Brac University

Department of Electrical & Electronic Engineering

Semester Spring-25

Course Number: EEE203L

Course Title: Electrical Circuits II Laboratory

Section: 06

Lab Report

Experiment no. 07

Name of the experiment: Cut-off Frequencies, Bandwidth & Selectivity (Software Simulation)

Prepared by:

Name: **Tanzeel Ahmed** ID: **24321367**

Group Number: 03

Other Group members:

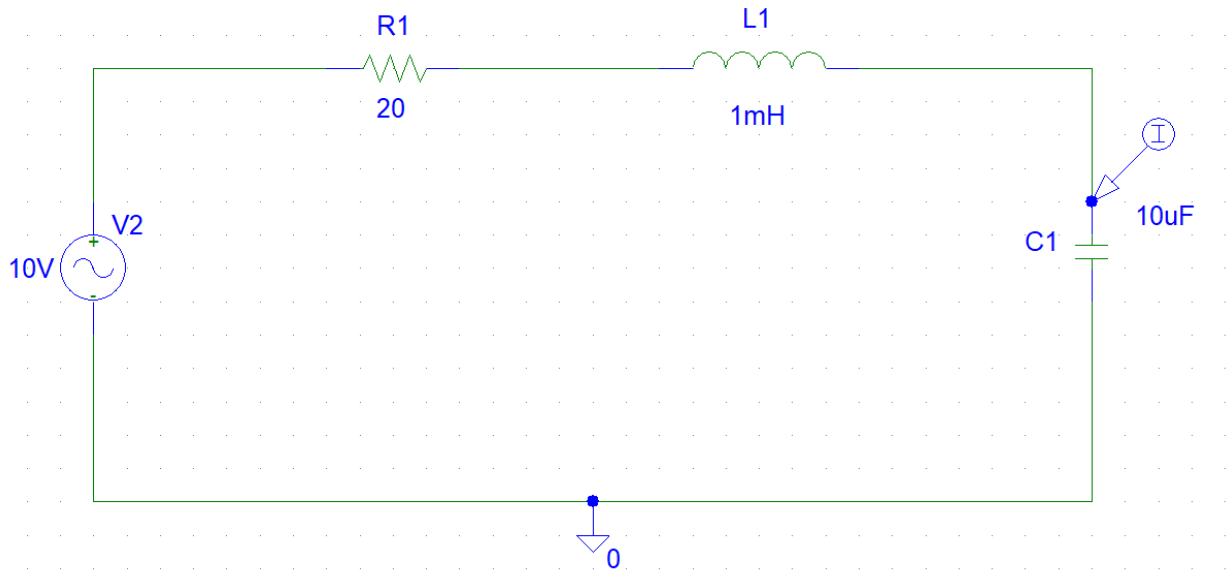
Sl.	***ID***	***Name***
1.	24121083	Abontika Das
2.	24121225	Aditi Gupta
3.	24121219	Subha Tasfia Chowdhury
4.	24321022	Sumya Zaman

Question:

1. Derive the relation between Q-factor and Bandwidth in a Series Resonant circuit.
2. A Series RLC circuit has the following parameters:
   R = 10Ω, L = 0.014H, C = 100uF. Compute the following:
   a. Resonant frequency in rad/sec.
   b. Q-factor of the circuit
   c. Bandwidth
   d. Upper and lower cut-off frequencies
   e. Maximum value of voltage appearing across the capacitor if the applied voltage is v(t) = 2 sin (1000t).

Experiment 7 (PSpice Simulation)

- Construct the circuit as follows and place a current marker to determine the current flowing through the series circuit.



- For the voltage source, use VAC and set the parameters accordingly.

Experiment 07 (Simulation):

Objective: This software experiment will be performed to learn the simulation steps for determining Cut-off frequencies, bandwidth and selectivity on a AC circuit using the Pspice software

Equipments required:

1. Pspice software (Schematics)
2. Suitable PC or Laptop

Components required in software:

- VAC voltage source
- Resistor (R)
- Capacitor (C)
- Ground (GND-Analog)
- Inductor (L)

Circuit diagram:

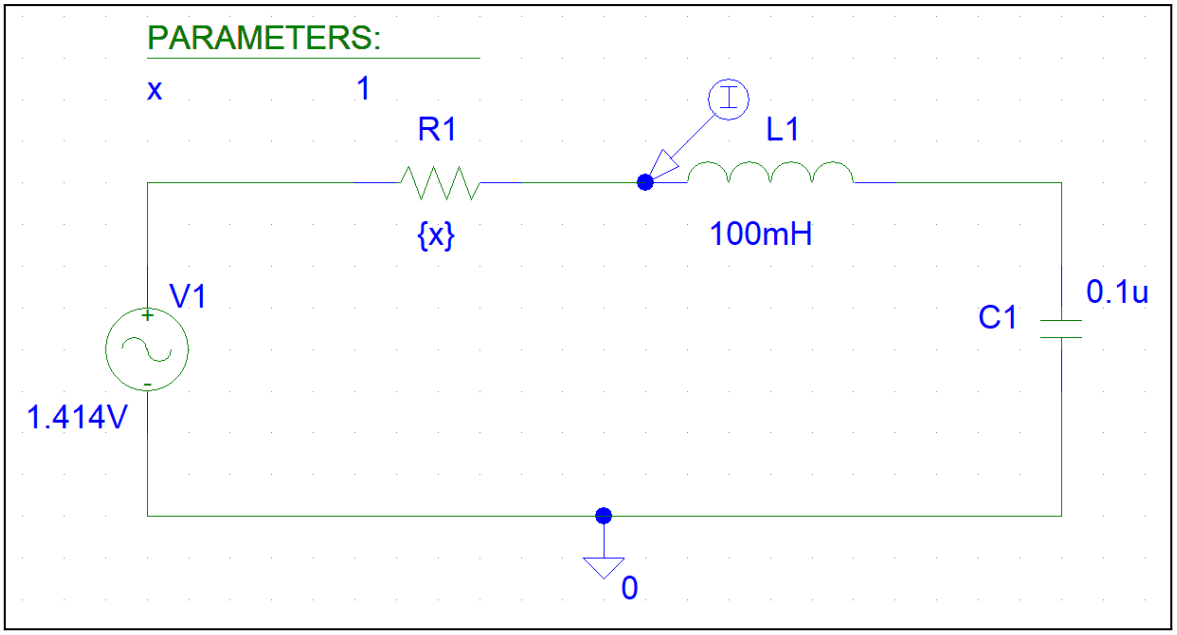
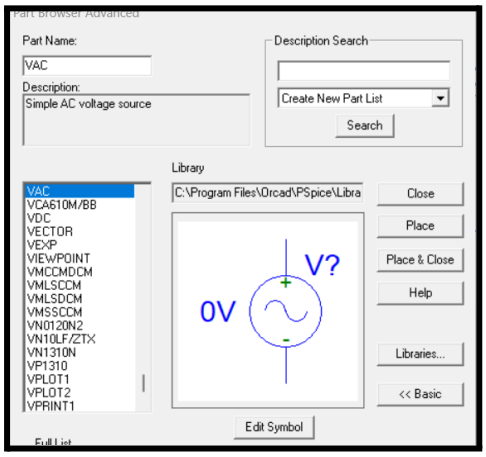
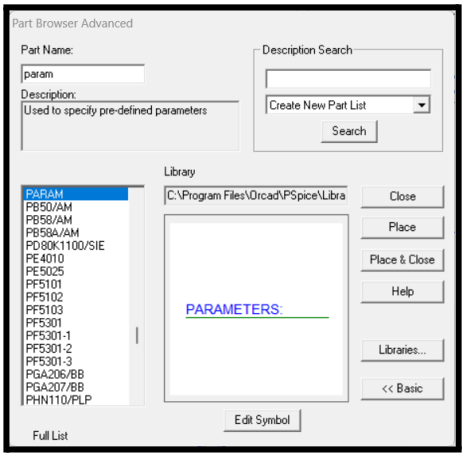


Figure: Circuit diagram for determining Cut-off frequency, bandwidth and selectivity simulated in Pspice Schematics

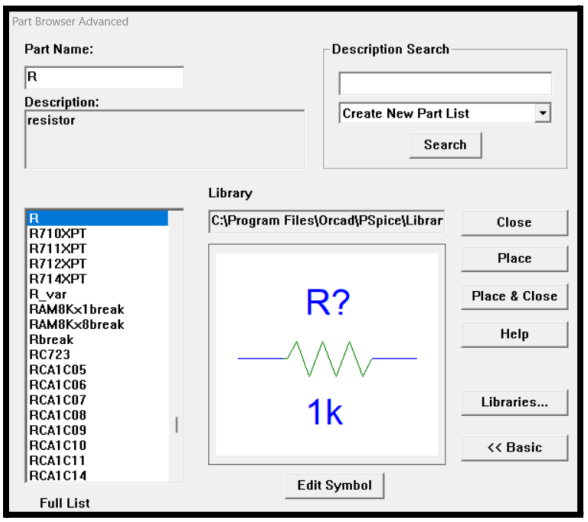
Tools, values and parameter setup menu:



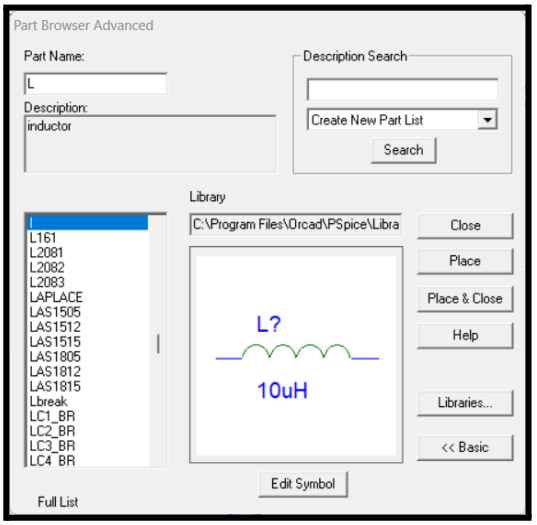
Voltage Source (VAC)



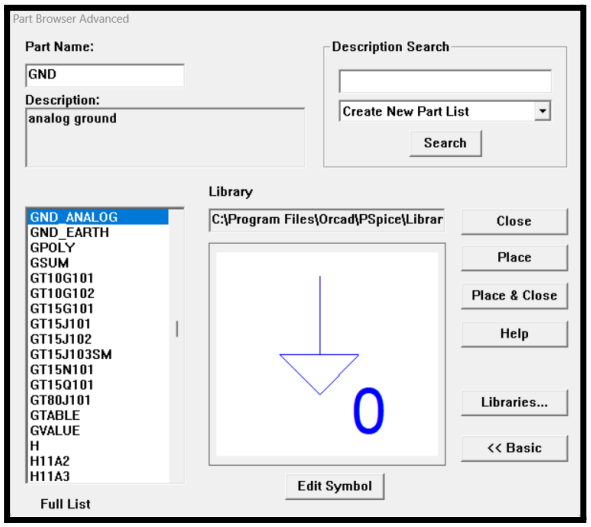
Selection of Parameters



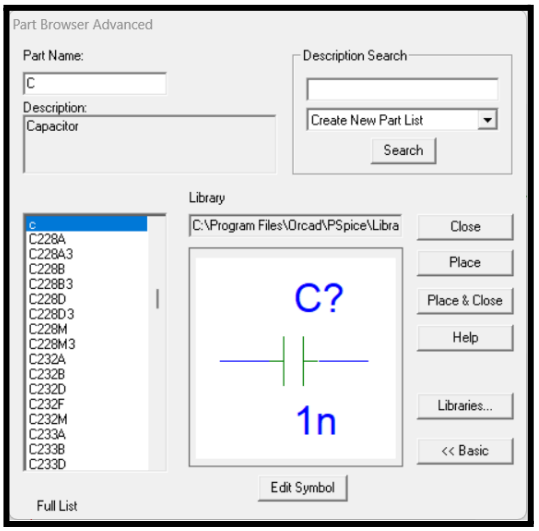
Selection of Resistor



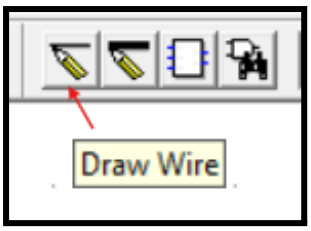
Selection of Inductor



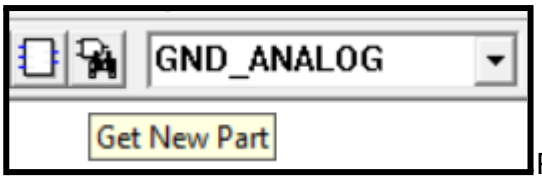
Selection of Ground



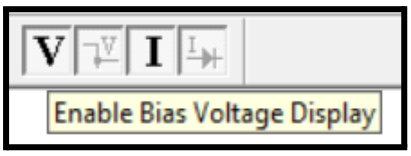
Selection of Capacitor



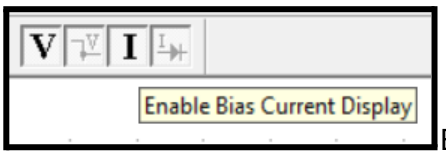
Wire tool



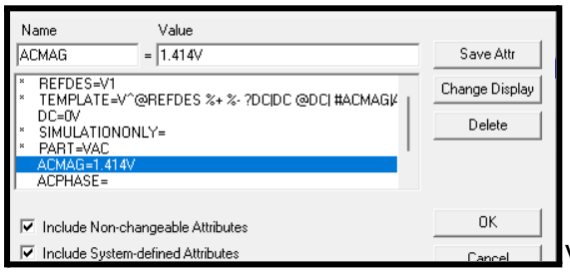
Parts menu



Bias Voltage



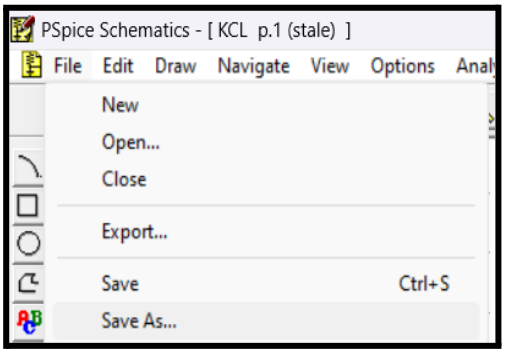
Bias Current



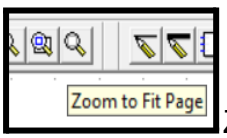
Values set in VAC

Resistor set (R value set)

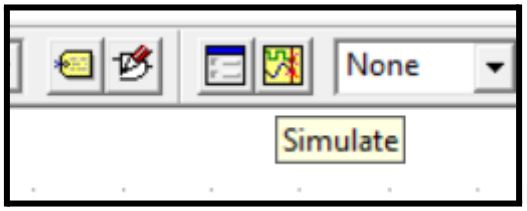
Text tools



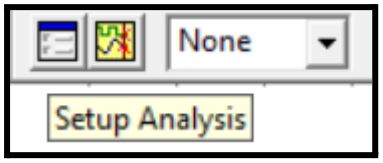
File saving



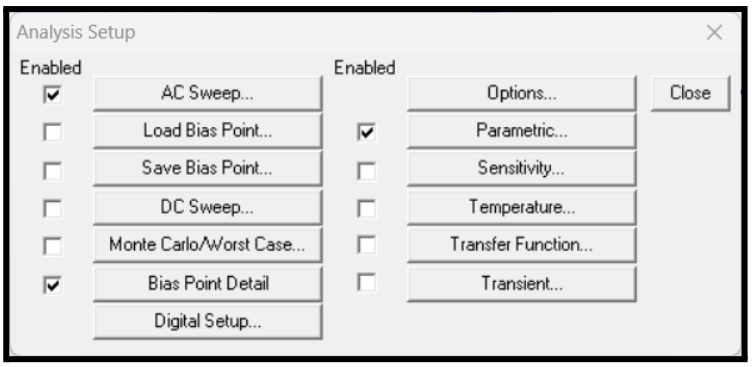
Zoom



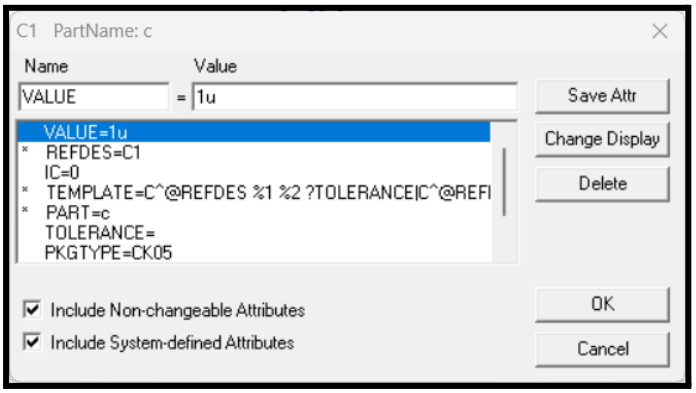
Begin Simulation



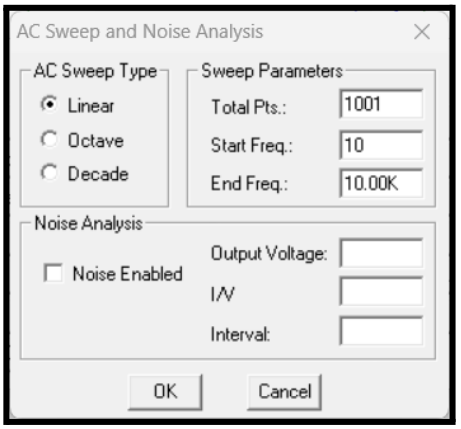
Setup Analysis Icon



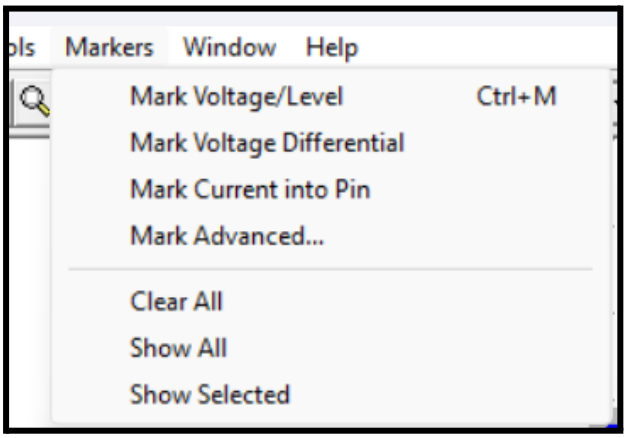
Analysis Setup Menu (AC Sweep and Parametric)



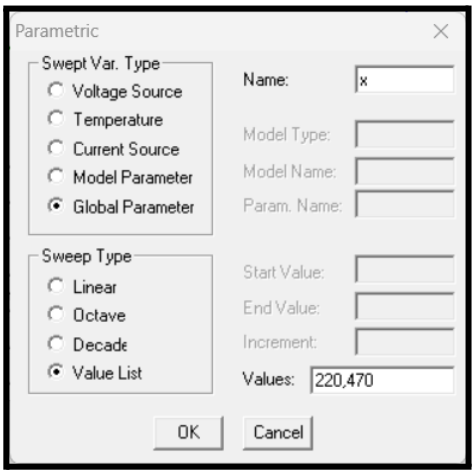
Values set in Capacitor



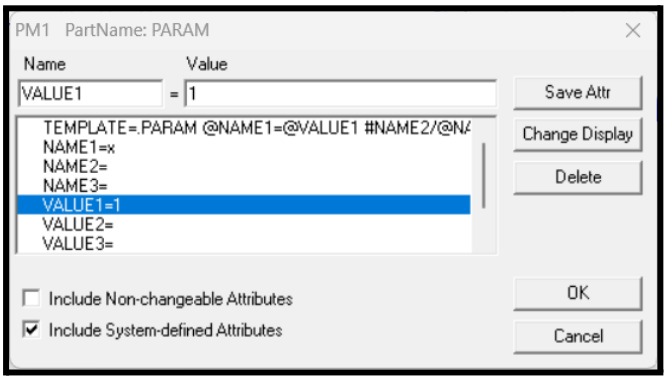
AC Sweep values



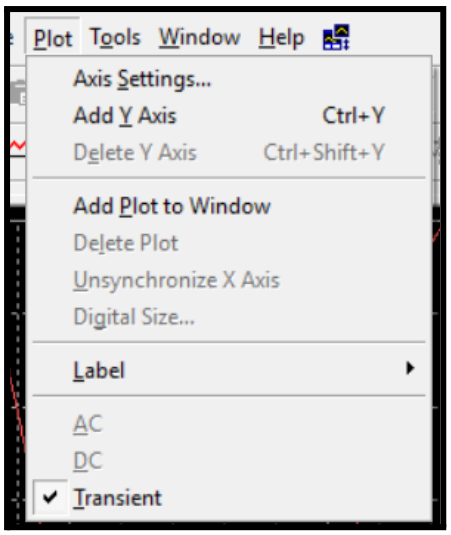
Marker Menu



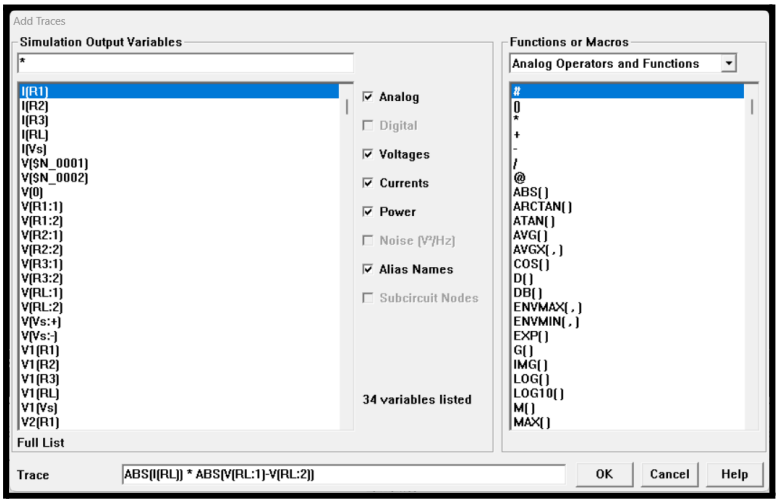
Parametric value



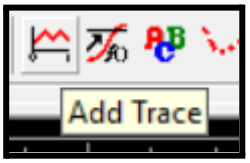
Values set inside parameters



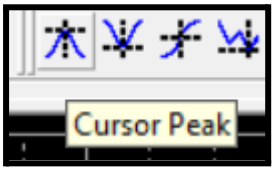
Adding new plot



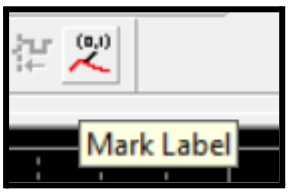
Add trace value



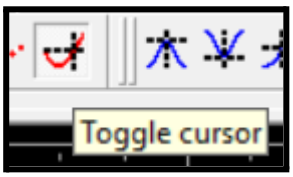
Add trace tool



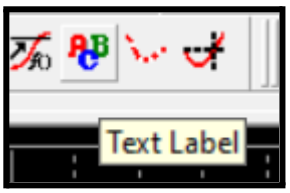
Cursor Peak tool



Mark Label Tool

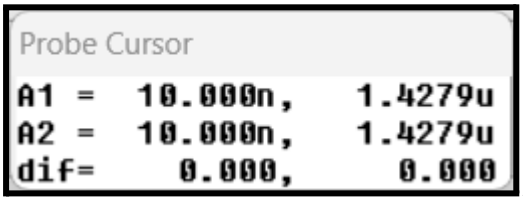


Toggle Cursor Tool



Text Label tool

Selected Plot



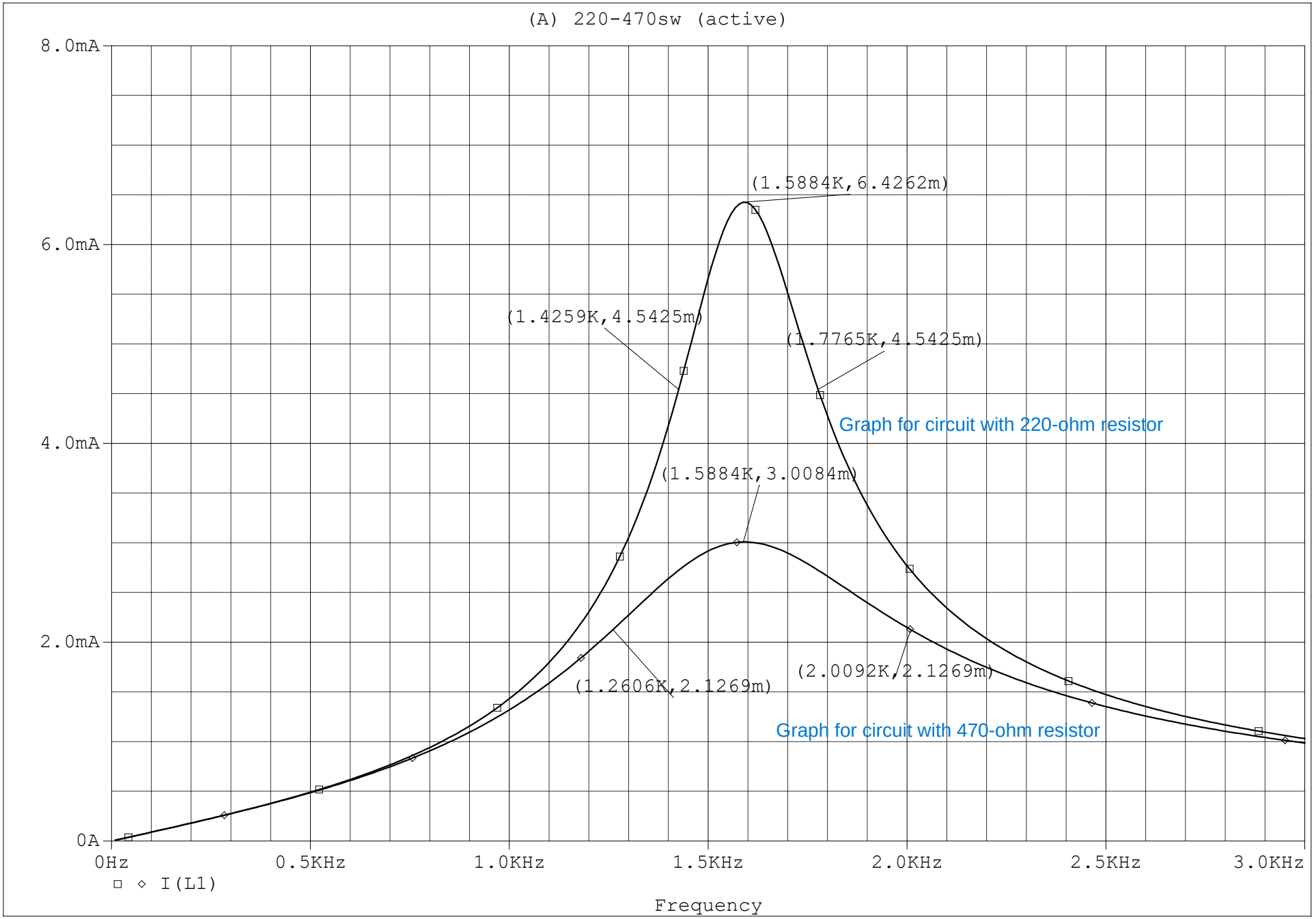
After enabling cursor peak (Probe Cursor Window)

Experiment Procedure:

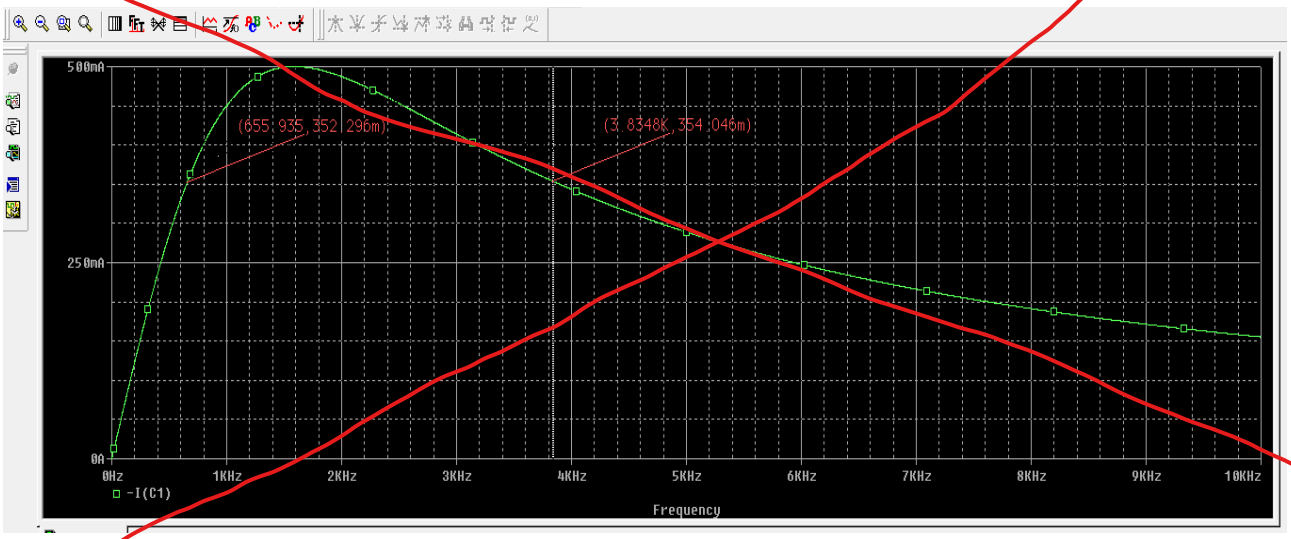
1. Open Pspice Schematics software
2. Open the parts menu
3. Search the necessary parts and place them according to the diagram
4. Using the wire tool connect all the parts in the circuit
5. Rename all the parts for easier identification
6. Use the draw text and text box tool to mark necessary information
7. Set the value of resistor and capacitor.
8. Double click on VAC and set ACMAG=1.414V.
9. Open setup analysis and select AC Sweep.
10. In AC Sweep, set type Linear, Total Pts. 1001, Start Freq, 10 and End 10K.
11. Select the Parametric menu and in there enable Global Parameter, Sweep Type Value list and Values 220, 470 along with the name x.
12. Double click on the capacitor and set IC=0V.
13. Double click the Parameters part and set the name and VALUE1.
14. Add Mark Voltage/Level and Current into Pin on the circuit.
15. Begin circuit simulation.
16. Shift to the graph interface menu
17. Use toggle cursor to mark peak of all graphs
18. Calculate the necessary information and fill the data tables.

* D:\BracU\4. Fourth Semester (SP-25)\EEE203\Lab\Simulations\220-470\simulation\220-470sw.sch

Date/Time run: 04/21/25 19:12:21 Temperature: 27.0



- For the cut-off frequencies, determine the value of 0.707 I_{max} and then mark those corresponding points to determine the lower (f_1) and upper (f_2) cut-off frequencies.



- Use the equations to determine Bandwidth and Q-factor.

Data:

R (Ω)	$\|I\|_{max}$ (mA)	f_S (kHz)	$0.707\|I\|_{max}$ (mA)	f_{CL} (kHz)	f_{CH} (kHz)	$BW = f_{CH} - f_{CL}$ (kHz)	$Q_S = \frac{f_{SP}}{BW}$
220	6.4262	1.5884	4.5433	1.4259	1.7765	0.3506	4.5305
470	3.0084	1.5884	2.1269	1.2606	2.0092	0.7486	2.1218

Calculation:

Table for 220-ohm = $0.707|I|_{max} = 0.707*6.4262 = 4.5433$

BW = 1.7765-1.4259 = 0.3506

$Q_s = fsp/BW = 1.5884/0.3506 = 4.5305$

Table for 470-ohm = $0.707|I|_{max} = 0.707*3.0084 = 2.1269$

BW = 2.0092-1.2606= 0.7486

$Q_s = fsp/BW = 1.5884/0.7486 = 2.1218$

Discussion:

We were able to build and observe a RLC series circuit which had an alternating current (AC) source using the Pspice Schematics software. Sinusoidal waveform was generated after the simulation was completed and after calculations, we correlated with practically measurable values such as rms, phase angle and time period. The usage of inductors in a simulated circuit was also established. From the simulation graph we successfully determined that it is possible to have resonance by varying the frequency while keeping L and C constant. The frequency at which the resonance occurs is known as resonant frequency ‘fr’. At this frequency, the current is maximum and the voltage drop across ‘R’ is also maximum. Beyond this frequency, the current and as well as the VR drops which gives rise to a profile. All steps necessary to complete the circuit and experiment were mentioned in the instructions. In conclusion, we were able to successfully verify cut-off frequencies, bandwidth and selectivity of the given circuit.

Schematics Drive Link: EEE203L-EXP7