Using Complex Numbers in Circuit Analysis
and
Review of the Algebra of Complex Numbers

The purpose of this note is to review the algebra of concept numbers and show how they can be used to simplify analyses of linear circuits. This is the basic theory behind how PSpice handles linear circuits (and linear small-signal approximations of non-linear circuits). The basic techniques are also widely used in many types of linear analysis found in physics and engineering (electrical or mechanical), so if you are majoring in one of those fields, you will need to become familiar with these techniques and concepts sooner or later, even though we will not use them extensively in this course. I illustrate the technique with a simple, standard LRC circuit and then with a non-trivial example, and I show how the same example can be easily solved by PSpice. At the end I discuss briefly how PSpice can handle much more complicated circuits, including non-linear circuits, by extending the same mathematical techniques.

Complex numbers are commonly used in electrical engineering, as well as in physics. In general they are used when some quantity has a *phase* as well as a *magnitude*. Such a situation occurs when one deals with sinusoidal oscillating voltage and current (other examples in physics include optics, where wave interference is important, and quantum mechanical wave functions). I want to emphasize that complex numbers are used to make calculations *easier*! Do not be intimidated by trying to imagine what an imaginary number is. There is no need for that. Instead, realize that there is nothing imaginary about the phase of a voltage waveform, and there is nothing particularly complex about working with complex numbers. Just look at them for now as a useful tool that you may as well start getting used to.[1] Complex numbers as used here are equivalent to the "phasors" used for this purpose in elementary physics textbooks. The advantage of calling them complex numbers instead of phasors is that you can make use of the (hopefully) familiar algebra of complex numbers.
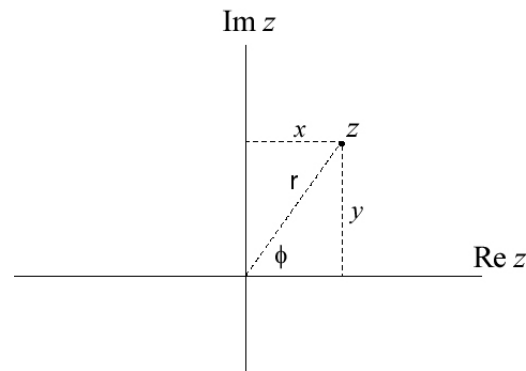
Figure 1. A complex number z shown in the complex plane. x is the real part of z, y is the imaginary part of z, r is the magnitude of z, and ϕ is the phase of z.

You may also refer to Appendix B of Horowitz and Hill for a review of complex numbers.

---

[1] If you haven't already, then later you can study the beautiful and seemingly magical mathematics of functions of complex variables, but there is no need for that in this course.

Representations of Complex Numbers

Let the symbol z represent a complex number, while x and y are its real and imaginary parts: $z = x + jy$, where $j \equiv \sqrt{-1}$.[2] The complex conjugate of z is $z^* = x - jy$. In general, to change a complex number into its complex conjugate, simply change j to $-j$ everywhere. Then all of the normal rules of algebra apply, with the understanding that $j^2 = -1$:
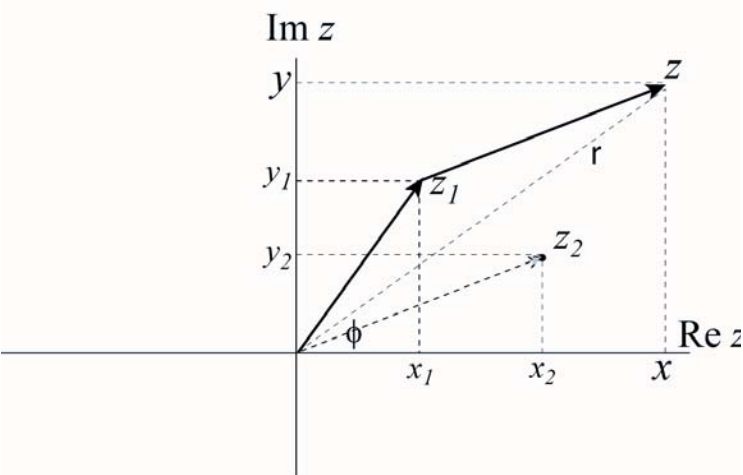
Figure 2. Graphical representation of the addition of two complex numbers. Note how the two numbers add as vectors in 2-D when the real and imaginary parts are simply added as $x=x_1+x_2$ and $y=y_1+y_2$ respectively.

$$z^2 = (x + jy)^2 = x^2 - y^2 + j2xy$$

$$|z|^2 \equiv z \cdot z^* = (x + jy) \cdot (x - jy) = x^2 + y^2$$

$$z_1 + z_2 = (x_1 + jy_1) + (x_2 + jy_2) = (x_1 + x_2) + j(y_1 + y_2)$$

Since we want to use complex numbers to represent phases of waveforms, it is essential to understand the polar, as well as Cartesian, form of a complex number (see Figure 1). This is no different from conversions between rectangular and plane polar coordinates, except that instead of labeling the axes x and y we label them Re and Im. Think of the complex number as a 2-dimensional vector in a plane. Addition of two complex numbers looks exactly like vector addition, either graphically or algebraically (as in the addition example above and in Figure 2). And $|z| = r = \sqrt{x^2 + y^2}$ is just the length of the vector.

Looking at Figure 1, you can see that

$$x = r\cos\phi$$

$$y = r\sin\phi$$

so we can write our complex number as

$$z = r\cos\phi + j \cdot r\sin\phi = r \cdot (\cos\phi + j \cdot \sin\phi) = re^{j\phi}$$

where the last step makes use of Euler's formula: $e^{j\phi} = \cos\phi + j \cdot \sin\phi$. This essential relation points directly to one reason why complex numbers make circuit analysis easier. Instead of representing a sinusoidal voltage or current as a sine or cosine function, we can represent it as an exponential. Exponentials are much easier to work with algebraically!

---

[2] Electrical engineers, and our textbook, use this notation, but physicists and physics textbooks (and mathematicians) generally use the symbol i instead of j.

Unless you love dealing with complicated trig identities, choose the complex exponential over the sine and cosine functions!

Here is a summary of the two representations of a complex number:

$$z = x + jy \qquad x = r\cos\phi \qquad\qquad y = r\sin\phi$$
$$z = re^{j\phi} \qquad r = \sqrt{x^2 + y^2} \qquad \phi = \arctan(y/x)$$

Keep in mind when calculating the phase ϕ that there is in general an ambiguity of $\pm\pi$ radians, which you have to resolve by looking at the signs of both y and x. The arctan function on your calculator will always return an angle in the range $-\frac{\pi}{2} \to +\frac{\pi}{2}$. You can avoid this ambiguity if you use the special function on your calculator for transforming between rectangular and polar coordinates. Also, computer languages usually include an inverse tangent function with two separate arguments for y and x, which will return the correct value of ϕ in the range $0 \to 2\pi$ or $-\pi \to +\pi$ (e.g. ATAN2 in FORTRAN).

Basic Algebra with Complex Numbers

Addition and subtraction of complex numbers are most easily done using the Cartesian (rectangular) form, for the same reason that vectors are most easily added and subtracted in Cartesian components.

$$z_1 + z_2 = (x_1 + x_2) + j\cdot(y_1 + y_2)$$
$$z_1 - z_2 = (x_1 - x_2) + j\cdot(y_1 - y_2)$$

However, multiplication and division are most easily done using the polar form, making use of the properties of the exponential function:

$$z_1 \cdot z_2 = r_1 \cdot r_2 \cdot e^{j(\phi_1 + \phi_2)}$$
$$\frac{z_1}{z_2} = \frac{r_1}{r_2} \cdot e^{j(\phi_1 - \phi_2)}$$

Nevertheless, multiplication in the rectangular form is straightforward:

$$z_1 \cdot z_2 = (x_1 + jy_1)\cdot(x_2 + jy_2) = (x_1 x_2 - y_1 y_2) + j\cdot(x_1 y_2 + x_2 y_1).$$

Division can be accomplished either by converting numerator and denominator to the polar form and using the equations above, or by multiplying the numerator and denominator by the complex conjugate of the denominator. This is an exercise that is frequently required in circuit analysis:

$$\frac{z_1}{z_2} = \frac{x_1 + jy_1}{x_2 + jy_2} \cdot \frac{x_2 - jy_2}{x_2 - jy_2} = \frac{(x_1 x_2 + y_1 y_2) + j\cdot(x_2 y_1 - x_1 y_2)}{x_2^2 + y_2^2}$$

In this way we can separate the real and imaginary parts of the ratio, from which we can calculate the magnitude and phase, if necessary. Do not try to memorize such a formula! It is the simple technique of multiplying the numerator and denominator by the complex conjugate of the denominator that you should remember. Executing this technique always guarantees that the resulting denominator will be real, with the imaginary number j appearing only in the numerator.

Working with Complex Impedance

Voltage and current are always real, observable quantities. In a linear A/C circuit with a sinusoidal stimulus, they will always have a form like $V(t) = V_0 \cos(\omega t + \phi)$. The algebraic complexities come in when we introduce capacitors and inductors, which produce $\pm 90°$ changes in phase. Adding sines and cosines with differing phases is algebraically painful, requiring expertise with trig identities. However, if the circuit is described by linear differential equations, then we can simplify life by adding an imaginary part to the voltage or current:

$$V(t) = V_0 \cos(\omega t + \phi) + j \cdot V_0 \sin(\omega t + \phi) = V_0 e^{j \cdot (\omega t + \phi)}$$

with the understanding that the observed voltage is just the real part of this expression. Now, when you do your circuit analysis you get to deal with the simple properties of the exponential function instead of nasty trig identities. When done, just take the real part of the final result, and that is your answer. As you will see, what this procedure will do for you is turn a set of linear differential equations into a set of linear algebraic equations.[3]

This works *only because the circuit is a linear circuit, described by linear differential equations.* Since linear equations do not involve any squares, square roots, and so forth of the voltage or current, or multiplication of one voltage or current by another, the real and imaginary parts don't get mixed up. Take a look at the equations in the previous section. The addition and subtraction equations do not mix up the real and imaginary parts, but the equations for multiplication and division do. Multiplying a complex number by a real constant also obviously does not mix up the real and imaginary parts. Essentially, a linear equation is one that will not mix up the real and imaginary parts of the voltages and currents. From a practical standpoint, a linear circuit is one that includes only passive components (resistors, capacitors, and inductors) plus voltage and/or current sources. No diodes, transistors, vacuum tubes, etc. are allowed.

It is perhaps worth mentioning here that the same formalism, with the same advantages of using complex numbers, works in mechanics when dealing with small, harmonic oscillations of mechanical systems.

The recipe for obtaining the steady-state[4] harmonic response of a linear circuit is straightforward. Write each non-static voltage or current source as a complex number:

$$V_0 e^{j\phi} \text{ or } I_0 e^{j\phi}$$

where the phase ϕ can be taken to be zero if there is only one source. Otherwise the relative phases of the sources must be taken into account. Then treat each passive component as an impedance

$$\text{Resistor: } Z = R$$

$$\text{Capacitor: } Z = \frac{1}{j\omega C}$$

$$\text{Inductor: } Z = j\omega L$$

---

[3] This procedure works for voltage and current sources that are sinusoidal (harmonic). However, a non-sinusoidal periodic source can be written as a Fourier series of sines and cosines. Each term in the series can be treated by the method described here. Since the circuit is linear, the response is just the linear superposition of the responses to the individual harmonic Fourier components.

[4] By steady-state, I mean turn all the switches on and then wait long enough for the transient behavior to dampen out and disappear. Usually the wait is very short, less than a blink of the eye.
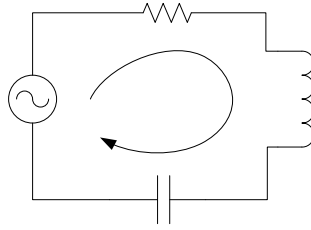
Figure 3. LRC series circuit.

where in general the impedance relates the voltage *across* a component to the current passing *through* the component according to a generalization of Ohm's law:

$$V = IZ$$

Use Kirchhoff's laws to write a set of linear equations for the currents and voltage in the circuit, exactly as you would do for a circuit made up of batteries and resistors. The only difference is that some of the "resistances" are imaginary, so what you end up with is a set of complex linear equations. Solve the equations for the currents and voltages. This is tedious to do by hand, but keep in mind that a computer can solve an amazingly large set of complex linear equations in an instant, using standard "canned" programs. Many scientific calculators also have built-in functions for solving sets of linear complex equations. Finally, express the resulting voltages and/or currents in polar form, from which you can read off the amplitude and phase of each current or voltage.

As an example not included in Horowitz and Hill, let's analyze the standard series LRC circuit (Figure 3) which has a voltage oscillator in series with a resistor, capacitor, and inductor. The differential equation for this circuit follows from adding up the voltage changes around the loop:

$$V_0 e^{j\omega t} - IR - L\frac{dI}{dt} - \frac{Q}{C} = 0 ,$$

where $V_0 e^{j\omega t}$ is the driving voltage, expressed as a complex quantity as suggested above, with an assumed phase $\phi = 0$. Using $Q = \int I dt$, we get an equation for the current:

$$L\frac{dI}{dt} + \frac{1}{C}\int I dt + RI = V_0 e^{j\omega t} .$$

This is readily solved by making the substitution $I = I_0 e^{j(\omega t + \phi)}$, which turns the differential equation into an algebraic equation:

$$\left(j\omega L + \frac{1}{j\omega C} + R \right) \cdot I_0 e^{j\phi} = V_0 .$$

The quantity in parentheses is exactly the "impedance" that one would get by using the impedance rules listed above for resistors, capacitors, and inductors, plus the rule that impedances in series simply add up. So, from now on do not bother to write down the differential equation! Just assume the rules for complex impedance and immediately write down the algebraic equation.
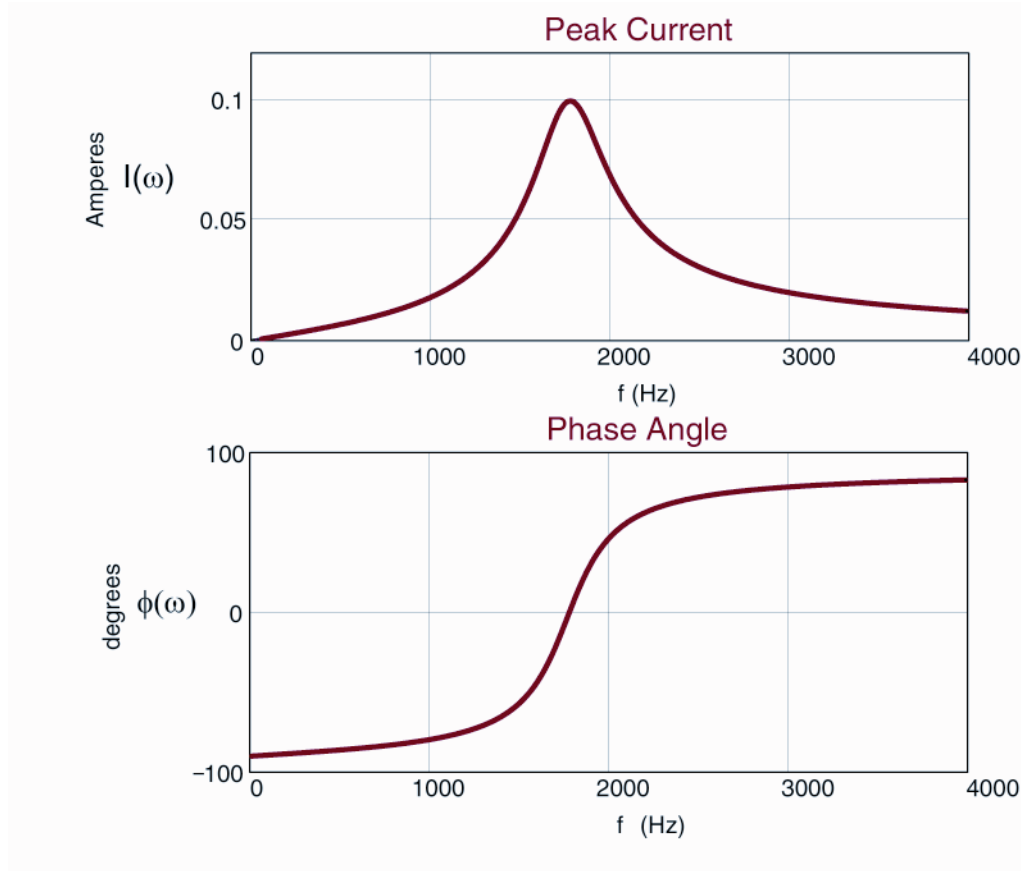
Figure 4. **Resonance curves for an LRC series circuit, with R=10 Ohms, C=2μF, and L=4mH.**

To analyze the series LRC circuit without writing any differential equation, we start with "Ohm's Law" for a reactive circuit:

$$ I = \frac{V_0}{Z} \text{ with } Z = R + \frac{1}{j\omega C} + j\omega L. $$

To do the division, I convert the impedance to polar form:

$$ Z = R + j \cdot \left(\omega L - \frac{1}{\omega C} \right) = \sqrt{ R^2 + \left(\omega L - \frac{1}{\omega C} \right)^2 } \cdot e^{j\phi_z} $$

with $\phi_Z = \arctan\left(\dfrac{\omega L - \dfrac{1}{\omega C}}{R} \right) = \arctan\left(\dfrac{\omega^2 - \omega_0^2}{\gamma\omega} \right)$ and $\omega_0 \equiv \dfrac{1}{\sqrt{LC}}$ and $\gamma \equiv \dfrac{R}{L}$.

So the current is given by

$$ I = \frac{V_0}{\sqrt{ R^2 + \left(\omega L - \frac{1}{\omega C} \right)^2 }} \cdot e^{-j\phi_z} = \frac{V_0 \cdot \dfrac{\omega}{L}}{\sqrt{ \gamma^2\omega^2 + \left(\omega^2 - \omega_0^2 \right)^2 }} \cdot e^{j\phi} $$

with $\phi = -\arctan\left(\dfrac{\omega^2 - \omega_0^2}{\gamma\omega}\right)$ for the phase of the current.

This result exhibits a resonance, with ω_0, the natural frequency of the circuit, being the frequency at which the impedance is minimum (and equal simply to R) and the current is maximum, with a phase shift of zero relative to the voltage. Also, γ is a measure of the amount of damping in the circuit and, thus, the width of the resonance curve. This resonance behavior is illustrated in Figure 4.

Analyzing a More Complex Linear Circuit

A more complicated looking example is shown in Figure 5, where the driving voltage is the real part of $V(t) = 10e^{i\omega t}$ volts, with angular frequency $\omega = 10^4$ radians/s. The impedance of the inductor is $j\omega L = 4j$ ohms, and the impedance of the capacitor is $1/j\omega C = -0.25j$ ohms. The objective is to find all the currents in the circuit and the equivalent impedance of the overall circuit, as seen by the voltage source. In this case there are 4 loops, so we will have 4 loop equations and 3 node equations. This goes beyond the complexity that you will see in homework or on any exam, but I throw it in as a random demonstration that the analysis is straightforward and can be formulated in a manner that makes a solution by computer fairly easy.

I prefer to work with the concept of "loop currents," in order to avoid having to write down the node equations. To understand this concept, look at the circuit as redrawn in Figure 6. The four loops are evident, and each is associated with a loop current. The current through the capacitor is clearly i_4, the current through the voltage source is i_1, and the current through the 2-ohm resistor is i_3. However, each of the other 4 components has two currents flowing through it. For example, the current flowing upward through the inductor is $i_3 - i_2$, and the current flowing downward through the leftmost resistor is $i_1 - i_2$. Now, let's apply Kirchhoff's loop law to loop #1, starting at the lower left corner and proceeding upwards through the voltage source, in the direction of loop current i_1:
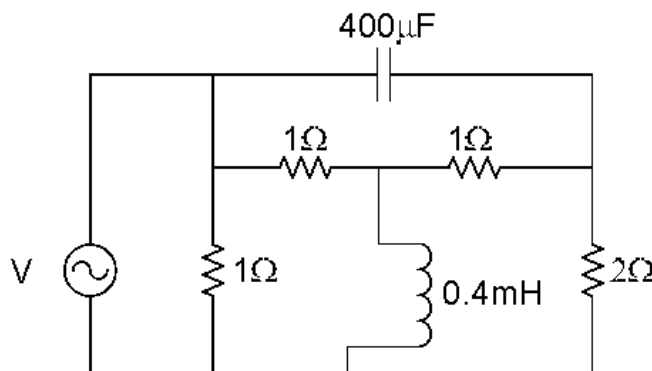
$$10 - (i_1 - i_2)\cdot 1 = 0$$

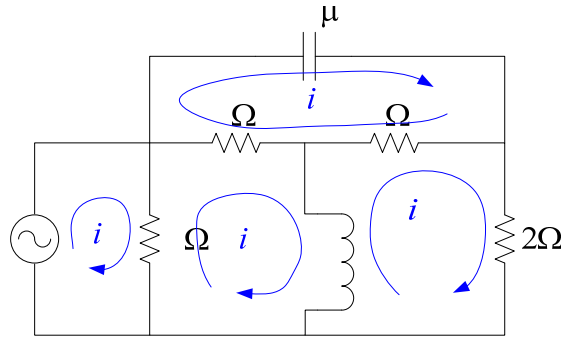**Figure 5. Example of a 4-loop linear circuit.**

Figure 6. The circuit redrawn with loop currents.

Do the same for loop #2, starting in the lower left hand corner and proceeding upwards through the 1-ohm resistor, in the direction of the loop current i_2 :

$$-(i_2 - i_1) \cdot 1 - (i_2 - i_4) \cdot 1 - (i_2 - i_3) \cdot 4 j = 0$$

The other two equations, for loops 3 and 4 respectively, are

$$-(i_3 - i_2) \cdot 4 j - (i_3 - i_4) \cdot 1 - i_3 \cdot 2 = 0$$

$$i_4 \cdot 0.25 j - (i_4 - i_3) \cdot 1 - (i_4 - i_2) \cdot 1 = 0$$

Such equations are easiest to deal with if organized in matrix notation:

$$\begin{pmatrix} 1 & -1 & 0 & 0 \\ 1 & -2-4j & 4j & 1 \\ 0 & 4j & -3-4j & 1 \\ 0 & 1 & 1 & -2+0.25j \end{pmatrix} \times \begin{pmatrix} i_1 \\ i_2 \\ i_3 \\ i_4 \end{pmatrix} = \begin{pmatrix} 10 \\ 0 \\ 0 \\ 0 \end{pmatrix}$$

Solving these equations by hand would be tedious and annoying, but doing it by computer with a program like Mathcad, Mathematica, or Matlab couldn't be easier. For example, in Mathcad let's call the matrix Z, so the equation looks like

$$Z \cdot I = V$$

Fill the 16 complex values into the matrix Z and the 4 values into V, and then type

$$I = Z^{-1} \cdot V$$

and you're done![5] The result is

$$I = \begin{pmatrix} 15.457 - 1.787 j \\ 5.457 - 1.787 j \\ 4.990 + 0.652 j \\ 5.213 + 0.084 j \end{pmatrix}$$

Here is how to interpret the result. For example, the current i_1 can be written in polar form as $i_1 = 15.56 e^{-j0.037\pi}$, so the current as a function of time is

$$i_1(t) = 15.56 \cdot \cos(\omega t - 0.037\pi).$$

---

[5] This is not the most efficient way to solve 4 linear equations, but for this purpose, who cares? The computer will finish the calculation before you can say "go"!
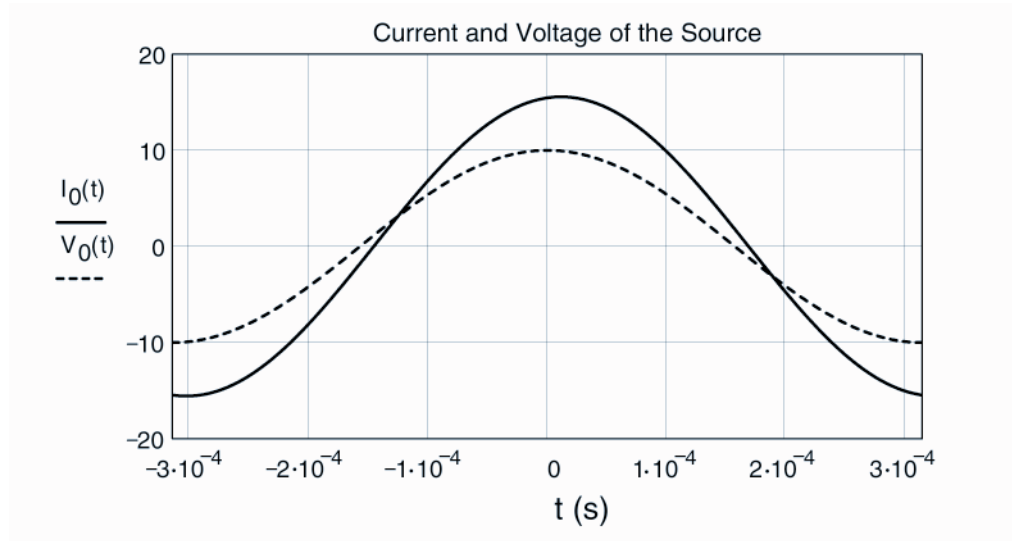
Figure 7. Plots of the voltage and current of the voltage supply as a function of time for a supply frequency of 10^4 radians/s. The current lags behind the voltage by several degrees.

That is, the current passing through the source lags behind the voltage by 0.037π radians, or about 6.7 degrees. Figure 7 shows how the current and voltage would look if displayed on an oscilloscope. The equivalent impedance of the circuit, as seen by the source, can be calculated from the ratio of the voltage and current of the source:

$$Z_{eq} = \frac{V}{i_1} = \frac{10}{15.56} \cdot e^{+j0.037\pi}.$$

Thus at this frequency, the circuit looks slightly inductive to the source.

Linear Circuit Analysis with PSpice

For illustration, let's analyze the same circuit as shown in Figure 5 using PSpice. (See my tutorial to learn how to so this.) Figure 8 shows the schematic as drawn using the "schematic capture" package that comes with PSpice. Although it is drawn slightly differently, you should convince yourself that it is the same schematic as in Figure 5. The
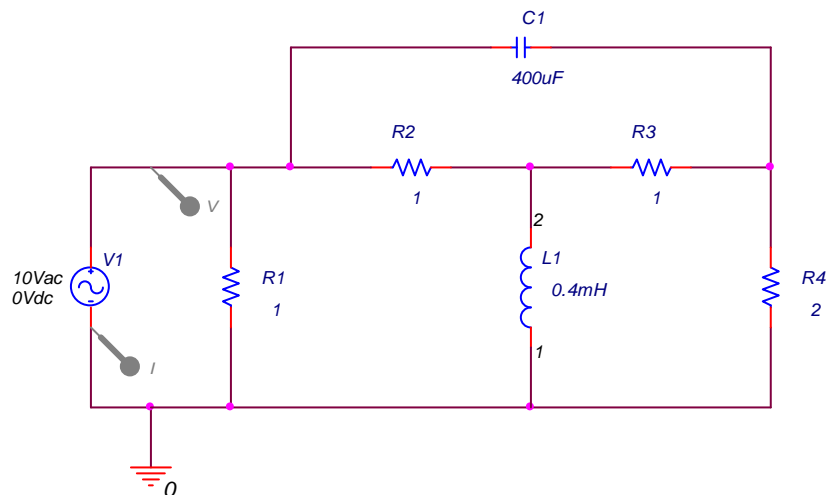

Figure 8. PSpice schematic of the same circuit shown in Figure 5. Note that I had to explicitly specify the ground point, and I also added two "probes," one for the current through the source and the other for the source voltage. The source frequency is not specified, because PSpice will scan the frequency over a large range.
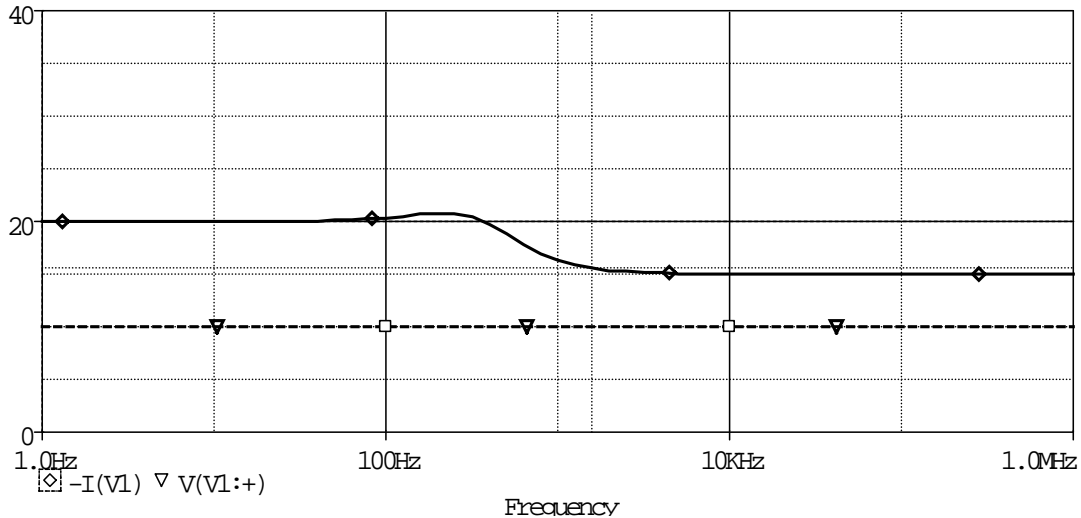
Figure 9. PSpice display of the supply voltage and current amplitudes versus frequency. The cursor crosshairs are placed on the frequency 1592 Hz (angular frequency 10,000 rad/s), where the current amplitude reads 15.56 amps.

PSpice "AC analysis" will analyze the circuit over a specified range of frequency, not just at a single frequency as we did above by hand. In this case I specified a logarithmic frequency scale from 1 Hz to 1 MHz, with 10 points analyzed per decade. There is really no point in plotting the result as sine waves (a la Figure 7, for example) because the relevant information is just the amplitude and phase at each frequency.[6] Figure 9 shows the voltage and current of the AC source (i.e. current i_1 in Figure 6), and Figure 10 shows the phase of the current (the voltage phase of the AC source is set to zero). In each plot the cursor is placed on the frequency analyzed above by hand, and the PSpice result agrees with that earlier calculation.

You should convince yourself that the results at very low frequency and very high frequency are obvious. At very low frequency the capacitor looks like an open switch while the inductor looks like a close switch of zero resistance (admittedly not a realistic inductor). Then the circuit looks like two 1-ohm resistors in parallel, for a total impedance of 0.5 ohms and a current of 20 amps. Similarly, at very high frequency the capacitor looks like a closed switch and the inductor looks like an open switch. In that case the circuit looks like a 1-ohm resistor in parallel with a 2-ohm resistor, for a total impedance of 2/3 ohms and a current of 15 amps.

---

[6] If you desire, you can run a "transient" analysis with a sinusoidal voltage source set at a specific frequency, in which case PSpice will output the result in the time domain, showing the voltage and current as a function of time. While that is rather pointless and a waste of CPU time for a linear circuit such as this one, a more interesting exercise is to input a non-sinusoidal source (e.g. a square wave) and look at the response (which could also be obtained by Fourier analysis as suggested in an earlier footnote).
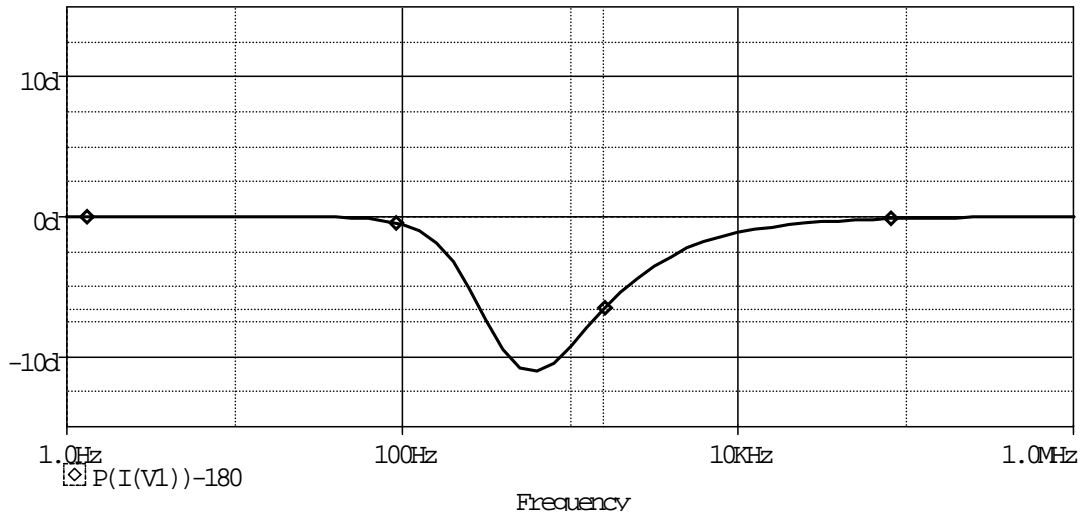
Figure 10. PSpice display of the supply current versus frequency. The cursor crosshairs are placed on the frequency 1592 Hz (angular frequency 10,000 rad/s), where the current phase reads −6.7 degrees. (Note that I plotted here the phase minus 180 degrees because PSpice takes the convention that a positive current is in the opposite sense of what is drawn in Figure 6. Subtracting 180 degrees is the same as multiplying the current by –1.)

Nonlinear Circuits

We have seen how complex numbers can make quick work of linear circuits, by turning a set of coupled linear differential equations into a set of linear algebraic equations that are easily solved with some help from a computer. But what about non-linear circuits? Well, one generally has to analyze a non-linear circuit by making a linear approximation around some initial guess (or around the actual bias point if that is already known). That may give a crummy result at first, but usually by iterating this procedure many times one can arrive at a good approximation to the solution. In any case, the analysis generally boils down to solving (perhaps many times) a set of linear equations, something that computers are very good at doing.[7]

The Spice program uses such a procedure. It uses various mathematical models of the nonlinear devices and treats the linear devices much as we have here (except that it can get more sophisticated and include imperfections such as leakage current in capacitors). First it calculates a "bias point," with the time dependence of all of the sources turned off. To do so, it starts with an initial guess for the DC currents everywhere and makes a linear approximation of all of the models around that guess. Then it solves the complete set of complex linear equations to get an improved set of currents. Then it makes a new linear approximation around those currents and solves the set of equations again. Eventually, with some luck, it finds that from one iteration to the next nothing changes much, at which point it assumes that the procedure has converged to the physical solution.

With the bias point in hand, Spice can then very quickly do an "AC" analysis. In such an analysis, the sources are assumed to be sinusoidal and have very small

---

[7] Be aware that in the case of a nonlinear circuit this process may not converge. That is never an issue with a linear circuit, for which just one iteration always gives the final solution.

amplitudes. Spice makes a linear approximation to the circuit around the bias point and calculates the response at each of a large set of frequencies, without doing any iteration. This can very quickly give you information on the frequency response of your circuit.

A more involved analysis is the "transient analysis." In this case, the sources are given whatever time dependence and amplitude you are interested in (square wave, triangle wave, or whatever). The amplitude is no longer assumed small, so Spice cannot get by with a linear approximation with a single iteration. Instead, it must take small time steps, such that in each step the voltages and currents don't change by very much. Then a few iterations can find a new convergence point after each step. Spice will vary the size of the step depending on how fast your source is changing. As you can imagine, it has to go very slowly in small steps when you feed it the edge of a square wave. For a large circuit this can take an enormous amount of computer time, but if you have the CPU cycles, it is worth it, because it will give you a very good idea of how your circuit will work before you build it. This is especially important if you are making integrated circuits, where after every new screw-up in your design you would have to wait 3 or 4 months, and spend at least several tens of thousands of dollars, to find out that it still doesn't work!