

International E-Conference on Life Sciences, Technology and Management In Association with International Journal of Scientific Research in Science and Technology Volume 9 | Issue 9 | Print ISSN: 2395-6011 | Online ISSN: 2395-602X (www.ijsrst.com)

# Study of Astable Multivibrator Using B2 Spice, TINA and Circuit Maker Software's

Dr. Sanjay K. Tupe

Assistant Professor, Department of Physics, Kalikadevi Arts, Commerce & science College, Shirur (Ka), Dist. Beed 413249, Maharashtra, India

# ABSTRACT

This paper addresses the performance of AstableMultivibratorusing B2 Spice, TINA and Circuit Makerelectronic circuit simulation software's. Traditionally electronic circuit design was verified by building prototypes, subjecting the circuit to the various stimuli and then measuring its response using appropriate laboratory equipment's. 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 verifying new ideas that could led to improve the circuit performance.

Key Words: - RC Coupled amplifier, Amplification, Transient Analysis, Smoke Analysis Simulation.

# I. INTRODUCTION

The evolution of electronics technology almost in to every facet because of low cost, reliability and ease of interface [1]. The electronic industry is getting progressively more and more efficiently more at new products in wide range and verity of circuits in service of human being. We also saw the more and more products coming in to the market in shorter time [2]. Hence low-cost circuit design, with an accurate, linear and faster testing techniques are addressed. A verity of electronic components PSpice commercially available which plays an important role in design development of accurate circuit design performance and optimum reliability [3].

# II. SIMULATION

Electronic simulation of circuit function is now a common practice in the design of individual circuit and the complete systems. The most of the circuit designer can simulate, and design the circuit and develop it as early as they can and hence in market [4]. Spice software models for common circuit elements, active as well as passive, and it is capable of simulating most electronic circuits. It is versatile programmed and is widely used both in Industries and Universities. [5]. The circuit performance and its reliability in any circuits for to



minimize the failure can be tested. To meet the required standard of the circuits and hence quality instruments, the circuit analysis is performed. In case of any failure or problem on can easily redesign it by modifying the very same circuit in a few minutes using highly sophisticated simulation tools [6].

The role of spice software's is very vital in all fields of engineering and industries for the design and built the electronic circuits. Today many groups around the world are investigating advanced software capable of responding a wide verity of components. Recent years have witnessed the excellent progress in the field of spice software. These improve the ability of users to integrate different types of electronic circuits in to their systems or applications. The spice software would have more capability to design and built electronic circuits in wider range of applications.



#### **III. ASTABLE MULTIVIBRATOR**

Fig 1 Circuit diagram for Astablemultivibrator

The 555 connection as an astable multivibrator is shown in figure 1. Assume that the capacitor is initially discharged and Q is high. The capacitor C will charge through R1 and R2 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 R2 lowering the voltage Vc. When the capacitor voltage becomes  $V_{LT} = (1/3)Vcc$ , the output of the comparator C2 becomes HIGH and the flip flop is again SET making the transistor OFF and again charging the capacitor through R1 and R2. 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

 $Vc(t) = Vcc - (Vcc - V_{LT})e^{-t/(R1 + R2)C}$ And at  $t = T_H$  $Vc(t) = (2/3)Vcc = V_{UT}$  and  $V_{LT} = (1/3)Vcc$ Therefore  $\frac{2}{3}$  Vcc =Vcc - (Vcc -  $\frac{1}{3}$ Vcc)  $e^{T_{H}/(R1 + R2)C}$  $T_{\rm H} = (R1 + R2)C \ln 2 = 0.69(R1 + R2)C$ We see from the figure that Vo is low during TL therefore, the discharge voltages across the capacitor can be written as  $Vc(t) = VUT e^{-t/R2C}$ (t = o is beginning of discharging of C) $At \quad t=T{\rm L}$  $Vc(t) = \frac{1}{3}Vcc = V_{LT}$ Hence  $\frac{1}{3}\text{Vcc} = \frac{2}{3}\text{Vcc} \text{ e}^{-\text{TL/R2C}}$ Or  $T_{L} = R2C \ln 2 \ 0.69R2C$ 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 is % duty cycle =  $\frac{\text{TH}}{T} \ge 100$ In this circuit the duty cycle is always be greater than 50%. If R1 << R2, it approaches 50%.



The Following Graphs Shows the Output of AstableMultivibrator in Different Software:

Fig 3 graph of square wave generator by TINA& Circuit Maker

### **IV. OBSERVATIONS**

- A. In B2spice Software: 1. Lower level of the output is 0.045V to 0.049V.2. Higher level of the output is 4.962 V.3. Rise time and fall time initially it is less but as the time increases it also increases.4.Initially the output frequency is maximum and decreases as the time increases. 5.We cannot get the perfect pulse; pulse width goes on increasing as the time increases.6.Current and potential are in phase.7.We get the maximum current up to 50µA.8.We get the minimum current up to .05µA.
- **B.** In TINAspice Software: 1. Output voltage is 0 V to 3.6 V. 2. In this software rise tie and the fall time are exactly equal to zero second. **3**. We get expected output but output that changes with the change as the values of resistor R1 and R2 changes. **4**. First maxima take more time. **5**. In this software, we cannot get the current response simultaneously in the graph window in transient analysis but in AC table analysis, we get the current the current as well as the potential value of any point of the circuit.
- C. In Circuit Maker Software: 1. The maximum output voltage at the peak is 4.950V.2. In this software rise time and the fall time are increases as the time increase.3. Initially the output frequency is maximum and decreases as the time increases. 4. We cannot get expected output. However, output which changes with the change with the values of resistor R1 and R2. 5. First peak take less time.6. 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 of the circuit. 7. The pulse starts from 0 V.

#### V. CONCLUSION

In above software's only in B2 Spice we observed simultaneously the potential & current curves. In Top spice we observe up to 500  $\mu$ s the output frequency is stable from start to end but in case of B2 spice& Circuit Maker it decreases 500  $\mu$ s.

#### VI. REFERENCES

- [1]. PSpice Schematics, Evaluation Version9.1 www.cadence.com
- [2]. TopSPICE/Win32 version 7.16c by penzar development. www.penzar.com
- [3]. B2 Spice A/D 5.2.3, Beige Bag Software www.beigebage.com info@ beigebage.com
- [4]. TINATM for Windows, The Complete Electronics Lab version 6.00.008SFS.
- [5]. CircuitMaker V6.2C Protel Technology, Inc. 5252N Edgewood Dr Ste175 Provo UT84604 USA.
- [6]. Muhammad H. Rashid, Introduction to PSpice Using OrCAD for circuits and electronics. Prentice hall of India private limited, New Delhi 110 001, 2006.
- [7]. Maheshwari L. K. and Anand M.M.S., Laboratory Experiments and PSPICE Simulation's in Analog Electronics., Prentice Hall of India Pvt. Ltd., New Delhi, (2007).
- [8]. http://www.ecircuitcenter.com/About SPICE.htm.
- [9]. Operational Amplifier G. B. Clayton
- [10]. Operational Amplifier and Linear Integrated Circuits R. A. Gaikwad
- [11]. Principles of Electronics V. K. Mehta
- [12]. Electronic Principles A. P. Malvino
- [13]. Comparative study of various circuit simulation software's", A.V.Mancharkar, S.K. Tupe, A.S. Jadhav, J.B. Patwardhan and S.H. Behere. 95th Indian Science Congress, Jan 2008, Andhra University, Visakhapatnam.
- [14]. "Comparative study of various circuit simulation software's by using triangular waveform generator circuit", S.K. Tupe, B. Deshmukh, and A.V. Mancharkar. Presented 97th Indian Science Congress, Jan 2010, to be held at Tiruanantpurum.