UNIVERSITY OF CALIFORNIA AT BERKELEY
College of Engineering
Department of Electrical Engineering and Computer Sciences

EE105 Lab Experiments

Prelab 1: Introduction to SPICE

Name:

Lab Section:

1. Before you begin, you will need some instructions on how to run and use HSPICE. We have a tutorial specifically for EE105 containing instructions for both the Windows and UNIX versions of HSPICE. The tutorial includes a number of examples to help you get started.

   - HSPICE Tutorial

2. Figure 1 shows a simple network of resistors and a capacitor. Find the transfer functions v_x/v_s and v_y/v_s (as functions of $j\omega$) by hand.

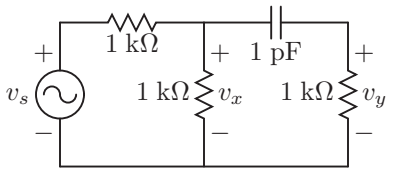
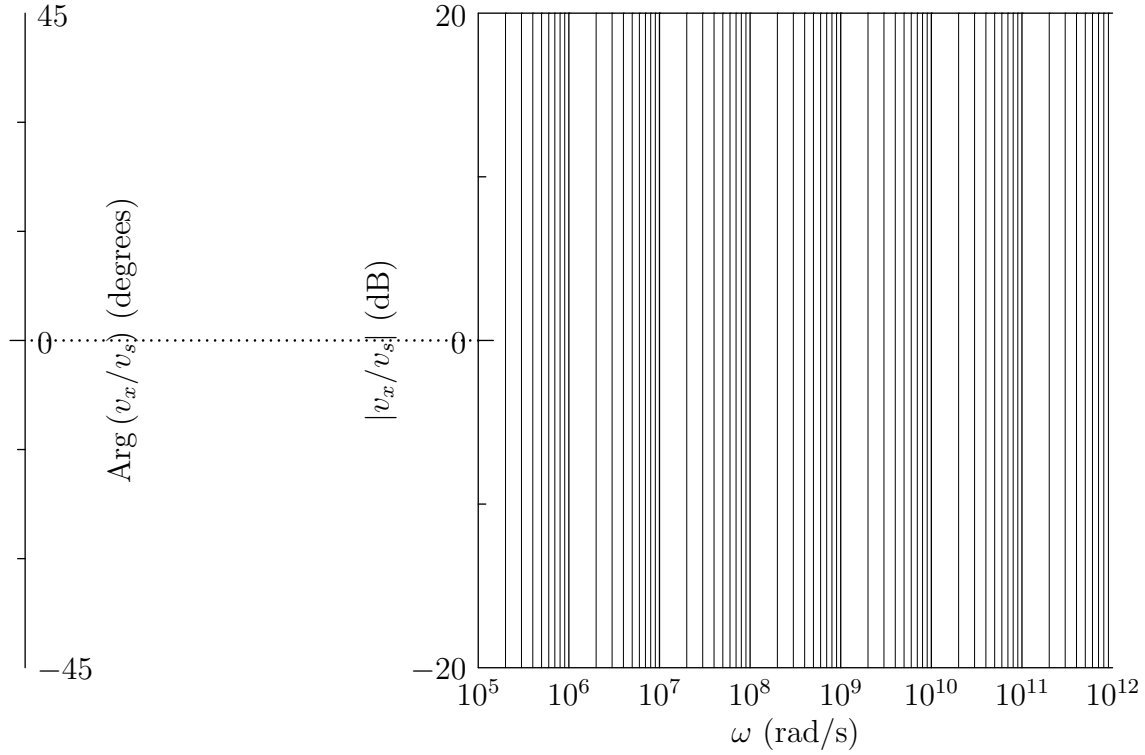


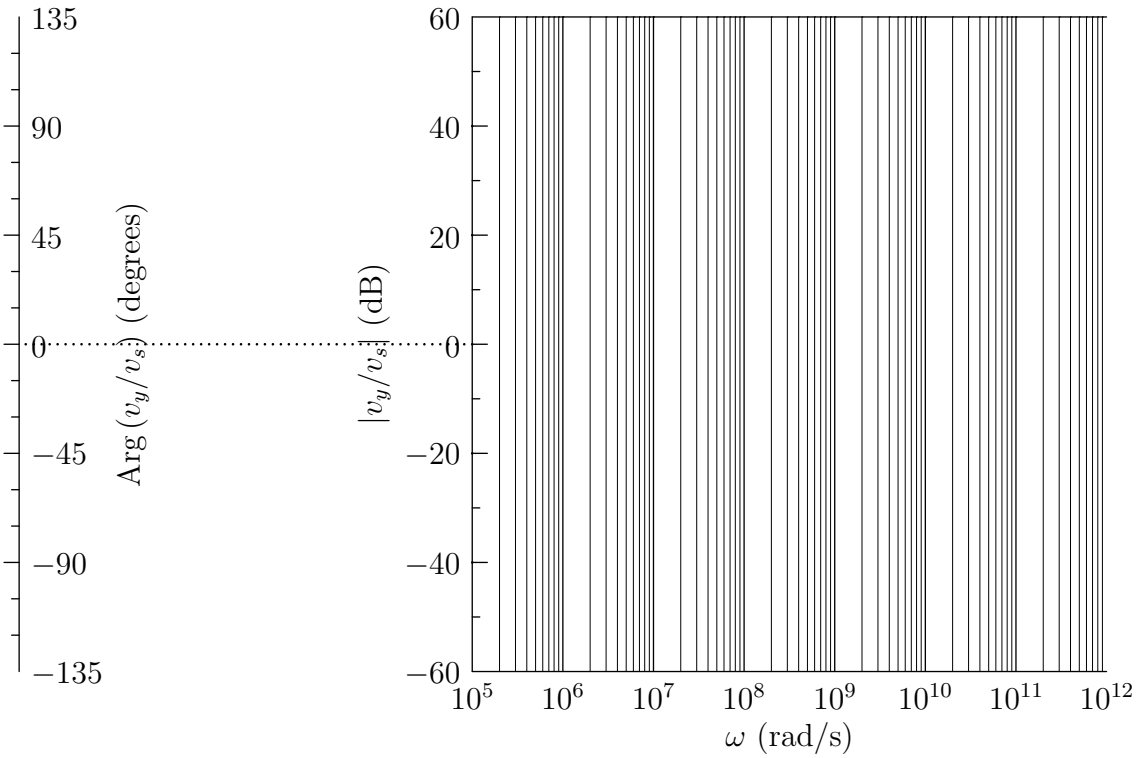
Figure 1: Circuit to simulate for the prelab

$\frac{v_x}{v_s} =$

$\frac{v_y}{v_s} =$

3. Use the transfer functions you derived to make Bode plots (magnitude and phase) for v_x/v_s and v_y/v_s. On each graph, indicate which curve is the magnitude and which is the phase. *Note: If you need a review of Bode plots, read the Bode Plot Tutorial.*





4. Now write an HSPICE netlist for this circuit.

5. Perform an AC analysis of this circuit from 100 kHz to 1 THz in HSPICE. Attach your netlist to this worksheet.

6. Use Awaves to generate Bode plots (magnitude and phase) for the circuit in Figure 1 for v_x/v_s and v_y/v_s. Do the results agree with your hand calculations? *Hint: The default x-axis in Awaves has units* Hz*, not* rad/s*. Note: If you do not have a 3-button mouse, you will not be able to generate a magnitude Bode plot. Instead, simply plot $|v_x/v_s|$ and $|v_y/v_s|$ on a log-log plot by right-clicking the axes and selecting "Set Logarithmic Scale".*

7. Turn in your hand calculations, netlist, and Bode plots (hand-made and simulated) at the beginning of your lab for checkoff. **Please print your plots with a white background. You can set a white background in Awaves by clicking *Window* → *Flip Color*.**