Research on the Application of Simulation Bench in Experimentation Teaching of Electrical Engineering and Electronics

Rongli Wang, Xiaojing Li, and Hongyue Liu

School of electrical engineering, Tianjin University of Technology, Tianjin, 300384, China

iobox@126.com

Keywords: Experimentation teaching; EWB; PSpice; Simulation Experimentation Bench

Abstract. To adapt the urgent demand of modern society, the graduates of higher education must not only know the theories deeply, but also have the high abilities in practice and innovation. There are some shortages in the traditional experimentation of Electrical Engineering and Electronics with the limit of condition, which does harm to students' abilities. EDA (Electronic Design Automation) technology is introduced into the teaching of practice due to its rapid development. Then, a simulation experimentation bench, based on PSpice and EWB, is combined with traditional Experimentation and served for students whose major is or not electricity. As a result, this method reforms the traditional experimentation.

1. Introduction

Electrical engineering and electronics courses require high abilities in practice, and hence experimentation teaching plays a significant role in training students' abilities in practice and innovation. Due to limitations in traditional teaching approaches, students could only finish verification of pre-designed lab contents, being lack of hands-on circuit design experience, which is harmful to increase their overall qualities and abilities [1].

To accommodate technology developments in the 21th century, we have introduced EDA style simulation software, EWB and PSpice. Taking advantages of high efficiency, high precision, zero loss, high flexibility, and realistic simulation, we have built up a simulation bench to provide an innovative experimental environment, making students more interested in, broadening their horizons, and increasing their innovative thinking and consciousness. This simulation bench can be fused with traditional experiment approaches to highlight the student-oriented, open-mind teaching mode. We briefly discuss this simulation bench and teaching mode in this paper.

2. Software Introductions

2.1 EWB.

Electronics Workbench (EWB) is an electrical circuit simulation software developed by a Canadian company Interactive Image Technologies in 1980s [2]. EWB has sufficient libraries of electronic components and common analytical tools, providing functions of electronic circuit design and detailed analysis of circuits. EWB has user-friendly interface and component libraries that are very close to practical circuit, and hence it could handle almost all electric and electronic experiments in lab. Parameters are easy to change, to make the simulation results close enough to practical. Therefore, EWB has advantages of easy to get started, short learning curve, close to practice for students.

The interface of the software is very intuitive, showing electronic circuit and simulation result directly. This is very suitable to teach electric and electronic experiments, and can be used as visualization of circuit theories to make the class content more comprehensive, enhance the understanding of concepts and theories, and train students the abilities of comprehensive analysis and innovative development.

2.2 PSpice.

PSpice is circuit simulation software developed by Microsim Company in US, based on SPICE (simulation program with integrated circuit emphasis) which was initially developed by Berkeley.

After merger of OrCAD and Microsim in 1998, PSpice product was merged into OrCAD company’s business EDA system and became OrCAD/PSpice.

PSpice has enormous component libraries, parameter model libraries, and all sorts of measuring instruments. It has strong schematic design function, circuit simulation function, graphics after-treatment function, and component symbols design function, based on which one can not only make DC/AC/transient circuit character analysis, but also complicated analysis such as Monte-Carlo statistical analysis, worst-case analysis, and optimization analysis etc. It has advantages of short design period, low cost, high quality, strong data processing capability, and open source, and has been widely applied in research and teaching fields.

3. Case Studies

3.1 Schematic.

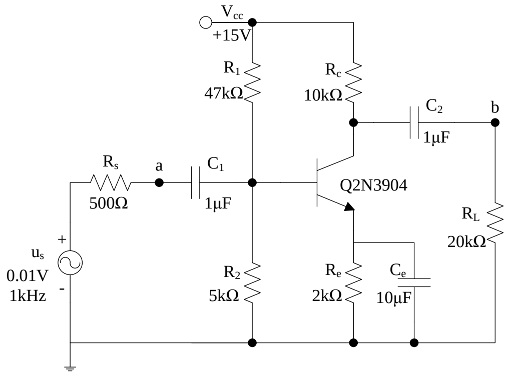

Fig.1 Common emitter amplifier（CE emitter amplifier）

Fig.1 shows a common used circuit in class, the common emitter amplifier [3], and we use it as the case study here, to analyze its amplifier function.

3.2 EWB Analysis.

3.2.1 Schematic Drawing

Getting started with EWB, we can create a new schematic in the working are, and refer Fig.1 to import corresponding components from libraries and modify symbols and parameters. The oscilloscope’s channel A connects to node a (input port), and channel B connects to node b (output port). The connected schematic will look like Fig.2.

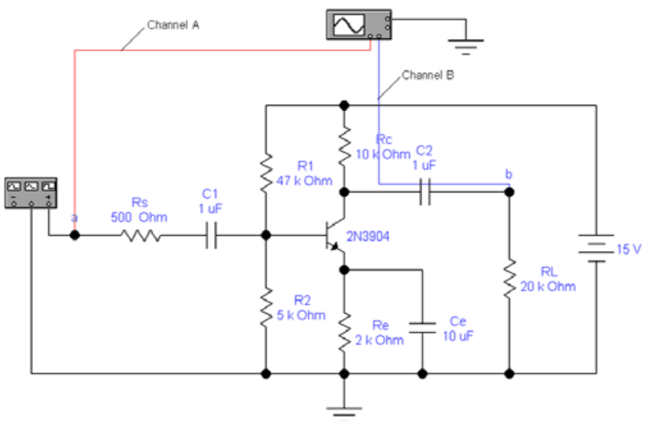

Fig.2 CE amplifier by EWB

Double check the schematic before saving the file, and simulation can be started then by pressing the Start/Stop 1 button on the right side of the EWB interface.

Before making the AC analysis, we need to choose the nodes that will be examined. Choose Circuit/Schematic Option/ Show nodes from the menu, and show the node number on the schematic, then select the corresponding node b. When doing the AC analysis, DC source will be reset to zero automatically, AC source, capacitors, inductances are under AC mode, and input signal will be set to sinusoidal mode. We can then select Analysis/AC Frequency from the menu to have the analysis result of frequency characteristics of CE amplifier, as illustrated in Fig.3.

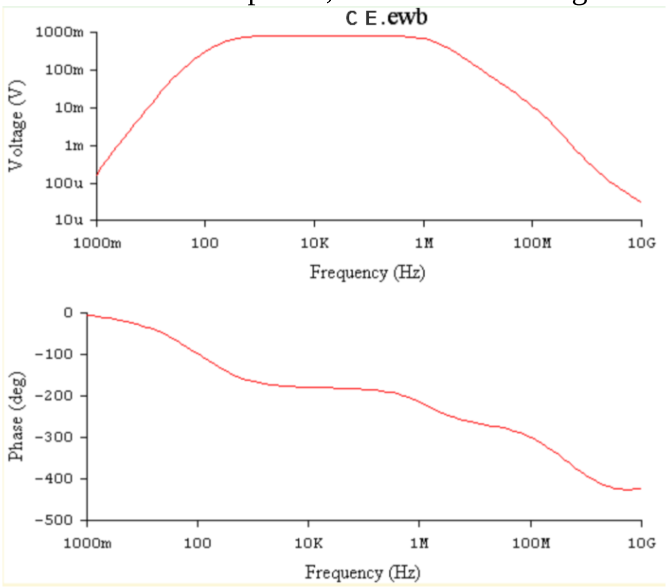

Fig.3 Frequency characteristic of CE amplifier by EWB

Besides, we can double click on the oscilloscope icon to retrieve the input and output waveforms. Now we can do the same settings as operating real oscilloscope to make better illustration. The waveform is shown in Fig.4.

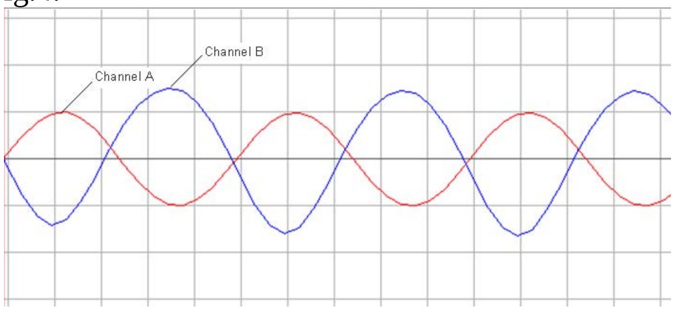

Fig.4 Analysis of transient state by EWB

3.2.2 Simulation Analysis

From the frequency characteristics of CE amplifier, we can have a series of indices such as IF gain, upper and lower cut-off frequency, and pass band. From the oscilloscope result showing in Fig.4, waveform from channel A (input signal) and waveform from channel B (output signal) have a 180-degree phase shift, as well as obvious amplification.

Students are able to design their own schematics, change component parameters, and select suitable parameters from observing waveforms and frequency characteristics. For example, by changing the resistance value of R2 and adjust the position of node Q, students are able to examine the corresponding waveform change caused by changing position of node Q, and hence they can see the instantaneous circuit operation condition and select a proper position for node Q to prevent any cutoff distortion and saturation distortion. After the simulation, students can bring their own schematic to build up practical circuit in lab.

3.3 PSPICE Analysis.

3.3.1 Schematic Drawing

After going into the Capture, we can create a Project and set the type to be Analog or Mixed Signal Circuit. Under the edit circuit window, import components from corresponding libraries and change symbols and parameters to be corresponding to Fig.1. Connect components then and provide numbers for nodes to make simulation easier. Connected schematic looks like Fig.5.

Double check the schematic and set the simulation type as AC Sweep (AC analysis). At the same time, analysis parameter such as start frequency and end frequency can be set as well.

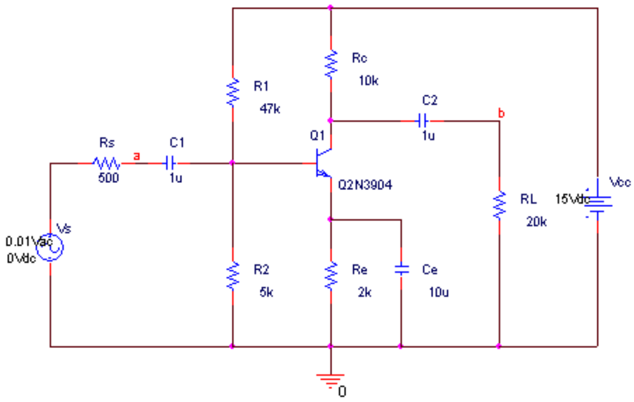

Fig.5 CE amplifier by PSpice

After saving the schematic, we are ready to run the simulation. After simulation, we need to select Add trace under the PSpice A/D Lite window menu Trace, type V(b)/V(a) in the pop up window to obtain the amplitude-frequency characteristic curve. Then the phase-frequency characteristic curve can be also added as shown in Fig.6, where the amplitude-frequency curve uses the left vertical axis, and the phase-frequency curve uses the right vertical axis. Same actions could be done to obtain the impedance curve. Briefly we can reset input source to zero, replace the RL load by a unit source on the output side, and redo the simulation to obtain the impedance curve.

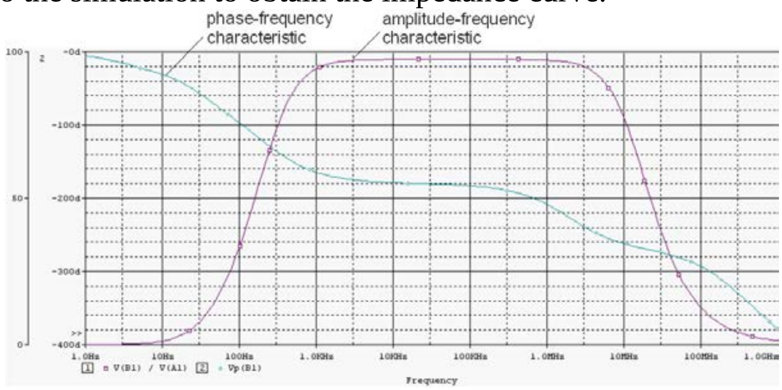

Fig.6 Frequency characteristics of CE amplifier by PSpice

When observation of transient input and transient output waveforms are required, we then need to analyze the transient response of the circuit. Set simulation type as TRAN, and then set other parameters such as start time, end time, sampling period (use VSIN source).

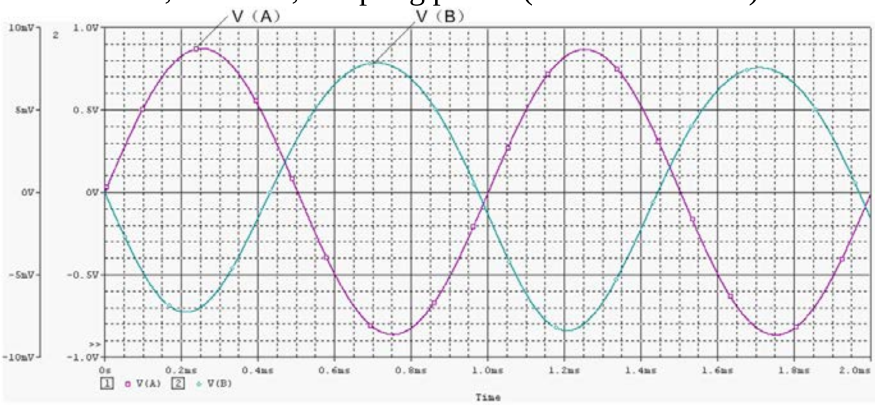

Fig.7 Analysis of transient state by PSpice

After saving the file, it is ready to start simulation. Once the simulation is finished, select Add trace under the PSpice A/D Lite window Trace, and type "V(A)" in the dialog to obtain the input waveform. Similarly, we add output signal V(B) as well, and the waveforms are shown in Fig.7, where the input signal waveform V(A) uses left vertical axis and the output signal waveform V(B) uses right one.

3.3.2 Simulation Analysis

Based on the frequency characteristics in Fig.6, we can use cursor tool to read any coordinate on the plot, and hence every pair of amplitude and phase angle can be obtained from the plot. From the transient state plot shown in Fig.7, there exists a phase shift close to 180 degrees, and vertical axis shows the amplification function as well. Compared with the EWB oscilloscope, this has a better result since observation and reading numbers are much easier.

3.4 Comparisons and Summary.

In those case studies above, we have only used one small function in EWB and PSpice. As a comparison, EWB has the advantages of easy dialog operation, easy to build schematic, full types of components, small program, low price, plenty of functions, and cost-effective [4]. The advantages of PSpice include significant size of component libraries, parameter model libraries, all sorts of measurement instruments, very strong analysis function especially at output waveform that it could handle many tasks that EWB cannot. Overall, EWB is easy to operate while PSpice is more powerful [5]. In practice, we introduce building up schematic from PSpice to electrical and electronic engineering students, and introduce EWB to non-electrical and electronic engineering students [6].

4. Conclusion

To achieve better teaching effect, we need to notice the significance of students' practical abilities and using simulation as a supporting role. Conventional lab experiments can improve students' abilities of solving practical problems, while software simulation can help improve abilities of designing and analyzing circuits. Both of them are indispensable, complementary, and combined with each other. Simulation cannot replace the practical lab experiment. The way we organize the experiment is to ask students design and analyze schematic using software before setting up lab experiment. This combines simulation and experiment, fuses software and hardware, and improves students' overall abilities of design and innovation. It is shown that applying novel approaches to help electrical and electronic engineering lab experiment is a revolution to conventional teaching style, and this is the inevitable trend of modern educational technology development [7].

5. Acknowledgment

We thank our colleagues from school of electrical engineering, Tianjin University of Technology, and we appreciate the help from our leaders and colleagues.

Project Fund: Tianjin Board of Education talent program, "Tianjin universities outstanding young teachers funding" (401007008); Tianjin University of Technology teaching fund, "Reform of electrical engineering teaching system for non-electrical non-mechanical majors" (YB11-63); Tianjin University of Technology teaching material fund (JC 15-06).

6. References

[1]. FU Guirong, SU Hongjuan, JI Gang, Enhancing the Training of Students' Engineering and Comprehensive Quality[J], Research and Exploration in Laboratory, 2006, 25(3), P363-365 (In Chinese).

[2]. WANG Yuqi, JIA Ke, GUO Mengcheng, EDA realizes conf irmation and tests of simulated electric circuit[J], Journal of Changchun University, 2006, 16(2), P33-35 (In Chinese).

[3]. KANG Huaguang, CHEN Daqin, ZHANG Lin, Fundamentals of Electronic Technique(Analog)[M]. Beijing: Higher Education Press, 2008.1 (In Chinese).

[4]. Wang Xiangting, Liu Tao, Zhang Xiaochun, et al. Electrician and electronic technology experimental teaching reform[J]. Journal of Experimental Technology and Management, 2013(4): 112-112 (In Chinese).

[5]. Li Zhensheng, Li Xiaofei, Li Xiaojing, et al. Electrician electronic experiment tutorial [M]. Beijing: Science Press, 2012 (In Chinese).

[6]. Jing Xinxing, Gao Yuan. The construction of practice teaching reform and implementation of local colleges and universities education "outstanding engineers training plan"[J]. Journal of Experimental Technology and Management, 2014(6): 24-26 (In Chinese).

[7]. Li Xiaojing. Explore and practice in improving teaching ability of young teachers in engineering schools [J]. Journal of Heilongjiang Province Higher Education Research, 2015(12): 109-111 (In Chinese).