The circuit shown in Figure 1 is in the **phasor domain** with impedances given for each of the passive components, in order to simulate with LTSpice we must find the equivalent component values. For an inductor we have $Z_L = j\omega L$ so $L = Z_L / (j\omega)$ and similarly for a capacitor $C = -j / (\omega Z_C)$. Of course for a resistor, $R = Z_R$. The simulation can be done at any frequency, but we can find the component values most easily if we simulate at $\omega = 1$ rad/s. Then, for the circuit shown in Figure 1, $R = 30 \, \Omega$, $L = 50 \, \Omega$, and $C = \{1/30\} \, \text{F}$ (if you enclose an expression in curly braces then LTSpice will “do the math”). You must then simulate at a frequency (in Hertz) of $f = \{1/(2*\pi)\}$ (again, let LTSpice do the math) so that the impedances will be the same as in the original circuit.

1. Use LTSpice and AC Analysis (as described in the preceding paragraph) to find the phasor node voltages and current in the circuit shown in Figure 1. Cut and paste the AC Analysis results as a comment onto the schematic. Include all lab partner names as a comment at the top of the schematic. Use the node voltages to determine the phasor voltages across each of the components.

   $V_a = \_\_\_\_$

   $V_b = \_\_\_\_$

   $V_c = \_\_\_\_$

   $I = \_\_\_\_$

   $V_R = \_\_\_\_$

   $V_L = \_\_\_\_$

   $V_C = \_\_\_\_$

2. Use the AC power formula derived in lecture ($P = \frac{1}{2} \Re\{V I^*\}$) and the phasor current and component voltages to compute the average power absorbed by (or delivered to) each component. Also compute the power absorbed by the voltage source ($P_S$).

   $P_R = \_\_\_\_$

   $P_L = \_\_\_\_$

   $P_C = \_\_\_\_$

   $P_S = \_\_\_\_$

**Figure 1: Phasor Domain Circuit**

**Deliverables:** This sheet (with all lab partner names at the top) and your LTSpice schematic.