comprehensive diagnostic suite, it can be packaged and shipped to a customer. Similarly, systems may come back from the field or from development work in the laboratory with damage from excessive handling, and diagnostics can be run to help isolate and fix the problems.

19.8 SCHEMATIC CAPTURE AND SPICE

Various types of computer aided design (CAD) software assist digital engineers in designing and implementing circuits and systems. Some types of CAD tools have already been discussed, including HDL simulators, PLD fitters, and electromagnetic field solvers. Perhaps the most universal CAD tool for electrical engineers of all disciplines is a schematic capture program. Schematic capture is to circuits what word processing is to text. At the most basic level, schematic capture programs allow the drawing and manipulation of graphical symbols. However, their utility extends to converting the information represented by the graphical connections into varying types of data formats used in subsequent stages of system development. Two of the most common and useful data conversion results are a *bill of materials* (BOM) and a *netlist*. A BOM is a complete summary of all components used in the schematic and generally has identical components grouped together so that one can quickly determine how many units of each component are used in a design.

A netlist is an exhaustive list of all the electrical connections in the schematic and is the means of transferring a schematic into a PCB layout program. With a schematic in hand, a circuit can be prototyped with either a manual wiring process or a tedious manual PCB design process. Both of these methods involve a person translating each drawn wire in the schematic into a wire on a circuit board. The assembly complexity and potential for errors increase as the design itself gets larger. The advantage of an automatically generated netlist is that errors are minimized as the entire design database is transferred from the schematic capture tool to the PCB layout tool. Each layout tool has its own proprietary data interchange format, so it is important for a schematic capture program to support the desired data format.

A wide variety of schematic capture programs are available, and they range in price from hundreds to many thousands of dollars, depending on their intended application. Major vendors include Cadence, OrCAD (owned by Cadence), Innoveda, and Mentor Graphics. At least one program, OrCAD, is available in a free student/demonstration edition that has limited features and can be used to draw circuits of moderate complexity.

Another CAD program that is applicable to many disciplines is the *Spice* family of analog simulators. Spice (or SPICE), an acronym for *simulation program with integrated circuit emphasis,* was originally developed at the University of California at Berkeley in the 1970s and has become a standard means of simulating circuit behavior. Many variants of Spice are available, and source code is available as well. Spice tools share a common basic syntax and feature set. Circuits have traditionally been represented manually in a netlist-like format for Spice processing. However, some vendors now enable schematic capture software to convert a drawn circuit into the Spice input format. PSpice is a well known variant that has been available for PC platforms for many years. It is now sold by Cadence, which has continued the practice of offering free student and demonstration versions. PSpice was used to evaluate some of the analog circuits and concepts discussed in this book.

Spice is a powerful tool. Two of its basic modes of operation are AC sweep and transient analysis. AC sweep performs frequency-domain analysis on a circuit and is used to characterize filters and the frequency response of general circuits. Transient analysis provides a time-domain view of an analog circuit and can be used to simulate a transmission line or view a filter's output in the time domain as it would be seen on an oscilloscope. Spice simulations are an excellent means of performing "what-if" evaluations of circuits while still in the design phase. Transmission line terminations can be evaluated to gauge signal integrity, and filters can be tested to determine the frequencies over which they are effective. The degree to which a Spice simulation matches reality depends on how closely the real conditions are modeled. Performing highly accurate simulations is a skill that requires a thorough understanding of circuit theory. However, useful first-order approximations of analog behavior can be readily achieved. A key source of divergence between simulation and reality in a digital design is the parasitic properties of wires and components that become significant at high frequencies. An idealized resistor or wire might require the explicit addition of parasitic inductance and capacitance to get a more accurate simulation.

A basic example of Spice simulation can be shown using the first-order RC filter in Fig. 19.11. This lowpass filter uses idealized components and has $f_C \approx 10$ MHz with a steady attenuation slope of 20 dB per decade.

Circuits are presented to Spice by uniquely naming or numbering each node and then instantiating circuit elements that reference those node names. Figure 19.12 shows the Spice circuit description for the idealized RC filter. Ground is represented as 0. Resistors and capacitors are designated with identifiers beginning with R and C, respectively. V denotes a voltage source, and this voltage source is specified with a 0-V DC component and a 1-V AC component. The `.AC` command instructs Spice to perform an AC sweep over a logarithmic (decade) range from 100 kHz to 1 GHz with 10 data points per decade. Note that the voltage source does not specify a frequency. This is because our simulation is an AC sweep that evaluates a range of frequencies. Finally, `.PROBE` instructs Spice to display the results graphically.

The expected filter transfer function in Fig. 19.13 is obtained following a brief simulation of the circuit description input.

Improving the simulation's realism can be achieved by introducing parasitic inductance in series with the ideal capacitor. The actual inductance varies with package type and the wiring scheme used.

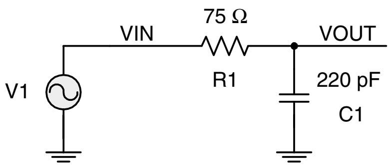

FIGURE 19.11 Idealized RC filter for Spice analysis.

```
V1          VIN 0 DC 0Vdc AC 1Vac
R1          VIN VOUT  75
C1          0 VOUT  220p
.AC DEC 10 1e5 1e9
.PROBE
.END
```

FIGURE 19.12 Spice circuit description for idealized RC filter.

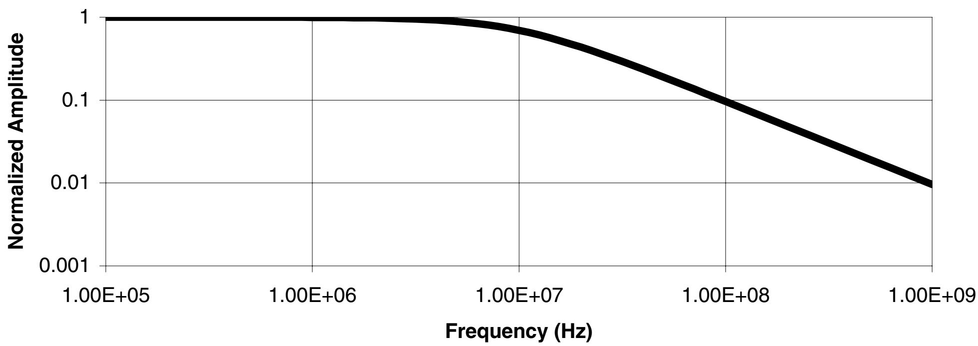

FIGURE 19.13 Bode magnitude plot for idealized RC filter.