SWECHA
Flat No. 201,Karan Center,S.D. Road,Secunderabad. 500003.

Swecha Documents	SF-SAC/ ECE / II-II/LM/2010 /ver. 1.0
LABMANAUALS	DEPARTMENT : ECE

ELECTRONIC CIRCUITS ANALYSIS LABORATORY MANUAL

ACADAMIC CHAPTER
OF
SWECHA
September- 2010

SWECHA
Flat No. 201,Karan Center,S.D. Road,Secunderabad. 500003.

INDEX

S.NO	NAME OF THE EXPERIMENT
1	Common Emitter Amplifier
2	Common Source Amplifier
3	Two Stage RC Coupled Amplifier
4	Current Shunt Feedback Amplifier
5	Cascode Amplifier
6	Colpitts Oscillato
7	RC Phase Shift Oscillator using Transistors
8	Class-A Power Amplifier(transformer less)
9	Class -B complementary symmetry Amplifier
10	Common Base(BJT)/Common Gate (JFET) Amplifier
11	Hartley Oscillator

Contributors List

1. Mr. L. Hari Venkatesh
2. Mr. A. Mahesh
3. Mr. P. Bhaskara Rao
4. Mr. T.V.S. Kishore
5. Mr. Akbar Hussain
6. Mr. Vishwanath
7. Prof. Satya Prasad Lanka
8. Dr. L. Pratap Reddy

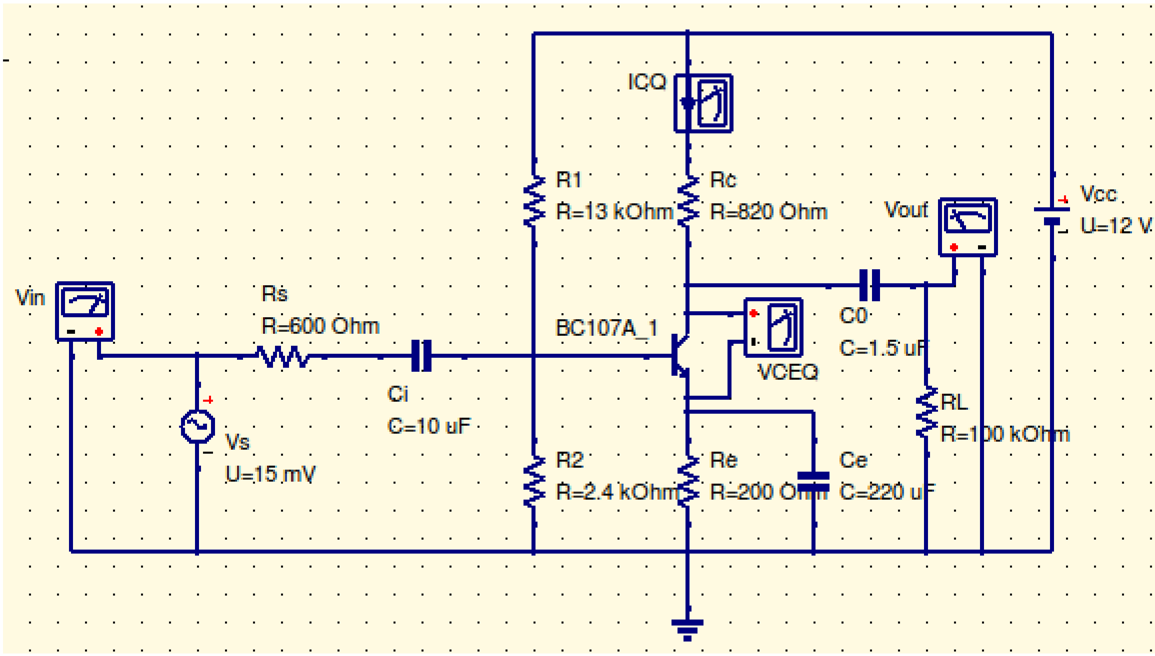
Experiment- I
Common Emitter Amplifier

Aim: To simulate the Common Emitter Amplifier and obtain the frequency response.

Design Specifications:
Voltage Gain(Av)=50, Bandwidth= 1MHz, Input Impedanc =2 kohm

Apparatus: Qucs Software

Circuit Diagram:



Design Equations:

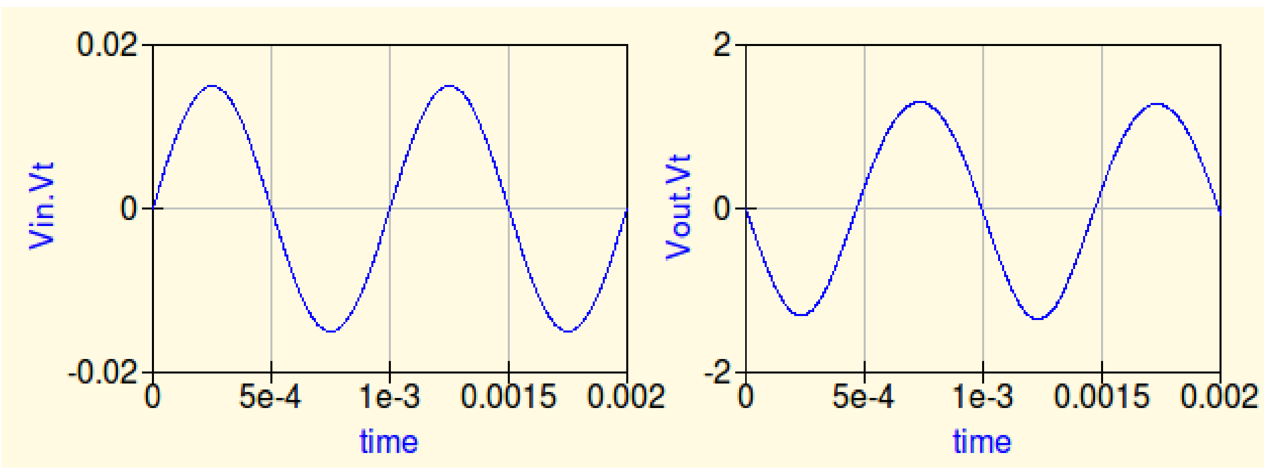
1. Select the transistor which has higher cutoff frequency of 1MHz
2. Assume V_{CC}=12V , V_{CE} =V_{CC}/2 , V_E=V_{CC}/10
3. Calculate Rc from $Av=-(h_{FE}(R_c||1/h_{oe}))/h_{ie}$, where h_{ie} , h_{oe} can be taken from the manufacturers datasheet of the transistor.
4. Calculate I_C from $V_{CC}-I_CR_C-V_{CE}-V_E=0$

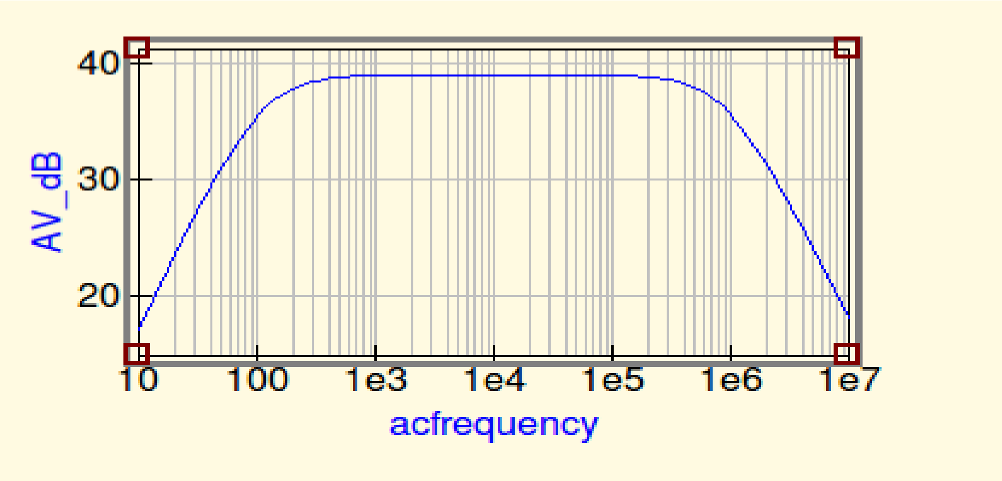
5. Assume $I_C = I_E$, Calculate R_E from $V_E = I_E R_E$
6. $S = 1 + (R_B/R_E)$, choose S=10, calculate $R_B = 9R_E$, where $R_B = R1||R2$
7. Calculate $V_B = V_{BE} + V_E$, where $V_{BE} = 0.65$ V
8. Calculate the ratio R1/R2 from $V_B = (R2 \cdot V_{CC}) / (R1 + R2)$
9. From steps 6 and 8 calculate R1, R2
10. Calculate emitter bypass capacitance (C_E) from $X_{CE} <= R_E/10$
11. Calculate input coupling capacitance (Ci)from $X_{Ci} <= Z_i/10$, where $Z_i = R_B||h_{ie}$
12. Calculate output coupling capacitance (C_o) from $X_{Co} <= Z_o/10$, where $Z_o = R_c||R_L$

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:





Result:

1. Gain=
2. Lower Cutoff Frequency f_L=
3. Upper Cutoff Frequency f_H=
4. Bandwidth= f_H- f_L
5. Input Impedance=

SWECHA
Flat No. 201,Karan Center,S.D. Road,Secunderabad. 500003.

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R_1	13 kohm
2	Resistor	R_2	2.4 kohm
3	Resistor	R_s	600 ohm
4	Resistor	R_c	820 ohm
5	Resistor	R_E	200 ohm
6	Resistor	R_L	10 kohm
7	Capacitor	C_i	10 uF
8	Capacitor	C_0	1.5 uF
9	Capacitor	C_e	220 uF
10	Transistor	Q_1	BC107A
11	Power supply	V_{CC}	12 V
12	Input Voltage Source	V_s	15 mV,1 kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

AC Simulation:

Sweep Parameter : frequency
Type: logarithmic
Start: 10Hz // Starting frequency of analysis //
Stop: 10MHz // Stop frequency of analysis //
Points Per Decade: 10
Number: 100 // Number of points in the graphs//

DC Simulation:

No changes are required

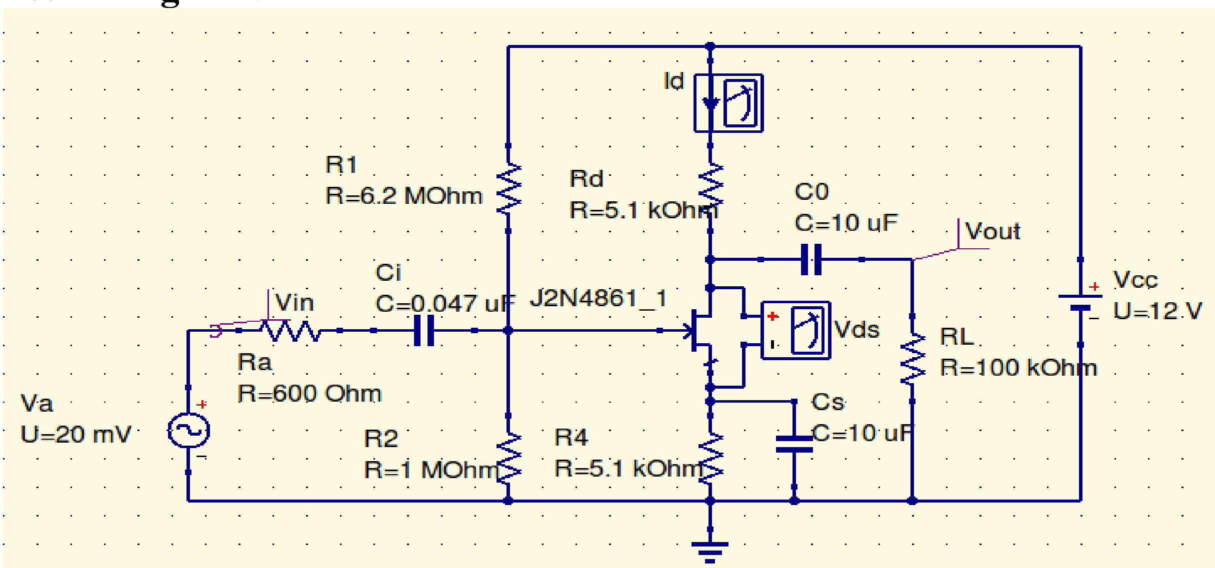
Experiment- 2
Common Source Amplifier

Aim: To simulate the Common Source Amplifier and obtain the frequency response.

Design Specifications:
AV=28dB, BW=1MHz,

Apparatus: Qucs Software

Circuit Diagram:



Design Equations:
1. Select the JFET which has higher cutoff frequency of 1MHz
2. Assume V_{DD}=12V , I_D = 1mA
3. Calculate $V_{DS(min)}=V_P+ 1 - V_{GS}$

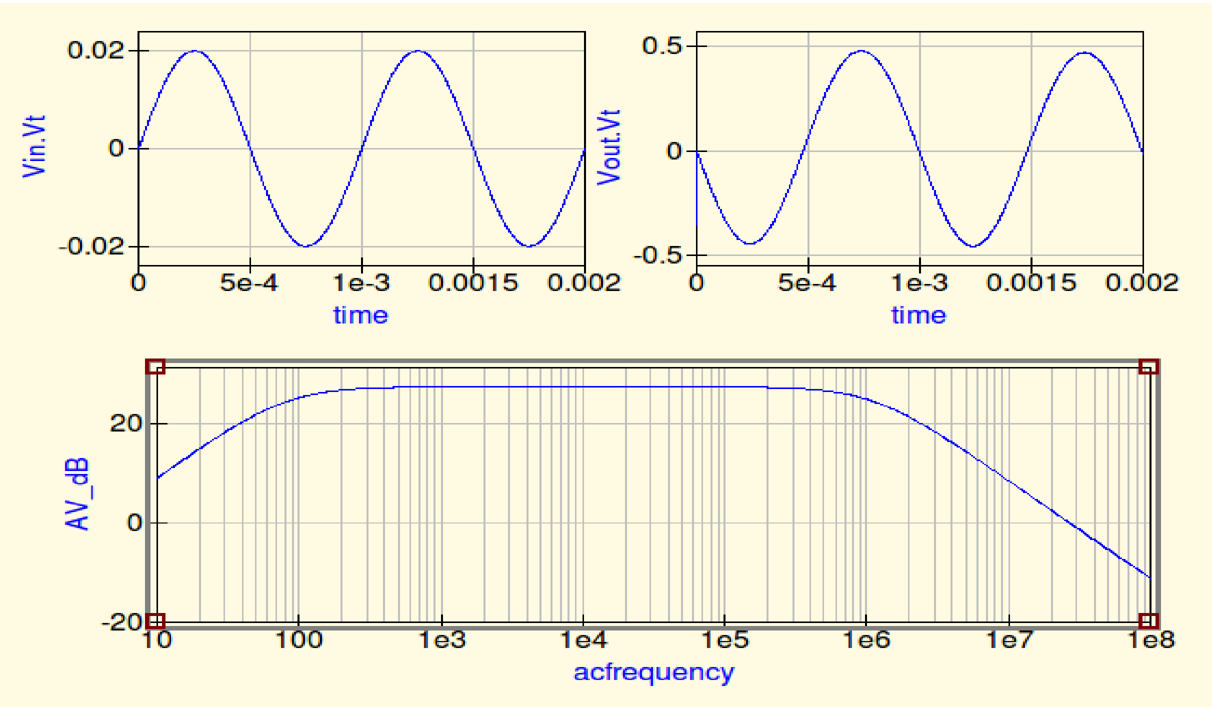
4. Calculate $V_S=(V_{DD}-V_{DS(min)}) / 2$
5. Calculate $R_S=R_D=V_S/I_D$
6. $V_{R2}=V_G= V_S-V_{GS}$
7. $V_{R1}=V_{DD}-V_G$

8. Assume R_2=1Mohm, Calculate $R1=V_{R1}R_2 / V_{R2}$, $R_{GS}=R_1||R_2$
9. $g_{m0}=2I_{DSS}/|V_P|$, $g_m=g_{m0}[1-V_{GS}/V_P]$, $r_m=1/g_m$
10. $A_V=-R_D/r_m$
11. $Xci<=R_{GS}/10$, $X_{C0}<=(R_D||R_L)/10$, $X_{CS}<=R_S/10$

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1. Voltage gain=
2. Bandwidth=

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R_1	6.2 Mohm
2	Resistor	R_2	1 Mohm
3	Resistor	R_a	600 ohm
4	Resistor	R_D	5.1 kohm
5	Resistor	R_S	5.1 kohm
6	Resistor	R_L	10 kohm
7	Capacitor	C_i	0.047 uF
8	Capacitor	C_0	10 uF
9	Capacitor	C_s	10 uF
10	Transistor	Q1	J2N4861_1
11	Power supply	V_{DD}	12 V
12	Input Voltage Source	V_a	20 mV,1 kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

AC Simulation:

Sweep Parameter : frequency
Type: logarithmic
Start: 10Hz // Starting frequency of analysis //
Stop: 100MHz // Stop frequency of analysis //
Points Per Decade: 10
Number: 100 // Number of points in the graphs//

DC Simulation:

No changes are required

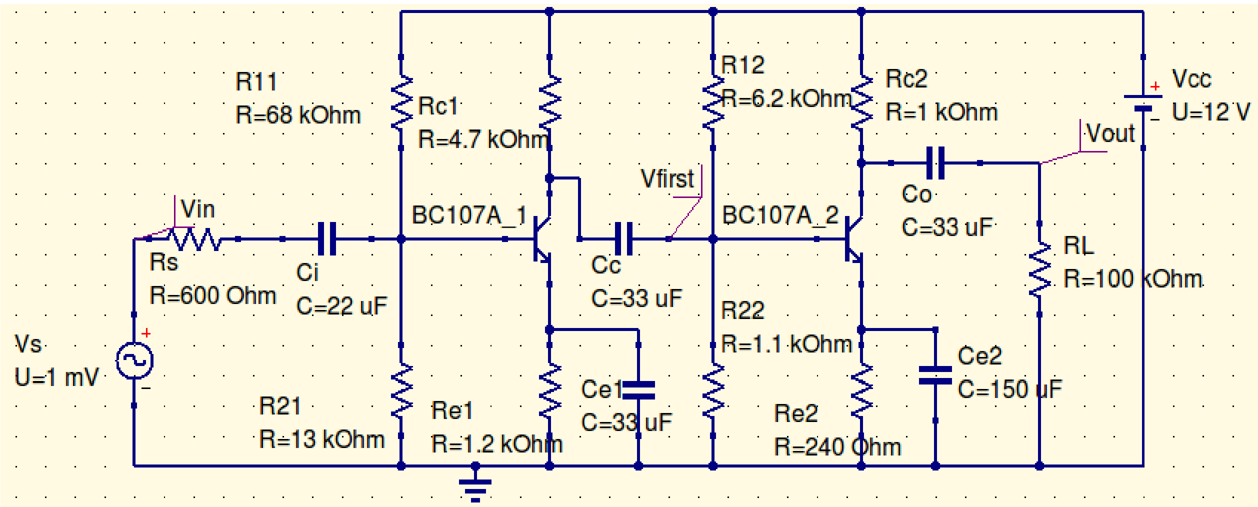
Experiment- 3
Two Stage RC Coupled Ampifier

Aim: To simulate the Two Stage RC Coupled Amplifier and obtain the frequency response.

Design Specifications:
Voltage Gain(Av1)=36dB, Voltage Gain(Av2)=11dB, Bandwidth= 700kHz, Input Impedanc =2 kohm

Apparatus: Qucs Software

Circuit Diagram:



Design Equations:

1. Select the transistors which has higher cutoff frequency of 1MHz

Design for Second Stage

2. Choose I_{C2}=5mA, Vcc=12, V_{CE2} = Vcc /2, V_{E2}= Vcc/10, S=5
3. Calculate $R_{E2}=V_{E2}/ I_{C2}$
4. Calculate R_C from $V_{CC}-I_{C2}R_{C2}-V_{CE2}-V_{E2}=0$

5. $R_{Leff2}=R_{C2}||R_L$
6. CalculateV_{B2} from $V_{B2}=V_{BE2}+V_{E2}$
7. Calculate R_{12}, R_{22} from $S=1+R_{B2}/R_E$, $V_{B2}=V_{CC}(R_2) / (R_1+R_2)$
8. $Z_{i2}=R_{B2}||\ [h_{ie2}+(1+h_{fe2})R_{E2}]$
9. $AV2=-h_{fe2}R_{Leff} / (h_{ie2}+(1+h_{fe2})R_{E2})$

Design for First Stage

10. Choose I_{C1}=1mA, Vcc=12, V_{CE1} = Vcc /2, V_{E1}= Vcc/10, S=10
11. Calculate R_{E1}, R_{C1},
12. $R_{Leff1}=R_{C1}||\ Z_{i2}$
13. $Z_{i1}=h_{ie}||R_{B1}$
14. $A_{V1}=-h_{fe1}R_{Leff1} / Z_{i1}$

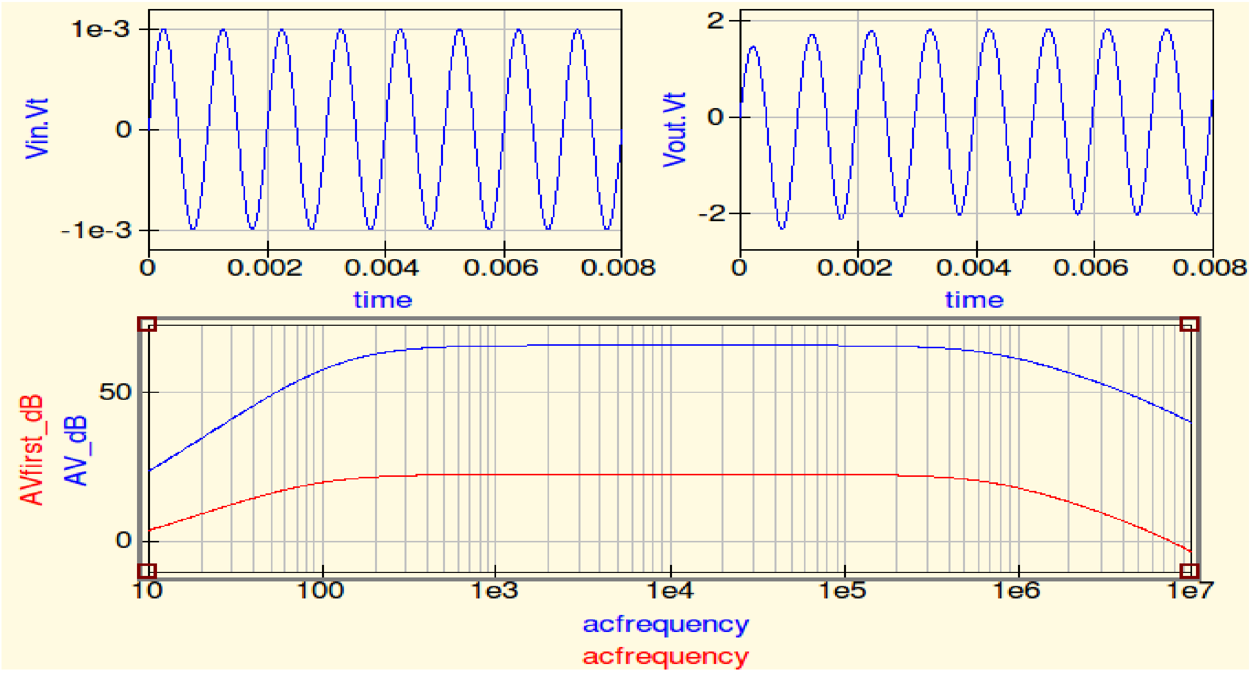
Calculation of Capacitor Values

15. $X_{ci}<=Z_{i1}/10$, $X_{ce}<=R_e/10$, $X_{cc}<=Z_{i2}/10$, $X_{c0}=R_{Leff2}/10$

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1. Overall Gain= ,
2. Gain of First stage=
3. Bandwidth of Two stage= f_H- f_L
4. Bandwidth of first stage= f_H- f_L
5. Input Impedance=

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R11	68 kohm
2	Resistor	R21	13 kohm
3	Resistor	R12	6.2 kohm
4	Resistor	R22	1.1 kohm
5	Resistor	Rs	600 kohm
6	Resistor	Rc1	4.7 kohm
7	Resistor	Rc2	1 kohm
8	Resistor	Re1	1.2 kohm
9	Resistor	Re2	240 ohm
10	Resistor	RL	100 kohm
11	Resistor	Rs	600 ohm
12	Capacitor	Ci	22 uF
13	Capacitor	Ce1	33 uF
14	Capacitor	Ce2	150 uF
15	Capacitor	Cc	33 uF
16	Capacitor	C0	33 uF
17	Transistor	Q1	BC107A
18	Transistor	Q2	BC107A
19	Power supply	VCC	12 V
20	Input Voltage source	Vs	1 mV, 1 kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

AC Simulation:

Sweep Parameter : frequency
Type: logarithmic
Start: 10Hz // Starting frequency of analysis //
Stop: 10MHz // Stop frequency of analysis //
Points Per Decade: 10
Number: 100 // Number of points in the graphs//

DC Simulation:

No changes are required

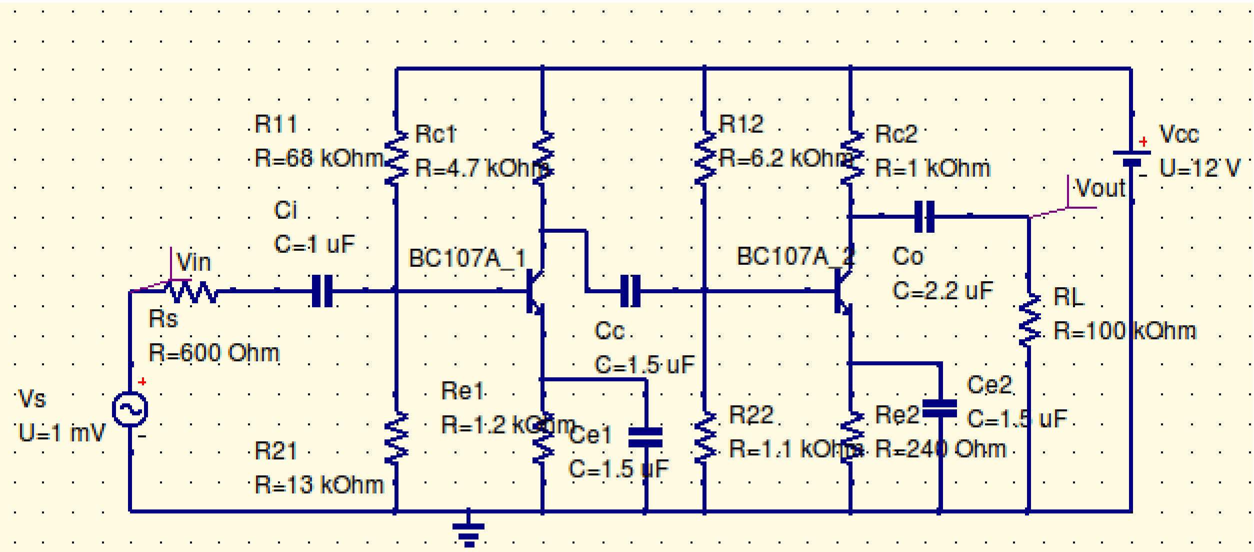
Experiment- 4
Current Shunt Feedback Amplifier

Aim: To simulate the Current Shunt Feedback Amplifier and obtain the frequency response.

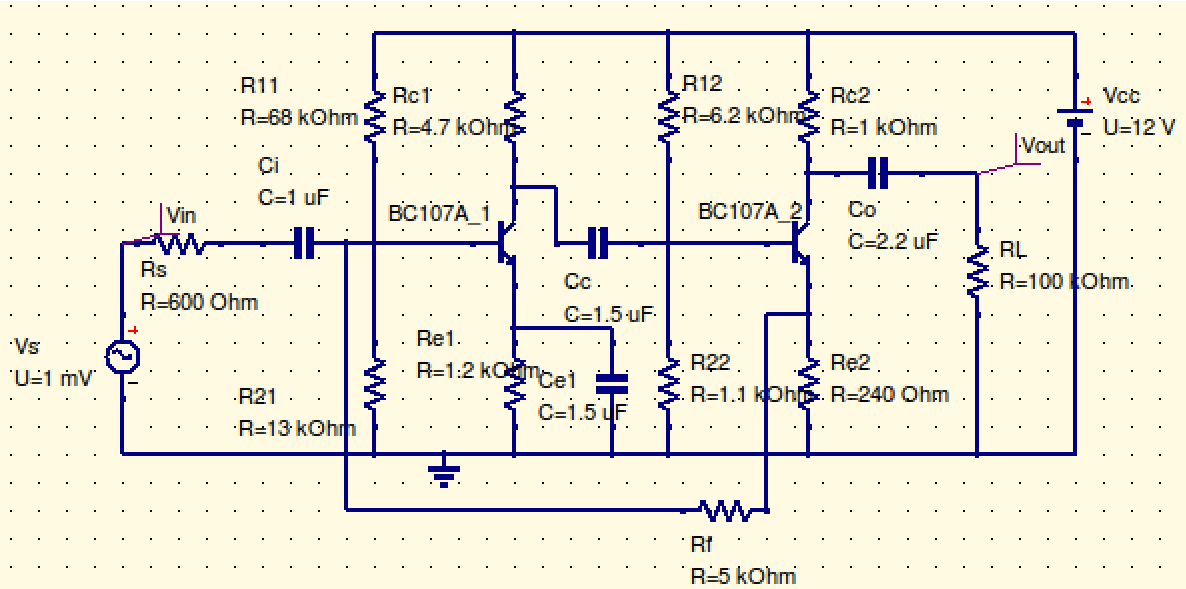
Design Specifications:
Voltage Gain(Av1)=36dB, Voltage Gain(Av2)=11dB, Input Impedanc =2kohm, f_L=1KHz without feedback
Apparatus: Qucs Software

Circuit Diagram:

Without Feedback



With Feedback



Design Equations:

1. Select the transistors which has higher cutoff frequency of 1MHz

Design for Second Stage

2. Choose I_{C2}=5mA, Vcc=12, $V_{CE2} = Vcc/2$, $V_{E2} = Vcc/10$, S=5
3. Calculate $R_{E2}=V_{E2}/I_{C2}$
4. Calculate R_C from $V_{CC}-I_{C2}R_{C2}-V_{CE2}-V_{E2}=0$
5. $R_{Leff2}=R_{C2}||R_L$
6. Calculate V_{B2} from $V_{B2}=V_{BE2}+V_{E2}$
7. Calculate R_{12}, R_{22} from $S=1+R_{B2}/R_E$, $V_{B2}=V_{CC}(R_2)/(R_1+R_2)$
8. $Z_{i2}=R_{B2}||[h_{ie2}+(1+h_{fe2})R_{E2}]$
9. $AV2=-h_{fe2}R_{Leff}/(h_{ie2}+(1+h_{fe2})R_{E2})$

Design for First Stage

10. Choose I_C=1mA, Vcc=12, $V_{CE} = Vcc/2$, $V_E = Vcc/10$, S=10
11. Calculate RE, RC,
12. R_{Leff1}=RC1|| Zi2
13. Zi1=hie||RB1
14. AV1=-hfeRLeff / Zi1

Calculation of Capacitor Values

15. $X_{ci}<=Z_{i1}/10$, $X_{ce1}<=R_{e1}/10$, $X_{ce2}<=R_{e2}/10$, $X_{cc}<=Z_{i2}/10$, $X_{c0}=R_{Leff2}/10$

Design With Feedback

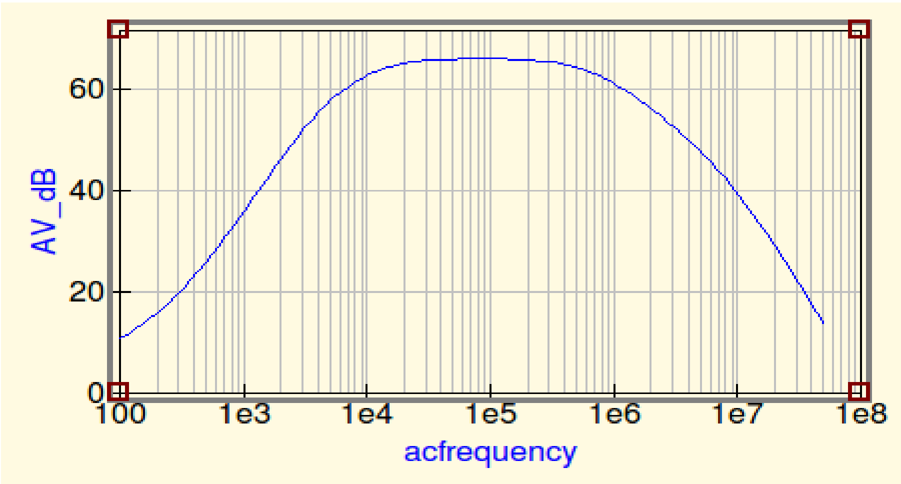
16. $\beta = -R_{e2} / (R_f+R_{e2})$, Choose R_f=5 Kohm
17. $D=1+ \beta A_I$, $A_I=(h_{fe1}h_{fe2})(R_{c1}||R_{B2}) / (Z_{i2}+(R_{c1}||R_{B2}))$
18. $A_{If}= A_I/D$
19. $A_{Vf}=A_{If}(R_{Leff2})/R_s$
20. $Z_0f=Z_{0D}$, $Z_{if}=Z_i/D$

Procedure:

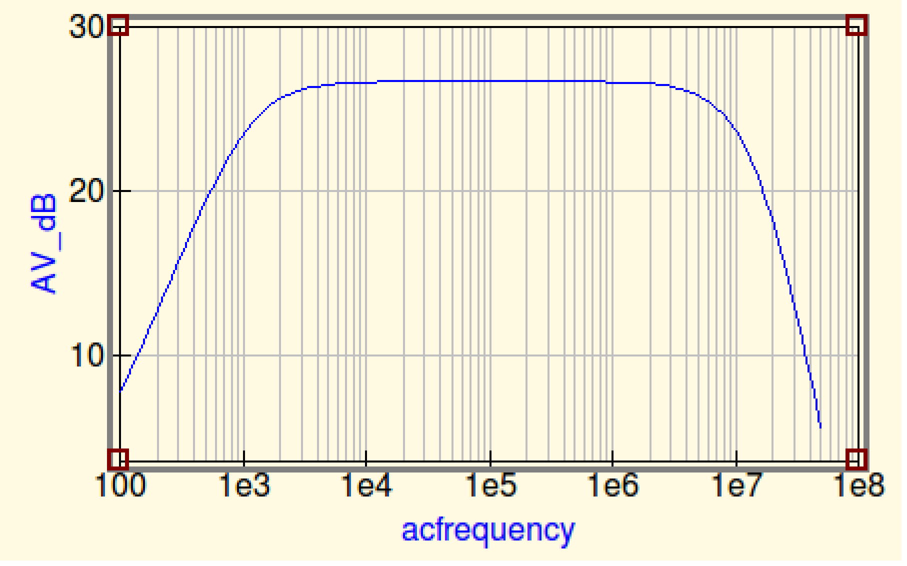
1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:

Without Feedback



With Feedback



Result:

1. Without feedback A_V= ,
2. With Feedback A_{vf}=
3. Without feedback BW=f_H-f_L
4. With feedback BW=f_H-f_L
5. Without feedback Z_i = , Z_0=
6. With feedback Z_i = , Z_0=

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R_{11}	68 kohm
2	Resistor	R_{21}	13 kohm
3	Resistor	R_{12}	6.2 kohm
4	Resistor	R_{22}	1.1 kohm
5	Resistor	R_s	600 kohm
6	Resistor	R_{c1}	4.7 kohm
7	Resistor	R_{c2}	1 kohm
8	Resistor	R_{e1}	1.2 kohm
9	Resistor	R_{e2}	240 ohm
10	Resistor	R_L	100 kohm
11	Resistor	R_s	600 ohm
12	Capacitor	C_i	1uF
13	Capacitor	C_{e1}	1.5 uF
14	Capacitor	C_{e2}	1.5 uF
15	Capacitor	C_c	1.5 uF
16	Capacitor	C_0	2.2 uF
17	Transistor	Q_1	BC107A
18	Transistor	Q_2	BC107A
19	Power supply	V_{CC}	12 V
20	Input Voltage source	Vs	1 mV, 1 kHz
21	Resistor	Rf	5 kohm

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

AC Simulation:

Sweep Parameter : frequency
Type: logarithmic
Start: 10Hz // Starting frequency of analysis //
Stop: 50MHz // Stop frequency of analysis //
Points Per Decade: 10
Number: 100 // Number of points in the graphs//

DC Simulation:

No changes are required

SWECHA

Flat No. 201,Karan Center,S.D. Road,Secunderabad. 500003.

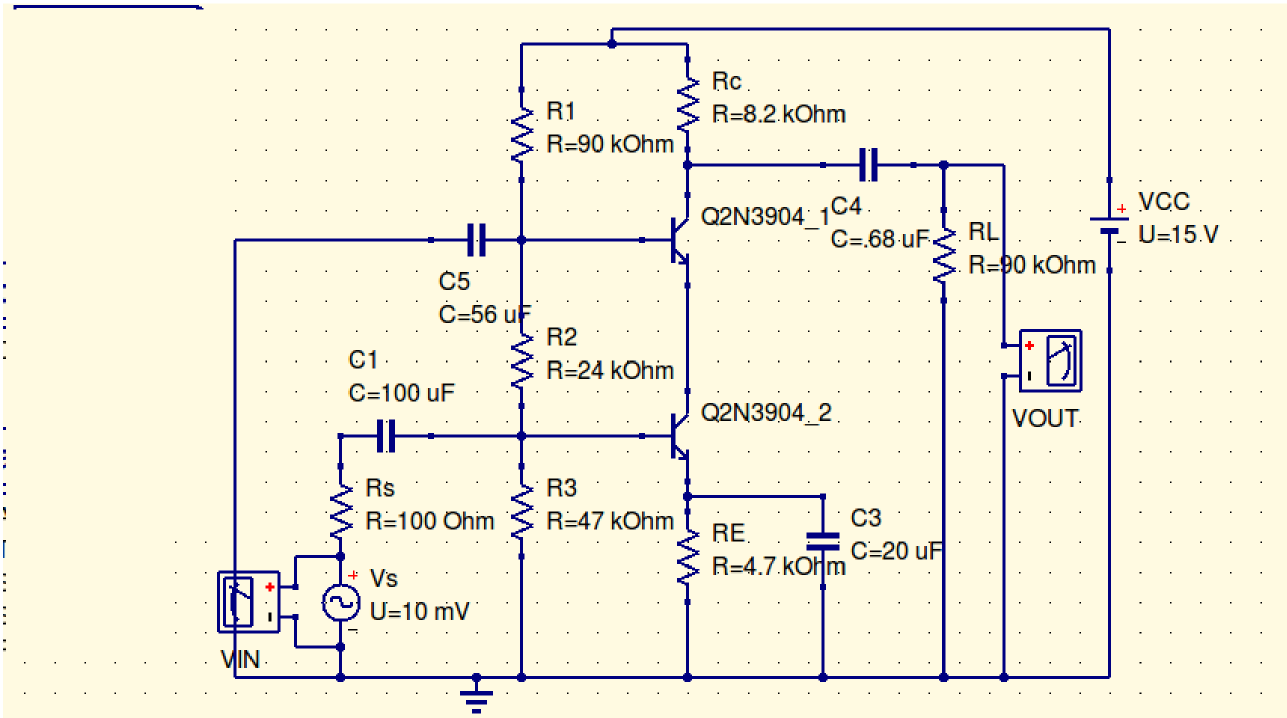
Experiment- 5
Cascode Amplifier

Aim: To simulate the Cascode Amplifier and obtain the frequency response.

Design Specifications:
Voltage Gain(Av)=100, Bandwidth= 1MHz

Apparatus: Qucs Software

Circuit Diagram:



Design Equations:

1. Select the transistor which has higher cutoff frequency of 1MHz
2. Assume V_{CC}=15V,V_{CE1} =V_{CE2} =V_{CC}/3.I_{E1}=I_{E2}=1mA,Rs = 600 ohm.
3. R_{Leff}= R_C||R_L.
4. re1 = 26mV/I_{E1}. hie1 = β1*re1. Since β1= β2,I_{E1}=I_{E2} =>re1=re2.

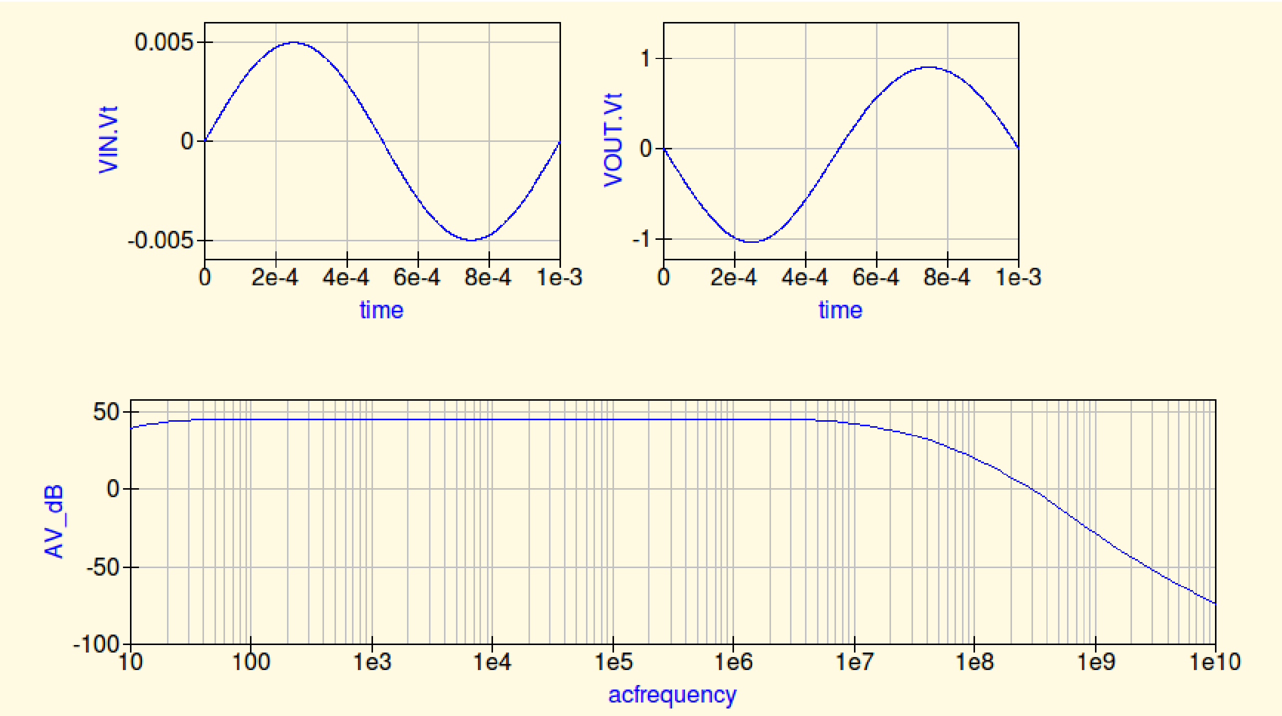
5.Gain for Q1 transistor $Av1=V01/Vi \approx - R_L/re1$.
With R_L=re2=hib2 of transistor-2 => Av1 = -re2/re1=-1.
6. Av2 = R_{Leff}/re2=?,total gain A_T=Av1*Av2 = 100.
calculate Av2 from above formula, from Av2 and R_{Leff} calculate Rc.
7.calculate R_E from

$$Vcc=IcRc + V_{CE2} + V_{CE1} + I_E R_E.$$

8.$I_{B1} = I_{B2} = I_{C1}/ \beta$, $R_3=10*R_E$,
find I_3 from $I_3=V_{B1}/R_3$ where $V_{B1}=V_{E1}+V_{BE1}$.
find I_2 from $I_2 = I_3+I_{B1}$
find R_2 from $R_2= [V_{B2}-V_{B1}]/I_2$.
find I_1 from $I_1 = I_2+I_{B2}$.
Find R_1 from $R_1=[Vcc-V_{B2}]/I_1$.
9.output coupling capacitor is given by $X_{C0} = (Rc||R_L)/10$.
X_{C0} = 1/2pi*f*C_0 where f is lower cutoff frequency. In diagram $C_0=C_4$.
Bypass capacitor is given by $X_{CE} = R_E/10$.
X_{CE} = 1/2pi*f*C_E. In diagram $C_E=C_3$.

Procedure:
1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1. Voltage Gain AV=
2. Bandwidth BW=f_H-f_L

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R1	90 kohm
2	Resistor	R2	24 kohm
3	Resistor	Rs	100 ohm
4	Resistor	R3	47 kohm
5	Resistor	Rc	8.2 kohm
6	Resistor	Re	4.7 kohm
7	Resistor	RL	90 kohm
8	Capacitor	C1	100 uF
9	Capacitor	C3	20 uF
10	Capacitor	C4	68 uF
11	Capacitor	C5	56 uF
12	Transistor	Q1	2N3904
13	Power supply	VCC	15 V
14	Input Voltage Source	Vs	10 mV,1 kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

AC Simulation:

Sweep Parameter : frequency
Type: logarithmic
Start: 10Hz // Starting frequency of analysis //
Stop: 2ms // Stop time of analysis //
Points Per Decade: 10
Number: 100 // Number of points in the graphs//

DC Simulation:

No changes are required

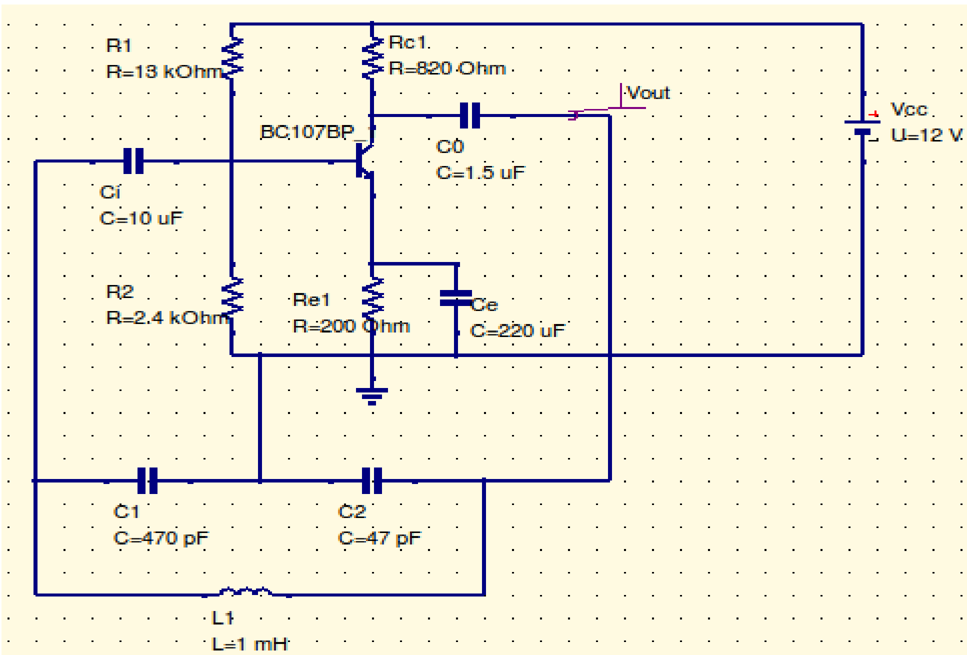
Experiment- 6
Colpitts Oscillator

Aim: To simulate the Colpitts Oscillator and obtain the transient response.

Design Specifications: 1. Voltage Gain(A_V)=50 ,
2. Frequency of the output signal=770 kHz

Apparatus: Qucs Software

Circuit Diagram:



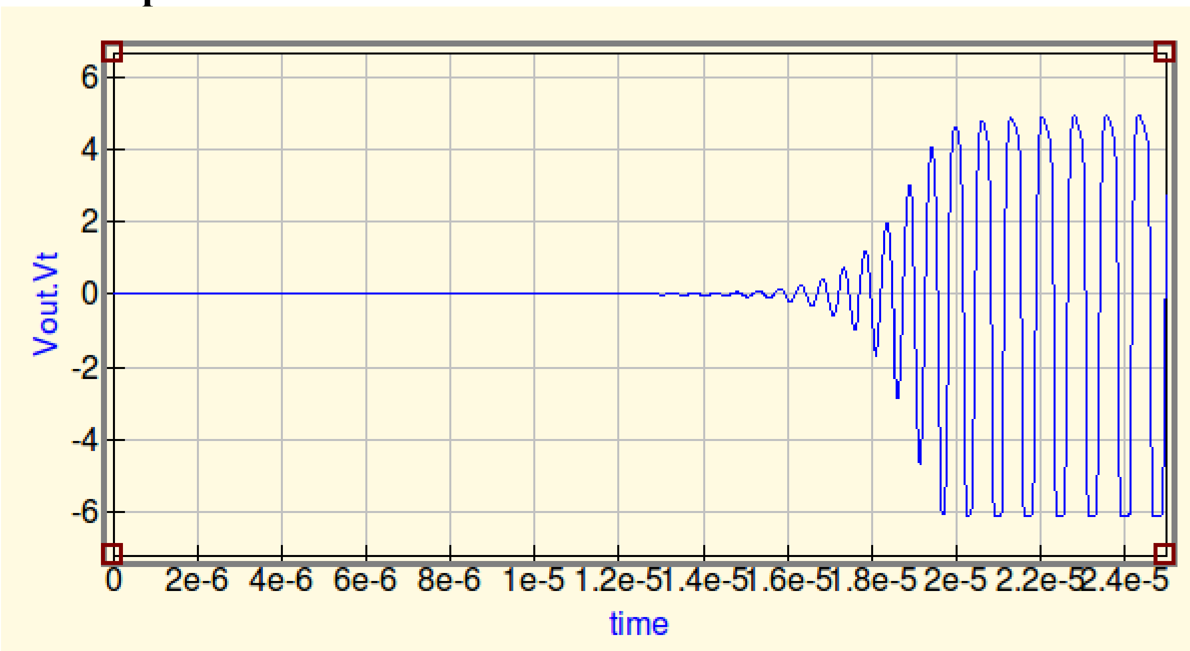
Design Equations:

1. Design the CE Amplifier for the given Gain.
2. Choose C_1
3. Calculate C_2 from $A_V > C_1/C_2$
4. Calculate C from $f=1/(2\Pi (L_1C)^{1/2})$, where $C= C_1C_2/(C_1+C_2)$

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the data sheet
3. Place the transiant simulation ,d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1. Theoritical Frequency (f_T)=(1/2Π) X($(C_1+C_2)/LC_1C_2)^{1/2}$
2. Practical Frequency (f_P)= $1/T_{measured}$

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R1	13 kohm
2	Resistor	R2	2.4 kohm
3	Resistor	Rc1	820 ohm
4	Resistor	Re1	200 ohm
5	Capacitor	Ci	10 uF
6	Capacitor	C0	1.5 uF
7	Capacitor	Ce	220 uF
8	Capacitor	C1	470 pF
9	Capacitor	C2	47 pF
9	Inductor	L1	1 mH
10	Transistor	BC107BP	BC107BP
11	Power supply	VCC	12 V

Simulation Properties Sheet

Transient Simulation:
Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 0.025ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

DC Simulation:
No changes are required

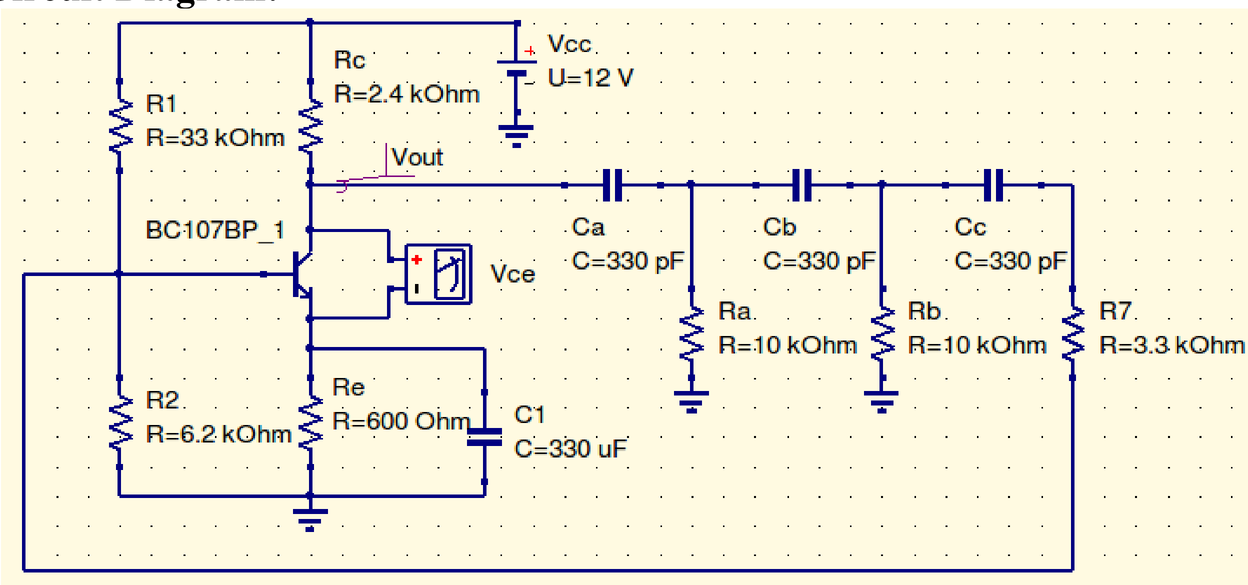
Experiment- 7
RC Phase Shift Oscillator using Transistor

Aim: To simulate the RC Phase Oscillator using Transistor and obtain the transient response.

Design Specifications: Frequency of output signal = 18kHz , $A_V >= 29$

Apparatus: Qucs Software

Circuit Diagram:



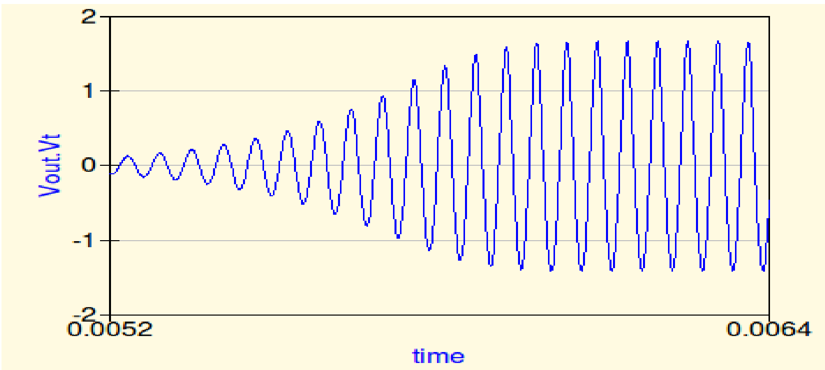
Design Equations:

1. Assume V_{CC}=12V , $V_{CE} = V_{CC}/2$, $V_E = V_{CC}/10$
2. Calculate Rc from $Av=-(h_{FE}\,(R_c||1/h_{oe}))\,/\,h_{ie}$, where h_{ie} , h_{oe} can be taken from the manufacturers datasheet of the transistor.
3. Calculate I_C from $V_{CC}-I_CR_C-V_{CE}-V_E=0$
4. Assume $I_C=I_E$, Calculate R_E from $V_E= I_ER_E$
5. $S=1+ (R_B/R_E)$, choose S=10, calculate $R_B =9R_E$,where $R_B=R1||R2$
6. Calculate $V_B=V_{BE}+V_E$, where V_{BE}=0.65 V
7. Calculate the ratio R1/R2 from $V_B=(R2\,.V_{CC})\,/\,(R1 + R2)$
8. From steps 5 and 7 calculate R1, R2

9. Calculate emitter bypass capacitance (C_E) from X_{CE} <= $R_E/10$
10. Choose R= Ra=Rb=10 kohm, calculate Ca=Cb=Cc using
 f=1/2πRC(6+4K)1/2 , where K= Rc/R
11. Calculate R7 from R7= R-h_{ie}
12. Choose the transistor such that $h_{oe}R_C < 0.1$, $h_{FE} > 4K+23+29/K$

Procedure:
1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:
1. Theoritical Frequency (f_T)=$1/2\Pi RC(6+4K)^{1/2}$,
 where K= Rc/R , R=Ra=Rb

2. Practical Frequency (f_P)= 1/ $T_{measured}$

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R1	33 kohm
2	Resistor	R2	6.2 kohm
3	Resistor	Re	600 ohm
4	Resistor	Rc	2.4 kohm
5	Resistor	Ra	10 kohm
6	Resistor	Rb	10 kohm
7	Resistor	R7	3.3 kohm
8	Capacitor	C1	330 uF
9	Capacitor	Ca, Cb, Cc	330 pF
10	Transistor	BC107BP	BC107BP
11	Power supply	Vcc	12 V

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 7ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

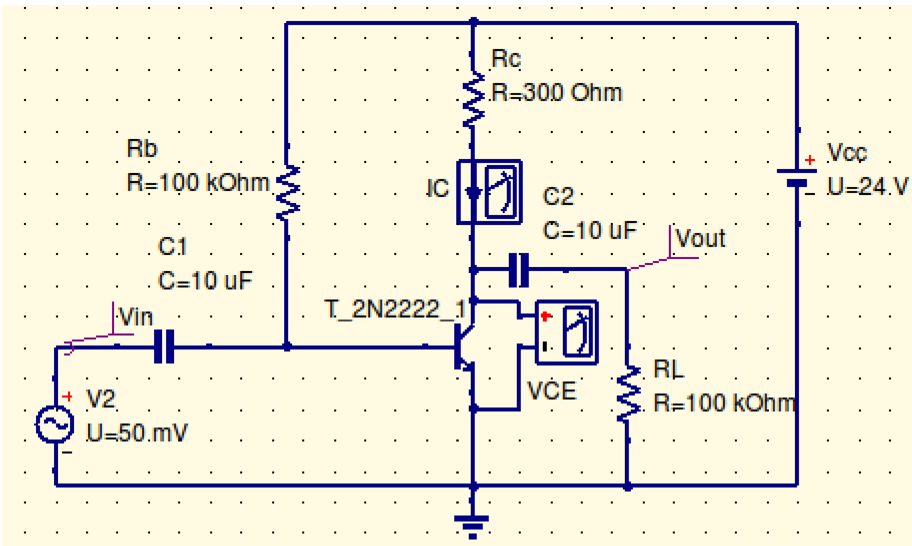
Experiment- 8
Class-A Power Amplifier (Transformerless)

Aim: To simulate the Class-A Power Amplifier and calculate the Efficiency.

Design Specifications: Efficiency (η) =10%

Apparatus: Qucs Software

Circuit Diagram:



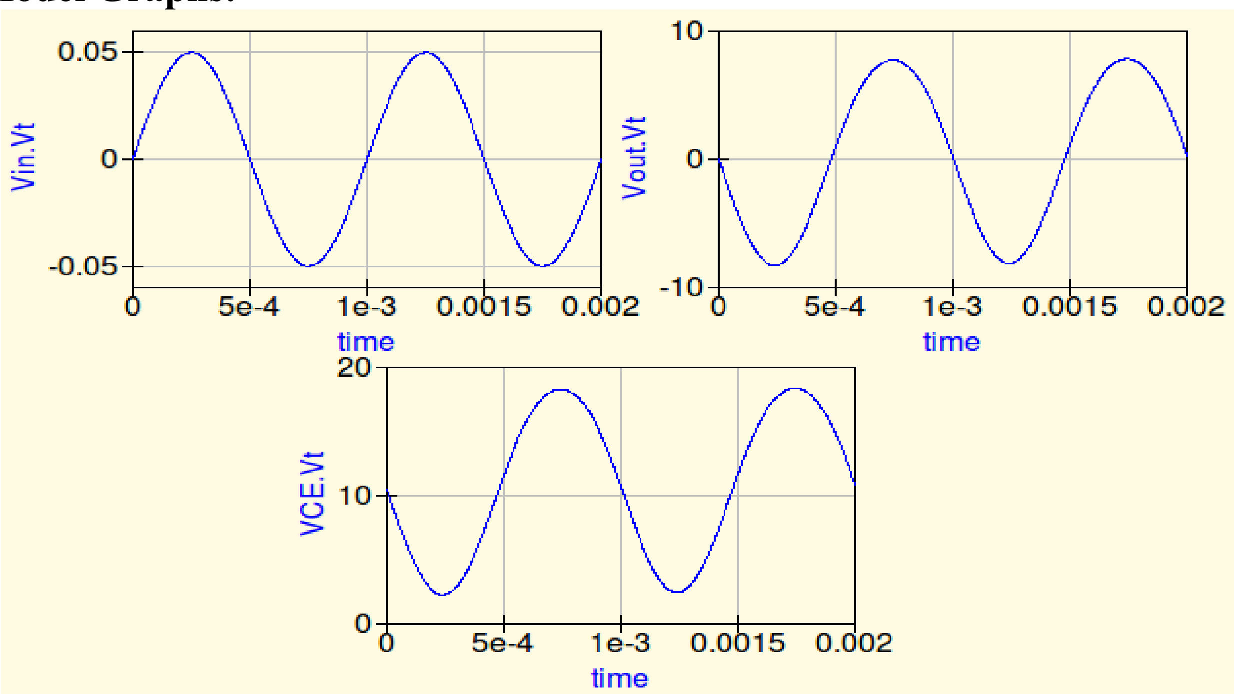
Design Equations:

1. Transistor Specifications will include I_{cmax} , CE breakdown Voltage BV_{CEO} and P_{Cmax}
2. Choose $2V_{CEQ} <= BV_{CEO}$ and $2I_{CQ} <= I_{cmax}$
3. Assume V_{CC} =24 ,$V_{CEQ} = V_{CC}/2$
4. Calculate Rc from $V_{CC} - I_{CQ}R_C - V_{CEQ} = 0$
5. Calculate R_B from $I_{BQ} = I_{CQ}/h_{FE}$, $I_{BQ} = (V_{CC} - 0.7) / R_B$
6. Choose C1,C2=10uF

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1. $P_{dc(i)} = V_{CC}I_{CQ}$ =
2. $P_{ac(o)} = (V_{CE(P-P)})^2 / (8R_C)$ =
3. $\eta = (Pac(o) / P_{dc(i)}) \times 100$ =

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	Rb	100 kohm
2	Resistor	Rc	300 ohm
3	Resistor	RL	100 kohm
4	Capacitor	C1	10 uF
5	Capacitor	C2	10uF
6	Transistor	2N2222	2N2222
7	Power supply	VCC	24 V
8	Input Voltage Source	VS	50mV, 1kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

DC Simulation:

No changes are required

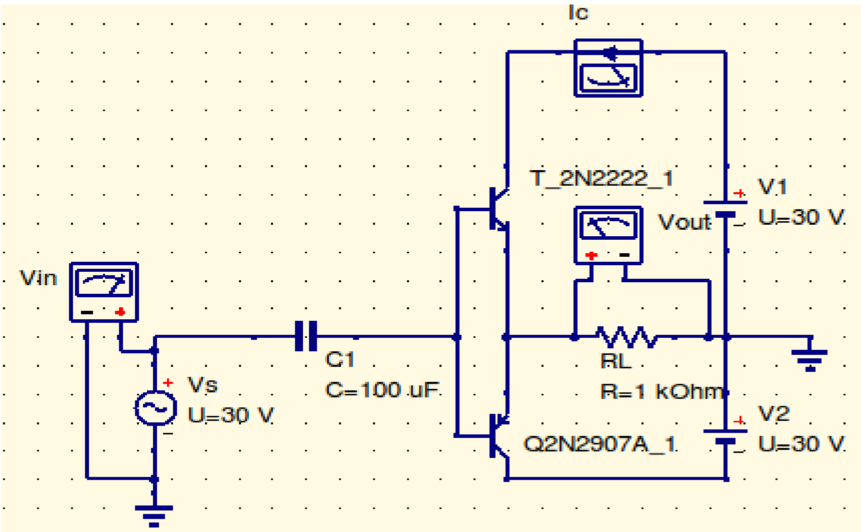
Experiment- 9
Class-B Complementary Symmetry Amplifier

Aim: To simulate the Class-B Complementary Symmetry Amplifier and calculate the Efficiency.

Design Specifications: Efficiency η=78%

Apparatus: Qucs Software

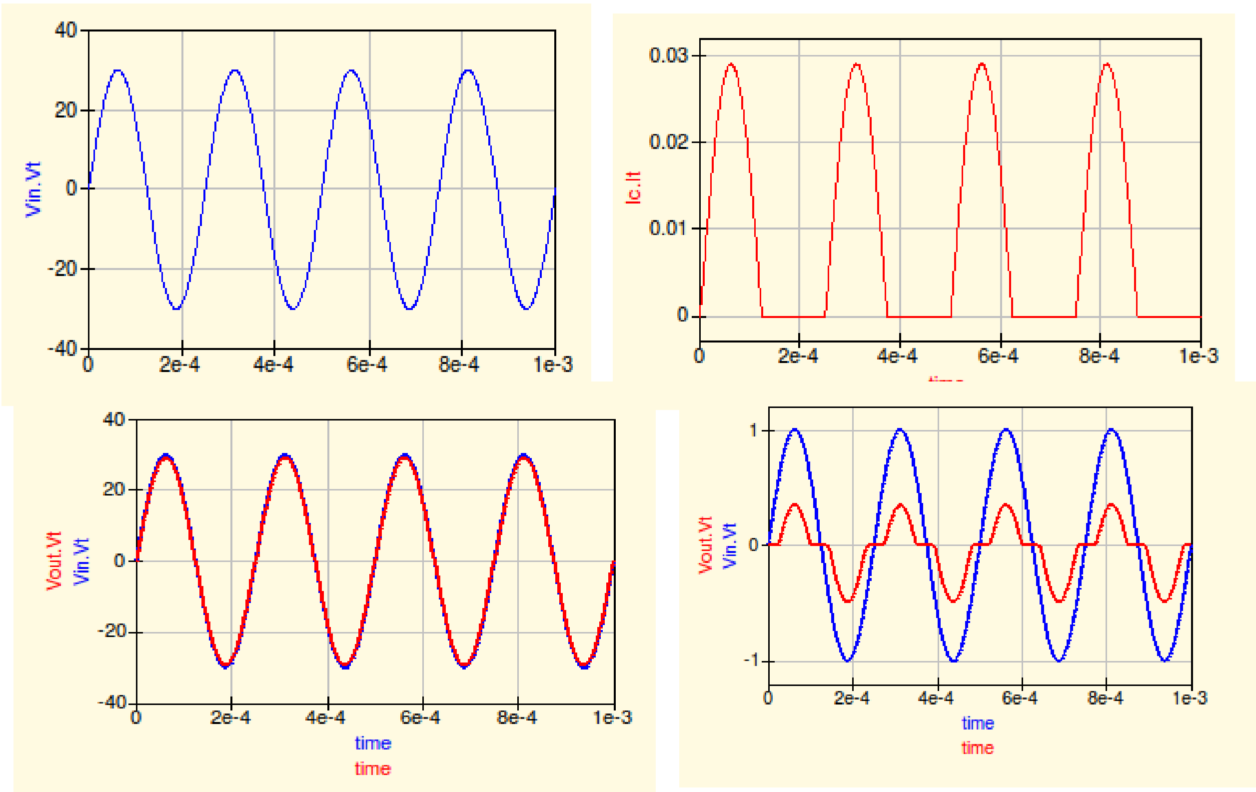
Circuit Diagram:



Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph by giving input voltage as 1V and 30V.

Model Graphs:



Result:

1. Pdc(i)= $V_{CC}(2I_{C(P)} / \pi)$
2. Pac(o)= $(V_{L(P-P)})^2 / 8R_L$
3. η =(Pac(o) / $P_{dc(i)}$) X 100

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	RL	1 kohm
2	Capacitor	C1	100 uF
3	Transistor	2N2907A	2N2907A (PNP)
4	Transistor	2N2222	2N2222 (NPN)
5	Power supply	V1	30 V
6	Power supply	V2	30 V
7	Input Voltage Source	V3	(1-30) V, 1 kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

DC Simulation:

No changes are required

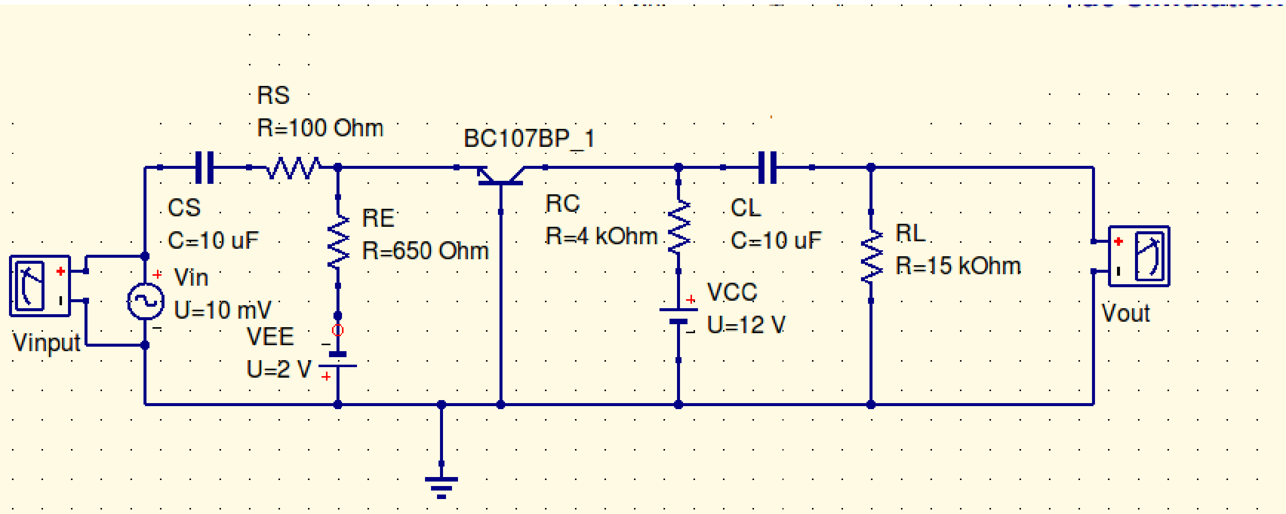
Experiment- 10
Common Base (BJT) Amplifier

Aim: To simulate the Common Base Amplifier and obtain the frequency response.

Design Specifications:
Voltage Gain(Av)=30, Bandwidth= 1MHz,

Apparatus: Qucs Software

Circuit Diagram:

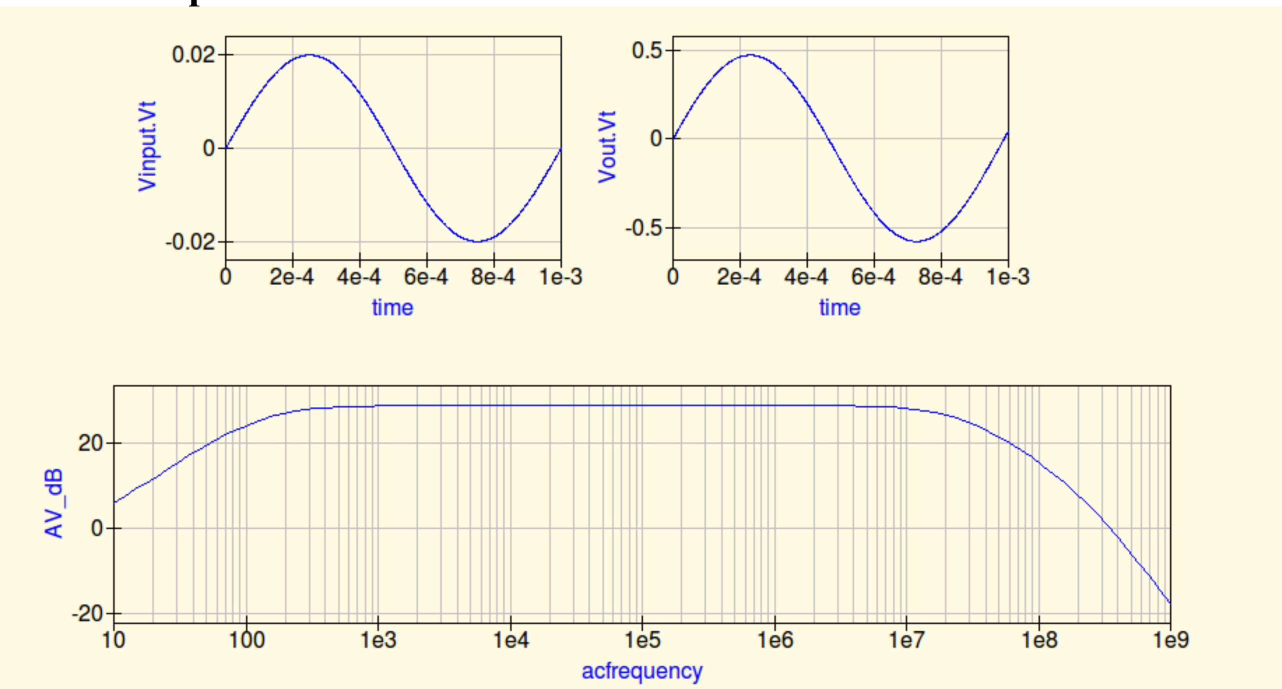


Design Equations:
1. Select the transistor which has higher cutoff frequency of 1MHz
2. Assume VCC=12V , VCB =VCC/2.
3. Calculate Rc from equation $Avs = -h_{fb}*R_L'/(Ri + Rs)$ where $R_L' = Rc|| R_L$
   $Ri = hib$,Rs is the sourcr resistance,R_L is the load resistance
4. Calculate Ic from equation $Vcc-IcRc-V_{CB} = 0$.
5.Assume $Ic = I_E$ and calculate RE from $-V_{EE} + I_ER_E-V_{CB}=0$.
6.Calculate Cs from equation
   $f_L = 1/(2pi(Rs+Ri)Cs)$ wher $f_{L\ is}$ the lower cutoff frequency.
   and take C_L =Cs.

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1.Voltage Gain=

2.Bandwidth BW= f_H-f_L

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	Rs	100 ohm
2	Resistor	RE	650 ohm
3	Resistor	RC	4 Kohm
4	Resistor	RL	15 kohm
5	Capacitor	Cs	10 uF
6	Capacitor	CL	10 uF
7	Transistor	BC107BP	BC107BP
8	Power supply	VCC	12 V
9	Power supply	VEE	2V
10	Input Voltage Source	Vin	10mV,1 kHz

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 2ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

AC Simulation:

Sweep Parameter : frequency
Type: logarithmic
Start: 10Hz // Starting frequency of analysis //
Stop: 100MHz // Stop time of analysis //
Points Per Decade: 10
Number: 100 // Number of points in the graphs//

DC Simulation:

No changes are required

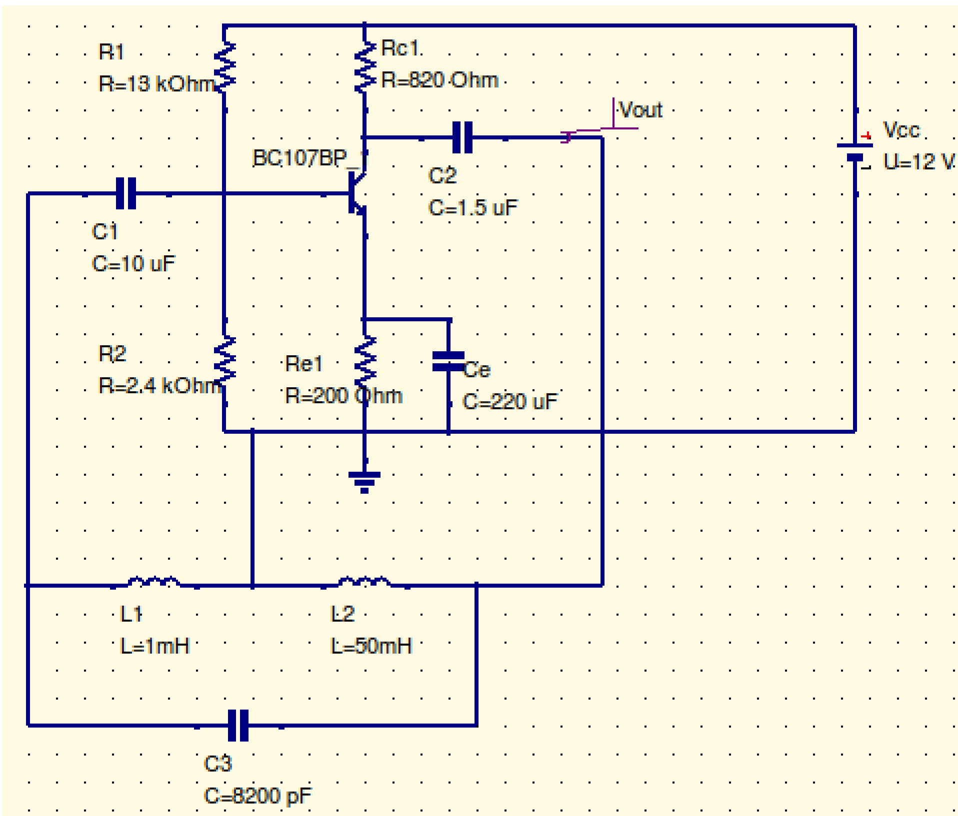
Experiment- 11
Hartley Oscillator

Aim: To simulate the Transistor Hartley Oscillator and obtain the transient response.

Design Specifications: Voltage Gain(A_V)=50 ,
Frequency of the output signal=7.7 kHz

Apparatus: Qucs Software

Circuit Diagram:



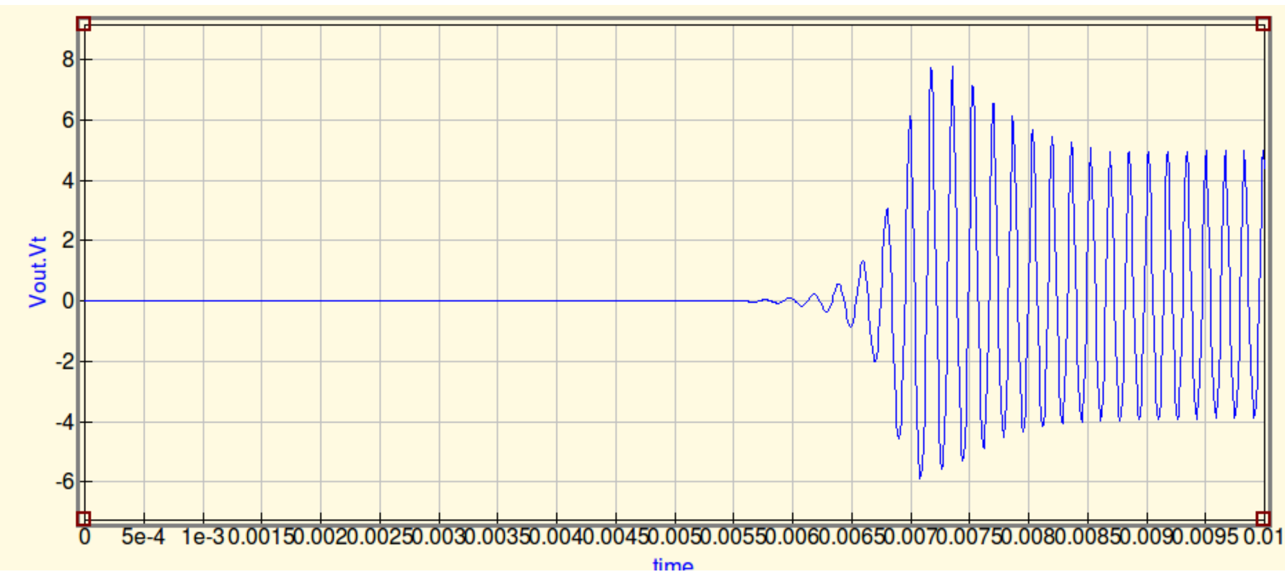
Design Equations:

1. Design the CE Amplifier for the given Gain.
2. Choose L_1
3. Calculate L_2 from $A_V=1/\beta=L_2/L_1$
4. Calculate C_3 from $f=1/(2\Pi\ (LC_3)^{1/2})$, where $L= L_1+L_2$

Procedure:

1. Connect the circuit as per the circuit diagram
2. Set the properties of components as per the components properties sheet
3. Place the transient simulation, d.c simulation and a.c simulations on editor.
4. Set the simulation properties
5. Simulate the circuit
6. Place the cartesian diagram and set the properties.
7. Note down the the graph.

Model Graphs:



Result:

1. Theoritical Frequency $(f_T)=1/\ (2\Pi\ ((L_1+L_2)C)^{1/2}\ )$
2. Practical Frequency $(f_P)=\ 1/T_{measured}$

Component Properties sheet

SNO	Component	Name	Value
1	Resistor	R1	13 kohm
2	Resistor	R2	2.4 kohm
3	Resistor	Rc1	820 ohm
4	Resistor	Re1	200 ohm
5	Capacitor	C1	10 uF
6	Capacitor	C2	1.5 uF
7	Capacitor	C3	2 uF
8	Inductor	L1	2 mH
9	Inductor	L2	2mH
10	Transistor	BC107BP	BC107BP
11	Power supply	VCC	12 V

Simulation Properties Sheet

Transient Simulation:

Sweep Parameter : time
Type: linear
Start: 0 // Starting time of analysis //
Stop: 10 ms // Stop time of analysis //
Step: 1.8018e-06 // Step Size or incrementing value//
Number: 1111 // Number of points in the graphs//

DC Simulation:
No changes are required

SWECHA

Flat No. 201,Karan Center,S.D. Road,Secunderabad. 500003.