SIMULATION OF PHYSICS EXPERIMENT THROUGH MULTISIM COMPUTER PROGRAM

[1]V D PATEL, [2]K R PATEL, [3]K P PATEL

[1]Municipal Arts & Urban Bank Science College, Mehsana

[2]Sheth M. N. Science College, Patan

[3]R. R. Mehta College of Science & C. L. Parikh College of commerce, Palanpur

ABSTRACT

The manuscript contained simulation of common physics experiment, Halfwave, Fullwave and Bridge Rectifiers using MULTISIM (Evolution mode) program. In the current paper circuit models designed in the computer program and analyze their wave forms (I/P & O/P) within the simulation computer program. Students can perform experiments inside the model and take out reading by varying Load. Multimeters and Oscilloscope provided in the simulation model for analyze purpose. The result obtained from simulation is compatible with the actual experiments. It is concluding that students who go through this simulation before actual experiments do not need faculty attention during the laboratory work.

Keywords:*MULTISIM, Rectifiers, Waveforms, Simulation,*

INTRODUCTION

MULTISIM is special software for electronic circuit design and simulation. It can complete the whole process from circuit simulation designed to circuit diagram created, thereby providing a new and convenient approach for electronic system exploitation, electronic product and electronic system engineering[1]. Virtual Lab has the parameter adjustment convenient, easy implementation, high reliability advantages. Higher education in the use of virtual laboratory that can fundamentally solve the serious shortage of funding for experimental and practical problems [2]. Ankit Rana discuss how a fundamental Colpitts oscillatory circuit can be designed using MULTISIM and optimizing its performance with change in several dependent characteristics in oscillation[3]. Shahane and teamStudied and found that students who used simulation prior to conducting actual experiments performed better than the students who conducted the laboratory experiments without conducting simulation first[4]. The combination of the demonstration and the virtual electronic laboratory constitute a bridge between theoretical lessons and laboratory classes. The professor can use the experiments of the virtual laboratory in the classroom to improve student retention [5]. An astable multivibrator is constructed by Rajender Kumar & Team for given specification one of which is, it must have a duty cycle of 0.75. The workability of the proposed circuits is confirmed through PSpice, SIMetrix and Tina pro simulations tools and finally the experimental work is compared [6]. P.Sathya discuss the design and implementation of high performance closed loop Boost converter for solar powered HBLED lighting system with the simulation carried out in MULTISIM[7]. Sachin Sharma design and implement a variable gain amplifier for biomedical signal using MULTISIM. Variable gain provide the facility to increase or decrease the gain depending upon the acquiring signal and same amplifier hardware can be used for acquiring various biomedical signals[8].

MULTISIM& ULTIboard

NI MULTISIM is an electronic schematic capture and simulation program which is part of a suite of circuit design programs, along with NI Ultiboard. MULTISIM is one of the few circuit design programs to employ the original Berkeley SPICE based software simulation. MULTISIM was originally created by a company named Electronics Workbench, which is now a division of National Instruments. MULTISIM includes microcontroller simulation (formerly known as MultiMCU), as well as integrated import and export features to the Printed Circuit Board layout software in the suite, NI Ultiboard. MULTISIM is widely used in academia and industry for circuits education, electronic schematic design and SPICE simulation.

Semiconductor manufacturers provide SPICE models to facilitate simulation of their devices by customers. SPICE technology is commonly used to model discrete components such as BJTs, Operational Amplifiers, and MOSFETs. Simulation models are also increasingly being developed for integrated systems such as integrated power converters and switching controllers where SPICE simulation plays a critical role in the evaluation of system performance, power efficiency, conduction, switching losses, and overall thermal behavior. In addition engineers are using behavioral simulation models of high-density general purpose amplifiers are to determine performance parameters such as gain and phase margins, noise behavior, and harmonic distortion.NI MULTISIM includes over 26,000 devices validated by leading semiconductor manufacturers such as Analog Devices, NXP, Infineon, Texas Instruments, ON Semiconductor, Microchip, Maxim, and many others.

NI ULTIboard is an electronic Printed Circuit Board Layout program which is part of a suite of circuit design programs, along with NI MULTISIM. One of its major features is the Real Time Design Rule Check, a feature that was only offered on expensive work stations in the days when it was introduced. ULTIboard was originally created by a company named Ultimate Technology, which is now a subsidiary of National Instruments. Ultiboard includes a 3D PCB viewing mode, as well as integrated import and export features to the Schematic Capture and Simulation software in the suite, MULTISIM.

Several separate tasks comprise the overall process of circuit simulation inMULTISIM. These include: (1) schematic capture, (2) device modeling, (3) setting up and running the simulation, and (4) output analysis.

SIMULATION

A simulation is designed in the NI MULTISIM 10.0 evolution mode. Three different experiments designed in three different windows of MULTISIM. Identical (Similar to the actual experiment) components placed in the circuit utility board. Required junctions and wire was used to connect components with each other. Two separate multimeters (for Input & Output) and one Oscilloscope placed in the circuit design zone. The circuit model and waveforms are shown in the below figures and their interpretation are as under.

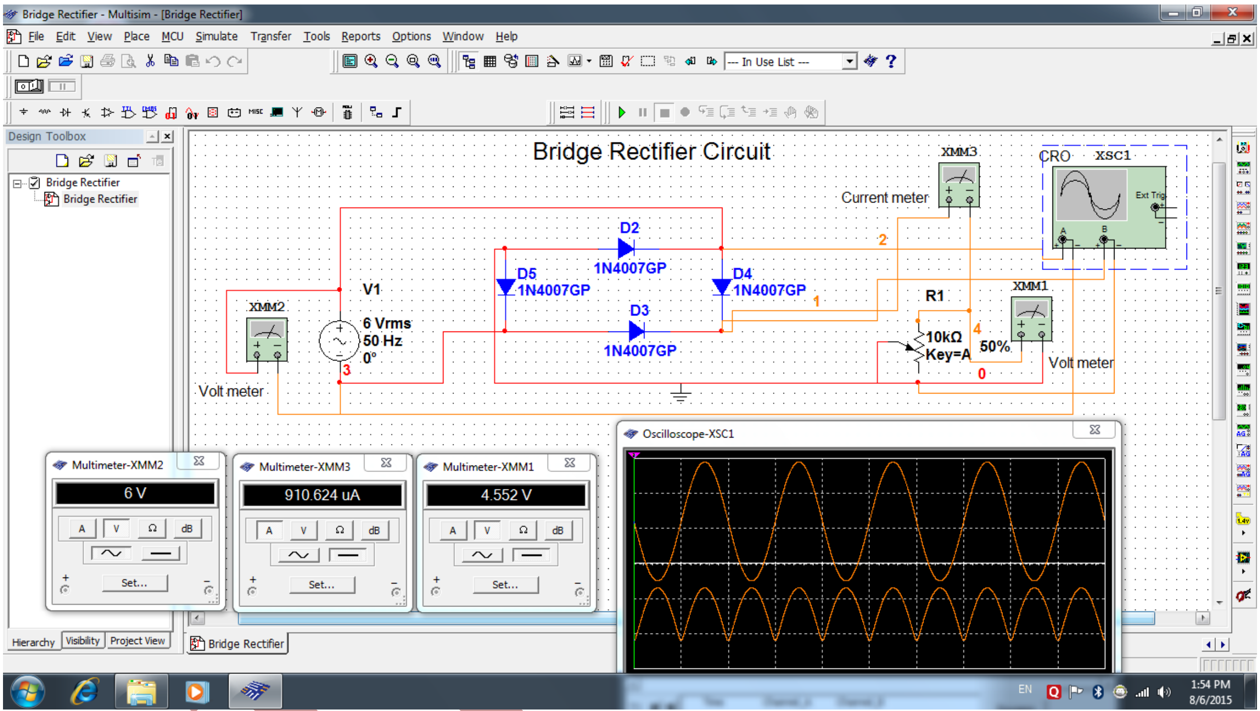


Figure – 1 Screenshot of Bridge Rectifiers in MULTISIM with required Components and Equipments

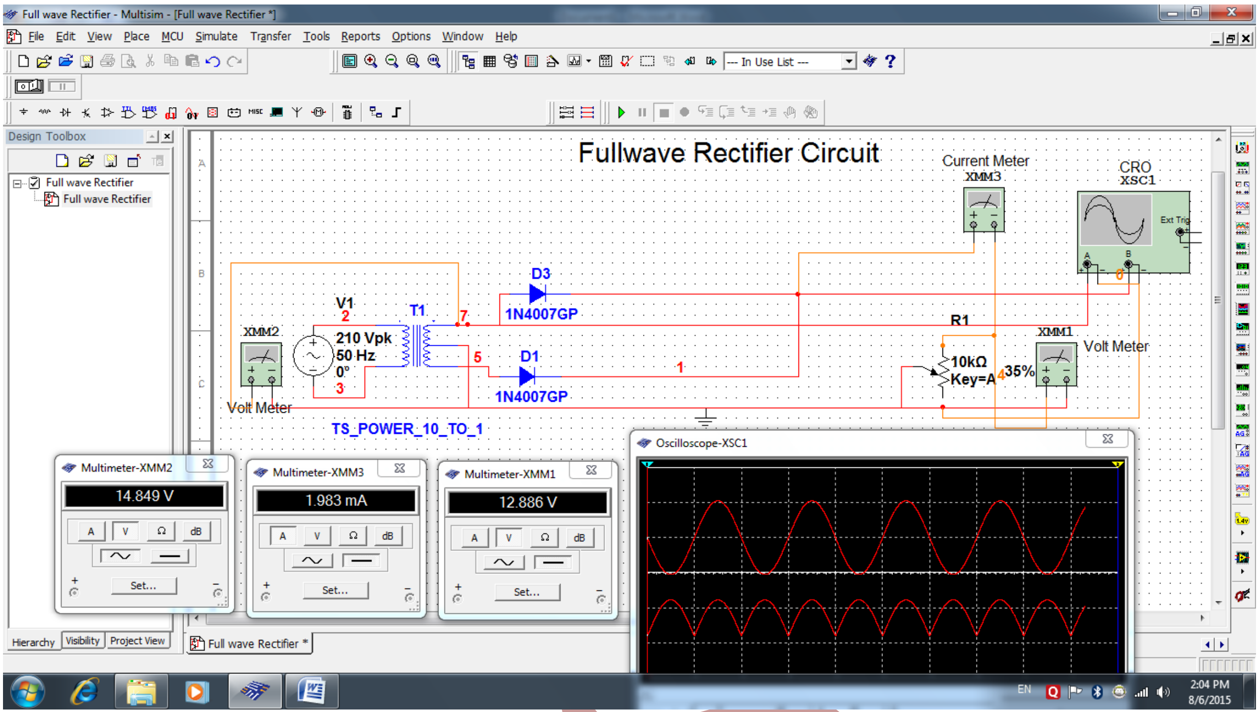


Figure – 2 Screenshot of Fullwave Rectifiers in MULTISIM with required Components and Equipments

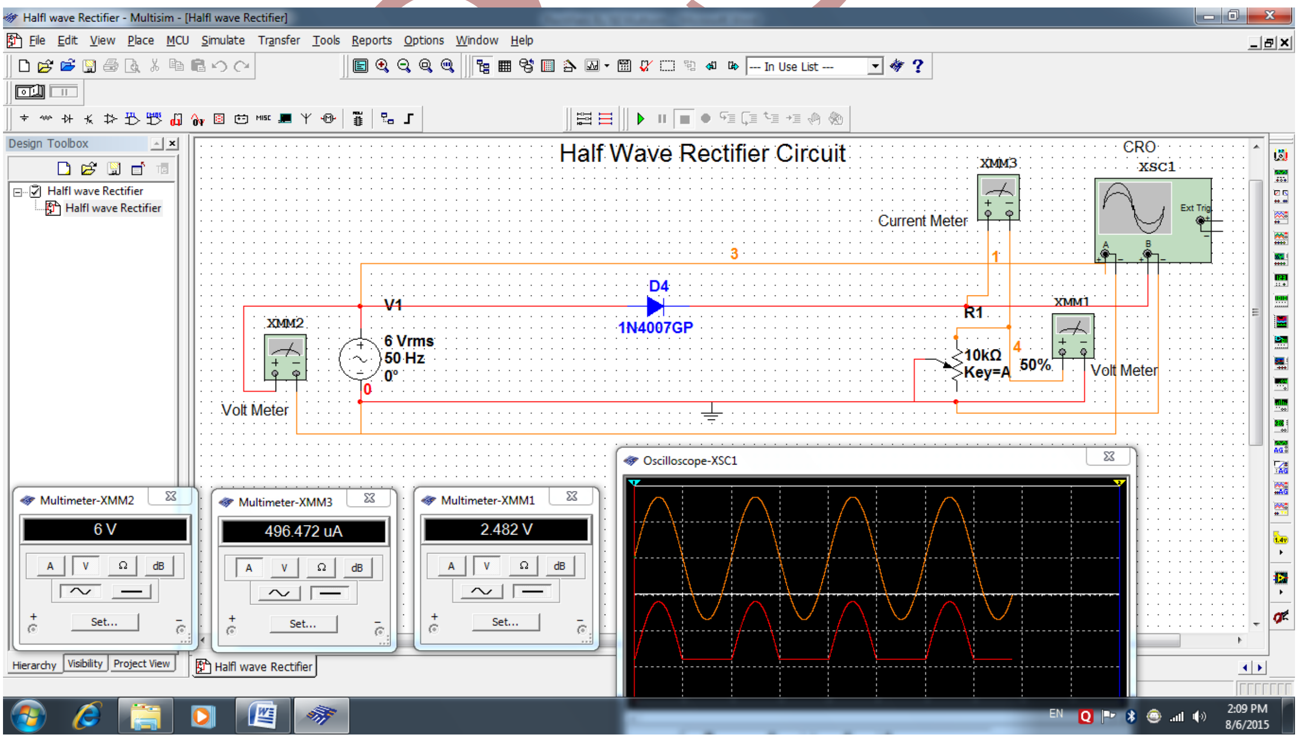


Figure – 3 Screenshot of Halfwave Rectifiers in MULTISIM with required Components and Equipments

EXPERIMENT

A reading of Voltage across load resistor and current pass through load resistor were taken by varying load resistor within the simulation model. The recorded data is displayed in below Table – 1.

Table – 1 Recorded Value of V & I through MULTISIM simulation model

R (KΩ)	Halfwave Rectifier I (µA)	Halfwave Rectifier Volt (V)	Fullwave Rectifier I (µA)	Fullwave Rectifier Volt (V)	Bridge Rectifier I (µA)	Bridge Rectifier Volt (V)
1	277	2.49	1434	12.90	513	4.61
2	311	2.49	1613	12.89	575	4.60
3	355	2.49	1841	12.89	655	4.59
4	414	2.48	2148	12.88	762	4.57
5	496	2.48	2575	12.87	910	4.55
6	618	2.47	3214	12.85	1132	4.52
7	822	2.46	4281	12.83	1498	4.49
8	1227	2.45	6406	12.81	2223	4.44
9	2435	2.43	12773	12.77	4369	4.36

By making plot of V v/s I, once students can calculate voltage regulation of concern rectifier. A MS Excel orORIGIN program can be used to prepare a plot of V → I for all the rectifier circuit. A plots obtained from MS Excel is illustrated in Figure 4 to Figure 6.

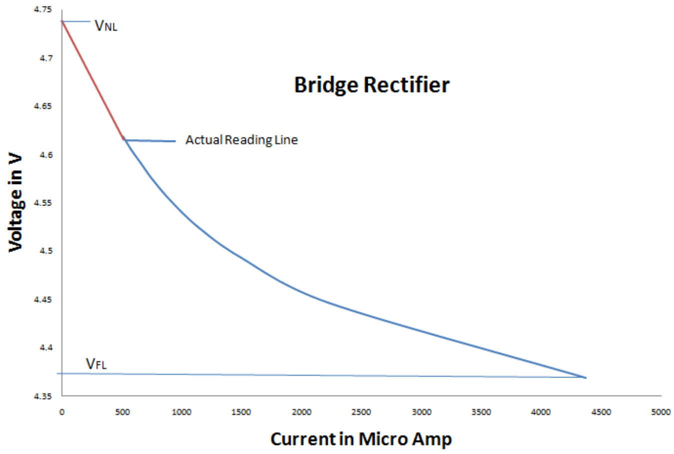


Figure – 4 V→I Plot of Bridge Rectifier

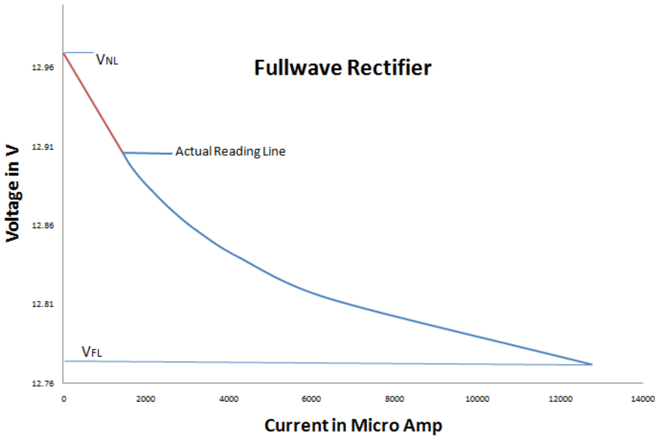


Figure – 5 V→I Plot of Fullwave Rectifier

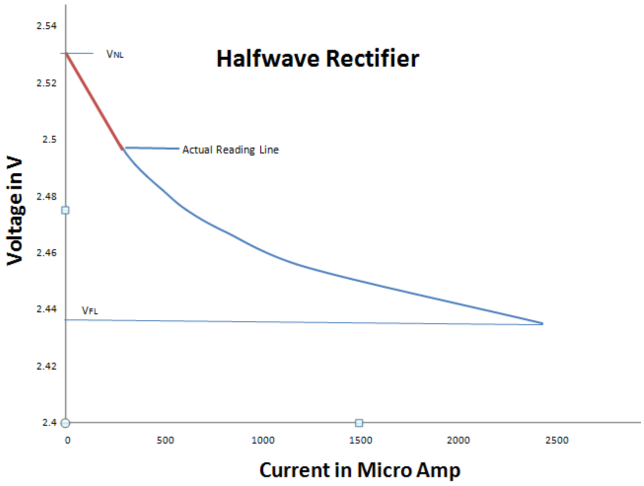


Figure – 6 V→I Plot of Halfwave Rectifier

Percentage of Regulation:

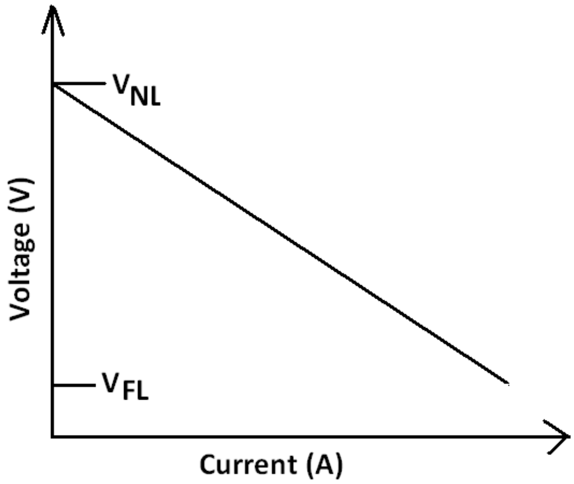
It is a measure of the variation of DC output voltage for variations in the load.

% of regulation = $\frac{V_{NL} - V_{FL}}{V_{FL}}$ X 100 %

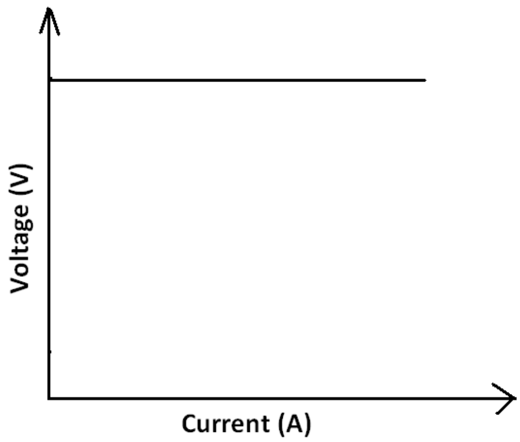
V_{NL} = DC voltage across load resistance, when minimum current flows it i.e. $R_L = \infty$ Ohm

V_{FL} = DC voltage across load resistance, when maximum current flows i.e. R_L = 0 Ohm

For an ideal half-wave rectifier, the percentage regulation is 0 percent.



Experimental Plot from Simulation



Ideal Plot of rectification

Figure – 7 V v/s I plot of rectifier

CONCLUSION

Groups of students were performed experiments using simulation model and understand all the required aspect of experiment. After that they bring to the laboratory for actual experiment. Significantly, they performed actual experiment and found voltage regulation (%) without attention of faculty. It shows that students can better understand visualization of experiments compared to traditional teaching method.

FUTURE PROSPECT

Lots of Physics experiments related to electronics background can be visualized by simulation model using MULTISIM. This will help teacher as well as students for better understanding of experiments and it all related theories.

REFERENCE

1. ZHANG Jing,LI Xin-guang, MULTISIM Based Schematic Design and Simulation, Computer Simulation, 05, 2005.
2. Yogendra Babarao Gandole, LabVIEW-based virtual model of the building Electronics Experiments, International Journal of Science & Emerging Technology, 2(1), 2011, 26-31
3. Ankit Rana, Colpitts Oscillator: Design and Performance Optimization, J Electr Electron Syst, 3(3), 2014, 132-140.
4. G. S. Shahane and P. A. Lohar, Role of Simulation Software in Enhancing The Quality of Electronics Teaching, Contemporary Research in India, 2(2), 2014, 11-13.
5. Yogendra B.Gandole, Virtual Instrumentation as an Effective Enhancement to Laboratory Experiment, International Journal of Computer Science and Information Technologies, 2 (6), 2011, 2728-2733.
6. Rajender Kumar, Sandeep Dahiya & Krishan Kumar, Design and Implementation of Astable Multivibrator for Different Applications in Communication System, International Journal of Advances in Electrical and Electronics Engineering, 2(2), 274-282.
7. P.Sathya , Dr.R.Natarajan, Design and Implementation of 12V/24V Closed loop Boost Converter for Solar Powered LED Lighting System, International Journal of Engineering and Technology, 5(1), 2013,254-264.
8. Sachin Sharma , Gaurav Kumar , Dipak Kumar Mishra , Debasis Mohapatra, Design and Implementation of a Variable Gain Amplifier for Biomedical Signal Acquisition, International Journal of Advanced Research in Computer Science and Software Engineering, 2(2), 2012, 193-198.