

Developing Digital Measurement and Analysis Laboratory in Circuits and Electronics Lab at TAMUK

Shuhui Li and Abrar A. Khan

**Department of Electrical Engineering & Computer Science
Texas A&M University – Kingsville
Kingsville, TX 78363**

ABSTRACT

The paper presents a laboratory development for measurement, analysis, design, and simulation of typical electric and electronic circuits using modern computer hardware and software. It shows computer measurement and data acquisition system using National Instruments (NI) LabView and NI-ELVIS, and gives methodologies for theoretical analysis and design of electric and electronic circuits using MathCad. It presents the development of a virtual laboratory using PSpice, and illustrates how Microsoft Excel spreadsheets and MatLab are used for data recording and/or result comparisons among measurements, theoretical calculations, and simulations. The paper shows the mechanisms and approaches developed for strong correlation between theories and hands-on experimentations using modern computer tools for the purpose of quality education of a laboratory course in electric and electronic circuits.

Index Terms – Education, laboratory, electric and electronic circuits, LabView, NI ELVIS, MathCad, PSpice simulation, Microsoft Excel spreadsheets, MatLab.

1. INTRODUCTION

Computers and microprocessors have changed a lot in the field of traditional Electrical Engineering (EE). They are also changing the traditional teaching styles in higher education. When digital computers as well as various software and hardware are used in laboratories and classrooms, they can provide much more effective and efficient ways in teaching and make many mathematical related engineering problems easy to understand.

The Circuits and Electronics Lab, a first semester junior EE laboratory course at Texas A&M University – Kingsville (TAMUK) draws on and correlates with the knowledge obtained by students from their sophomore and early junior level EE courses such as electric circuits and electronics. Traditionally, all of the lab experiments in this course were based on traditional equipments, devices, methods and techniques for measurements, data recording, and result analysis. This usually makes experiments time-consuming and inefficient, and therefore, greatly limits the effectiveness in the students' understanding of fundamental concepts and theories from

the hands-on experimentation. This paper gives methodologies for the significant changes and improvements of the course by developing a completely computer assisted measurement, analysis, and design environment that incorporates National Instruments (NI) LabView, NI ELVIS, MathCad, PSpice, Microsoft Excel, and MatLab.

The paper first presents the development of a complete computer measurement and data acquisition system using LabView and NI ELVIS in section II. Section III shows methodologies and approaches used to introduce students to the computer measuring system, and to basic LabView concepts and programming skills. Section IV illustrates how MathCad is used to let students more effectively understand theoretical, analytical and design aspects of various experimental circuits. Section V presents the virtual laboratory using PSpice for circuit simulation, analysis, and design. Section VI shows the methodologies for result comparisons among MathCad calculation, PSpice simulation, and LabView and NI ELVIS measurement by using Microsoft Excel and MatLab for the purpose of strong correlation between theories and hands-on experimentation. Section VII gives the conclusions.

2. COMPUTER MEASUREMENT AND DATA ACQUISITION USING NI LABVIEW AND NI ELVIS

LabVIEW is a graphical development environment with built-in functionality for simulation, data acquisition, instrument control, measurement analysis, and data presentation [1, 2]. The LabView graphical development environment does not require any lines of traditional text-based codes to create applications. Instead, a user interface for an application can be created quickly and simply by dragging and dropping pre-built objects and then system functionality can be specified by assembling block diagrams (a natural design notation for Electrical and Computer Engineering students).

Since the introduction of LabView in the Circuits and Electronics Lab at TAMUK, it has greatly improved the ways of teaching and learning. Rather than focusing on tedious and/or time consuming methods of gathering data, students can obtain measured results quickly, efficiently and effectively. Results can be obtained by capturing the screen and/or by saving to data files for further analysis. The saved results in a data file can be reloaded in a computer and then analyzed by different engineering and mathematical software. This not only makes it possible to have much more lab contents in an experiment but also makes students focus more on results, concepts, relations to theories, and/or design aspects of the experiments, providing a more effective and efficient methodology for quality education.

NI ELVIS, a suite of more professional virtual instruments developed by the National Instruments and introduced into the laboratory in 2003, has further improved the quality education of the lab course at TAMUK. The NI ELVIS uses LabVIEW-based software, a multifunction DAQ board, and a custom designed workstation (Figure 1) to provide functionality for a suite of instruments. This specially designed suite of LabVIEW-based virtual instruments provides functionality of common laboratory instruments, such as an oscilloscope, a function generator, a DMM, a variable power supply, a bode analyzer, a dynamic signal analyzer, an impedance analyzer, and current-voltage analyzers. All the functions of the NI ELVIS have been significantly used in the fourteen lab experiments of the course for measurement, analysis, and

design of typical electric and electronic circuits, filters, operational amplifiers, and D/A and A/D converters [3].

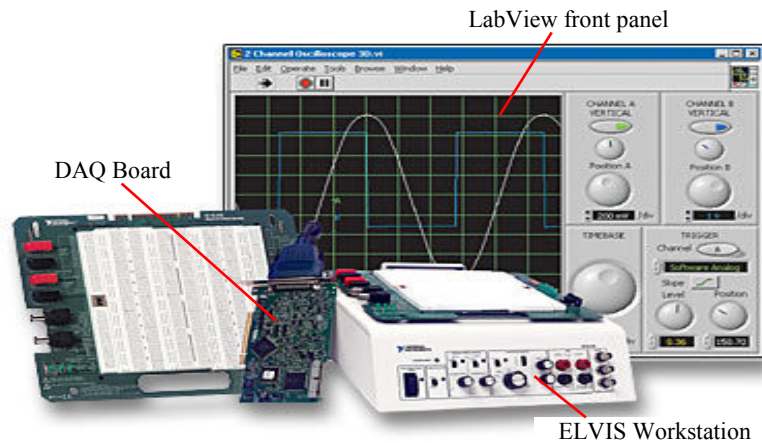


Figure 1. NI ELVIS (Educational Laboratory Virtual Instrumentation Suite)

Figure 2 presents an example of using LabView/Ni ELVIS for measurement of the transients in a series RC circuit under a square wave input voltage. After the measuring circuit is set up and is provided with necessary connections to NI DAQ analog input channels, measuring results can be displayed in LabView front panel by clicking the running icon. Any result in the front panel can be screen captured as shown in Figure 2 a) or saved to a file by clicking the Save button in Figure 2 a). This greatly reduces the time needed to get the results. On the other hand, large amounts of data saved in a file can be analyzed and compared with Mathcad calculation or PSpice simulation results (Figure 2 b) so as to most effectively re-emphasize students' understanding of theories.

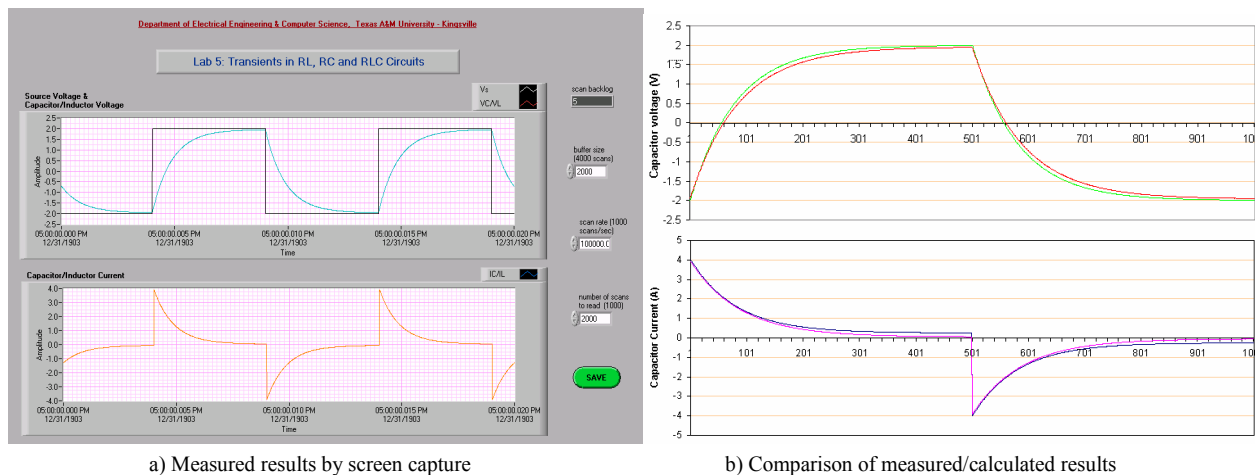


Figure 2. Measured and calculated results for transient of a RC circuit

3. UNDERSTANDING LABVIEW AND NI ELVIS

It is important for students to understand the significance of LabView and NI ELVIS virtual instruments as replacements of traditional instruments, and to understand the basic NI LabView and graphical programming concepts. This would develop students' interests and

confidence and give them a clearer understanding to use LabView and NI ELVIS for measurement, analysis, and design of various electric and electronic circuits as well as more advanced design projects.

3.1. Comparing LabView and NI ELVIS with Traditional Instruments

Result comparisons between traditional and virtual instruments usually give students a better understanding and/or stronger confidence on LabView and NI ELVIS virtual instruments for experimentation. Figure 3 shows a circuit of an experiment related to network analysis techniques such as node voltage, mesh current, superposition, and Thevein equivalent methods. It is one of the experiments for this comparison purpose. Specially designed Microsoft Excel tables have been used for data recording. Normally, the comparisons show very small differences among results from traditional instruments, the virtual instruments, and theoretical calculations as shown in Table 1. Those comparisons usually motivate students' interests to learn LabView and to use LabView/NI ELVIS for more extensive measurement, analysis, and design of diverse electric and electronic circuits.

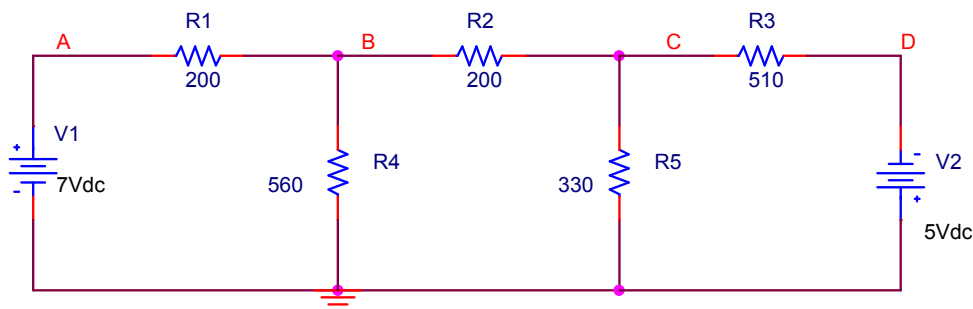


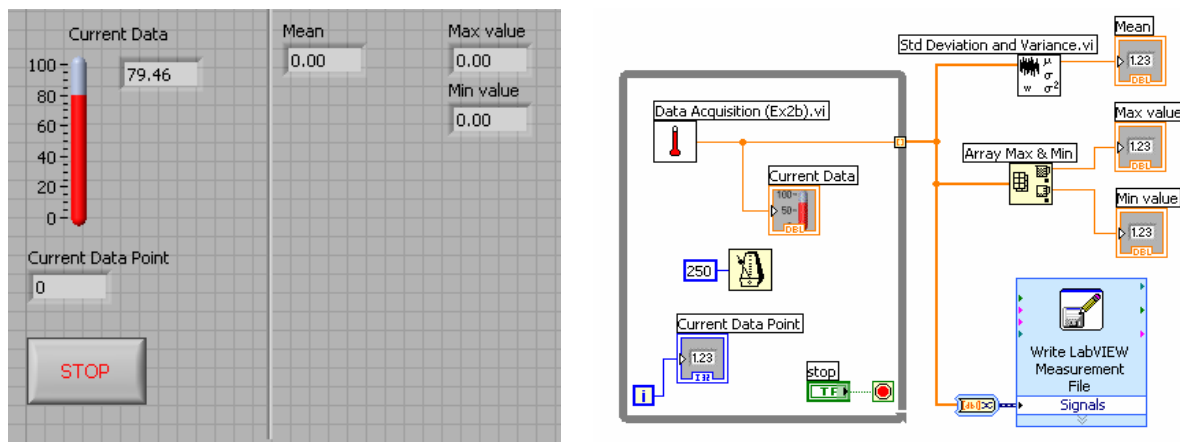
Figure 3. Circuit for experiment of network analysis techniques

Table 1: Comparison of measurement by NI ELVIS DMM and FLUKE 45 DMM

Measurement	Mesh Current			Node Voltage		
	Calculated	Measured		Calculated	Measured	
		DMM	NI ELVIS		DMM	NI ELVIS
I_{R1} (mA)	18.79	18.49	18.38	18.79	18.71	18.7
I_{R2} (mA)	13	12.85	12.4	13.005	13.0015	13.01
I_{R3} (mA)	11.06	10.92	10.42	11.060784	11.0935294	11.094118
I_{R4} (mA)	5.79	5.64	5.98	5.7892857	5.81785714	5.8214286
I_{R5} (mA)	1.94	1.93	1.98	1.9424242	1.9930303	1.9939394
V_A (V)	7	7	7	7	7	7
V_B (V)	3.2424	3.1584	3.3488	3.242	3.258	3.26
V_C (V)	0.6402	0.6369	0.6534	0.641	0.6577	0.658
V_D (V)	-5	-5	-5	-5	-5	-5

3.2. Introduction of LabView Basics and Graphical Programming

Teaching basic LabView concepts and programming skills is important to let students understand the fundamentals of LabView data acquisition and the graphical programming, and the significance to use LabView for measurement and automation. However, unlike other text-based programming languages, it is possible to teach the LabView fundamentals in one or two lab experiment hours. An environment that has facilities for students to exercise while the instructor is teaching is generally needed. A typical example for this introduction is to develop a simple temperature monitoring and recording system as shown in Figure 4 [4]. The temperature sensor can also be replaced by a voltage source for simplicity or convenience. This example can be subdivided into four or five exercises that quickly teach students step-by-step from basic LabView concepts and programming skills to the final implementation of the simple temperature data acquisition and recording system. All these exercises will be able to cover and give students opportunities to learn and practice LabView basics such as subVIs, loops, arrays, charts and graphs, DAQ and waveforms, and file I/O.



a) Measured results by screen capture

b) Comparison of measured/calculated results

Figure 4. Measured and calculated results for transient of a RC circuit

4. UNDERSTANDING THEORETICAL BACKGROUND OF AN EXPERIMENT USING MATHCAD

One of the important purposes of the experimentation is to re-emphasize students' understanding of theories. In order to relate the lab experiments more effectively and efficiently with theories, MathCad is used for theoretically related background, analysis, and/or design of most lab experiments. MathCad has been a very successful mathematical tool to be able to effectively show the relationship between mathematical equations and electrical variables represented by data, curves, and plots. Its powerful animation tools can provide a strong dynamic environment for students to see both theories/equations and what may happen in a circuit under the theories/equations [5].

Figure 5 shows the MathCad worksheet used for calculation of the mesh currents of the circuit given in Figure 3. In the MathCad worksheet, a student types equations based on learned theories [6] and then the MathCad will automatically do all the calculations. This not only makes it much faster to get the results but also makes the theoretical background of an experiment

clearer and easier to understand. For any condition change for the same circuit, MathCad quickly gives a new solution. For example, when changing $V1=0V$ or $V2=0V$ respectively in Figure 3, a student can quickly get theoretical calculated results for the superposition theorem method of the circuit [6].

Lab Experiment 2: Network Analysis Techniques

Resistors of the circuit:

$$R1 := 200\Omega \quad R2 := 200\Omega \quad R3 := 510\Omega \quad R4 := 560\Omega \quad R5 := 330\Omega$$

Voltage sources:

$$V1 := 7V \quad V2 := 5V$$

1. Mesh current method:

$$IR1 := 0A \quad IR2 := 0A \quad IR3 := 0A$$

Given

$$\begin{pmatrix} R1 + R4 & -R4 & 0 \\ -R4 & R2 + R5 + R4 & -R5 \\ 0 & -R5 & R3 + R5 \end{pmatrix} \cdot \begin{pmatrix} IR1 \\ IR2 \\ IR3 \end{pmatrix} = \begin{pmatrix} V1 \\ 0 \\ V2 \end{pmatrix}$$

$$\begin{pmatrix} IR1 \\ IR2 \\ IR3 \end{pmatrix} := \text{Find}(IR1, IR2, IR3) \quad \begin{pmatrix} IR1 \\ IR2 \\ IR3 \end{pmatrix} = \begin{pmatrix} 18.792 \\ 13.003 \\ 11.061 \end{pmatrix} \text{ mA}$$

Figure 5. Calculating mesh currents using MathCad

Lab 8: Design and Evaluation of First-order RC Low and High Pass Filters

$$\text{Design specification of cut-off frequency:} \quad f_{co} := 1000\text{Hz} \quad \omega_{co} := 2\pi \cdot f_{co}$$

• Low pass filter design and evaluation:

1. Choose a capacitor: $C := 1\mu\text{F}$

2. Design a resistor for cut-off frequency requirement: $R := \frac{1}{C \cdot \omega_{co}} \quad R = 159.155 \text{ ohm}$

3. Frequency response plots: $\omega := 0, 0.1 \dots 6 \quad \omega_1(\omega) := 2\pi \cdot 10^3 \cdot s^{-1} \quad H(\omega) := \frac{1}{1 + \frac{j \cdot \omega_1(\omega)}{\omega_{co}}}$

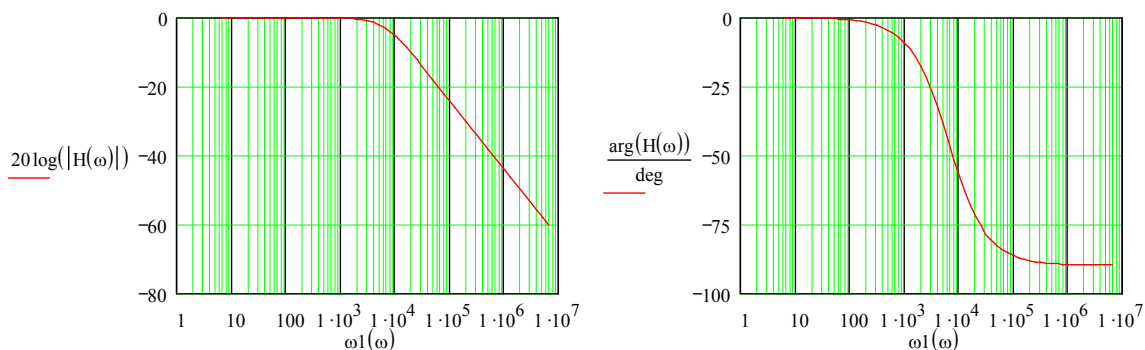


Figure 6. MathCad worksheet used for designing and evaluating a first-order RC low-pass filter

MathCad is also used to draw plots for given theoretical equations of an experiment. Figure 6 shows an example of using MathCad to calculate and draw the bode plot of a series RC low pass filter [6]. Again it can be seen as an advantage to use MathCad to provide a clear view on the theoretical background of the experiment represented in both equations and plots. For any design specification change represented in the cut-off frequency, the MathCad can quickly recalculate new design data and redraw the bode plot, making students able not only to rapidly find an appropriate design for their need but also to see the design performance in bode plot.

It is also convenient to export MathCad data such as the bode plots of Figure 6 to a data file. Data saved in a file can be easily reloaded in a computer and then compared with NI LabView or NI ELVIS measured results so as to most effectively re-emphasize students' understanding of theories (see section VI).

5. VIRTUAL LABORATORY USING PSPICE

PSpice is a general-purpose circuit simulation program that simulates electric and electronic circuits and also has various circuit analysis tools for simulation data [7]. The PSpice student version is free to download [8]. Although it is significant to incorporate NI ELVIS and MathCad for quality education of the laboratory course, the valuable role of the circuit simulation cannot be replaced [9]. First, PSpice simulation can provide a strong virtual prototype for a practical experiment. Second, it is much faster to build or modify an electric circuit or system in simulation than in hardware. Third, with rich analytical tools and functions in PSpice, analysis and design of an electric or electronic circuit is much easy to perform in software simulation environment.

A virtual laboratory using PSpice has been shown successful for quality education in the Circuits and Electronics Lab at TAMUK. For each experiment, there is one hour for lecture and three hours for experimentation. PSpice simulation is required before experimentation. With the PSpice simulation, students can get clear virtual understanding of an experiment before building actual measuring circuits, which will prepare them for more productive measured results. Figure 7 shows a circuit layout of an experiment for designing and implementing digital to analog converter (DAC) [3]. Students start their designs of the DAC in simulation. In PSpice simulation, they can quickly set up a circuit, check design errors, and simulate the circuit to view transients of any node voltage or branch current as shown in Figure 8. Figure 8 shows the four bit digital

Lab 13: Design and Implementation of Digital to Analog Converter

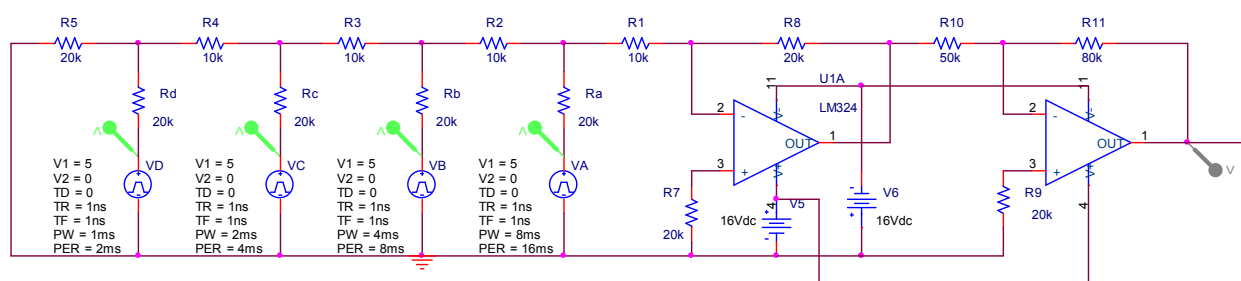


Figure 7. Simulation circuit of Digital to Analog Converter in PSpice

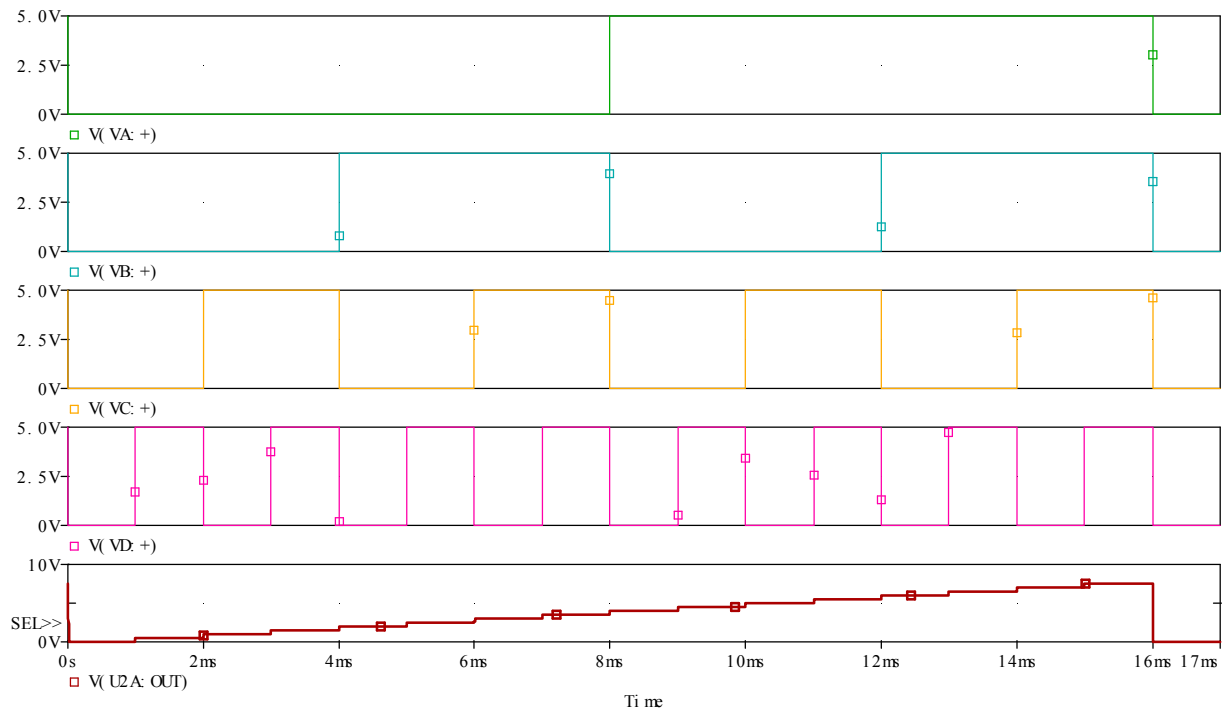


Figure 8. Simulation results of the digital inputs and analog output of the DAC

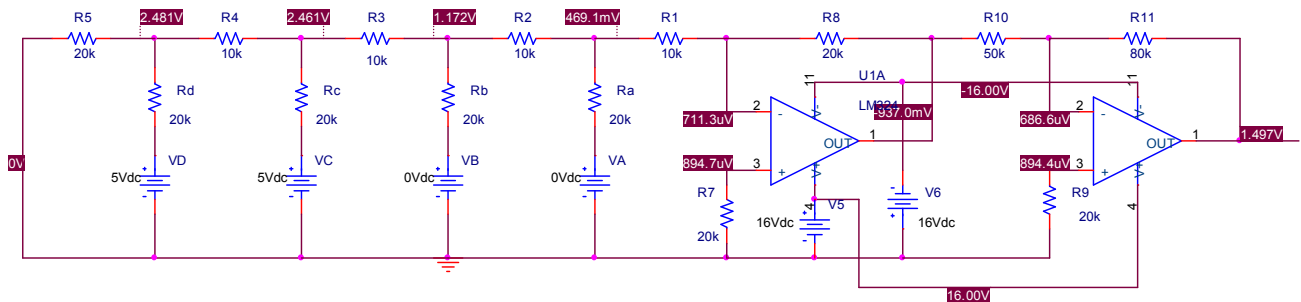
inputs representing a counter and the triangular analog output. Through the simulation, students can see and understand the basic concepts of DAC such as the gain, range, and resolution and how those parameters can affect the DAC or can be modified [10]. After the simulation analysis and design, students can quickly build a practical DAC circuit, avoiding possible errors or mistakes that could damage any electric and electronic component or device, and obtain more meaningful experimental results.

The simulation is also used during a lab experiment to assist students debugging a laboratory circuit or understanding the reasons for some evident inconsistencies between measurement and pre-theoretical calculations. For example, after setting up a DAC circuit of Figure 7, students usually have difficulties to get the circuit to work properly. With PSpice simulation, they can simulate voltages of various nodes for a digital bit combination as shown in Figure 9 a, use the LabView program specifically designed for the experiment (Figure 9 b) to generate this bit combination, measure node voltages of the practical DAC circuit, and compare them with the simulated results. This way, possible problems of the DAC circuit can be quickly identified.

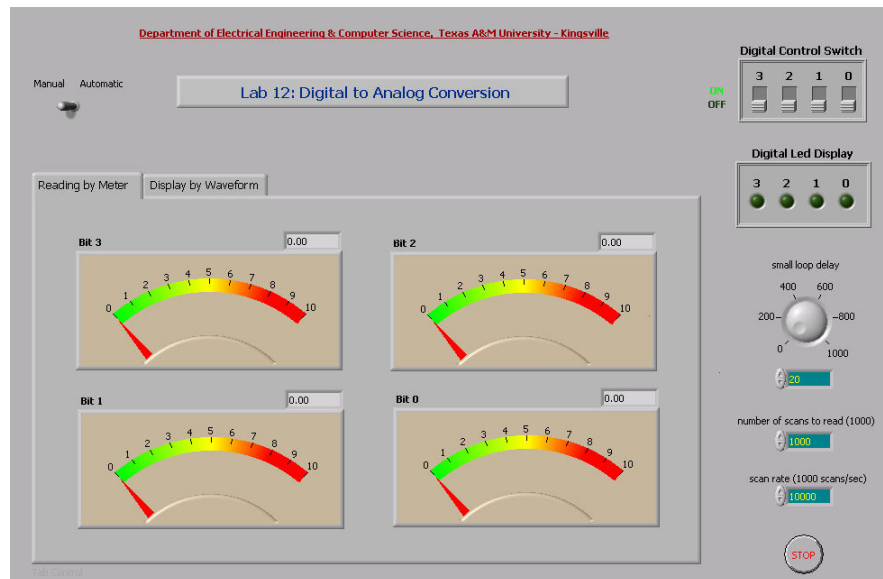
In some situations where measured and simulated results do not match due to factors such as internal impedance in a source and/or in electric/electronic components, it is very easy for students to modify the corresponding schematic circuit, re-simulate the modified circuit, and then compare new simulated results with measured results. In this way, not only can theories be clarified but also the deviation of the measured results caused by such factors can be understood.

It is critical to point out that PSpice simulation results can be easily exported to a file or other software and then compared with measured results. This is not only true for small amount

of data obtained from bias point simulation but also true for large amounts of data obtained through DC sweep, AC sweep, and transient simulations, giving students a fast way to compare results obtained through LabView/NI ELVIS measurement, PSpice simulation, and MathCad calculation.



a) Node voltage simulation for a digital bit combination



b) LabView front panel for measurement and debugging of DACs

Figure 9. Debug a DAC Circuit with the Assistance of PSpice Simulation

6. COMPARING MEASURED, SIMULATED AND THEORETICALLY CALCULATED RESULTS

Comparisons among measured, simulated, and theoretically calculated results are important for effectively re-emphasizing students' understanding of theories. Two computer software packages, Microsoft Excel and MatLab, have been used for the result comparisons.

For each lab experiment, tables in Microsoft Excel spreadsheets [11] are designed. In addition to the purpose of the comparisons, they are also used for data recording and some automatic theoretical calculations. This electronic data recording and calculation could greatly save the time of an experiment, making students focus more on other aspects of measurement and analysis.

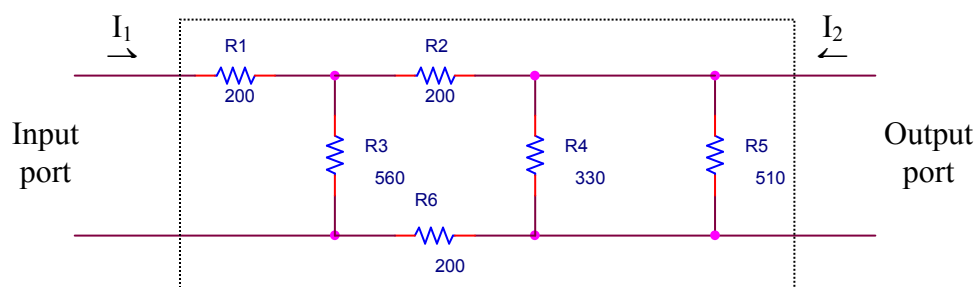


Figure 10. A circuit layout for lab experiment of Two-port Networks and Equivalent Circuits

B10		=		=IF(OR(B6="",B7=""),"",B6/B7*1000)		
	A	B	C	D	E	F
1	Table 1: Z parameters of a two-port network					
2						
3	a) Measuring Z parameters					
4	Output port open circuit test			Input port open circuit test		
5	Variables	Measured	Simulated*	Variables	Measured	Simulated*
6	V ₁ (V)	10.01	10	V ₁ (V)	5.8	5.833
7	I ₁ (mA)	20.37	20.42	I ₁ (mA)	0	0
8	V ₂ (V)	1.96	1.974	V ₂ (V)	9.98	10
9	I ₂ (mA)	0	0	I ₂ (mA)	59.8	60.33
10	Z ₁₁	491.4089347	489.7159647	Z ₁₂	96.98996656	96.68489972
11	Z ₂₁	96.21993127	96.66993144	Z ₂₂	166.8896321	165.7550141
12						
13	b) Equivalent T network of measuring circuit					
14	Equivalent T network parameters					
15	Za (ohms)			395.1890034		
16	Zb (ohms)			69.89966555		
17	Zc (ohms)			96.21993127		
18						
19	c) Use of Z parameters - measured/simulated load voltage and current					
20	Method used				Load voltage	Load current
21					V _L (V)	I _L (mA)
22	Measurements from original network				1.71	1.72
23	Simulations from Z-parameter network*				1.72	1.72
24	Measurements from equivalent T network				1.72	1.72

Figure 11. A circuit layout for lab experiment of Two-port Networks and Equivalent Circuits

Figure 10 shows a circuit for measurement of Z parameters in a two-port network [3, 6] and Figure 11 gives corresponding spreadsheet tables designed for data recording and partial theoretical calculations. The Z parameters of the two-port network can be automatically calculated in Table 1 of Figure 13 when relevant measured/simulated data of the open circuit tests at input and output ports are recorded in the table. For example, the Z parameters of Z_{11} and Z_{21} are automatically calculated when data in cells B6, B7, and B8, or C6, C7, and C8 are provided by a student based on the measured or simulated results from output port open circuit test. Cells B9 and C9 are pre-specified constants of 0 mA and are password protected. The formula used for B10, for instance, is

$$=IF(OR(B6="",B7=""),"",B6/B7*1000) \quad (1)$$

According to (1), the calculated Z parameters are shown only after measured results in cells B6 and B7 are provided. The parameters for the equivalent T network of the measured circuit are also automatically calculated when results in cells B10, B11 and E10, E11 are available. The formula used for calculation of Z_a in cell D15 is

$$=IF(OR(B10="",B11=""),"",B10-B11) \quad (2)$$

Besides the example presented in Figures 12 and 13, Microsoft Excel is also used significantly to compare in plots and curves among results obtained through MathCad calculations, PSpice simulations, and NI ELVIS measurements [12]. Before the comparison, results from MathCad, Pspice, and NI ELVIS are normally exported into different files. Then, data from those files is loaded into an Excel spreadsheet, and comparison plots can be easily drawn.

However, for comparison in spreadsheets, data needs to be collected consistently such as having the same starting and ending times and time intervals, or having the same starting and ending frequencies and frequency intervals. But these conditions may not be always true for measurement, calculation, and simulation in three different environments. In order for comparisons under these conditions, MatLab has been used.

Figure 12 gives the main MatLab code lines for comparing results saved in three different files for a high-pass filter experiment. The program loads in data on line 6, and draws results from Mathcad calculations, PSpice simulations, and NI ELVIS measurements on line 10, 14, and 18 respectively. The MatLab “hold on” command on line 11 and 15 makes it possible to draw results obtained in different software environments in the same plot without the requirement for all the curves to have the same starting and ending frequencies and frequency intervals. It is necessary to point out that the basic structure of the program is suitable for comparison purpose of all the other experiments of the course. Figure 13 shows the comparison of bode plots generated/obtained by Mathcad, PSpice, and NI ELVIS for a RC highpass filter. The comparison in MatLab gives more flexible and effective ways to compare measured, simulated and calculated results generated in different software environments so as to more effectively re-emphasize students’ understanding of theories.

```

1 %*****
2 %   Matlab Program for Comparing Mathcad, PSpice and NI ELVIS   %
3 %*****
4
5 % 1. Loading data
6 load Mathcad_hp.txt; load pspice_hp.txt; load ELVIS_hp.txt;
7 subplot(2,1,1);
8
9 % 2. Plot Mathcad results
10 x1=Mathcad_hp(:,1); y1=Mathcad_hp(:,2); semilogx(x1,y1);
11 hold on;
12
13 % 3. Plot PSpice results
14 x2=pspice_hp(:,1); y2=pspice_hp(:,2); semilogx(x2,y2,'go');
15 hold on;
16
17 % 4. Plot NI ELVIS measured results
18 x3=ELVIS_hp(:,1); y3=ELVIS_hp(:,2); semilogx(x3,y3,'r+'); grid;
19 grid on;
20

```

Figure 12. MatLab program for comparing Mathcad, PSpice, and NI ELVIS results

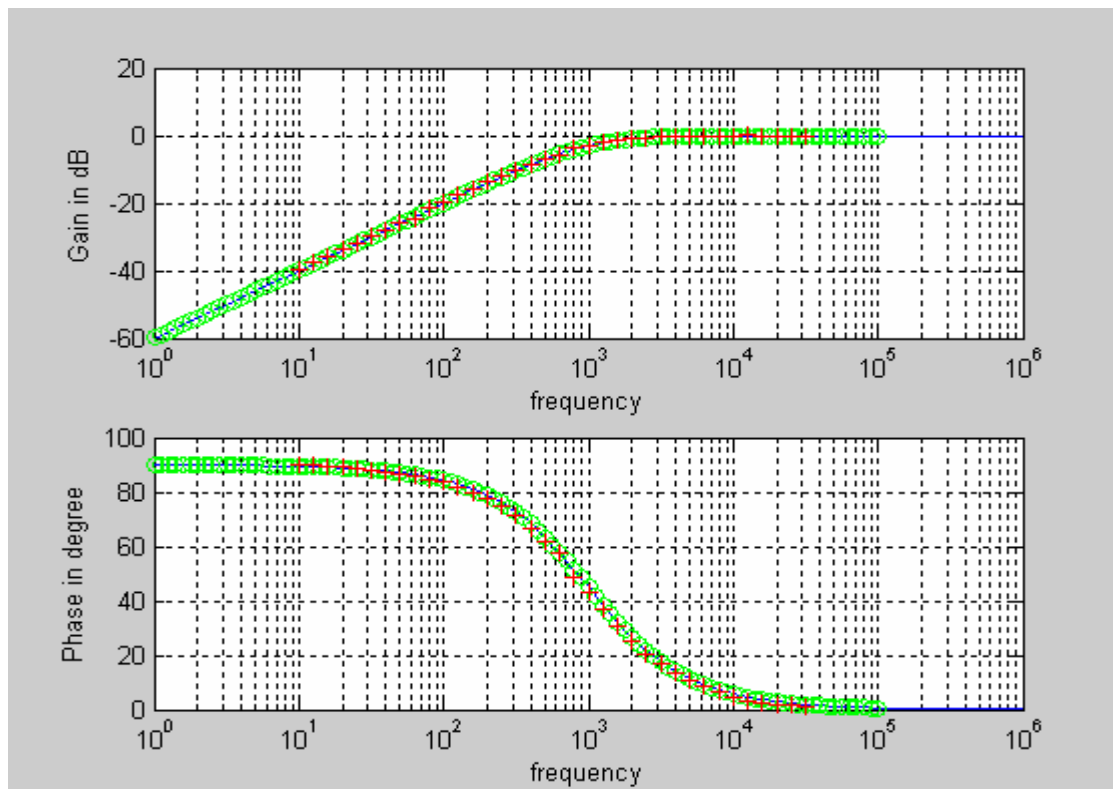


Figure 13. Result comparison among Mathcad calculation '—', Pspice simulation '—o—', and NI ELVIS measurement '—+—'

7. CONCLUSIONS

The paper gives the methodologies and approaches of developing a digital laboratory for analysis, design, simulation, and measurement of typical electric and electronic circuits using modern computer hardware and software. It has been shown in the paper that the use of LabView and NI ELVIS makes it possible to get experimental results more efficiently, to have much more lab contents, and to let students focus more on results, concepts, relations to theories and/or design aspects of an experiment. MathCad has been illustrated to be able to effectively show the relationship between mathematical equations and electrical variables represented in data, curves, and plots, making the theoretical background of an experiment easy to see and understand. PSpice simulations provide strong virtual laboratory environment, preparing students for more productive results from a practical experiment. The simulations also show significant advantages in assisting students on debugging a laboratory circuit or understanding the reasons for some evident inconsistency between measurements and pre-theoretical calculations. The mechanisms and approaches designed for comparisons among measured, simulated, and theoretically calculated results are important for more effectively re-emphasizing students' understanding of theories for the purpose of quality education.

8. REFERENCES

- [1] Robert H. Bishop, "LabView 6i," Prentice Hall, 2001.
- [2] National Instruments, "LabView 7 Express User Manual," National Instruments Corporate, Austin, TX, 2003.
- [3] Shuhui Li, "Lab Manual for Circuits and Electronics Lab," available from <http://www.engineer.tamuk.edu/departments/eecs/Faculty/Li/EEEN3212/eeen3212.html>.
- [4] National Instruments, "LabVIEW Introduction Course – 3 Hours," available from http://www.ni.com/academic/lv_training/how_learn_lv.htm.
- [5] "Mathcad 11 User Guide," Mathsoft Engineering & Education, Inc., Cambridge, MA, 2002.
- [6] A. B. Carlson, "Circuits", Brooks/Cole, 2000.
- [7] Paul W. Tuinenga, *SPICE - A Guide to Circuit Simulation & Analysis Using PSpice*, Prentice Hall, 1995.
- [8] "PSpice 9.1 Student Version," available from http://www.cadence.com/products/si_pk_bd/index.aspx.
- [9] J. Olin Campbell, John Bourne, Pieter Mosterman, and Arthur J. Brodersen, "The Effectiveness of Learning Simulations for Electronic Laboratories," *Journal of Engineering Education*, Vol. 91, No. 2, 2001.
- [10] Jonathan W. Valvano, "Embedded Microcomputer Systems: Real Time Interfacing," Brooks-Cole Publishers, 2000.
- [11] "Microsoft Excel User's Guide," Microsoft Corporation, Redmond, WA, Version 9.0, 1999.
- [12] A. El-Hajj, K. Y. Kabalan, and M. Al-Husseini, "Antenna Array Design Using Spreadsheets", *IEEE Transactions on Education*, Vol. 46, No. 3, August 2003.

SHUHUI LI received his B.S. and M.S. degrees in Electrical Engineering from Southwest Jiaotong University in Chengdu, China in 1983 and 1988 respectively and Ph.D. degree in Electrical Engineering in 1999 from Texas Tech University. From 1988 to 1995, he was with the School of Electrical Engineering at Southwest Jiaotong University, where his research interests were in the areas of modeling and simulation of large dynamic systems, dynamic process simulation of electrified railways, power electronics, power systems, and power system harmonics. From 1995 to 1999, he involved into the research areas of renewable energies, neural networks, and applications of

massively parallel processing. He joined the Department of Electrical Engineering and Computer Science at Texas A&M University - Kingsville (TAMUK) in 1999. He is currently an associate professor at TAMUK. He is a member of IEEE.

ABRAR A KHAN completed his Bachelor of Engineering from Government College of Engineering Amravati Maharastra, India. He is currently pursuing his Masters degree in Electrical Engineering at Texas A&M University – Kingsville. He was the Recipient of Graduate CS Scholarship 2002-2003. His research topic is to develop Circuits and Electronics Lab using National Instruments software and Hardware. His interests include Automotive Electronics, Control systems, Power systems, System Design and Microprocessors.