Session 2548

COMPUTER AIDED DESIGN PROGRAM

H. Javan, Ph.D., L.Drouin, Ph.D., S.Southall, M.S.
Department of Engineering Technology
The University of Memphis
Memphis, TN 38152

Abstract

A new computer aided design course is now being offered to the students in Engineering Technology Program. The Program is an integration of three different software, namely Electronic Work Bench, Microcap, and PSPICE. Students have carried out a total of fifteen projects covering DC, AC, time and frequency domain analysis, simulation and design. During the final week students were given list of twenty five projects to choose from, simulate, analyze, and make final report. Our evaluation of student's works shows that with this method students have developed a great confidence in applying simulation programs in their carrier.

I. Introduction

Computer Aided Design course in the Department of Engineering Technology of the University of Memphis is a dynamic program offered to all engineering technology students majoring in electronics and computer engineering technology. The Program is composed of using three different software to design a given circuit for desired specifications. The emphasis is on simulation, analysis and design rather than just a CAD. Basic structure of the Program is shown below:

```
                         TECH 3281
                             |
                 ------------------------------------------------
                 |                              |              |
           ---------------------        -----------     ----------
           Elec.Work Bench               Microcap         Pspice
           ---------------------        -----------     ----------
                 |
      ----------------------------------------
      Analog                      Digital
      -----------                 ----------
         |
----------------------------------
Discrete            Integrated
----------          ------------
```

The other segments, Microcap, and Pspice follow the same structure.

II. Exercises Developed

ELCTRONIC WORK BENCH As an example we start with EWB, analog, discrete circuits; we discuss DC, AC, sources using passive RC, RL, and RLC resonance circuits, simulating, analyzing, and designing these circuits, then we discuss active circuits such as designing rectifier power supply, amplifiers, operational amplifiers, and combination of these circuits. In each section first student is given a lecture then they are asked to design a certain circuit for a certain specification, display the result, measure and compare the simulated value with the theoretical values. Each section includes an assignment as a project. For example at the end of analog segment, students were assigned to design an amplifier using discrete and integrated technology to meet the following specifications:

Av >30db

Distortion<10db

The result of this project is indicated in figure 1. As can be determined from this figure the gain is about 40db. At this time, since students were not introduced to frequency analysis, harmonic distortion was not measured, but it was simulated that incorrect biasing of the transistor caused clip, resulting in harmonic distortion. Observation and removing clip from the wave form by selecting proper bias point was very instructive since experimenting this type of circuit on the bread board can be very time consuming.

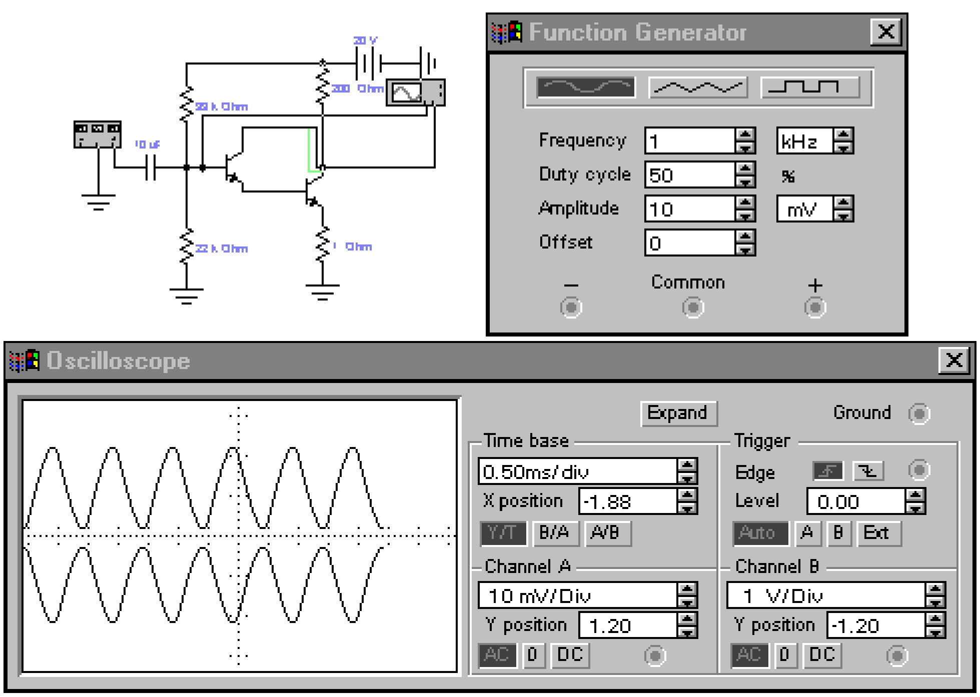


Fig. 1

In the second project student were assigned to simulate, analyze, and to calculate the resonance frequency of a simple RLC circuit using Spectrum analyzer. Figure 2 is the result of this example. The calculated value of resonance frequency is;

$$Fr = 1/2\pi \sqrt{LC}$$

Using the values from the circuit, it is 500 kHz in excellent agreement with the number shown in the lower right hand corner of the figure, you can only verify such a result with a $4000 spectrum analyzer! In addition, students checked the circuit selectivity with different source resistance such as 50 Ohms instead of 10 Ohms. All the obtained results were in good agreement with the calculated and the experimental values.

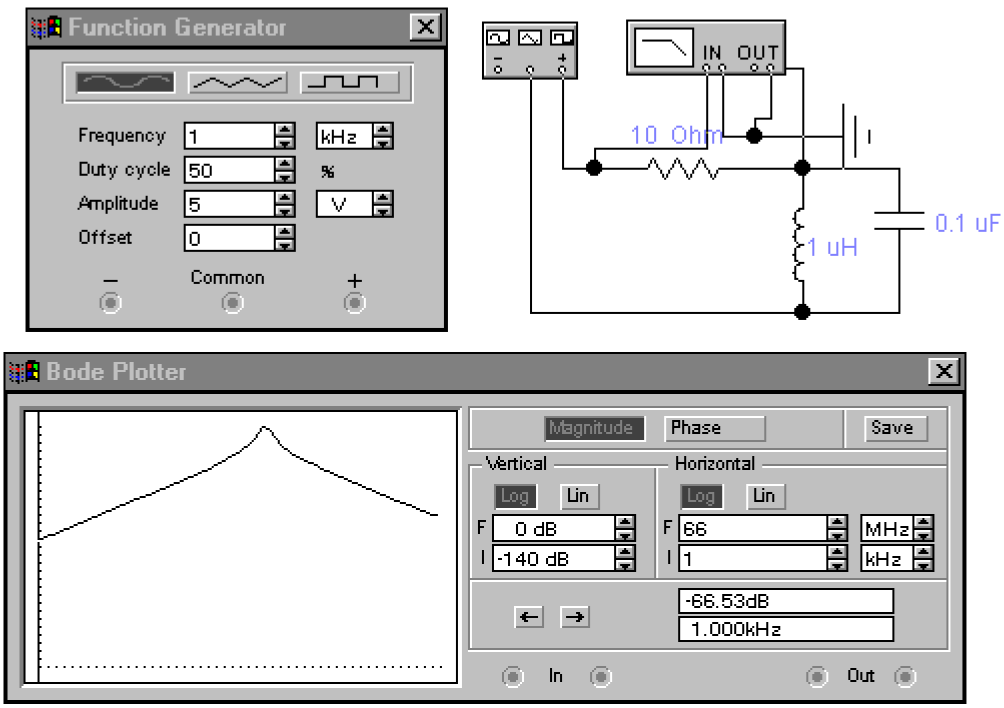


Fig. 2

On the digital part, students were given a count sequence and they were asked to synthesize the circuit using T, R-S, J-K, or D flip-flop which will yield the optimum design from the standpoint of component and the number of pins. Students applied K-Map to implement the circuit, then used the logic analyzer to verify their answers. Table 1 is the required count sequence and figure 3 is the result of one of the students' work. Display of logic analyzer in fact confirms the count sequence. Student applied other flip flops, but J-K resulted in optimum design.

Table 1- Required Count Sequence

	present state Q2	Q1	Q0	next state Q2	Q1	Q0
S0	0	0	0	0	0	1
S1	0	0	1	0	1	1
S2	0	1	1	0	1	0
S3	0	1	0	1	1	0
S4	1	1	0	0	0	0
S5	1	1	1	0	1	0
S6	1	0	1	0	1	1
S7	1	0	0	0	0	1

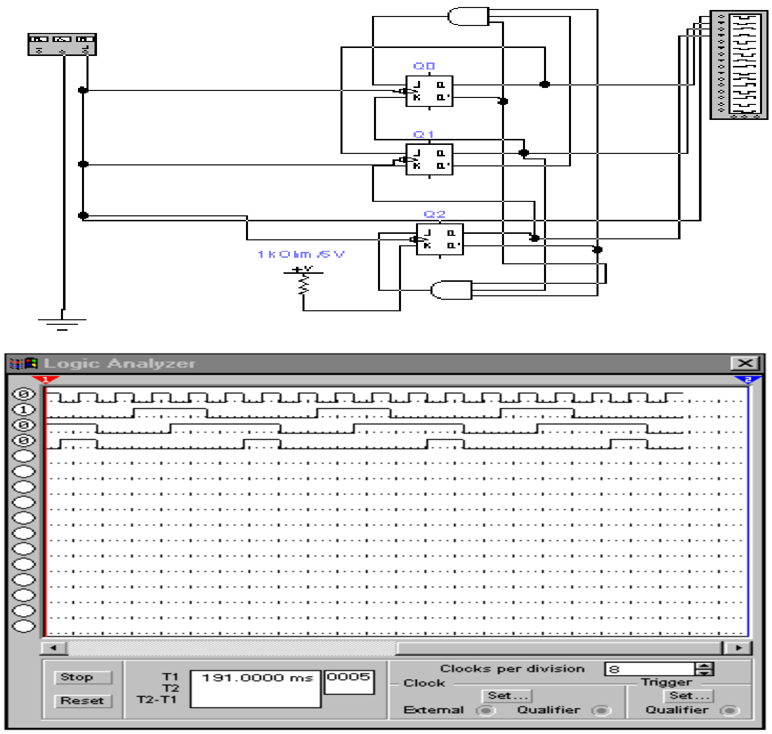


Fig. 3

MICRO-CAP Miro-cap was applied for investigating time and frequency responses. The first one included determination of time constant for two different RC circuit. The first one with different charging and discharging time constant and the second one was equivalent circuit of a simple probe we use with scopes. Both were very instructive, the first with unequal time constant, and the second show the inside of a simple probe which we use almost exclusively every day. The results are shown in figure 4. The calculated and measured time constants were in perfect agreement, except for the second one.

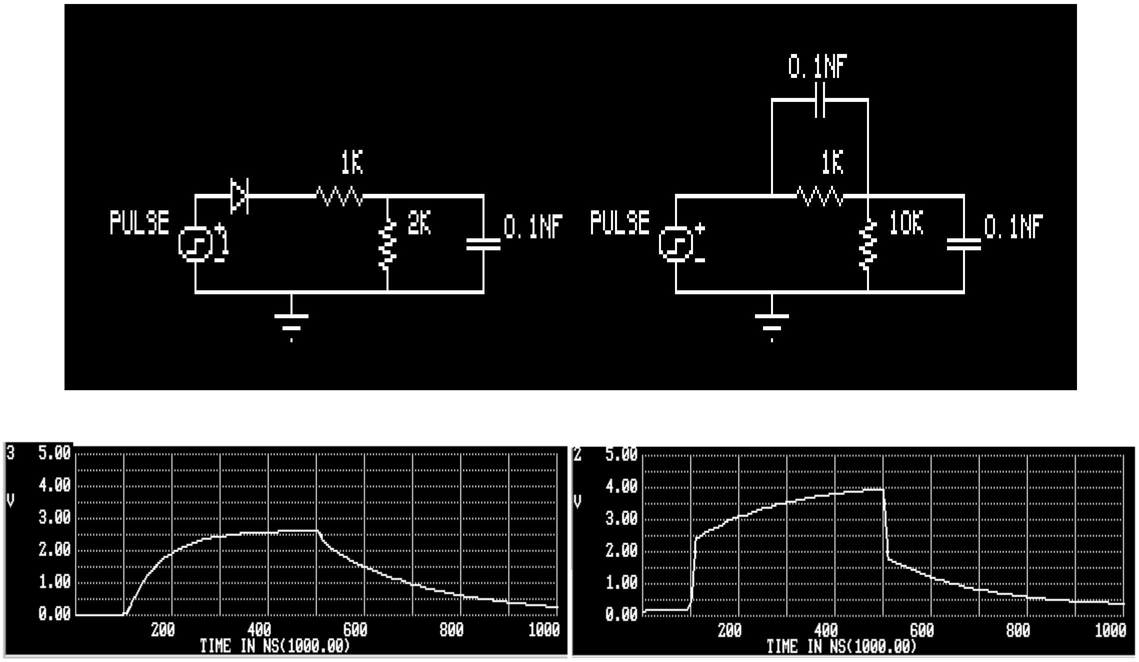
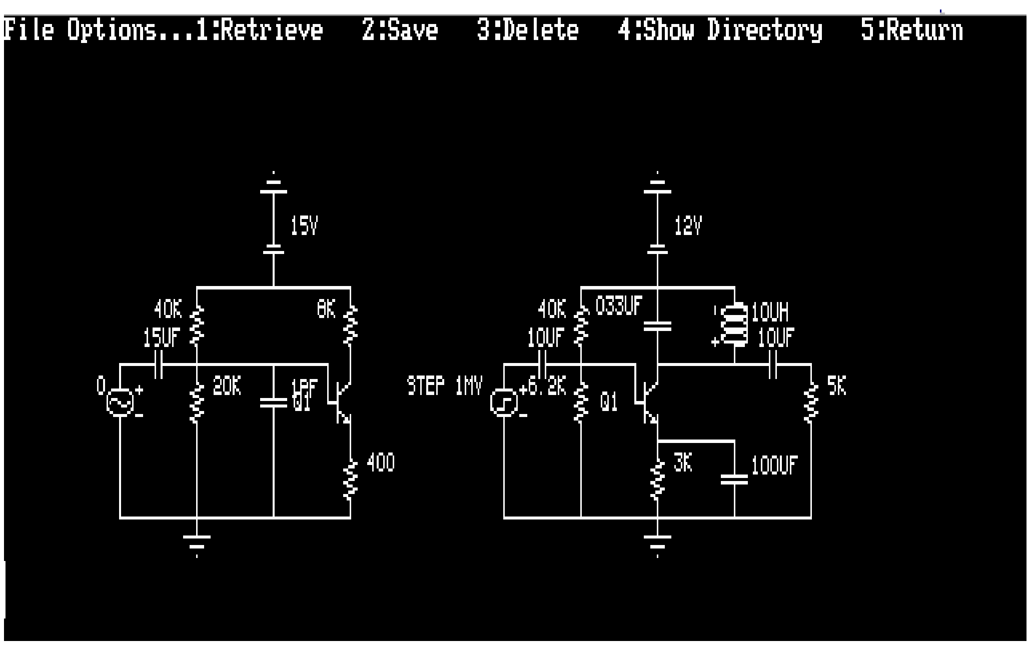


Fig. 4

In the frequency part of Micro-cap students were given two amplifiers, one wide band and the other narrow band. They were asked to determine the band-width and the gain of each amplifier. The results were very satisfactory since other wise we had to use sweeper and spectrum analyzer to determine these parameters. Figure 5 shows the simulated results.



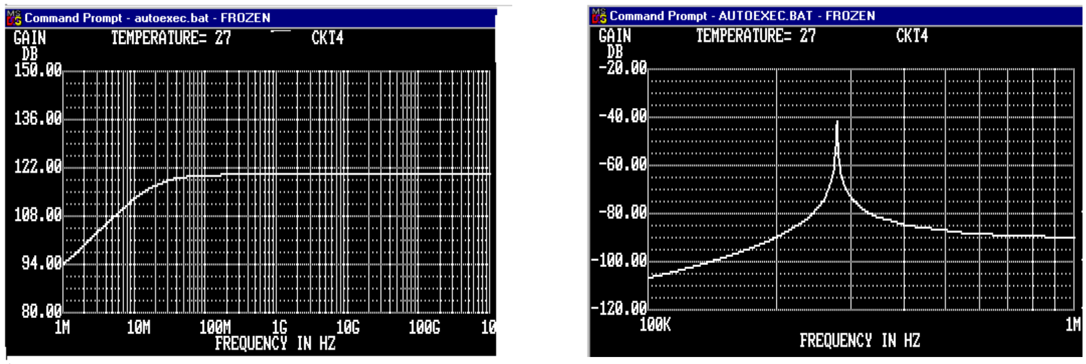


Fig. 5

PSPICE Micro-cap is a little similar to Electronic WorkBench and at the same time applies nodal analysis. Thus after the Micro-cap we introduced PSPICE. In this segment we prefer to write programs. We have used DC, AC, and time domain analysis. Spice is extremely useful to determine DC current especially for complicated and time consuming circuit such as shown in figure 6. The first one is a balanced bridge, a 10 Ohms resistor was inserted in the circuit intentionally, but the circuit is balanced since

$$R1/R2 = R3/R4 \qquad (2)$$

Under this condition, I(RL) should be zero which simulation results confirmed this. The other circuit is a π (delta) networks requiring Mesh Analysis or obtaining Thevenin equivalent circuit . Application of PSPICE, however, makes the analysis of these circuits very simple.

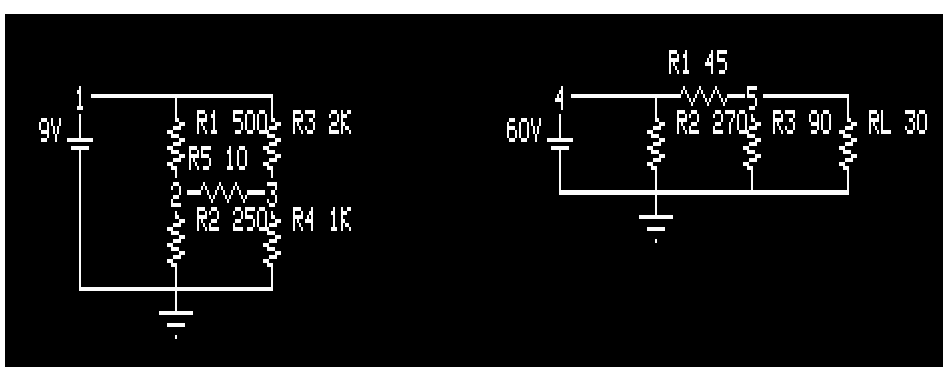


Fig. 6

```
VS 1 0 9V
R1 1 2 500
R2 2 0 250
R3 1 3 2K
R4 3 0 1K
R5 2 3 10
.DC VS 9V 9V 9V
.PRINT DC I(R5)
.END
```

```
I(R5)
0.000E+00
```

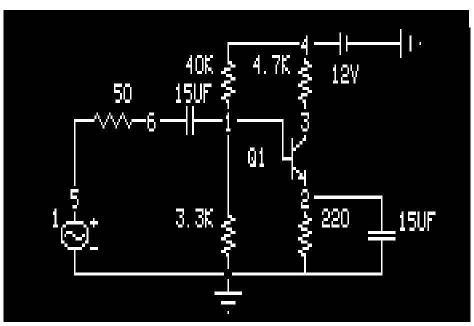
```
VS 4 0 60V
R1 4 5 30
R2 4 0 10
R3 5 0 60
RL 5 0 30
.DC VS 60V 60V 60V
.PRINT DC I(R1) I(R2)
.END
```

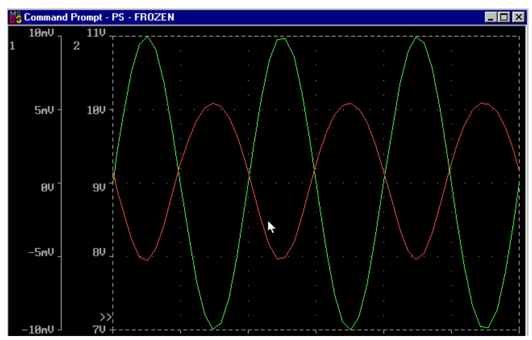
```
I(RL)
6.667E-01
```

Finally as an AC analysis, a simple common emitter amplifier without by pass emitter capacitance was assigned to write a program to simulate the circuit, measure the gain and compare with the calculated values. The result is shown in figure 7. The approximate gain of this amplifier is found from the following equation:

$$Av = Rc/r_e \tag{3}$$

The input file shown, simulated, and calculated values are in excellent agreement.






```
VS 5 0 SIN(0 10mV 5KHz)
Vcc 4 0 12V
RS 5 6 50
R1 1 4 40K
R2 1 0 3.3K
Rc 3 4 4.7K
Re 2 0 220
Cb 6 1 15UF
Ce 2 0 15UF
Q1 3 1 2 BJT
.MODEL BJT NPN
.TRAN 0.02ms 0.6ms
.PROBE
.END
```

Fig. 7

III. Future Work

We are considering adding Low Cost PCB Prototype System so that students can actually prepare dry printed boards of their works and test them with the existing instruments. This will make our course work a complete computer aided design and at the same time it will be an introduction to VLSI technology.

IV. Conclusions

We have introduced this method so that students can develop a complete confidence in simulating a complicated circuit since combination of three different methods makes a good collection of well known software used in industries. Our emphasize is on simulation, design, and analysis. Our approach has following advantages:

1. Students become expert in simulation and analyzing a complicated circuit in a very short time
2. Virtual labs are 100% safe.
3. virtual lab is cost effective. It takes a few $ to update a software, it takes $$$$$ to update the hardware.
4. Some electronic instrument like **Spectrum Analyzer, Logic Analyzer** are very costly and chairmen do not want to spend that much money! On the other hand these instrument are built in the initial cost of software.
5. Virtual labs need no calibration or maintenance.

These features suggest that using a virtual lab can be a great experience for the students. We will incorporate other software such as Puff, Antenna, and Transmission Lines, in our future courses in Microwave and Communication courses. Although virtual labs are really an excellent tool for teaching but they do not replace eye to eye contact of live instruction.

V. Acknowledgements

Several students have made significant contributions to this course, Mr. Bill Pappas, Eric Burrage, and Darrell Wirt deserve special thanks for their patience and hard works. The authors also would like to thank computer expert Mr. Craig Brookover for his assistance in extracting all different programs from different software and incorporating them in this document.

VI. Bibliography

1. ---------------Electronic WorkBench User’s Guide, Interactive Image Technology Ltd. 1995.
2. Roden, Martin, S., Micro-Cap II, An Electronic Circuit Analysis Progrm, California State University, The Benjamin/Cummings Publishing Company, Inc.,1988.

3. Keown, John, Pspice and Circuit Analysis, Southern College of Technology, Macmillan Publishing Company, 1993.
4. Yarbrough, Raymond, B., Electrical Engineering Reference Manual, Professional Publication Inc., 1990.

HANK JAVAN

Hank Javan received a B.S.E.E. in 1965, M.S.E.E. in 1970, and a D.Sc.E.E in 1980 from University of California and Washington University respectively. Dr. Javan is currently an Assistant Professor in the College of Engineering, University of Memphis. Dr. Javan has several years of academic and industrial experiences including University of North Carolina, Research Consultant for IBM, NASA, and Navy. He has published over twenty referee articles in IEEE, IJE, and RF Design. Dr. Javan is a Senior Member of IEEE, and a regular member of ASEE.

LEE DROUIN

Lee Drouin received a B.S.E.E. from Purdue University, M.S.E.E. from Naval Postgraduate School, and Ed.D. from Memphis State University. Dr. Drouin is currently an Assistant Professor in the College of Engineering and is the Coordinator of Electronics Engineering Technology Program. Dr. Drouin has over 29 years of electronics experiences with Civil Engineer Corps and Navy. Dr. Drouin has extensive academic experiences in the field of Industrial Electronics, Industrial Power & Motor Controls, and Electronic Communication.

SCOTT SOUTHALL

Scott Southall received B.S.E.T. in 1987 and a M.S. in 1991 from Memphis State University. Mr. Southall is currently an Assistant Professor of Engineering Technology at the University of Memphis. Prior to the University of Memphis, he was a Senior Project Manager with Productive systems, Inc. in Memphis, TN where he was responsible for industrial control systems development for automated material handling systems and machinery. He also developed and conducted a number of customized programmable controllers training seminars for industrial clients. Mr. Southall is a member of ASEE and The Society of Manufacturing Engineers (SME).