Department of Electrical Engineering
College of Engineering
University of Hail

Laboratory Manual
EE 303 – Electronics II

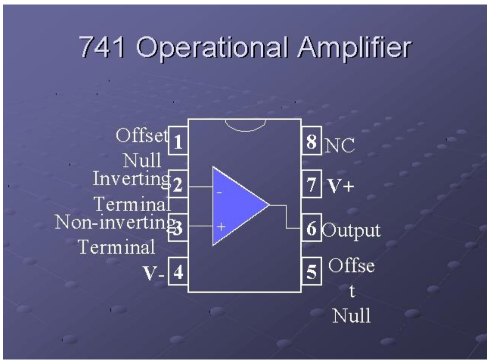


Table of Contents

Preface

This manual contains laboratory experiments for the course "***Electronics II***" (EE 303). These experiments are designed to support, verify and supplement the theory taught in the course "Electronics II (EE 303)". It will also help students gain a workable knowledge on design, analysis and simulation of various basic analog electronic circuits such as amplifiers, filters, oscillators, etc. which are essential circuit building blocks for any real life application.

Most of the experiments have been taken from KFUPM lab manual with some modifications.

Comments and suggestions are welcome from both instructors and students that will help in designing new experiments or modifying the existing ones.

Abdullah Alisher, Lecturer

Introduction

The manual starts with two tutorials on PSPICE, in which the first tutorial shows the simulation of circuits using simple Netlist command codes and then introduces the use of advance models for transistor in PSPICE. After the end of these two tutorials, students are expected to carry out the simulation of rest of the experiment circuits using Netlist command codes.

There are total 8 experiments. The first two experiments deal with the frequency response of transistor amplifier circuits. The third and the fourth deal the linear application and non-ideal characteristics of operational amplifier. The rest of the experiments deal with the filters, feedback and oscillator circuits.

Laboratory Guidelines:

Every week before lab, each student should read over the laboratory experiment and work out the various calculations, etc. that are outlined in the prelab. The student should refer to **Microelectronic Circuits, 4th edition by Sedra and Smith** for the fundamental theory.

- The students should work as a group.
- Most experiments have several parts; students must alternate in doing these parts as they are expected to work in group.
- Static sensitive devices should be handled carefully.
- All equipment, apparatus and tools must be RETURNED to their original place after use.
- Students are NOT allowed to work alone in the laboratory.
- Report immediately to the Lab Supervisor any damages to equipment, hazards, and potential hazards.
- Do not put suspected defective parts back in the bins. Give them to the Lab Technician for testing or disposal.
- Report all equipment problems to Lab Instructor or Lab Technician.
- Each student must have a laboratory notebook. The notebook should be a permanent document that is maintained and witnessed properly, and that contains accurate records of all lab sessions.
- Laboratory and equipment maintenance is the responsibility of not only the Lab Technician, but also the students. A concerted effort to keep the equipment in excellent condition and the working environment well-organized will result in a productive and safe laboratory.

Safety Tips

It is the duty of all concerned who use the electrical laboratory to take all reasonable steps to safeguard health and safety of themselves and the safety of the equipment they use.

Safety of Yourself:

- Inspect all cords, plugs, and equipment for possible damage. Notify your instructor if you notice any damage or any other trouble, such as, loose wall sockets or sparks.
- Be careful when inserting or removing a plug. Do not remove the plugs by pulling on the cord.
- While making connections, keep the power off.
- Do not touch bare wires or parts. If you have to do so, turn off all the power first and unplug the equipment.
- Do not work when your skin is wet.

Safety of Equipment:

- Before plugging ICs into a board, be sure that their pins are straight.
- Unplug ICs very carefully, to avoid bending the pins. You may want to try an IC extractor if one is available in your lab.
- Be aware of the fact that ICs, especially those made with MOS technology, can be instantly damaged by static electricity, such as that accumulated by your body. You should make sure you are "discharged" before handling them, by momentarily touching the metal case of a properly grounded instrument.
- Be sure capacitors are discharged before plugging them into a circuit. If in doubt, short their leads.
- When finished wiring a circuit, inspect all connections to make sure you have made no mistakes. Turn on the circuit only when you are happy with the result.
- If your circuit is connected to a signal generator (such as, function generator), do not turn it on until after you have turned on the power supplies. If you later want to turn off the power, turn off the signal source first.

Laboratory Ethics:

- Do not eat, drink or smoke in the lab.
- Do not use any equipment unless it is mentioned in the lab manual or otherwise advised by the lab instructor to do so.
- Switch off the equipment and disconnect the power supplies from the circuit before leaving the laboratory.
- Observe cleanliness and proper laboratory housekeeping of the equipment and other related accessories.

Tutorial #1: Net Listing and Simulation Analysis Using PSPICE

Objective:

In this tutorial, the students will learn about PSPICE specially in net listing and simulation analysis. There are three basic types of analysis: DC, Transient and AC analysis.

- DC analysis allows you to see the behavior of the circuit in response to DC input voltage/current.
- Transient analysis displays the input-output waveforms as a function of time so you can see if the signal is as expected or distorted.
- AC analysis will show the behavior of the circuit as a function of frequency (frequency response).

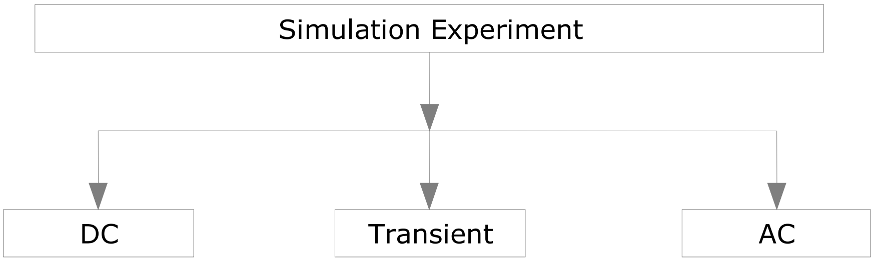


Figure 1

Additional number of SPICE commands in simulation analysis will be introduced and more circuits will be tested.

Introduction:

An electronic circuit does not have to be very complex or contain many elements before the hand calculation effort becomes unwieldy. In fact, for many of the examples worked in the lectures we were forced to use a number of device-model approximations and circuit simplifications. These circuit solutions are usually quite adequate for a first look; however, for a more detailed design and analysis approach, Computer Aided Design (CAD) and Computer Aided Engineering (CAE) tools are used.

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 PSPICE 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 PSPICE by specifying their various components and their terminal connections (net listing).

A typical PSPICE input file format is as follows:

TITLE STATEMENT

CIRCUIT ELEMENTS:
Power Supplies / Signal Sources

Circuit description/Element Descriptions
Model Statement

CONTROL STATEMENTS:
Analysis Requests
Output Requests

.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. Anytime a line starts with an "*", Pspice ignores the whole line. Using an "*" is also handy to block out a command line.
4. You can use upper or lower case letters.
5. Don't forget to add a carriage return after the .END statement.

Control Statements:

.OP
The inclusion of the statement .OP makes SPICE perform DC analysis to find the operating point of the circuit and to print detailed results of the operating point analysis. The general form is .OP

.DC
The inclusion of the statement **.DC** makes the spice perform the dc analysis of the circuit over specified range of input. The general format is

.DC SOURCE_NAME START_VAL STOP-VAL INCREMENT_VAL

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.
Example:
.DC VIN -5 5 1
The above statement will make spice vary the voltage V_{IN} from -5V to 5V in steps of 1V.

.AC
The **.AC** control statement is used to perform ac analysis on a circuit. The general format of the **.AC** statement is

.**AC FREQ-VAR NP START STOP**

Where **FREQ_VAR** 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; its interpretation depends on keyword (DEC, OCT, or LIN) in the FREQ. **FSTART** is the starting frequency. FSTART cannot be zero. **FSTOP** is the final or ending frequency.

Example:
.AC Dec 100 100k 100Meg
The above statement will make Pspice vary the frequency in decade from 100K to 100Meg over 100 point.
Dec means decade (Multiple of 10).
Oct means octave (Multiple of 8).
Lin means linear.

.TRAN
.TRAN makes SPICE perform a time domain transient analysis of the circuit. The general form is:
.TRAN TSTEP TSTOP
where TSTEP is the time increment used for plotting and/or printing results of the transient analysis. TSTOP is the time of the last transient analysis.
Example:
.Tran 0.5ns 200us
The above statement will perform transient analysis up to 200µs in steps of 0.5ns.

.TF
The inclusion of a .TF statement makes SPICE perform a small signal DC analysis yielding the transfer function between a specified output node and a specified input source. The output of the program can print the input resistance at the specified input source, the output resistance at the specified output node and transfer function. The general function is
.TF OUTPUTVAR INPUTSRC
where OUTPUTVAR is a small signal output variable (voltage or current) and INPUTSRC is a small signal source (voltage or current).

.PRINT
The inclusion of the .PRINT statement makes SPICE perform a print of a specified output variable resulting from a specified type of analysis. The general form is
.PRINT ANALYSISTYPE OV
where ANALYSISTYPE is the type of analysis performed from which the output variable OV is obtained (DC, TRAN, & AC).

.PROBE
Probe in PSPICE 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 PSPICE Probe.

.Temp
The above statement will be used to simulate the circuit for different temperature.
Example:
.Temp 0 75 25
The above example will simulate the circuit with temperature varying from 0 to 75 in steps of 25.

Time Dependent Functions for Independent Sources:

In SPICE there are five kinds of time-dependent functions for transient analysis using independent sources. Here we shall introduce only three of them.

PULSE

The general form of is

V…	N+	N-	PULSE(V1 V2 TD TR TF PW PER)
I …	N+	N-	PULSE(V1 V2 TD TR TF PW PER)

Figure 1 shows a pulse form which illustrates the parameters mentioned above.

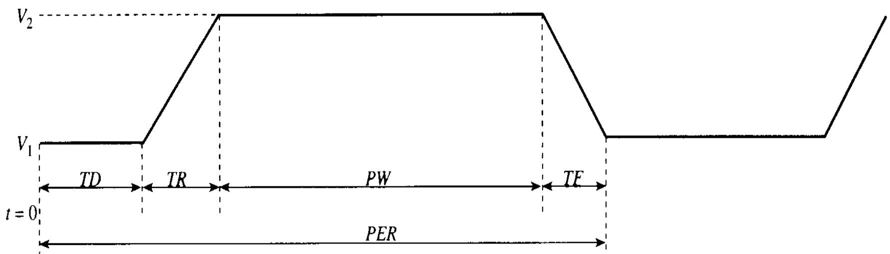


Figure 2

SIN

The general form is

V…	N+	N-	SIN(VO VA FREQ TD THETA)
I …	N+	N-	SIN(VO VA FREQ TD THETA)

Figure 3 shows a sinusoidal form which illustrates the parameters mentioned above.

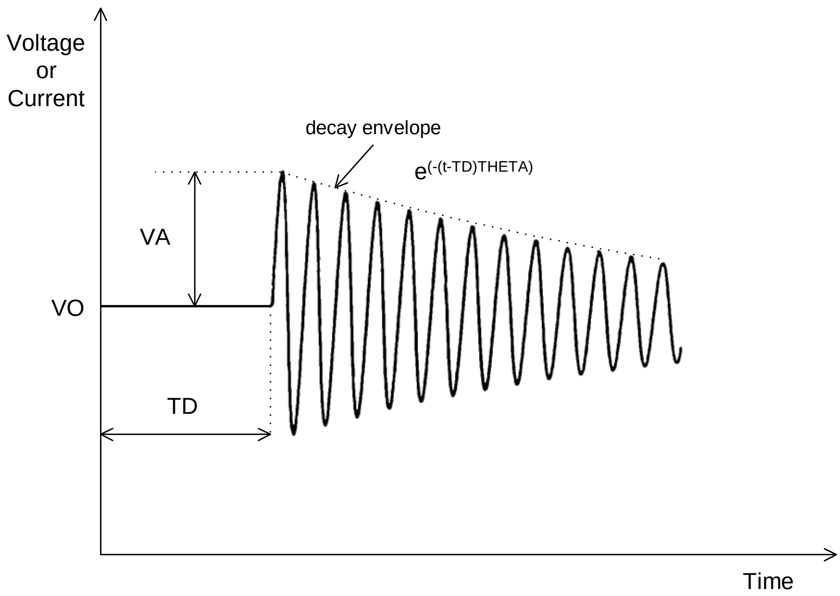


Figure 3

PWL

The general form of piecewise linear function (PWL) is

V…	N+	N-	PWL(T1 V1 T2 V2 T3 V3 ……)
I …	N+	N-	PWL(T1 V1 T2 V2 T3 V3 ……)

Figure 4 shows a piecewise linear form which illustrates the parameters mentioned above.

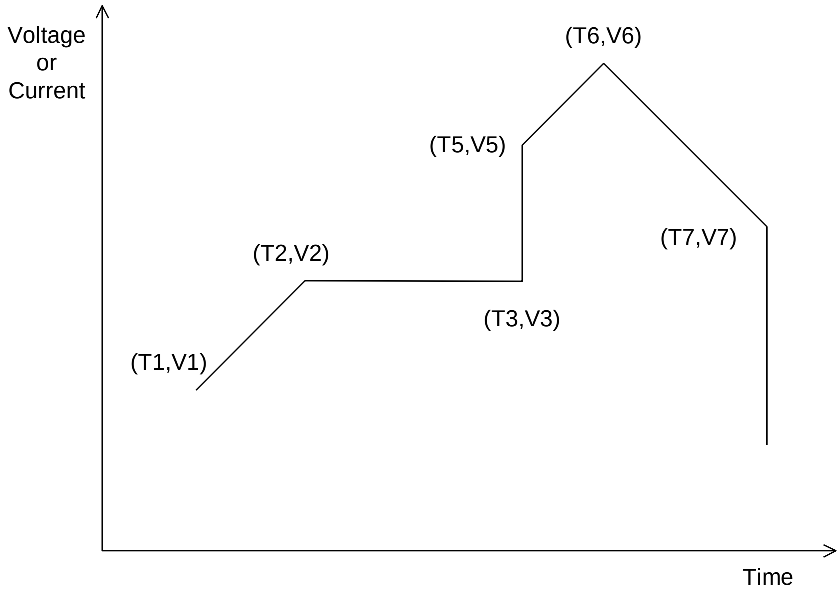


Figure 4

PSPICE Probe:

Probe in PSPICE 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 PSPICE Probe. The available operators and functions are as follows:

Function	**Actual Meaning**	**Example**
+	Addition of voltage or current	V(3)+V(5)
-	Subtraction of voltage or current	I(VSEN1)-I(VSEN2)
*	Multiplication of voltage or current	V(1)*I(VSEN1)
/	Division of voltage or current	V(10)/V(1)
ABS(X)	X-Absolute value of X	ABS(V(10))
SQRT(X)	X-Square root of X	SQRT(I(VSEN))
EXP(X)		EXP(V(5))
LOG(X)	ln(X) → log base e	LOG(V(1))
LOG10(X)	Log_{10}(X) → log base 10	LOG10(V(10))
DB(X)	20*log_{10}(X)	VDB(6)
PWR(X,Y)	X^Y → X to the power of Y	PWR(V(5),2)
SIN(X)	Sin(X) → X in radians	SIN(V(1))
COS(X)	Cos(X)	COS(V(1))
TAN(X)	Tan(X)	TAN(V(1))
ATAN(X)	Atan(X)	ATAN(V(1))
D(X)	Derivative of X with respect to the x-axis variable d(V(3))	
S(X)	Integral of X over the x-axis variable	S(I(VSEN1))
AVG(X)	Running average of X	AVG(V(5))
RMS(X)	Running RMS average of X	RMS(I(VSEN10))

The general form of Probe is **.PROBE**, in this case all variables are available in the data file. Another form is **.PROBE V(1) V(4) V(7)**, in this case only voltage at nodes 1, 4 and 7 will be stored in the data file.

WRITING AND RUNING THE PROGRAM:

1. **Create an input file** (source file) **or Circuit description file** for PSpice.

You can run PSpice by going to **programs→PSpice Student→PSpice 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 PSpice recognize it as a valid circuit description file.

2. **Run the program**

Once you are in PSpice, 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.

3. **Look at the output file and print the results**

The output file always generated by PSpice is the text file that has the file type “OUT”. Let’s say you submit a data file to PSpice 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.

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 5 is:

```
PWRSUP.CIR
VAC       10 0 SIN(0 100 50)
D1        10 11 MODA
.MODEL MODA D
CF        12 0 40U
RL        12 0 1K
RS        11 12 2
.TRAN     1M 40M
.PLOT TRAN V(12)
```

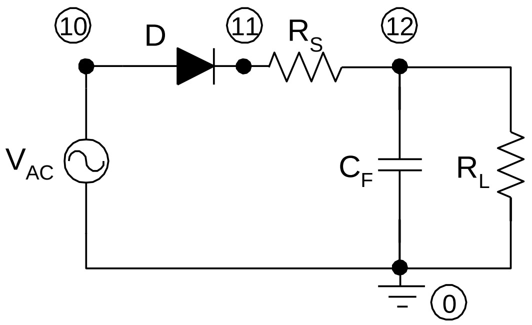


Figure 5

```
.PROBE V(12)
.END
```

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

```
DIFF.CIR
RS1        1 2 500
RS2        6 0 500
VIN        1 0 AC 50M
RC1        11 3 10K
RC2        11 5 10K
VCC        11 0 15
VEE        9 0 -15
Q1         3 2 4 MOD1
Q2         5 6 4 MOD1
Q3         4 7 8 MOD1
R1         7 0 82K
R3         7 9 82K
R2         8 9 8.2K
.MODEL MOD1 NPN
.TF V(3) VIN
.END
```

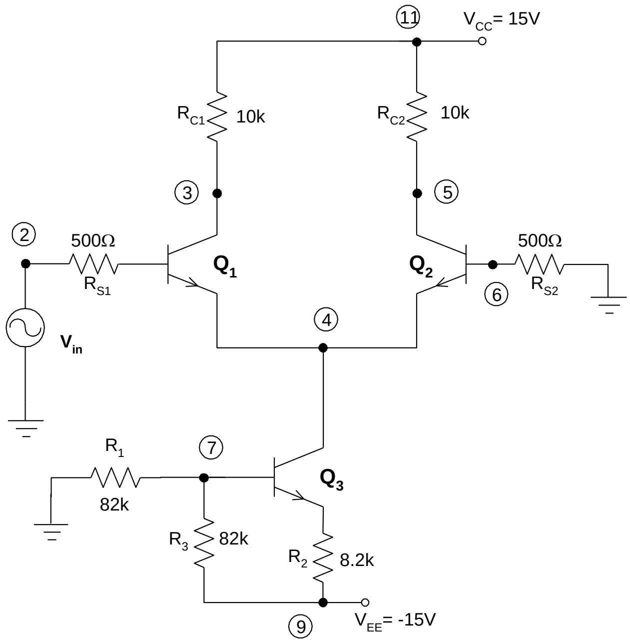


Figure 6

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

```
TRANRC.CIR
VIN 5 0 PULSE(0 4 10N 2N 2N 20N 48N)
R 5 8 1K
C 8 0 400P
.TRAN 3N 105N
.PLOT TRAN V(8)
.PROBE
.END
```

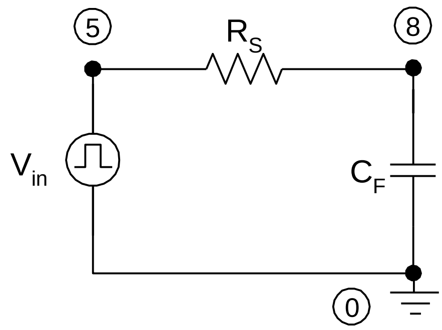


Figure 7

.SUBCKT Line

Another important command is the SUBCKT statement.

The general form is

.subckt *subname n1, n2 …*] [*param1=val1 param2=val2 …*]

Example:

.subckt opamp 1 2 3 4

.ends

A subcircuit definition begins with a *.subckt* line. The parameter *subnam* is the subcircuit name, and *n1, n2 ...* are the external nodes, which cannot be zero. The group of elements which immediately follows the *.subckt* line defines the subcircuit. The last line in a subcircuit definition is the *.ends* line.

Subcircuit Calls

The general form is

xname n1 [*n2 n3* …] *subnam* [*param1=val1 param2=val2* …]
Example:
x1 5 6 7 8 opamp

Subcircuits are used in *pspice* by specifying pseudo-elements beginning with the letter x, followed by the circuit nodes to be used in expanding the subcircuit.

Note: At this stage you are requested to write and run the above three programs, obtain the results from SPICE simulation. Try to compare your results with your hand calculations using the approximate analysis techniques of EE 203.

We also encourage simulating and comparing with hand calculations for circuits of your own; try some of the circuits you studied in EE 203.

Tutorial #2: Transistor Models for PSPICE

Objective:

The objective of this experiment is to familiarize the students with the concept of net listing in PSPICE

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

Introduction:

Separate SPICE models are used for the BJT, JFET, MOSFET and diode. These models are generally complex. For example, the BJT model can include the ohmic base resistance r_b and the current dependent collector resistance r_o. The MOSFET models can include the effects of charge controlled capacitances, short channel effects to the degree they are understood, and the channel length variations as a function of terminal voltages. All these information, and many others, can be included in the model statement used for describing semiconductor devices. 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

DXXX NA NC MNAME [AREA]

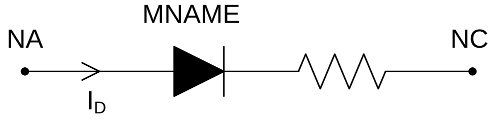


Figure 1

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. The default value for the junction cross-sectional area is 1. 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

QXXX NC NB NE MNAME [AREA]

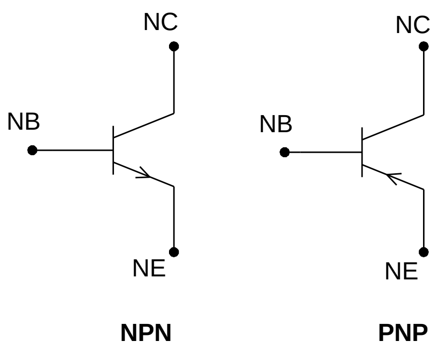


Figure 2

The associated model statements is for npn

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

and for pnp

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

The default value of the base cross-sectional AREA is 1. 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

MXXX ND NG NS NB MNAME [W=VALW][L=VALL][AD=VALD][AS=VALS]

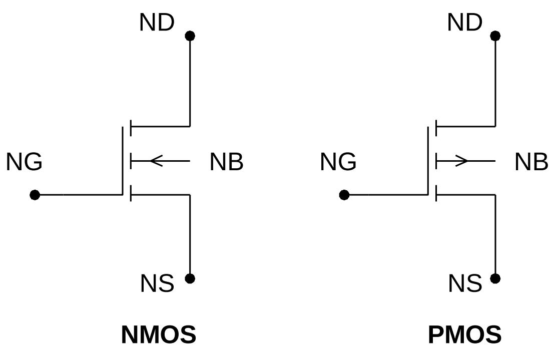


Figure 3

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…]

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. The drain and source areas are VALD and VALS respectively, and the default values are $10^{-6}cm^2$. Detailed model parameters are provided in Table 3.

Examples:

1. The input file of the circuit of Figure 4 is:

```
Figure 4 spice file
vsig 1 0 ac 1 sin(0 5m 100k)
VDD 6 0 DC 5V
Rsig 1 2 1K
C1 2 3 0.15uF
.op
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)
.tran 0.01u 20u
.print tran v(7)
.END
```

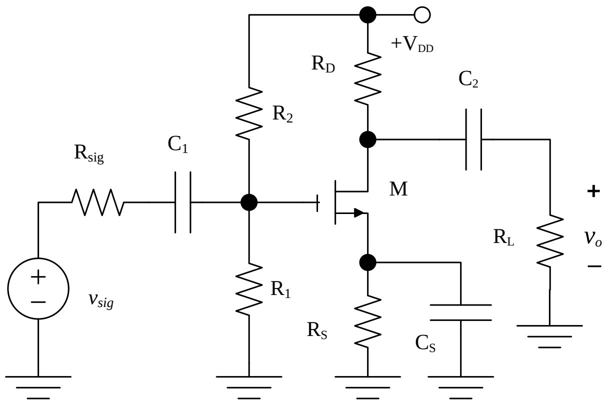


Figure 4

2. The input file of the circuit of Figure 5 is:

```
Circuit 5
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
```

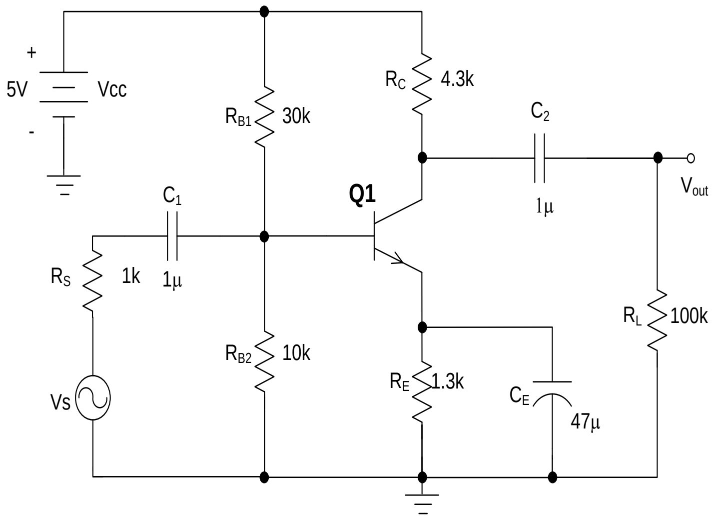


Figure 5

Assignment:

A. Consider the BJT amplifier circuit shown in Figure 6 and perform the following:

1. Hand calculation to find the medium frequency gain and the upper and lower 3dB points. Assume v_s = 10mV, β_F = 100, C_π = 1pF, C_μ = 3pF.
2. 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.
3. Repeat step 2 using β_F = 100, $I_S = 10^{-15}$, r_b = 100Ω, V_A = 150V, Cπ = 1pF, C_μ = 3pF and τ_F = 0.2ns.
4. Compare between the results obtained using hand calculations and SPICE simulations.
5. Comment on your results.

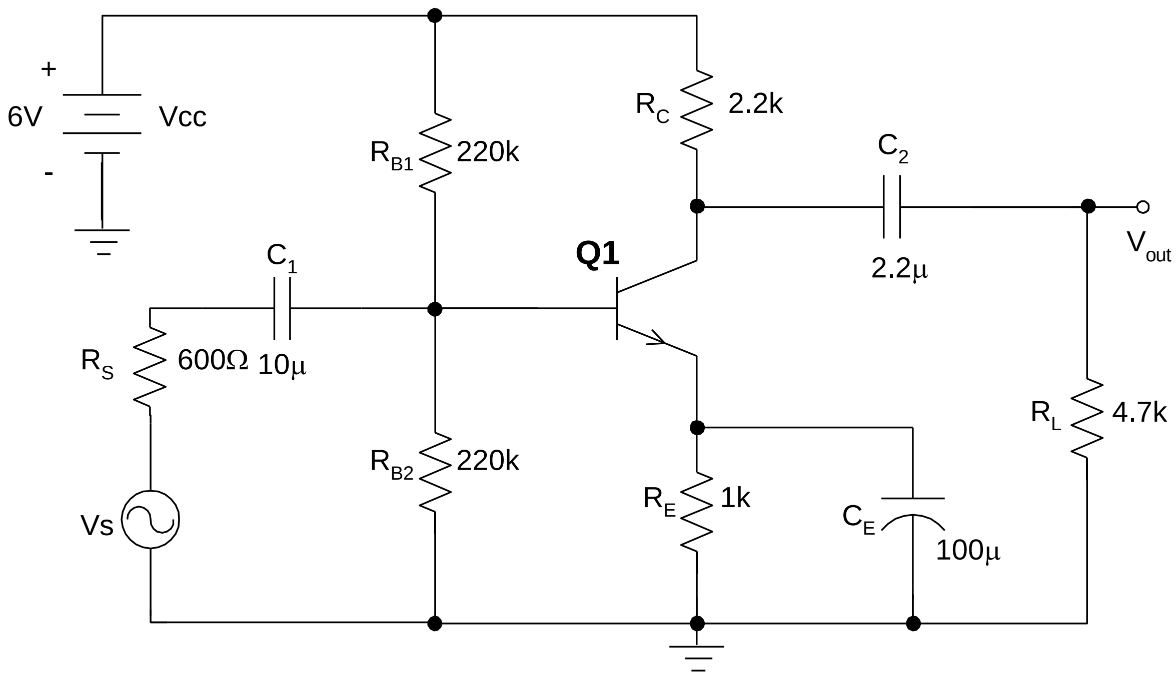


Figure 6

B. Consider the NMOS amplifier shown in Figure 7.
   - Use hand calculations to find the DC operating point and voltage gain.
   - Use Pspice default model and simulate the circuit.
   - Use the following NMOS parameters and simulate the circuit again.

For NMOS:
.MODEL MN NMOS LEVEL=2 LD=0.15U TOX=200.0E-10
+ MSUB=5.36726E+15 VTO=0.743469 KP=8.00059E-05 GAMMA=0.543
+ PHI=0.6 U0=655.881 UEXP=0.157282 UCRIT=31443.8
+ DELTA=2.39824 VMAX=55260.9 XJ=0.25U LAMBDA=0.0367072
+ NFS=1E+12 NEFF=1.001 NSS=1E+11 TPG=1.0 RSH=70.00
+ CGDO=4.3E-10 CGSO=4.3E-10 CJ=0.0003 MJ=0.6585
+ CJSW=8.0E-10 MJSW=0.2402 PB=0.58

For PMOS:
.MODEL MP PMOS LEVEL=2 LD=0.15U TOX=200.0E-10
+ NSUB=4.3318E+15 VTO=-0.738861 KP=2.70E-05 GAMMA=0.58
+ PHI=0.6 U0=261.977 UEXP=0.323932 UCRIT=65719.8
+ DELTA=1.79192 VMAX=25694 XJ=0.25U LAMBDA=0.0612279
+ NFS=1E+12 NEFF=1.001 NSS=1E+11 TPG=-1.0 RSH=120.6
+ CGDO=4.3E-10 CGSO=4.3E-10 CJ=0.0005 MJ=0.5052
+ CJSW=1.349E-10 MJSW=0.2417 PB=0.64

- Comment on your results.

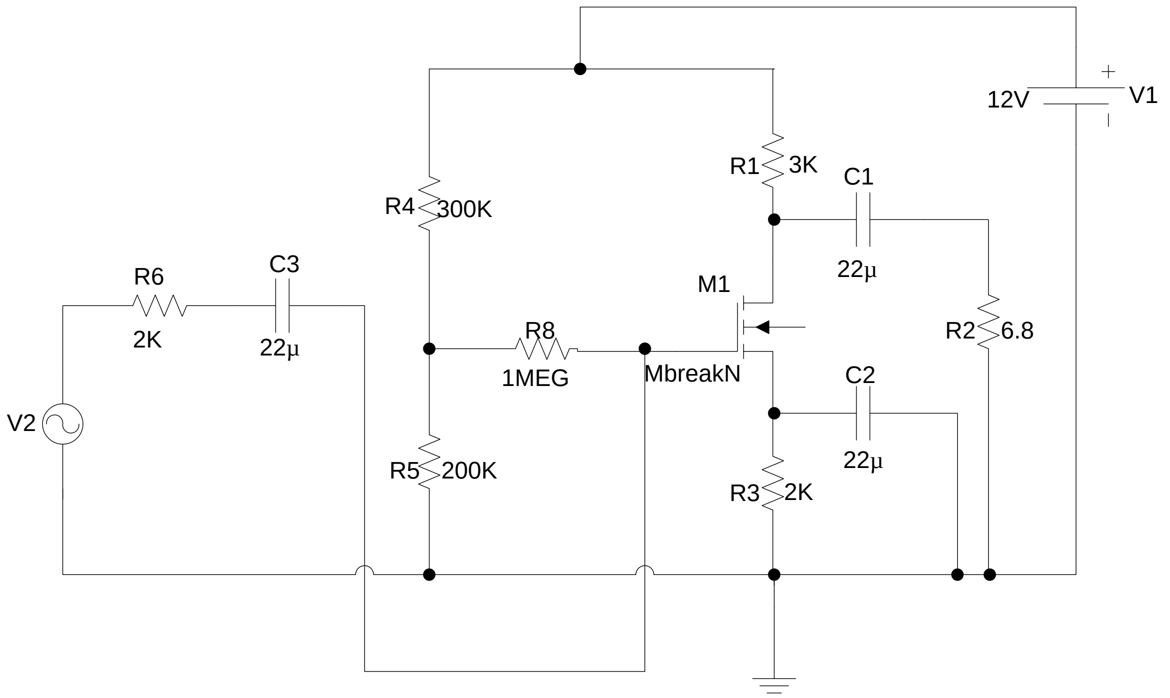


Figure 7

TABLE 1: DETAILED DIODE MODEL PARAMETERS

D Diode

D<name><+node><-node><model name>[area]

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

Q Bipolar Transistor

Q<name><collector node><base node><emitter node><[substrate node]
<model name>[area value]

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

M MOSFET

M<name><drain node><gate node><source node><bulk/substrate node>
<model name>[L=<value>][W=<value>][AD=<value>][AS=<value>]

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	$volt^{l/2}$
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	

Experiment #1: Gain Frequency Characteristics of Single Transistor Amplifiers

Objective:

To study the effects of coupling capacitors on the gain and frequency response of single transistor amplifiers.

Prelab work:

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

1. For the circuit shown in Figure 1 perform a complete small signal ac analysis and obtain the MF gain, the LF poles, the HF poles and bandwidth of this amplifier. Also find the small signal input and output resistances.

2. From the results obtained in step 1, try to deduce the effect of introducing a 50μF capacitor in parallel with R_E. Specifically, what is the effect of such a capacitor on the MF gain and the bandwidth?

3. Using SPICE simulate your circuit with and without the capacitor C_E and try to deduce from the SPICE output file, the MF gain, the LF poles, the HF poles (corner frequencies) and the bandwidth. You can also obtain the input resistance and the output resistance using SPICE. For the SPICE analysis use the frequency range 100Hz to 8MHz. Use β=100, C_μ=C_{bc}=8pF and C_π=C_{be}=30pF.

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

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

Experimental work:

1. Construct the circuit shown in Figure 1 without the capacitor C_E. Apply a small ac signal v_s and make sure by monitoring the oscilloscope that the output voltage is not distorted. Change the input frequency from 100Hz to 3MHz. At each frequency measure the small signal voltage gain and plot it on the same graph supplied by SPICE output file.

2. Calculate the MF range, LF poles, HF poles and bandwidth from your measured gain-frequency characteristic.

3. Insert your experimental results into Table I.

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

5. Repeat steps 1-4 after connecting the capacitor C_E.

6. Comment on your results.

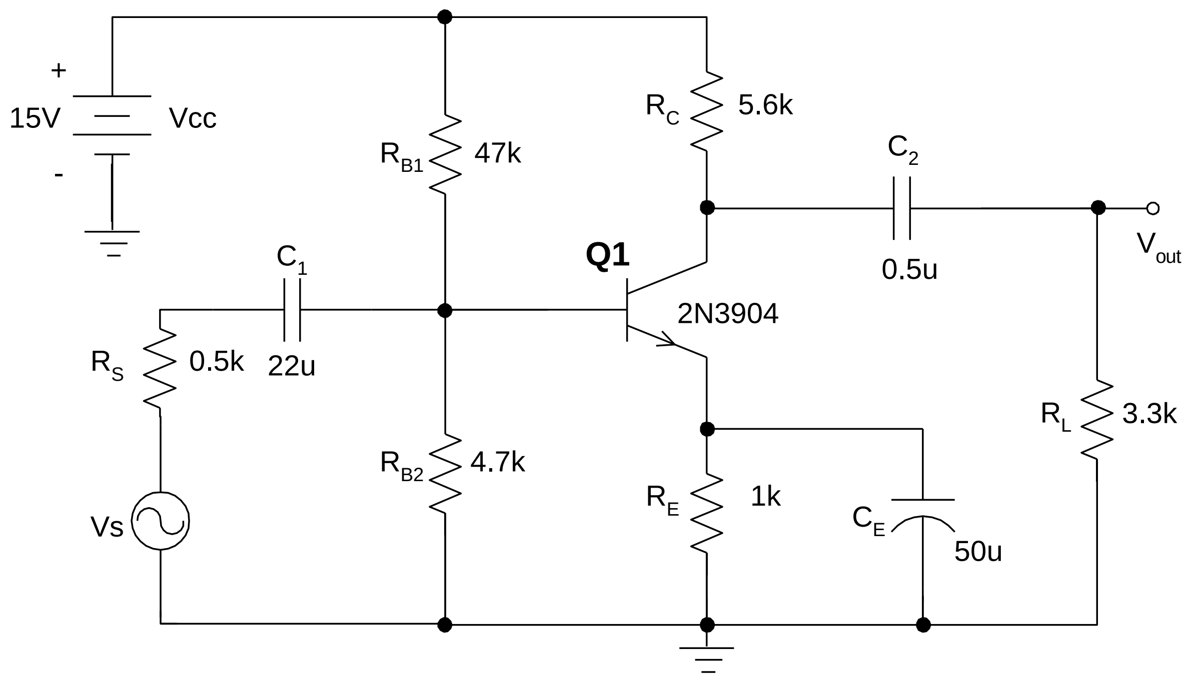


Figure 1

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

Parameter	**Hand Calculation**	**SPICE Simulation**	**Experimental Result**
MF Gain			
LF Poles (Corner Frequencies			
HF Poles (Corner Frequencies)			
Bandwidth			

Experiment #2: Gain Frequency Characteristics of Multistage Transistor Amplifiers

Objective:

To study the effects of coupling and junction capacitors on the gain and frequency response of multistage transistor amplifiers.

Prelab:

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

1. For the two stage amplifier circuit shown in Figure 1 perform a complete small signal ac analysis and obtain the MF gain, the LF poles, the HF poles (corner frequencies) and bandwidth of this multistage amplifier. Also find the small signal input and output resistances.

2. Using SPICE simulate your circuit and try to deduce from the SPICE output file, the MF gain, the LF poles, the HF poles and the bandwidth. Also obtain the input resistance and the output resistance using SPICE. For the SPICE analysis use the frequency range 100Hz to 8MHz. Use β=100, C_μ=C_{bc}=8pF and C_π=C_{be}=30pF.

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

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

Experimental Work:

1. Construct the circuit shown in Figure 1. Apply a small ac signal v_s and make sure by monitoring the output on oscilloscope that the output voltage is not distorted. Change the input frequency from 100Hz to 3MHz. At each frequency measure the small signal voltage gain and plot it on the same graph supplied by SPICE output file.

2. Calculate the MF range, LF poles, HF poles and bandwidth from your measured gain-frequency characteristic.

3. Insert your experimental results into Table I.
4. Compare your hand calculations, SPICE simulations and experimental measurements.

5. Comment on your results.

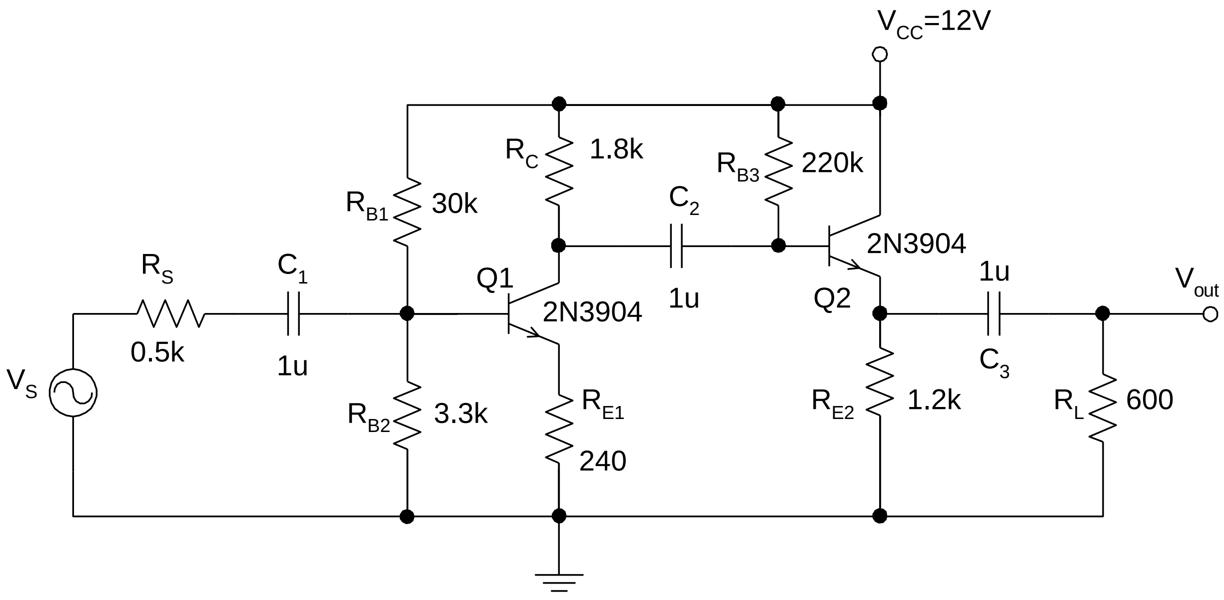


Figure 1

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

Parameter	Hand Calculation	SPICE Simulation	Experimental Result
MF Gain			
LF Poles (Corner Frequencies			
HF Poles (Corner Frequencies)			
Bandwidth			

Experiment #3: Linear Applications of Operational Amplifiers

Objective:

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

Prelab:

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

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

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

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

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

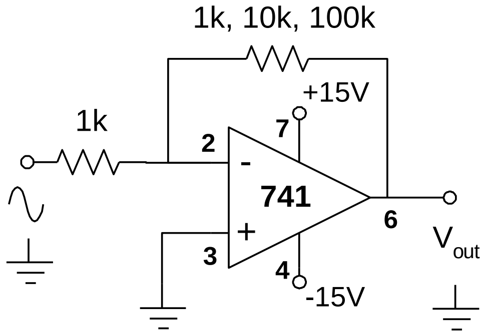
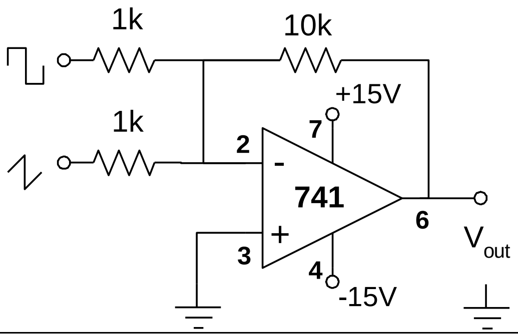
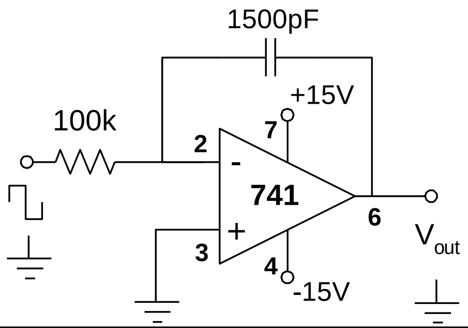
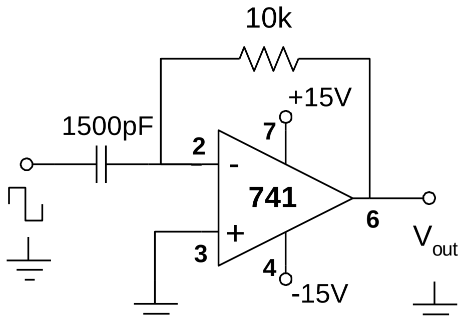
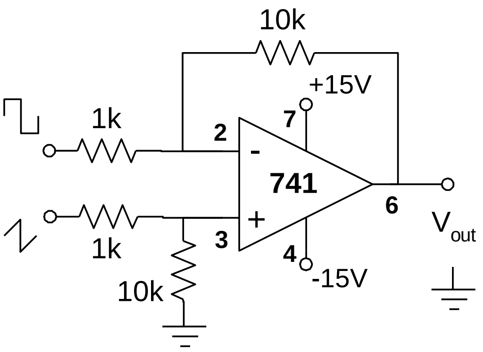
CIRCUIT	Hand Calculation	SPICE Simulation	Experimental Result
			
			
			
			
			

Figure 1

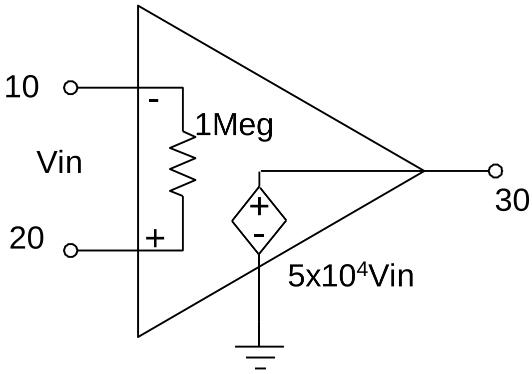


Figure 2

<u>Example</u>

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

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

X……. NA NB NC … SUBNAME

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

XOP1 2 3 6 OPAMP

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

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

Experimental Work:

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

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

3. Comment on your results.

Experiment #4: Determination of Operational Amplifier Characteristics

It is well known that the characteristics of commercially available operational amplifiers are different from the ideal characteristics. Although it is possible to use some of these non ideal characteristics to advantage; for example the finite bandwidth and the finite gain characteristic can be used to construct capacitor less filters and oscillators, in general the non ideal characteristics of the operational amplifiers may degrade the circuit performance. Therefore, 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. It is, therefore, important to know how to measure the operational amplifier characteristics using simple equipments available in any laboratory.

1. Measurement of Open Loop Gain

Direct measurement of the open loop gain is not feasible because of the large values involved. Instead, measurement of open loop gain can be carried out with the operational amplifier embedded in a negative feedback circuit. Such an arrangement is shown in Figure 1. Obtain an expression for the output voltage in terms of the voltage V_{in}, an the voltage V_x. If we select

$$10R_1 = R_2 = R_F = 10k\Omega,\ R_3 = 10\Omega$$

then is is easy to show that for large values of operational amplifier gain, the overall gain: that is with the feedback loop closed will be ≈ -10. This value is not important in itself; its significance is to assure us that there is sufficient negative feedback so that reasonable values of V_{in} can be used without driving V_{out} to saturation levels. What is important is the simple relation between V_{out} and V_x; obtain this relation. Clearly, it is a simple matter to measure V_x and V_{out} and hence to calculate the gain of the operational amplifier.

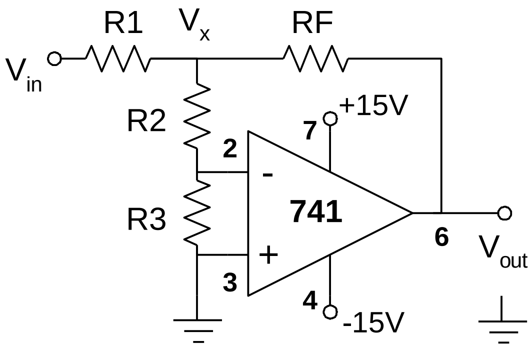


Figure 1

Some additional caution has to be exercised in measuring the operational amplifier gain using this method. This is because the open loop break frequency of the op-amp is as low as few Hz. Therefore, to measure the op-amp gain correctly we must choose a frequency that is lower than the break frequency of the op-amp. A good way to assure that we have selected the right frequency is to display both V_{out} and V_x on the oscilloscope in a Lissajou pattern. At frequencies above the break frequency we will obtain an ellipse. Why? Then by reducing the frequency until the ellipse is converted to a straight line then we can assume that we are using the right frequency and consequently the measured value of the op-amp gain can be considered correct. Explain this. You may face some difficulties in deciding whether you are seeing a straight line or not on the oscilloscope. Why? Anyway, try to avoid this.

2. Measurement of open loop break frequency

Consider the circuit shown in Figure 2. We know that at relatively high frequencies (w.r.t. the open loop break frequency), the gain of the op amp can be expressed by

$$A = \frac{A_o}{1 + jw / w_o}$$

At the frequency w_t corresponding to unity gain, it is easy to show that

$$A_o = w_t / w_o$$

The above equation can be easily proved since we know that $w_t >> w_o$. Therefore the gain of the op-amp can be expressed by

$$A = \frac{A_o}{1 + jA_o w / w_t}$$

It is easy to show that when $R_1 = R_2$ the gain V_{out}/V_{in} will be

$$A_{VF} = \frac{-1}{1 + 2/A}$$

Substituting the value of A and since A_o is very large it is easy to show that

$$A_{VF} = \frac{-1}{1 + 2w / w_t}$$

From the last equation it is obvious that the gain will drop to $1/\sqrt{2}$ when $w_m = w_t/2$. We can measure w_m. Since we know the open loop voltage gain A_o, then it is easy to calculate w_o.

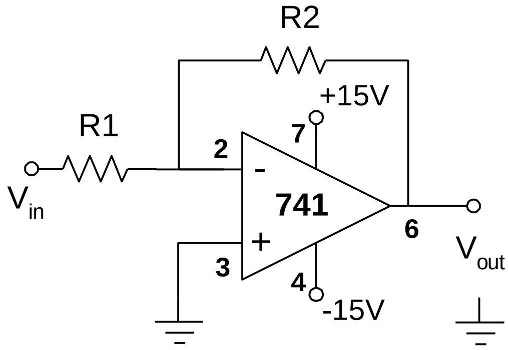
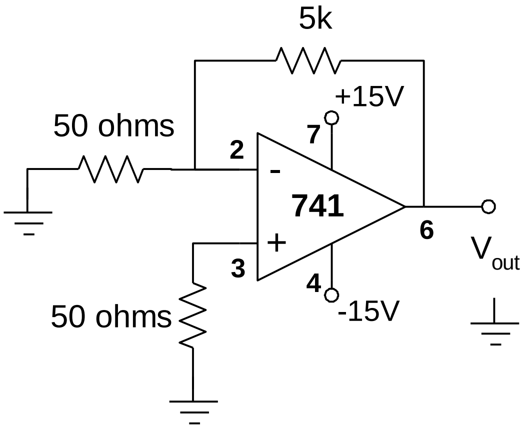


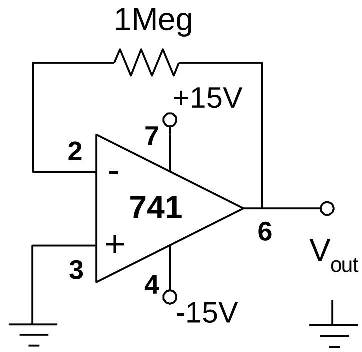
Figure 2

3. Input offset voltage, Bias current, Offset current

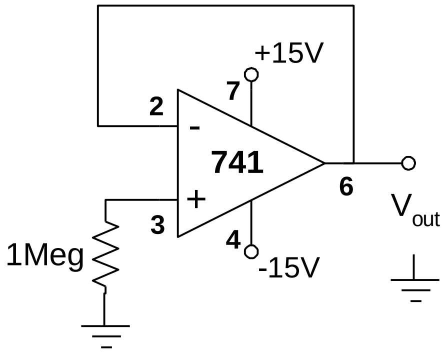
In order to measure offset voltage (Vos), I_1 and I_2 of general purpose op-amps using inexpensive methods, the circuits shown in Figure 3 are proposed. Verify the usefulness of these circuits by obtaining expressions for the output voltage for each circuit. Show how the offset parameters can be deduced from the measurement of the output voltages.



(a) Circuit to measure offset voltage; effect of I_{OS} is minimised



(b) Circuit to measure bias current in inverting terminal. Donot neglect V_{OS}



(c) Circuit to measure bias current in non-inverting terminal. Donot neglet V_{OS}.

Figure 3

4. <u>Slew rate and full power bandwidth</u>

From the discussion of section 2, we found that $w_t = w_o A_o$. Therefore, if we consider the circuit of Figure 2, its gain can be expressed as

$$\frac{V_{out}}{V_{in}} = \frac{-R_2/R_1}{1+(1+R_2/R_1)/A}$$

Therefore, substituting for $A = w_t/s$ the gain can be expressed as

$$\frac{V_{out}}{V_{in}} = \frac{-R_2/R_1}{1+(1+R_2/R_1)s/w_t}$$

which corresponds to an amplifier with dc gain of $-R_2/R_1$ and a 3dB corner frequency of $w_t/(1+R_2/R_1)$. Therefore, if we measure the frequency response of a closed loop amplifier with a gain of, say, 10, the 3dB frequency of $w_t/11$ would be achieved. This is true only if the output voltage is quite small (less than a volt). On the other hand, op-amps are capable of providing output signal swings that approach the voltages of the power supplies used. (Typical values are ±10V for ±15V power supplies). The large signal frequency response of op-amps is limited by the slew rate. Specifically, there is an upper limit for the rate of change of the output voltage with time. This upper limit is called slew rate. This slew rate limiting causes distortion in large signal output sine waves. Specifically, as the frequency of the sine wave is increased, its slope, which is highest at the zero crossings, increases until that slope equals the op-amp slew rate. Increasing the frequency further will obviously result in a distorted output. The op-amp data sheets usually specify the frequency at which a sine wave output, whose peak amplitude is equal to the maximum rated voltage, starts to show distortion. This frequency is called the full power bandwidth and is denoted by f_M. It is easy to show that

$$f_M = \frac{slewrate}{2\pi V_{\max}}$$

where V_{max} is the maximum specified output voltage of the op-amp.

To measure the slew rate and full power bandwidth, consider the circuit shown in Figure 4. If the input voltage is 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. It is

$$\text{Slew Rate} = V_{out}/T_{SR}$$

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. Determine this frequency. This is f_M. Verify the relationship between f_M and the slew rate.

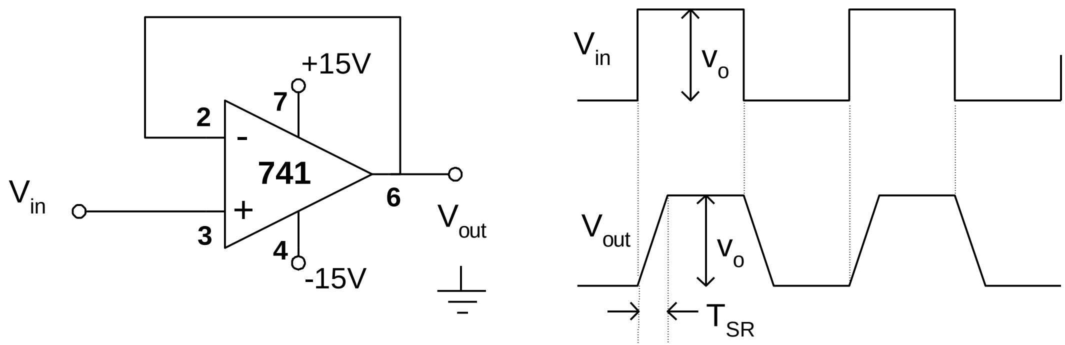


Figure 4

Table I: Typical Performance of Operational Amplifiers

	741	**LM118**	**AD507**
Input offset voltage (mV)	≤ 5	≤ 4	≤ 5
Bias current (nA)	≤ 500	≤ 250	≤ 15
Offset current (nA)	≤ 200	≤ 50	≤ 15
Open loop gain (dB)	106	100	100
CMRR (dB)	80	90	100
Input resistance (MΩ)	2	5	300
Slew rate (V/µs)	0.5	≥ 50	35
Unity gain bandwidth (MHz)	1	15	35
Full power bandwidth (kHz)	10	1000	600

Experiment #5: Active Filters

Objective:

To measure the transfer functions of several active filters and to determine their corner frequencies, or center frequency and to determine their roll-off rates from the frequency responses.

Prelab:

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

1. For the different active-filter configurations shown in Figure 1 and 2 perform an approximate hand calculation assuming that the operational amplifiers are ideal. In each case sketch the expected transfer function and calculate the corner frequency, or the center frequency, the gain of the filter configuration and its pass band, or bandwidth.

2. Using SPICE, simulate the different configurations and from SPICE output file calculate the corner frequency, or the center frequency, the gain of the filter configuration and its pass band, or bandwidth. The second model for simulation op-amps is 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. Figure 3 also shows an example of how to call the op-amp SUBCIRCUIT. Try to use the SUBCIRCUIT concept in your simulation.
   For the SPICE simulations take the input voltages of the order of 1V amplitude and obtain the frequency range from your hand calculations in step 1.

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

Experimental Work:

1. Construct the circuits shown in Figure 1 and 2. Apply sinusoidal input voltage with constant amplitude, of the order 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. Measure the ratio between output and input voltages and plot your results on the same sheet obtained from SPICE output.
2. From your measurements obtain the corner frequency, or the center frequency, the gain of the filter in the pass band and its band, or bandwidth.

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

4. Comment on your results.

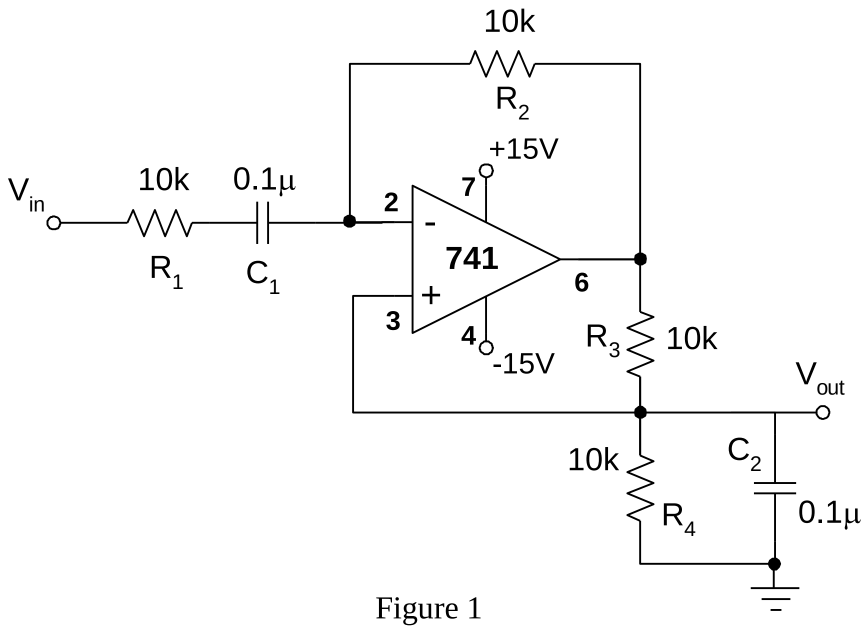


Figure 1

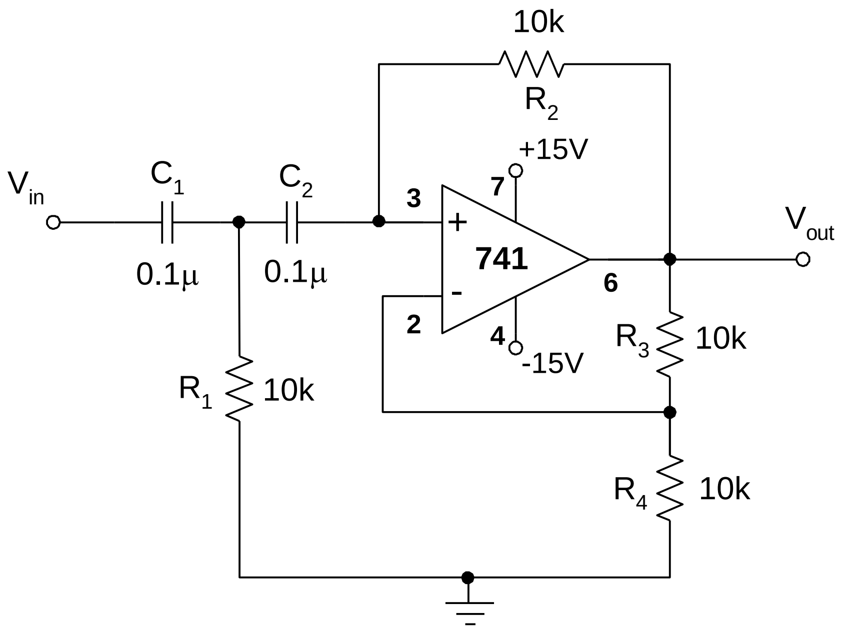
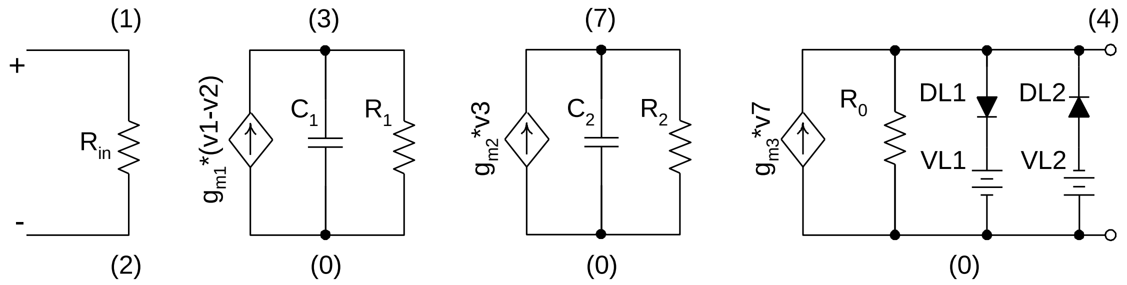


Figure 2




```
.SUBCKT OPAMP     1    2    4
RIN    1      2      400K
GM1    3      0      1      2      200m
R1     3      0      1MEG
C1     3      0      0.0159uF
GM2    0      7      3      0      1m
R2     7      0      1K
C2     7      0      39.8pF
GM3    0      4      7      0      5m
R0     4      0      200K
DL1    4      5      DIODE
DL2    6      4      DIODE
VL1    5      0      DC     13V
VL2    0      6      DC     13V
.MODEL        DIODE         D
.ENDS
```

A SUBCKT call for an op-amp connected between points 11, 17, 30 where 11 is the positive input, 17 is the negative input and 30 is the output is:

```
x…..   11     17     30     OPAMP
```

Figure 3. SPICE model for operational amplifier

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

Circuit		Hand Calculation	SPICE Simulation	Experimental Result
Figure 1	MF Gain			
	Center Frequency			
	Bandwidth			
Figure 2	MF Gain			
	Center Frequency			
	Bandwidth			

Experiment #6: Feedback and Non-linear Distortion

Objective:

The major objective of this experiment is to illustrate how negative feedback can reduce nonlinear distortion.

Introduction:

Consider the operational amplifier circuit shown in Figure 1. Because the output resistance of the op-amp is about 50Ω then it is expected that the power delivered to the loudspeaker will not be the maximum power. Thus if you listen to the output from the speaker using an input of around 0.5V p-p then you will not hear a loud sound. A possible solution for this problem is to connect an output stage between the op-amp output and the speaker, as shown in Figure 2. If you connect such a circuit you will notice that the sound is now more loud. However, it will be distorted. There are two sources for this distortion. The first as you know is the crossover distortion is to connect the two diodes D1 and D2 as shown in Figure 3. Now it is expected that the crossover distortion will disappear. Unfortunately it will not disappear completely unless the two transistors are identical and also the two diodes are identical and that all the transistors and diodes easily satisfied if you are using discrete elements. So what is the solution? The solution is simply apply negative feedback across the whole circuit as shown in Figure 4.

Prelab work:

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

1. Using simple hand calculation try to sketch the voltage across the speaker in all cases.

2. Using SPICE simulate the circuits of Figures 1 to 4 and in each case obtain an output file including the output voltage across the speaker. Assume default values for the transistors and diodes. Also assume that the input voltage is a pure sine wave with frequency 1kHz and amplitude around 0.25V.

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

Experimental work:

1. Construct the circuit in Figure 1 to 4 one by one and in each observe the output on the oscilloscope and listen to it from the speaker. Sketch your output.

2. Compare the calculated, simulated and measured results.

3. Now rather than obtaining the input from a function generator, obtain it from the output of a microphone. You may need to change the 10k resistance and make it 1k, this because the output of the microphone is usually small so we need more amplification from the op-amp. Observe the output on the oscilloscope and listen to it, in the four cases of Figure 1 to 4. In each case try to sense how clear and loud your output is. Comment on your results.

4. Try to write a conclusion to illustrate the usefulness of negative feedback as a powerful mean for reducing nonlinear distortion. Are there any advantages for negative feedback?

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

	Output Waveforms			**Sound Level Hi or Lo**
	Hand Calculation	**SPICE**	**Experiment**	
Figure 1				
Figure 2				
Figure 3				
Figure 4				

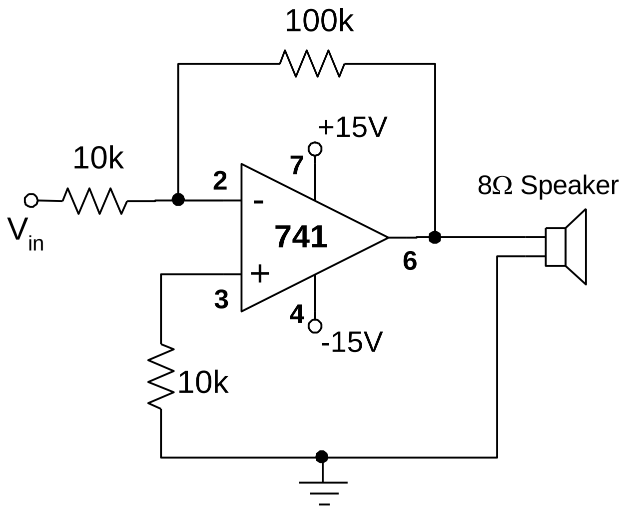


Figure 1

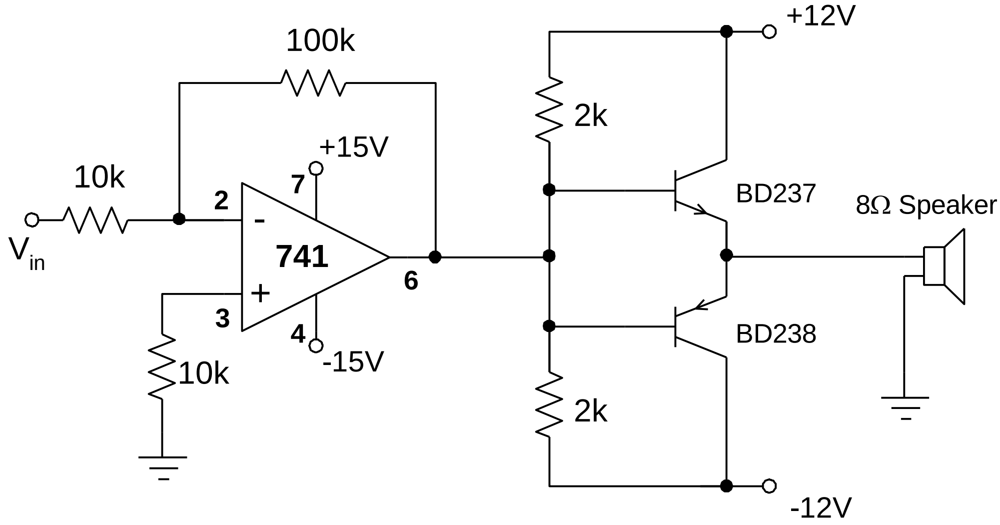


Figure 2

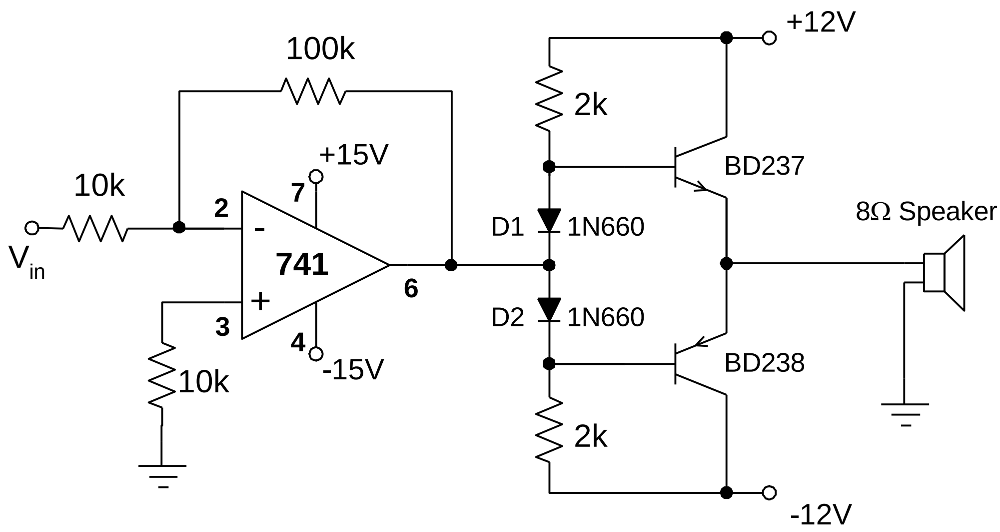


Figure 3

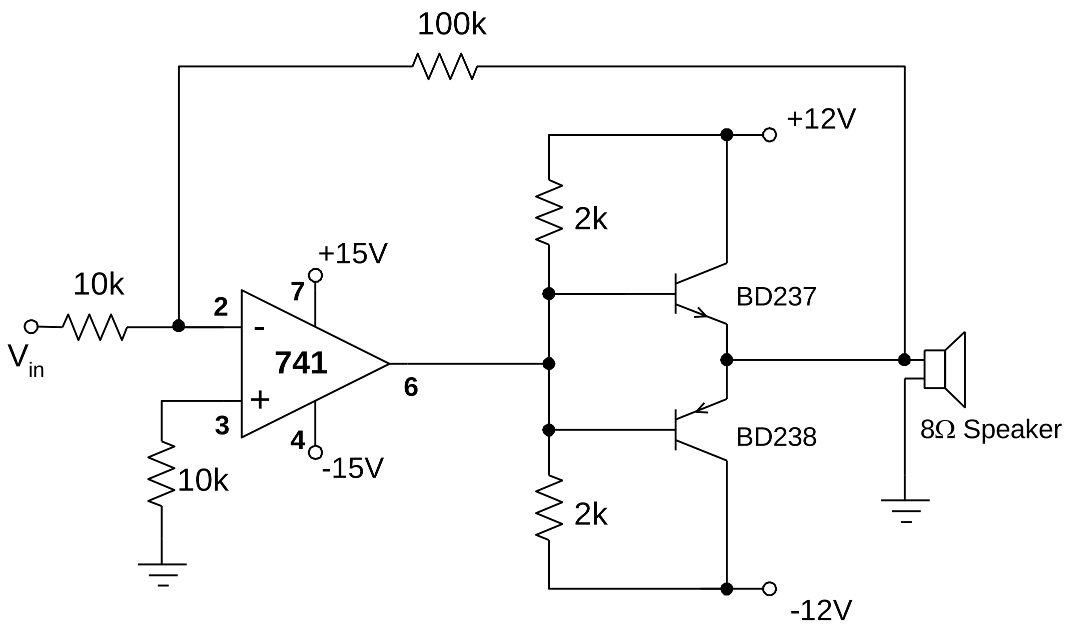


Figure 4

Experiment #7: Feedback Amplifiers

Objective:

To study the properties of negative feedback amplifiers.

Prelab work:

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

1. For the circuit shown in Figure 1, use the feedback techniques to perform a complete ac small signal analysis and obtain the MF gain, the LF poles, the HF poles, the bandwidth, and the input resistance, the output resistance of this amplifier with and without feedback.

2. Using SPICE simulate your circuit and from SPICE output file calculate the parameters of the amplifier obtained in step 1. For the SPICE analysis use the frequency range 100Hz to 8MHz. Use β=100, C_μ=C_{bc}=8pF and C_π=C_{be}=30pF.

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

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

Experimental work:

1. Construct the circuit shown in Figure 1. Apply a small ac signal v_s and make sure by monitoring the oscilloscope that the output voltage is not distorted. Change the input frequency from 100Hz to 3MHz. At each frequency measure the small signal voltage gain and plot it on the same graph supplied by SPICE output file.

2. Calculate the MF range, LF poles, HF poles and bandwidth from your measured gain-frequency characteristic.

3. Measure R_{in} and R_{out} at medium frequency and tabulate the results of steps 1, 2 and 3 in Table I.

4. Now remove the resistor R_F and capacitor C_F and repeat steps 1, 2 and 3.
5. Compare your hand calculations, SPICE simulations and experimental measurements.

6. Comment on your results.

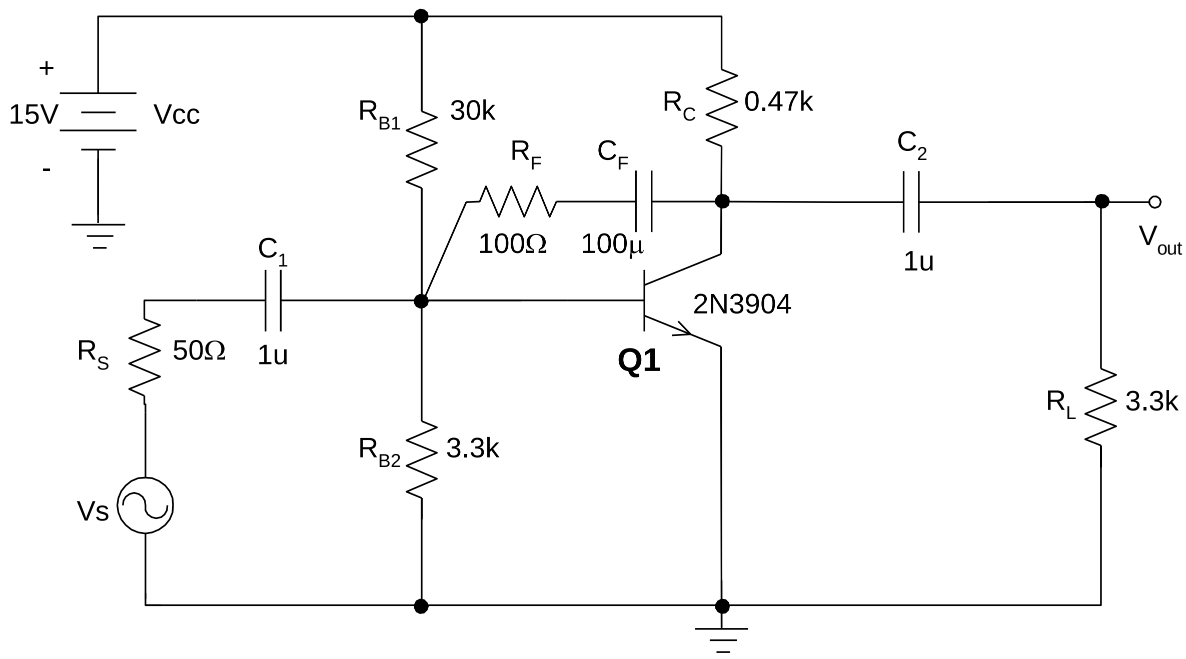


Figure 1

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

	With Feedback			Without Feedback		
	Hand Calculation	**SPICE**	**Experiment**	**Hand Calculation**	**SPICE**	**Experiment**
MF Gain						
Bandwidth						
R_{in}						
R_{out}						

Experiment #8: Oscillators

Objective:

To investigate the operation of sinusoidal oscillators using operational amplifiers, specifically, the phase shift oscillator, the Wein-Bridge oscillator and the quadrature oscillator.

Prelab work:

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

1. For the different oscillator circuits shown in Figure 1, perform an approximate hand calculation assuming ideal operational amplifiers. In each case obtain an expression for the frequency of oscillation and the condition of oscillation.

2. Using SPICE simulate the different configurations and from SPICE output file obtain the oscillation frequency. For simulating the op-amp you can use the first model presented in Experiment # 3 or the second model presented in Experiment # 7. Try the second model to see the effect of the gain frequency characteristic of the op-amp on the frequency and condition of oscillation. For the SPICE simulation it is essential to calculate the closed loop gain of your circuit. As you know the closed loop gain is the overall gain of the amplifier and the feedback network. Of course in oscillator circuits, we do not have an input ac signal. However, we can open the loop at an appropriate point and assume that an input voltage of say, 1V ac is applied at this input. Then we calculate the output voltage and phase angle as functions of the input frequency. If we find that at certain frequency the overall gain is unity and the overall phase angle is zero, then this is the possible frequency of oscillation. Notice that for oscillation to start the condition of oscillation must be satisfied. Thus, do not be disappointed if you notice that for certain value of R_1 the overall gain cannot be unity and phase shift is not zero. Try again using a larger value of R_1.

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

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

Experimental work:

1. Construct the circuit shown in Figure 1. In each case change the variable resistance until you get an output on the oscilloscope. This means that your circuit is oscillating. In each case record the value of the resistance R_1 at which oscillation just starts to appear on the oscilloscope. Check whether it satisfies the condition of oscillation obtained from your theoretical work.

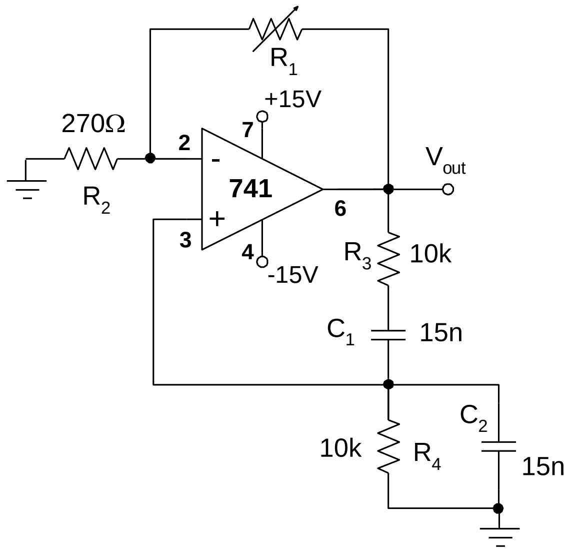
2. In each case observe the output waveform. Is it pure sinusoidal signal? If not, what are the sources of distortion in your opinion?

3. Also observe the amplitude of the output waveforms. Can we control it? If the answer is yes, how?

4. Tabulate your results in Table I.

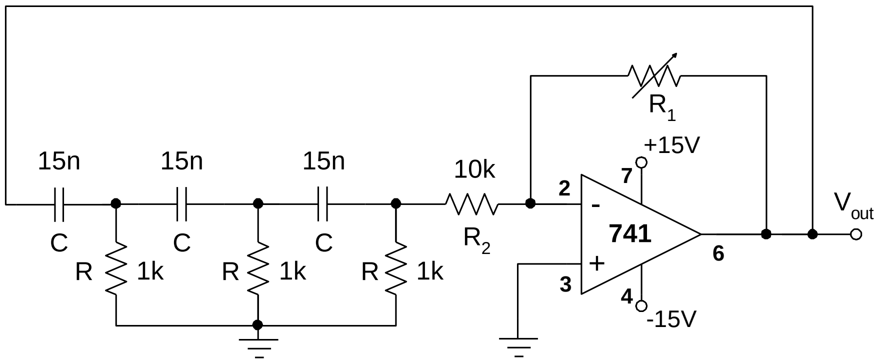
Table I: Summary of hand calculations, SPICE simulation and experiment

Circuit		**Hand Calculation**	**SPICE Simulation**	**Experimental Result**
Figure 1	Frequency			
	Condition of Oscillation			
Figure 2	Frequency			
	Condition of Oscillation			
Figure 3	Frequency			
	Condition of Oscillation			



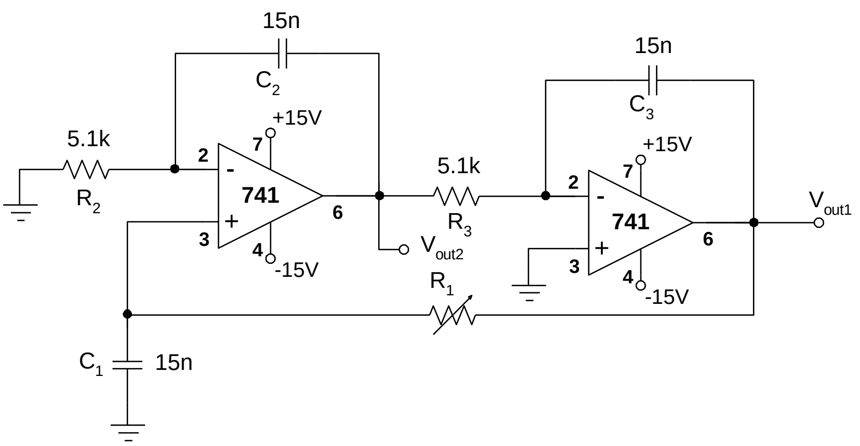
$f_o = 1/2\pi CR$
$C = 15nF$, $R = 10k\Omega$
$R_2 = 270\Omega$, $R_1 > 2R_2$

(a) Wein Bridge Oscillator



$f_o = 1/2\pi\sqrt{6}\,CR$
$C = 15nF$, $R = 1k\Omega$
$R_2 = 10k\Omega$, $R_1 > 29R_2$

(b) Phase shift Oscillator



$f_o = 1/2\pi\sqrt{C_2 C_3 R_2 R_3}$
$C_1=C_2=C_3=15nF$, $R = 1k\Omega$
$R_2=R_3=5.1k\Omega$, $R_1 > R_2$

(c) Quadrature Oscillator

Figure 1