**Student/Demo Version of Electrical and Electronics Simulation Software –**
**An Enhancement to the Learning of Electrical and Electronics Technology**

by

**Chan Chee Meng**
Politeknik Port Dickson
Negeri Sembilan

## ABSTRACT

*In recent years, many electrical and electronics simulation software packages are made available through the Internet. These simulation software packages are for designing, simulating, and analyzing analog, digital and mixed electronic circuits. Analysis results can be displayed as sophisticated diagrams or on a range of virtual instruments. Some of these simulation software packages are capable of a full-featured analog and mixed-signal simulator supporting everything from high-frequency systems to low-power IC designs and have advanced features enable "what if" analyses, allowing engineers in the electrical and electronics industries to explore various design configurations before committing to final implementation. A few of the simulation software development companies provide software configuration for the teaching and learning electrical and electronics technology. This discussion focuses on how the student/demo copies of these simulation software packages are used in the form of visual aids to enrich students' learning capacities.*

## INTRODUCTION

The prices of the simulation software packages are costly and even with discount for students are still beyond the reach of the majority of the students in our country. However, companies of these software packages provide student or demonstration version. These copies are free of charge to students/users. The student or demo simulation software versions, utilized as visual aids, though with their limitations are a plus point in enriching and enhancing the learning of basic electrical and electronics concepts. Basic electrical and electronics circuit theories that are taught in a classroom environment are reinforced when related circuits are simulated using these simulation software packages. Students are able to view the simulated results concurring with the learning axiom; "you hear you forget, you see you remember and when you do you understand." An added feature of these simulation software packages is students are at liberty to repeat simulation of a circuit with different parameters without much hassle. The repetition of the simulation activity will enhance one's learning process.

**Application of Student/Demo Version of Electrical and Electronics Simulation Software**

**CircuitMaker Student Version [1]**

This simulation software edition of CircuitMaker is only authorized for use by students on their personal home computers. It is not to be used by schools or other educational institutions of any kind. The CircuitMaker Student Version is the same as CircuitMaker Version 6, with the following limitations:
- Maximum of 50 devices per design (any type);
- Device library limited to 1,000 models;
- Symbol editor and Macro feature disabled;
- PCB netlist export limited to TraxMaker format.

**Example 1: Ohm's Law**

Ohm's Law states that the supply voltage, E, in an electrical circuit is directly proportional to the current, I, flowing in the circuit for any one value of the load, R. Figure 1 in Appendix A shows how this concept can simulated by using the DC Sweep Analysis.

**Example 2: Phase relationship between the input and output of small signal amplifiers**

One of the characteristics of common base, common collector and common emitter amplifiers that electronics engineering students need to know is the phase relationship between the input and output of these amplifiers. The common emitter amplifier circuit and its simulation result in Figures 2 and 3 shown in Appendix B, provided by the CircuitMaker simulation software package, illustrates the phase relationship between the input and output of the circuit.

**Example 3: Electronic Fault Diagnosis**

Students who are interested to improve their electronic fault diagnosis skills can make use of this simulation software package. Using the DC voltage comparison method, the fault of an electronic component can be determined. This is illustrated in Figures 4 and 5 as shown in Appendix C when the resistor 3.3k, in the circuit is set at open circuit condition.

**PSpice Student Version Release 9.1 [2]**

The Student Version of PSpice can be distributed freely, providing all copyrights are observed and the software is not redistributed under another name. Circuit simulation of the Pspice A/D is limited to circuits with up to:

- 64 nodes
- 10 transistors
- 65 digital primitive devices
- 10 transmission lines in total (ideal or non-ideal)
- 4 pairwise coupled transmission lines.

The sample library includes 39 analog and 134 digital parts. Device characterization in the PSpice Model Editor is limited to diodes. Stimulus generation in the PSpice Stimulus Editor is limited to sine waves (analog) and clocks (digital). Circuit optimization with the PSpice Optimizer is limited to one goal, one parameter, and one constraint.  CSDF format data files cannot be created. A maximum of 50 parts can only be placed on a schematic design.

**Example 4: Harmonics in a neutral conductor**

In a balanced three-phase four-wire power system, no current will flow in the neutral conductor but when the lines are loaded with nonlinear loads, odd harmonics current flowing in each line will add arithmetically at the neutral conductor. This phenomenon can be visualized after simulating the circuit shown in Figure 5 and the Fourier analysis result is shown in Table 1 in Appendix D.
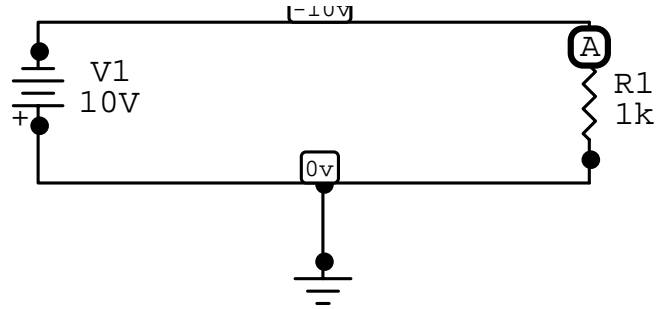

## CONCLUSION

Besides CircuitMaker and Pspice, companies that developed Tina pro [3], Electronics Workbench (Multisim) [4] and Micro-Cap [5] also provide demo copies of their simulation software packages.  The restrictions of the student/demo version of the simulation software packages should not be a deterrent or a demotivating factor to the users. Working within the limit boundary of the simulation software packages will reap much benefit. Students employing the simulation software package as an add-on to the traditional teaching and hands-on methods of learning the electrical and electronics engineering technology will find it an enriching experience.


## REFERENCES

1       CircuitMaker is a registered trademark of Protel International Ltd., Home page: www.circuitmaker.com

2.      Pspice is a registered trademark of Cadence Design Systems, Inc., Home page: www.orcad.com

3.      Tina pro 6 is a registered trademark of Designsoft, Inc., Home page: www.designsoftware.com

4.      Electronics Workbench is a registered trademark of Interactive Image Technology Ltd., Home page: www.electronicsworkbench.com

5.         Micro-Cap is a registered trademark of Spectrum Software, Inc., Home page:
www.spectrum-soft.com

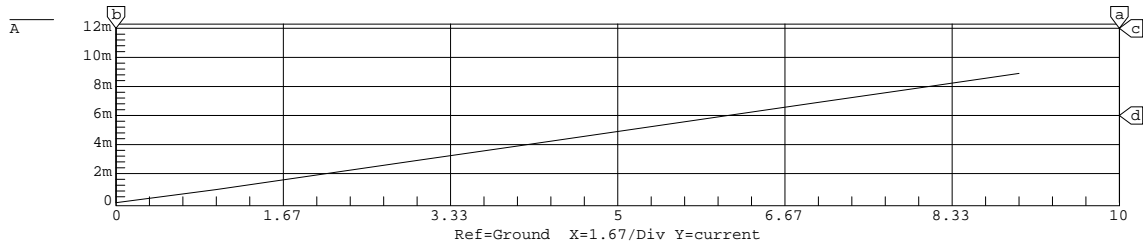Xa: 10.00    Xb: 0.000    a-b: 10.00
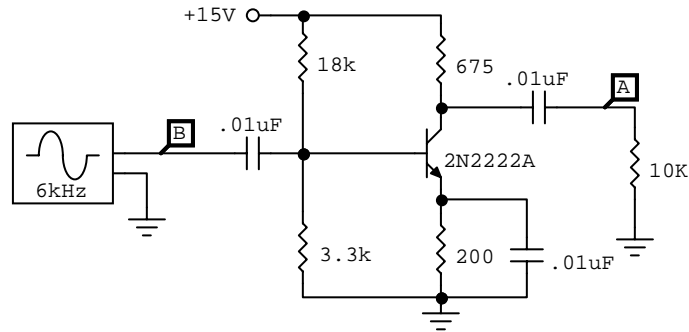Yc: 12.00m   Yd: 6.000m   c-d: 6.000m

Ref=Ground   X=1.67/Div   Y=current

**Figure 1**

+15V

18k

675 .01uF

A

.01uF

B

2N2222A

6kHz

10K

3.3k

200 .01uF

**Figure 2: Common emitter amplifier**

Xa: 833.3u   Xb: 0.000     a-b: 833.3u  freq: 1.200k
Yc: 600.0m   Yd:-600.0m    c-d: 1.200

A

B

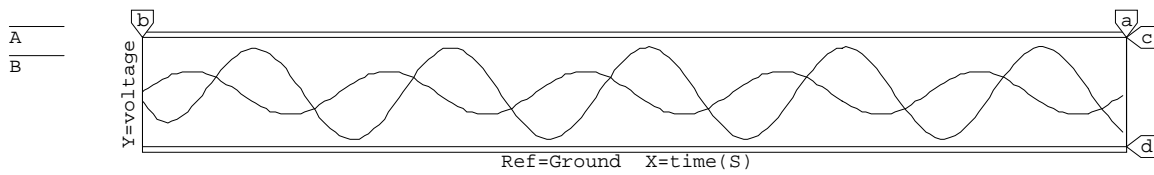b                                                                a

c

Y=voltage

d

Ref=Ground   X=time(S)

**Figure 3: Phase relationship between the input and output**
**of a common emitter amplifier**
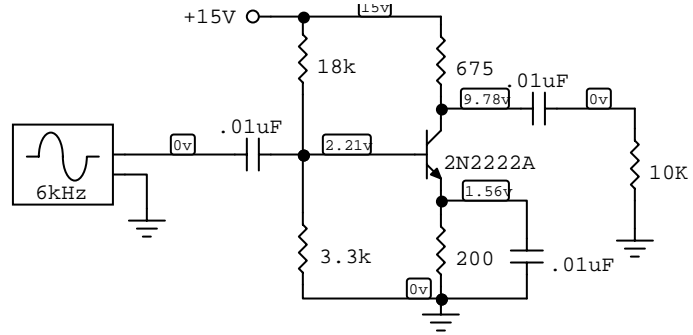
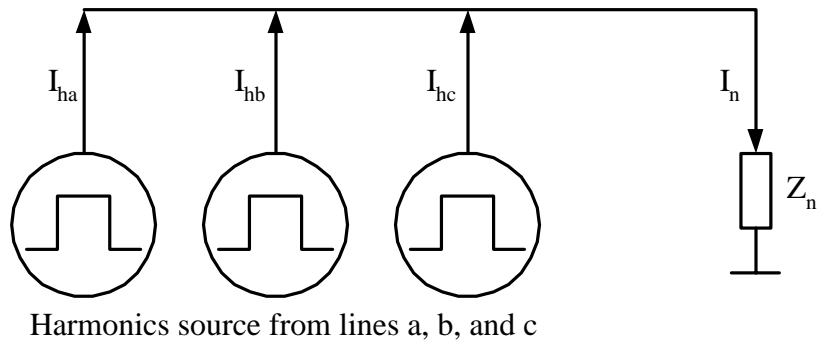**Figure 4: Normal DC voltages**
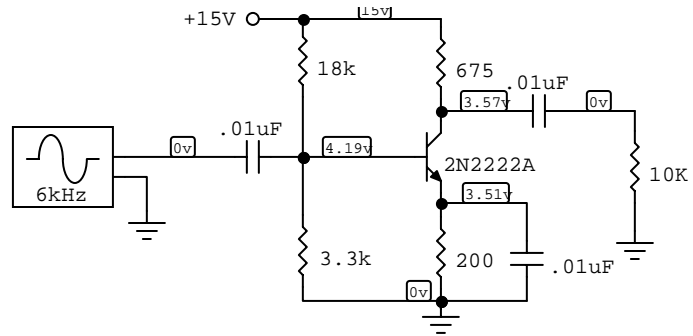


**Figure 5: Abnormal DC voltages**

Harmonics source from lines a, b, and c

**Figure 5: Harmonic currents in a neutral conductor**

| Harmonic Number | $I_{ha}$ | Phase (°) | $I_{hb}$ | Phase (°) | $I_{hc}$ | Phase (°) | $I_n$ |
|---|---|---|---|---|---|---|---|
| 1 | 25.05 | -18.01 | 25.05 | -138 | 25.05 | 102 | 0.0098 |
| 2 | 3.867 | 53.84 | 3.872 | 174 | 3.867 | -65.91 | 0.1071 |
| 3 | 7.292 | -53.97 | 7.281 | -53.98 | 7.289 | -54.06 | 21.86 |
| 4 | 3.502 | 17.71 | 3.499 | -101.9 | 3.5 | 138.2 | 0.0238 |
| 5 | 3.249 | -89.73 | 3.237 | 30.01 | 3.245 | 149.8 | 0.0159 |
| 6 | 2.953 | -18.35 | 2.941 | -17.88 | 2.949 | -17.84 | 8.843 |
| 7 | 1.343 | -124.9 | 1.336 | 113.9 | 1.339 | -6.869 | 0.0457 |
| 8 | 2.301 | -54.29 | 2.281 | 66.14 | 2.295 | -173.9 | 0.0299 |
| 9 | 0.3117 | -155.7 | 0.3123 | -162.8 | 0.3077 | -166.5 | 0.9288 |
| 10 | 1.632 | -90.02 | 1.611 | 150.1 | 1.625 | 29.88 | 0.0226 |
| 11 | 0.2142 | -27.38 | 0.1988 | 103.7 | 0.2142 | -131.7 | 0.0653 |
| 12 | 1.027 | -125.4 | 1.012 | -125.9 | 1.02 | -126.6 | 3.058 |
| 13 | 0.3972 | -58.16 | 0.3764 | -173.1 | 0.3963 | 68.68 | 0.0241 |
| 14 | 0.5459 | -160 | 0.5398 | -42.05 | 0.5397 | 76.42 | 0.0276 |
| 15 | 0.3699 | -92.78 | 0.3486 | -89.23 | 0.368 | -88.3 | 1.086 |

**Table 1: Harmonic currents in a neutral conductor**