure-of-merit for the amplifier is its gain-bandwidth product, which is defined as GB=$|A_M|$BW. It will be seen that in amplifier design there is usually trade-off between gain and bandwidth.

EQUIPMENTS & COMPONENTS

1. Signal generator, Bread Board, Digital Multi-Meter, and Digital Oscilloscope.
2. DC supply V_{DD}=5V from the Board
3. MOSFET 2N4351.
4. Resistors: R_{sig} = 1kΩ, R_1 = 300kΩ, R_2 = 100kΩ, R_D = 4.3kΩ, R_S = 1.3kΩ, R_L=100kΩ or 10kΩ.
5. Capacitors: C_1 = 0.15μF, C_2 = 0.15μF, C_S = 10μF.

PRELAB WORK

Students must perform the hand calculations and SPICE before the lab.

Hand Calculation:

1. With C_S, calculate A_M, f_L, f_H, BW, and GB for R_L=100kΩ and R_L=10kΩ assuming g_m=1.04mA/V. Try to deduce the trade-of between gain and bandwidth from these results.
2. From the results obtained in step 1, try to deduce the effect of R_L on the A_M and the bandwidth?
3. Without C_s, calculate A_M, f_L, f_H, and BW when R_L=100kΩ.
4. From the results obtained in step 1 and 3, try to deduce the effect of C_s on the A_M and the BW?
5. Record your results in Table II.

SPICE Simulation:

SPICE simulations can be used to verify the hand calculations. In practice, howevere, stray and bread board capacitances will affect high frequency pole significantly. So, to compare your experimental results for f_H with SPICE you need to add parasitic capacitances in your SPICE file. For example, use three parastic capacitances between each two terminals of the transistors.

6. Use PISCE program developed in the pervious SPICE tutorial to generate three output files for the cases: (a) With C_s and R_L=100kΩ (b) With C_s and R_L=10kΩ (c) Without C_s and R_L=100kΩ. Use no parastic capacitances.
7. Determine A_M, f_L, f_H, BW, and GB for each case.
8. Record your results in Table II.
9. Repeat steps 6 and 7 but using parastic capacitances of about 20pF.
10. Record your results in Table III.

EXPERIMENTAL WORK

See pin configurations of MOSFET 2N4351 in the data sheet given in the Appendix at the end of the manual.

1. Construct the circuit shown in Figure 1 with the capacitor C_S and R_L=100kΩ. Apply a small ac signal v_{sig} with a frequency in the midband (about 10kHz). Keep increasing the amplitude and make sure by monitoring the oscilloscope that the output voltage is not distorted. Using v_{sig} that results in maximum undistorted output (approximately 20mVp-p), in the remaining steps.

   (a) Measure the output amplitude in midband (the output should be constant over wide range of frequencies). Calculate A_M by dividing the amplitude of the output signal by that of the input signal.
   Notes:
   - *i. You may use Table I to record your readings.*
   - *ii. Keep monitoring the input value during measurement since it may vary. Always adjust it to 20mVp-p in order to get correct data.*
   - *iii. Note that at low-frequency the output signal will be noisy, to reduce this effect use the average function on the oscilloscope by pressing the acquire knob, do this for frequencies above 1MHz also.*

   (b) Change the input frequency gradually from 10kHz to about 20Hz. At each frequency measure the small signal voltage gain. Give a special attention to the frequency where gain reduces to $A_M/\sqrt{2}$.

   (c) Now set the frequency back to 10kHz and gradually increase the frequency up to 2MHz and measure the small signal voltage gain at each step. Again give a special attention to the frequency where A_M reduces to $A_M/\sqrt{2}$

2. Plot this set of data on the provided graph sheet (Figure 2).

3. Calculate the A_M, f_L, f_H, BW and GB from your measured gain-frequency characteristics.

4. Change R_L to 10kΩ repeat step 1 through 3.

5. Remove Cs and change R_L back to 100kΩ repeat step 1 through 3.

6. Insert your experimental results into Table III.

7. Compare your hand calculations, SPICE simulations and experimental measurements.

8. Comment on your results.

Table I: Measurement results

With C_s and R_L=100kΩ			With C_s and R_L=1kΩ			Without C_s and R_L=100kΩ		
f	Gain	Gain in dB	f	Gain	Gain in dB	f	Gain	Gain in dB

This page to be submitted with the report

Table II: Summary of hand calculations, SPICE simulations.

Parameter	Hand Calculation			SPICE Simulation		
	Without C_S R_L=100k	**With C_S R_L=100K**	**With C_S R_L=1K**	**Without C_S R_L=100k**	**With C_S R_L=100K**	**With C_S R_L=1K**
A_M						
f_L						
f_H						
BW						
GB=$\lvert A_M \rvert$BW						

Table III: Summary of SPICE simulation including parasitic capacitances and experimental results.

Parameter	SPICE Simulation With parasitic capacitances			Experimental Result		
	Without C_S R_L=100k	**With C_S R_L=100K**	**With C_S R_L=1K**	**Without C_S R_L=100k**	**With C_S R_L=100K**	**With C_S R_L=1K**
A_M						
f_L						
f_H						
BW						
GB=$\lvert A_M \rvert$BW						

This page to be submitted with the report

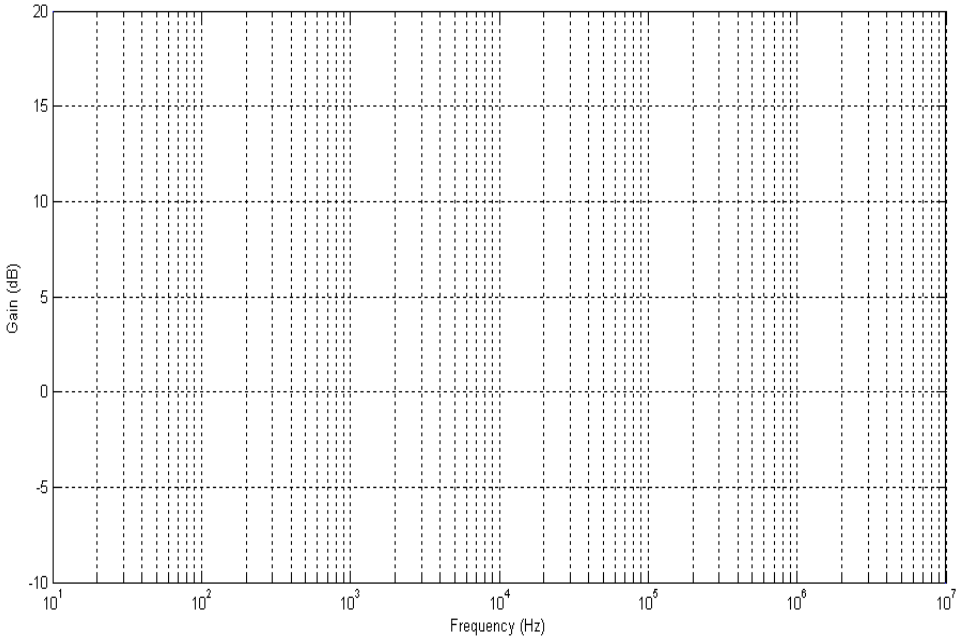


Figure 2: Frequency Response of CS amplifier

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

Experiment # 2

Frequency response of multistage transistor amplifiers

OBJECTIVE

1. To measure the frequency response of common emitter (CE) amplifier.
2. To study the effect of the load resistance R_L on the frequency response of the CE amplifier.
3. To explore some advantages of using multistage amplifier Common Emitter-Common-Collector amplifier.

BACKGROUND

A typical common emitter (CE) amplifier is shown in Figure 1.

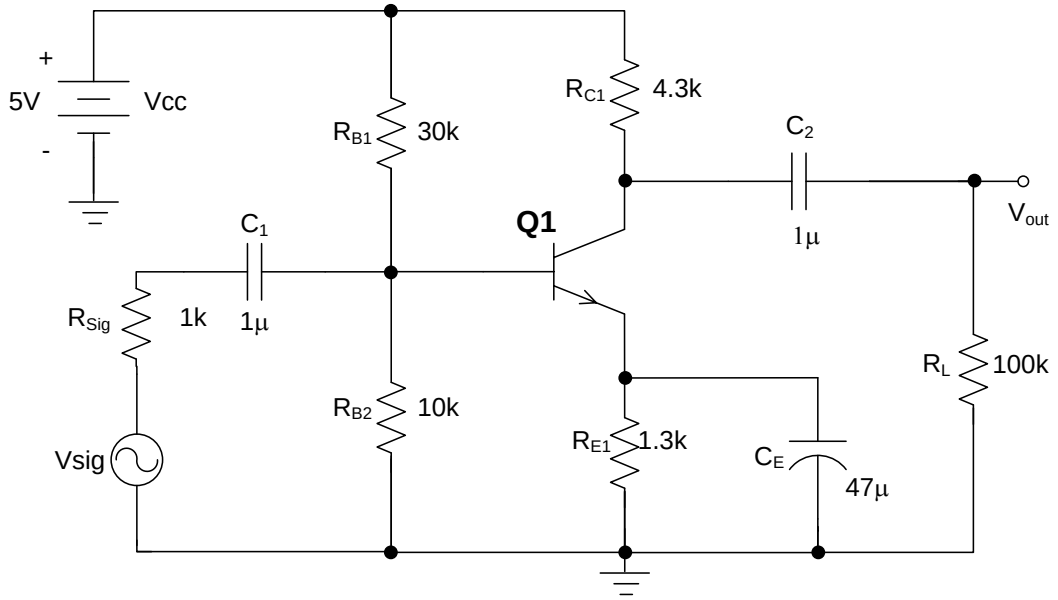


Figure 1: Common Emitter amplifier

As you have studied in your lectures, it can be shown using small signal ac analysis that:
1. The midband frequency A_M gain is given by:

$$A_M = \frac{v_o}{v_{sig}} = \frac{v_{be}}{v_{sig}}\frac{v_o}{v_{be}} = -\frac{R_{in}}{R_{sig}+R_{in}} g_m R_L' \qquad \textbf{(1)}$$

where $R_{in} = R_{B1} // R_{B2} // r_{\pi 1}$ and $R_L^{'} = R_L // R_{C1}$

2. The low frequency pole can be approximated by the pole due to the capacitor assoiciated with the lowest resistance namely C_E:

$$\omega_L \cong \frac{1}{C_E (R_{E1} // \frac{r_\pi + R_S // R_{B1} // R_{B2}}{1+\beta})} \quad \textbf{(2)}$$

3. The high frequency pole can be approximated by the pole due to input capacitance which becomes significantly large because of miller's capacitance $C_{eq} = C_{\mu 1}(1 + g_m R_L^{'})$:

$$\omega_H \cong \frac{1}{[C_{\pi 1} + C_{eq}]R_{in} // R_s} \approx \frac{1}{C_{\mu 1} g_m R_L^{'} (R_{in} // R_s)} \quad \textbf{(3)}$$

Where $g_m R_L^{'}$ represents the voltage gain between the two terminals of $C_{\mu 1}$ (i.e. the gain between the collector and base of Q1). Since R_L is connected at the collector of Q1, changing its value will directly alter both A_M and ω_H. In order to maintain almost constant gain and hence constant bandwidth a buffer stage or common-collector (CC) amplifier can be used to isolate the load from the basic CE amplifier or $C_{\mu 1}$ as shown in Figure 2. Now the input resistance of the CC amplifier (R_{in2}) will act as the new load derived by the collector of Q1. The input resistance of Q2 is slightly dependent on R_L and hence changing R_L will lead to small variation in the gain and ω_H. In other words, since the CC amplifier has a relatively small output resistance which can derive various loads while maintaining almost constant gain. Keeping the gain almost constant will also results in constant bandwidth. Also, note that since CC amplifier has usually much larger bandwidth than that of the CE amplifier, the overall bandwidth of the multistage amplifier will mainly be decided by the poles of CE amplifier.

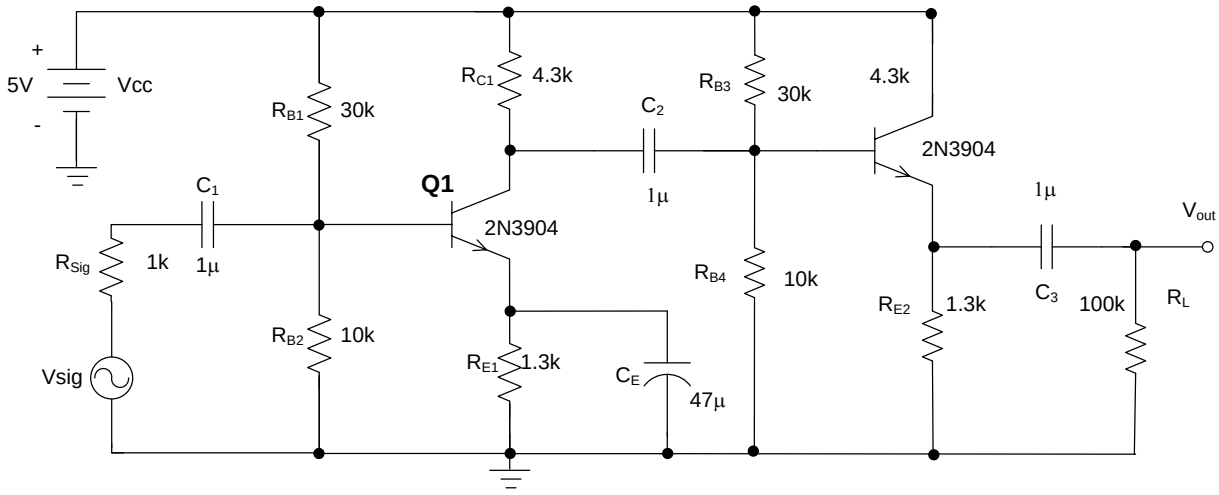


Figure 2: Common Emitter amplifier with Buffer

With the buffer connected the midband the various parameters become:

1. $A_M = \frac{v_o}{v_{sig}} = -\frac{R_{in}}{R_{sig} + R_{in}} g_m R_L^{''} A_{Buffer}$ **(4)**

Where $R_L^{"} = R_{C1} // R_{in2} = R_{C1} // R_{B3} // R_{B4} // [r_{\pi 2} + (R_L // R_{E2})(1+\beta)]$ and

$$A_{Buffer} = \frac{R_L // R_{E2}(1+\beta)}{r_{\pi 2} + R_L // R_{E2}(1+\beta)} \quad \textbf{(5)}$$

2. Note that C_3 is associated with relatively small resistance and hence may significantly contribute to value of the low frequency pole particularly when R_L is small:

$$\omega_L \cong \frac{1}{C_E(R_{E1} // \frac{r_\pi + R_S // R_{B1} // R_{B2}}{1+\beta})} + \frac{1}{C_3[(R_{E2} // \frac{r_{\pi 2} + R_{C1} // R_{B1} // R_{B2}}{1+\beta}) + R_L]} \quad \textbf{(6)}$$

3. The high frequency pole can be expressed as:

$$\omega_H \cong \frac{1}{[C_{\pi 1} + C_{\mu 1}(1 + g_m R_L^{"})]R_{in} // R_s} \approx \frac{1}{C_{\mu 1} g_m R_L^{"}(R_{in} // R_s)} \quad \textbf{(7)}$$

It can be seen that unlike $R_L^{'}$, the value of $R_L^{"}$ will be almost constant for different load values.

EQUIPMENTS & COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator.
2. DC supply V_{cc}=5V from the Board.
3. Two BJT 2N3904.
4. Resistors: R_{sig}=1k,R_{B1}=R_{B2}=30k, R_{B3}=R_{B4}=10k, R_{C1}=4.3k,R_{E1}=R_{E2}=1.3k, R_L=100k and 1K.
5. Capacitors: C_E=47μF,C_1=C_2=C_3=1μF.

PRELAB WORK

Students must perform the following calculations and SPICE before the lab.
Hand Calculation:

1. Given that g_{m1}=g_{m2}=16.9mA/V and $r_{\pi 1}$= $r_{\pi 2}$=7.2kΩ, complete the Table I.

R_L	$R_L^{'}$	$R_L^{"}$	A_{Buffer}
1 kΩ			
100 kΩ			
% Change			

2. For the two amplifier circuits shown in Figure 1 and Figure 2 calculate A_M, f_L, f_H, BW and GB required to complete Table II.

SPICE Simulation:

3. Use the programs developed in the pervious SPICE tutorial to generate the required outputs for Figure 1 and Figure 2 to determine A_M, f_L, f_H, BW and GB to complete Table II. For the SPICE analysis use the frequency range 10Hz to 8MHz. Use the BJT model given in the tutorial.

4. Use three parastic capacitances of about 10pF between each two terminals of the transistors. Repeat step 3 and determine f_H, BW and GB and record your results in Table II.

EXPERIMENTAL WORK

See pin configurations of BJT 2N3904 in the data sheet given in the Appendix at the end of the manual.

1. Construct the circuit shown in Figure 1. Apply a small ac signal v_{sig} and frequency in the midband about 10kHz. Keep increasing the amplitude and make sure by monitoring the output on oscilloscope that the output voltage is not distorted. Calculate $A_{M.}$ Use the corresponding value of v_{sig} in the remaining steps.

2. Reduce the input frequency from 10kHz gradually to find f_L.

3. Go back with frequency to 10kHz and increase it gradually to find f_H.

4. Calculate BW and GB from your measured gain-frequency characteristic.

5. Repeat steps 1 through 4 when R_L is changed to 1kΩ.

6. Construct the circuit shown in Figure 2 with R_L=100kΩ.

7. Repeat steps 1 through 4 for Figure 2.

8. Repeat steps 1 through 4 for Figure 2 when R_L is changed to 1kΩ.

9. Insert your experimental results into Table IV.

10. Compare your hand calculations, SPICE simulations and experimental measurements.

11. Comment on your results.

This page to be submitted with the report

Table II: Summary of hand calculations

Parameter	Figure 1			Figure 2		
	R_L=1 kΩ	**R_L=100 kΩ**	**% Change**	**R_L=1 kΩ**	**R_L=100 kΩ**	**% Change**
A_M						
f_L						
f_H						
BW						
GB=$\lvert A_M \rvert$BW						

Table III: Summary of SPICE simulations

Parameter	Figure 1			Figure 2		
	R_L=1 kΩ	**R_L=100 kΩ**	**% Change**	**R_L=1 kΩ**	**R_L=100 kΩ**	**% Change**
A_M						
f_L						
f_H						
f_H (with parasitic capacitances)						
BW						
BW (with parasitic capacitances)						
GB=$\lvert A_M \rvert$BW						
GB (with parasitic capacitances)						

This page to be submitted with the report

Table IV: Summary of Experimental results

Parameter	Figure 1			Figure 2		
	R_L=1 kΩ	**R_L=100 kΩ**	**% Change**	**R_L=1 kΩ**	**R_L=100 kΩ**	**% Change**
A_M						
f_L						
f_H						
BW						
GB=$\lvert A_M \rvert$ BW						

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

Experiment # 3

Linear Applications of Operational Amplifier

OBJECTIVE

1. To measure the characteristics of several linear circuits based on the operational amplifier namely inverting amplifier, inverting summer, inverting integrator, inverting differentiator and differential amplifier.
2. Learn factors involved in circuit design using op amps.
3. To design circuits to implement simple linear functions.

BACKGROUND

This section provides brief discussion of the main characteristics of the op amp based circuits shown in Figure 1:

Unity-gain Buffer:
It has a voltage gain, input resistance and output resistance of $A_v = v_o / v_i = 1$, $R_i = \infty$ and $R_o = 0$, respectively. It does not take any input current and can drive any desired load resistance without loss of signal voltage. Thus, it used to provide excellent impedance-level transformation while maintaining signal voltage level.

Inverting Amplifier:
It has a voltage gain, input resistance and output resistance of $A_v = v_o / v_i = -R_2 / R_1$, $R_i = R_1$ and $R_o = 0$, respectively. The minus sign means a 180^o phase shift between the output and input signals

Summing Amplifier:
The output voltage can be expresses as $v_o = -[(R_3 / R_1)v_1 + (R_3 / R_2)v_2]$.
It can be seen that the scale factors for the two inputs can be independently adjusted by proper choice of R_2 and R_1. Also, more inputs can be added simply by connecting them same way as v_1 and v_2. Hence, it can be used as a simple digital-to-analog converter.

Inverting Integrator:

The output voltage in time domain can be expresses as $v_o(t) = -\frac{1}{RC}\int_{t_o}^{t} v_i(\tau)d\tau + v_i(t_o)$

This means that the output voltage at time *t* is given by the initial capacitor voltage plus the integral of the input signal from start of integration interval, here, *t*=0. Note that dc gain is infinity which means any small dc component of $v_i(t)$ results in ∞ output. In practice, the op amp will saturates at a voltage close to positive or negative supply depending on input voltage polarity.

Differentiator:

The output voltage in time domain can be expresses as $v_o(t) = -RC\frac{dv_i(t)}{dt}$

This means that the output is scaled version of derivative of input voltage. The differentiator is noise magnifier (i.e. spikes may be produced at output due to sharp changing in $v_i(t)$).

Difference Amplifier:

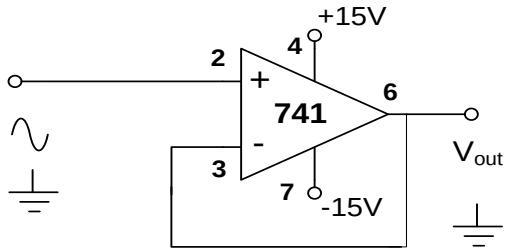
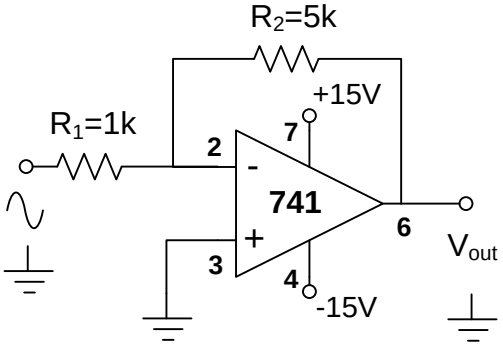
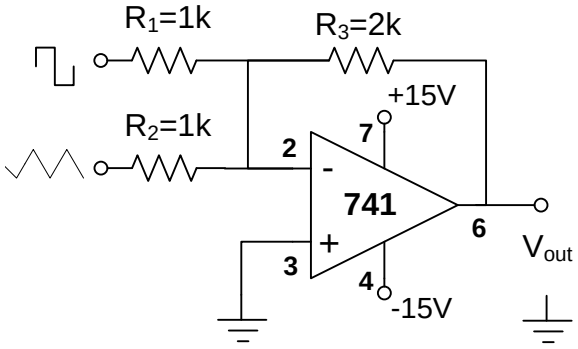
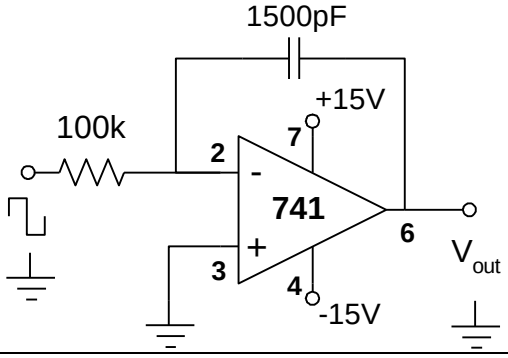
This circuit amplifies difference between two input signals. The output voltage can be expresses as $v_o = -(R_2/R_1)(v_1 - v_2) = (R_2/R_1)(v_2 - v_1)$.

Voltage to Current Converter:

Such a converter is capable of producing a current into a load that is independent of the load value and also proportional to the input voltage. It can be shown for the simple voltage to current converter given in Figure 1 that $I_{out}=V_i/R$ regardless of the value of Z_L. Can you prove that?

EQUIPMENTS & COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator.
2. DC power Supplies of 10V and -10V from the Board.
3. Op amps 741.
4. Resistors: 1kΩ, 2kΩ, 5kΩ, 10kΩ, 100kΩ.
5. Capacitors: 1.5nF, 0.1μF.

Function	CIRCUIT
Unity-gain Buffer	
Inverting Amplifier	
Summing Amplifier (Adder)	
Inverting Integrator	

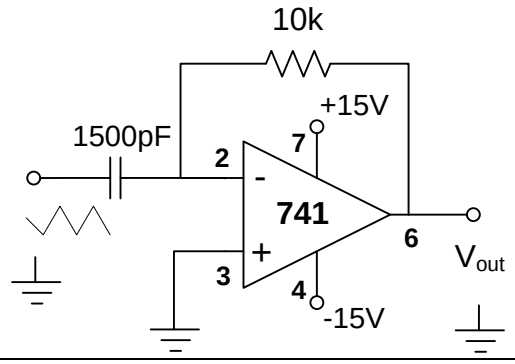
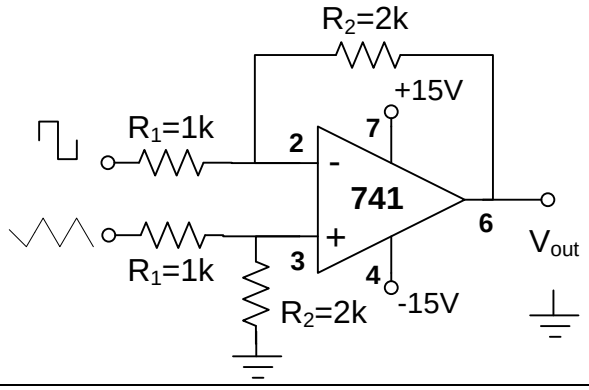
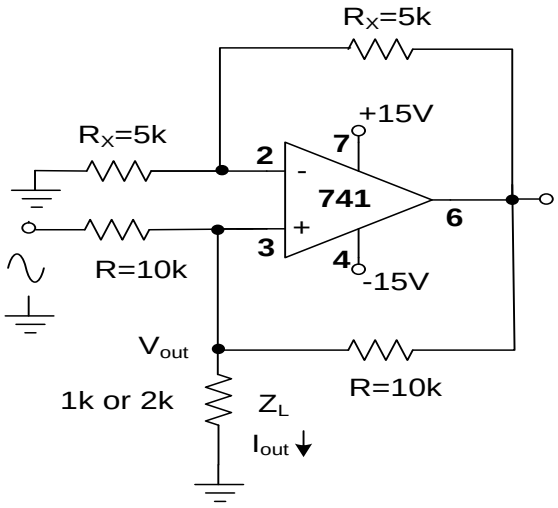
Differentiator	
Difdifference Amplifier	
Voltage to Current Converter	

Figure 1: Basic applications based on op amp.

PRELAB WORK

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

During design phase, use typical values of resistors and capacitors from the list provided in the Appendix of this manual.

Hand Calculation:

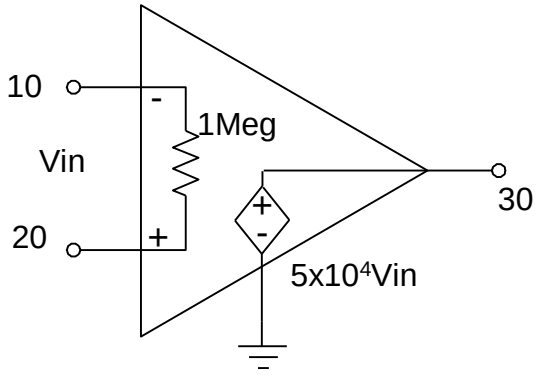
1. For the different configurations shown in Figure 1, perform an approximate hand calculation assuming that the operational amplifier is ideal. In each case sketch the expected output waveform in Table I. Assume that the sine and triangular waves have amplitude of 1V, the square wave varies from 0 to 5V with 50% duty cycle, and frequency of all signals is set to 1kHz.
2. Redesign the integrator circuit so that when the input is sine wave with frequency 1kHz the output voltage will have same amplitude. Use C=0.1μF.
3. Redesign the differentiator circuit so that when the input is sine wave with frequency 1kHz the output voltage will have same amplitude. Use C=0.1μF.

SPICE Simulation:

4. Using SPICE simulate the different configurations and submit the output waveforms for each case. At this stage, the op-amp can be simulated using the simplified model of Figure 2. Usually, the op amp model is written as SUBCIRCUIT in which the model of the op-amp is written only once and then recalled whenever needed. The concept of SUBCIRCUIT is very useful when simulating large systems containing several identical devices. The general format of a SUBCIRCUIT is

```
.SUBCKT SUBNAME N1 N2 N3 …….
CIRCUIT DESCRPTION
.ENDS
```

The first line define the SUBNAME which is the name given to the SUBCIRCUIT and N1, N2, N3, ... are the nodes to which the SUBCIRCUIT will be connected. Then element statements are given. The last line must be the .ENDS control line.



(a)

```
*                  V-    V+   Vo
.SUBCKT OPAMP      10     20    30
RIN  20 10 1MEG
EOUT 30 0 20 10 5E4
.ENDS
```

(b)

Figure 2: A simple SPICE model for the op amp: (a) Circuit (b) SPICE subcircuit

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.

Also, in this experiment you need to generate various types of inputs with 1V amplitude and 1kHz frequency. The square wave input illustrated in Figure 3 can be simulated using the PULSE function with the following general form:

V… N+ N- PULSE(V1 V2 TD TR TF PW PER)

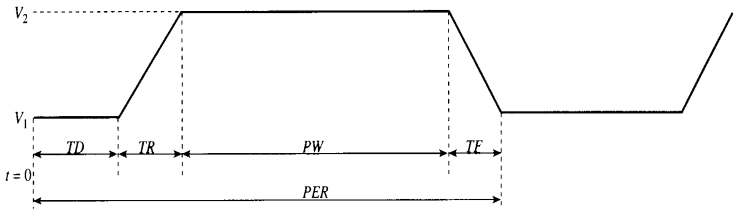


Figure 3: Pulse function.

.

Whereas the triangular wave illustrated in Figure 4 can be simulated using piecewise linear (PWL) function having the following general form:

V… N+ N- PWL(T1 V1 T2 V2 T3 V3 ……)

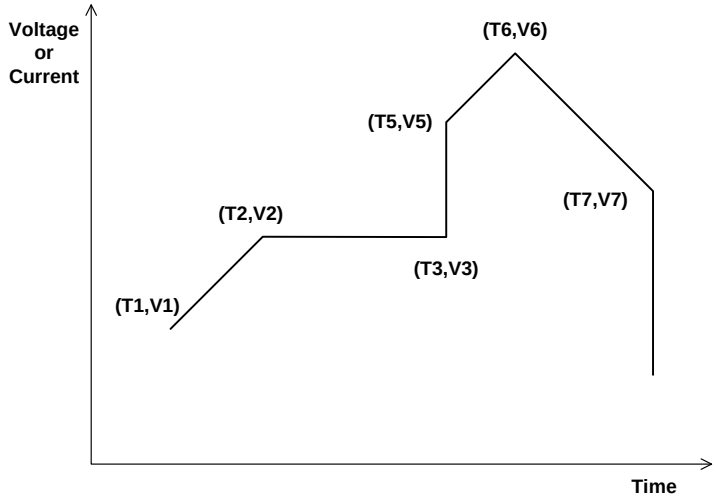


Figure 4: General PWL function.

Figure 5 shows the spice file needed to simulate the adder circuit. Familiarize your self with it. Then use it to develop the programs for other circuits of Figure 1.

```
Adder Circuit Simulation
* Statement for square wave input
*                    TD
VSQ 1 0 PULSE(0   5    0    0.1p    0.1p    0.5m    1m)
*Statement for triangular wave input
VTR 5 0 PWL(0 0 0.25m 1 0.75m -1 1.25m 1 1.75m -1
+2.25m 1 2.75m -1)
R1 1 2 1K
R2 5 2 1K
RF 2 6 1K
* Calling the op amp subcircuit.
X1 2 0 6 OPAMP
*op amp subcircuit
.SUBCKT OPAMP 10 20 30
RIN  20 10 1MEG
* Voltage controlled voltage source
EOUT 30 0 20 10 5E4
.ENDS
.TRAN 0.01m 2m 0
.PROBE
.END
```

Notes:
1. Nodes 2, 0, and 6 in basic circuit will be assigned to nodes 10, 20, and 30 in the subcircuit, respectively.
2. General format for Voltage controlled voltage source is: **Ename N1 N2 NC1 NC2 Value**
In the example, it is specified with positive node 30, negative node 0 while nodes 20 and 10 being the possitive and negative nodes of the controling volatge. The value of the controlling constant is 5E4.

Requiremnt:
Change the input file such that the two inputs are not synchronized. This can be achieved by changing TD of the pulse from 0 to 0.1ms and run the program again.

Figure 5: SPICE file for the adder circuit

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

EXPERIMENTAL WORK

See pin configurations of 741 op amp in the data sheet given in the Appendix at the end of the manual.
Also, note when testing the adder and difference circuits that the two input are not synchronized and hence be careful when you plot the results.

1. Construct the voltage buffer circuit shown in Figure 1. Apply a sine wave of 1V and frequency of 1kHz. Monitor the input and output wave forms and sketch the output in Table I.
2. Repeat step 1 for the inverting amplifier.
3. Construct the adder circuit. In this case, two inputs are need. Generate the triangular signal normally from the function generator whereas use its SYNC output to provide the square wave input. Note that square wave varies from 0 to 5V and its amplitude cannot be changed. Sketch the output in Table I. Be careful when you plot the results the two inputs may not be synchronized.
4. Construct the inverting integrator. Apply a square wave normally from the signal generator of 1V and frequency of 1kHz. Monitor the input and output wave forms and sketch the output in Table I.
5. Test your design of the prelab and sketch the output in Table I

6. Construct the differentiator circuit. Apply a triangular wave of 1V and frequency of 1kHz. Monitor the input and output wave forms and sketch the output in Table I.
7. Repeat step 3 for the difference amplifier.
8. Test your design of the prelab and sketch the output in Table I.
9. Construct the voltage to current converter circuit. Apply a sine wave of 1V and frequency of 1kHz. Since the oscilloscope can not be use to measure current, measure the amplitude of the voltage V_{out} for the two cases Z_L=1kΩ and Z_L=2kΩ. Then calculate I_{out} in each case. Record your results in Table I.
10. Comment on your results.

This page to be submitted with the report

Table I: Summary of the Results

Circuit	Hand Calculation	Experimental Result
Unity-gain Buffer		
Inverting Amplifier		
Summing Amplifier (Adder)		
Inverting Integrator	Figure 1	
	Your design	
Differentiator	Figure 1	
	Your design	
Difference Amplifier		
Voltage to Current Converter		

MINI-SYSTEM

Now, you should be apply to combine the basic circuit of Figure 1 to design more general linear functions, see for example the circuit of Figure 3.

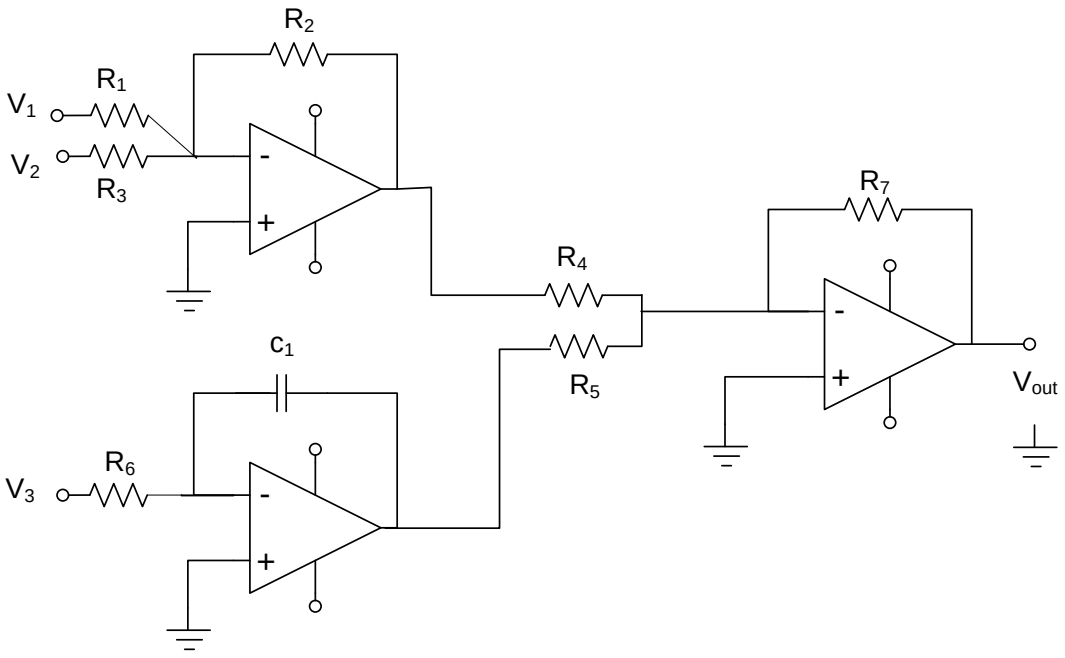


Figure 3: An example of combining some circuit of Figure 1 to design a more general linear function.

Design Problem:

An individual design problem will be assigned by the lab instructor for example:

$$V_{out} = 7V_1 - \int V_2 dt$$

$$V_{out} = -V_1 + 5\frac{dV_2}{dt}$$

$$V_{out} = 3\frac{dV_1}{dt} - \frac{1}{2}\int V_2 dt$$

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

Experiment # 4

Frequency response of op amp based amplifiers

OBJECTIVE

1. To study the effects of the limited op amp bandwidth in the frequency response op amp amplifiers.
2. Investigate how to achieve wider bandwidth for a given gain using multi-stage amplifiers.
3. Introduce the use of Network Analyzer based testing.

EQUIPMENTS & COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator.
2. DC power Supplies of 10V and -10V from the Board.
3. Op amps 741.
4. Resistors: R=1kΩ, R=2kΩ, 5kΩ, 10kΩ, 100kΩ.

BACKGROUND

Just like any amplifier the op amp gain is frequency dependent. General-purpose op amps, internally compensated for stability, can be represented by a single-pole low-pass transfer function:

$$A(s) = \frac{A_o}{1 + s/\omega_b} \Rightarrow A(j\omega) = \frac{A_o}{1 + j\omega/\omega_b} \qquad (1)$$

At high frequencies $\omega >> \omega_b$, the gain can be approximated as:

$$A(j\omega) = \frac{A_o\omega_b}{j\omega} = \frac{\omega_T}{j\omega} \qquad (2)$$

Where A_o is the DC gain, ω_b *is* open loop bandwidth of op amp and ω_T is the unity gain frequency of the op amp or gain bandwidth product (frequency at which magnitude of gain becomes unity). Analyzing the inverting amplifier using the model given by (2) yields:

$$\frac{v_o(s)}{v_i(s)} \approx -\frac{R_2/R_1}{1+\dfrac{s}{\omega_T/(1+R_2/R_1)}}$$

This means that the inverting amplifier has frequency response of a low pass function with DC gain of -R_2/R_1 and –3dB pole at $\omega_T = (1+R_2/R_1)$. Also, it can be seen that as the gain increases the bandwidth decreases. In fact, this is considered to be the most serous disadvantage of using op amp (i.e. the conflict between gain and bandwidth of op amp based circuits). Also, it is worth mentioning that the unity gain inverting amplifier has 3dB frequency of $f_T/2$. One solution to circumvent this problem is through cascading several (N) simple amplifiers as shown in Figure 1.

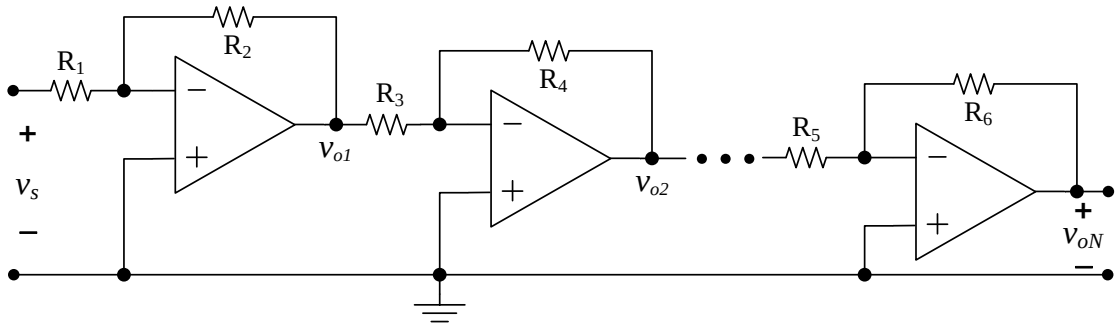


Figure 1: Cascaded amplifier using inverting amplifier

If each sub-amplifier is modeled by a single pole, it can be shown easily that the dc gain of the amplifier is equal to the product of dc gains of individual amplifiers and the bandwidth of the cascade amplifier is $\omega_H = \omega_{H1}\sqrt{2^{1/N}-1}$. Noman Check this relation This implies that distributing the gain over several amplifiers will results in wider bandwidth than using a single stage amplifier.

PRELAB WORK

Students must perform the following calculations and SPICE before the lab.

Hand Calculation:

1. For a single stage inverting amplifier let R_1=1kΩ and complete Table I assuming f_t=1MHz for op amp 741.

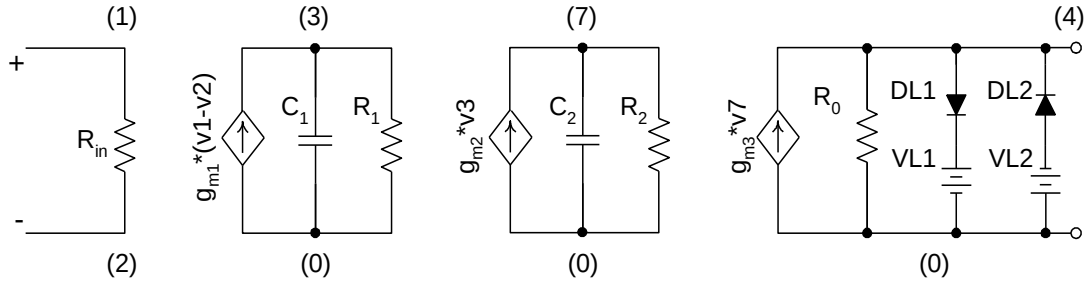
Table I: Hand calculation for prelab.

R_2	DC gain	BW	GB
1k			
10k			
100k			

2. Design a two stage amplifier to provide total gain of 100V/V such that
   (a) Each stage provides gain of -10V/V.
   (b) The first stage provides gain of -5V/V while the second stage provides gain of -20V/V.
3. Estimate the BWs of the two amplifiers required by step 2.

SPICE Simulation:

4. Using SPICE simulate the different amplifiers of step 1 and 2 and submit the output waveforms for each case. Use a more detailed model for simulation op-amps as shown in Figure 3. This model is more sophisticated than the first model presented in Experiment 3, as it models the finite input resistance, the finite differential gain, the finite output resistance, the frequency dependence of the differential gain and the limiting characteristics of the op-amp.




```
*                       V-      V+      Vo
.SUBCKT OPAMP           2       1       4
RIN     1       2       2MEG
GM1     0       3       1       2       20m
R1      3       0       1MEG
C1      3       0       0.031uF
GM2     0       7       3       0       10m
R2      7       0       1K
C2      7       0       39.8pF
GM3     0       4       7       0       1m
R0      4       0       1K
```

```
DL1   4     5     DIODE
DL2   6     4     DIODE
VL1   5     0     DC    13V
VL2   0     6     DC    13V
.MODEL      DIODE       D
.ENDS
```

Figure 2: Detailed Model for the op amp

Comments about Figure 2:

1. The general SPICE statement for voltage controlled current source is
   Gname N1 N2 NC1 NC2 Value
   Where
   N1 and N2 are the positive and negative terminals of the dependent source, respectively.
   NC1 and NC2 are the positive and negative terminals of the controlling voltage source, respectively.

2. DC gain= (GM1x R1)(GM2x R2)(GM3x R0)
3. Dominant pole ω_b=1/R1C1
4. A second pole ω2=1/R2C2 at much higher frequency, usually neglected.
5. The DC voltage sources and diodes set the maximum and minimum output.

EXPERIMENTAL WORK

1. Construct an inverting amplifier and measure the gain and BW for fixed R_1=R_2=1kΩ. Use an input of 1V sine wave with 1 kHz frequency and find the gain. Then gradually increase the frequency until you find the 3dB BW. Estimate the f_t of your op amp.

2. Change R_2 to 10kΩ and measure the gain and BW.

3. Repeat step 2 for R_2=100kΩ. But you need to use a sine wave with input of 0.1V. Why?

4. Record your results for steps 1 through 2 in Table II.

5. Construct two stage amplifiers to test your designs of step 2 of the prelab.

6. Record your results for step 5 in Table III.

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

8. Comment on your results.

This page to be submitted with the report

Table II: Summary of hand calculation and experimental results for one stage.

R_2	Hand Calculations			Experimental Results		
	Gain	**BW**	**GB**	**Gain**	**BW**	**GB**
1kΩ						
10kΩ						
100kΩ						

Table III: Summary of hand calculation and experimental results for two stage amplifiers.

Amplifiers	Hand Calculations			Experimental Results		
	Gain	**BW**	**GB**	**Gain**	**BW**	**GB**
Equal gain for each stage						
Different gains						

DEMONSTRATION: NETWORK ANALYZER

The Lab instructor will demonstrate this part to each group upon the time of finishing the experiment.

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

Experiment # 5

DC imperfections and large signal non-idealities of op amps

OBJECTIVE

4. To measure DC imperfections of the op amp namely offset voltage, biasing and offset currents.
5. To study the large signal limitations of the op amps including output voltage limitation, slew rate, and full-power bandwidth.
6. To see the variations in the performance for different op amps.

BACKGROUND

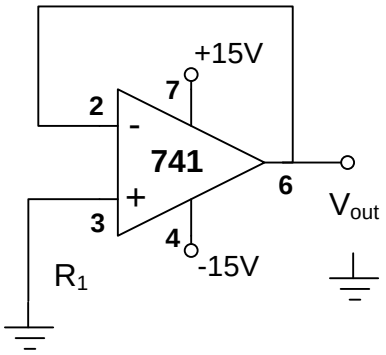
Non-ideal behavior of op amps causes various error terms in practical. In this experiment, DC imperfections of the op amp including offset voltage, biasing and offset currents are measured. Also, various large signal limitations are explored.

Input-Offset Voltage:

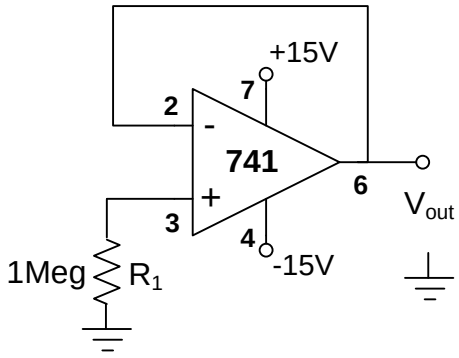
Op amps are direct-coupled devices with large DC gains. Any small DC offset voltage of the input causes the op amp to saturate. Even with inputs being zero, the amplifier output rests at some dc voltage offset level instead of zero. This offset voltage is usually referred to the input port as input offset (V_{os}) by dividing its value by the op amp gain. Typical values for V_{OS} are in the range of 1 to 5mV. Actual sign of V_{OS} is unknown as only upper bound is given. The input offset voltage can be measured as shown in Figure 1(a). Here, the amplifier is connected as voltage-follower to give output voltage equal to offset voltage.

Input-Bias and Offset Currents:

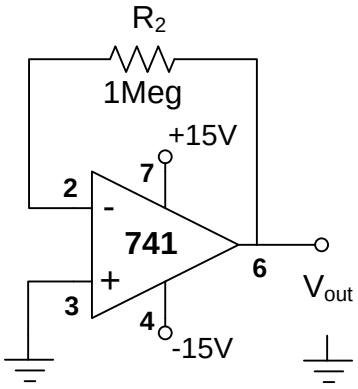
The input currents in both the non-inverting terminal (I_{B1}) and inverting terminal (I_{B2}) are not zeroes in practical op amps particularly those based on BJT. They are similar in value with directions depending on internal amplifier circuit type. The difference between the bias currents is known as input offset current $I_{os} = |I_{B1} - I_{B2}|$ having unknown sign. The circuit of Figure 2 (b) can be used to measure I_{B1} since $V_o = V_{os} - R_1 I_{B1}$ whereas the circuit of Figure 2 (c) can be used to measure I_{B2} since $V_o = V_{os} + R_2 I_{B2}$



(a) Circuit to measure offset voltage



(b) Circuit to measure bias current in non-inverting terminal. Do not neglect V_{OS}



(c) Circuit to measure bias current in inverting terminal. Do not neglect V_{OS}.

Figure 1: Circuit to measure DC offsets.

Offset Compensation:

In general, to measure the offset voltage of a given circuit, the input signal is set to zero as demonstrated for amplifier shown in Figure 2. In this case the circuits for the inverting and non-inverting amplifiers become the same. Apart from the V_{os}, it can be shown without R_2 (short circuit) that $V_o = R_2 I_{B2}$. This means that the V_o is proportional to the magnitude of the biasing current. But when R_B is used and its value is selected as $R_B = R_1 // R_2$ will lead to $V_o = I_{OS} R_2$. Since, offset current (I_{OS}) is typically 5-10 times smaller than individual bias currents, dc output voltage error is reduced by this method. When V_{os} is not neglecting $V_o = (1 + \frac{R_2}{R_1}) V_{os} + I_{OS} R_2$ why?

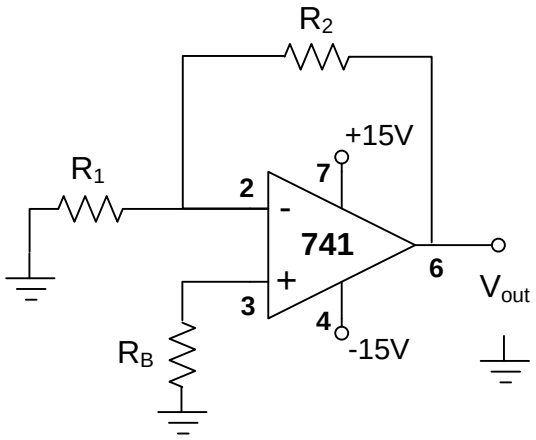


Figure 2: Equivalent circuit for measuring the output offset voltage for both the inverting and non-inverting amplifiers

In addition, the output offset voltage of most IC op amps can be manually adjusted by adding a potentiometer between pins 1 and 5 as shown in Figure 3. Here a small voltage with opposite polarity of the offset voltage is introduced to cancel its effect.

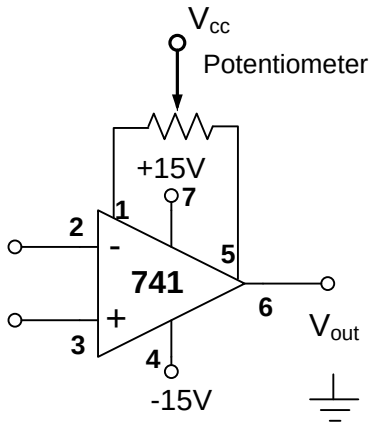


Figure 3: Manual offset nulling

Output Voltage:

General purpose op amps have their output voltage limited to several volts less than power supply span. Like other amplifiers the op amp has a linear range for the output swing before saturation or clipping. It is known as the rated output voltage.

Slew Rate:

The second important large signal limitation of the op amp is known as the slew rate. Slew rate (SR) is defined as the maximum rate of change of voltage at output of op amp. Typical values are in the range from 0.1V/ms to 10V/ms. Slew rate is usually studied by

considering the op amp connected as a voltage buffer. For large step input the output will be ramp. This is because the op amp output is unable to increase at required rate and the op amp is said to be "slewing". Slew rate phenomenon not only causes distortion in large output signal but also in sinusoidal waveforms. Assume the input to the buffer is a sine wave input with frequncy ω_o and amplitude V, if the rate of change of this signal ($\omega_o V$) is more than SR the output will be distorted as shown in Figure 3.

Full-power bandwidth:

Another important term related to slew rate is known as full-power bandwidth. It is usually given in op amp data sheets. It is defined as the maximum frequency (f_M) at which a full-scale signal (maximum possible output V_{omax}) can be processed without slewing. This means $f_M = SR/(2\pi V_{o\max})$.

EQUIPMENTS &COMPONENTS

6. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator.
7. Op amps 741.
8. DC supplies ±15V from the Board.
9. Resistors: 1MΩ, 0.05kΩ, 5kΩ.

PRELAB WORK

Students must perform the following calculations and SPICE before the lab.
Hand Calculation:

Manufacturers usually provide users with the most important parameters of the operational amplifiers. Table I shows the typical performance of selected operational amplifiers. These data, however give the average performance of a selected type. The actual performance of a particular operational amplifier may be different from its typical characteristic. Use the data sheet of 741 op amp given in the appendix to complete Table I.

Table I : Typical Performance of Operational Amplifiers

Parameters	**741**
Input offset voltage (mV)	
Bias current (nA)	
Offset current (nA)	

Open loop gain (dB)	
CMRR (dB)	
Input resistance (MΩ)	
Slew rate (V/μs)	
Unity gain bandwidth (MHz)	
Full power bandwidth (kHz)	

SPICE Simulation:

Transistor level op amp circuit is needed to be used in simulation to determine these non-ideality. Behavioral and linear op amp model presented in experiment 3 and 4 are unsuitable for this purpose.

EXPERIMENTAL WORK

Read the steps before you start you may decide to do part III inside part I and part II.
Part I: DC measurements. Use the multi-meter and record your results in Table I.

1. Construct the circuit of Figure 1(a) and measure the output voltage.
2. Construct the circuit of Figure 1(b) and measure the output voltage and deduce the value of I_{B1}.
3. Construct the circuit of Figure 1(c) and measure the output voltage and deduce the value of I_{B2}. Calculate the value of I_{os} using results of steps 2 and 3.
4. Construct the circuit of Figure 2, with R_B short circuit measure the output. Now use R_B=50Ω measure the output. Do you see the reduction in the output offset?

Part II: Measurements of large signal non-idealities

5. Construct an inverting amplifier with gain -1 using two resistors of 10kΩ. Apply a sine wave input with amplitude of 1V and frequency 1 kHz. Keep increasing the input amplitude until you observe clipping in the output signal. What is the maximum output swing. Record your results in Table II.
6. Construct the circuit of Figure 4, apply a square wave of 20V p-p (here we assume that the dc supply voltage of the op-amp is ±15V i.e. the 20V p-p represents the maximum output voltage of the op-amp) and if we keep the

frequency at, say 1kHz, then the output will be as shown in Figure 4. Notice the effect of slew rate. The slew rate can be easily measured from the output Slew Rate = V_{out}/T_{SR}. Record your results in Table II.

7. Now apply a sine wave input of 20V p-p. Keep increasing the frequency of the input sine wave while monitoring the output until it starts to show distortion as shown in Figure 5. Determine this frequency. This is f_M. Verify the relationship between f_M and the slew rate. Record your results in Table II.

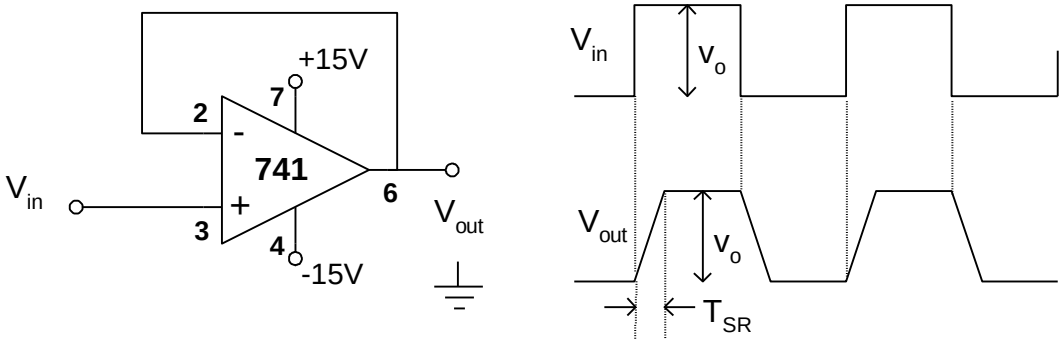


Figure 4: Set up to measure slow rate

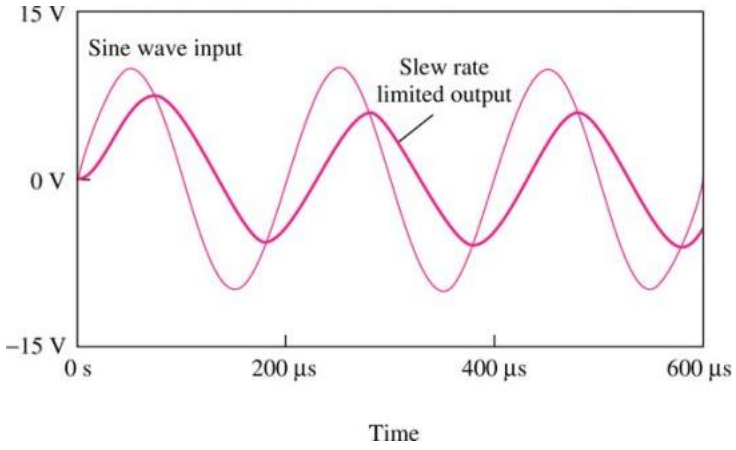


Figure 5: Effect of slew rate on sinusoidal signals

Part III:

8. Repeat steps 1 to 7 using another different op amp. Record your results in Table I and Table II.
9. Comparing the results of part I and II with their counterparts obtained from part III comment on your results.

Part IV:

10. Measure the SR for another type of op amp the will be assigned by the lab instructor.
11. Comment on your results.

This page to be submitted with the report

Table I: Summary of experimental results for part I

Parameter	Op amp 1	Op amp 2
V_{os}		
I_{B1}		
I_{B2}		
I_{os} (from I_{B1} and I_{B2})		
I_{os} (from Figure 2)		

Table II: Summary of experimental results for part II

Parameter	Op amp 1	Op amp 2
Max. Output		
Slew Rate		
Full-power bandwidth		

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

Experiment # 6

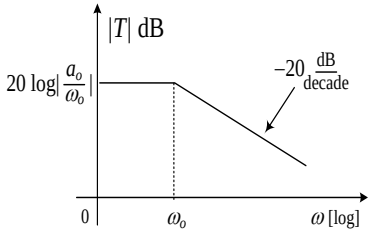
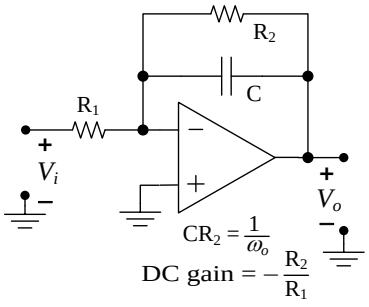
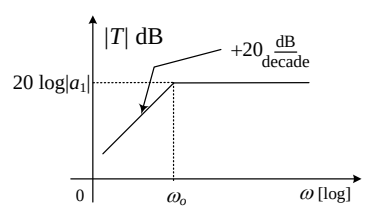
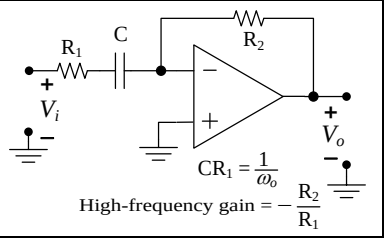
Various types of first-order active filters and their applications

OBJECTIVE

1. To design various type of first-order filters.
2. To measure the characteristics of these filters.
3. To see the high frequency limitation due to op amp limited bandwidth.
4. To cascade first-order sections to design new functions.

BACKGROUND

High order-filters can be realized by cascading first and second order sections. These filters can be implemented by passive RC or LC circuit. However, using active-RC filters based on op amps provides several advantages: gain which can be set to a desired value, independent of some of the filter parameters without affecting others and the output impedance is very low (ideally zero) allowing cascading. Figure 1 gives summary of various types of first order filters including the transfer functions, bode plots and op amp based circuit realizations.

Type	T(s)	Bode plot	Circuit
Low-pass filter	$T(s)=\frac{a_o}{s+\omega_o}$		
High-pass filter	$T(s)=\frac{a_1 s}{s+\omega_o}$		

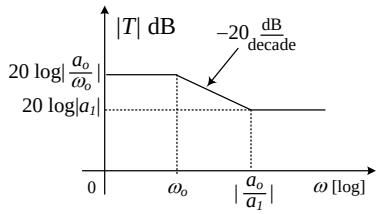
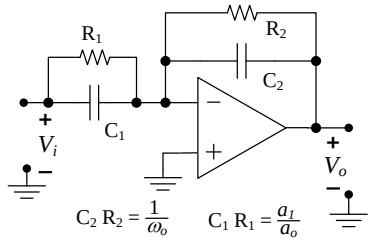
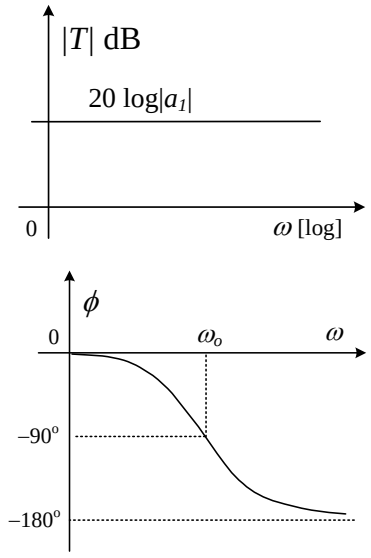
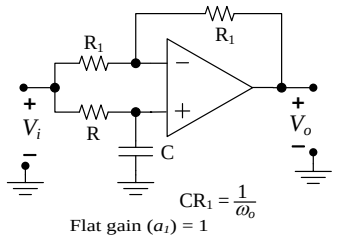
General (bilinear) filter	$T(s) = \frac{a_1 s + a_0}{s + \omega_o}$	 When the zero greater than the pole.	 $C_2 R_2 = \frac{1}{\omega_o}$ $C_1 R_1 = \frac{a_1}{a_o}$ DC gain $= -\frac{R_2}{R_1}$ HF gain $= -\frac{C_1}{C_2}$
All-pass filter	$T(s) = -a_1 \frac{s - \omega_0}{s + \omega_o}$ $\phi(j\omega) = \tan^{-1}(-\omega / \omega_o)$ $-\tan^{-1}(\omega/\omega_o)$ $=-2\tan^{-1}(\omega/\omega_o)$		 $CR_1 = \frac{1}{\omega_o}$ Flat gain (a_1) = 1

Figure 1: Different types of first-order filters

EQUIPMENTS &COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator
2. DC supplies ±15V from the Board.
3. Resistors: 10kΩ (2 NOs), 3.84kΩ, 3.46kΩ, 0.1kΩ, 10kΩ, 100kΩ.
4. Capacitors: C=50nF, 3.54nF, 1μF

PRELAB WORK

Students must perform the following calculations and SPICE before the lab.

Design:

1. Design the LPF to have a 3dB frequency at 10kHz, dc gain of 10 and input resistance of 10kΩ.
2. Using R_1=10kΩ, design the HPF to achieve a corner frequency at 10^4 rad/s and HF gain of 10.

3. Design bilinear circuit to have a zero at 830 Hz a pole frequency at 13kHz and HF gain of 14.1. Select C1=50nF. What will be the DC gain?
4. Design the all-pass filter to realize a 120^o phase shift at 60Hz. Use resistors of R_1=10kΩ and C=1μF.

SPICE:

5. Using SPICE simulates the different filters and submits their frequency responses. Use op amp model of experiment 4.

EXPERIMENTAL WORK

1. Assemble the circuits shown in Figure 1 (Do not forget the supplies of the op amp). Apply sinusoidal input voltage with constant amplitude of 1V, and vary the frequency within the range decided by your hand calculations of the prelab. In each case monitor the input and output voltages on a dual trace oscilloscope.

2. Measure the output voltage and plot your results on the provide graph sheet.

3. From your measurements determine the various parameters of each filter type.

4. For the HPF, bilinear, and allpass filters keep increasing the frequency and monitor the output until the gain starts roll-off (decrease). What is the reason for this problem?

5. Compare your hand calculations, SPICE simulations and experimental measurements and tabulate them in Table I.

6. Comment on your results.

This page to be submitted with the report

Table I: Summary of hand calculations, SPICE simulation and experiment

Circuit		Hand Calculation	SPICE Simulation	Experimental Result
LPF	DC gain			
	f_o			
	Att. at $f = 10f_o$			
HPF	HF gain			
	f_o			
	Att. at $f = 0.1f_o$			
Bilinear Function	DC gain			
	f_o			
	f_z			
	HF gain			
All-pass	Gain			
	Ang. at $0.1f_o$			
	Ang. at f_o			
	Ang. at $10f_o$			

MINI-SYSTEM

Individual design problem will be assigned by the lab instructor.

Now, you should be able to combine the filter circuit of Figure 1 to design different frequency spectrum. Examples include:

1. By cascading a first-order LPF with a first-order HPF one can provide a wideband band-pass filter. Provide such a design for the case in which the midband gain is 10 and the 3 dB bandwidth extends from 100 Hz to 10 kHz. Select the input resistance to be 10kΩ and equal value for the two capacitors.
2. Design of bandstop filter using two Bilinear filters. This response can be achieved by implementing a transfer function such as

$$T(s) = \frac{s+10^3}{s+10^2} x \frac{s+10^4}{s+10^5}.$$

This page to be submitted with the report

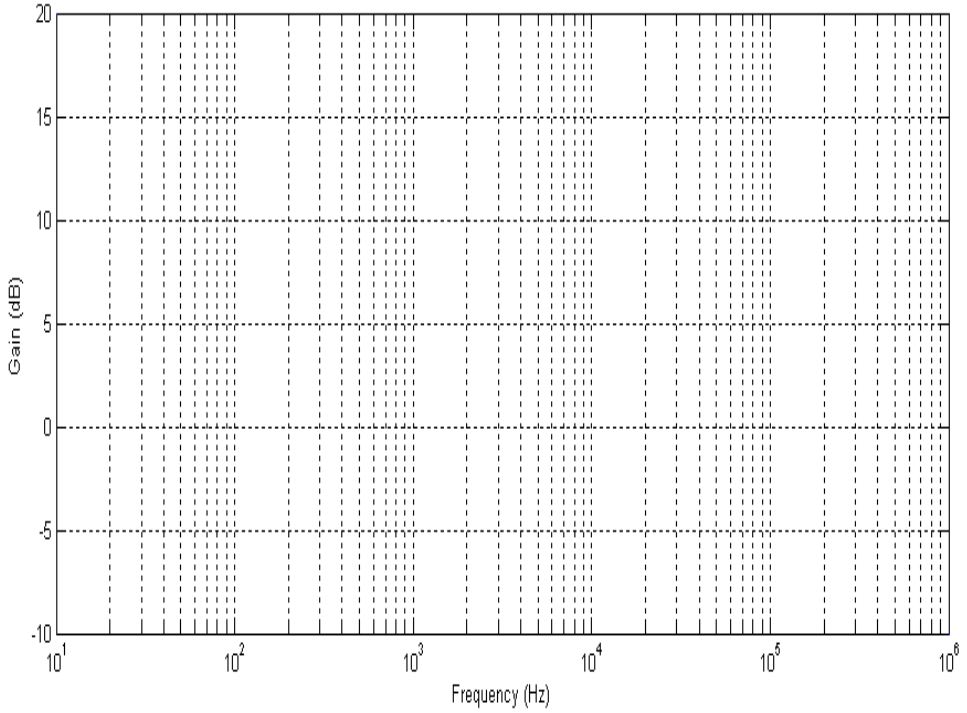


Figure 2: Frequency response of the LPF

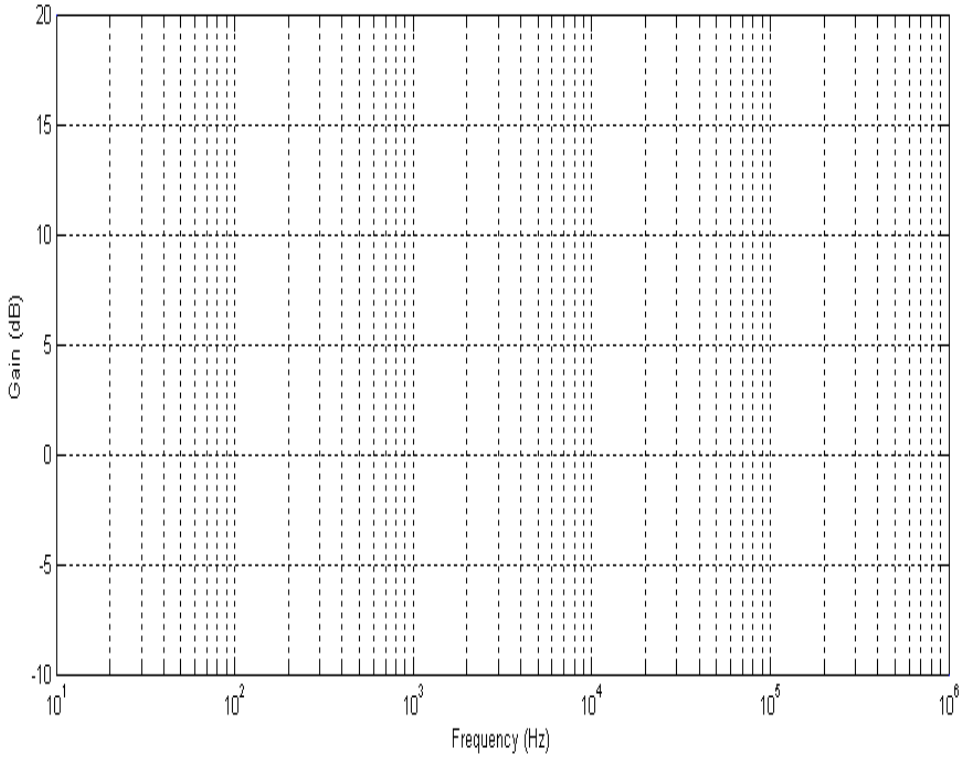


Figure 3: Frequency response of the HPF

This page to be submitted with the report

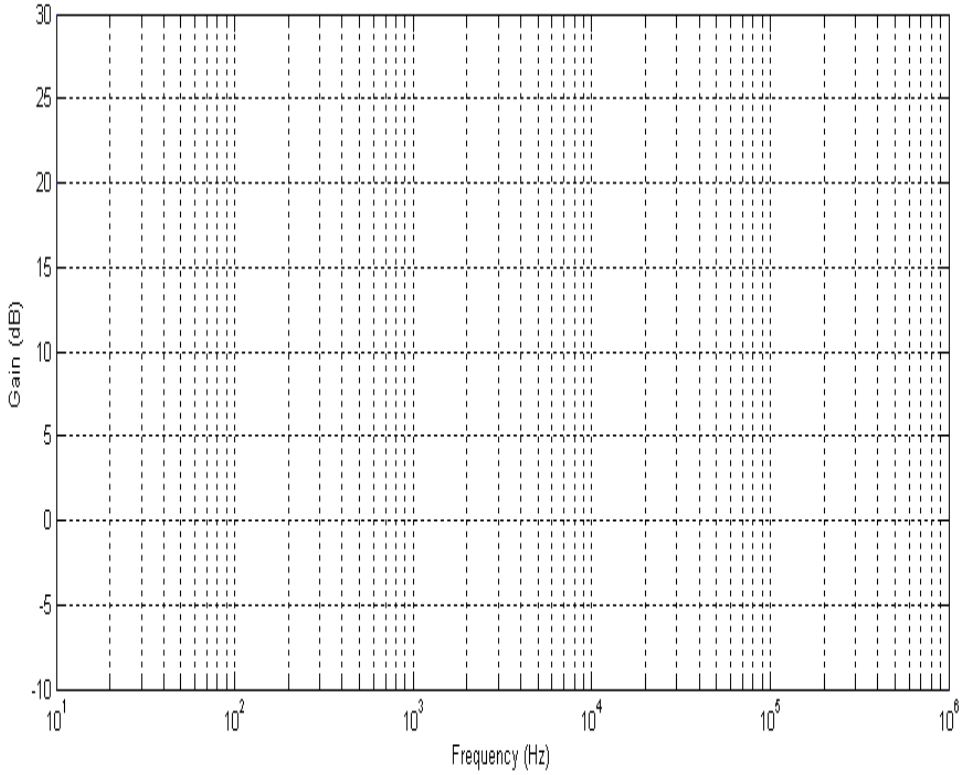


Figure 4: Frequency response of the Bilinear filter

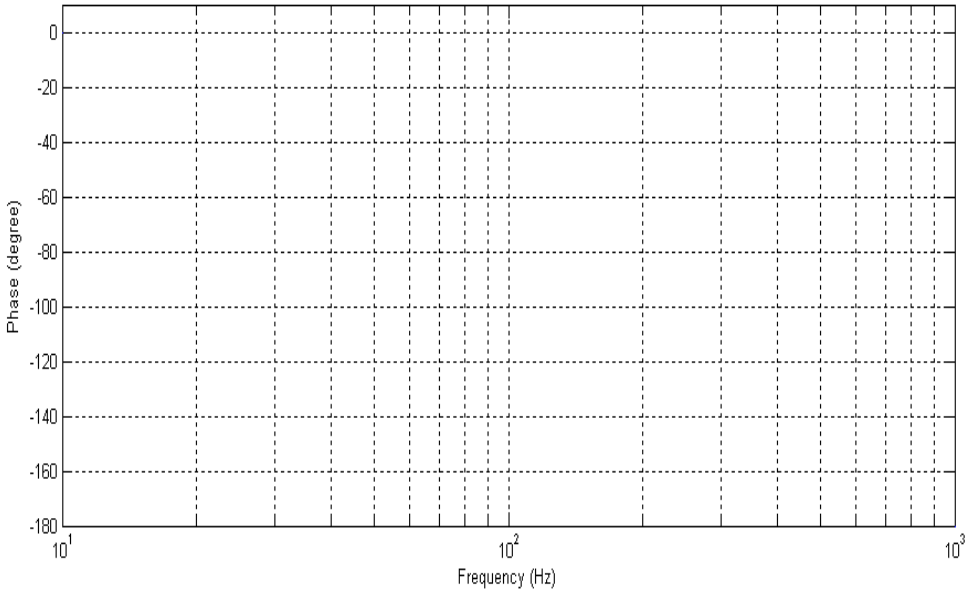


Figure 5: Phase response of the allpass filter

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

Experiment # 7

Second-order active filters

OBJECTIVE

1. To design several second-order filters.
2. To measure the characteristics of these filters.
3. To see the high frequency limitation due to op amp limited bandwidth.

BACKGROUND

Low-Pass Filter Realization

One famous realization of LPF biquad is the Sallen-Key as shown in Figure 1.

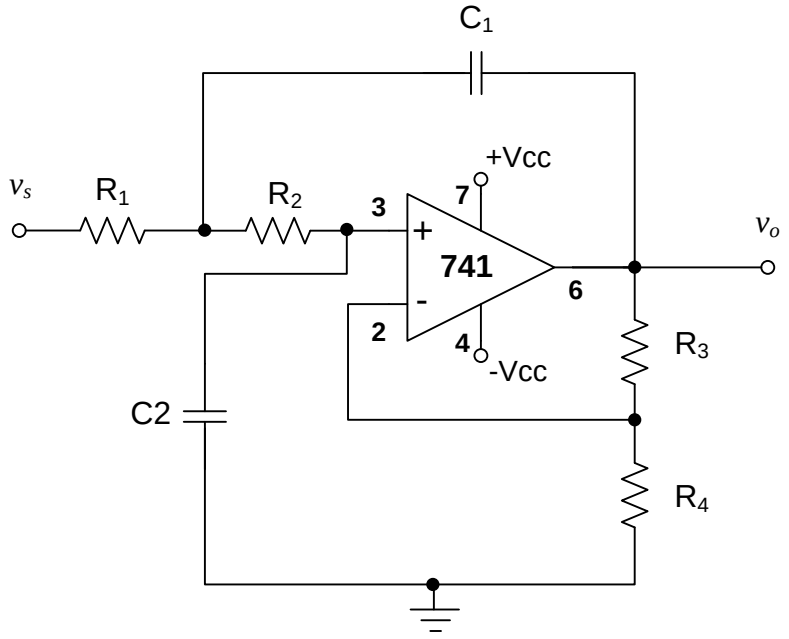


Figure 1 Sallen-Key lowpass filter

$$T_{LP}(s)=\frac{V_o}{V_s}=\frac{K}{s^2+s(C_1R_1+C_2R_1+C_2R_2-C_1R_1K)/(C_1C_2R_1R_2)+1/(C_1C_2R_1R_2)}v_s$$

Where $K=(1+R_3/R_4)$.

The filter parameters are as follows:

a) Low frequency gain=K

b) $\omega_o = 1/\sqrt{C_1 C_2 R_1 R_2}$

c) $\dfrac{\omega_o}{Q} = \dfrac{C_1 R_1 + C_2 R_1 + C_2 R_2 - C_1 R_1 K}{C_1 C_2 R_1 R_2}$

d) $Q = \left(\sqrt{\dfrac{C_2}{C_1}} \dfrac{R_1 + R_2}{\sqrt{R_1 R_2}} + (1-K) \sqrt{\dfrac{R_1 C_1}{R_2 C_2}} \right)^{-1}$

High pass realization

The associated HPF is obtained from LPF by exchanging places of resistors and capacitors (CR/RC transformation).

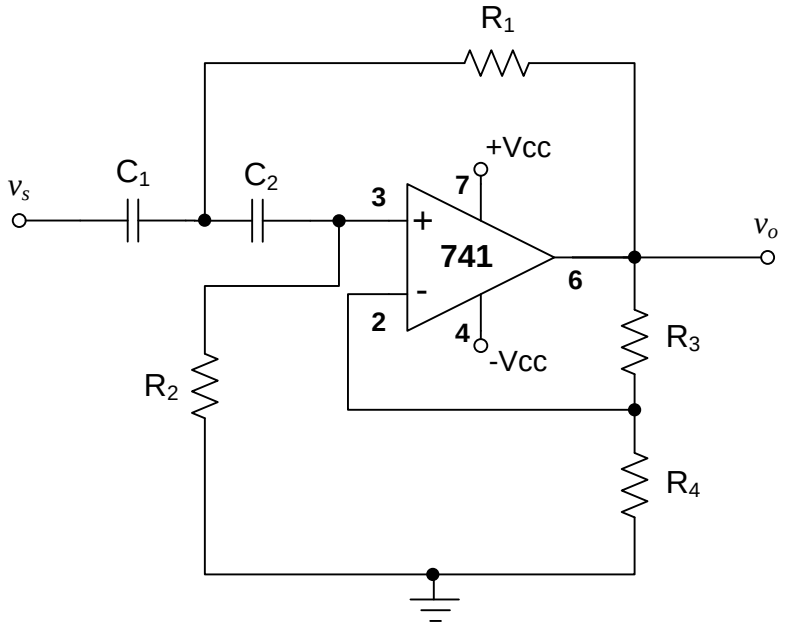


Figure 2: Second-order high pass filter

$$T_{HP}(s) = \frac{v_o}{v_s} = \frac{s^2 K}{s^2 + s(C_1 R_1 + C_2 R_1 + C_2 R_2 - C_2 R_2 K)/(C_1 C_2 R_1 R_2) + 1/(C_1 C_2 R_1 R_2)} v_s$$

Thus, the filter parameters are as follows:

a) High frequency gain=K

b) $\omega_o = 1/\sqrt{C_1 C_2 R_1 R_2}$

c) $\dfrac{\omega_o}{Q} = \dfrac{C_1 R_1 + C_2 R_1 + C_2 R_2 - C_2 R_2 K}{C_1 C_2 R_1 R_2}$

d) $Q = \left(\sqrt{\dfrac{R_1}{R_2}} \dfrac{C_1 + C_2}{\sqrt{C_1 C_2}} + (1-K) \sqrt{\dfrac{R_2 C_2}{R_1 C_1}} \right)^{-1}$

Bandpass realization

A circuit that realize biquadratic BPF based on a single op amp is shown in Figure 3 known as Delyiannis-Friend biquad

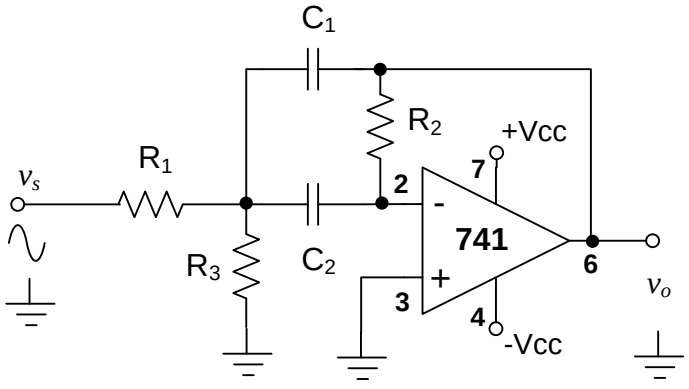


Figure 3: A second-order BPF based on a single op amp.

$$T_{BP}(s)=\frac{v_o}{v_s}=-\frac{s/C_1R_1}{s^2+s(C_1+C_2)/(C_1C_2R_2)+(R_1+R_3)/C_1C_2R_1R_2R_3}$$

Therefore, the filter parameters are as follows:

a) gain= $\dfrac{1/C_1R_1}{(C_1+C_2)/C_2R_2}=\dfrac{C_2R_2}{C_1R_1(C_1+C_2)}$

b) $\omega_o=\dfrac{\sqrt{R_1+R_3}}{\sqrt{C_1C_2R_1R_2R_3}}$

c) $BW=\dfrac{\omega_o}{Q}=\dfrac{(C_1+C_2)}{C_1C_2R_2}$

d) $Q=\dfrac{\omega_o}{BW}=\dfrac{\sqrt{R_1+R_3}}{\sqrt{R_1R_3}}\dfrac{\sqrt{C_1C_2R_2}}{C_1+C_2}$

PRELAB WORK

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

Design:

1. Design the circuit of Figure 1 such that f_o=12.5kHz, Q=5, the dc gain is not specified. Use C_1=C_2=C and R_1=R_2=R.
2. Design the circuit of Figure 3 to achieve f_o=10kHz, Q=3 and gain of 5. Use C_1=C_2=5nF.

SPICE simulation:

3. Use SPICE simulations to verify your designs and submit their frequency responses. Use op amp model of experiment 4.

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

EXPERIMENTAL WORK

1. Construct the filter of Figure 1 and measure the frequency response for your design.

2. Plot your data in the provided graph sheet and determine the parameters required by Table I.

3. Use same components of Figure 1 to assemble the filter of Figure 2.

4. Repeat step 2 for the filter of Figure 2.

5. Construct the filter of Figure 3 and measure the frequency response for your design.

6. Repeat step 2 for the filter of Figure 3.

7. Compare your hand calculations, SPICE simulations and experimental measurements.

8. Comment on your results.

This page to be submitted with the report

Table I: Summary of hand calculations, SPICE simulation and experiment

Circuit		Hand Calculation	SPICE Simulation	Experimental Result
Figure 1	MF Gain			
	Corner Frequency			
	Att. at $f = 10f_o$			
Figure 2	MF Gain			
	Corner Frequency			
	Att. at $f = 0.1f_o$			
Figure 3	MF Gain			
	Center Frequency			
	Bandwidth			
	Att. at $f = 10f_o$			
	Att. at $f = 0.1f_o$			

This page to be submitted with the report

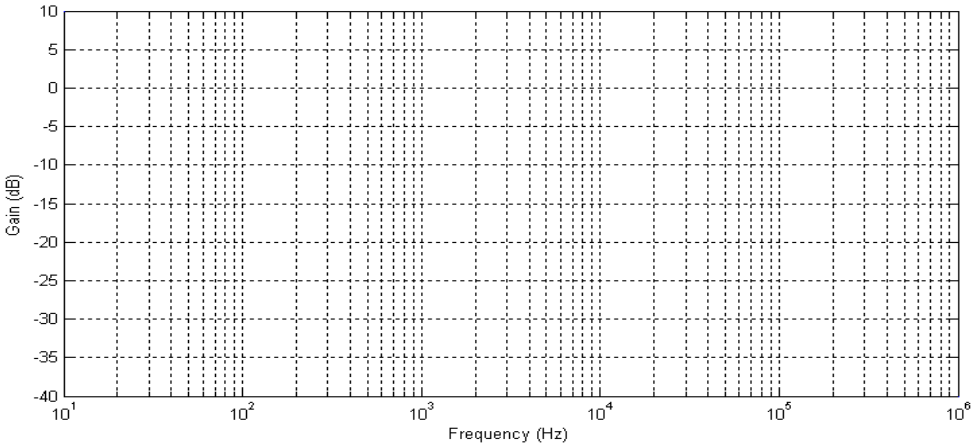


Figure 4: Frequency response of the LPF

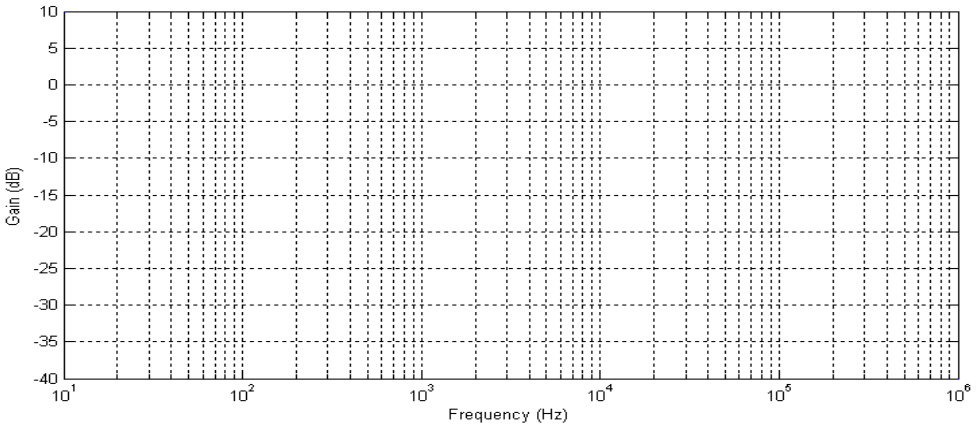


Figure 5: Frequency response of the HPF

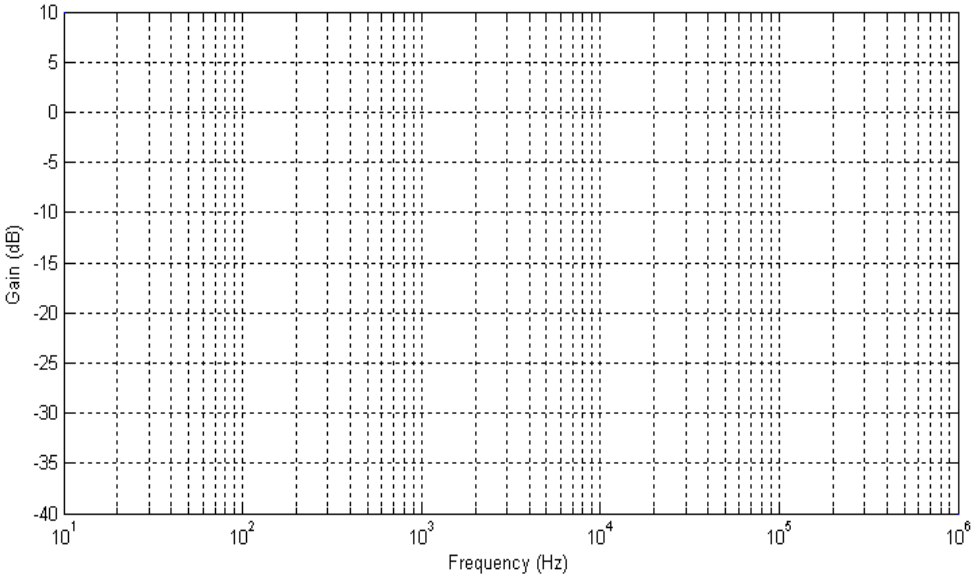


Figure 6: Frequency response of the BPF

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

Experiment # 8

Applying Negative Feedback on Amplifiers and Rectifiers

OBJECTIVE

1. To investigate the properties of negative feedback amplifiers.
2. To test the operation of a shunt-shunt feedback.
3. To illustrate how negative feedback can improve rectifier circuits (reduce non-linear distortion).

BACKGROUND

Shunt-Shunt

A typical MOSFET based shunt-shunt feedback amplifier is shown in Figure 1.

It can be shown that the open loop gain $A = -g_m(R_L // R_D // R_F)(R_1 // R_2 // R_F // R_{sig})$ V/A and $\beta = -1/R_F$ A/V. The closed loop gain can be calculated $A_f = V_o / I_s = A/(1+\beta A)$ leading to voltage gain of $V_o / V_{sig} = A_f / R_{sig}$.

The input resistance $R_{in} = (1/R_{if} - 1/R_{sig})^{-1}$ with $R_{if} = R_{ia}/(1+\beta A)$ where $R_{ia} = R_1 // R_2 // R_F // R_{sig}$.

The output resistance $R_{out} = (1/R_{of} - 1/R_L)^{-1}$ with $R_{of} = R_{oa}/(1+\beta A)$ where $R_{oa} = R_L // R_D // R_F$.

Also it is known that negative feedback cause the upper 3 dB frequency pole to increase by the amount of feedback (i.e. $\omega_{Hf} = \omega_H(1+AB)$).

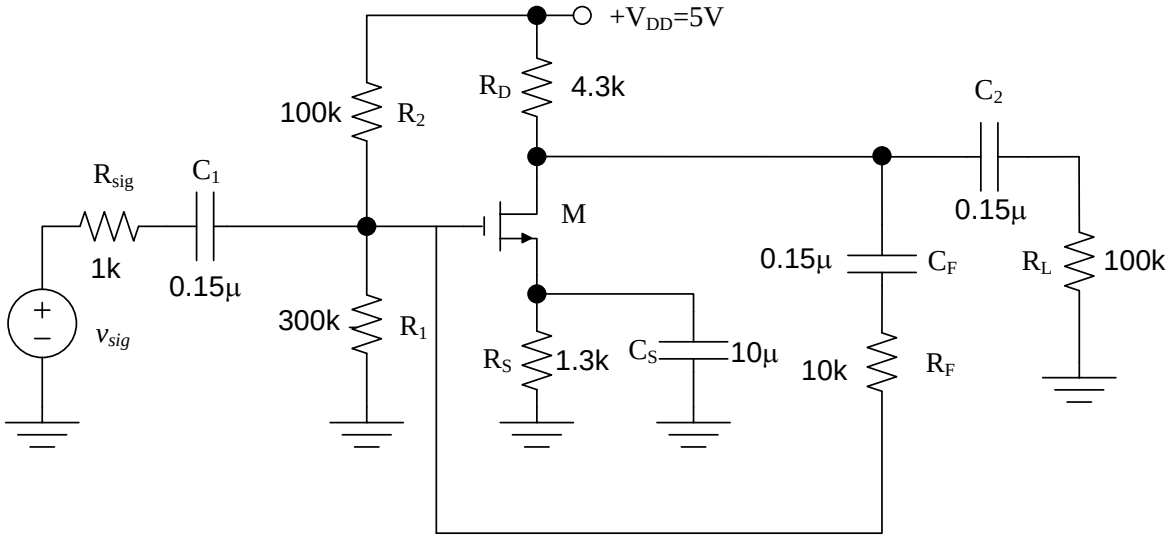


Figure 1 Shunt-Shunt Amplifier based on MOSFET

Precision Rectifiers

The half wave rectifier circuit studied in EE203 suffer from having one diode drop in the signal path. Thus this circuit is suitable for power-supply design where the input signal is much larger than the diode drop. But for applications such as in instrumentation where the signal to be rectified is small (less than 0.5V) cannot be used. The solution is to apply negative feedback across as shown in Figure 2 to form what is called superdiode or precision half-wave rectifier. Similar idea can be used to remove the dead region or crossover distortion encounter in class-AB output stages. The operation of this circuit assuming ideal op amp can be described as follows: For v_I>0, the circuit acts as a voltage follower (i.e. $v_O = v_I$). The feedback loop is closed through the forward biased diode. The feedback mechanism will adjust the op amp output v_{O1} to exactly absorb the forward voltage drop of the diode. The positive load current will force the op amp to provide an equal positive diode current to follow. For v_I<0, the output voltage will tend to go negative. This will tend to induce negative load and diode currents. But the diode cannot pass current from cathode to anode. Thus the diode will be cutoff and the loop will be broken. This force the load current to be zero and thus the output voltage will be zero.

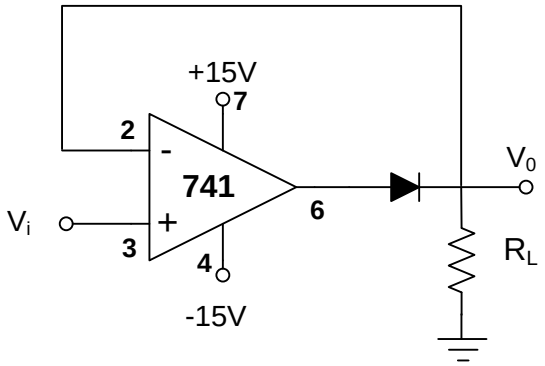


Figure 2: Precision half-wave rectifier

COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator
2. DC supplies from the Board.
3. Resistors:100kΩ (2 NOs), 300kΩ, 1.3kΩ, 4.3kΩ, 10kΩ, 1kΩ.
4. Capacitors: C=10μF, 0.15μF (3NOs)
5. Op-amps 741

PRELAB WORK

Students must perform the following calculations and SPICE before the lab.

Hand calculations:

1. Recall from Experiment 1, the values of A_M, f_L, f_H, BW and GB for the amplifier of Figure 1 but without feedback. Also, calculate the input resistance and the output resistance without feedback.

2. For the circuit shown in Figure 1, use the feedback techniques to calculate A_M, f_L, f_H, BW and GB for this amplifier with feedback.

SPICE simulation:

3. Using SPICE simulate your circuit of Figure 1 and from SPICE output file calculate the parameters of the amplifier obtained in step 2. Use the model of MOSFET given in the tutorial. Run SPICE two times without and with parasitic capacitances of 20pF as demonstrated in Experiment 1.

4. Tabulate the results obtained from your hand calculations and from SPICE simulation in Table I.

5. Draw the voltage transfer characteristic of precision half-wave rectifier.

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

EXPERIMENTAL WORK

Part I:

1. Construct the circuit shown in Figure 1 and apply a small ac signal v_{sig} with a frequency in the midband (about 10kHz) and make sure by monitoring the oscilloscope that the output voltage is not distorted. Measure the midband gain. Use v_{sig} that results in maximum undistorted output in the remaining steps.

   Keep monitoring the input value during measurement to keep it fixed.

2. Decrease the input frequency gradually to determine f_L.

3. Go back with the frequency to 10kHz and increase it gradually to find f_H.

4. Remove Rs (short circuit) and measure the midband gain for the amplifier of Figure 1. Use the result of this step and step 1 to calculate the input resistance using the formula: $R_{in} / (R_{in}+R_{sig}) = (Gain\ with\ R_{sig})/(Gain\ without\ R_{sig})$

5. Reconnect Rs and remove RL (open circuit) and measure the midband gain for the amplifier of Figure 1. Use the result of this step and step 1 to calculate the output resistance using the formula: $R_L/ (R_L+R_{out}) = (Gain\ with\ R_L)/(Gain\ without\ R_L)$

6. Now remove the resistor R_F and capacitor C_F and repeat steps 1 through 5.

7. Record your results in Table I.

8. Compare your hand calculations, SPICE simulations and experimental measurements.

9. Comment on your results.

Part II:

10. Connect the circuit of Figure 2. Apply sinusoidal signal of 10V (P-P), and frequency 1 KHz and monitor both the input and output signals using the oscilloscope.

11. Use the oscilloscope to display the transfer characteristics with output V_o vertically and input V_i horizontally. Make sure to establish convenient axes near the lower left corner of your oscilloscope screen. Note to observe the transfers characteristics use the X-Y format, which you can find on the display setting of the oscilloscope.

12. Design the precision full-wave rectifier shown in Figure 3. (Hint the circuit uses superdiode for positive half cycle and "inverting superdiode" for the negative half cycle).

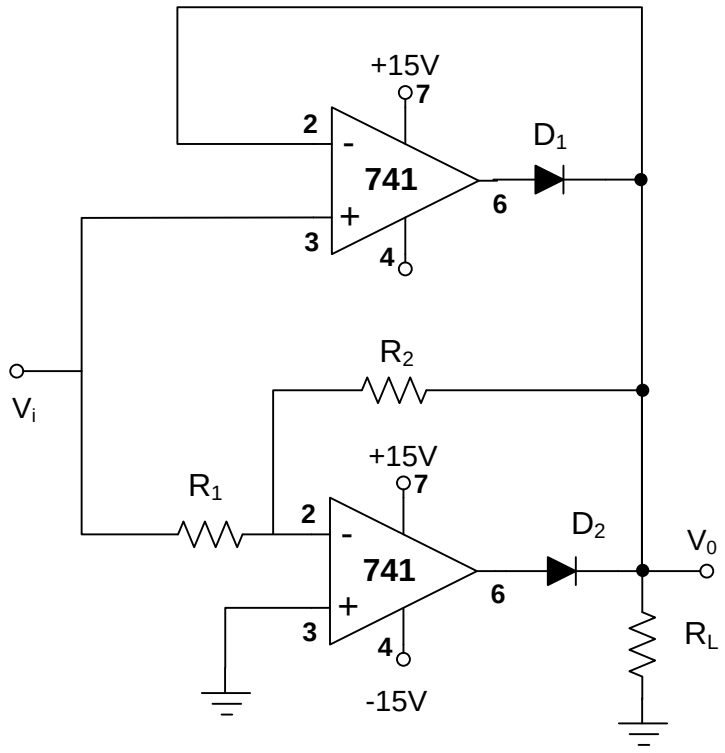


Figure 3: Precision full-wave rectifier

This page to be submitted with the report

Table I: Summary of hand calculations, SPICE simulation and experiment

	With Feedback			Without Feedback		
	Hand Calculation	**SPICE***	**Experiment**	**Hand Calculation**	**SPICE***	**Experiment**
Gain						
f_L						
f_H						
BW						
GB						
R_{in}						
R_{out}						

* For f_H record the SPICE results with and without parasitic capacitances for better comparisons.

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

Experiment # 9

Op amp Based Sinusoidal Oscillators

OBJECTIVE

1. To investigate the operation of sinusoidal oscillators using operational amplifiers, specifically, the Wein-Bridge oscillator, phase shift oscillator, and the quadrature oscillator.
2. To vary the frequency and amplitude of oscillation.

BACKGROUND

The Wien-Bridge Oscillator

The Wien-Bridge oscillator is shown in Figure 1. It can be shown that when $R_p = R_s = R$ and $C_p = C_s = C$, the frequency and condition of oscillation will be $\Rightarrow \omega_o = \frac{1}{CR}$, $\frac{R_2}{R_1} \geq 2$ respectively. However, one must select R_2/R_1 slightly greater than 2 to make sure that oscillations will start. Thus R_2 will be replaced by a variable resistor (potentiometer) that is varied until oscillations starts. Stop at the value of R_2 that results in minimum required gain to sustain oscillations. This is because loop gain greater than unity causes distorted oscillations.

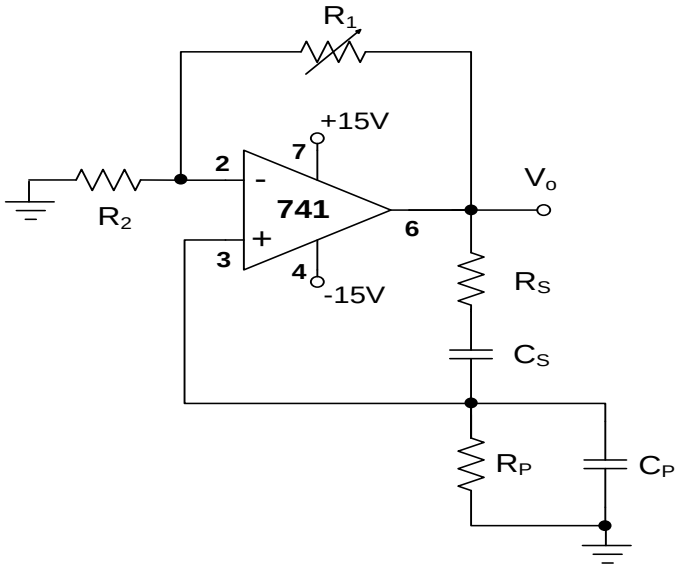


Figure 1: The Wien-Bridge Oscillator

Amplitude Controll

Althogh the frequncy of oscillation of an oscillator circuit can be therotically calculated. The amplitude of oscillation remains unknown. A simple limiter can be used for amplitude control as shown in Figure 2. When R_6=R_7 and/or R_5=R_8 are varied the output swing will change.

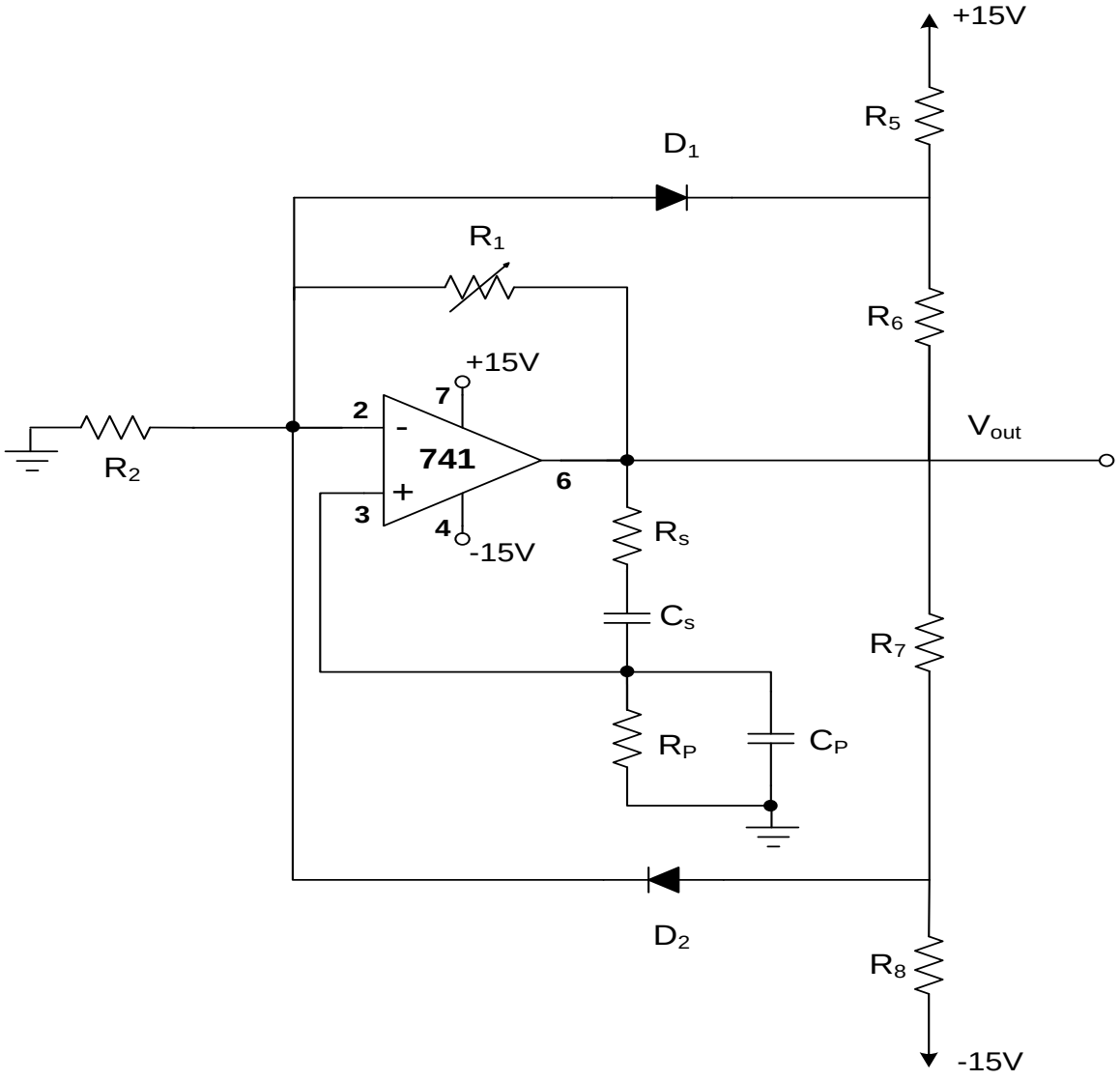


Figure 2: The Wien-Bridge Oscillator with amplitude control

The phase shift Oscillator

The circuit of the phase shift oscillator is shown in Figure 3. The circuit will oscillate when $\Rightarrow R_f \geq 12R$ and the oscillation frequency will be $\Rightarrow \omega_o = \frac{1}{\sqrt{3}CR}$

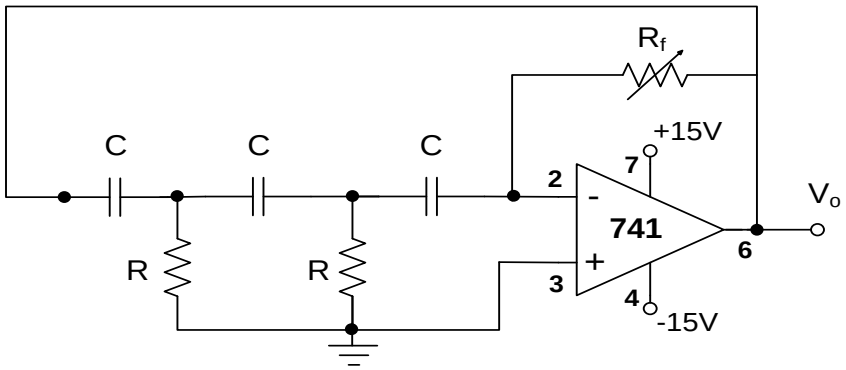


Figure 3: A practical phase-shift oscillator

The Quadrature Oscillator:

The circuit of the Quadrature Oscillator is shown in Figure 4. Since amplifier 1 is used as an integrator. The phase shift between the signal at V_o and V_{o1} will be always 90 (i.e. the circuit provide two sinusoidal output in quadratue). Such quadrature oscillator has wide range of application in communication and control.

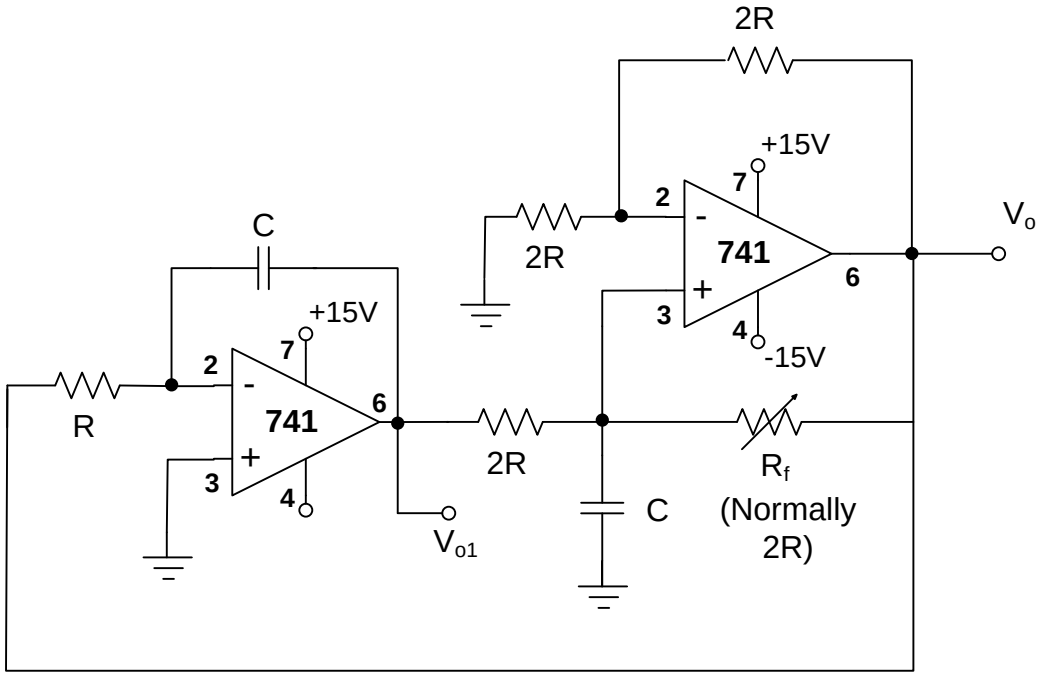


Figure 4: The Quadrature Oscillator

It can be shown that frequency of oscillation $-\omega_o^2 C^2 R^2 R_f + R_f = 0 \Rightarrow \omega_o = \frac{1}{CR}$ whereas the condition of oscillation is $\Rightarrow \frac{1}{2} R_f - R = 0 \Rightarrow R_f \geq 2R$.

Finally, it is worth mentioning that these oscillators are suitable for low frequency applications (up to few hundreds of kHz). This is mainly due the small slew rate and limited gain-bandwidth product of the op amp.

COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator.
2. DC supplies from Board.
3. Resistors:1kΩ (2 NOs),0.27kΩ,10kΩ (3NOs),120kΩ,5kΩ.
4. Capacitors: C=0.1μF (2 NOs),15nF (3NOs)
5. Op-amps 741(2NOs)

PRELAB WORK

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

Design:

1. Design the oscillator of Figure 1 to provide $\omega_o=10^4$ rad/s. Use equal capacitors of 0.1μF.
2. Design the oscillator of Figure 3 such that the frequency of oscillation will be 610Hz. Use equal capacitors of 15nF.
3. Design the oscillator of Figure 4 to provide frequency of oscillation at 2.1kHz. Use equal capacitors of 15nF.

SPICE Simulation:

Using SPICE simulate the different configurations and from SPICE output file obtain the oscillation frequency. For simulating the op-amp you can use the second model presented in Experiment # 4. The simulation of oscillator circuits using SPICE usually produces no output. This is because the oscillator theoretically has no input signal. In practice, an oscillator starts oscillating because there is a small voltage or noise present at the circuit terminals. This small voltage must be included in the simulation of oscillator circuits to overcome the start up problems. To illustrate this point, consider the SPICE program given in Figure 5. It can be seen that to start oscillation a small voltage "VIMAG" is applied for a short time (1μs). This will make sure that oscillations will start. Once the oscillations starts, they will sustain even when the VIMAG goes to zero after 1μs.

```
Wein Bridge Oscillator Simulation
R1        2 5 270
R2        2 6 541
RS        6 1 10K
CS        1 3 15n
RP        3 0 10K
CP        3 0 15n
VIMAG 5 0 PWL(0 0 0.1u 0.1 0.5u 0.1 1u 0)
X1 2 3  6 OPAMP
.TRAN 0.01m 652m 650m 0.01m
.PROBE
*                          v-   v+   vo
.SUBCKT OPAMP              2    1    4
RIN       1          2     2MEG
GM1       0          3     1          2          20m
```

```
R1      3       0       1MEG
C1      3       0       0.031uF
GM2     0       7       3       0       10m
R2      7       0       1K
C2      7       0       39.8pF
GM3     0       4       7       0       1m
R0      4       0       1K
DL1     4       5       DIODE
DL2     6       4       DIODE
VL1     5       0       DC      13V
VL2     0       6       DC      13V
.MODEL          DIODE   D
.ENDS
.END
```

Figure 5: Example for SPICE file to simulate sinsoudal oscillators.

4. Tabulate the results obtained from your hand calculations and from SPICE simulation in Table I.

EXPERIMENTAL WORK

1. Assemble the circuit shown in Figure 1 using your design values. Change the variable resistance until you get an output on the oscilloscope. This means that your circuit is oscillating. Record frequency of oscillation and the value of the resistance at which oscillation just starts to appear on the oscilloscope in Table I.

2. Repeat step 1 for circuit shown in Figure 2 for two cases $R_5=R_8=5k\Omega$ and $R_5=R_8=5k\Omega$. Use $R_6=R_7=1k\Omega$.

3. Repeat step 1 for circuit shown in Figure 3.

4. Repeat step 1 for circuit shown in Figure 4.

This page to be submitted with the report

Table I: Summary of hand calculations, SPICE simulation and experiment

Circuit		Hand Calculation	SPICE Simulation	Experimental Result
Figure 1	Frequency			
	R_1			
	Amplitude			
Figure 2 $R_5=R_8=5k\Omega$	Frequency			
	R_1			
	Amplitude			
Figure 2 $R_5=R_8=3k\Omega$	Frequency			
	R_1			
	Amplitude			
Figure 3	Frequency			
	R_f			
	Amplitude			
Figure 4	Frequency			
	R_f			
	Amplitude			

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

Experiment # 10

Operational amplifier based signal generators

OBJECTIVE

1. To study how to generate various signal types.
2. To understand the operation of comparator based circuit.
3. To design a square wave generator.
4. Integrate previous lab. Experience to build a mini-system.
5. To test your understanding.

BACKGROUND

Comparator:

The simplest way to implement a comparator is to use the op amp in open loop configuration as shown in Figure 1. For input $V_s > V_{REF}$, output saturates at V_{CC} while for $V_s < V_{REF}$, output saturates at $-V_{EE}$.

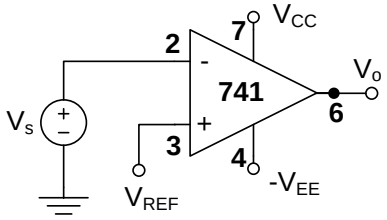


Figure 1: A simple op amp based comparator

Schmitt Trigger:

If an external input Vi is used to trigger the bistable circuit as shown in Figure 2, the circuit becomes known as Schmitt Trigger circuit. The operation of this circuit can be explained as follows:

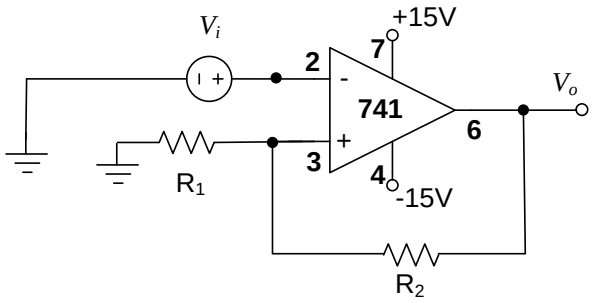


Figure 2: Schmitt Trigger circuit

Before applying Vi, the output will be either at the positive (L_+) or negative (L_-) rated output of the op amp. Assuming the output starts at L_+, this means $v+=V_{TH}= \beta L_+$ with $\beta=R_1/(R_1+R_2)$. The output will remain at this value (L_+) until the input exceeds V_{TH} where the differential input of the op amp becomes negative. Thus the output will change to L_-. Similarly, if the output starts at L_-, the output will stay there unless the input becomes less than $V_{TL}= \beta L_-$ where the differential input of the op amp becomes positive and the output will switch to L_+.

Square wave generation

A simple circuit capable of generating square wave oscillation utilizing Schmitt Trigger is shown in Figure 3.

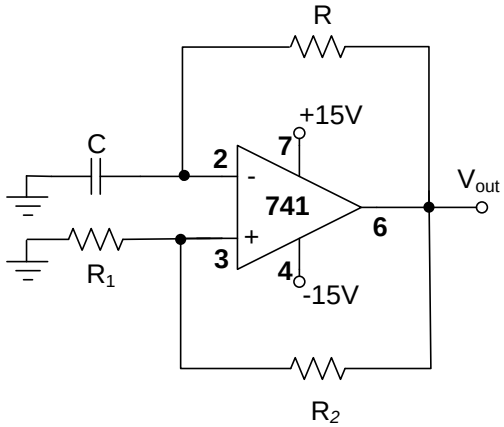


Figure 3: A simple square wave generator circuit

For $R_1=R_2$, the frequency of oscillation will be approximately 1/(2.2RC)

Generating other wave forms from square wave:

A square wave signal can be changed to triangle wave simply by integration. Also, a simple way to produce sin wave from a triangle wave is to use a lowpass filter. As you know a triangle wave can be expressed in terms of infinite some of sinusoidal components. Filtering all components except the fundamental will result in sin wave output. A general signal generation circuit capable of producing square, triangle and sin waves is shown in Figure 4.

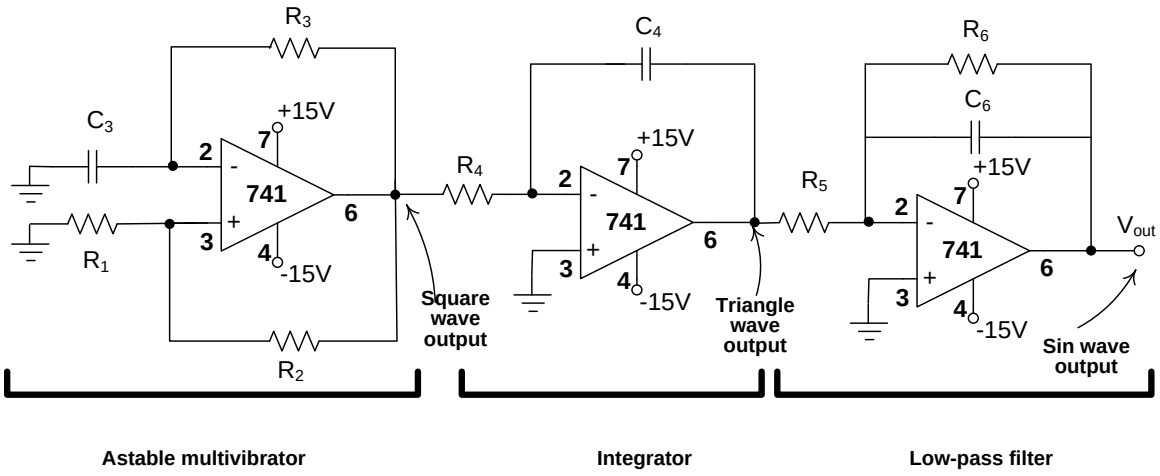


Figure 4: A simple circuit to generate triangle and sin waves from square wave.

Generating other wave forms from sine wave:

You may also start with a sin wave signal and generate the other two waves. Using a comparator after the sin wave will produce a square wave output. Then use an integrator to change the square wave to triangle wave as shown in Figure 5.

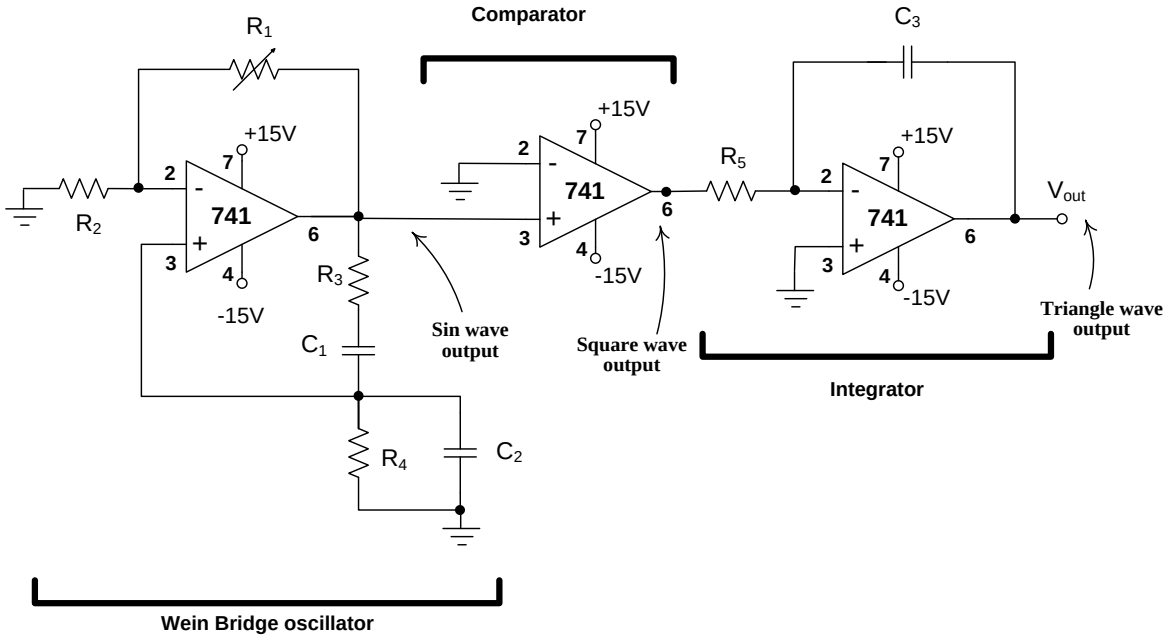


Figure 5: A simple circuit to generate square and triangular waves from Sin wave.

COMPONENTS

1. Digital Oscilloscope, Bread Board, Digital Multi-Meter, Signal generator.
2. DC supplies from the board.
3. Resistors: 1kΩ(2 NOs),0.27kΩ,10kΩ (3NOs),120kΩ,5kΩ

4. Capacitors: C=0.1μF (2 NOs), 15nF (3NOs), 20nF (2 NOs)
5. Op-amp 741(3NOs)

PRELAB WORK

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

Design:

1. Design the oscillator of Figure 3 to provide square wave with frequency 2.5kHz use C=15nF.
2. Design the integrator of Figure 4 such that when the input is square wave with frequency 2.5kHz the output voltage will have same amplitude. Use capacitor of 20nF.
3. Design the LPF of Figure 4 such that it will pass the 2.5kHz signal and attenuate its harmonics as much as possible. Use capacitor of 20nF.
4. Design the integrator of Figure 5 such that when the input is square wave with frequency 1.53kHz the output voltage will have same amplitude. Use capacitor of 20nF.

SPICE Simulation:

5. Using SPICE simulate the circuit of Figure 3 from SPICE output file obtain the oscillation frequency. For simulating the op-amp you can use the model presented in Experiment 4.

EXPERIMENTAL WORK

1. Assemble the circuit shown in Figure 3 using your design. Measure the frequency of oscillation and duty cycle of the square wave output. Sketch the output wave form.

2. Construct the circuit of Figure 4 and test your design. Sketch the different outputs.

3. Use signal generator to provide sine wave with frequency of 1.5kHz with amplitude of 18V (P-P). Construct the circuit of Figure 5 to generate square and triangle waves. Sketch the different outputs.

4. Finally, can you think if you start with triangle wave how a sin and square wave can be generated?

MINI-SYSTEM

Consider the circuit of Figure 6. Observing that it consists of an inverting integrator followed by a sharp comparator (with output of 14V or -14V) in a feedback loop. Answer the following:

1. Assume at t=0, V_{o2} is 14V, what is the value of V_{ol} that makes V_i negative?
2. What will happen when V_{o2} switches to -14V?
3. What are the elements that determine the rate of change of V_{ol}?
4. What are the elements that control the threshold that V_{ol} must reach before the comparator switches states?
5. Draw the output wave from at the output of each op amp.
6. Show for the values given in Figure 6 that the value of the period T = (1.3)RC.

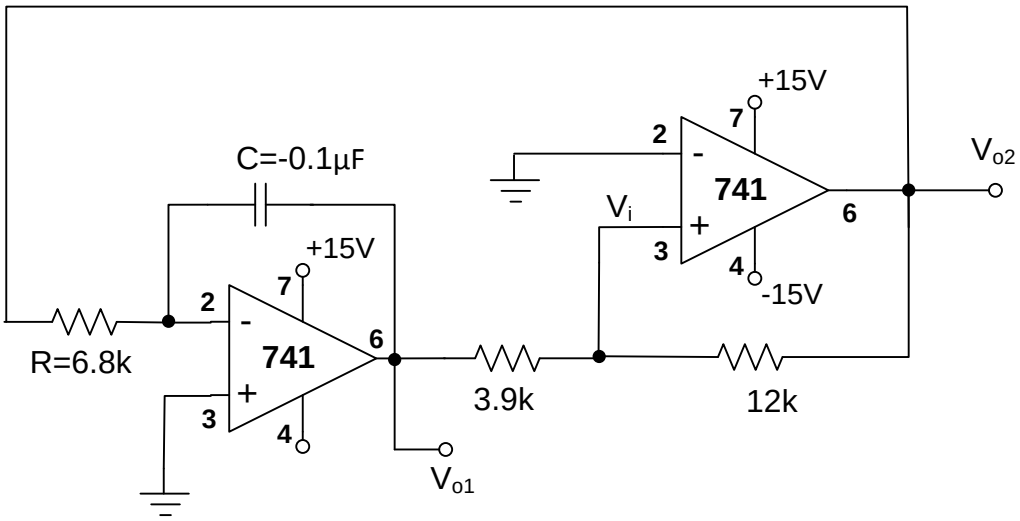


Figure 6: Triangular wave generator (Note that polarity of the op amp inputs

Your lab instructor will assign you to redesign the circuit of Figure 6 for a given frequency of oscillation.

APPENDIX

1. Capacitors and Resistor available in stores are listed below.

CERAMIC	CERAMIC	ELECTROLYTE
1P	1000P	0.47µ
1.5P	2000P	1µ
2P	0.1µ	2.2µ
4.7P	0.01µ	3.3µ
5P	0.05µ	4.7µ
10P	0.0001µ	6.8µ
18P	0.00022µ	10µ
22P	0.00047µ	15µ
27P	0.001µ	22µ
33P	0.0015µ	33µ
47P	0.22µ	47µ
68P	0.0047µ	100µ
100P	0.015µ	220µ
180P	0.022µ	330µ
220P	0.047µ	380µ
270P	0.47µ	
330P	0.15µ	
470P		
560P		

10	**200**	**3.9K**	**82K**
11	**220**	**4.3K**	**91K**
12	**240**	**4.7K**	**100K**
13	**270**	**5.1K**	**110K**
15	**300**	**5.6K**	**120K**
16	**330**	**6.2K**	**130K**
18	**360**	**6.8K**	**150K**
20	**390**	**7.5K**	**160K**
22	**430**	**8.2K**	**180K**
24	**470**	**9.1K**	**200K**
27	**510**	**10K**	**220K**
30	**560**	**11K**	**240K**
33	**620**	**12K**	**270K**
36	**680**	**13K**	**300K**
39	**750**	**15K**	**330K**
43	**820**	**16K**	**360K**
47	**910**	**18K**	**390K**
51	**1K**	**20K**	**430K**
56	**1.1K**	**22K**	**470K**
62	**1.2K**	**24K**	**510K**
68	**1.3K**	**27K**	**560K**
75	**1.5K**	**30K**	**620K**
82	**1.6K**	**33K**	**680K**
91	**1.8K**	**36K**	**750K**
100	**2K**	**39K**	**820K**
110	**2.2K**	**43K**	**910K**
120	**2.4K**	**47K**	**1M**
130	**2.7K**	**51K**	**2.2M**
150	**3K**	**56K**	**10M**
160	**3.3K**	**62K**	
180	**3.6K**	**68K**	

2. Data sheet for MOSFET # 2N4351

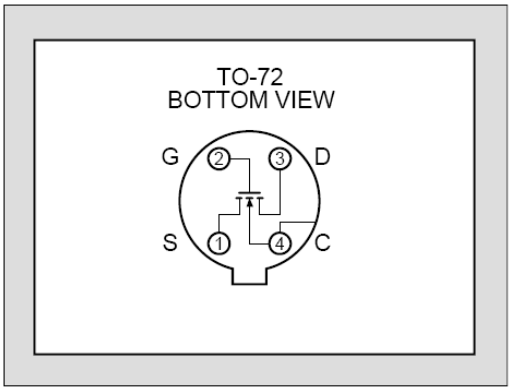
LINEAR SYSTEMS

Linear Integrated Systems

2N4351

N-CHANNEL MOSFET ENHANCEMENT MODE

FEATURES	
DIRECT REPLACEMENT FOR INTERSIL 2N4351	
HIGH DRAIN CURRENT	I_D = 100mA
HIGH GAIN	g_{fs} = 1000µS
ABSOLUTE MAXIMUM RATINGS[1] @ 25 °C (unless otherwise stated)	
Maximum Temperatures	
Storage Temperature	-65 to +200 °C
Operating Junction Temperature	-55 to +150 °C
Maximum Power Dissipation	
Continuous Power Dissipation	375mW
Maximum Current	
Drain to Source	100mA
Maximum Voltages	
Drain to Body	25V
Drain to Source	25V
Peak Gate to Source[2]	±125V



* Body tied to Case.

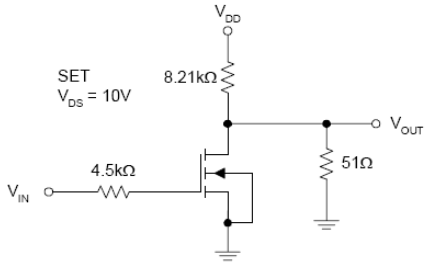
ELECTRICAL CHARACTERISTICS @ 25 °C (unless otherwise stated) (V_{SB} = 0V unless otherwise stated)

SYMBOL	CHARACTERISTIC	MIN	TYP	MAX	UNITS	CONDITIONS
BV_{DSS}	Drain to Source Breakdown Voltage	25			V	I_D = 10µA, V_{GS} = 0V
$V_{DS(on)}$	Drain to Source "On" Voltage			1		I_D = 2mA, V_{GS} = 10V
$V_{GS(th)}$	Gate to Source Threshold Voltage	1		5		V_{DS} = 10V, I_D = 10µA
I_{GSS}	Gate Leakage Current			10	pA	V_{GS} = ±30V, V_{DS} = 0V
I_{DSS}	Drain Leakage Current "Off"			10	nA	V_{DS} = 10V, V_{GS} = 0V
$I_{D(on)}$	Drain Current "On"	3			mA	V_{GS} = 10V, V_{DS} = 10V
g_{fs}	Forward Transconductance	1000			µS	V_{DS} = 10V, I_D = 2mA, *f* = 1MHz
$r_{DS(on)}$	Drain to Source "On" Resistance			300	Ω	V_{GS} = 10V, I_D = 0A, *f* = 1kHz
C_{rss}	Reverse Transfer Capacitance			1.3	pF	V_{DS} = 0V, V_{GS} = 0V, *f* = 140kHz
C_{iss}	Input Capacitance			5.0		V_{DS} = 10V, V_{GS} = 0V, *f* = 140kHz
C_{db}	Drain to Body Capacitance			5.0		V_{DB} = 10V, *f* = 140kHz

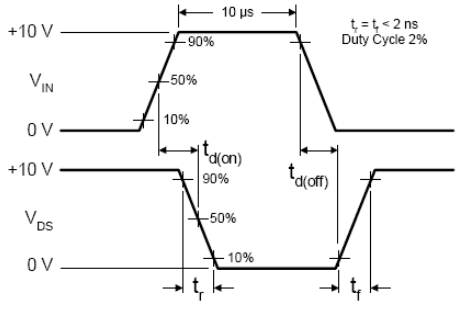
SWITCHING CHARACTERISTICS

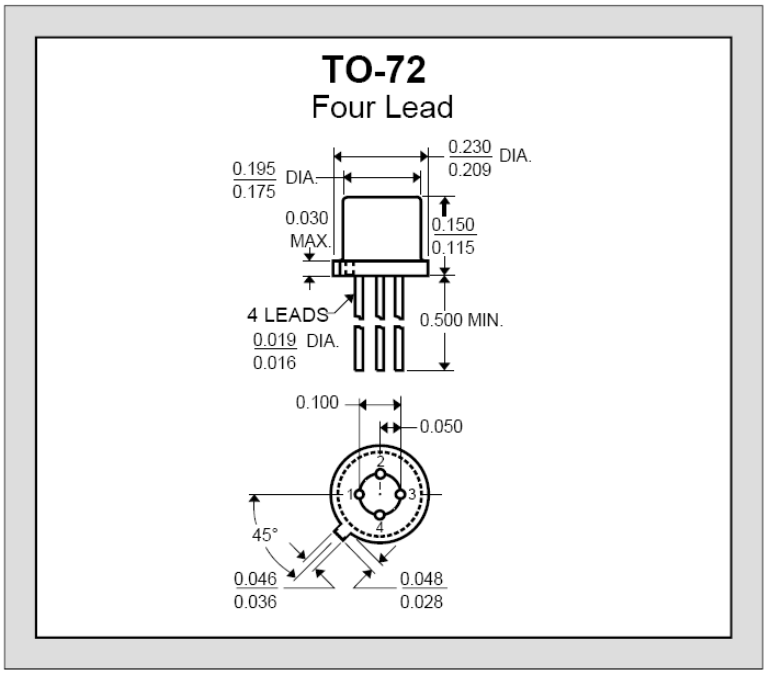
SYMBOL	CHARACTERISTIC	MAX	UNITS
$t_{d(on)}$	Turn On Delay Time	45	ns
t_r	Turn On Rise Time	65	
$t_{d(off)}$	Turn Off Delay Time	60	
t_f	Turn Off Fall Time	100	

SWITCHING TEST CIRCUIT



TIMING WAVEFORMS





3. Data sheet for BJT #2N3904

2N3904

SMALL SIGNAL NPN TRANSISTOR

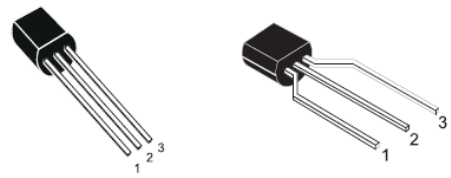
PRELIMINARY DATA

Ordering Code	Marking	Package / Shipment
2N3904	2N3904	TO-92 / Bulk
2N3904-AP	2N3904	TO-92 / Ammopack

- SILICON EPITAXIAL PLANAR NPN TRANSISTOR
- TO-92 PACKAGE SUITABLE FOR THROUGH-HOLE PCB ASSEMBLY
- THE PNP COMPLEMENTARY TYPE IS 2N3906

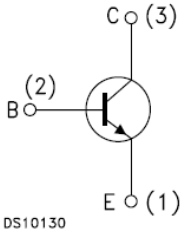
APPLICATIONS

- WELL SUITABLE FOR TV AND HOME APPLIANCE EQUIPMENT
- SMALL LOAD SWITCH TRANSISTOR WITH HIGH GAIN AND LOW SATURATION VOLTAGE



TO-92 Bulk **TO-92 Ammopack**

INTERNAL SCHEMATIC DIAGRAM



DS10130

ABSOLUTE MAXIMUM RATINGS

Symbol	Parameter	Value	Unit
V_{CBO}	Collector-Base Voltage (I_E = 0)	60	V
V_{CEO}	Collector-Emitter Voltage (I_B = 0)	40	V
V_{EBO}	Emitter-Base Voltage (I_C = 0)	6	V
I_C	Collector Current	200	mA
P_{tot}	Total Dissipation at T_C = 25 °C	625	mW
T_{stg}	Storage Temperature	-65 to 150	°C
T_j	Max. Operating Junction Temperature	150	°C

THERMAL DATA

$R_{thj\text{-}amb}$ •	Thermal Resistance Junction-Ambient	Max	200	°C/W
$R_{thj\text{-}case}$ •	Thermal Resistance Junction-Case	Max	83.3	°C/W

ELECTRICAL CHARACTERISTICS (T_{case} = 25 °C unless otherwise specified)

Symbol	Parameter	Test Conditions	Min.	Typ.	Max.	Unit
I_{CEX}	Collector Cut-off Current (V_{BE} = -3 V)	V_{CE} = 30 V			50	nA
I_{BEX}	Base Cut-off Current (V_{BE} = -3 V)	V_{CE} = 30 V			50	nA
$V_{(BR)CEO}$*	Collector-Emitter Breakdown Voltage (I_B = 0)	I_C = 1 mA	40			V
$V_{(BR)CBO}$	Collector-Base Breakdown Voltage (I_E = 0)	I_C = 10 μA	60			V
$V_{(BR)EBO}$	Emitter-Base Breakdown Voltage (I_C = 0)	I_E = 10 μA	6			V
$V_{CE(sat)}$*	Collector-Emitter Saturation Voltage	I_C = 10 mA I_B = 1 mA I_C = 50 mA I_B = 5 mA			0.2 0.2	V V
$V_{BE(sat)}$*	Base-Emitter Saturation Voltage	I_C = 10 mA I_B = 1 mA I_C = 50 mA I_B = 5 mA	 0.65		0.85 0.95	V V
h_{FE}*	DC Current Gain	I_C = 0.1 mA V_{CE} = 1 V I_C = 1 mA V_{CE} = 1 V I_C = 10 mA V_{CE} = 1 V I_C = 50 mA V_{CE} = 1 V I_C = 100 mA V_{CE} = 1 V	60 80 100 60 30		 300	
f_T	Transition Frequency	I_C = 10 mA V_{CE} = 20 V f = 100 MHz	250	270		MHz
C_{CBO}	Collector-Base Capacitance	I_E = 0 V_{CB} = 10 V f = 1 MHz		4		pF
C_{EBO}	Emitter-Base Capacitance	I_C = 0 V_{EB} = 0.5 V f = 1 MHz		18		pF
NF	Noise Figure	V_{CE} = 5 V I_C = 0.1 mA f = 10 Hz to 15.7 KHz R_G = 1 KΩ		5		dB
t_d t_r	Delay Time Rise Time	I_C = 10 mA I_B = 1 mA V_{CC} = 30 V			35 35	ns ns
t_s t_f	Storage Time Fall Time	I_C = 10 mA I_{B1} = -I_{B2} = 1 mA V_{CC} = 30 V			200 50	ns ns

* Pulsed: Pulse duration = 300 μs, duty cycle ≤ 2 %

4. Data sheet for op amp 741.

National Semiconductor

August 2000

LM741
Operational Amplifier

General Description

The LM741 series are general purpose operational amplifiers which feature improved performance over industry standards like the LM709. They are direct, plug-in replacements for the 709C, LM201, MC1439 and 748 in most applications.

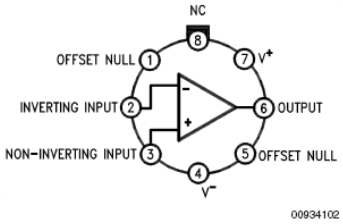
The amplifiers offer many features which make their application nearly foolproof: overload protection on the input and output, no latch-up when the common mode range is exceeded, as well as freedom from oscillations.

The LM741C is identical to the LM741/LM741A except that the LM741C has their performance guaranteed over a 0˚C to +70˚C temperature range, instead of −55˚C to +125˚C.

Features

Connection Diagrams

Metal Can Package

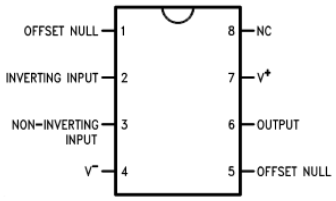


00934102

Note 1: LM741H is available per JM38510/10101

Order Number LM741H, LM741H/883 (Note 1), **LM741AH/883 or LM741CH**
See NS Package Number H08C

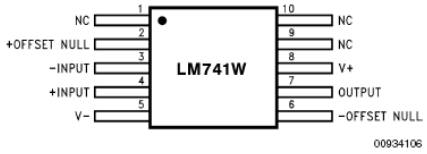
Dual-In-Line or S.O. Package



00934103

Order Number LM741J, LM741J/883, LM741CN
See NS Package Number J08A, M08A or N08E

Ceramic Flatpak

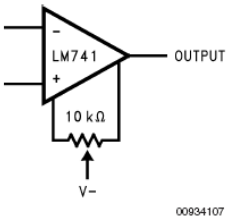


00934106

Order Number LM741W/883
See NS Package Number W10A

Typical Application

Offset Nulling Circuit



00934107

Absolute Maximum Ratings (Note 2)

If Military/Aerospace specified devices are required, please contact the National Semiconductor Sales Office/ Distributors for availability and specifications.

(Note 7)

	LM741A	LM741	LM741C
Supply Voltage	±22V	±22V	±18V
Power Dissipation (Note 3)	500 mW	500 mW	500 mW
Differential Input Voltage	±30V	±30V	±30V
Input Voltage (Note 4)	±15V	±15V	±15V
Output Short Circuit Duration	Continuous	Continuous	Continuous
Operating Temperature Range	−55˚C to +125˚C	−55˚C to +125˚C	0˚C to +70˚C
Storage Temperature Range	−65˚C to +150˚C	−65˚C to +150˚C	−65˚C to +150˚C
Junction Temperature	150˚C	150˚C	100˚C
Soldering Information			
N-Package (10 seconds)	260˚C	260˚C	260˚C
J- or H-Package (10 seconds)	300˚C	300˚C	300˚C
M-Package			
Vapor Phase (60 seconds)	215˚C	215˚C	215˚C
Infrared (15 seconds)	215˚C	215˚C	215˚C
See AN-450 "Surface Mounting Methods and Their Effect on Product Reliability" for other methods of soldering surface mount devices.			
ESD Tolerance (Note 8)	400V	400V	400V

Electrical Characteristics (Note 5)

Parameter	Conditions	LM741A			LM741			LM741C			Units
		Min	Typ	Max	Min	Typ	Max	Min	Typ	Max	
Input Offset Voltage	$T_A = 25˚C$										
	$R_S \le 10\ k\Omega$					1.0	5.0		2.0	6.0	mV
	$R_S \le 50\Omega$		0.8	3.0							mV
	$T_{AMIN} \le T_A \le T_{AMAX}$										
	$R_S \le 50\Omega$			4.0							mV
	$R_S \le 10\ k\Omega$						6.0			7.5	mV
Average Input Offset Voltage Drift				15							µV/˚C
Input Offset Voltage Adjustment Range	$T_A = 25˚C$, $V_S = \pm 20V$	±10				±15			±15		mV
Input Offset Current	$T_A = 25˚C$		3.0	30		20	200		20	200	nA
	$T_{AMIN} \le T_A \le T_{AMAX}$			70		85	500			300	nA
Average Input Offset Current Drift				0.5							nA/˚C
Input Bias Current	$T_A = 25˚C$		30	80		80	500		80	500	nA
	$T_{AMIN} \le T_A \le T_{AMAX}$			0.210			1.5			0.8	µA
Input Resistance	$T_A = 25˚C$, $V_S = \pm 20V$	1.0	6.0		0.3	2.0		0.3	2.0		MΩ
	$T_{AMIN} \le T_A \le T_{AMAX}$, $V_S = \pm 20V$	0.5									MΩ
Input Voltage Range	$T_A = 25˚C$							±12	±13		V
	$T_{AMIN} \le T_A \le T_{AMAX}$				±12	±13					V

Electrical Characteristics (Note 5) (Continued)

Parameter	Conditions	LM741A			LM741			LM741C			Units
		Min	Typ	Max	Min	Typ	Max	Min	Typ	Max	
Large Signal Voltage Gain	T_A = 25°C, $R_L \geq 2$ kΩ										
	V_S = ±20V, V_O = ±15V	50									V/mV
	V_S = ±15V, V_O = ±10V				50	200		20	200		V/mV
	$T_{AMIN} \leq T_A \leq T_{AMAX}$, $R_L \geq 2$ kΩ,										
	V_S = ±20V, V_O = ±15V	32									V/mV
	V_S = ±15V, V_O = ±10V				25			15			V/mV
	V_S = ±5V, V_O = ±2V	10									V/mV
Output Voltage Swing	V_S = ±20V										
	$R_L \geq 10$ kΩ	±16									V
	$R_L \geq 2$ kΩ	±15									V
	V_S = ±15V										
	$R_L \geq 10$ kΩ				±12	±14		±12	±14		V
	$R_L \geq 2$ kΩ				±10	±13		±10	±13		V
Output Short Circuit	T_A = 25°C	10	25	35		25			25		mA
Current	$T_{AMIN} \leq T_A \leq T_{AMAX}$	10		40							mA
Common-Mode	$T_{AMIN} \leq T_A \leq T_{AMAX}$										
Rejection Ratio	$R_S \leq 10$ kΩ, V_{CM} = ±12V				70	90		70	90		dB
	$R_S \leq 50$Ω, V_{CM} = ±12V	80	95								dB
Supply Voltage Rejection	$T_{AMIN} \leq T_A \leq T_{AMAX}$,										
Ratio	V_S = ±20V to V_S = ±5V										
	$R_S \leq 50$Ω	86	96								dB
	$R_S \leq 10$ kΩ				77	96		77	96		dB
Transient Response	T_A = 25°C, Unity Gain										
Rise Time			0.25	0.8		0.3			0.3		µs
Overshoot			6.0	20		5			5		%
Bandwidth (Note 6)	T_A = 25°C	0.437	1.5								MHz
Slew Rate	T_A = 25°C, Unity Gain	0.3	0.7			0.5			0.5		V/µs
Supply Current	T_A = 25°C					1.7	2.8		1.7	2.8	mA
Power Consumption	T_A = 25°C										
	V_S = ±20V		80	150							mW
	V_S = ±15V					50	85		50	85	mW
LM741A	V_S = ±20V										
	$T_A = T_{AMIN}$			165							mW
	$T_A = T_{AMAX}$			135							mW
LM741	V_S = ±15V										
	$T_A = T_{AMIN}$					60	100				mW
	$T_A = T_{AMAX}$					45	75				mW

Note 2: "Absolute Maximum Ratings" indicate limits beyond which damage to the device may occur. Operating Ratings indicate conditions for which the device is functional, but do not guarantee specific performance limits.

Electrical Characteristics (Note 5) (Continued)

Note 3: For operation at elevated temperatures, these devices must be derated based on thermal resistance, and T_j max. (listed under "Absolute Maximum Ratings"). $T_j = T_A + (\theta_{jA} P_D)$.

Thermal Resistance	Cerdip (J)	DIP (N)	HO8 (H)	SO-8 (M)
θ_{jA} (Junction to Ambient)	100˚C/W	100˚C/W	170˚C/W	195˚C/W
θ_{jC} (Junction to Case)	N/A	N/A	25˚C/W	N/A

Note 4: For supply voltages less than ±15V, the absolute maximum input voltage is equal to the supply voltage.

Note 5: Unless otherwise specified, these specifications apply for V_S = ±15V, −55˚C ≤ T_A ≤ +125˚C (LM741/LM741A). For the LM741C/LM741E, these specifications are limited to 0˚C ≤ T_A ≤ +70˚C.

Note 6: Calculated value from: BW (MHz) = 0.35/Rise Time(μs).

Note 7: For military specifications see RETS741X for LM741 and RETS741AX for LM741A.

Note 8: Human body model, 1.5 kΩ in series with 100 pF.

Schematic Diagram