

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.

Name of the experiment: Familiarization with Series Resonance in AC Circuits (Software Simulation)

Prepared by:

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

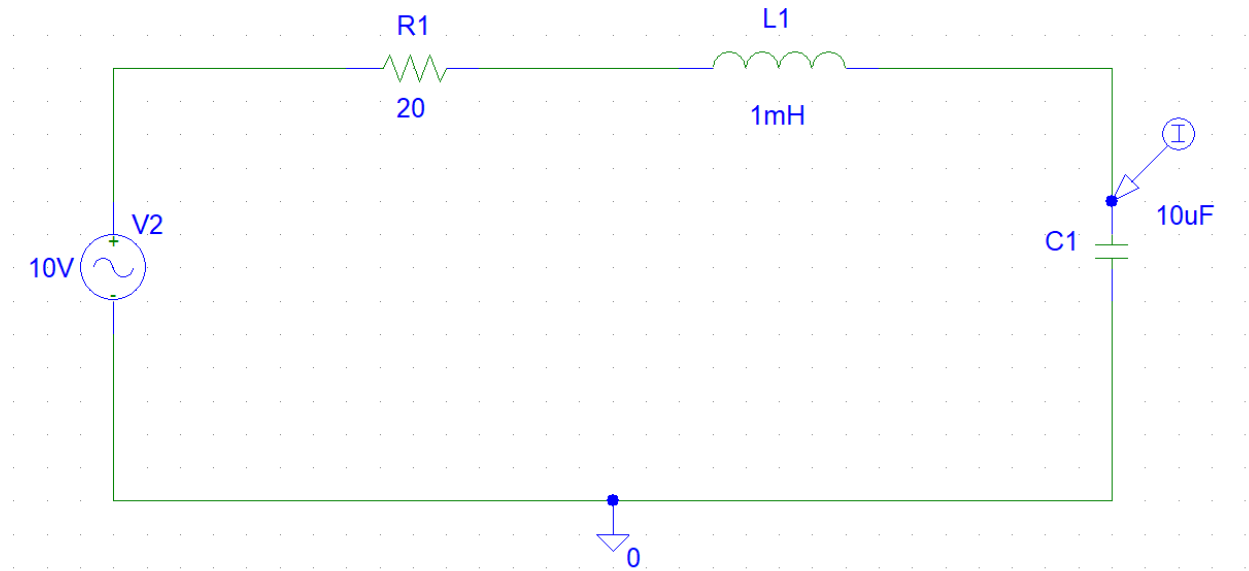
Group Number: 03

Other Group members:

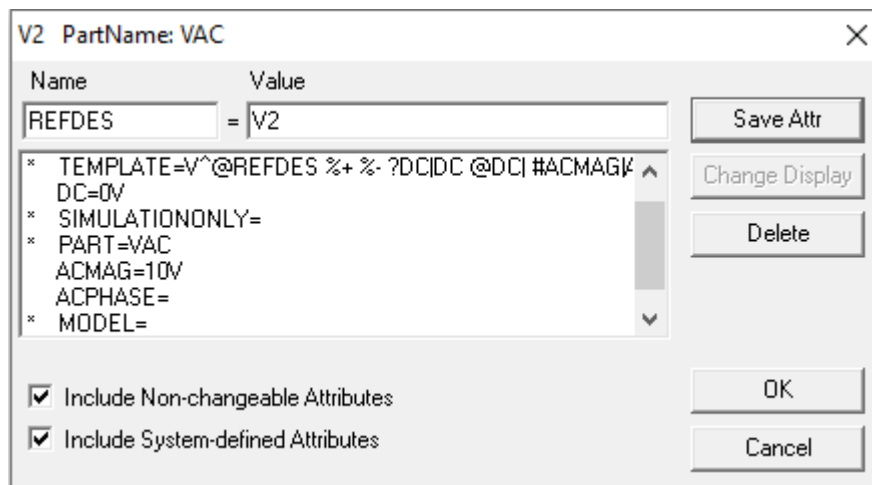
| <i>Sl.</i> | <i>ID</i> | <i>Name</i> |
|-------------------|------------------|------------------------|
| 1. | 24121083 | Abontika Das |
| 2. | 24121225 | Aditi Gupta |
| 3. | 24121219 | Subha Tasfia Chowdhury |
| 4. | 24321022 | Sumya Zaman |

Experiment 6 (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 06 (Simulation):

Objective: This software experiment will be performed to learn the simulation steps for being familiar with series resonance in AC Circuits 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:

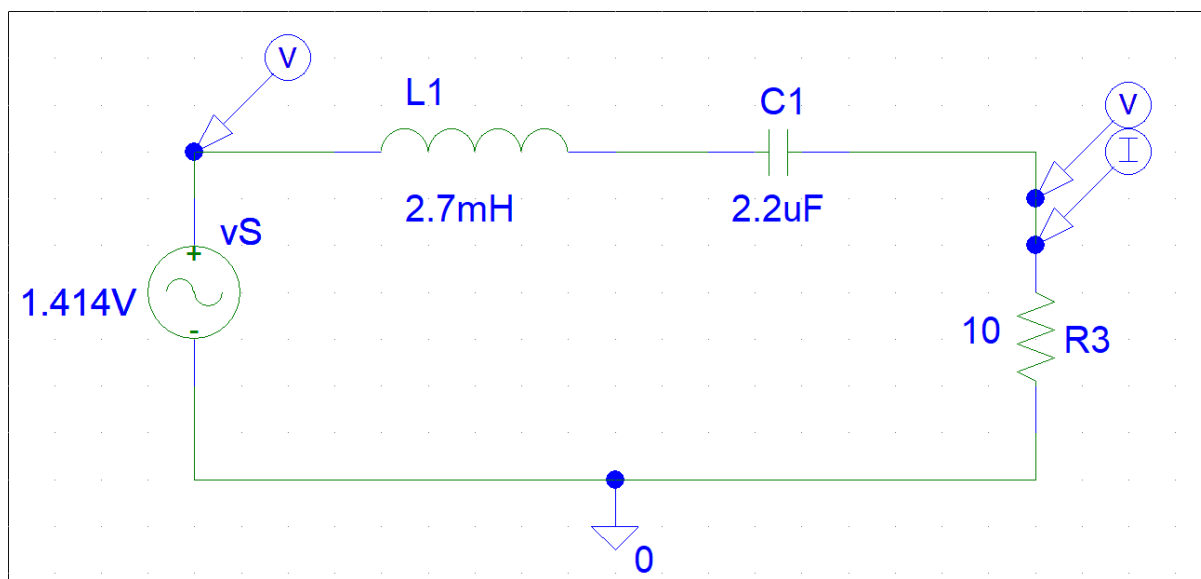
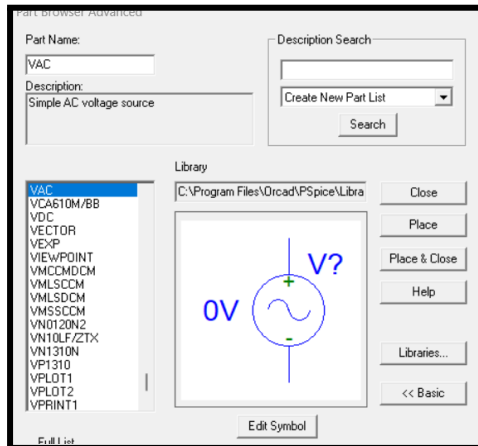
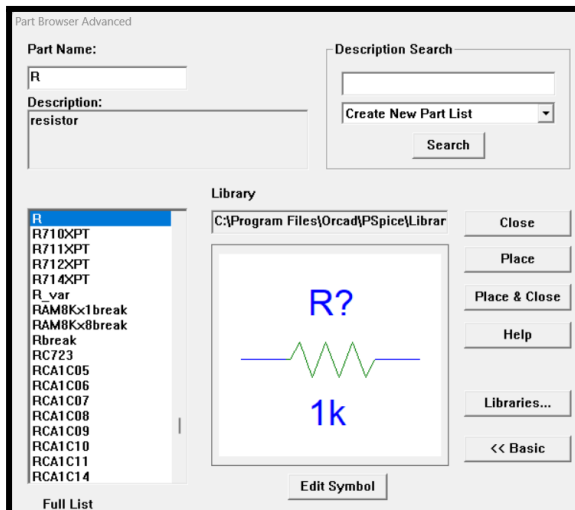


Figure: Circuit diagram for determining resonance simulated in Pspice Schematics

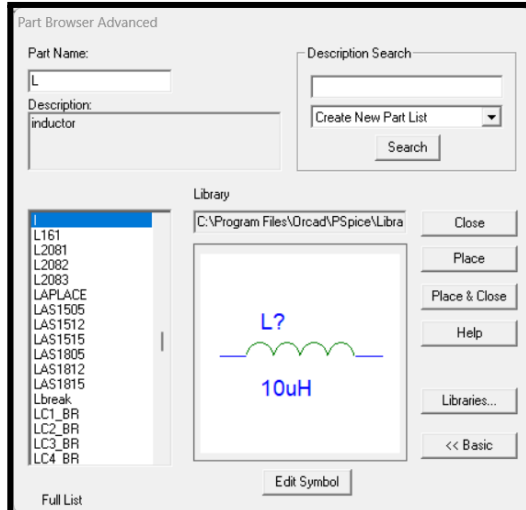
Tools, values and parameter setup menu:



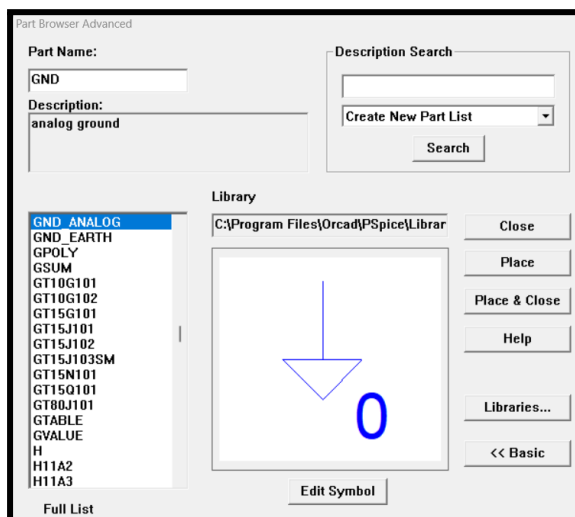
Voltage Source (VAC)



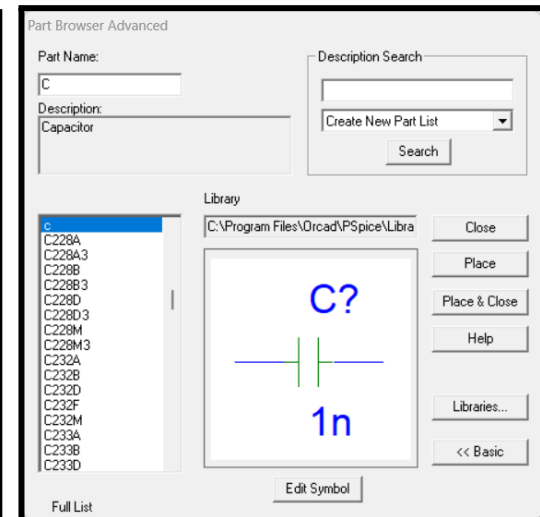
Selection of Resistor



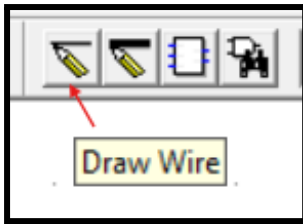
Selection of Inductor



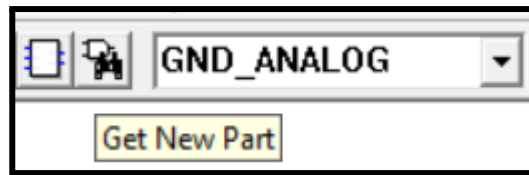
Selection of Ground



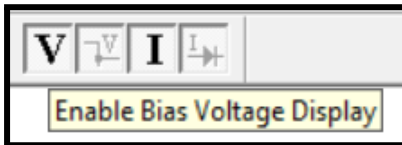
Selection of Capacitor



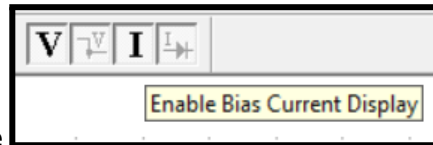
Wire tool



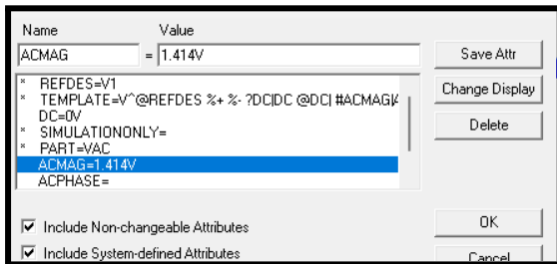
Parts menu



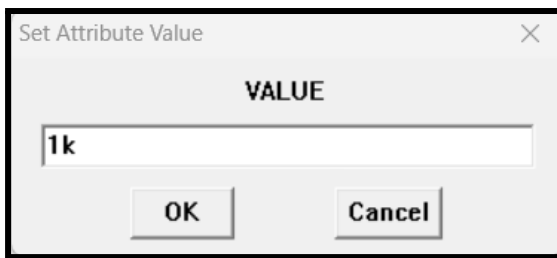
Bias Voltage



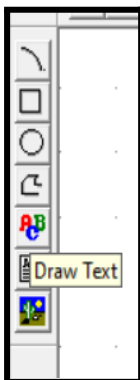
Bias Current



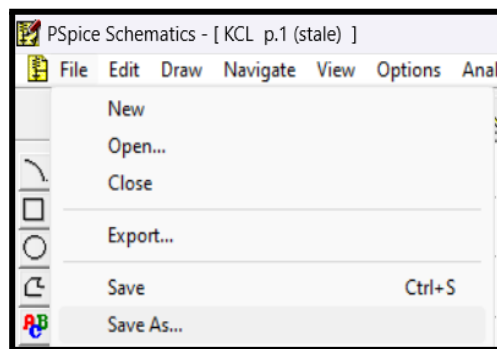
Values set in VAC



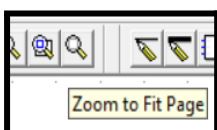
Resistor set (R value set)



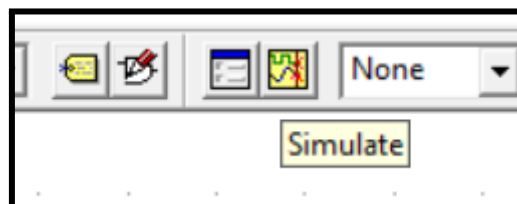
Text tools



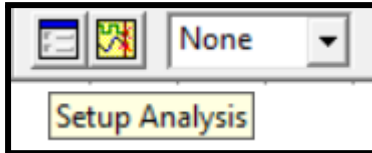
File saving



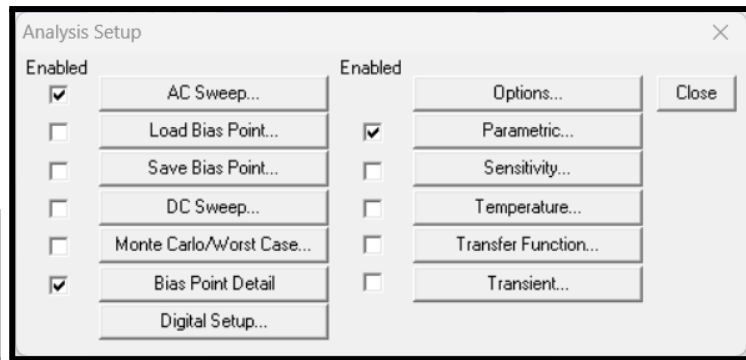
Zoom



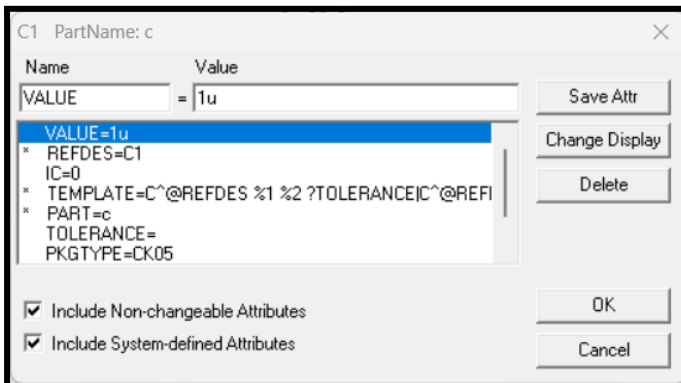
Begin Simulation



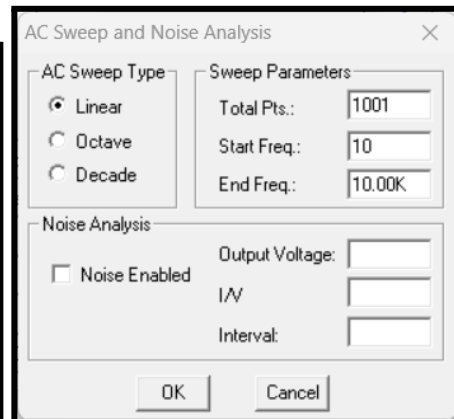
Setup Analysis Icon



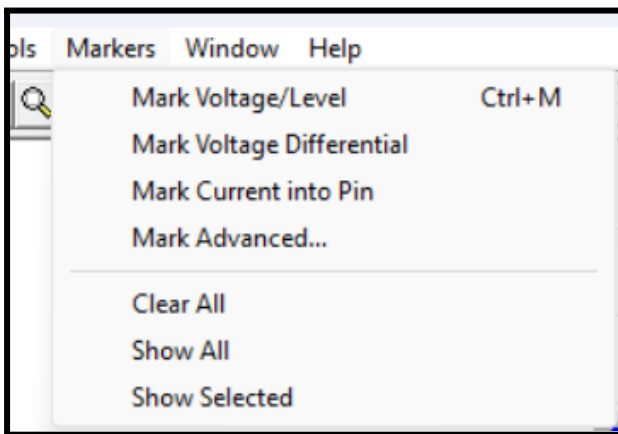
Analysis Setup Menu (AC Sweep)



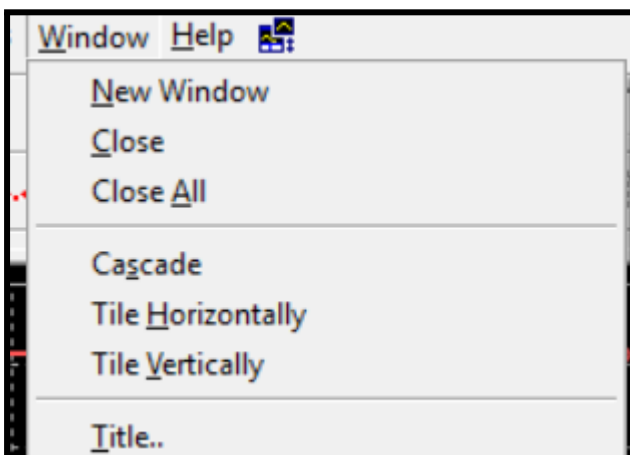
Values set in Capacitor



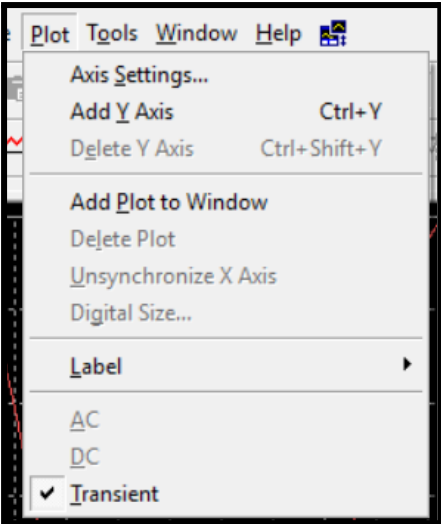
AC Sweep values



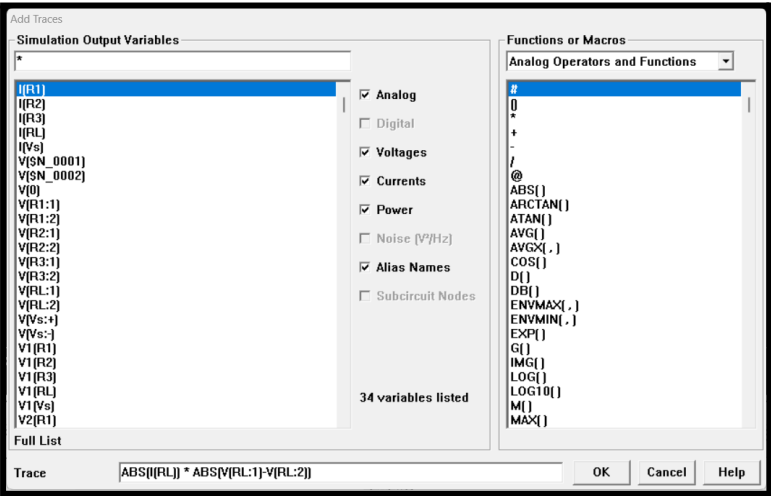
Marker Menu



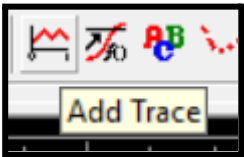
New Window Menu



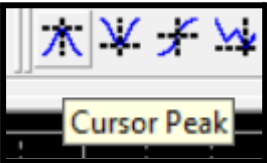
Adding new plot



Add trace value



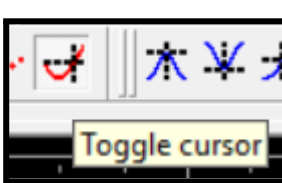
Add trace tool



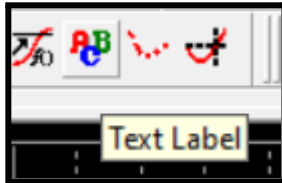
Cursor Peak tool



Mark Label Tool



Toggle Cursor Tool



Text Label tool



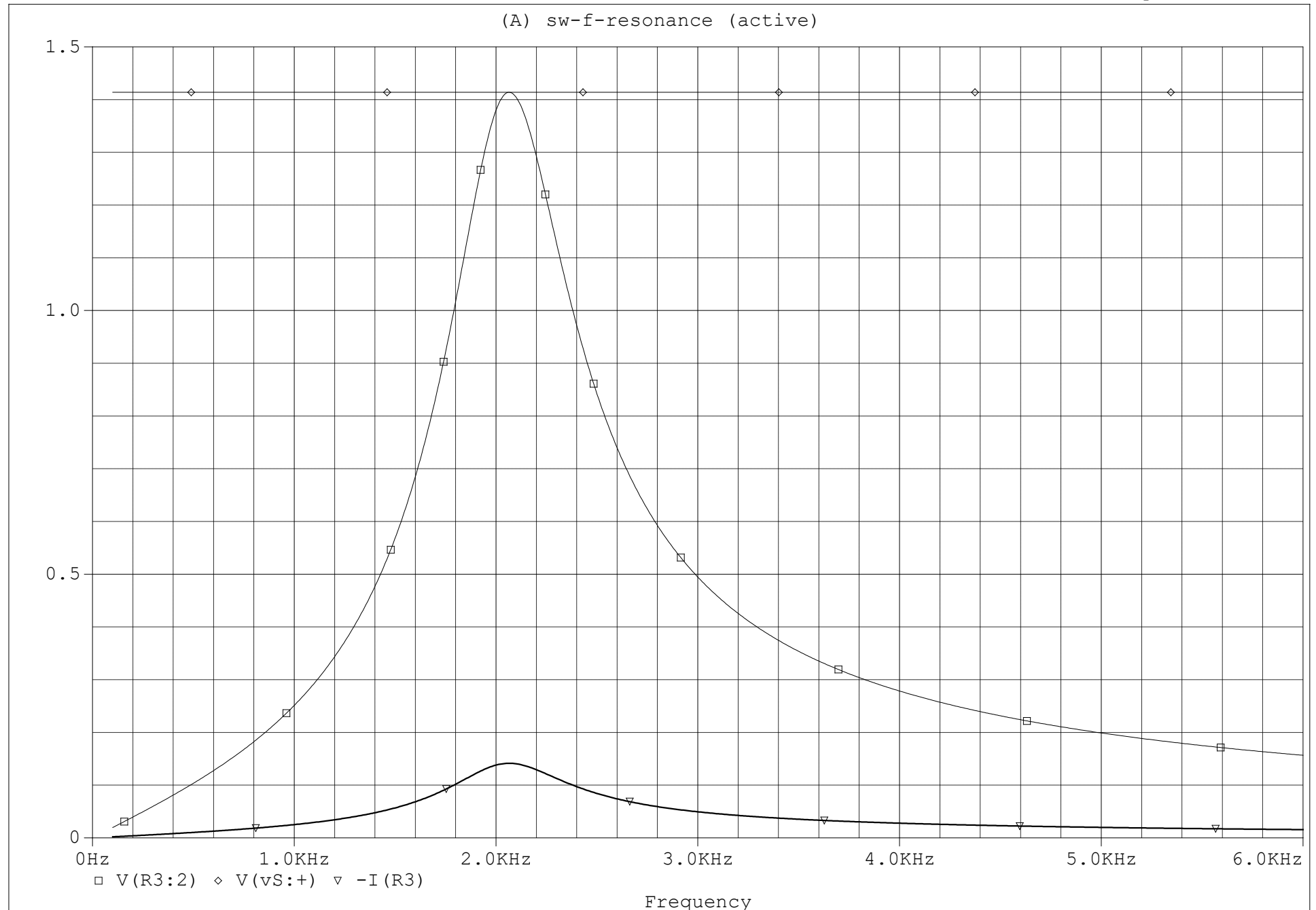
Selected Plot

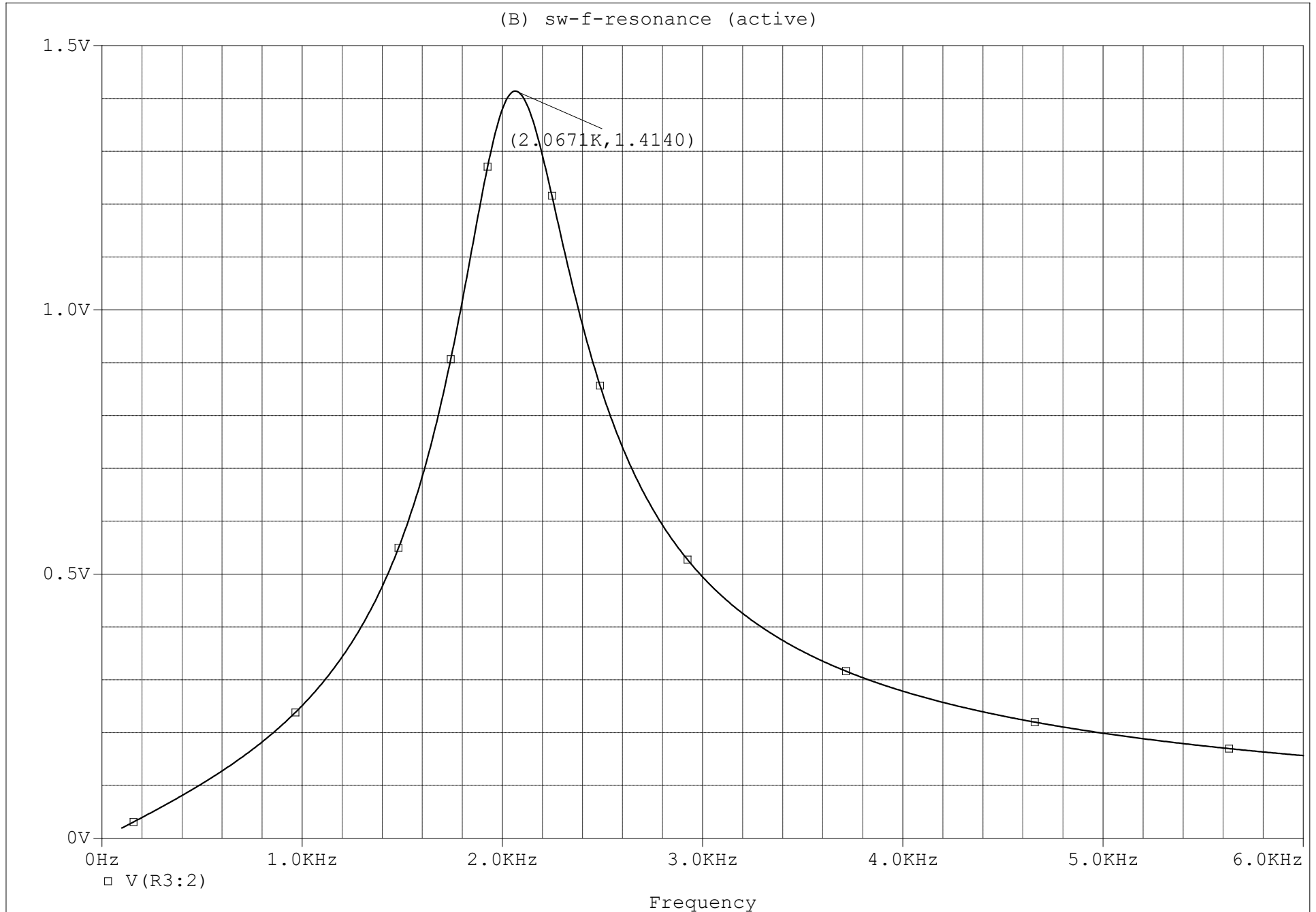


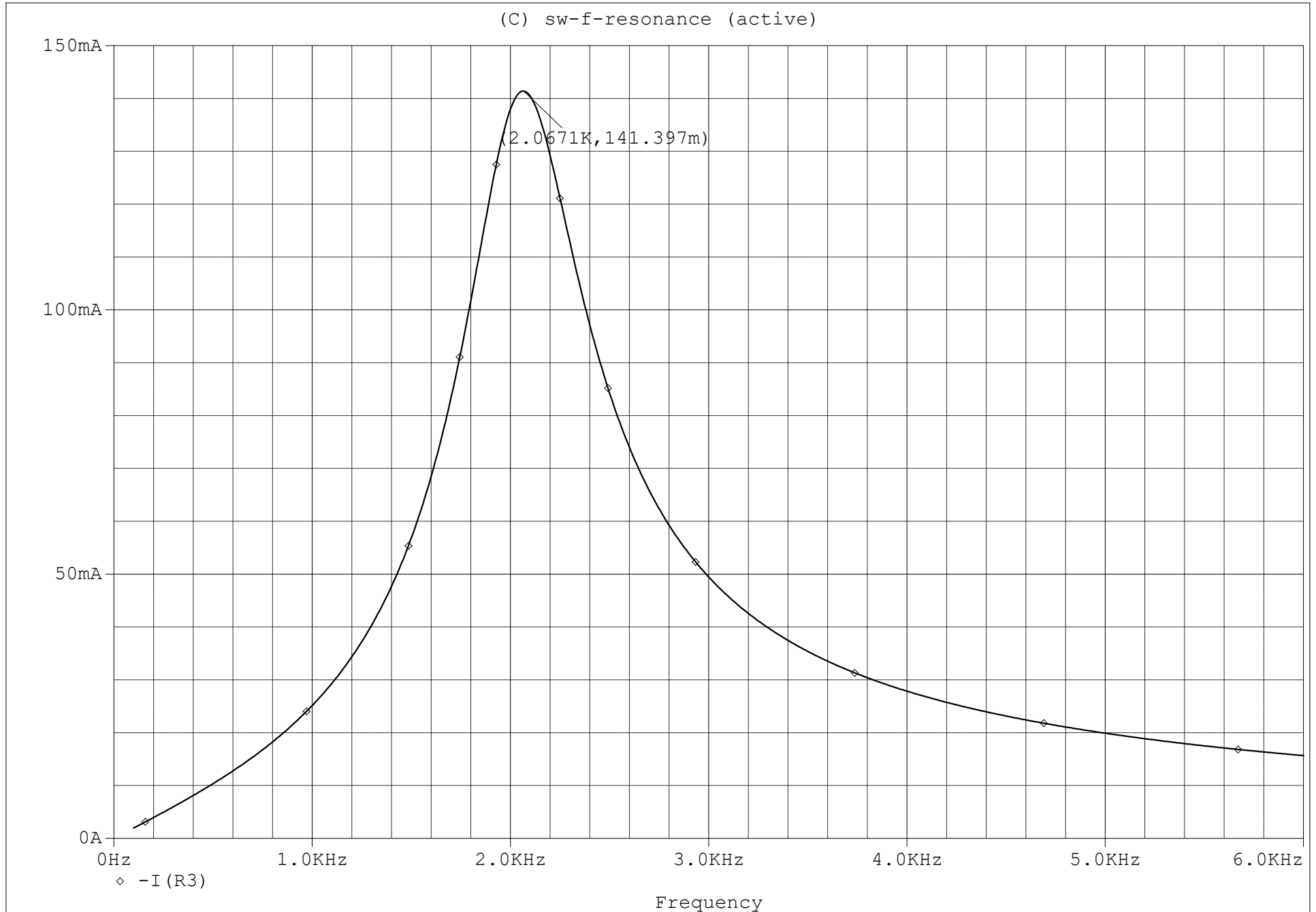
After enabling cursor peak (Probe Cursor Window)

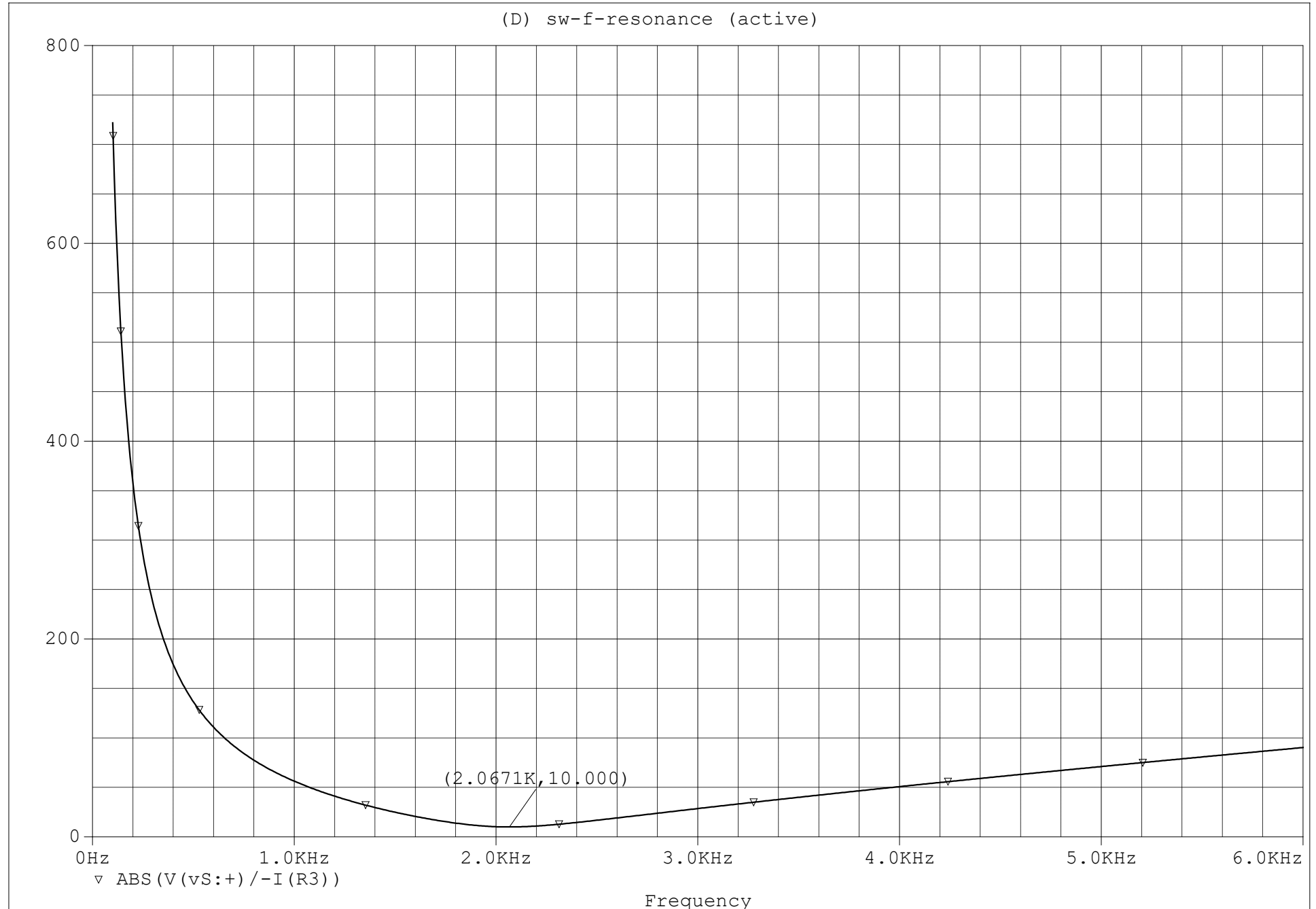
Experiment Procedure:

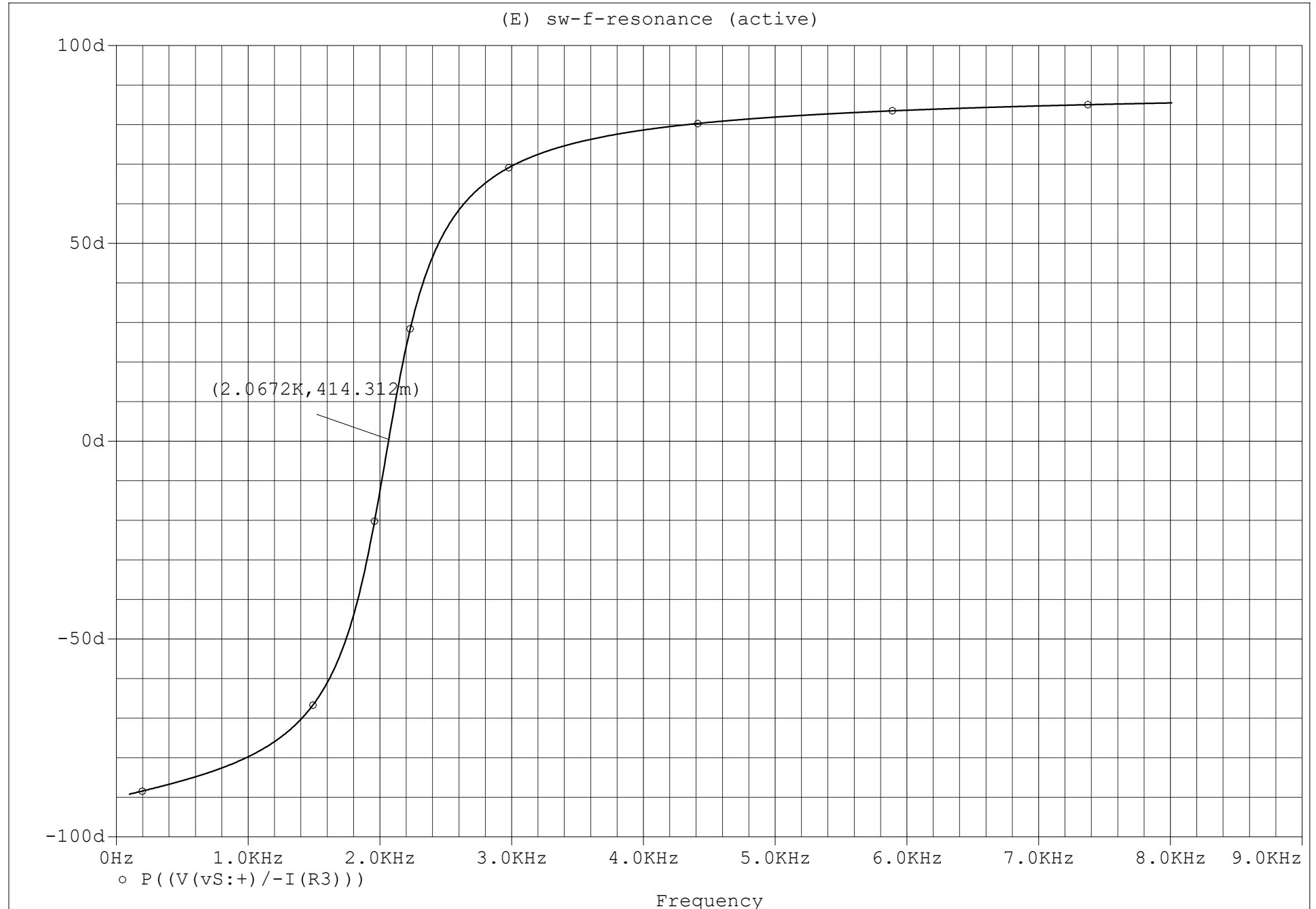
1. Open Pspice Schematics software
2. Open the parts menu
3. Search the necessary parts and place them according to the diagram
4. Using the wire tool connect all the parts in the circuit
5. Rename all the parts for easier identification
6. Use the draw text and text box tool to mark necessary information
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, 100 and End 8K.
11. Double click on the capacitor and inductor to set $IC=0V$.
12. Add Mark Voltage/Level and Current into Pin on the circuit.
13. Begin circuit simulation.
14. Shift to the graph interface menu
15. Use toggle cursor to mark peak of all graphs
16. Calculate the necessary information and fill the data tables.

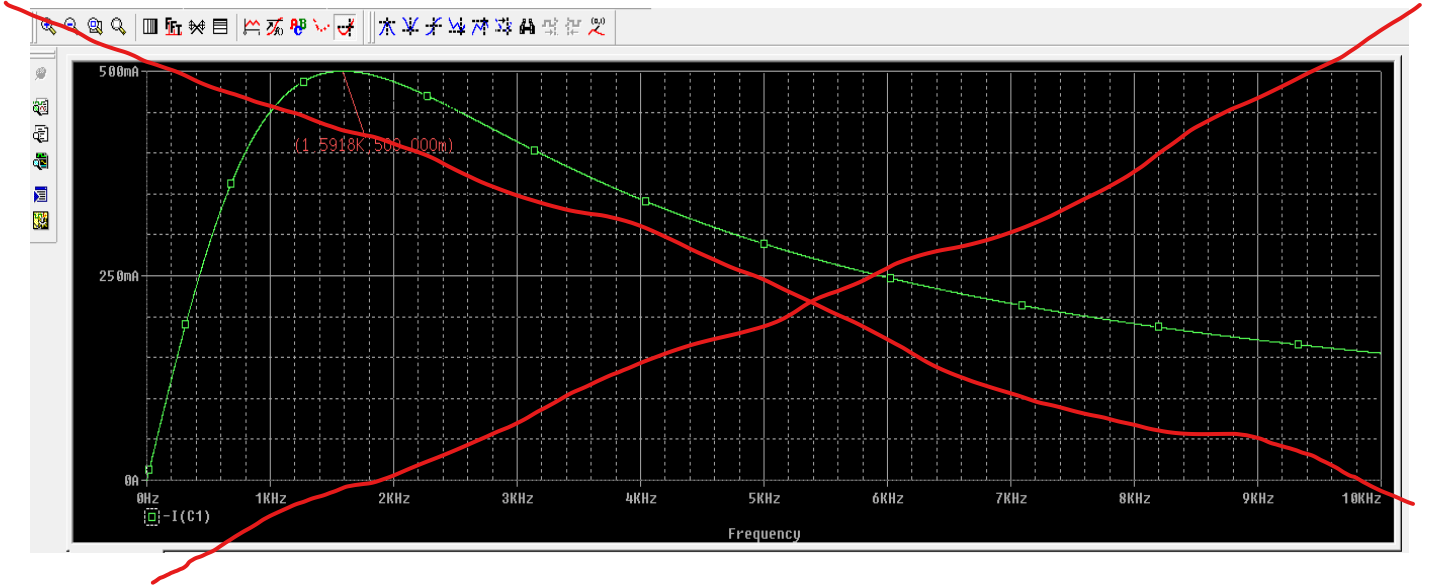












- The X co-ordinate represents the Resonant frequency, f_0

Data:

| Table for $ V_R $ | |
|----------------------|----------------|
| $ V_R _{max}$ (V) | f_s (kHz) |
| 1.414 | 2.0671 |

| Table for $ I $ | |
|---------------------|----------------|
| $ I _{max}$ (mA) | f_s (kHz) |
| 141.397 | 2.0671 |

| Table for $ Z $ | |
|---------------------|----------------|
| R (Ω) | f_s (kHz) |
| 10 | 2.0671 |

| Table for $\angle Z$ | |
|----------------------|--|
| f_s (kHz) | |
| 2.0672 | |

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 resonance occurs when the resultant reactance is zero. Resonance is a particular situation that may occur in an electric circuit containing both inductive and capacitive elements. All steps necessary to complete the circuit and experiment were mentioned in the instructions. In conclusion, we were able to successfully get familiarized with the series resonance phenomenon of the given circuit.

Schematics Drive Link: [EEE203L-EXP6](#)