

Using Simulation Tools to Verify Laboratory Measurements*

JAY R. PORTER and SANJAY TUMATI

Department of Engineering Technology and Industrial Distribution, 106 Fermier, 3367 TAMU, Texas A&M University, College Station, TX 77843-3367. E-mail: porter@entc.tamu.edu

The Electronics Engineering Technology (EET) program at Texas A&M University is currently working with industry to incorporate both digital and analog testing techniques into the curriculum. One area that has been identified as important by industry is helping future engineers understand the concept of integrating simulation into the design verification and testing process. To this end, the EET faculty is working to develop tools that allow the measurement process to be automated and compared directly to simulated results. For example, the faculty has already developed a LabVIEWTM tool that imports simulation data calculated using Altera's MaxPlusII and automatically creates a comparison between simulated and measured results on digital logic circuits. This paper presents a similar tool that was created recently for analog measurements allowing the comparison of Cadence's PSpice simulations to benchtop measurement data.

INTRODUCTION

THE TYPICAL ELECTRONICS engineering technology program emphasizes circuit analysis and design in two major areas: analog electronics and digital electronics. In courses emphasizing digital electronics, students are taught analysis and design skills and are asked to apply them to problems from simple combinational logic circuits to more complex problems such as the design of a simple microprocessor. In the analog area, students are taught to work with passive components, semiconductor devices, and more complex analog integrated circuits. They are then asked to use these components to design circuits such as filters, amplifiers, etc. Because of the applied nature of engineering technology, these courses generally have a lab component where concepts taught in the classroom are reinforced with hands-on experience. Circuits designed on paper are constructed, tested and evaluated.

At a minimum, students verify that the tools and theories they are learning actually work. However, the laboratory setting offers the ability to teach much more. Students should be exposed to the difference between the ideal and the real world and allowed to try 'what-if' scenarios on the bench. More abstract concepts such as signal timing and device loading can be explored in digital courses, while the effects of real components with tolerances can be studied on the analog side. Unfortunately, lab time is limited and the measurements required to investigate these effects can be very time-consuming. Thus, students are very limited in how much they can accomplish.

Today's circuit simulators have come a long way

and simulation offers a possible solution to this problem. By allowing graphical schematic capture and rapid calculation of circuit responses, students can easily prototype circuits inside and outside of class time. They can then experiment with different designs and watch the effects of changing components [1, 2]. For these reasons, simulation tools have become commonplace in electronics engineering technology. However, while simulations have a place in the educational environment, they should not be considered a complete replacement for hands-on laboratory experiments [3]. Instead they should be used to augment the laboratory experience.

Currently, students often use simulation tools in the lab to plot simulated and measured results together in a lab report and make a qualitative comparison. Fortunately, while these tools are often decoupled from the laboratory measurement process, there have been previous efforts to integrate them more fully into the laboratory experience [4].

The Electronics Engineering Technology (EET) program at Texas A&M University is currently working with industry to incorporate both digital and analog testing techniques into the curriculum. One area that has been identified as important by industry is helping future engineers understand the concept of integrating simulation into the design verification and testing process. This has led to ongoing efforts by the EET faculty to bring simulation into the laboratory in both the analog and digital electronics course sequences. One approach currently used requires the students to manually use simulation tools while testing their circuits in the laboratory to debug and verify design requirements. A better method is to integrate simulation results into the test equipment through the use of a virtual instrumentation development environment such as

* Accepted 14 July 2004.

LabVIEW™. For example, consider the case of a frequency response test of a filter. A measurement tool that could import circuit simulations, read the simulated test parameters, and then perform the exact same test on the bench would allow students to make direct comparisons of their simulation data to their measured data. These comparisons could then lead to a feedback loop where students improve their circuit performance by correcting for implementation errors and real-world effects.

As part of this effort, the EET faculty has already developed an integrated simulation/measurement tool for the digital course sequence [5]. In these courses, Altera PLD devices are used to implement digital circuits such as simple microprocessors designed by the students. Because of the complexity of these circuits, repeated simulation and testing is critical. To help automate this process, the MaxTester tool was created. MaxTester creates an interface between the simulation tools in Altera's MaxPlusII and the measurement process, allowing the student to physically apply the test vectors used in the simulation to the circuit under test. The measured output is then plotted against the simulated output to allow comparisons to be made. The use of MaxTester has improved the efficiency of the design cycle, giving students time to create better circuit designs.

This paper discusses a new set of tools that allows direct correlation between circuit simulations using Cadence's ORCAD PSpice and actual measured circuit responses. Using frequency response testing as an example; the methodology,

system implementation and results are presented. Finally, the benefits of these tools are discussed in the context of an undergraduate electronics laboratory.

METHODOLOGY

Overview

There are a variety of simulators available for teaching analog and digital design. Companies including Cadence and Protel provide analog (and mixed-signal) simulators that allow students to graphically enter schematics and then perform time and frequency domain analyses. The analog course sequence currently performs simulations with Cadence's ORCAD software. Fortunately, most simulator tools are comparable in form and function, so it would not be difficult to extend the processes discussed here to other tools. In fact, National Instruments now supports access to data from several popular simulation programs [6].

The basic concept is presented in the block diagram in Fig. 1. Using a virtual instrumentation tool such as National Instruments' LabVIEW, an integrated measurement and simulation tool can be developed. By using a personal computer at the laboratory bench, students can dynamically model and simulate circuits during lab time. Once the simulation is complete, a virtual instrumentation tool can be used to access the simulation data. The tool can use this data to determine simulation parameters. This includes information such as

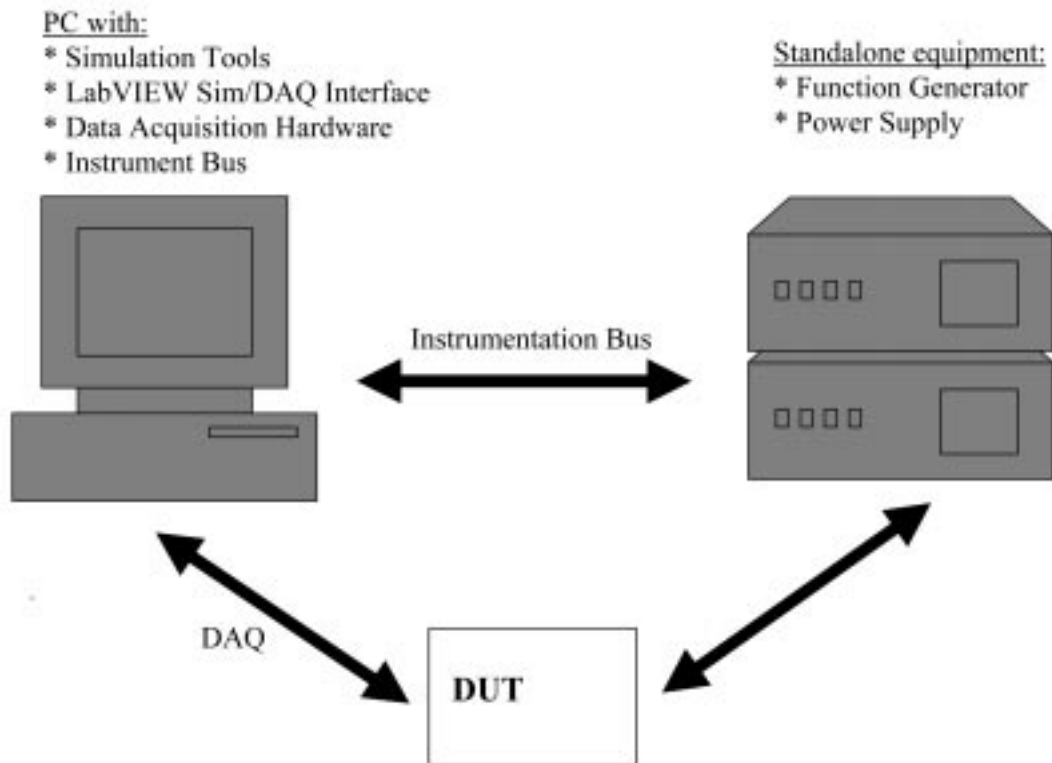


Fig. 1. Block diagram of the integrated simulation/measurement tool.

total simulation time and time step for a transient analysis, frequency range and step size for an AC analysis, functional data for a digital test, and information needed to correlate the data contained in the simulation to actual test points on the circuit. Once the simulation parameters are known, the tool repeats the simulated measurement on the actual circuit using PC-based data acquisition cards and/or GPIB (General Purpose Instrumentation Bus) controlled instrumentation to automate the procedure. Finally, the simulated and measured data are presented graphically and allow the students to perform real-time comparisons.

System hardware

A picture of the hardware setup can be seen in Fig. 2. A 1 GHz Pentium-based personal computer was used to host the measurement tool. ORCAD PSpice 9.1 (Cadence; San Jose, CA) served as the simulation environment and LabVIEW 6.1 (National Instruments; Austin, TX) was used to develop the virtual simulation/measurement tools. The PC contained a PCI-6024 (National Instruments; Austin, TX) data acquisition card. The card's 12-bit, 200kS/s digitizer was used to measure both the input and output signals from the device under test. While this card limits testing to signals below approximately 20 kHz, faster cards are available.

The PC also contained a PCI-GPIB (National Instruments; Austin, TX) to allow communications with GPIB test and measurement equipment on the bench. An Agilent 33120A 15MHz signal generator (Agilent; Palo Alto, CA) served as the source for the input signal and was controlled through GPIB. An Agilent E3631 power supply (Agilent; Palo Alto, CA) was used as the power

supply and was also GPIB controlled. Finally, the circuit to be tested was constructed on a typical breadboard and connected to the measurement equipment through standard test leads.

LabVIEW/PSpice interface software

To create an instrument that integrates simulation results into the measurement process, a software interface between LabVIEW and ORCAD PSpice had to be designed first. Fortunately, as with most Spice-based simulation packages, ORCAD supports the output of a text file, or a .CSD file that contains the simulated voltage and current data at each node. In ORCAD PSpice 9.1, this is enabled through the PSpice » Edit Simulation Profile menu. One must check the box labeled 'Save the data in CSDF format' under the Data Collection tab. This will then cause PSpice to generate an ASCII file similar to the one in Fig. 3, upon completion of the simulation. Looking at the .CSD file, one can see that it is made up of three components. First is a header section (#H) that contains information such as simulation type, number of circuit nodes, output data format, etc. Second is a name section (#N) that contains the naming convention for the voltages and currents at each of the nodes in the circuit. Finally, there is a data section (#C) that has the data for each of the nodes at every simulation point.

To read in the .CSD file data, a generic LabVIEW Virtual Instrument VI was developed. This VI was designed modularly so that it could be used in any LabVIEW-based instrument that required PSpice simulation data. The front panel of the VI can be seen in Fig. 4. First, the user selects a .CSD simulation output file that corresponds to the measurement they want to make.



Fig. 2. Hardware setup of the integrated simulation/measurement tool used for frequency response testing.

```

bandpass-SCHEMATIC1-bandpass1.csd - Notepad
File Edit Format View Help
#H
SOURCE='PSPICE' VERSION='Pspice 9.1 (Mar 1999)'
TITLE='** circuit file for profile: bandpass'
SUBTITLE='
TIME='15:59:12' DATE='01/12/02' TEMPERATURE='2.700E01'
ANALYSIS='AC Sweep' SERIALNO='1108061965'
ALLVALUES='YES' COMPLEXVALUES='YES' NODES='64'
SWEEPVAR='Frequency' SWEEPMODE='LOG'
XBEGIN='1.000E01' XEND='1.000E04'
FORMAT='0 R_VOLTS0rAMPS;EFLOAT / I_VOLTS0rAMPS;EFLOAT : NODE0rBRANCH;NODE
DGTLDATA='NO'
#N
'V(R5:2)'; 'v(R1:1)'; 'V(C2:2)'; 'V(N00045)'; 'v2(C2)'; 'v1(R1)'; 'V2(R5)'; 'v(R2:1)'; 'v(C1:1)'; 'v(R1:2)
'; 'v(R6:1)'; 'v(U1:+)'; 'V(N00048)'; 'v1(C1)'; 'vB(X_U1.X_lm741.q2)'; 'v2(R1)'; 'v1(R2)'; 'v1(R6)
'; 'v(R2:2)'; 'v(C1:2)'; 'V(R6:2)'; 'v(U1:OUT)'; 'V(N00119)'; 'v2(C1)'; 'v1(X_U1.X_lm741.dc)';
'v2(X_U1.X_lm741.de)'; 'v2(R2)'; 'v2(R6)'; 'v2(X_U1.X_lm741.ro1)'; 'v(R5:1)'; 'v(v3:+)'; 'v(C2:1)';
'V(N00406)'; 'v1(C2)'; 'v1(R5)'; 'v1(v3)'; 'v(v1:-)'; 'v(U1:v-)'; 'V(N00091)'; 'v1(X_U1.X_lm741.dp)';
'v2(X_U1.X_lm741.fee)'; 'v2(X_U1.X_lm741.rp)'; 'v2(v1)'; 'v2(X_U1.X_lm741.ve)'; 'v(v2:+)';
'v(U1:v+)'; 'V(N00082)'; 'v2(X_U1.X_lm741.dp)'; 'v1(X_U1.X_lm741.rc1)'; 'v1(X_U1.X_lm741.rc2)';
'v1(X_U1.X_lm741.rp)'; 'v1(v2)'; 'v1(X_U1.X_lm741.vc)'; 'v(X_U1.X_lm741.11)'; 'v1(X_U1.X_lm741.c1)';
'v(X_U1.X_lm741.q1)'; 'v2(X_U1.X_lm741.rc1)'; 'v(X_U1.X_lm741.12)'; 'v2(X_U1.X_lm741.c1)';
'vc(X_U1.X_lm741.q2)'; 'v2(X_U1.X_lm741.rc2)'; 'v(X_U1.X_lm741.6)'; 'v1(X_U1.X_lm741.c2)';
.
.
...
#C 1.0000000000E01 64
2.583E-02/5.427E-02:1 3.827E-06/-5.974E-06:2 -2.831E-01/-5.158E-01:3 1.000E00/0.000E00:4 0.000E00/0.
000E00:5 0.000E00/0.000E00:6 1.910E-06/-2.981E-06:7 -1.910E-06/2.982E-06:8 3.046E-05/5.655E-05:9 -2.
963E-01/-5.435E-01:A 0.000E00/0.000E00:B 0.000E00/0.000E00:C -2.635E-01/-5.539E-01:D 0.000E00/0.000E
00:E 0.000E00/0.000E00:F 0.000E00/0.000E00:10 1.913E-06/-2.986E-06:11
1.245E-06/-1.943E-06:12 2.581E-06/-4.029E-06:13 0.000E00/0.000E00:14 -2.963E-01/-5.435E-01:15 -3.111
E-05/1.708E-05:16 3.158E-05/5.668E-04:17 3.245E-15/2.079E-15:18 -1.025E-09/5.586E-10:19 8.340E-11/-4
.645E-11:1A -8.340E-11/4.645E-11:1B -2.635E-13/-5.539E-13:1C 2.635E-13/5.539E-13:1D 0.000E00/0.000E
00:1E 0.000E00/0.000E00:1F 2.635E-04/5.539E-04:20 2.895E-04/5.274E-04:21 2.319E-04/-1.292E-05:22 5.
148E-06/9.378E-06:23 3.046E-10/5.655E-10:24 -3.600E-10/5.619E-10:25
3.601E-10/-5.622E-10:26 -3.638E-10/5.679E-10:27 3.639E-10/-5.682E-10:28 1.450E-13/-2.264E-13:29
.
.
...

```

Fig. 3. Example of the .CSD file output from Orcad PSpice 9.1



Fig. 4. The LabVIEW simulation import interface.

Once this selection is made, the VI automatically determines which voltages and currents have been calculated and are available. It then populates the drop down lists so that the user can select the input stimulus and the output signal. The action of selecting these variables also gives the student a chance to ensure that they have their test equipment properly connected to the circuit under test. Once the selections are made, the VI will parse the .CSD file to extract the following:

- The number of measurements the virtual instrument will have to make.
- An array of the test points where measurements will be made. In the case of the frequency response tester, this is an array of the frequencies at which the response was calculated.
- An array of the input value at each of the test points. For frequency response, this is the magnitude and phase of the input signal at each test point.
- An array of the output value at each of the test points. For frequency response, this is the magnitude and phase of the output signal at each test point.

This data can then be passed to the measurement VI so that the necessary measurements can be performed and the data compared to simulation.

EXAMPLE OF A FREQUENCY RESPONSE TESTER

Similar to most electronics engineering technology programs, the EET program at Texas A&M has an analog course sequence that begins with courses on DC and AC circuit analysis and culminates in a course on semiconductor electronics. The course sequence is laboratory intensive, seeking to give students as much hands-on experience as possible. Simulation is routinely used in the laboratory to aid in design verification and debugging. For example, a topic stressed in the AC circuit course and the semiconductor electronics course is that of frequency response, especially in the context of filters. In a traditional lab experiment, students may simulate a filter circuit's frequency response and then build and test the circuit on the bench. Unfortunately, typical frequency response testing requires that the student measure the magnitude and phase of the input and output signals at many different frequencies. Due to limited laboratory time, they will then usually wait until after the lab is over to plot the data and compare it to simulation.

For this reason, a new virtual instrumentation tool that automates frequency response measurements and integrates simulation has been created. The tool works by first importing PSpice simulation data as discussed previously. Using the list of frequencies from the PSpice interface VI, the Agilent function generator then sweeps the input signal. The data acquisition card digitizes the

circuit's input and output signals at each frequency. In this manner, performing a frequency response test at 60 separate frequencies takes less than a minute. Once the measurement is complete, the data is processed using Fourier Transform analysis and the amplitude and phase responses of the circuit are calculated. The graphical user interface allows the student to quantitatively compare and analyze both the simulation and measured data. Active cursors allow the student to make frequency and voltage measurements. The interface also allows students to view the output using either linear or logarithmic scales.

Figure 5 demonstrates an example of an active bandpass filter that students can test using the integrated virtual instrumentation tool. The bandpass filter circuit diagram is shown in Fig. 5a. One can note the ideal component values that were calculated as part of the filter design process. These same ideal components values were assumed in the simulation. Using the LabVIEW frequency response VI, the students can measure the performance of their actual circuit and compare it to the simulation, as in Fig. 5b. They will see that both the maximum gain and the center frequency of the simulated filter versus the actual filter are different. With this in mind, they can be asked to reconcile these differences. Through interactive discussions and testing of their circuits, they should arrive at the conclusion that most of the discrepancies are related to the difference between ideal versus 'real-world' components. Rerunning their simulations using actual component values measured on an impedance meter will show much better agreement between their simulated and measured results, as in Fig. 5c. An interesting lesson learned with this particular circuit was related to the capacitors used in the circuit. Because one of them was electrolytic (due to parts availability), good agreement could not be reached until both the capacitance and leakage resistance were modeled.

Figure 6 shows a second example of results from the tester. This time, a three-pole, lowpass, Chebyshev active filter was implemented and tested. In this case, even reasonable agreement required using actual component values in the simulation. Fig. 6b shows the amplitude and phase comparisons produced by the integrated verification tool. One can see very good agreement on both plots; however, the phase plot shows some noise at high frequencies (> 6 kHz). This is due to two reasons: the limited sampling rate of the PCI-6024 data acquisition card and the multiplexed sampling of the input and output signal. At high frequencies, the phase errors become noticeable. Solutions to this problem include using a higher speed digitizer or using a GPIB oscilloscope to capture the data.

DISCUSSION

Over the past two semesters, the integrated simulation/measurement VI tool was used in the

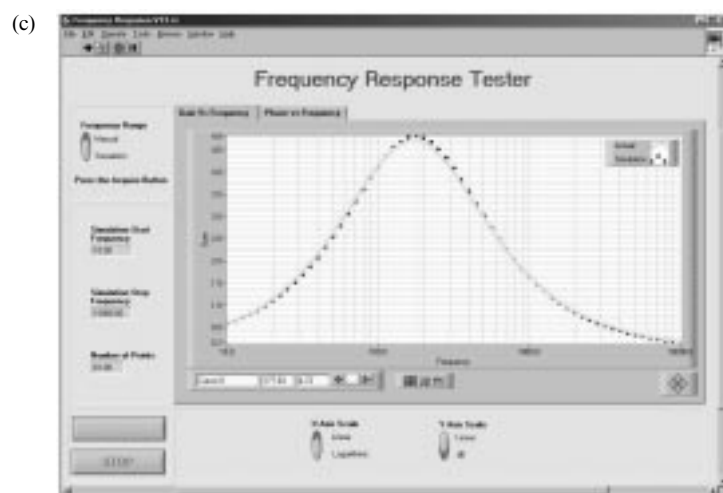
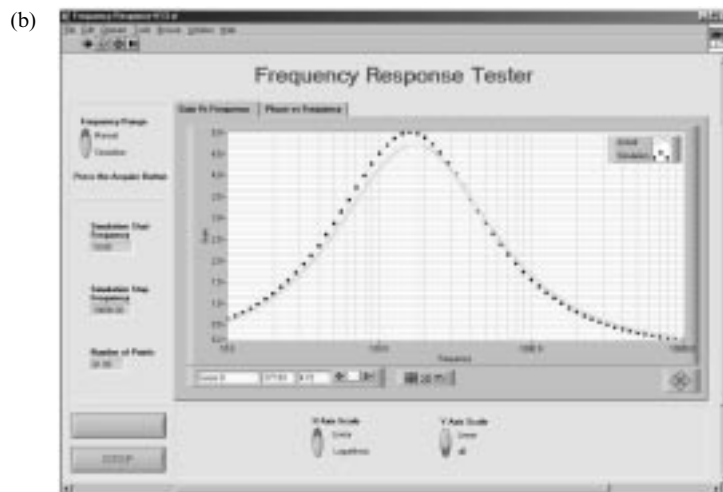
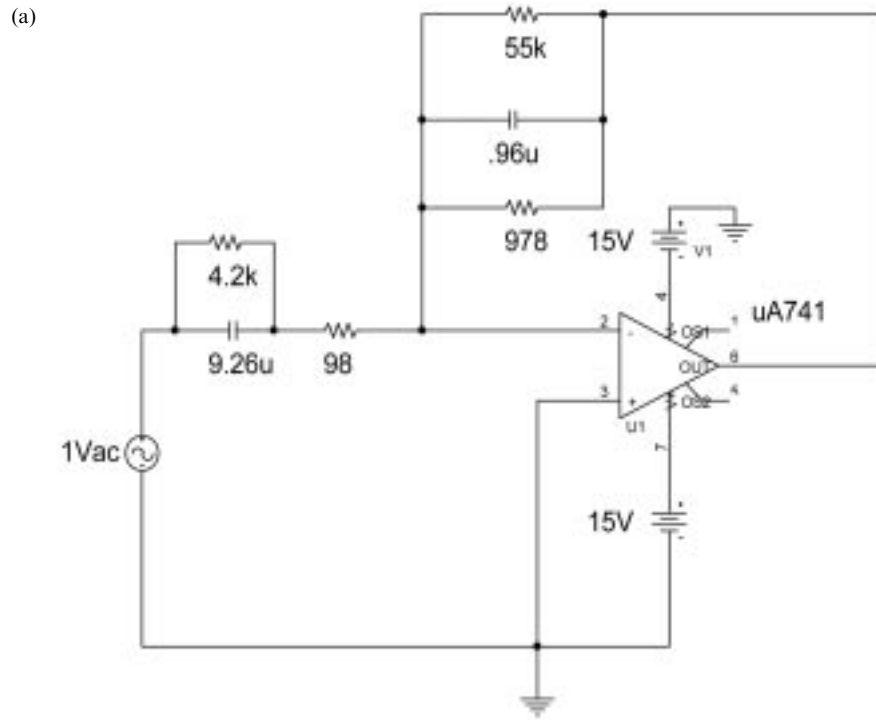


Fig. 5. a) Circuit diagram of the active bandpass filter. b) The simulated (dashed) and measured (solid) frequency response of an active bandpass filter. The simulation assumed ideal components. c) Same as before but the simulation used actual component values.

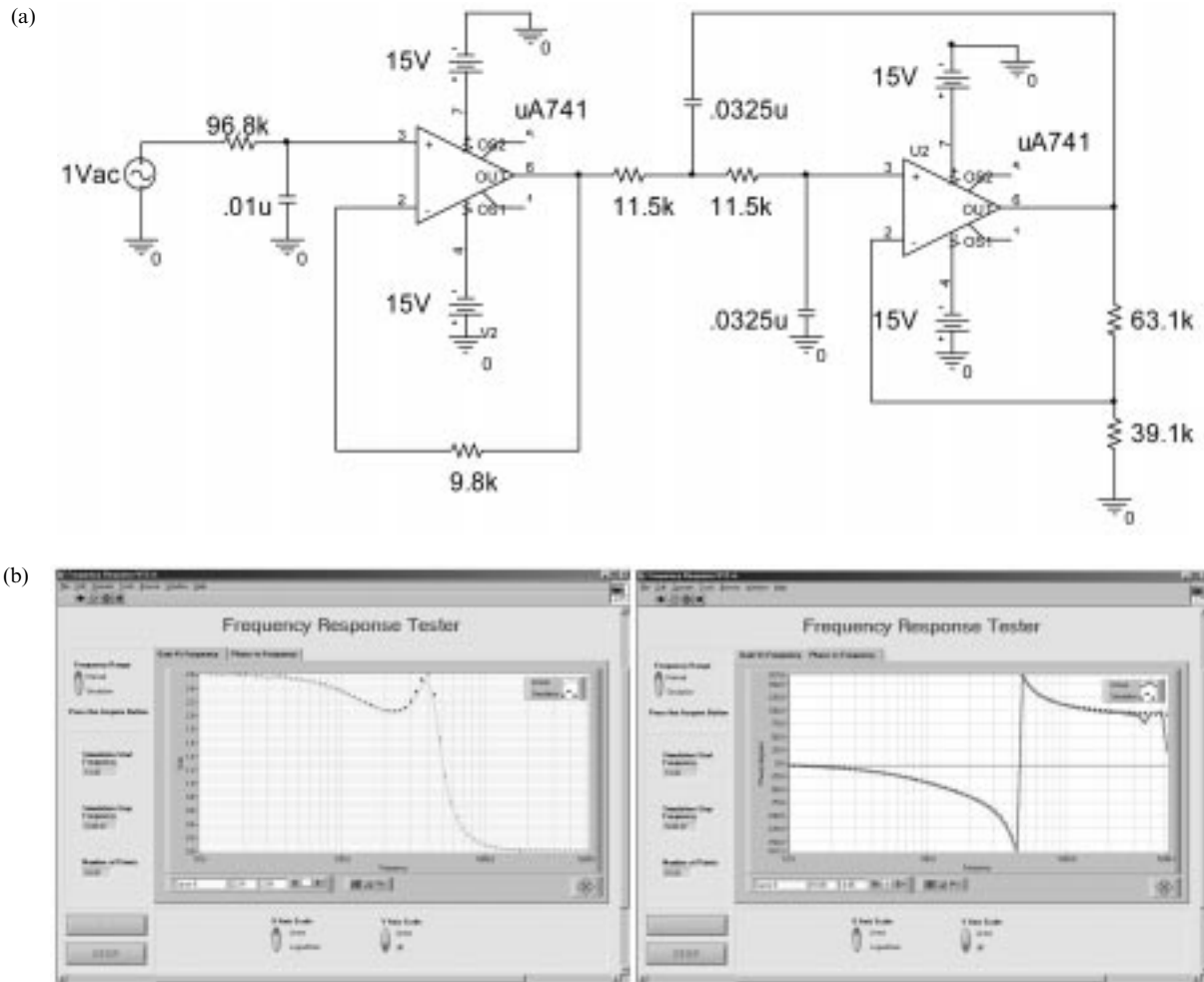


Fig. 6. a) Circuit diagram for a three-pole Chebyshev lowpass filter. b) Comparison of the amplitude and phase response of a three-pole Chebyshev active low-pass filter.

semiconductor electronics course. Prior to the use of this tool, students were expected to measure the frequency response of filters by manually changing the frequency on the signal generator, making complex signal measurements at each point, and then plotting the magnitude and phase of the frequency response. While this method gave the students a good understanding of frequency response, it was very time-consuming. This meant that the scope of the laboratory experiments was limited. Students simply plotted the frequency response of the assigned filters and did not have time to think about circuit non-idealities that required repetitive experimentation. One can begin to understand how this led to a very limited hands-on experience. Not only were there many potential sources for error, such as making amplitude and phase measurement errors, not taking enough data points, etc., but once the student had left the laboratory there was little chance of returning to repeat the experiment if problems occurred. There was even less chance that a student would have the time to truly investigate a circuit and experiment with component values, etc.

Once the new integrated simulation/measurement tool was introduced in the labs, the students were able to complete the same work in a quarter of the time. This time saving was exploited to let the students explore ‘what-if’ scenarios in the lab to gain a better understanding of real-world circuit design issues. It was thus possible for the instructor to assign more tasks for the students. Some of these tasks included:

- Understanding the limitations of PSpice modeling by comparing it with the LabVIEW-based measurements.
- Understanding the effects of non-idealities in real components.
- Learning to model the non-idealities in real components.
- Building and testing filters of higher orders.
- Changing individual component values to demonstrate the effect of each component on filter parameters (gain, cutoff corner, etc.) and circuit sensitivity.

From the instructor’s point of view, this was a substantial improvement, because it became

possible to cover more material in a short time. Some phenomena such as circuit non-idealities and simulator limitations are better appreciated in the laboratory than in the classroom. The new tool thus promoted a rapid accumulation of facts through simulation and measurement with only the occasional overhead of rewiring a new circuit. Understandably, the students' reactions varied from excitement at the working of the virtual instrumentation software to relief at not having to make the same types of frequency response measurements as they had in earlier courses.

It should be noted that the authors do not advocate replacing all testing with automated procedures. When the students manually test a few devices, they gain test techniques and debugging skills. However, once the student understands the procedure, rapid automation opens up a whole new area of learning, where students can iterate on the circuit design without the overhead of time-consuming testing.

CONCLUSION AND FUTURE WORK

This paper discusses the development and implementation of measurement tools that can be used to integrate design simulation and verification into a single process. The authors are using the LabVIEW virtual instrumentation programming environment to create measurement tools that seamlessly compare simulation of both analog and digital circuits to measured results. Successful examples have been implemented, including a digital stimulus and response tester for PLD circuits and a frequency response tester for analog devices. These tools will lead to more efficient use of lab time, allowing students to spend more time investigating concepts and less time debugging their measurement system. Future work includes the development of new analog instruments that will allow students to investigate concepts such as analog transients and harmonic distortion. These new integrated tools are also being incorporated into ongoing undergraduate research projects in mixed-signal and digital tests.

REFERENCES

1. T. Hall, Using simulation software for electronics engineering technology laboratory instruction, 2000 American Society of Engineering Education Annual Conference, 3547, St. Louis, MO (18–21 June 2000).
2. R. A. Shackelford, Learning about real components, IEE Colloquium on Computer Based Learning in Electronic Education, 12/1–12/3 (10 May 1995).
3. S. Pisarski, Impact of simulation software in the engineering technology curriculum, 1999 American Society of Engineering Education Annual Conference, 2548, Charlotte, NC (20–23 June 1999).
4. J. Ross, The use and misuse of circuit simulation in electronics courses, 1998 IEEE Frontiers in Education Conference, vol. 3, 1101 (4–7 November 1998).
5. J. Ochoa and M. Landrum, An approach to advanced digital design for undergraduate students, ASEE Society of Engineering Education Gulf Southwest Annual Conference (7–9 March 1999).
6. <http://www.ni.com/design/eda.htm>.

Jay R. Porter joined the Department of Engineering Technology and Industrial Distribution at Texas A&M University in 1998 and currently works in the areas of mixed-signal circuit testing and virtual instrumentation development. He received a B.Sc. degree in electrical engineering (1987), an M.Sc. in physics (1989), and a Ph.D. in electrical engineering (1993) from Texas A&M University.

Sanjay Tumati is a student in the Department of Electrical Engineering at Texas A&M and is pursuing his M.Sc. degree. He is currently teaching analog electronics laboratory classes for the Electronics Engineering Technology Program.