Apr 14th, 12:00 AM - 12:00 AM

Rapid Development of Advanced Virtual Labs for In-Person and Online Education

Yiyang Li
Old Dominion University

Pauline Delacruz
Old Dominion University

Yuzhong Shen
Old Dominion University

RAPID DEVELOPMENT OF ADVANCED VIRTUAL LABS FOR IN-PERSON AND ONLINE EDUCATION

Yiyang Li, Pauline Delacruz, Yuzhong Shen
Department of Computational Modeling and Simulation Engineering
Old Dominion University
Norfolk, VA, USA
{yli013, pdela003, yshen}@odu.edu

ABSTRACT

This abstract discusses methodologies and preliminary findings on rapid development of advanced virtual labs using modeling and simulation for in-person and online education, including rapid generation of virtual environment, integration of state-of-the-art industry leading software tools, advanced software design techniques that enables large scale software reuse, and innovative user interface design that facilitate the configuration and use of virtual labs by instructors and students. The latest design and development of the virtual lab for electronic circuits is presented.

Keywords: virtual environment, 3D scanning, circuit simulation

1 INTRODUCTION

With the advancement in computer technologies and the changes COVID-19 has made to the way students attend classes, online education has become increasingly important. Instructors can conduct traditional teaching activities through online conference tools such as ZOOM and WebEx, however the lab courses in general do not match the quality of in-person labs. This is because in most cases, instructors use live stream or recorded video for lab demonstration that does not allow the learner to participate in the experimental process, thus the content and concepts learned by the students are limited. Computer-based modeling and simulation (M&S) approaches have great potential to address this problem. By using the same virtual 3D environment as the lab classroom, students can carry out experiments and observe the results on their electronic devices. Moreover, student can also avoid potential hazards on human or lab equipment. For example, an explosion effect or warning message can be prompt when the student has built a short circuit.

2 METHODOLOGY

The correct lab instruments proportions consistent with the real world is crucial in order to build a realistic lab environment. To obtain accurate representations of the instruments and reduce development time, 3D scanning was employed, and the Structure Sensor 3D scanner was used to scan the lab instruments. Then the model was generated based on the collected raw data. Figure 1 (a) shows the original oscilloscope model directly generated by the Structure Sensor and (b) the final 3D model that was generated using Maya based on the original model.

The virtual lab was designed for PHYS 303 Intermediate Experimental Physics in the Physics Department. The equipment involved includes oscilloscope, digital multimeter, DC power supply, digital/analog trainer, and function generator. Users can use these instruments to build circuits and perform circuit analysis in the virtual lab. Figure 2(a) shows the 3D model of the digital/analog trainer.

Figure 1. (a) Oscilloscope model generated by the Structure Sensor (b) Final 3D model.

Currently, a virtual lab prototype capable of handling circuit component connection logic has been developed. The prototype allows user to place circuit components, connect circuit components to a breadboard, and to place jump wires between instruments. Figure 2(b) shows the breadboard and an test connection between components.

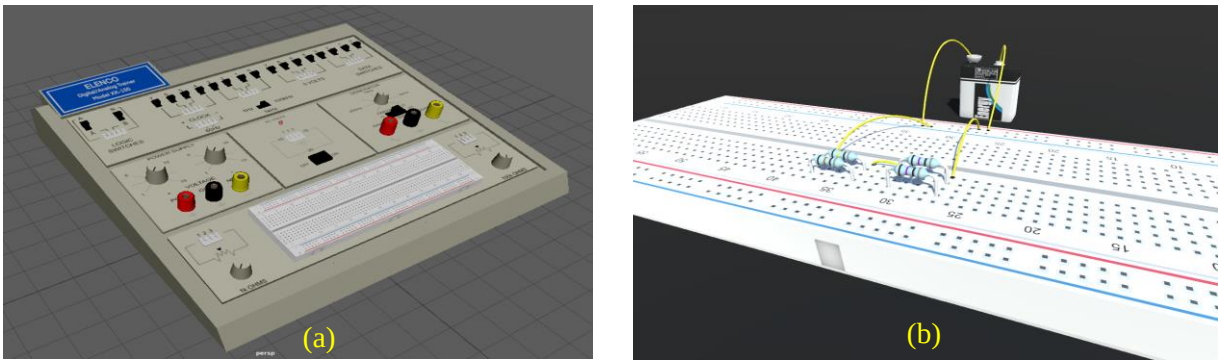

Figure 2. (a) Digital/analog trainer model (b)Circuit component connection example

In order to obtain accurate circuit simulation results, SPICE (Simulation Program with Integrated Circuit Emphasis), an open-source general-purpose analog circuit simulator, was utilized as the simulation engine. After the user finishes building the circuit, the program extracts the topology of the circuit and generates a SPICE script, and then the program calls SPICE to simulate the generated script and outputs the results. Figure 3 (a) shows the SPICE script generated from an example circuit and (b) the simulation output.

```
Combined series & parallel voltage divider

*resistor connected between nodes 1 and 0
R1 1 0 7
*dc voltage connected between nodes 2 and 0
V1 2 0 dc 9
*resistor connected between nodes 1 and 0
R2 1 0 7
*resistor connected between nodes 2 and 1
R3 2 1 1
*resistor connected between nodes 2 and 1
R4 2 1 4

.op
.end
```

(a)

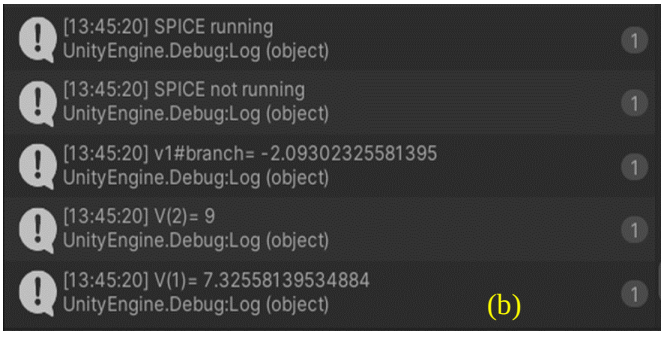

Figure 3. (a) The SPICE script generated from the circuit. (b) The simulation output.

3 CONCLUSIONS

In this project, the virtual lab prototype is presented. The program implements the extraction of the circuit topology, the generation of SPICE scripts, and the interaction with the SPICE simulator. In future work, the real labs will be modeled, as well as the design and implementation for the different sections of the lab course content.