



*Guide to*

**LTSPICE XVII**

# Index

Serial No.	Topic/Activity	Page number
Introduction		
1.	Brief Introduction on the Tool	1
2.	How to install and Setup LTSpice XVII	2
3.	Tips and tricks	3
4.	Basic syntax and simulation commands and Simulation steps	4-5
Example circuits		
5.	RC circuits	7-9
	a) Low-pass filter	7
	b) High-pass filter	8
	c) Band-pass filter	9
7.	RL circuits	10-12
	a) Low-pass filter	10
	b) High-pass filter	11
	c) Band-pass filter	12
8.	Astable multivibrator using NE555	13
9.	Diode circuits	14-18
	a) Clippers	14
	i) Positive clipper	15
	ii) Negative clipper	16
	iii) Double-ended clipper	17
	b) Bridge Rectifier	18
10.	Mini-project: AC to DC converter using Bridge rectifier and filters	19
11.	BJT circuits	
	a) IV characteristics of NPN and PNP	20-21
	b) CE single stage amplifier with VDB (voltage divider bias)	22-24
	c) Differential Amplifier	25-26
12.	Oscillators	27-32
	a) Hartley Oscillator	27-29
	b) Colpitts Oscillator	30-32
13.	Op-amps	33-36
	a) Basic characteristics (Gain)	33
	b) Adder	34
	c) Subtractor	35
	d) Logarithmic	36
14.	MOSFET circuits	37-49
	a) IV characteristics NMOS and PMOS	37-40
	b) NMOS current sink	41
	c) PMOS current source	42
	d) NMOS CS single stage amplifier with resistive load	43-47
	e) NMOS CS differential amplifier with balanced resistive load	48-49

# Introduction to the tool

---

As you probably know SPICE stands for Simulation Program with Integrated Circuit Emphasis i.e. we use SPICE models specifically in design and simulation of Integrated circuits and ASIC(Application Specific IC) chipsets.

With advancement of techniques in design, there are so many tools and software to help you focus on this domain. One such tool is LTSpice.

LTSpice stands for Linear Technology SPICE. It is very clear from the Brand itself that we are focusing on the linear models of systems and components, though there are ways to simulate non-linearity as seen in chapter 14 MOSFET circuits.

LTSpice is a high-performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of Analog circuits. Included in the download of LTSpice are macro-models for a majority of Analog Devices switching regulators, amplifiers, as well as a library of devices for general circuit simulation.

## Benefits of LTSpice

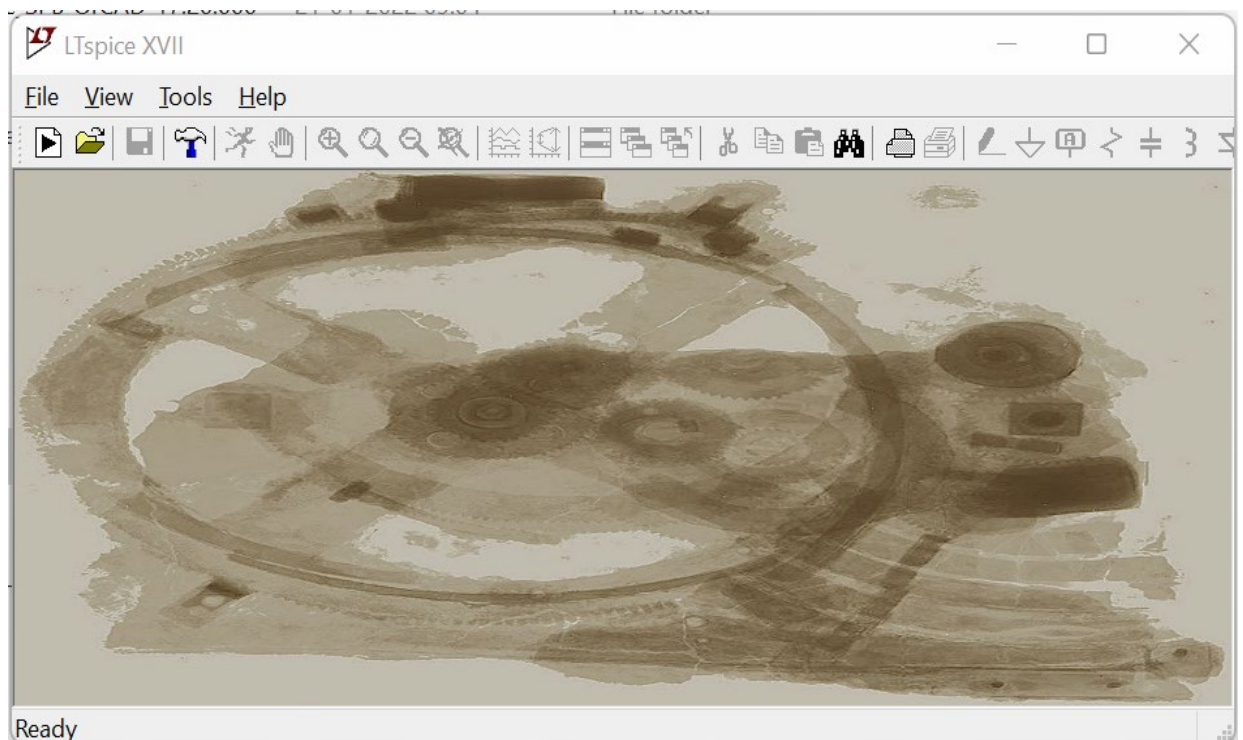
It's enhancements to SPICE have made simulating switching regulators extremely fast compared to normal SPICE simulators and easy to simulate wide range of circuits and systems, allowing the user to view waveforms for most switching regulators in just a few minutes. This manual provides an overview of the advantages of using LTSpice in an Analog circuit design with the help of many example-circuits.

# How to install and setup LTSpice

---

A short guide to install and setup LTSpice is mentioned here

1. go to the below-mentioned link and scroll down to the downloads section.  
<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
2. Download the software based on compatibility of your OS
3. Go to the downloaded executable (.exe) file and start the setup as per the prompts provided by the setup wizard.
4. As soon as you finish the setup, a screen should appear as shown below



5. From here, clicking on “file” and “new schematic” will open up a schematic window and it will be autosaved as Draft1 in C:/Users/username/Documents/LTspiceXVII.
6. That is all and you are all set for simulating circuits

Note: For further Assistance, you can go to the “help” and “help topics”

# Tips and Tricks

---

Since you will be making many schematics and doing a lot of schematics, you might want to save lot of time. For this, LTSpice provides few useful hotkeys (you can always edit them in the control panel under “drafting options”)

## Few useful Hotkeys

V= Voltage source

G = Reference/Ground

R= linear resistance

C= Ceramic capacitor

L = Inductor

D = Diode

S = Spice directives (explained in detail in the next section)

T = Comment/Non-executable text

LTSpice also provides settings for few options such as Grid-point toggle in schematic page and also the plots page. This enables easier and more systematic way to analyse the plot and rig up the circuit.

Few other useful features provided are :

- Colour scheme choices
- Multiple plots in (in different panes) {right click → “add plot pane” → move the trace label to empty pane}
- Adding custom traces (discussed in next section)
- Sliding markers to get the Parameter values on X and Y Axes : Add this by clicking on the trace label on top of plot pane (the difference along the X and Y individually are calculated and it can be viewed in the marker coordinates box)
- Changing scale of X,Y Axes (right-click when ruler appears)
- You can plot Fourier transform spectrum of a curve (right click →view→FFT)

## Basic Simulation and SPICE commands

---

Command syntax	Description
Simulation commands (included only the once used in this book... There are more...Refer to the “Help topics” included in the software)	
.dc source_name, from, to, increment	DC sweep used to analyse characteristics
.ac type, points, from, to	AC sweep to analyse frequency response
.tran stoptime	Transient small-signal analysis to analyse linearity *(Initialize frequency of source before using)
.opt	Print value of operating points of given circuit *(Initialize static source values before using for correct outputs)
SPICE Commands (included only the once used in this book... There are more...Refer to the “Help topics” included in the software)	
.step parameter_name, from, to, increment	Used for parametric analysis on component variable
.model model_name model_type (parameters=values)	Used to initialize parameter values of components of specific model
.include (or .lib) file.txt	Used to include model files or model libraries *(make sure to keep the model file in the same directory where the schematics are saved)
.param	Used for initialising parameters common to multiple components *(initialize the component parameter as {Var name} in all components you want to set the parameter for)
.meas type var_name param_type eqn	Used to measure desired quantities of the given expression based on type Result of .meas can be viewed by view→spice-log

## Steps for simulation:

Common for All : Right click → Edit simulation Command → choose the type of analysis

## DC analysis :

choose the values as appropriate

Run the simulation and obtain the curve (a red probe indicates voltage output and a black one indicates current output)

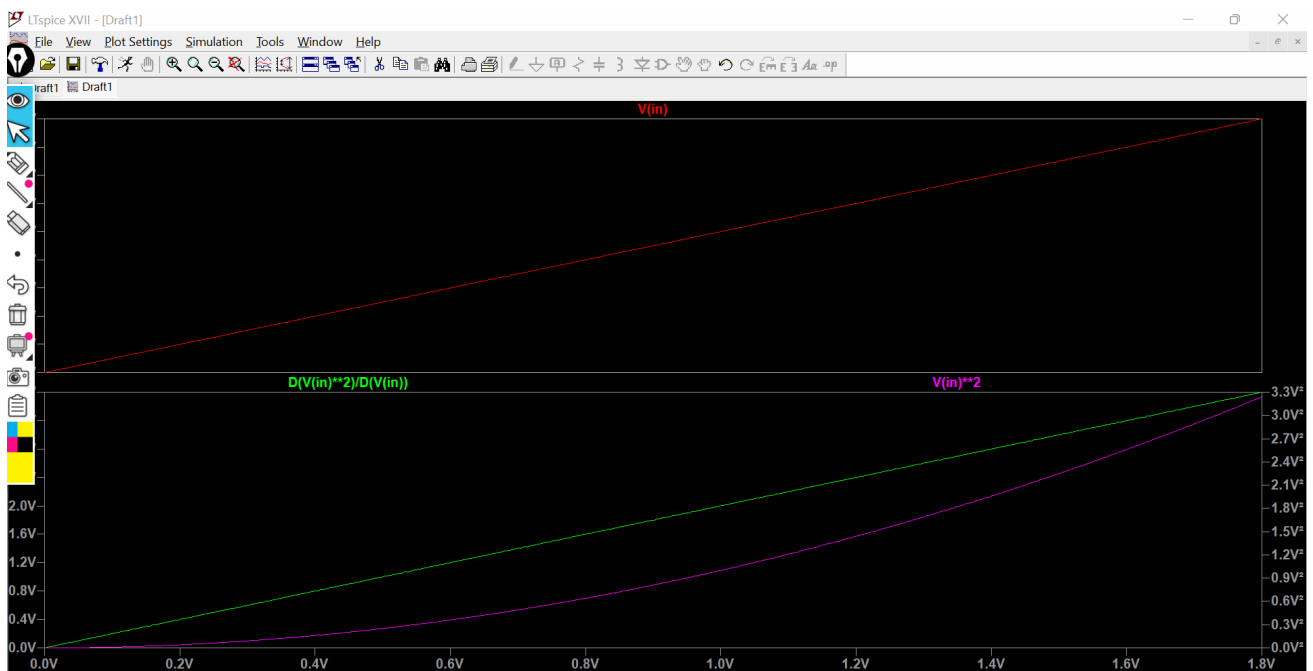
Sometimes you may want to add a parameter which is composed of equation of existing parameters (slope for example)

To do so, right-click on empty space on plot plane → click on add trace → add equation by typing in the equation

Basic syntax in equation editing :

- $D(A)/D(B)$  => differential of A with respect to B; (used to plot variation in slope for a given trace)
- $A**B$  => A to the power of B (used in algebraic expressions)

An example demonstrating above two syntax is shown below



# EXAMPLE CIRCUITS



# RC Circuits

## Problem statement:

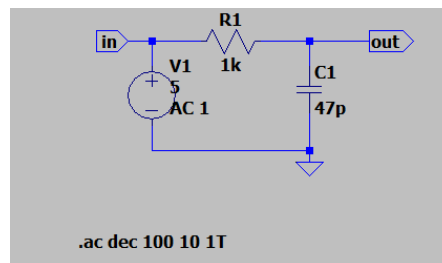
- a) verify that theoretical cut-off frequency and practical ( $F_c$ ) is equal for:
  - i) low pass filter
  - ii) high pass filter
- b) Design a bandpass circuit and find it's bandwidth

NOTE: Take  $V_{in}=5V, 600mV_{pp}$   $R=1k\Omega$   $C_1=47pF$   $C_2=94nF$

## Solution:

a)

i)



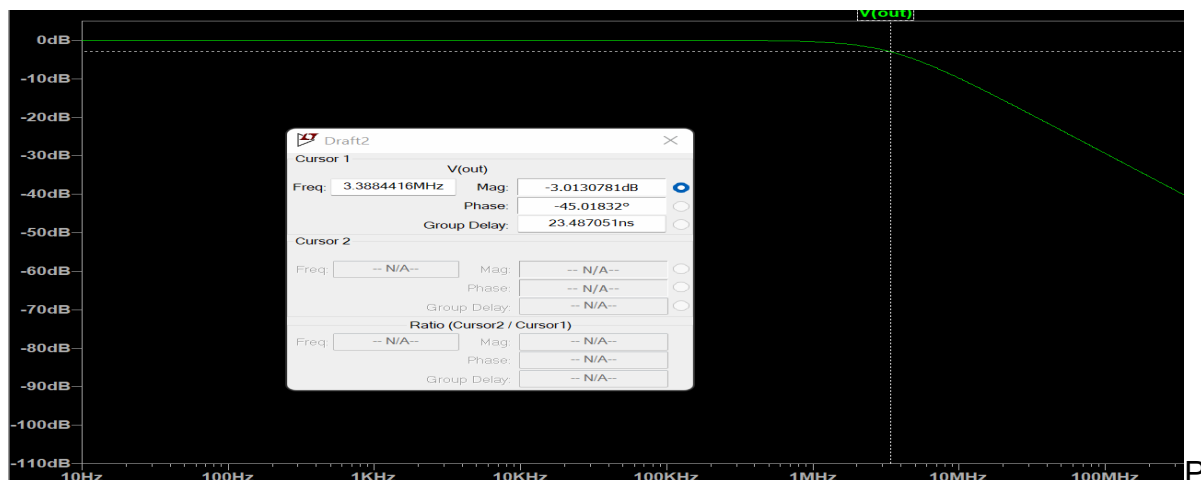
Doing the AC Analysis, we get the 3db frequency as 3.388MHz that gives  $F_c=3.388MHz$

Theoretically,  $F_c=1/2\pi R C$

$$F_c=1/(6.28 \cdot 1k \cdot 47p)$$

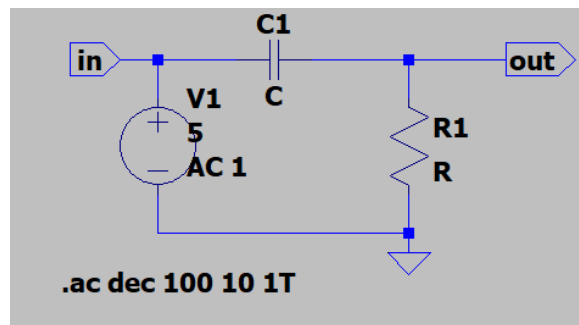
$$F_c= 3.3879MHz$$

## Frequency response



a)

ii)

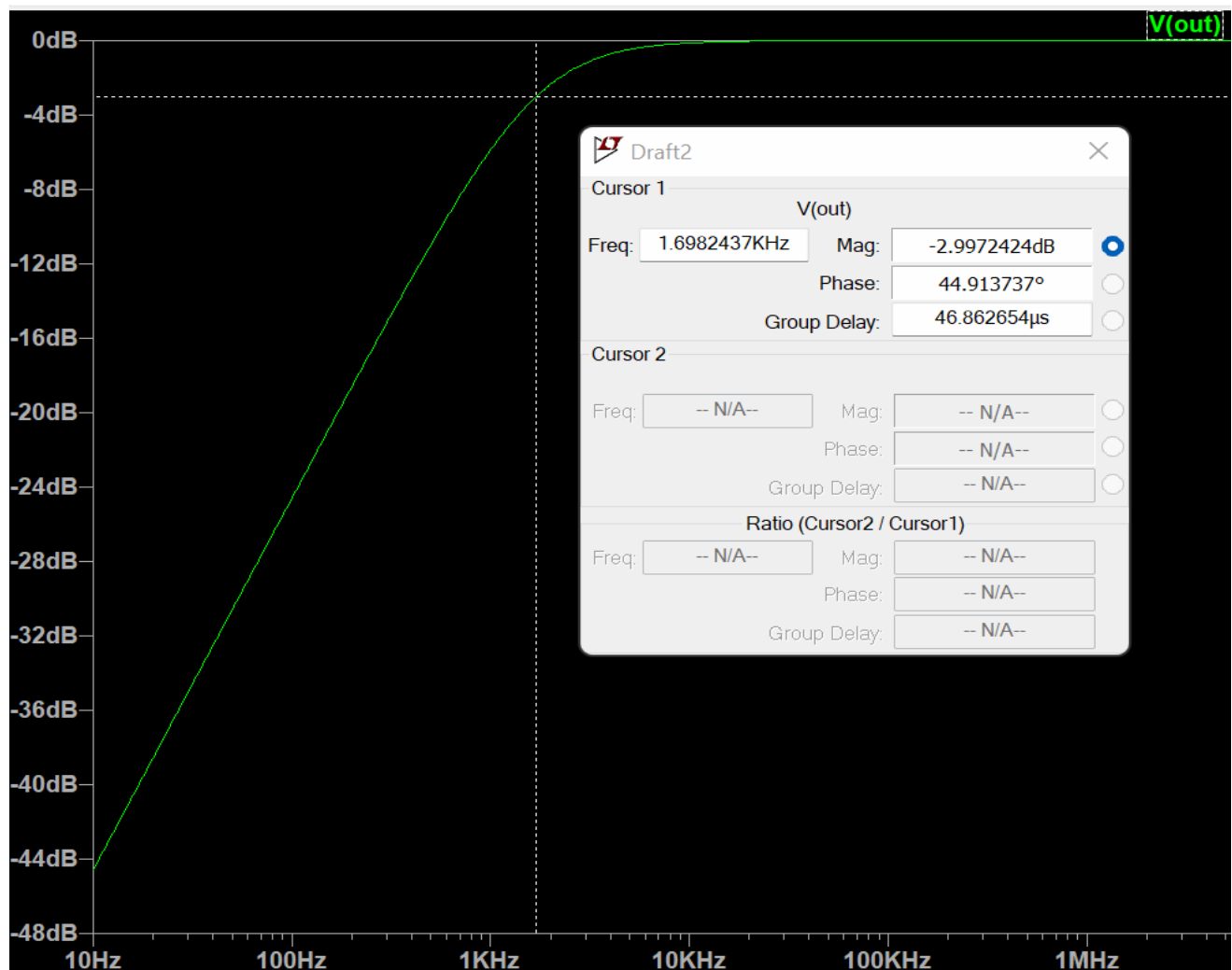


Doing the AC analysis we get  $F_c = 1.698\text{KHz}$

Theoretically,  $F_c = 1/(2\pi RC)$

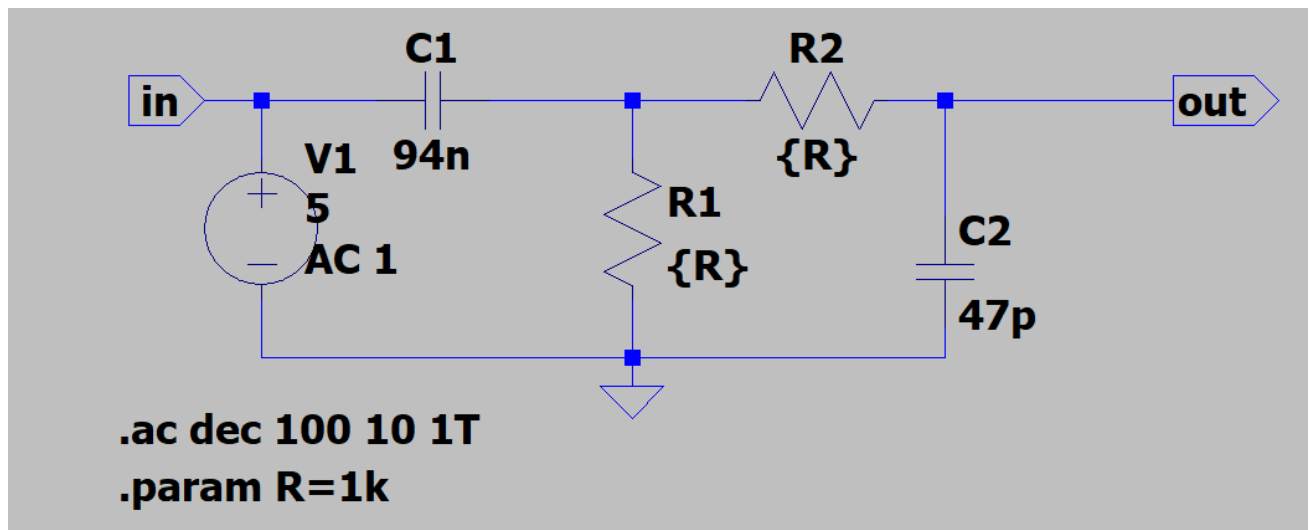
$$F_c = 1/(6.28 \cdot 1\text{k} \cdot 94\text{n})$$

$$F_c = 1.693\text{kHz}$$



b)

Cascading a Low-pass and High-pass we get a Band-Pass filter



Plotting the frequency response, we get  $F_h=3.388\text{MHz}$  and  $F_l=1.698\text{KHz}$

$BW = F_h - F_l = 3.388\text{M} - 1.698\text{K} \Rightarrow BW=3.386\text{MHz}$



# RL circuits

## Problem statement:

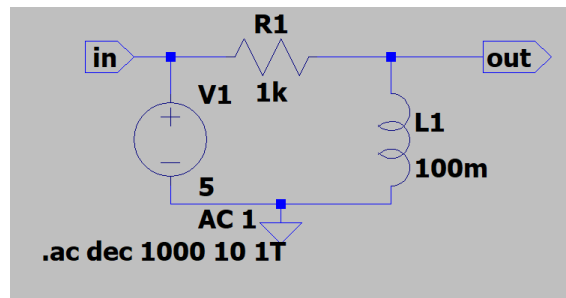
- a) verify that theoretical cut-off frequency and practical ( $F_c$ ) is equal for:
- high pass filter
  - low pass filter
- b) Design a bandpass circuit and find it's bandwidth

NOTE: Take  $V_{in}=5V, 600mV_{pp}$ ;  $R=1k\Omega$ ;  $L_1=100mH$ ;  $L_2=1mH$

## Solution:

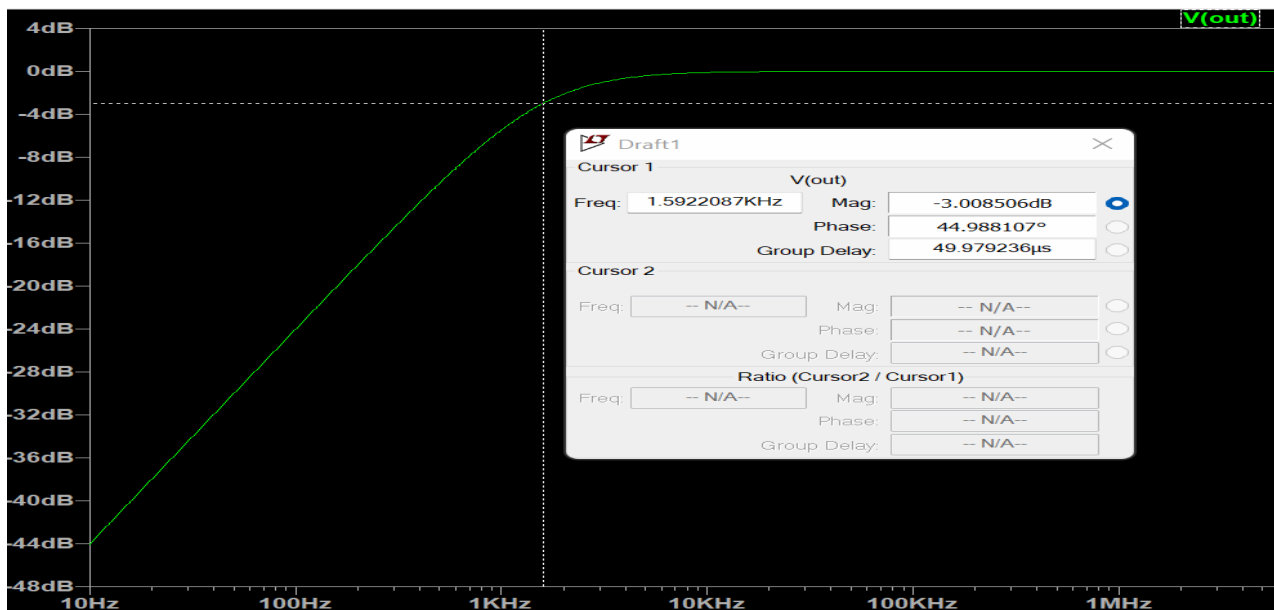
a)

i)



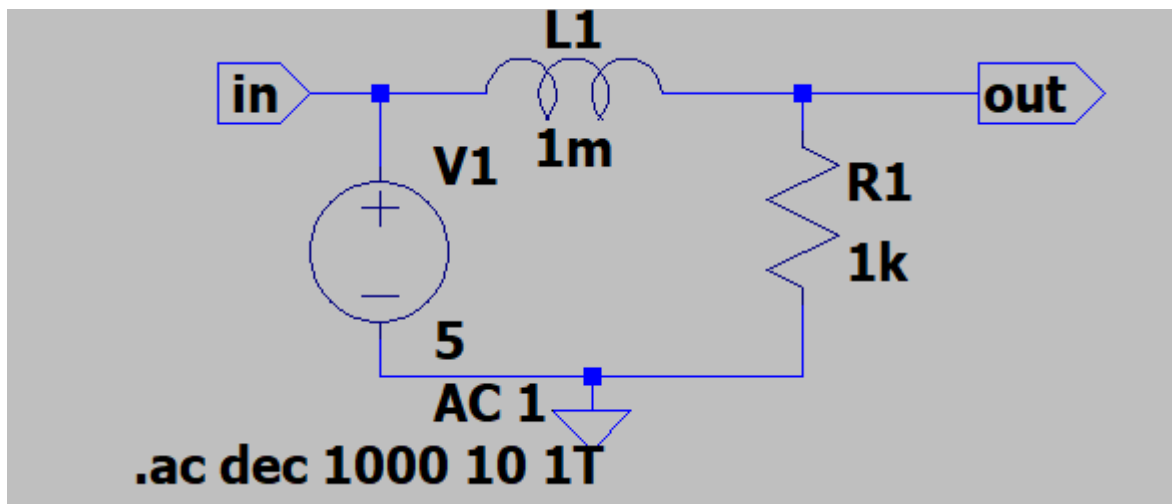
Theoretically,  $F_c = R / (2\pi L) = 1k / (2\pi \cdot 100m) \Rightarrow F_c = 1.592KHz$

By plotting frequency response as below we get  $F_c = 1.592KHz$



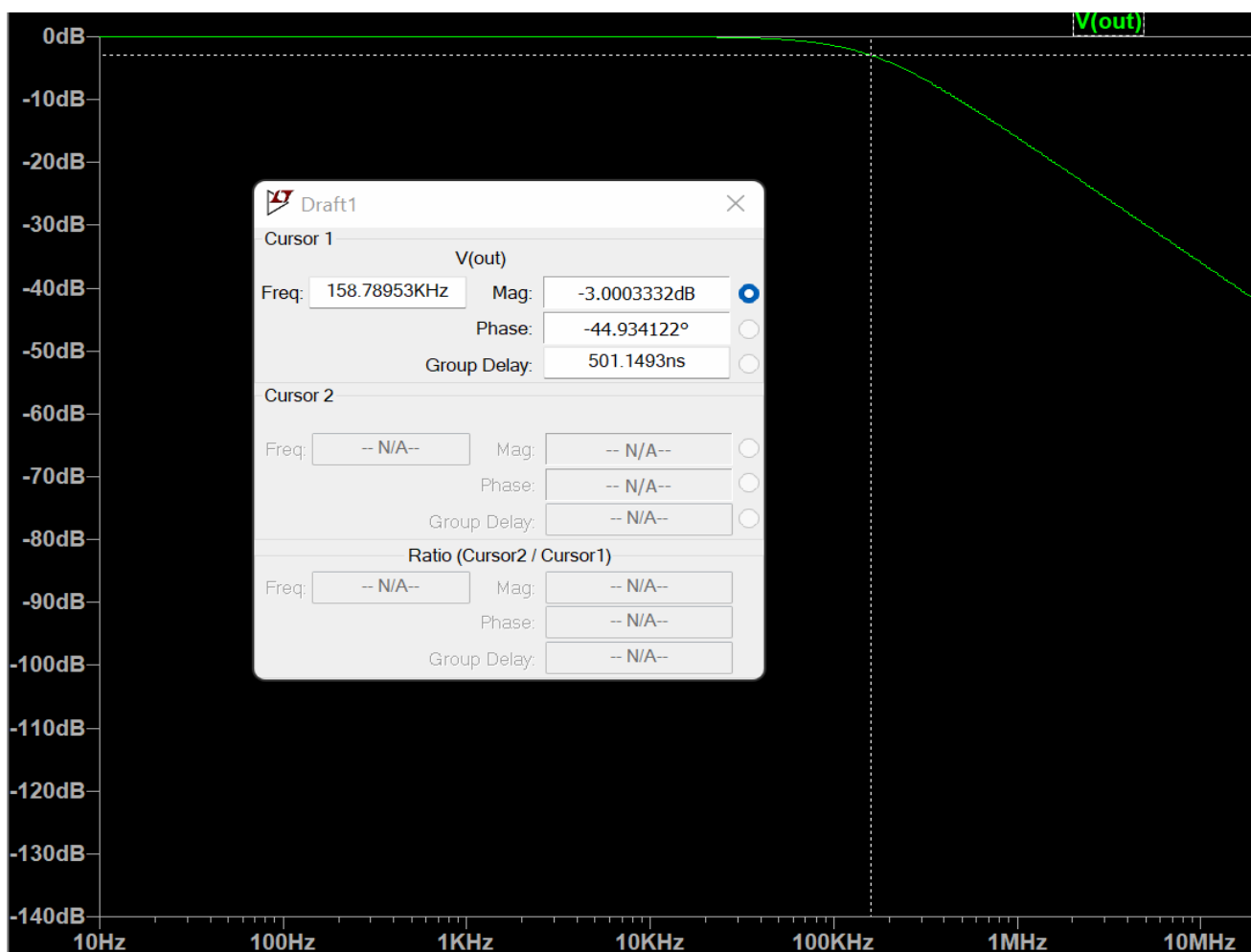
a)

ii)



Theoretically,  $F_c = R / (2\pi L) = 1k / (2\pi \cdot 1m) \Rightarrow F_c = 159.2KHz$

By plotting frequency response as below we get  $F_c = 158.79KHz$

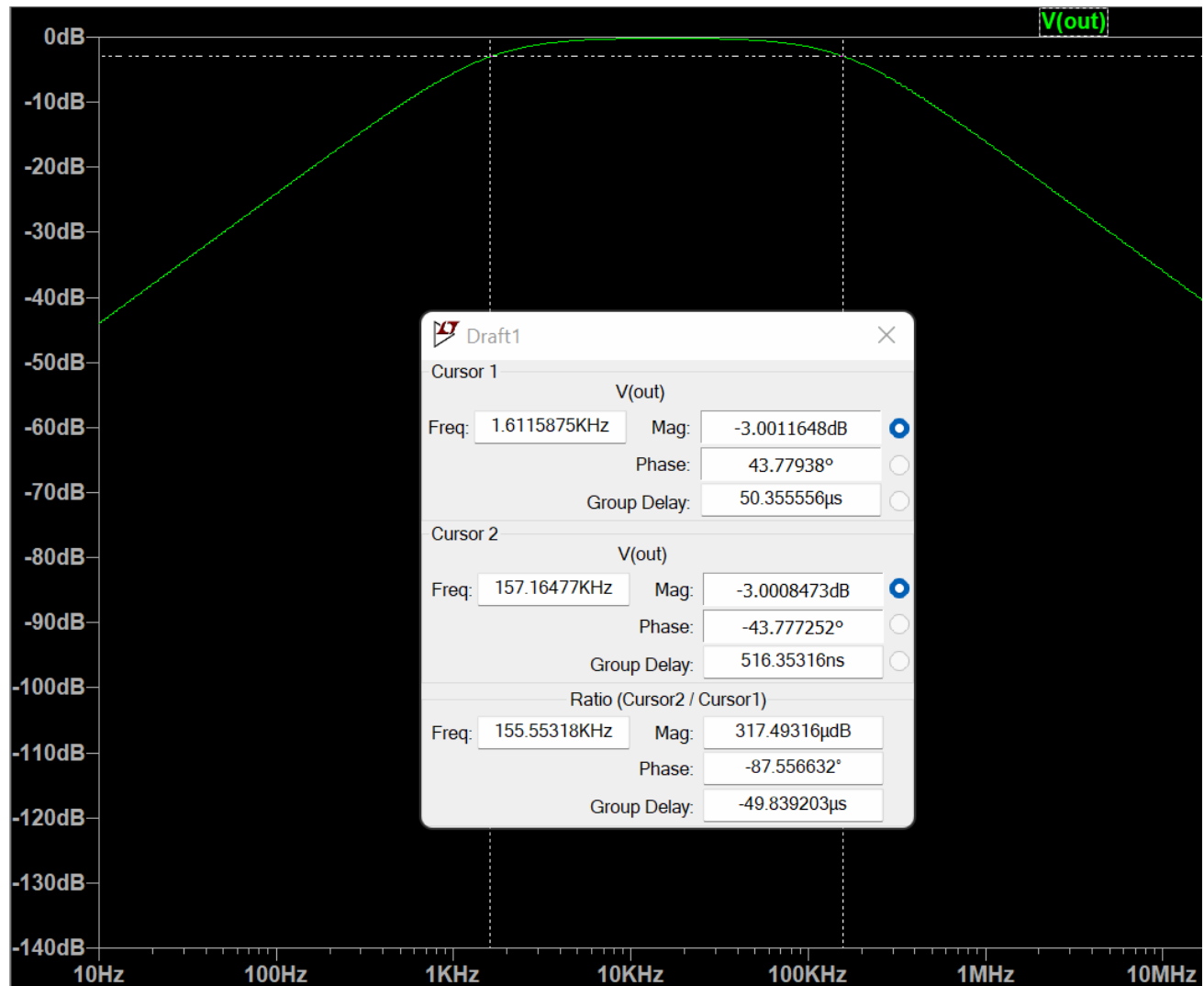


b)

ii)

Plotting the frequency response of the bandpass filter, we get

$$BW = F_h - F_l = 157.16\text{K} - 1.61\text{K} \Rightarrow \underline{BW = 155.55\text{KHz}}$$

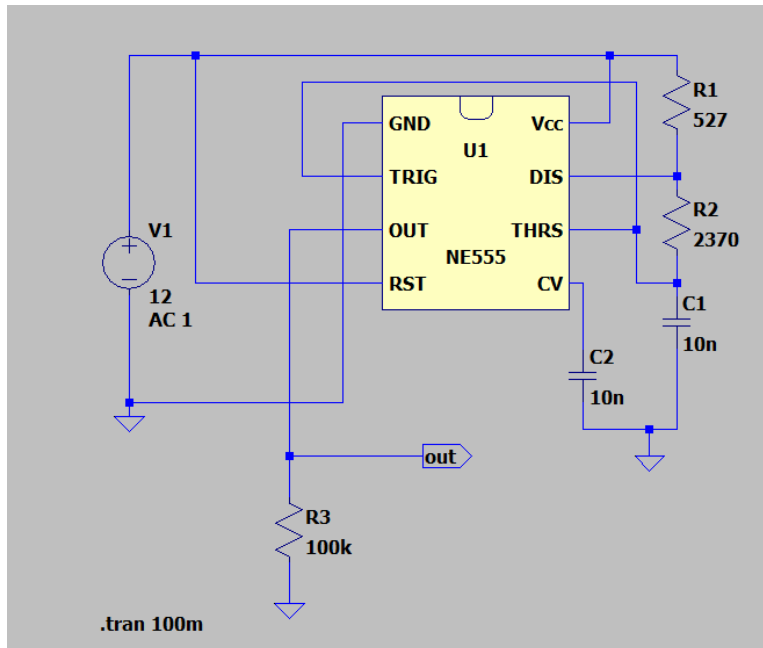


# Astable multivibrator

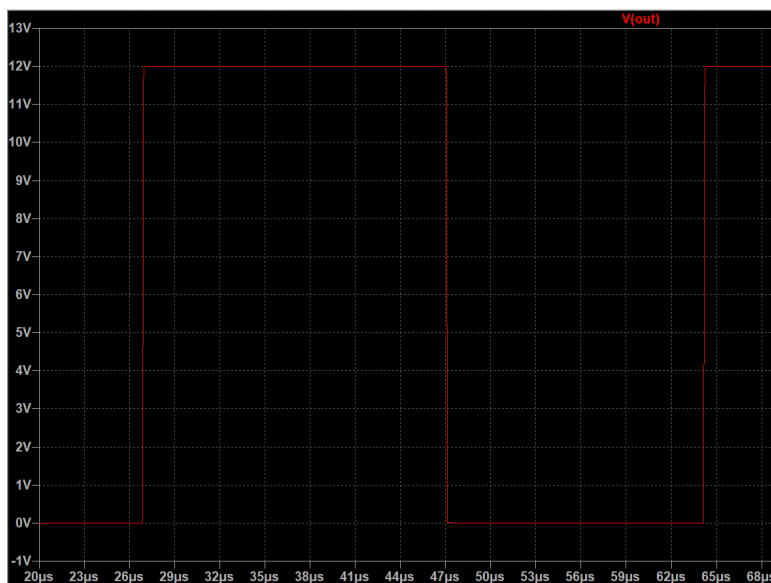
**Problem statement:** Design an astable Multivibrator and show the output on a waveform. Calculate  $T_h$  (time in high state),  $T_l$  (Time in low state) and its frequency of oscillations from the output.

**Solution:**

Schematic is shown below



Waveform:



## Diode Circuits Part A

---

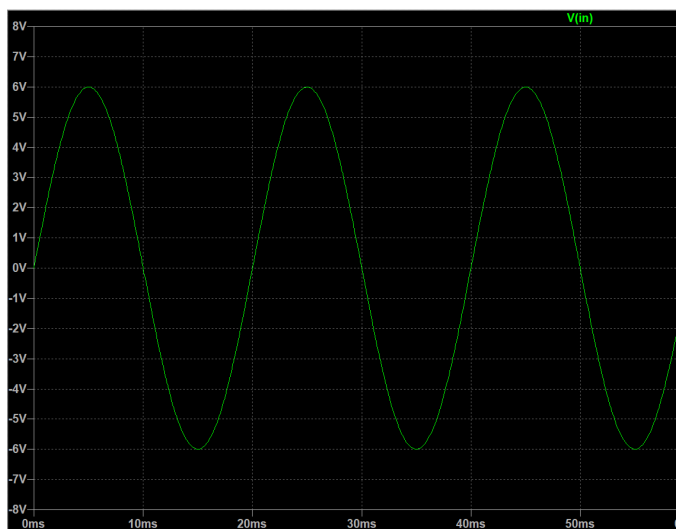
Problem statement: Design the following and show the output waveform for each of them:

- i) Positive Clipper
- ii) Negative Clipper
- iii) Double ended clipper

Note: take Source = 12Vpp 50Hz and R=1k. Use 1N4148

Solution:

Input Wave :

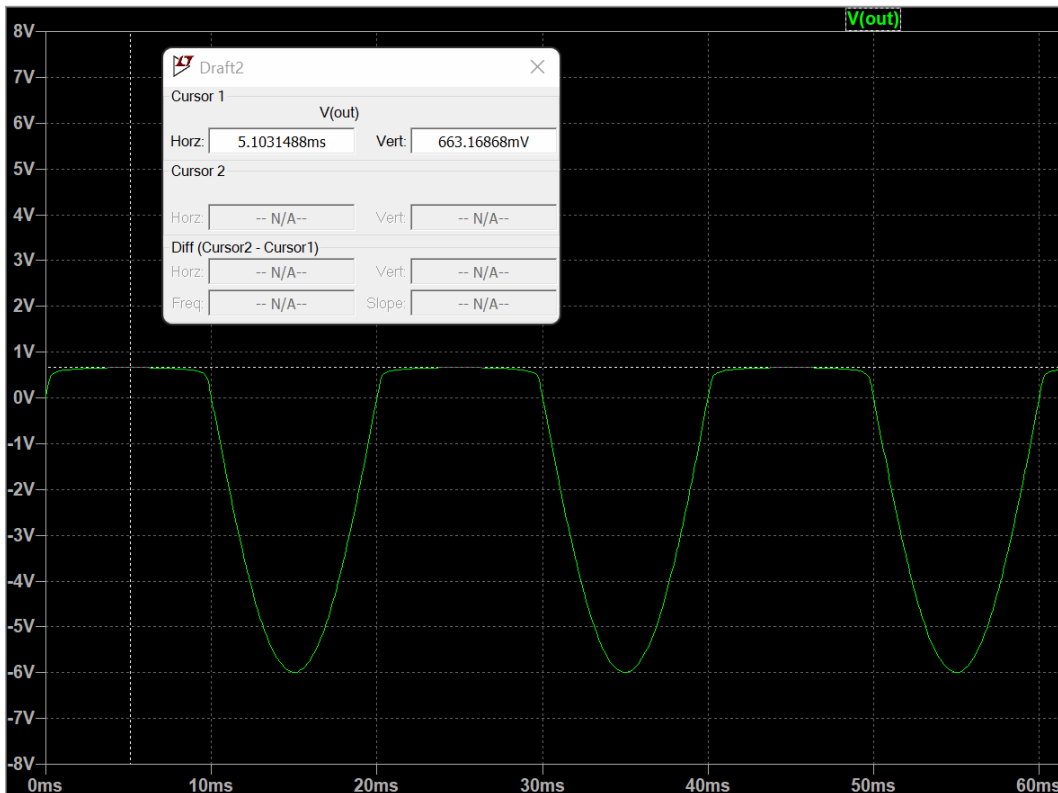
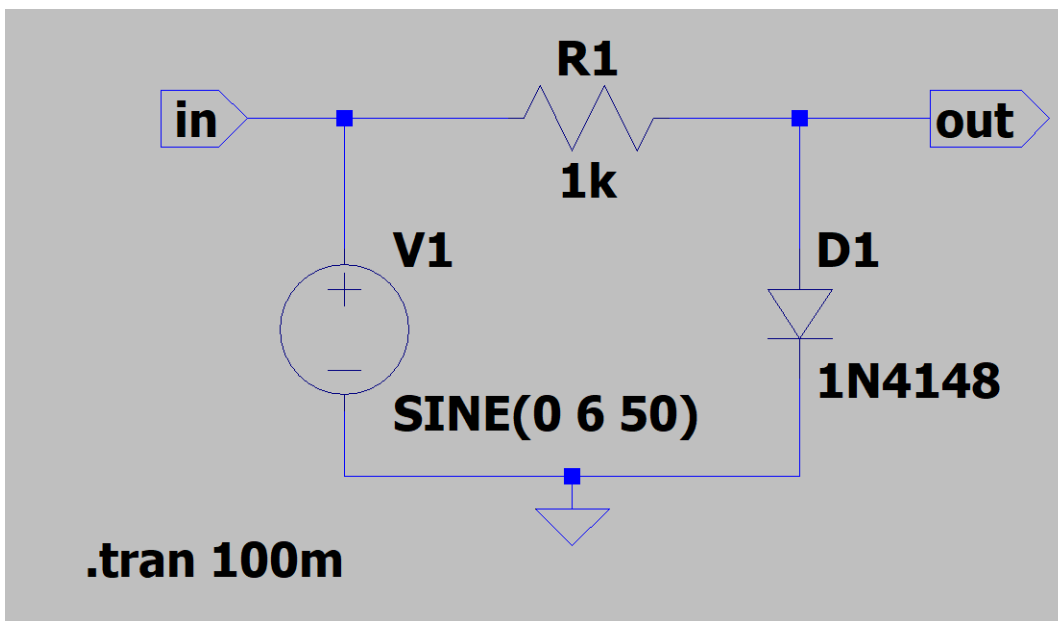


⚠ Note: Here we use 1N4148 which is a silicon diode. As you know, Silicon has a threshold of 0.7V approximately. So we can see the clipping at +/- 0.7V and not 0.

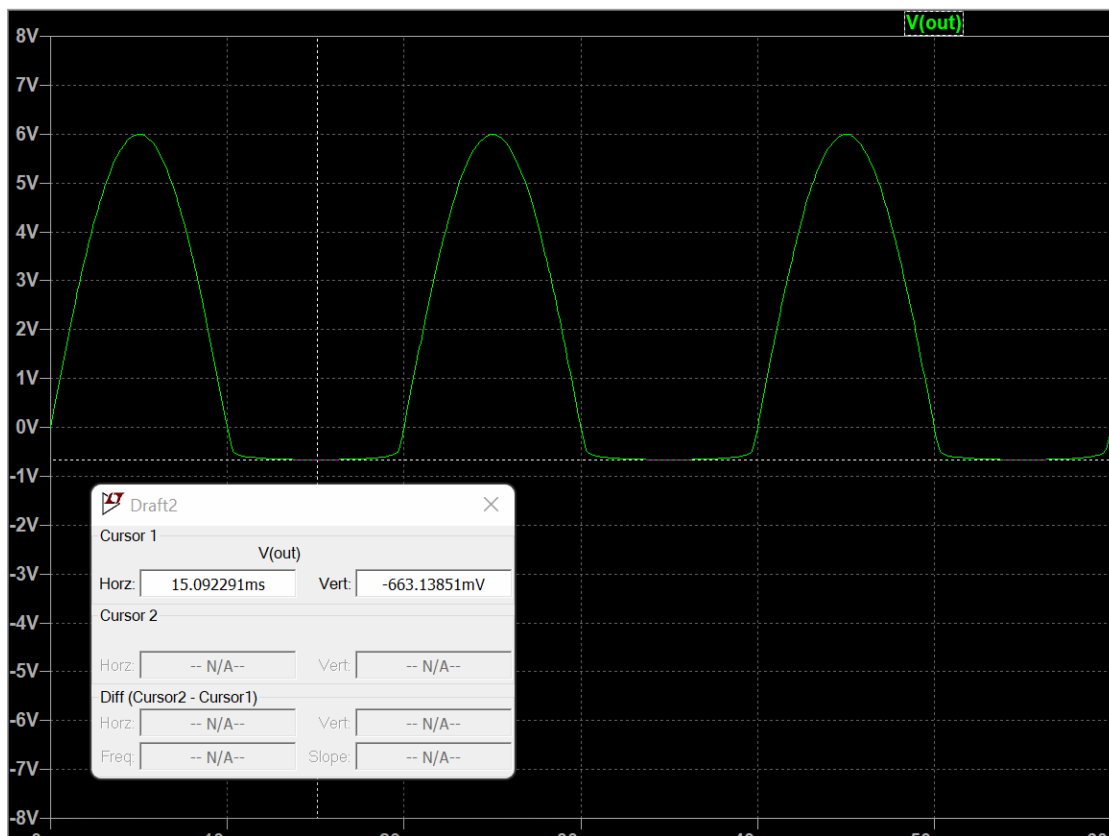
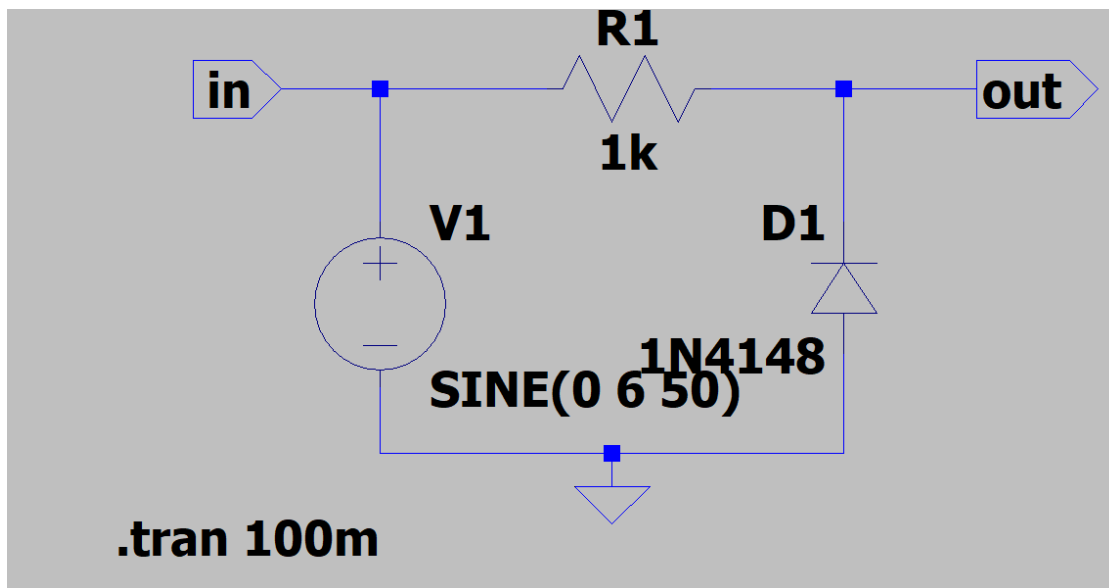
This in double ended clipping at high frequency is used in generating pulsating voltage/current depending on necessity and the frequency of rectified wave is adjusted to clock the digital circuits (Rectifiers shown in next section). ⚠



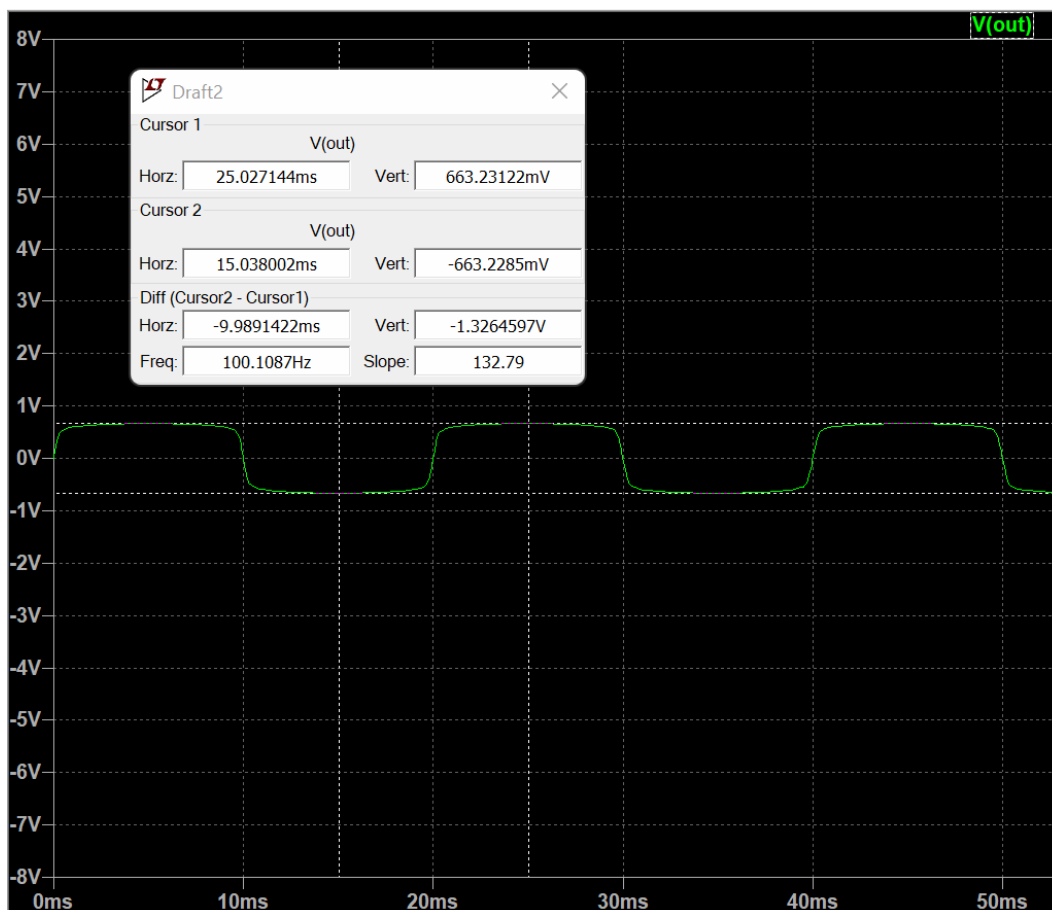
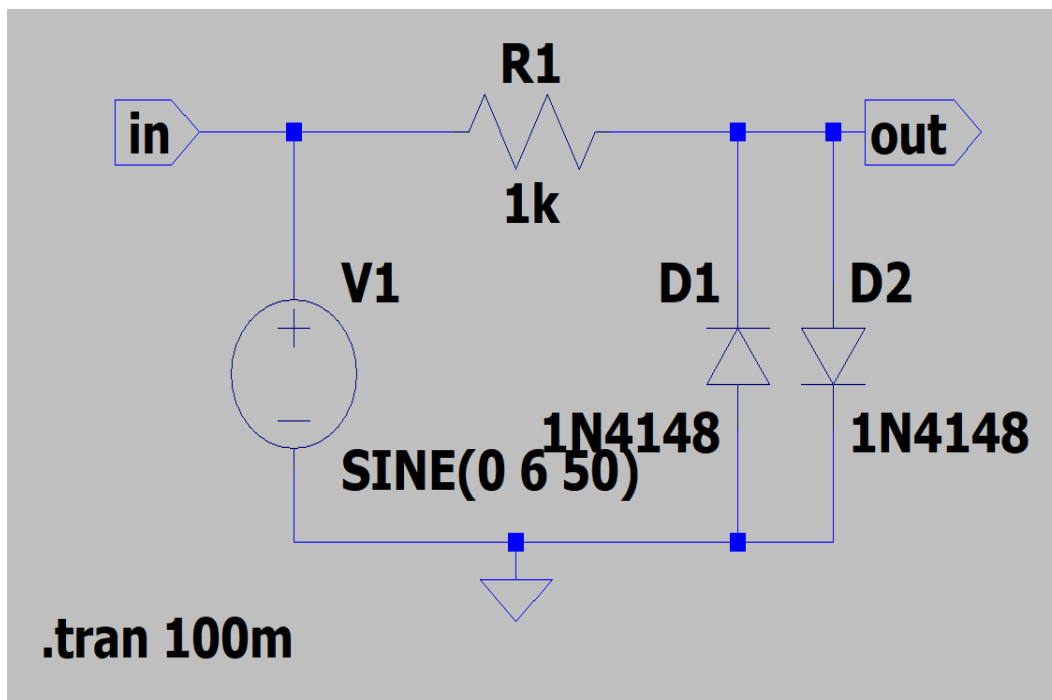
i) Positive clipping



ii) Negative clipping



iii) Double ended clipping

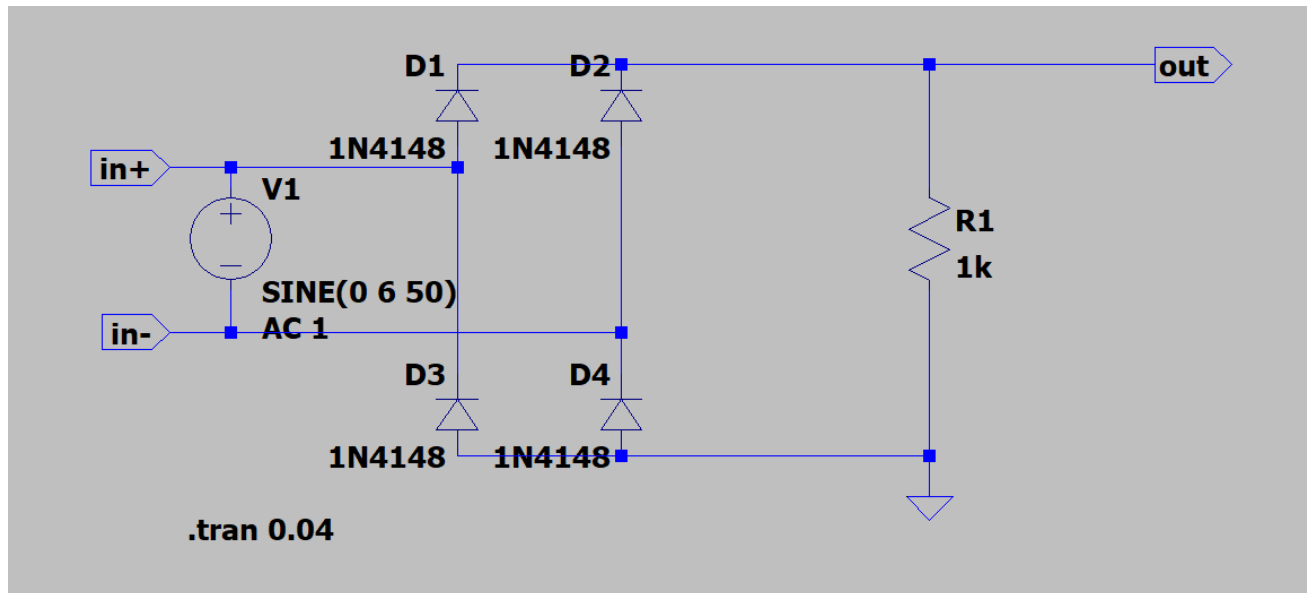


## Diode Circuits Part B

Problem statement: Design a Bridge Rectifier and compare input and output frequencies.

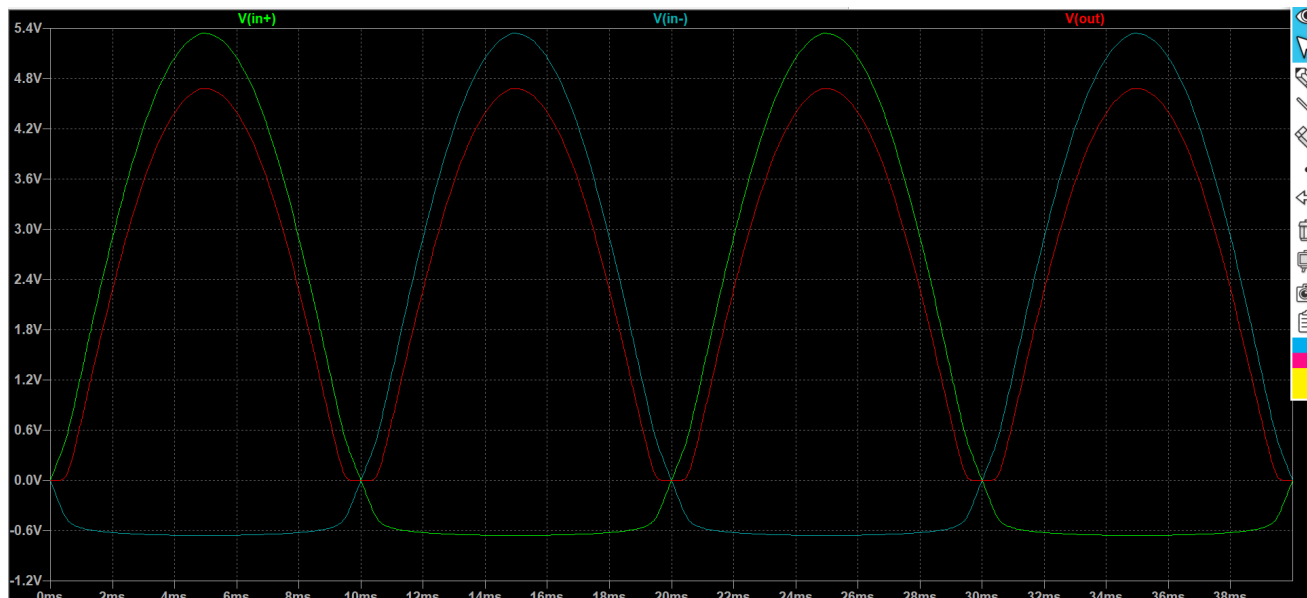
Note: Use  $R=1k$ , Diode part: 1N4148,  $V_{in}=12V_{pp}$ ,  $F_{in} = 50Hz$

Solution:



Waveform:

$V(in+)$  = Positive half cycle;  $V(in-)$  = Negative half cycle;  $V(out)$  = output wave

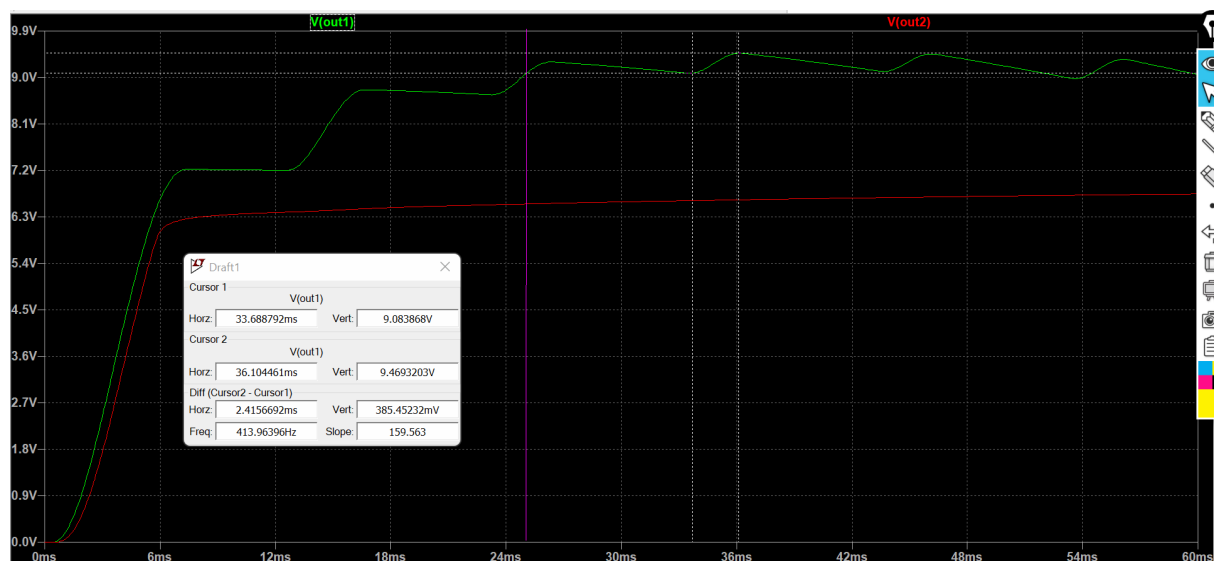
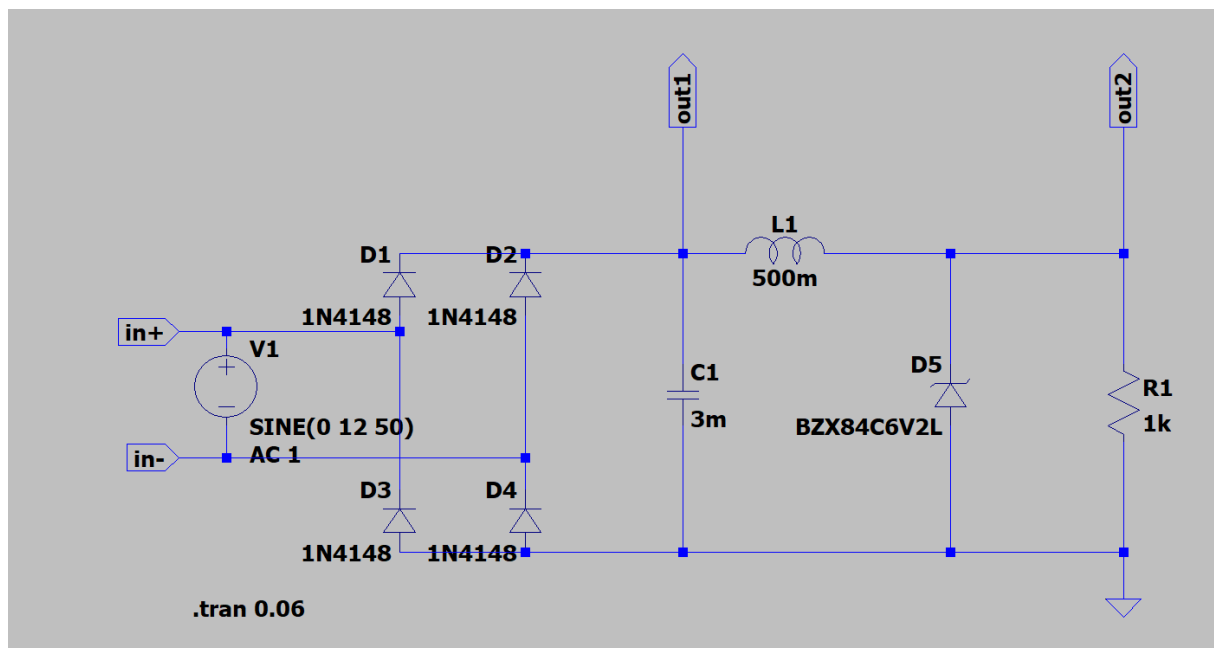


Clearly, the rectifier is active for both positive and negative half cycle. So, the frequency would be double the input i.e. 100Hz

# Project: AC to DC converter using Bridge rectifier and filters

**Problem statement:** Construct a AC to DC converter using the rectifier designed in previous section, AC and DC filters and regulator. Show the output at each stage. regulated at 6.2V (Use Zener diode model BZX84C6V2L)

**Solution:** By now it should be clear that Transient equivalent (response to AC) of capacitors are bypass and steady equivalent (response to DC) of inductor are bypass. Therefore, we connect capacitor in parallel and inductor in series. For Regulators, we use Zener diode in reverse bias conditions so as to mirror the Zener breakdown voltage at the output voltage



# BJT Circuits

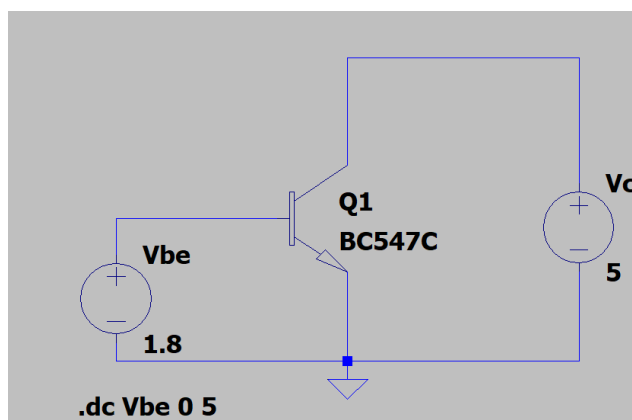
Problem statement: Use BC547C for all the following circuits

- a) Verify the IV characteristics of an NPN BJT
- b) Design a CE amplifier and calculate the gain in each of the following analysis:
  - i) DC analysis
  - ii) AC analysis
  - iii) Transient analysis
- c) Construct a CE differential amplifier , plot  $\Delta V_{out}$  and  $V_{in}$ , and calculate the double-ended gain

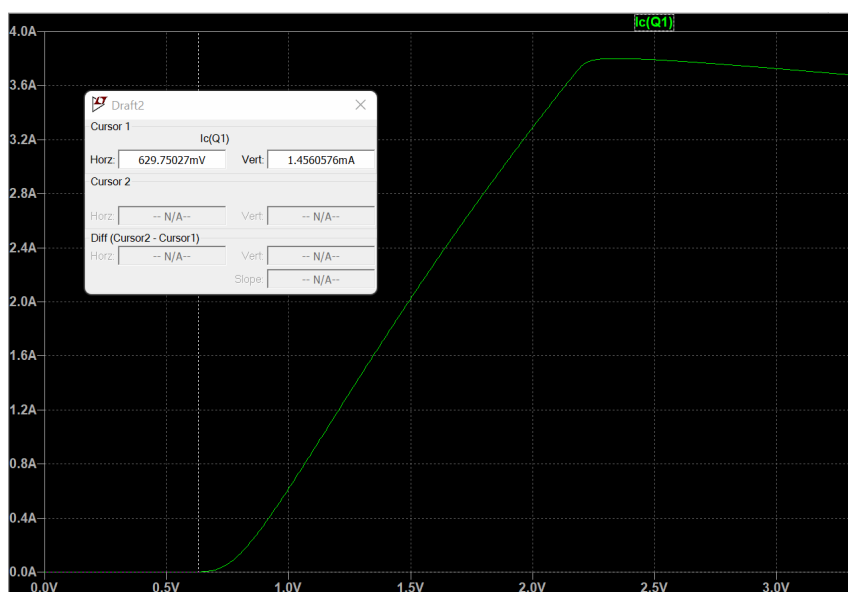
Note: Take  $R_c = 1.2k, R_e = 220, V_{cc} = 5V$

Solution:

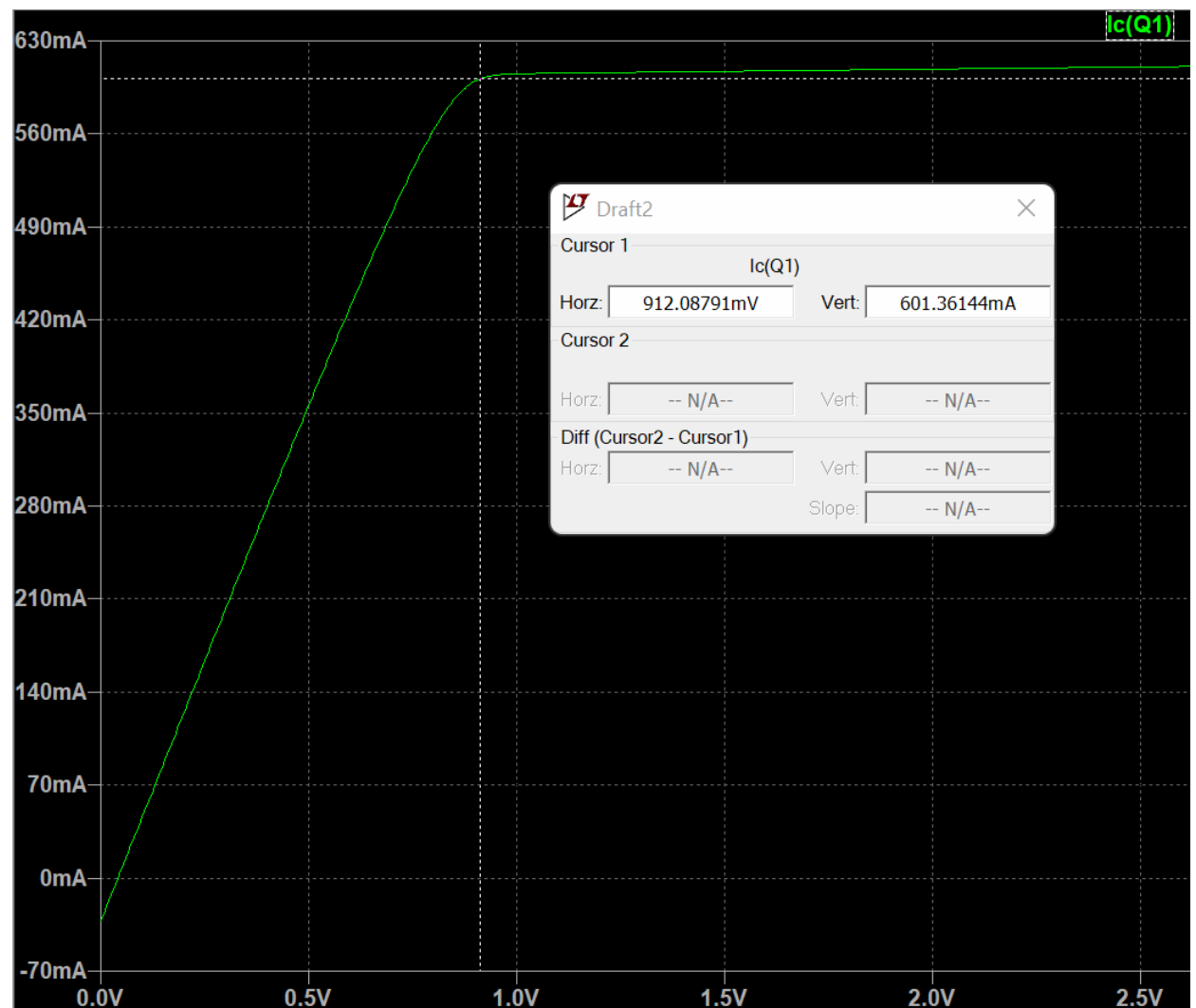
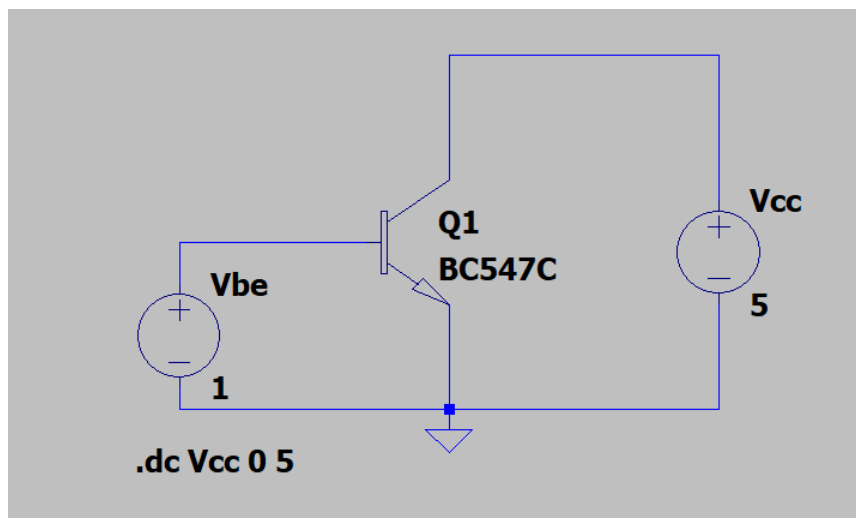
a) Input characteristics



From the DC response we find that  $V_{th} = 630mV$  approximately.

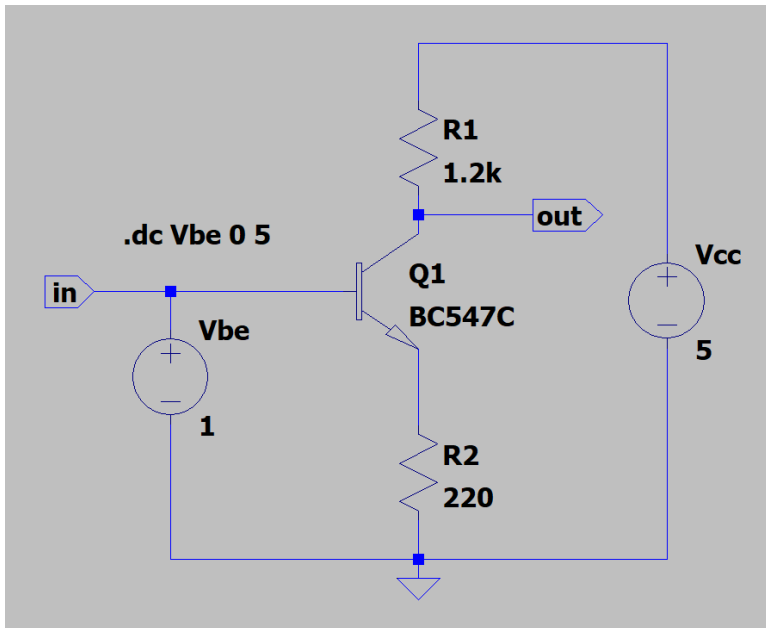


Output characteristics:



b) CE amplifier

i) DC analysis

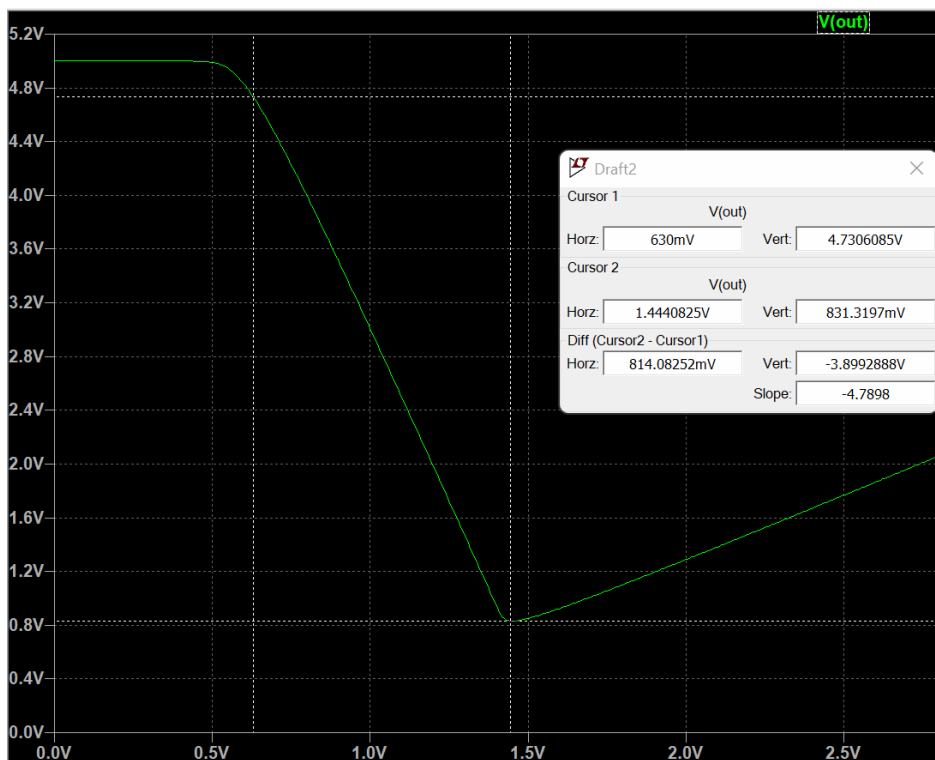


Taking average of the limits of active region,

$$V_{beq} = (630m + 1444m) / 2$$

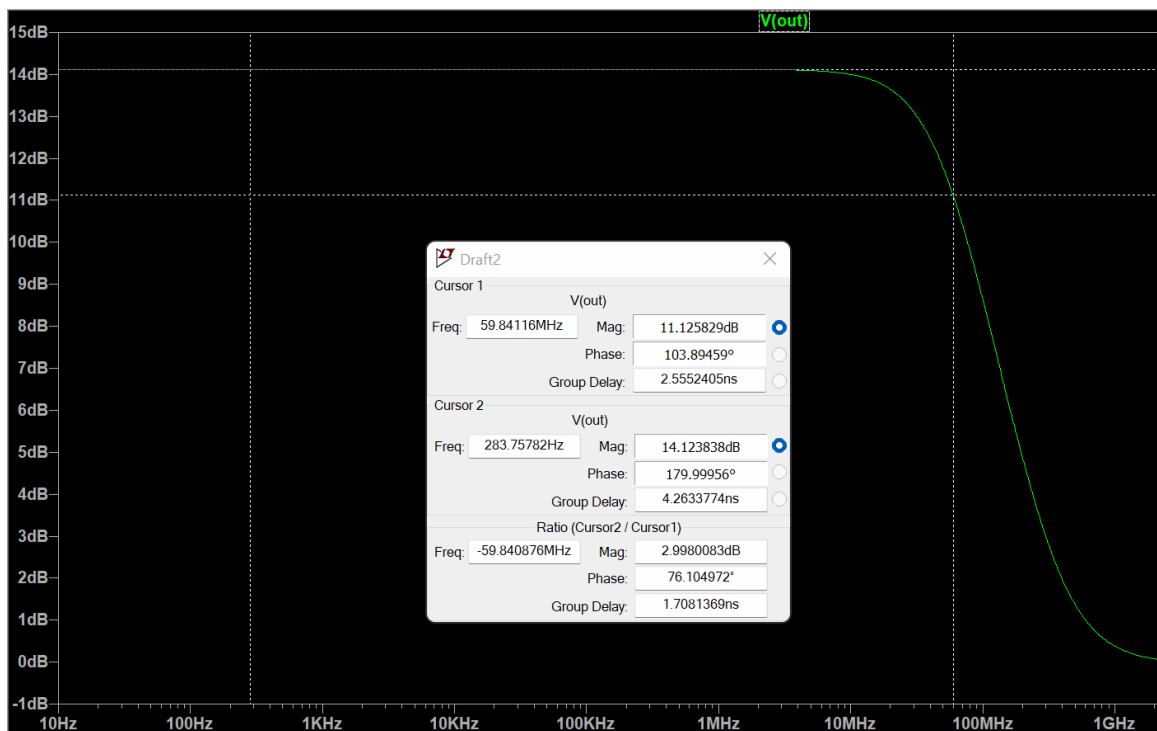
$$= 1037m = 1.037V$$

$$A_{dc} = -R_c / R_e = 1.2k / .22k > \underline{A_{dc} = 5.45}$$





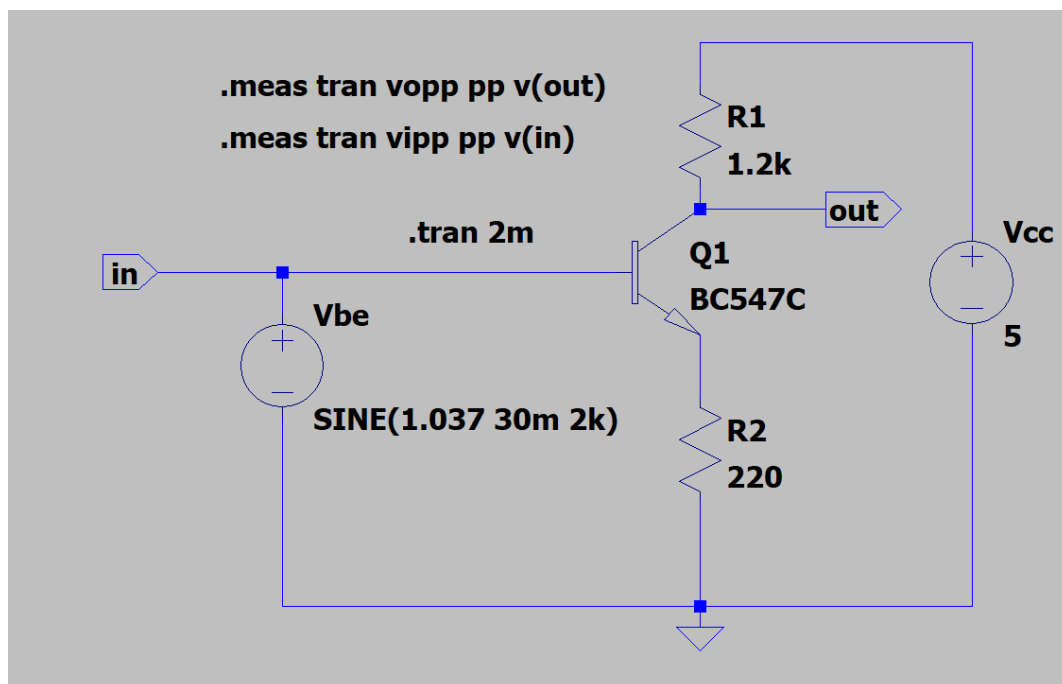
ii) AC analysis



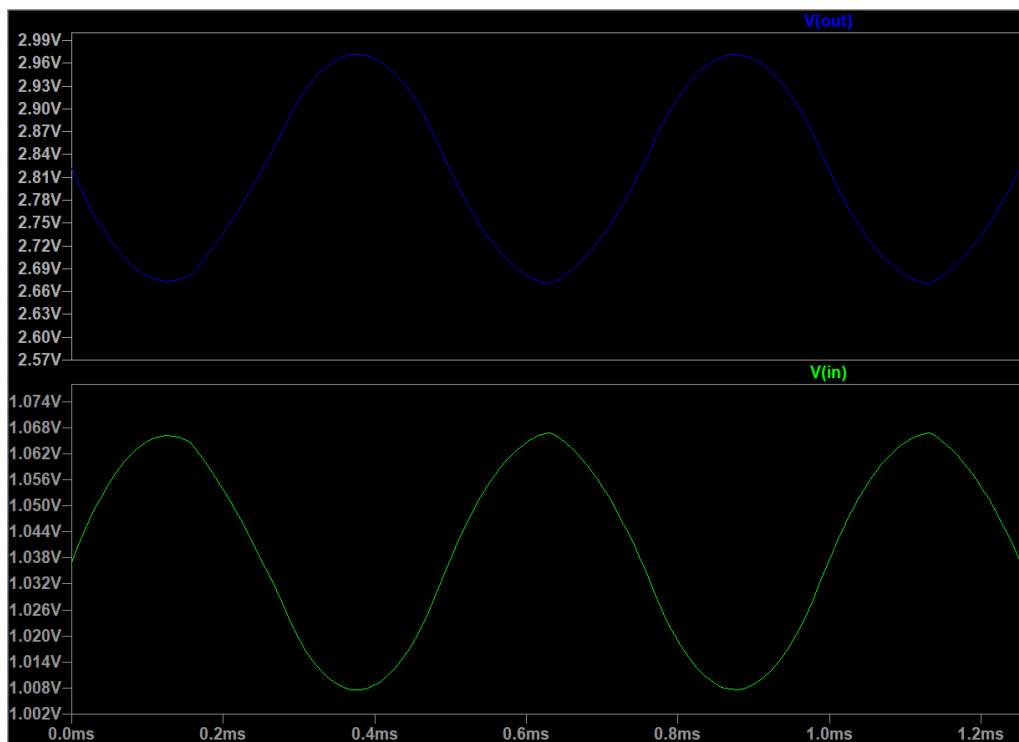
$$A(ac) = 14dB = 10^{(14.128/20)} \Rightarrow \underline{A(ac)=5.14}$$

$$\underline{F_c = 59.8MHz}$$

iii) Transient analysis



## Waveform

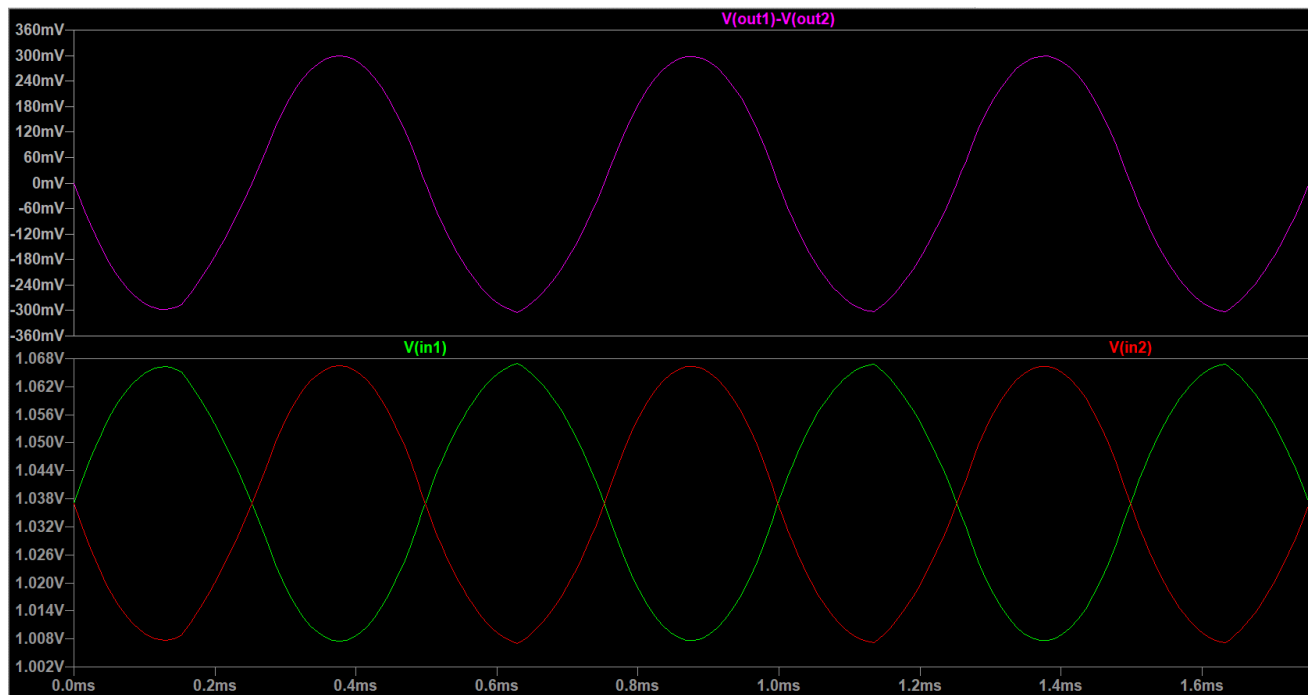
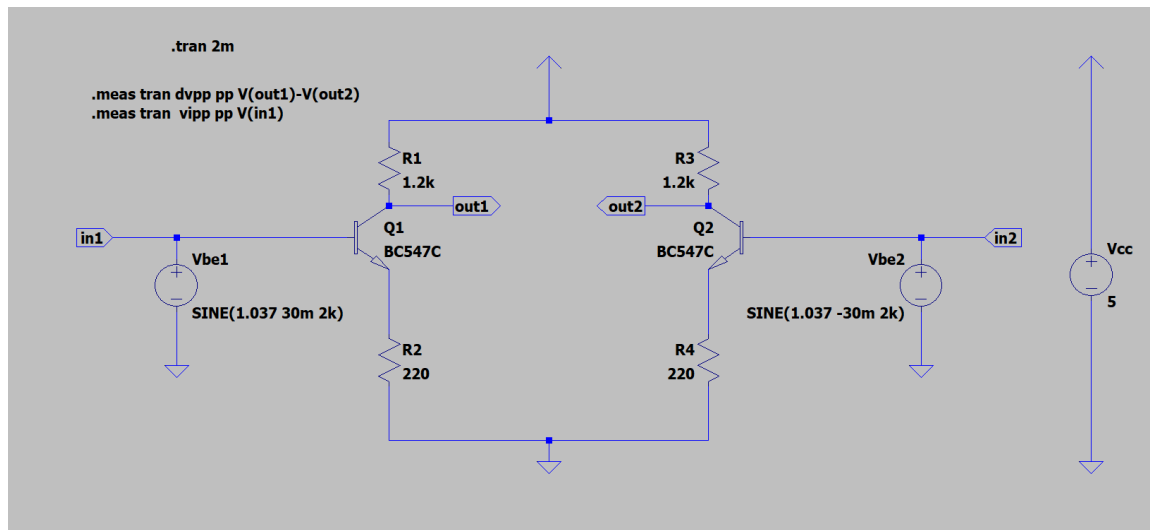


## SPICE log

```
SPICE Error Log: C:\Users\1107s\Documents\LTspiceXVII\Draft2.log
Circuit: * C:\Users\1107s\Documents\LTspiceXVII\Draft2.asc
Direct Newton iteration for .op point succeeded.
vopp: PP(v(out))=0.302517 FROM 0 TO 0.002
vipp: PP(v(in))=0.0595145 FROM 0 TO 0.002
Date: Tue Feb 01 16:09:45 2022
Total elapsed time: 0.066 seconds.
```

$$A(\text{tran}) = V_{\text{opp}}/V_{\text{ipp}} = 302.517\text{m}/59.514\text{m} \Rightarrow \underline{A(\text{tran})=5.083}$$

### c) Differential amplifier



### Calculation of gain

```
SPICE Error Log: C:\Users\1107s\Documents\LTspiceXVII\Draft2.log
Circuit: * C:\Users\1107s\Documents\LTspiceXVII\Draft2.asc
Direct Newton iteration for .op point succeeded.
dvpp: PP(v(out1)-v(out2))=0.605499 FROM 0 TO 0.002
vipp: PP(v(in1))=0.0595653 FROM 0 TO 0.002

Date: Tue Feb 01 16:45:22 2022
Total elapsed time: 0.051 seconds.
```

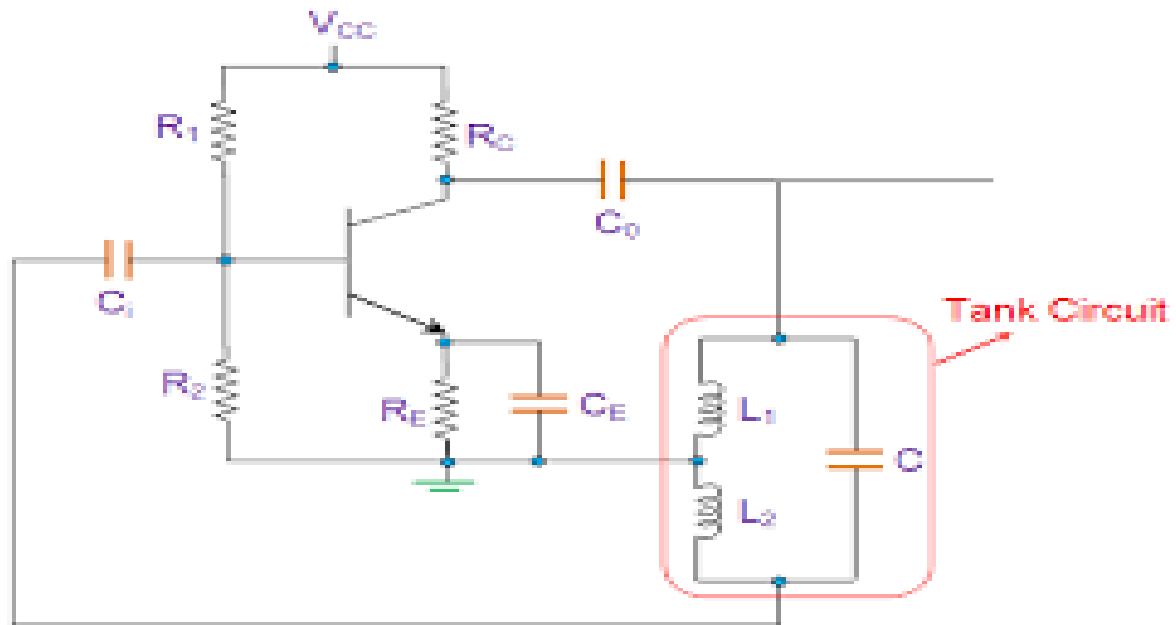
$$A(\text{de}) = dv_{pp}/v_{ip} = 605.5\text{m}/59.56\text{m}$$

$$\Rightarrow \underline{A(\text{de}) = 10.166}$$

⚠ Note that double ended gain of a differential amplifier with differential inputs is twice the gain of each half-amplifier of the whole system. ⚠

# Oscillators

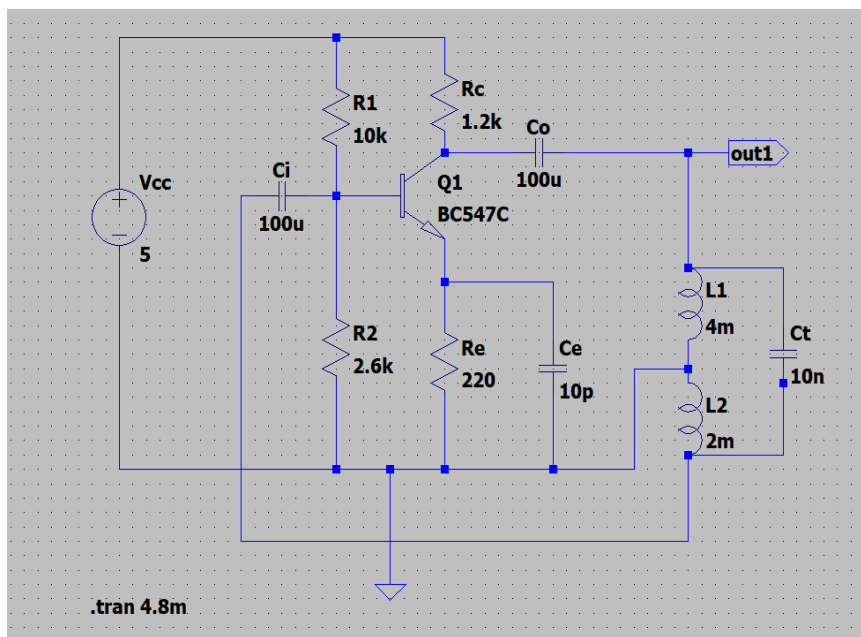
- a) Problem statement: Design a Hartley Oscillator as following and obtain the output waveform and spectrum



Design parameters:

- i)  $R_1=10k$ ;  $R_2 = 2.6k$ ;  $R_c=1.2k$ ;  $R_e=220$
- ii)  $C_i=C_o=100\mu F$ ;  $C_t=10nF$ ;  $C_e=10pF$
- iii)  $L_1=4mH$ ;  $L_2=2mH$
- iv)  $V_{cc}=5V$ , Transistor = BC547C

Solution:



Theoretical Frequency:

$$L = L1 + L2 = 6mH$$

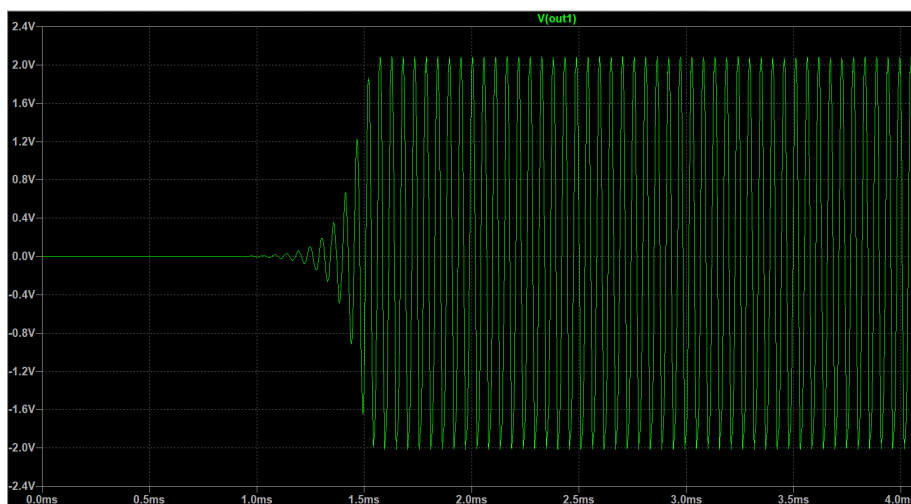
$$C = Ct = 10n$$

$$F = 0.15923 / \sqrt{LC}$$

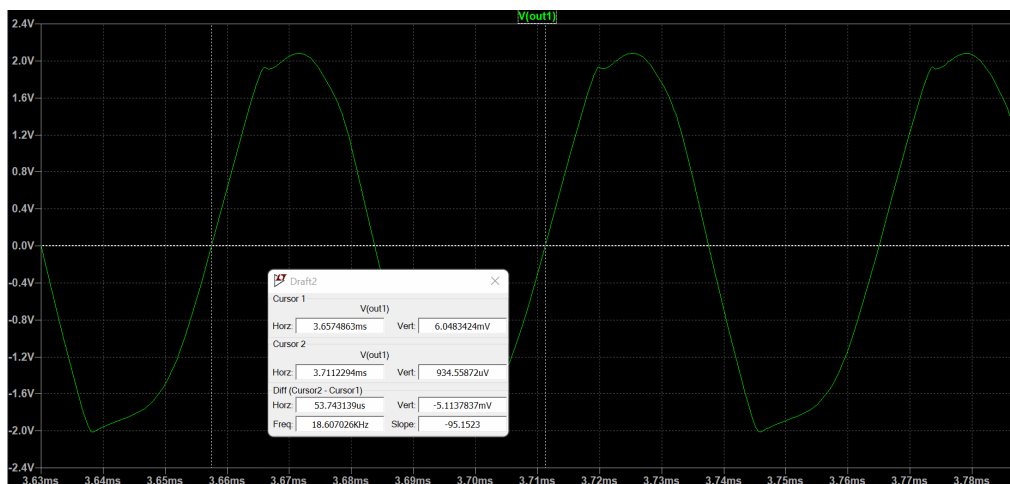
$$F = (0.15923) / (7.746u)$$

$$\Rightarrow \underline{F = 20.56KHz}$$

Original waveform

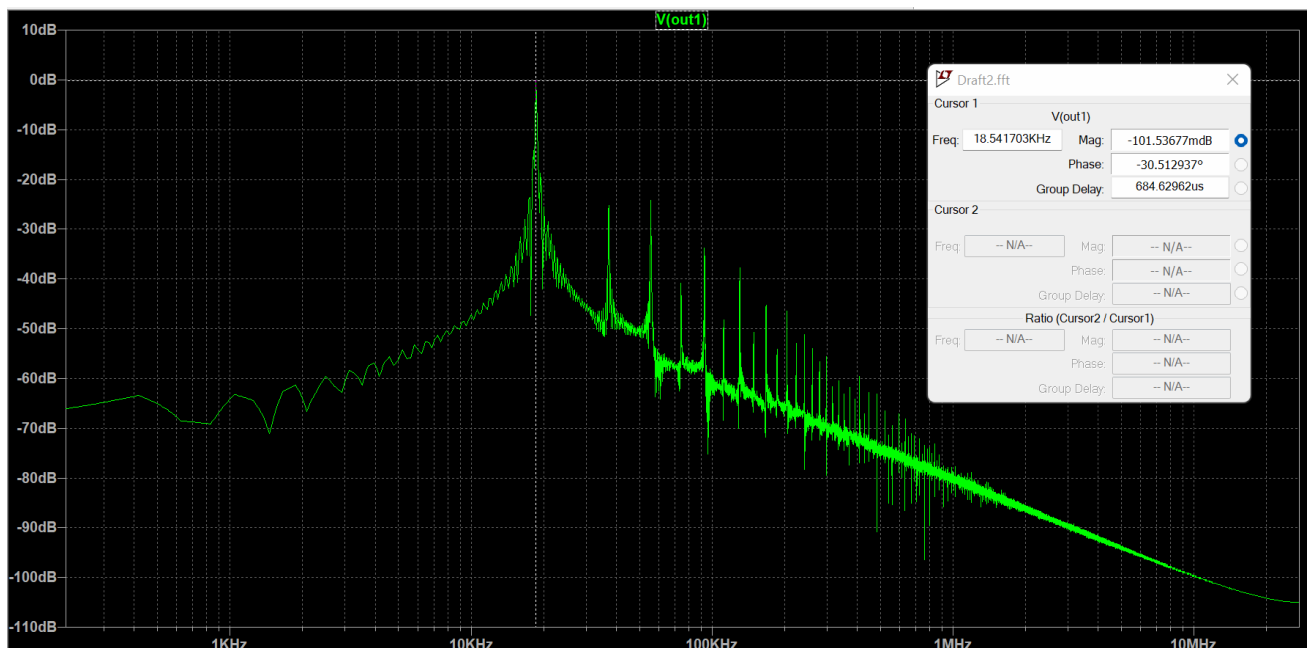


Frequency calculated from waveform:



The frequency obtained from the waveform is F=18.6KHz

Frequency calculation from spectrum:

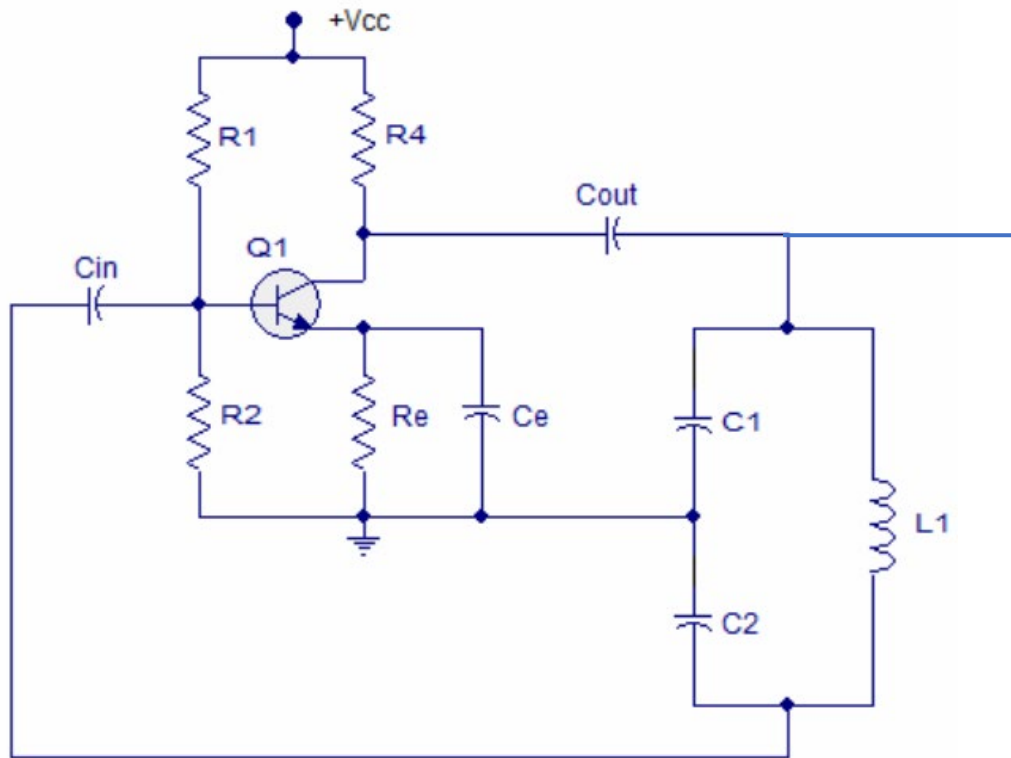


Frequency of oscillator obtained from spectrum is F=18.54KHz

# Oscillators

---

b) Problem statement: Design a Colpitts Oscillator as following and obtain the output waveform

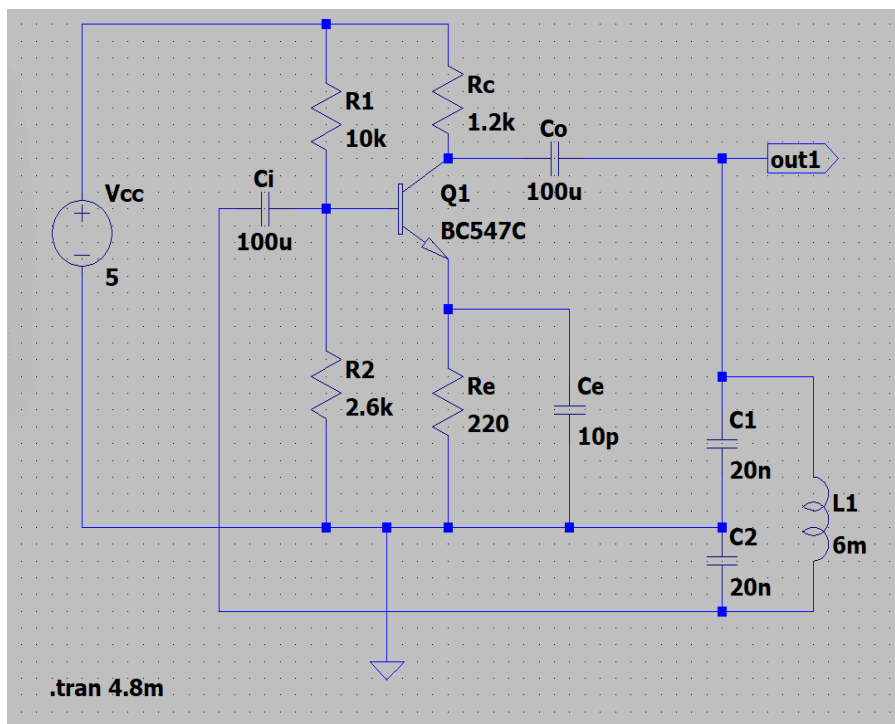


Design parameters:

- i)  $R1=10k$ ;  $R2 = 2.6k$ ;  $Rc=1.2k$ ;  $Re=220$
- ii)  $Ci=Co=100\mu F$ ;  $Ce=10pF$ ;  $C1=C2=20nF$
- iii)  $Lt=6mH$
- iv)  $V_{cc}=5V$ , Transistor = BC547C



## Solutions:



## Theoretical Frequency:

$$L=L_t=6\text{mH}$$

$$C=C_1 \cdot C_2 / (C_1 + C_2) = 20\text{n} \cdot 20\text{n} / 40\text{n}$$

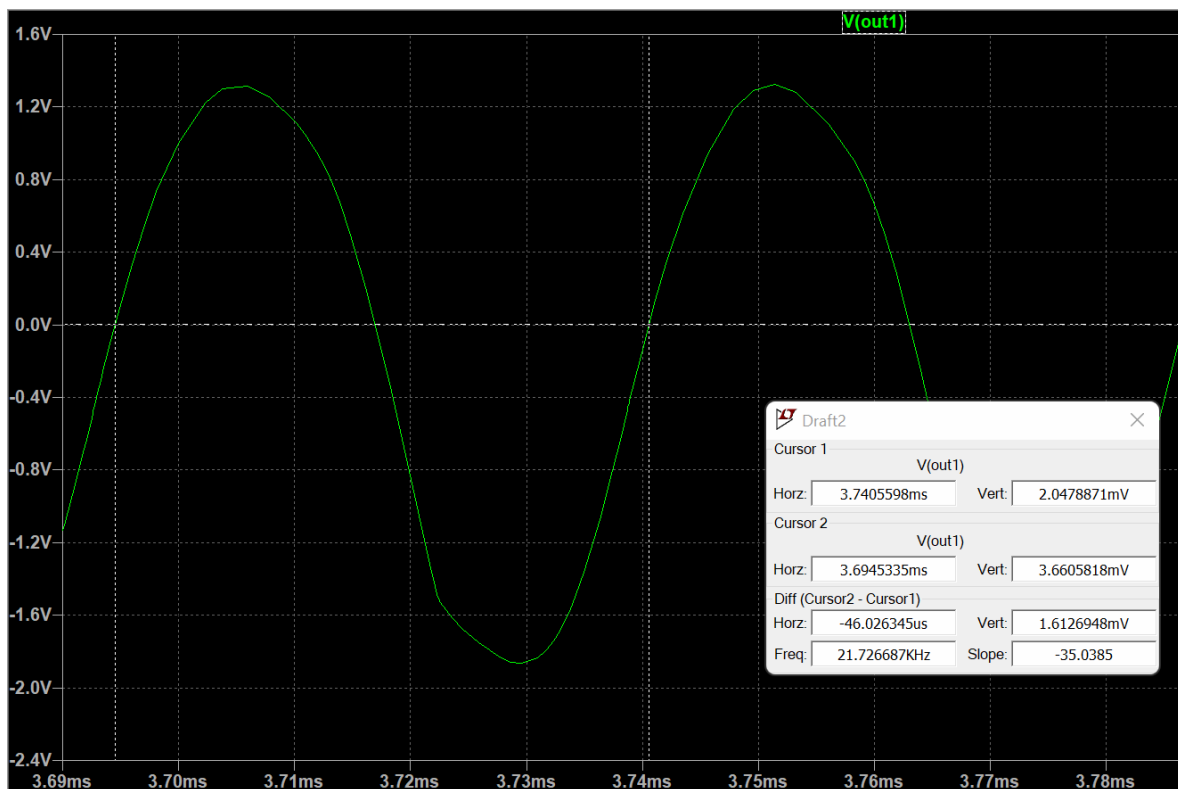
$$C=10\text{nF}$$

$$F=0.15923/\sqrt{LC}=0.15923/7.746$$

## Original waveform

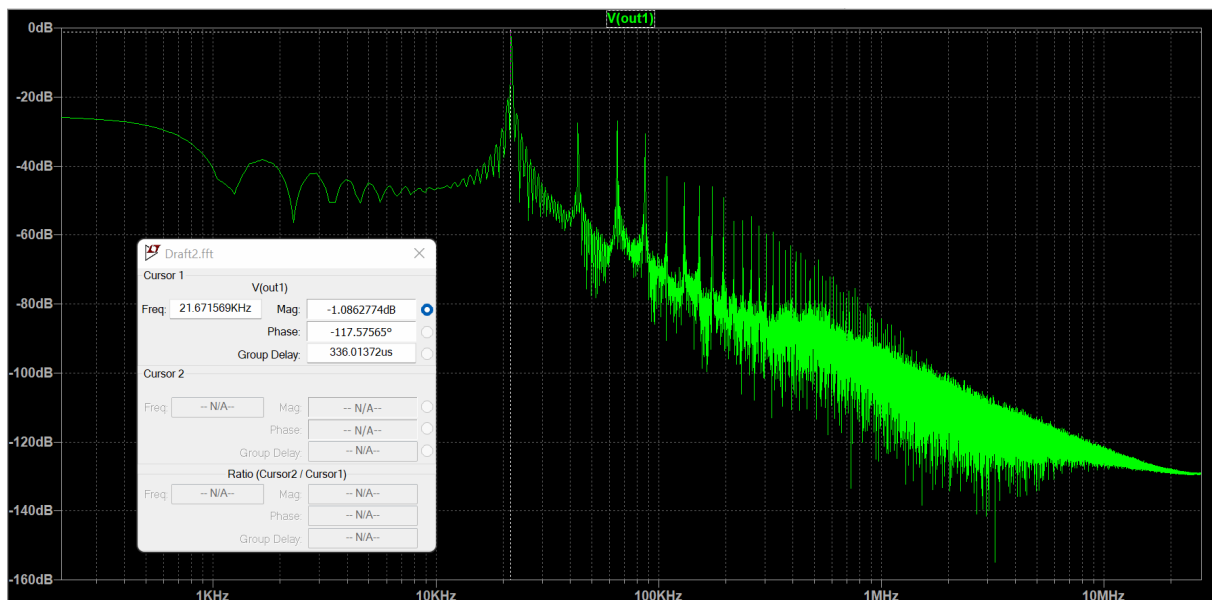


## Frequency calculation from Waveform



The frequency obtained from the waveform is F=21.72KHz

## Frequency Obtained from spectrum:



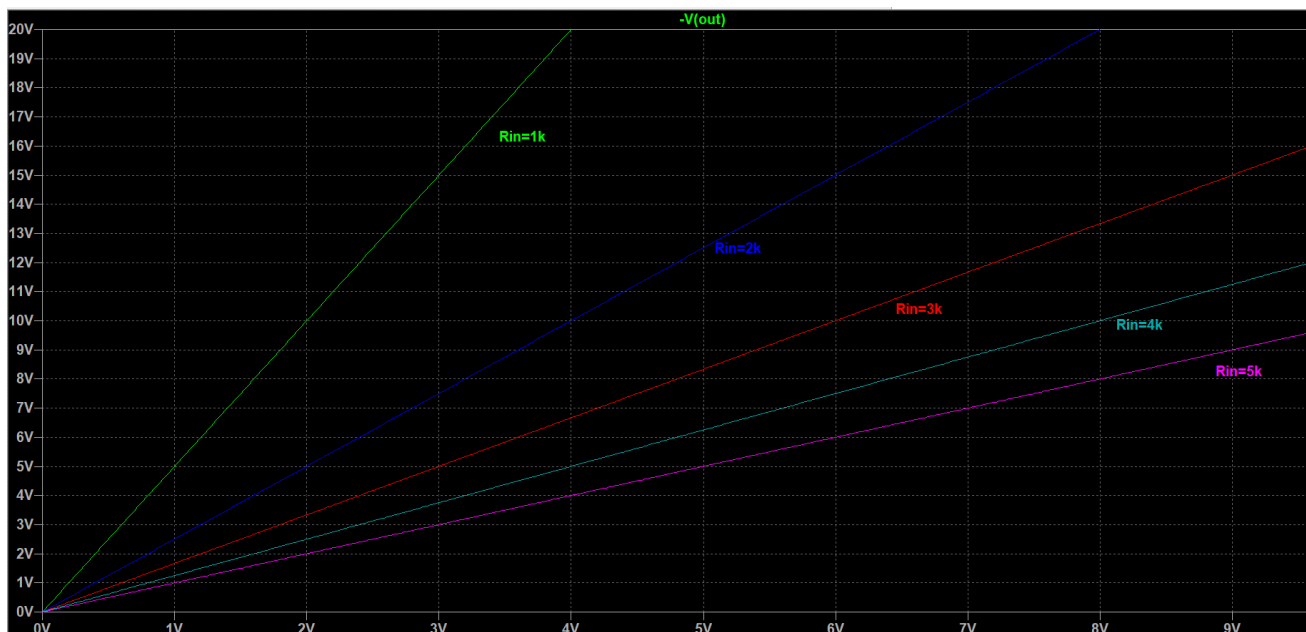
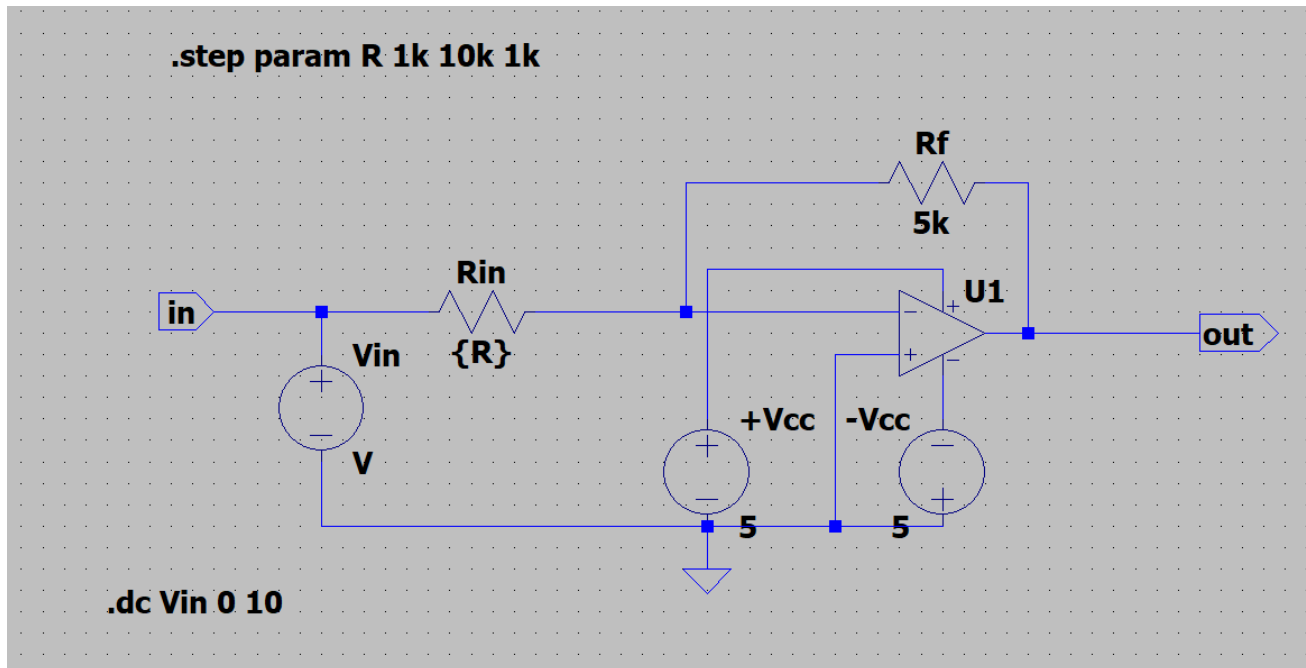
Frequency of oscillator obtained from spectrum is F=21.67KHz

# Op-amp

- a) Problem statement: Obtain a plot that verifies dependence of  $R_f$  and  $R_{in}$  on gain of the op-amp. Use Universal OP-Amp 1,  $R_f = 5k$ ;  $\pm V_{cc} = \pm 5V$

Hint:  $\text{gain} = |V_{out}/V_{in}| = |R_f/R_{in}|$

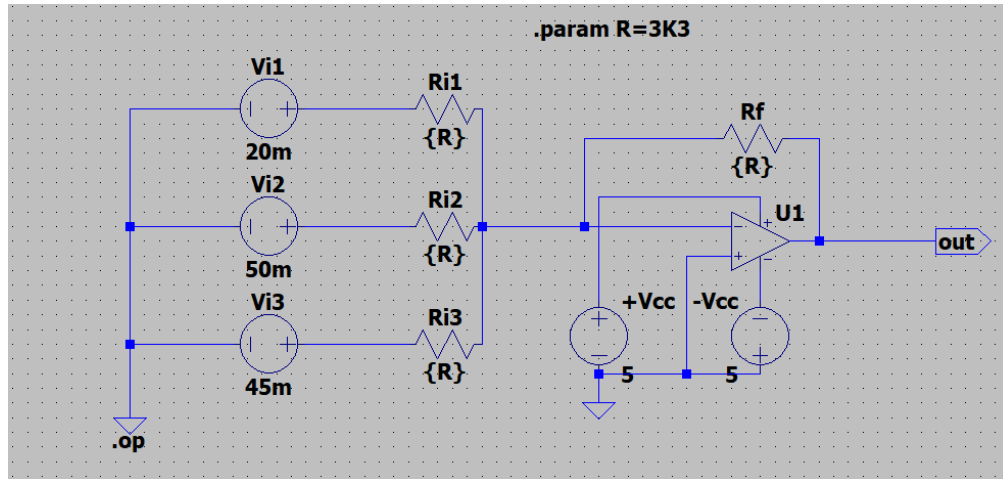
Solution:



b) Op-amp adder/Subtractor:

Problem statement: Design an op-amp Adder with  $V_1 = 20\text{mV}$   $V_2 = 50\text{mV}$   $V_3 = 45\text{mV}$   
 $R_1 = R_2 = R_3 = R_f = 3.3\text{k}$

Solutions:

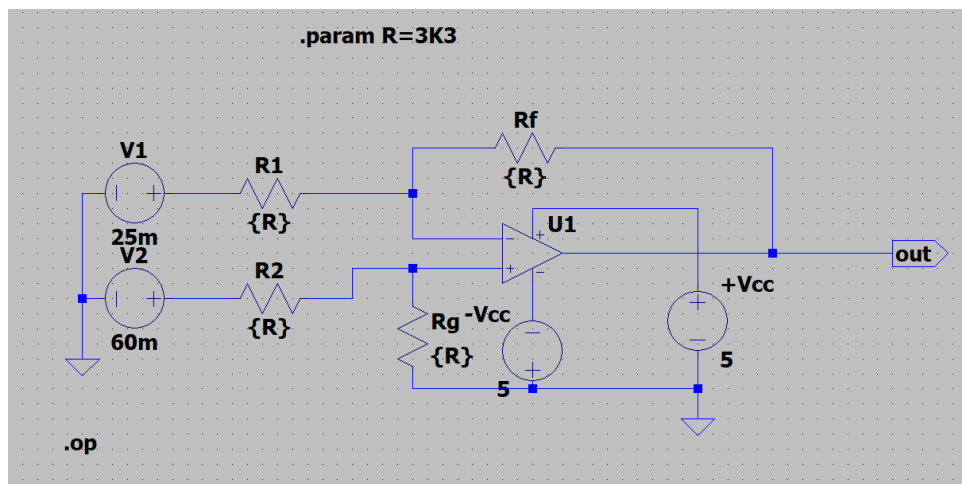


```

LT * C:\Users\1107s\Documents\LTspiceXVII\Draft1.asc
--- Operating Point ---
V(n002):      1.15003e-007  voltage
V(out):       -0.115      voltage
V(n003):      5           voltage
V(n005):      -5          voltage
V(n004):      0.05        voltage
V(n001):      0.02        voltage
V(n006):      0.045       voltage
I(Ri1):       -6.06057e-006 device_current
I(Ri3):       -1.36363e-005 device_current
I(Ri2):       -1.51515e-005 device_current
I(Rf):        -3.48484e-005 device_current
I(Vi3):       -1.36363e-005 device_current
I(Vi1):       -6.06057e-006 device_current
I(Vi2):       -1.51515e-005 device_current
I(+vcc):      -1e-008     device_current
I(-vcc):      -1e-008     device_current
Ix(u1:1):     0           subckt_current
Ix(u1:2):     2.30006e-016 subckt_current
Ix(u1:3):     1e-008      subckt_current
Ix(u1:4):     -1e-008     subckt_current
Ix(u1:5):     3.48499e-005 subckt_current

```

c) Subtractor:



\* C:\Users\1107s\Documents\LTspiceXVII\Draft1.asc

--- Operating Point ---

V(n001) :	0.0299999	voltage
V(n004) :	0.0299999	voltage
V(out) :	0.0349999	voltage
V(n003) :	5	voltage
V(n006) :	-5	voltage
V(n005) :	0.06	voltage
V(n002) :	0.025	voltage
I(R1) :	1.51511e-006	device_current
I(R2) :	-9.09094e-006	device_current
I(Rg) :	9.09088e-006	device_current
I(Rf) :	1.51517e-006	device_current
I(V2) :	-9.09094e-006	device_current
I(V1) :	1.51511e-006	device_current
I(+vcc) :	-9.94e-009	device_current
I(-vcc) :	-1.006e-008	device_current
Ix(u1:1) :	5.99998e-011	subckt_current
Ix(u1:2) :	5.99997e-011	subckt_current
Ix(u1:3) :	9.94e-009	subckt_current
Ix(u1:4) :	-1.006e-008	subckt_current
Ix(u1:5) :	-1.51541e-006	subckt_current

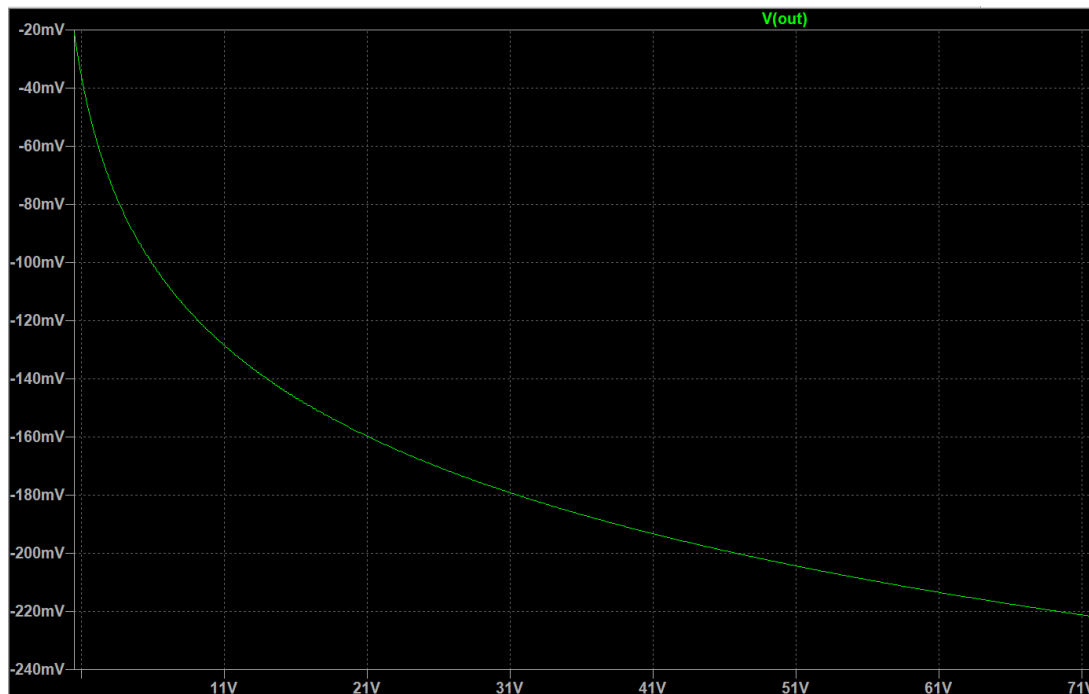
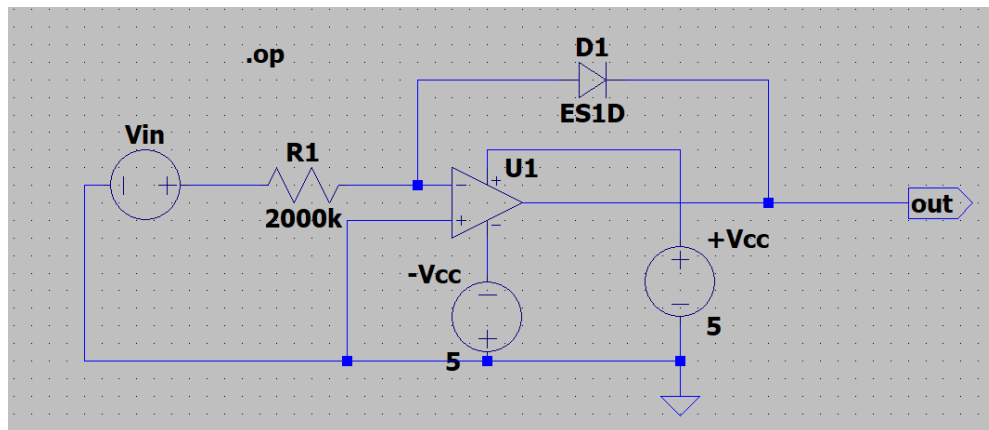
d) Logarithmic

Problem statement: Design a logarithmic op-amp circuit take diode ES1D( $I_S=0.5\mu$ ,  $V_T=0.068$ ),  $R=2M(2000k)$  and plot the output curve.

Note:

$$V_{out} = -V_T \ln\left(\frac{V_{in}}{I_S R}\right)$$

Solution:

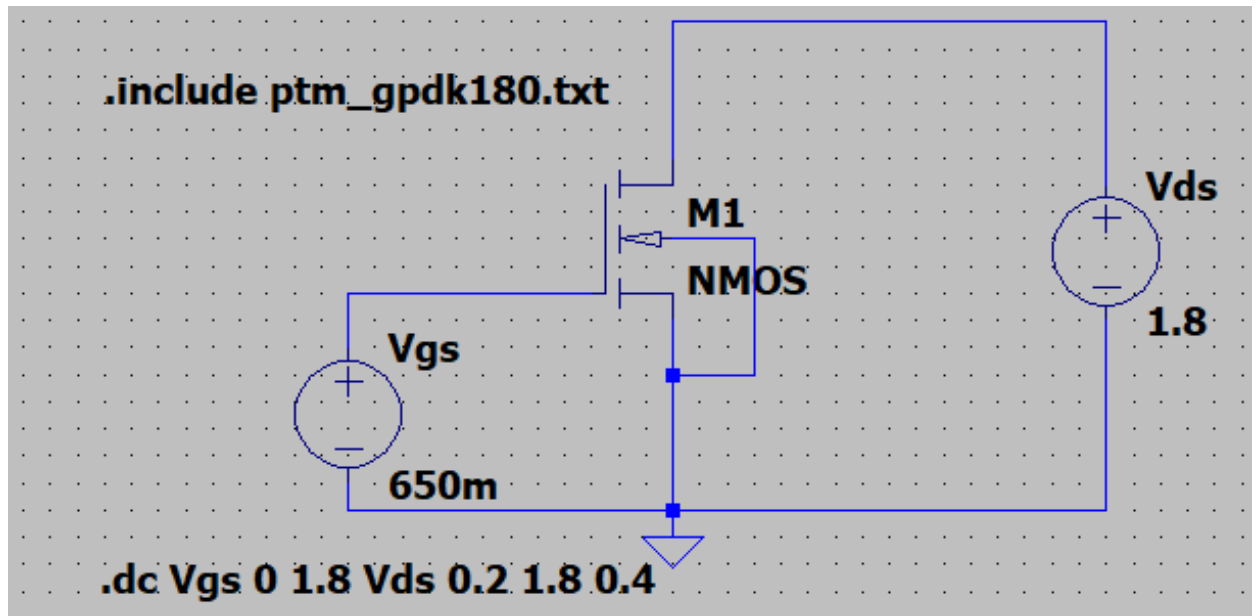


# MOSFET Circuits

- a) Problem Statements: Derive the IV Characteristics of NMOS, PMOS (make sure to use 180m technology). Given that  $W/L=2\mu/180n$ .

Solution:

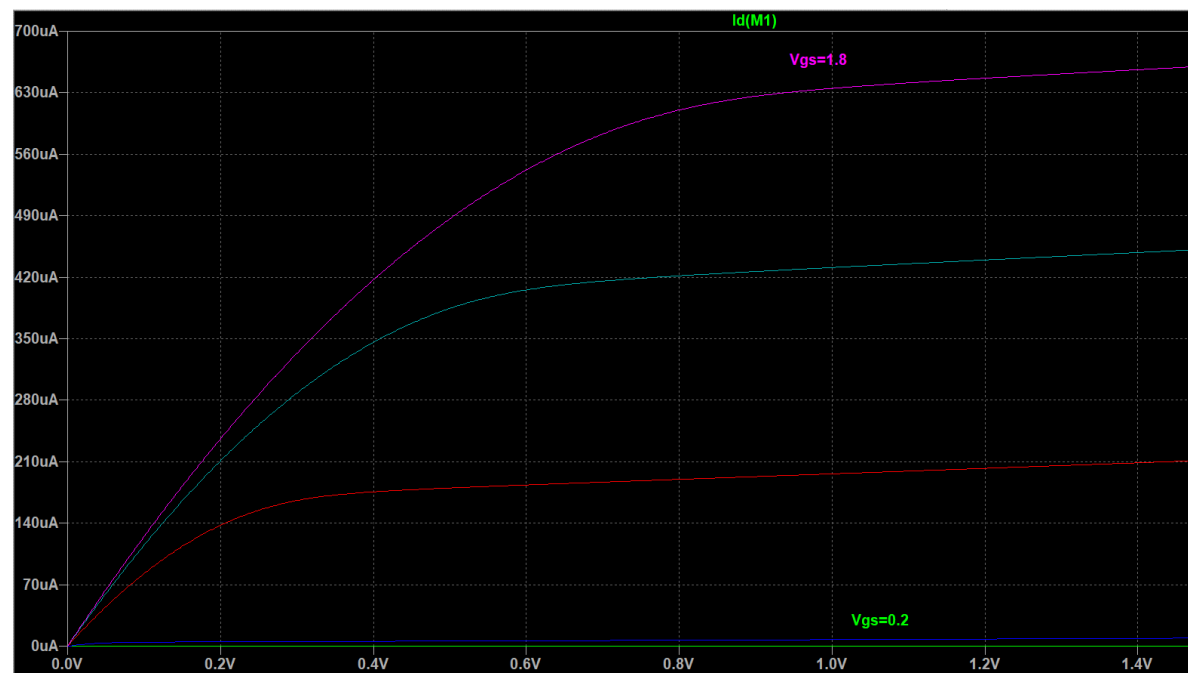
NMOS:



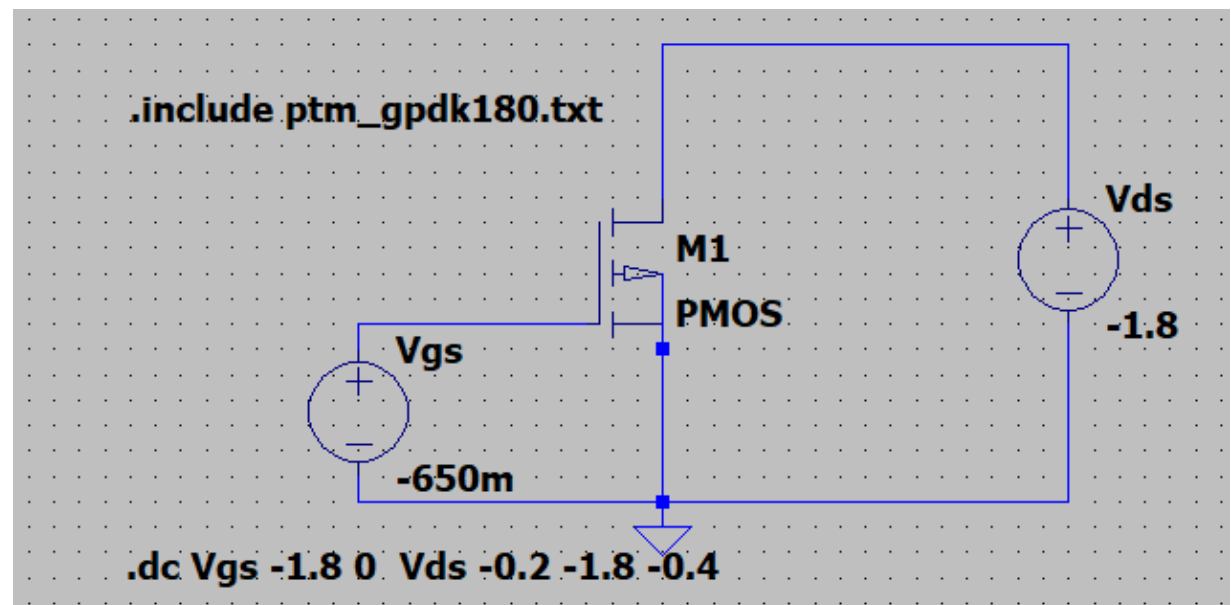
Input Characteristics:



### Output Characteristics:

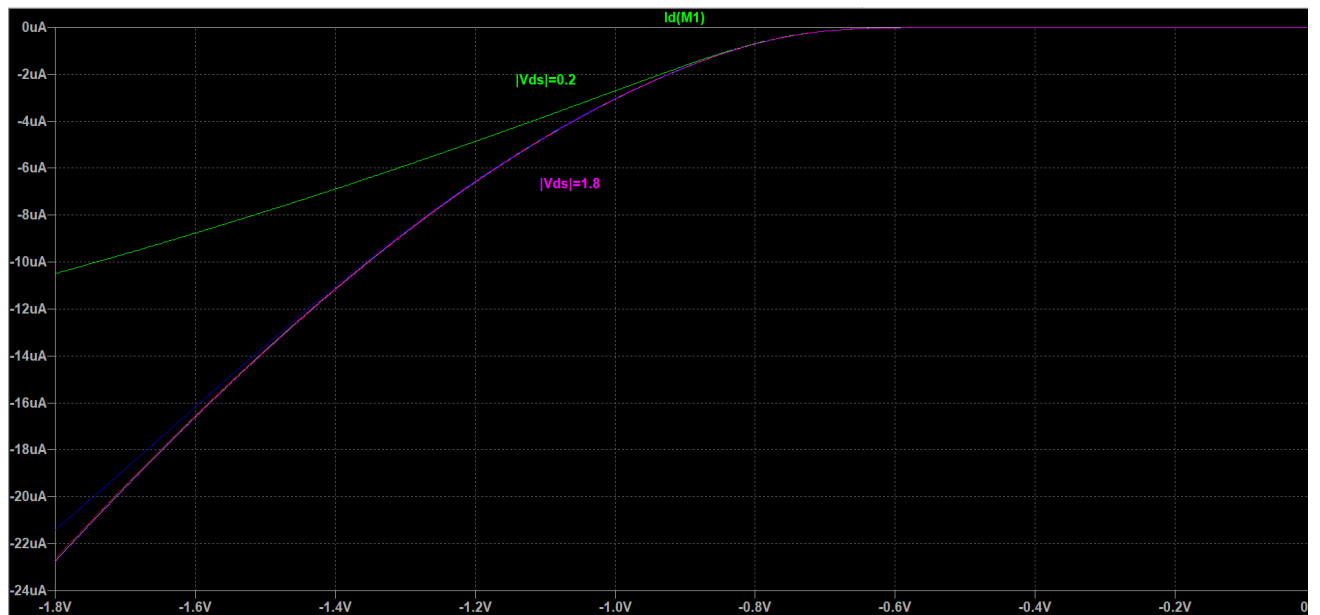


### PMOS:

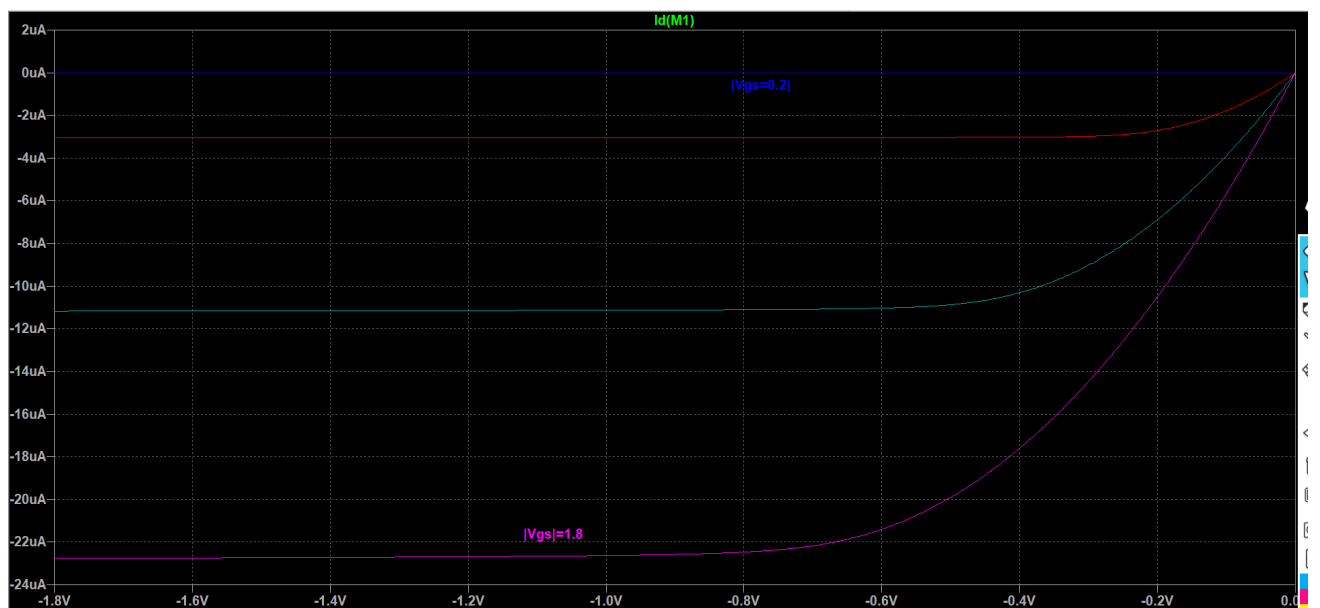




### Input Characteristics:



### Output Characteristics:

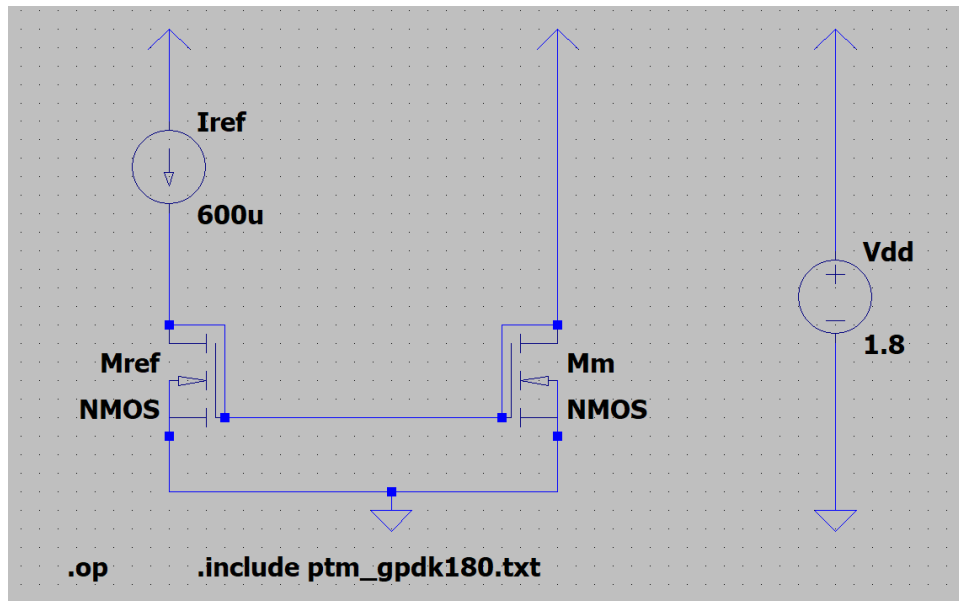


b) NMOS Current Sink :

Problem Statement: Design an NMOS current sink of current Reference 600uA  
 $V_{dd}=1.8V$  and  $W/L(1)=W/L(2)=2\mu/198.5n$

🔍 Exploration work: Vary number of transistors in parallel for reference and mirror devices and find out how the current varies. (Ans: If P Mirrors and Q References are used, the current is scaled by a factor P/Q)

Solution:

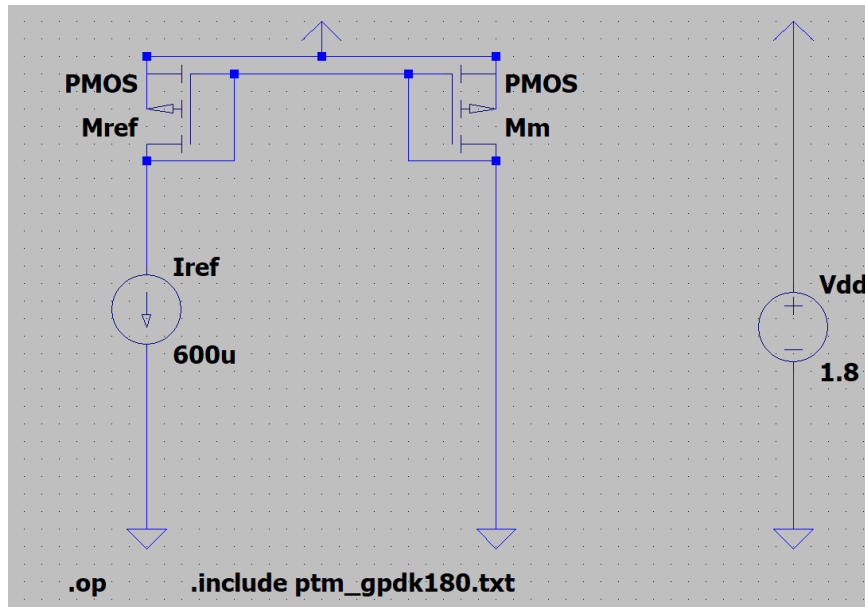


--- BSIM3 MOSFETS ---		
Name:	mref	mm
Model:	nmos	nmos
Id:	6.00e-04	6.00e-04
Vgs:	1.80e+00	1.80e+00
Vds:	1.80e+00	1.80e+00
Vbs:	0.00e+00	0.00e+00
Vth:	5.27e-01	5.27e-01
Vdsat:	8.81e-01	8.81e-01
Gm:	4.14e-04	4.14e-04
Gds:	3.34e-05	3.34e-05
Gmb:	3.88e-04	3.88e-04
Cbd:	0.00e+00	0.00e+00
Cbs:	0.00e+00	0.00e+00
Cgsov:	1.09e-15	1.09e-15
Cgdov:	1.08e-15	1.08e-15

- c) PMOS Current Source: Design an PMOS current sink of current Reference 600uA  
 $V_{dd}=1.8V$  and  $W/L(1)=W/L(2)=2.75u/200n$

🔍 Exploration work: Vary number of transistors in parallel for reference and mirror devices and find out how the current varies. (Ans: If P Mirrors and Q References are used, the current is scaled by a factor P/Q)

Solution:



--- BSIM3 MOSFETS ---		
Name:	mm	mref
Model:	pmos	pmos
Id:	-6.00e-04	-6.00e-04
Vgs:	-1.80e+00	-1.80e+00
Vds:	-1.80e+00	-1.80e+00
Vbs:	0.00e+00	0.00e+00
Vth:	-5.44e-01	-5.44e-01
Vdsat:	-6.15e-01	-6.15e-01
Gm:	5.71e-04	5.71e-04
Gds:	1.17e-04	1.17e-04
Gmb:	6.11e-05	6.11e-05
Cbd:	0.00e+00	0.00e+00
Cbs:	0.00e+00	0.00e+00
Cgsov:	1.50e-15	1.50e-15
Cadov:	1.48e-15	1.48e-15

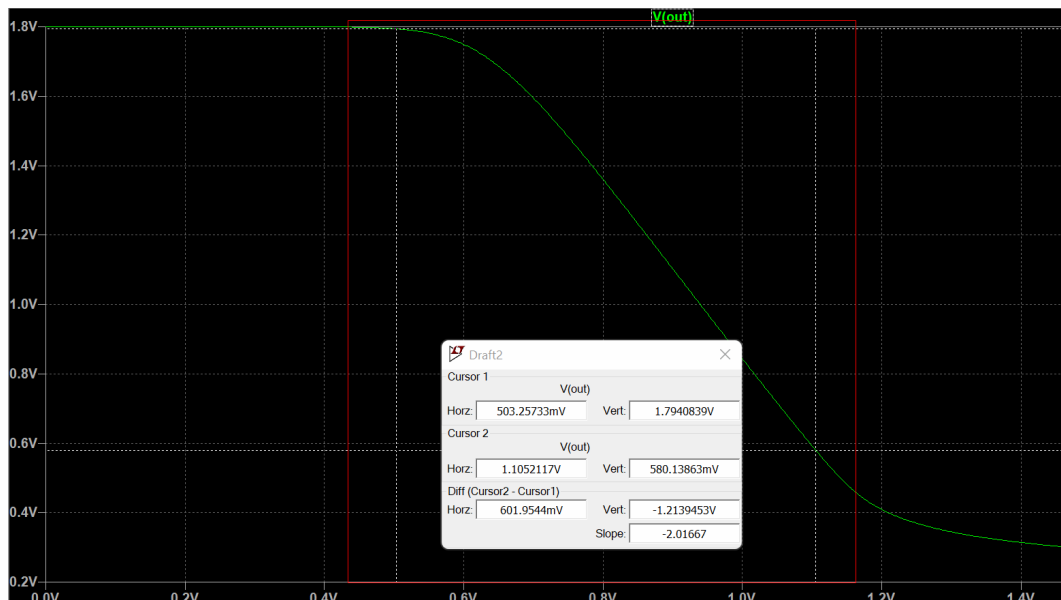
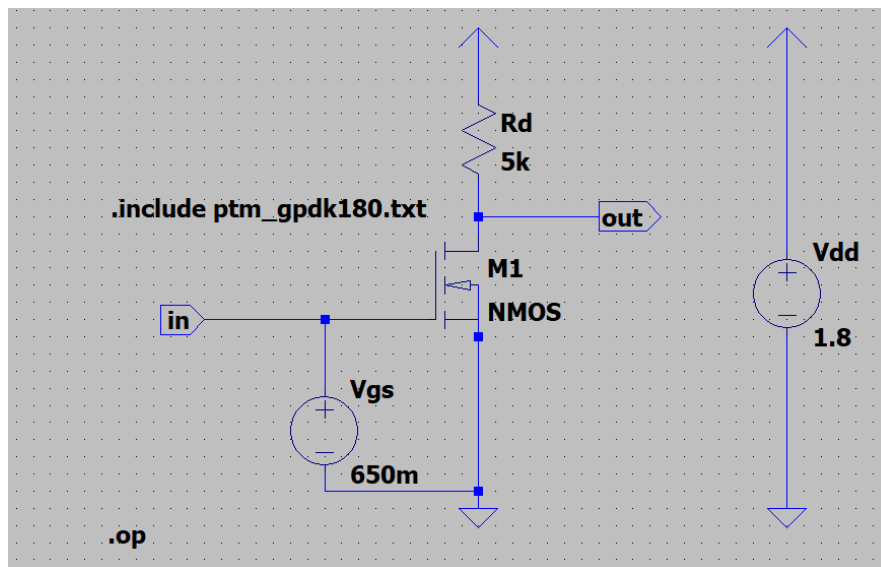
d) NMOS CS single stage Amplifier with resistive load

Problem Statement: Design a Single stage Common Source NMOS Amplifier with resistive load and calculate the gain with following parameters:

- i)  $R_d = 5k$
- ii)  $W = 2\mu$ ;  $L = 180n$
- iii) Technology = 180nm
- iv)  $V_{th} = 0.44$  (Ignore  $V_{th}$  in result window)

Solution:

DC Analysis:



Operating point at Vgs=800mV

--- BSIM3 MOSFETS ---	
Name:	m1
Model:	nmos
Id:	8.80e-05
Vgs:	8.00e-01
Vds:	1.36e+00
Vbs:	0.00e+00
Vdsat:	1.96e-01
Gm:	5.55e-04
Gds:	2.03e-05
Gmb:	2.27e-04
Cbd:	0.00e+00
Cbs:	0.00e+00
Cgsov:	1.09e-15
Cgdov:	1.01e-15
Cgbov:	0.00e+00
dQgdVgb:	7.34e-15
dQgdVdb:	-9.83e-16
dQgdVsb:	-5.90e-15
dQddVgb:	-1.02e-15
dQddVdb:	1.01e-15

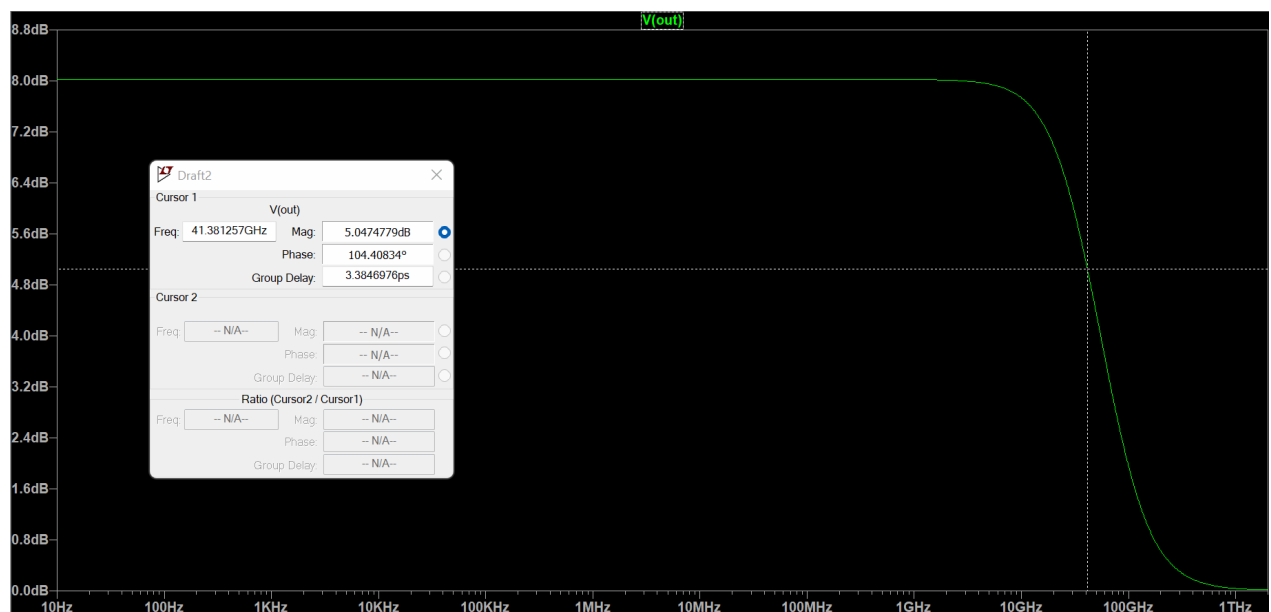
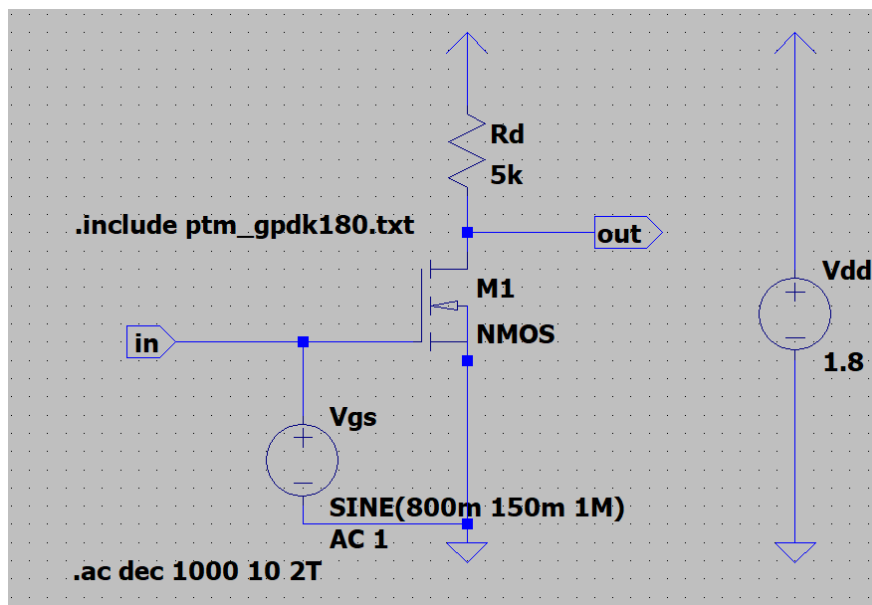
$$R_o = \left| \frac{4I_d V_{ds}}{4I_d^2 - G_m^2 (V_{gs} - V_{th})^2} \right|$$

$$= 58.08k$$

$$A_{dc} = -G_m (R_o || R_d)$$

$$A_{dc} = 2.53 \text{ V/V}$$

## AC Analysis:

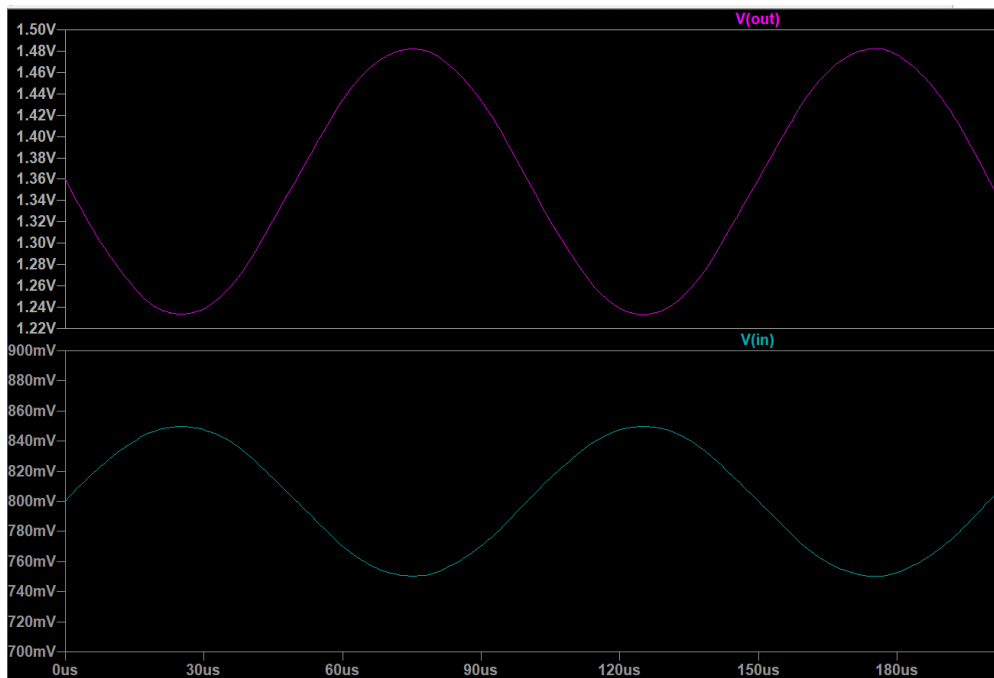
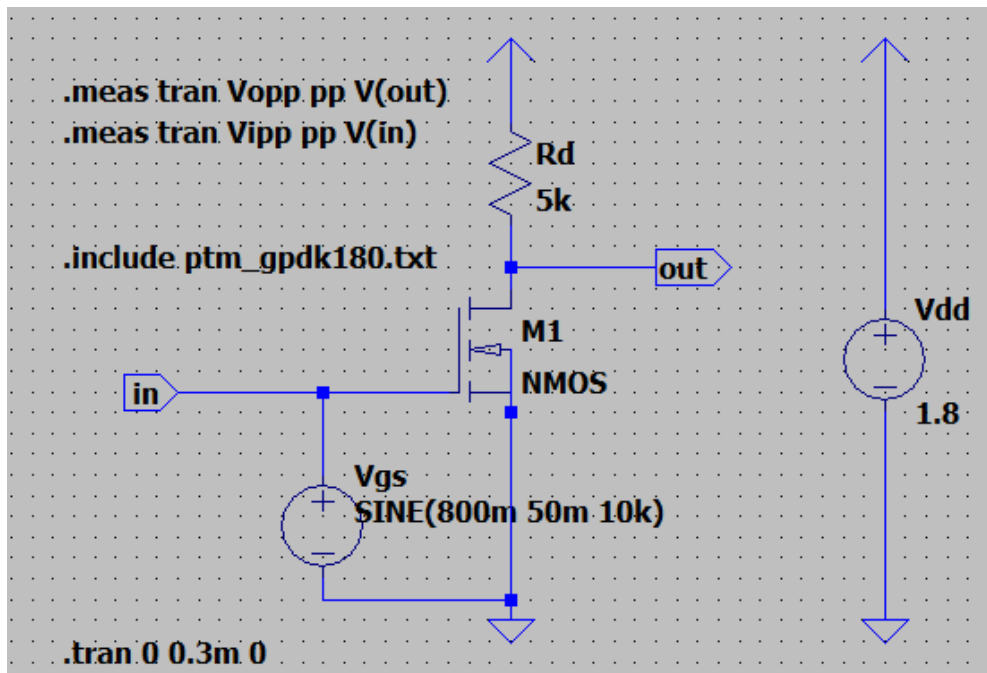


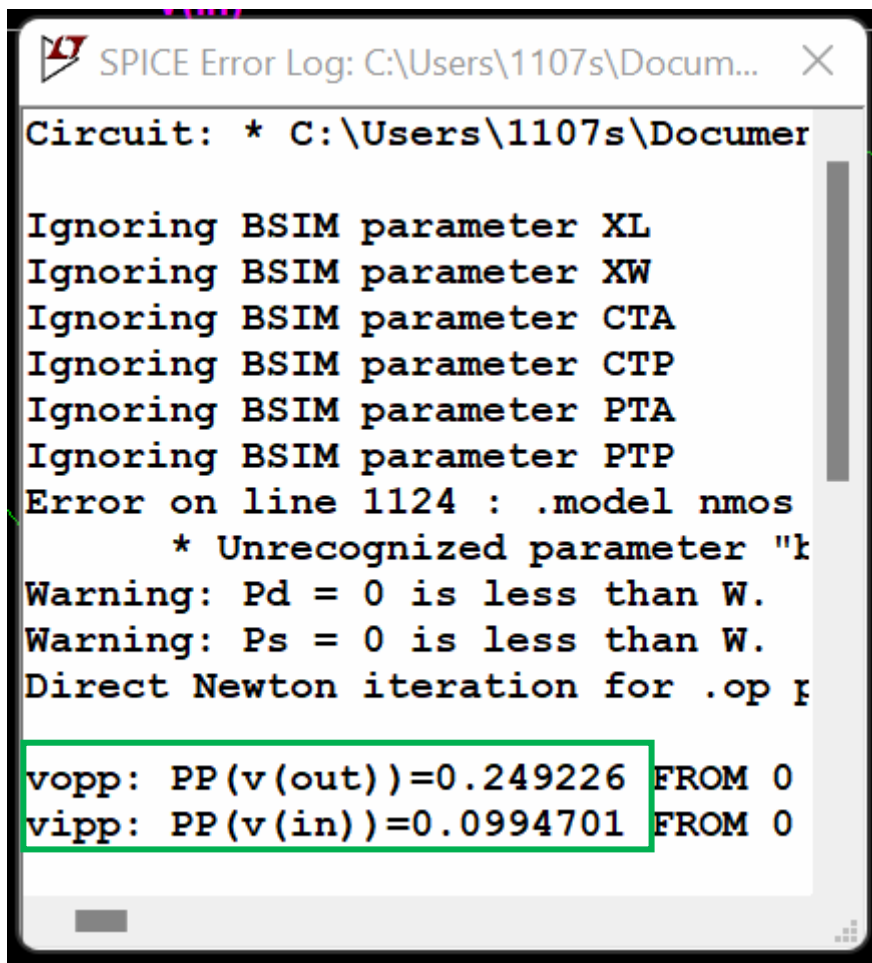
Gain(dB)=8dB

$\Rightarrow A(AC) = 10^{8/20}$

$A(ac) = 2.52 \text{ V/V}$

## Transient Analysis:





The image shows a SPICE Error Log window with the title bar 'SPICE Error Log: C:\Users\1107s\Docum...'. The log contains the following text:

```
Circuit: * C:\Users\1107s\Documer  
  
Ignoring BSIM parameter XL  
Ignoring BSIM parameter XW  
Ignoring BSIM parameter CTA  
Ignoring BSIM parameter CTP  
Ignoring BSIM parameter PTA  
Ignoring BSIM parameter PTP  
Error on line 1124 : .model nmos  
      * Unrecognized parameter "k  
Warning: Pd = 0 is less than W.  
Warning: Ps = 0 is less than W.  
Direct Newton iteration for .op p  
  
vopp: PP(v(out))=0.249226 FROM 0  
vipp: PP(v(in))=0.0994701 FROM 0
```

The last two lines are highlighted with a green box.

$A(\text{tran}) = v_{\text{opp}} / v_{\text{ipp}}$

$A(\text{tran}) = 2.504$



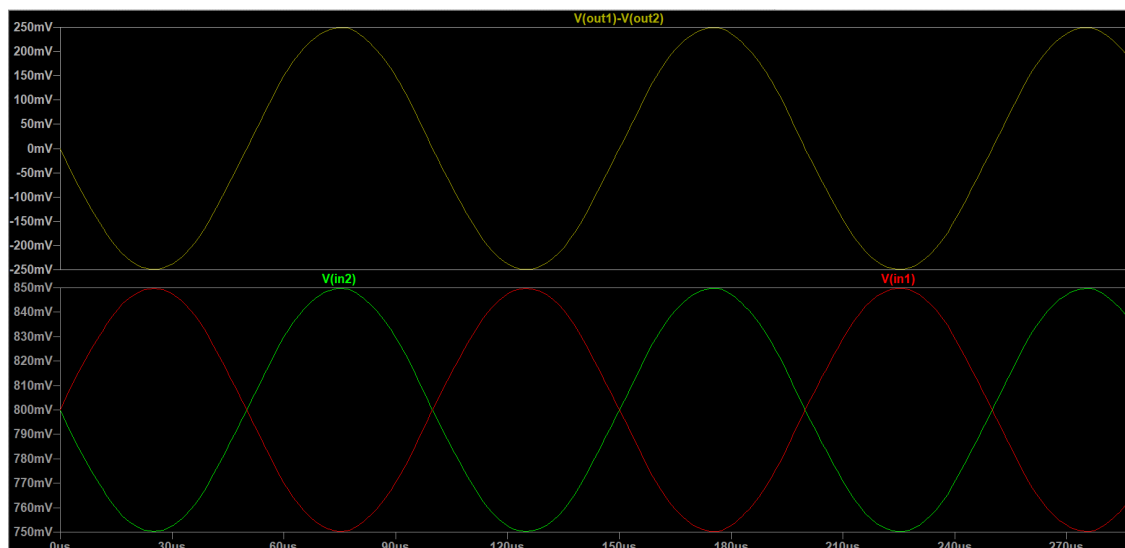
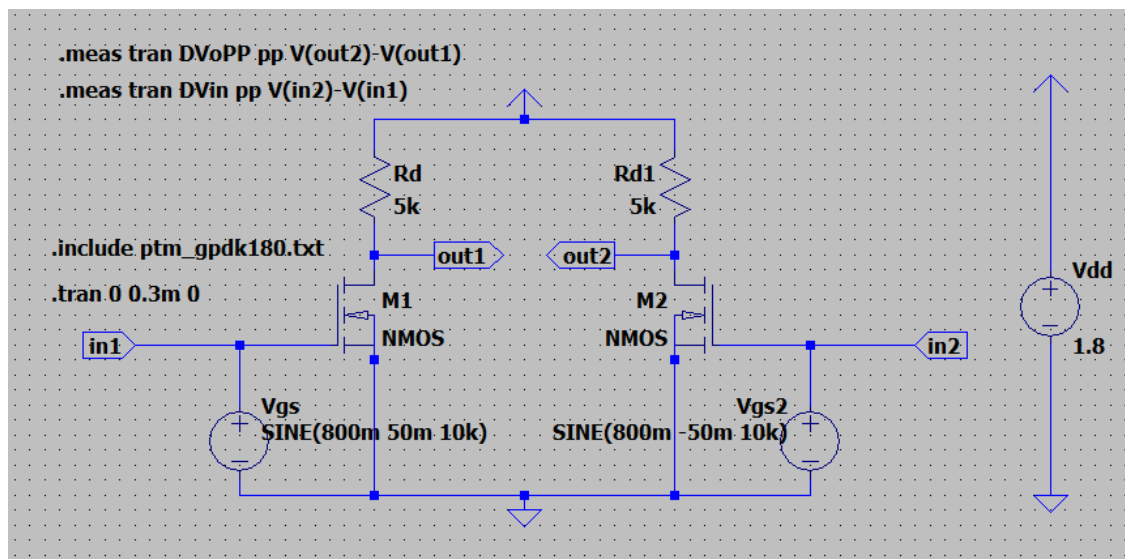
e) NMOS CS differential amplifier with balanced resistive load

Problem Statement: Construct a CS differential amplifier , plot  $\Delta V_{out}$  and  $V_{in}$ , and calculate the double-ended gain

Parameters:

- I.  $R_d = 5k$
- II.  $W = 2\mu$ ;
- III.  $L = 180n$
- IV. Technology = 180nm

Solution:





SPICE Error Log: C:\Users\1107s\Documents\LTspiceXVII\Draft2.log

Circuit: \* C:\Users\1107s\Documents\LTspiceXVII\Draft2

Ignoring BSIM parameter XL

Ignoring BSIM parameter XW

Ignoring BSIM parameter CTA

Ignoring BSIM parameter CTP

Ignoring BSIM parameter PTA

Ignoring BSIM parameter PTP

Error on line 1127 : .model nmos nmos level = 49 lint :  
\* Unrecognized parameter "binflag"

Warning: Pd = 0 is less than W.

Warning: Ps = 0 is less than W.

Direct Newton iteration for .op point succeeded.

dvopp: PP(v(out2)-v(out1))=0.498694 FROM 0 TO 0.0003

dvin: PP(v(in2)-v(in1))=0.199049 FROM 0 TO 0.0003

$$A = \frac{dV_{opp}}{dV_{ipp}}$$

$$A = 2.504V/V$$