Performance Comparison of Astable Multivibrator Circuit Using Various Circuit Designing SPICE Softwares

A. V. Mancharkar
Head, Dept. of Physics, New Arts, Commerce and Science College, Parner 414302, Maharashtra, India

Abstract

This paper addresses performance of astable multivibrator circuit using different spice softwares. Traditionally, electronic circuit design was verified by building prototypes, subjecting the circuit to various stimuli and then measuring its response using appropriate laboratory equipments. Prototype building is somewhat time consuming, but produces practical experience from which we judge the manufacturability of the design. Computer programs that simulate the performance of an electronic circuit provide a simple cost effective means of confirming the intended operation prior to circuit construction and of verifying new ideas that could lead to improve the circuit performance. Such computer programs are revolutionized the electronic industry which provides the resent products in the service of human being. In this paper I have performed the transient analysis of the same circuit in different spice softwares. The accuracy of the circuit performance and reliability is reported by using these test tools. The results are explored with browsing output data facility.
KEYWORDS: Astable multivibrator, Transient analysis, Simulation.

I. INTRODUCTION

The evolution of process control has seen the infusion of electronics technology in to almost every facet because of low cost, reliability, and ease of interface [1]. The electronic industry is getting progressively more and more efficiently at developing new products in wide ranges and verity of circuits in service of human being. We also saw the more and more products coming into the market with shorter time and product lives and sometimes float at times [2]. Hence low cost circuit design, with an accurate, linear and faster testing technique is addressed. A verity of electronic components is commercially available which plays an important role in design and development of accurate circuit performance and optimum reliability. Here the product has been practically designed first and then tested for its intended results, but any degradation in the output signal is found circuit parameters and values have to be replaced and circuit is re-tested for improved performance and low drift in specified parameters over the operating range. To complete one such process involves large amount of time, higher cost and accurate component values this critical problem of product design and testing is simplified by the use of spice software simulations. This allows user to design test and perform various analysis to optimize the accuracy, circuit performance and its reliability before any product is actually made and transferred to the costumer for their use [3]. Comparative study of various spice softwares is made by using the transient analysis are discussed in this paper.

II. SIMULATION CONCEPT

Electronic simulation of circuit function is now a common practice in the design both of individual circuits and complete systems. The more of the circuit a designer can simulate, the faster the circuit can get in to production and hence to market. PSPICE (PC Version, Simulation Program with Integrated Circuit Emphasis) has become a common tool for analogue simulation and widely used, even for some mixed mode circuit designs [4]. Dr. Lwrence Negal at University of California, Berkeley, originally developed PSPICE; it is a general purpose circuit simulator program that simulates electronic circuits. It performs various types of analysis of electronic circuits. SPICE software's contains models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is versatile program and is widely used both in industries and universities [5]. Sophisticated instrumentation systems rely heavily on the application of wide range of electronic circuits. PSPICE allow the designer to construct entirely new circuit using hard wires, components and PCBs. The circuit performance and its reliability in any circuit to minimize failure can be tested. To meet the required standards of a circuit and hence quality instrument, the circuit analysis is performed. In case of any failure or problems observed, one can easily redesign it by modifying the very same circuit in few minute using highly sophisticated simulation tools [6]. With adequate number of design and redesign interactions on a computer platform where it consume only a small amount of time and no material. The design can be made robust. The simulated circuit can then be subjected to different analysis i.e. actual tests. The performance and reliability of circuit and instrument definitely shows results of up most levels. Thus it is the faster and low cost cumbersome process. The different steps involved in simulating a circuit with capture are as follows. 1. Circuit creation with capture--- create new analog, mixed AD project. Place circuit parts. Connect the parts. Specify values and names. 2. Specify types of simulation--- Create a simulation profile. Select type of analysis (Basic, Dc sweep, Ac sweep, Transient, Fourier, Temperature, Parametric etc.). Run the simulation program. 3. View the results--- Add trace to the probe window. Use cursers to analyze waveforms. Check the output file, if needed. Save or print the results [7]. The design centre software package has three major interactive programs; Schematics, Spice and probe. Schematics are a powerful program that lets you build circuits by drawing in a window on the screen. Spice analyzes the circuit created by schematics and generates voltage and current solutions. Probe is a graphic postprocessor that allows you to display plots of parameter such as voltage, current, impedance and power [8].

III. THEORY OF SQUARE WAVE GENERATOR CIRCUIT

The astable multivibrator derives its name because there are no stable states in this circuit. As shown in Fig 1 a circuit diagram of astable multivibrator is shown where in the IC 555 is used. Assume that the capacitor is initially discharged and Q is high. The capacitor C_2 will charge through R_1 and R_2 and the voltage Vc across it will rise exponentially towards Vcc. However as soon as this voltage reaches Vu_T = (2/3)Vcc, the comparator output goes high, reseating the flip flop. Q' becomes high and the transistor conducts and the capacitor discharges through R_2 lowering the voltage Vc. When the capacitor voltage becomes V_{LT} = (1/3)Vcc, the output of the comparator C_2 becomes high and the flip flop is again SET making the transistor OFF and again charging the capacitor through R_1 and R_2. The cycle repeats continuously and the pulse waveform is obtained at the output.

Assuming that t = o is the instant when charging of C begins, we can write the voltage across the capacitor during charging as

$$V_c(t) = V_{cc} – (V_{cc} – V_{LT})e^{-t/(R1 + R2)C}$$

And at t = T_H, $V_c(t) = (2/3)V_{cc} = V_{UT}$ and $V_{LT} = (1/3)V_{cc}$

Therefore

$\frac{2}{3}V_{cc} = V_{cc} – (V_{cc} - \frac{1}{3}V_{cc})\, e^{T_H / (R1 + R2)C}$

Where $T_H = (R1 + R2)C \ln 2 = 0.69(R1 + R2)C$

We see from the figure that Vo is low during T_L therefore, the discharge voltages across the capacitor can be written as

$V_c(t) = V_{UT}\, e^{-t / R2C}$

(t = o is beginning of discharging of C)

At t = T_L

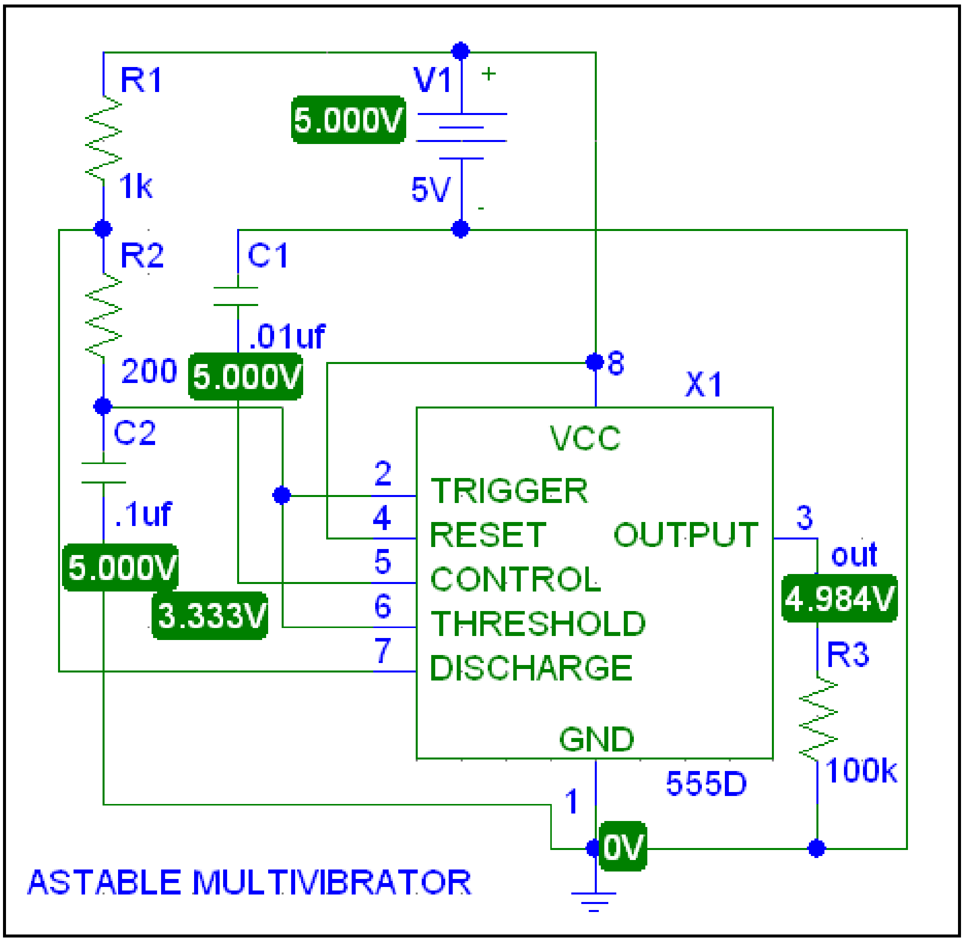

Figure 1 Astable multivibrator

$V_c(t) = \frac{1}{3}V_{cc} = V_{LT}$, Hence $\frac{1}{3}V_{cc} = \frac{2}{3}V_{cc}\, e^{-TL/R2C}$

Or T_L = R2C ln2 0.69R2C

The total time period, T = $T_H + T_L$, T = 0.69(R1 + 2R2)C

$f = \frac{1}{T} = \frac{1.443}{(R1+2R2)C}$ The % duty cycle = $\frac{TH}{T}$ x 100

In this circuit the duty cycle is always be greater than 50%. If R1 << R2, it approaches 50%.

IV. GRAPHS OF ASTABLE MULTIVIBRATOR OUTPUT IN DIFFERENT SOFTWARE:

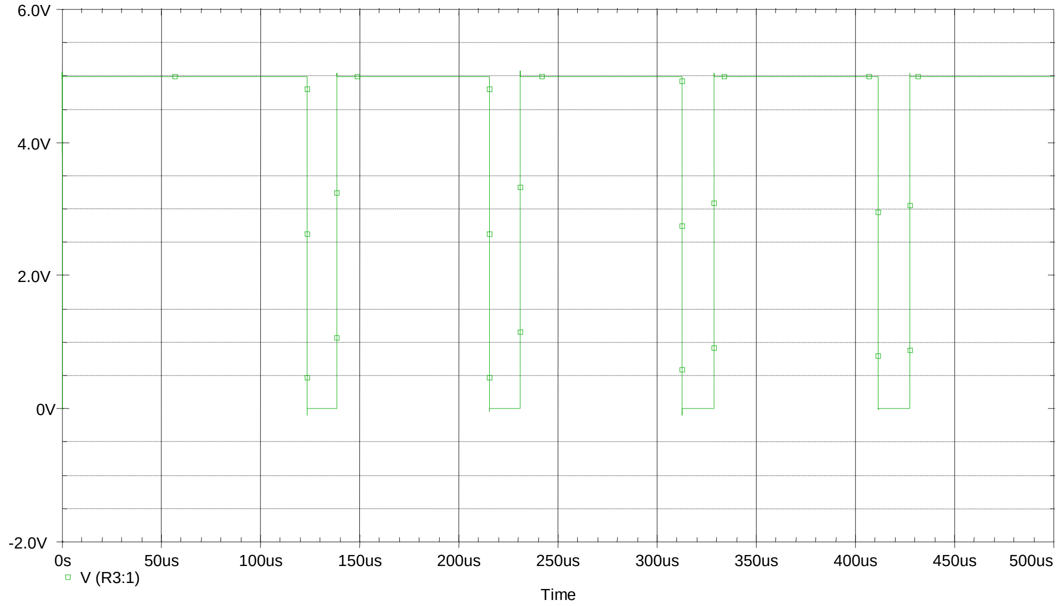

Figure 2 Output Voltage of astable multivibrator using PSpice

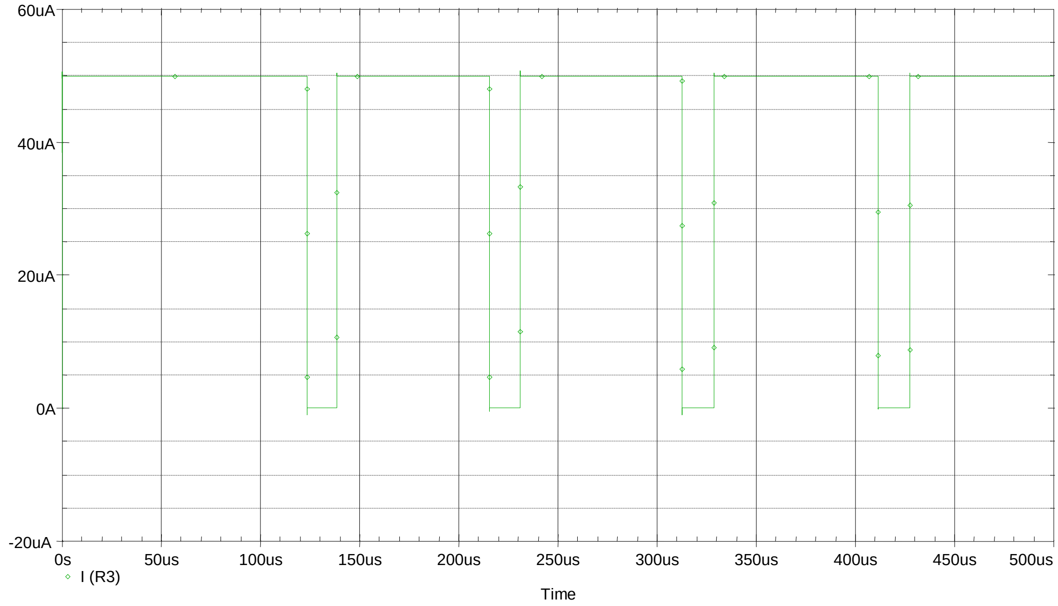

Figure 3 Output current of astable multivibrator using PSpice

- Output starts 3.8364µV to 4.9836 V.
- Rise time and fall time in this software is exactly equal to 598µs.
- The pulse width depends upon the values of R_1 and R_2.
- Except first all the pulses are equally spaced.
- Current and potential are in phase.
- We get the maximum current up to 49.936µA.

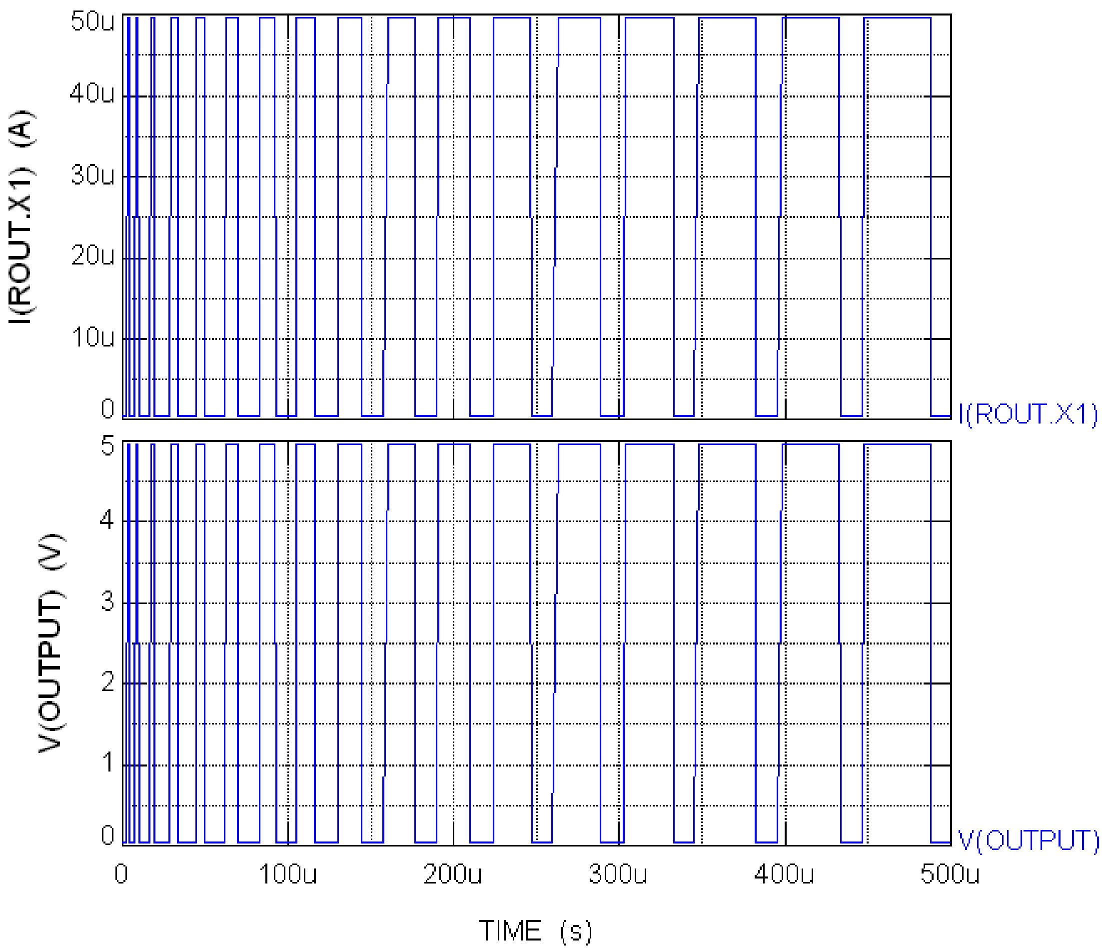

Figure 4 Output current and potential of astable multivibrator using Top Spice

- Output starts -38.976 µV to 4.962 V.
- We observe rise time and the fall time are exactly equal to zero sec.
- We cannot get the perfect pulse; pulse width goes on increasing as the time increases.
- Current and potential are in phase.
- We get the maximum current up to 49.62µA.
- Rise time and fall time; initially it is less but as the time increases it also increases.

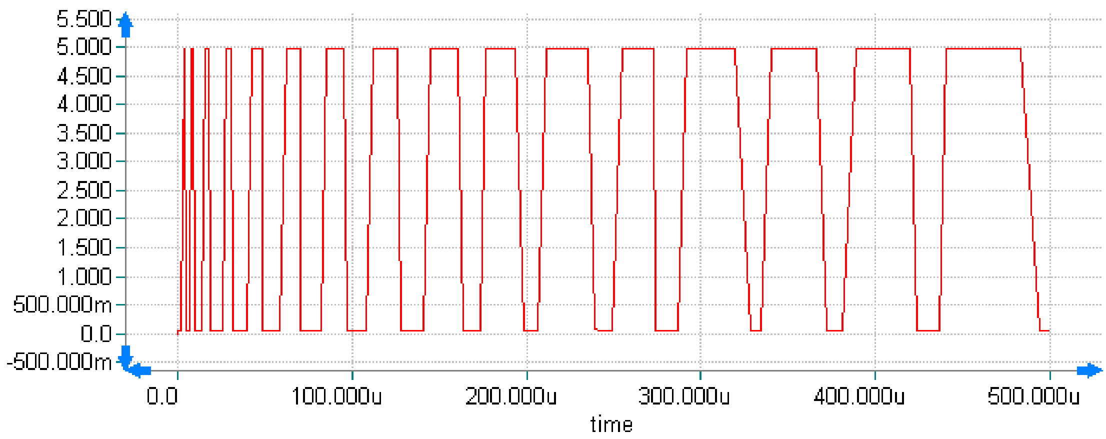

Figure 5 Output potential of astable multivibrator using B2 Spice

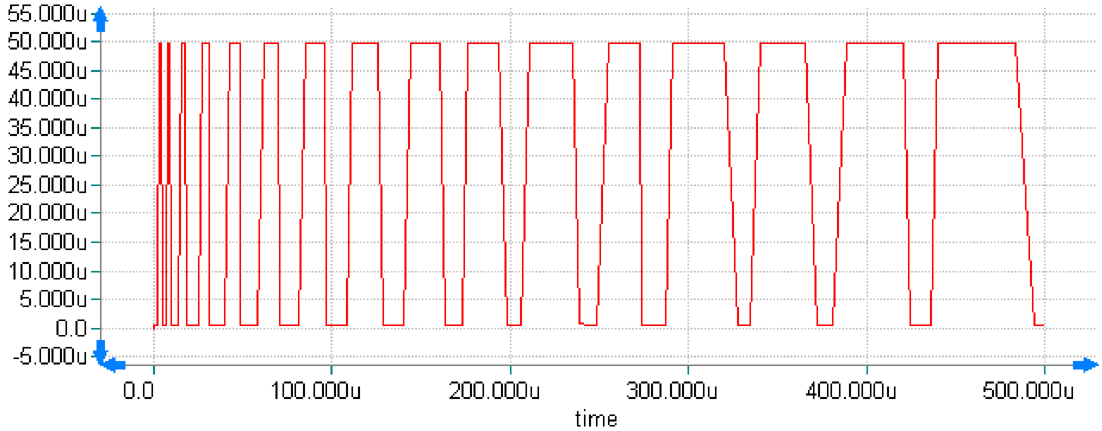

Figure 6 Output current of astable multivibrator using B2 Spice

- Lower level of the output is 0.045 V to 0.049V.
- Higher level of the output is 4.962 V.
- Rise time and fall time; initially it is less but as the time increases it also increases.
- Initially the output frequency is maximum and decreases as the time increases
- We cannot get the perfect pulse; pulse width goes on increasing as the time increases.
- Current and potential are in phase.
- We get the maximum current up to 50 µA.
- We get the minimum current up to .05 µA.

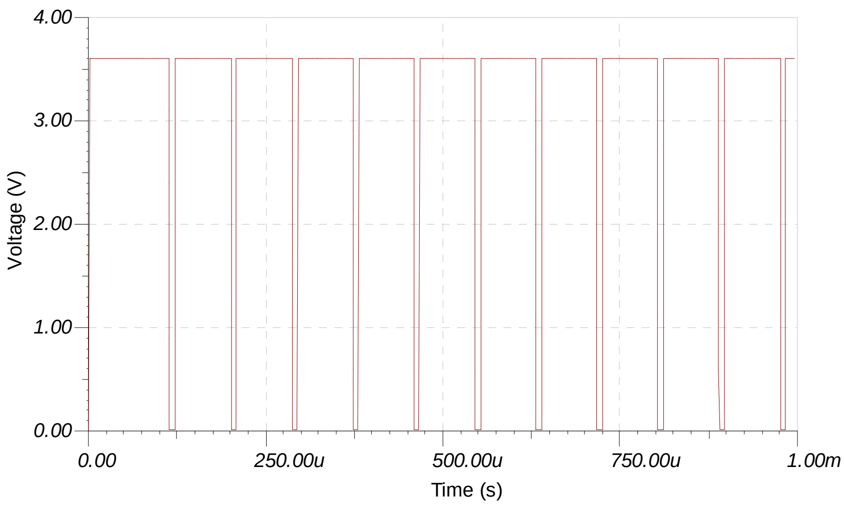

Figure 7 Output current of astable multivibrator using Tina

- Output voltage is 0 V to 3.6 V
- In this software rise time and the fall time are exactly equal to zero second.
- We get expected output but output that changes with the change as the values of resistors R_1 and R_2 changes.
- First maxima take more time to appear.
- In this software, we cannot get the current response simultaneously in the graph window in transient analysis but in AC table analysis, we get current as well as the potential values of potential at any point of the circuit.

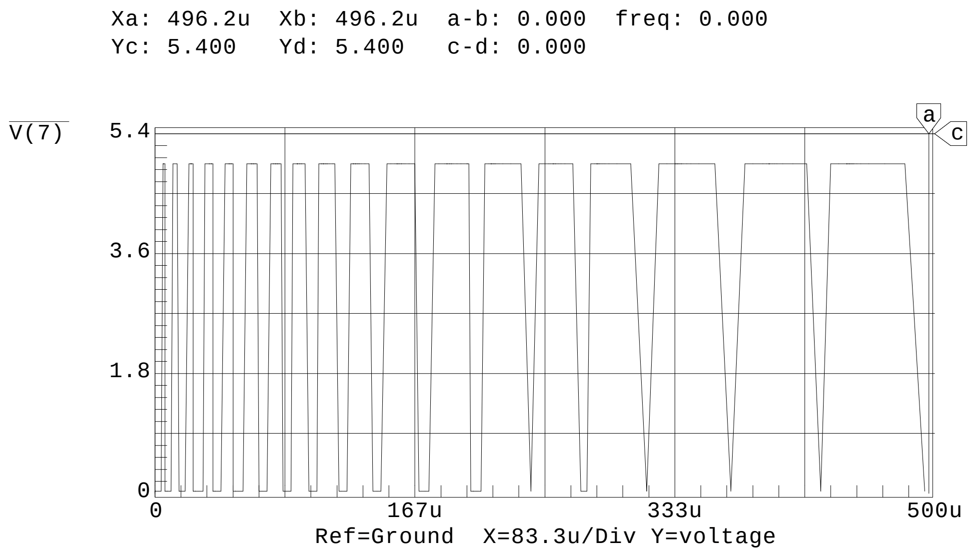

Figure 8 Output current of astable multivibrator using Circuit Maker

- The maximum output voltage at the peak is 4.950V.
- In this software rise time and the fall time increases as the time increases.
- Initially the output frequency is maximum and decreases as the time increases
- We cannot get expected output. However the output changes with the values of resistors R_1 and R_2.
- First peak take less time.
- In this software, we cannot get the current response simultaneously in the graph window in transient analysis. However, in multimeter we get the current as well as the potential value of any point in the circuit.
- The pulse starts from 0 V.

Table 1: Data for simulated astable multivibrator circuit using Pspice, TopSpice, B2 Spice, Tina and Circuit Maker.

Software	Start Time	Start Pot.	Positive Potential	Negative Potential
Pspice	100	-13.884	0.005	4.9836
TopSpice	0	-38.98	0.049	4.962
B2 Spice	0	0	0.045	4.962
Tina	0	0	0	3.6
Circuit m.	0	45	0	4.95

V. RESULTS AND DISCUSSION

This paper reports that the results obtained after simulating the square wave generator using astable multivibrator (same circuit) using different spice software test tools i.e. Pspice, B2 Spice, Top Spice, TINA and Circuit Maker. We observed the amplitudes of the square waves are different in different software’s are given in Table 1. The results obtained after performing simulation and theoretical results are not matching exactly, these are slightly different. Since Spice is industry standard for electronic circuits simulations, it is used extensively for the design of other circuits such as pressure, flow, displacement, force vibration etc [16]. Here square wave generator circuit is used as a example for the comparative study of different software's in circuit analysis. Also these simulations studies are uniformly may be applied for testing the circuit performance and reliability under changing the physical conditions under which they are operating before actual circuit or the product is made [17]. Hence it avoids the complex, cumbersome, cyclic testing procedure for a newly developed product.

VII. REFERENCES

[1] Murty D V S, Tranducers and instrumentations, Printice Hall of India, New Delhi: 1995.

[2] Ward A E, Angus A S, Electronic Product Design, Chapman & Hall Oxford: 1996.

[3] Walunj A K, Mancharkar A V and Shaligram A D, PSpice Simulation for performance and reliability testing of sensor signal conditioning circuits, J. Instrument. Soc. India 30 (20): 65-74,2001

[4] Martin O' Hara, Modeling board- level DC-DC convertors in SPICE (Electronic Product Design), July: 1998.

[5] Tunenga Paul W, SPICE: A guide to circuit simulation and analysis using PSPICE, Englewood Cliffs N. J. USA, and Prentice Hall: 1998.

[6] Rashid M. H. Microelectronic Circuit Analysis and Design, PWS Publishing Co: 1999.

[7] University of Pennsylvania, Department of Electrical & System Engineering, PSPICE – A brief primer. Jan Vander Spiegel @ 2006 jan-at-seusepenn.cdu.

[8] Rangan C S, Sharma G R, Mani V S V, Instrumentation Devices & System, Tata McGraw-Hill Ltd, New Delhi : 1997.

[9] L K. Maheshwari M. M. S. Anand, Laboratory Experiments and PSPICE simulations in Analog Electronics Prentice – Hall of India Private limited New Delhi – 110 001, 2007.

[10] Mumhad H. Rashid, Introduction to PSpice using ORCAD for Circuits and Electronics, Prentice – Hall of India, Private Limited New Delhi – 110 001, 2006.

[11] PSpice Schematics, Evaluation Version 9.1, Cadence Design Systems, www.Cadance.com.

[12] B2 Spice A/D 5.2.3 Copyright @ 2004-2007, BeigeBag Software, www.beigebag.com, info @ beigebag.com.

[13] TopSPICE/Win32 Schematic Editor Demo version 7.16c Copyright@ 1991-2008, by panzer Development. http://www.penzar.com.

[14] TINATM for windows, the Complete Electronics Lab, Version 6.02006DT.

[15] CircuitMaker@ Student Version6.2c, Portal Technology, inc. 5252N Edgewood Dr Ste175, ProvoUT 84604 USA.

[16] Mancharkar A V and S H Behere, PSpice simulations for Performance Testing of signal Conditioning Circuit for Temperature Sensor, Technical College – Bourgas, March, 2000.

[17] Walunj A K, Mancharkar A V and Shaligram A D, Temperature Sensors SCC PSpice performance and reliability testing due to resistive tolerances and effects, Proceeding of 7th NSPTS, University of Pune, 14-16Feb, 2000, pp50.1-50.6.