

Chapter 3

AC Analyses

3.1 Introduction

The AC analyses are a family of frequency-domain analyses that include AC analysis, transfer function (XF) analysis, scattering parameter (SP, TDR) analyses, and noise analysis. All of these analyses are based on the same underlying mathematical technique: phasor analysis [chua87]. As such, they all share common assumptions and characteristics. Phasor analysis computes the small-signal sinusoidal steady-state response of the circuit. This implies that the solutions computed by the AC analysis contain only sinusoids at the same frequency as the input signal. Thus, each signal is represented with only two numbers, one that gives the magnitude, and another that gives the phase. With such a simple representation for a signal, it is not possible to represent transient behavior. Indeed, the AC analyses do not compute the transient response of a network. Thus, unlike with transient analysis, it is difficult to judge the stability of a circuit by a simple inspection of the results. For example, if one used AC analysis to compute the response of a simple RC circuit, and then made the circuit unstable by replacing the R with $-R$, the response of the two circuits would be the same, except for the phase, which would be negated.

AC analysis computes the small-signal behavior of a circuit by first linearizing the circuit about a DC operating point. Since the AC analyses operate on a linear time-invariant representation, the re-

sults computed by the AC analyses cannot exhibit the effects normally associated with nonlinear and time-varying circuits: distortion and frequency translation. However, the AC analyses do provide a wealth of information about the linearized circuit and so are invaluable in certain applications. They are also, on the whole, much less temperamental than DC or transient analysis. The AC analyses are not subject to the convergence problems of DC, and the accuracy problems of transient. If the AC analyses are inaccurate, it is almost always because the component models are incorrect.

While the mathematical basis of all of the AC analyses is the same, the results they compute are different and they are applied in different situations. Both AC and XF analyses are used to compute transfer functions. Which one you used is decided solely on the basis of efficiency. AC analysis is used if the number of independent outputs is greater than the number of independent inputs. A typical example of when AC analysis is called for is when you want to debug a circuit by applying a single stimulus and inspecting the response by probing arbitrary node voltages and terminal currents at will. XF analysis is used if the number of independent inputs is greater than the number of independent outputs. An example of when XF analysis is preferred is when you want to compute several transfer functions, such as differential- and common-mode gains as well as positive and negative power supply rejection, to a single output.

S -parameter analysis is used to create an S -parameter N -port description of the circuit. This analysis is most commonly used on high-frequency analog circuits. Time-domain reflectometry (TDR) analysis is the time-domain equivalent of S -parameter analysis. Converting S -parameters into the time domain sometimes provides insight not available from frequency-domain results.

Finally, noise analysis is used to predict the steady-state response of the linear time-invariant circuit to the small random noise signals generated by the components. It does so in the frequency-domain where such calculations are very efficient.

3.2 AC Analyses Theory

One question engineers often ask about their circuits is “What is its frequency response?” or “What is its transfer function?”. This is equivalent to asking “How will it respond if I stimulate it with a small sinusoidal signal?”. To answer such a question, circuit simulators formulate a new set of equations by linearizing the circuit about the DC solution using a Taylor series expansion and solving them using phasor analysis. Given a circuit described with (1.3) with a DC operating point of v_{DC} , the Taylor series expansion is used to calculate the change in the solution (Δv) as a function of the change in the stimulus (Δu). The Taylor series expansion for

$$f(v) + u = 0 \quad (3.1)$$

is

$$f(v + \Delta v) + u + \Delta u = f(v) + u + \Delta u + \frac{\partial f}{\partial v} \Delta v + \frac{\partial^2 f}{\partial v^2} \Delta v^2 + \dots \quad (3.2)$$

Since $v + \Delta v$ is the solution to the perturbed circuit,

$$f(v + \Delta v) + u + \Delta u = 0. \quad (3.3)$$

The equation is simplified by substituting (3.1) and (3.3) to (3.2) and by assuming the perturbation in the stimulus (Δu) is small, which makes the higher order derivatives negligible.

$$\frac{\partial f}{\partial v} \Delta v + \Delta u = 0 \quad (3.4)$$

Applying these arguments to the circuit equation (1.3) and replacing $f(v)$ with $i(v) + \frac{d}{dt}q(v(t))$ results in

$$\frac{\partial}{\partial v} i(v_{\text{DC}}) \Delta v(t) + \frac{\partial}{\partial v} \frac{d}{dt} q(v_{\text{DC}}) \Delta v(t) + \Delta u = 0 \quad (3.5)$$

Let

$$G = \frac{\partial}{\partial v} i(v_{\text{DC}}) \quad (3.6)$$

be the small-signal conductance at the operating point, and

$$C = \frac{\partial}{\partial v} q(v_{\text{DC}}) \quad (3.7)$$

be the small-signal capacitance. Then (3.5) becomes

$$G\Delta v(t) + C\frac{d}{dt}\Delta v(t) + \Delta u = 0 \quad (3.8)$$

In phasor analysis one computes the small-signal sinusoidal steady-state solution by assuming that all signals take the form of complex exponentials. Let $\Delta v(t) = Ve^{j\omega t}$ and $\Delta u(t) = Ue^{j\omega t}$ where V and U are complex, then

$$GVe^{j\omega t} + C\frac{d}{dt}(Ve^{j\omega t}) + Ue^{j\omega t} = 0 \quad (3.9)$$

$$GVe^{j\omega t} + j\omega C(Ve^{j\omega t}) + Ue^{j\omega t} = 0 \quad (3.10)$$

$$GV + j\omega CV + U = 0 \quad (3.11)$$

$$(G + j\omega C)V = -U \quad (3.12)$$

Equation (3.12) is a complex linear system of equations that is solved for V . V is a complex number (or vector) that gives both the magnitude and the phase of the solution.

3.3 AC Analyses Practice

In this section, the practical details of the various AC analyses are presented.

3.3.1 AC Analysis

With AC analysis, one specifies the stimulus for the circuit and computes the sinusoidal steady-state response. The stimulus may consist of signals on several sources, but they must all be sinusoidal and have the same frequency. For example, simulating a differential amplifier often involves applying two sinusoidal sources to the inputs, one shifted 180° from the other. It is not possible to simulate a mixer because for mixers the two input signals must be at different frequencies.

Prior to performing the AC analysis, the circuit is linearized. Thus, the absolute magnitude of the stimulus loses some of its significance.

Since the circuit is linearized, large signal effects such as distortion or clipping are not modeled. Furthermore, since the circuit is linearized about a constant operating point, effects from the time-varying nature of a circuit are not modeled, such as energy being converted from one frequency to another (as in mixers). For this reason, most AC analyses are performed with the stimulus magnitude set to unity. Doing so is a convenience that results in the simulator directly computing the transfer function rather than computing the actual signal level. For example, consider the simulation of an open-loop opamp with a low frequency gain of 100 kV/V using AC analysis. By specifying the stimulus magnitude to be one and stimulus phase to be zero, the simulator directly computes the gain of the opamp.

3.3.2 XF Analysis

Spectre provides an AC-like analysis that is referred to as the transfer function or XF analysis. Like AC analysis, it is a small-signal analysis (in other words, it operates on the linearized circuit equations) that is based on phasors. Unlike AC analysis, which allows you to simultaneously compute the transfer function from a single stimulus to every node in the circuit, XF analysis simultaneously computes individual transfer functions from every independent source to a single output. Consider the amplifier shown in Figure 3.1 on the following page. Typically, a designer might be interested in transfer functions between several sources in the circuit to the output. For example, the gain is the transfer function from V_{in} to $V(out)$. The positive power supply rejection is the inverse of the transfer function from V_{CC} to $V(out)$. Similarly the negative power supply rejection is the inverse of the transfer function from V_{ee} to $V(out)$. Finally, the output impedance is the transfer function from I_{out} to $V(out)$. The XF analysis computes all of these transfer functions at once.

By default Spectre computes the transfer function from every independent source in the circuit to the chosen output. However, it is also possible to have Spectre compute all possible transfer functions. This is useful when it is not known in advance which transfer functions are interesting. Transfer functions for nodes are computed assuming that a unit magnitude current source is connected from

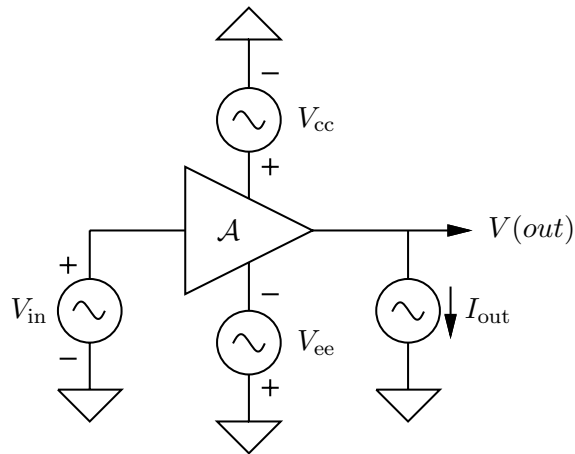


Figure 3.1: XF analysis simultaneously computes the transfer function from all sources in the circuit to the output. In this circuit, it allows the gain, the power supply rejection from both supplies, and the output impedance to be computed with one analysis.

the node to ground. Transfer functions for terminals are computed assuming that a unit magnitude voltage source is connected in series with the terminal.

3.3.3 SP Analysis

S -parameters are used by high-frequency circuit designers to characterize linear or near-linear circuits and components. Spectre's `sp` analysis directly computes S -parameters in the frequency domain and its `tdr` analysis directly computes them in the time domain. Both analyses are small-signal analyses based on phasor analysis, and so both start by linearizing the circuit about the DC operating point. To measure S -parameters the circuit must be configured as an N -port. Spectre provides a `port` component that is used to define the ports of the N -port. Every port component defines one

port of the network. A port component acts like a voltage source in series with a resistor. The value of the resistor is referred to as the reference resistance of the port. The default value is $50\ \Omega$, though it can be set to any value. When the S -parameters are computed as a function of frequency, they may be saved for plotting and they may be written to an ASCII file that is readable by Spectre's `nport` component. In this way, the characteristics of a linear component or circuit are encapsulated in a file by the `sp` analysis, and can be read into a different larger circuit by the `nport` component and analyzed using any analysis supported by Spectre, including transient analysis.

It is possible to compute S -parameters as a function of frequency with SPICE, though it takes some work. Start by modeling each port by a resistor in series with a voltage source. The value of the resistor is set to the reference resistance of the port, and the AC magnitude of the voltage sources is initially set to zero on all ports. Assume for simplicity that the S -parameters for a two port are being computed. To compute s_{11} and s_{21} , set the AC magnitude of the voltage source on the first port to 2 V and run an AC analysis over the frequency range of interest. Then, s_{11} is equal to the voltage across port one minus 1, and s_{21} is equal to the voltage across port two. To compute s_{12} and s_{22} , return the AC magnitude on the first port to zero and set the AC magnitude of the voltage source on the second port to 2 V. Again, run AC analysis. s_{22} is equal to the voltage across port two minus 1, and s_{12} is equal to the voltage across port one. This procedure is diagramed in Figure 3.2 on the next page. For clarity, the disabled voltage sources have been removed from the figure. To avoid the subtraction by one when computing s_{11} and s_{22} , install 1 V voltage sources in the circuit to automatically perform the subtraction.

3.3.4 Noise Analysis

Noise analysis predicts the noise performance of a linearized circuit in the frequency domain. The noise is caused by stochastic fluctuations in certain types of components. Examples of noise sources in components include thermal noise in resistors (also referred to as

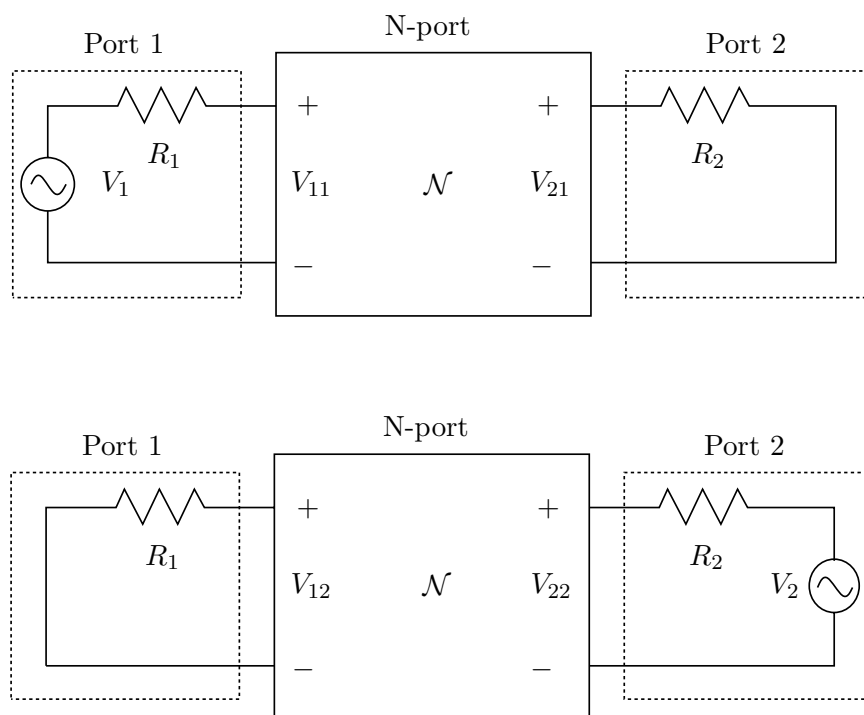


Figure 3.2: Illustration of how S -parameters are computed with SPICE. The top circuit computes $s_{11} = V_{11} - 1$ and $s_{21} = V_{21}$, where $V_1 = 2$ V. The bottom circuit computes $s_{22} = V_{22} - 1$ and $s_{12} = V_{12}$, where $V_2 = 2$ V.
