REVIEW OF RESEARCH
ISSN: 2249-894X
IMPACT FACTOR : 5.7631(UIF)
UGC APPROVED JOURNAL NO. 48514
VOLUME - 8 | ISSUE - 9 | JUNE - 2019

COMPARATIVE STUDY OF VARIOUS CIRCUIT DESIGNING SPICE SOFTWARE'S – A SIMULATION STUDY PERFORMED USING TRIANGULAR WAVE GENERATOR

Sanjay K. Tupe
Assistant Professor, Department of Physics, Kalika Devi Arts, Commerce & Science College, Shirur (k.) Dist.- Beed.

ABSTRACT:
This paper addresses performance of triangular wave generator using different spice software's. Traditionally, electronic circuit design was verified by building prototypes, subjecting the circuit to 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 of verifying new ideas that could lead to improve the circuit performance.

KEYWORDS: *Triangular wave generator, Transient 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 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 [3]. Comparative study of various spice software's by using triangular wave generator is made by using the transient mode of analysis is 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 [4]. 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]. 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 [7]. The design center 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 Triangular Wave Generator Circuit:

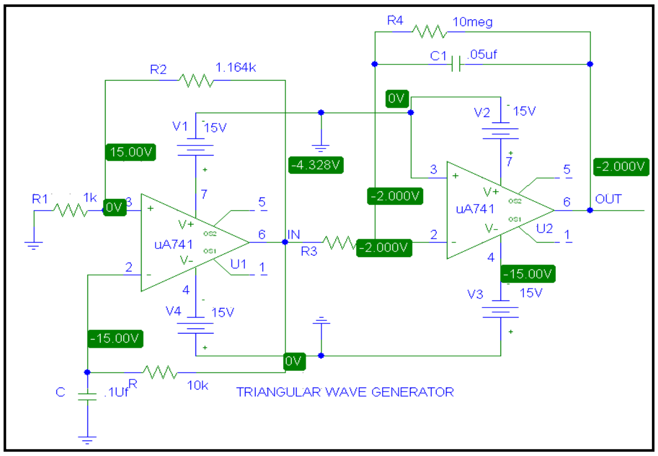

Figure 1 Tringular wave form generator circuit

The simplest method of forming a triangular waveform generator is to integrate the square waveform. By connecting an integrator to a square waveform generator, a triangular waveform can be generated as shown in figure 1

In this circuit the astable multivibrator output is connected to an integrator. The frequencies of the square wave and the triangular waveforms are identical. R_3C_1 is chosen equal to T and the R_4 10MΩ resistor shunts the capacitor C_1 to obtain a triangular waveform.

The operation can be described by assuming that the output of op-amp A_1 is at $+V_{sat}$. this will force a current $+V_{sat}$ / R through the integrator capacitor C causing the integrator output to decrease linearly. This will continue until the non-inverting terminal voltage of A_1 crosses zero and becomes negative. At this instant, $V_0 = -V_{sat}$ and the current through R and C will reverse direction. The integrator output starts increasing. This continues till the voltage V^+ crosses zero and becomes positive, thereby making $V_0' = V_{sat.}$ the cycle thereafter repeat itself.

Let V_0' $+V_{sat}$ at $t = t_o$. The current flowing in to the integrator is constant, given by $I^+ = +V_{sat} / R$ and the integrator output is-

$$V_0(t) = V_0(t_o) - \frac{1}{C}\int_{t0}^{t} Idt$$

$$V_0(t) = V_0(t_o) - I\frac{(t-t0)}{C}$$

The voltage V+ can be obtained by superposition. Therefore,

$$V^+(t) = \frac{VsatR1}{R1+R2} + \frac{VoR2}{R1+R2}$$

When V^+ goes through zero and becomes negative, the comparator output $V_0' = -V_{sat}$. at this instant $V^+(t_1) = 0$ or $V_0(T_1) = -(R_1 / R_2) V_{sat}$. the current supplied to the integrator for $T_2 > t > T_1$, is $I^- = (-V_{sat}) / R$.

The integrator output becomes a positive going ramp with the same slope as the negative going ramp. At a time $t = T_2$

$$V_0(T_2) = \left(\frac{R1}{R2}\right)V_{sat}$$

The comparator switches again to $+V_{sat}$ and the cycle repeats. The frequency can be determined by substituting the values in the V_0 (t) relationships. At $t = T_1$,

$$V_0(T_1) = -\left(\frac{R1}{R2}\right)V_{sat}$$

Hence

$$-\left(\frac{R1}{R2}\right) V_{sat} = \left(\frac{R1}{R2}\right) V_{sat} -- \left(\frac{Vsat}{RC}\right)\frac{T}{2}, \text{ since } t_1 = \frac{T}{2}$$

Therefore $$T = 4RC\frac{R1}{R2}$$

Or the frequency of the triangular waveform,

$$f = \frac{R2}{4\ RCXR1}$$

The amplitude can be controlled by the ratio (R1 / R2) V_{sat}.

The amplitude can be controlled by a back to back connected Zener diode combination at the output of the comparator [9, 10].

IV. Triangular Wave Form Generator Output in Different Software:

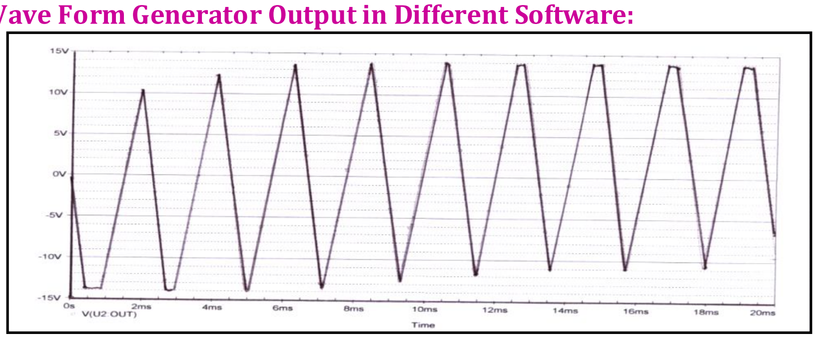

Figure 2Output Voltage of Triangular wave form generator using PSpice

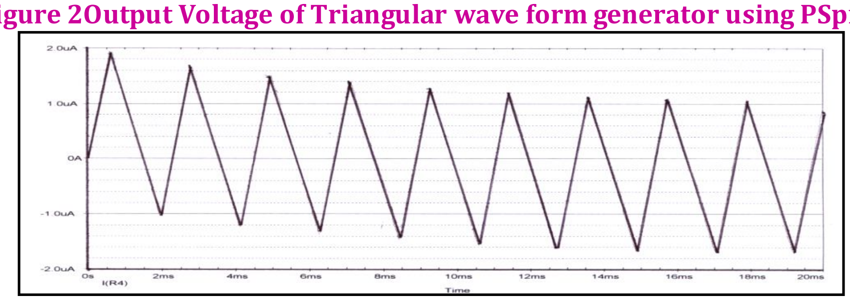

Figure 3Output current of Triangular wave form generator using PSpice

- Here wave starts from 200 ns. At this time the potential is 8.1905 mV.
- Initially at the positive half we get the sharp peak. But opposite to it there is thick peak but after the time 7.1277 ms at the positive half we get the thick peak but opposite to it there is sharp peak.
- At the starts Pulse goes to -Ve amplitude.
- Initially at the positive amplitude peak potential is 10.374 V and it increases up to the 13.853 V peak potential, but at the negative amplitude initial peak is at the maximum Potential i.e. -13.853 V and up to 20ms it decreases to the -9.1836 V.
- Frequency is near about 460 Hz.
- The potential and currents are out of phase shown in the above graph.
- Current wave starts at 200 ns.
- Initially at the positive amplitude peak current is 1.9143 μA and it decreases up to the 919.120 μA peak current, but at the negative amplitude initial peak is at the -1.0340 μA and up to19.204 ms it decreases to the -1.7145μA.
- Minimum current is -1.7145 μA.
- Maximum current is 1.9143 μA [11].

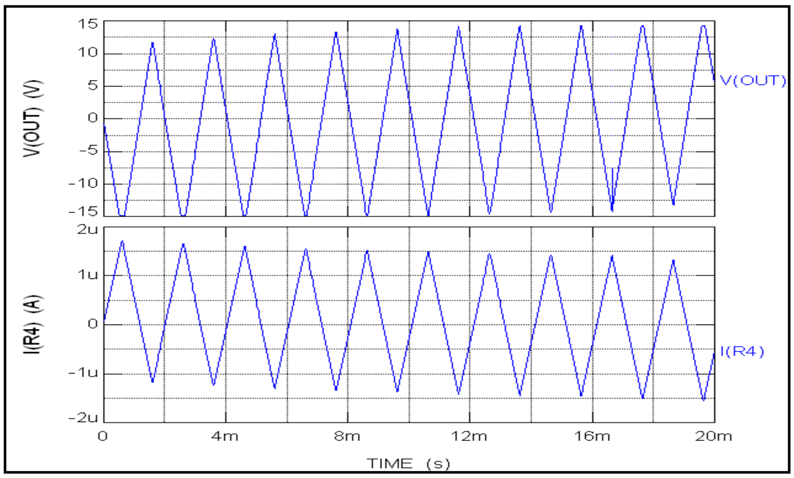

Figure 4 Output potential & current of Triangular wave generator sing Top Spice

- Here the wave starts from 27.714 μs. At this time the potential is -803.58 mV.
- Initially at the positive half we get the sharp peak but opposite to it there is thick peak but after the time 17.63 msat the positive half we get the thick peak but opposite to it there is sharp peak.
- At the starts Pulse goes to -Ve amplitude.
- Initially at the positive amplitude peak potential is 11.71 V and it increases up to the 14.30 V peak potential, but at the negative amplitude initial peak is at the maximum Potential i.e. -14.796 V and up to 20ms it decreases to the -13.13 V.
- Frequency is near about 485 Hz.
- The potential and currents are out of phase shown in the above graph.
- Current wave starts at 27.714 μs.
- Initially at the positive amplitude peak current is 1.71 μA and it decreases up to the 1.32 μA peak current, but at the negative amplitude initial peak is at the -1.23 μA and up to19.65 ms it decreases to the -1.55 μA.
- Minimum current is -1.55 μA.
- Maximum current is 1.71 μA
- In this software, dead time is 27.714 μs [12].

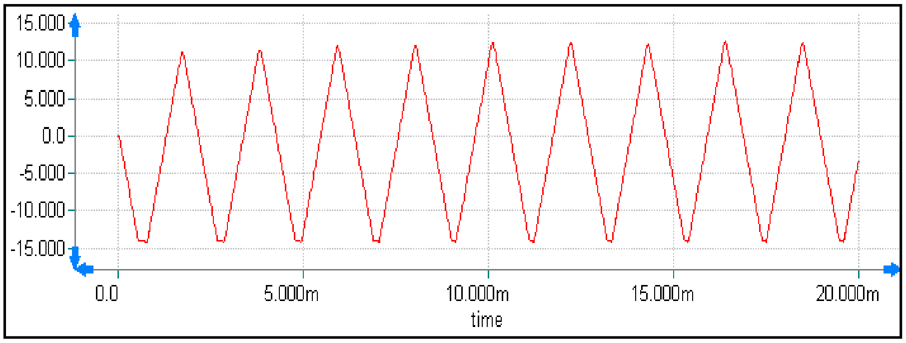

Figure 5 Output potential of Triangular wave generator using B2 Spice

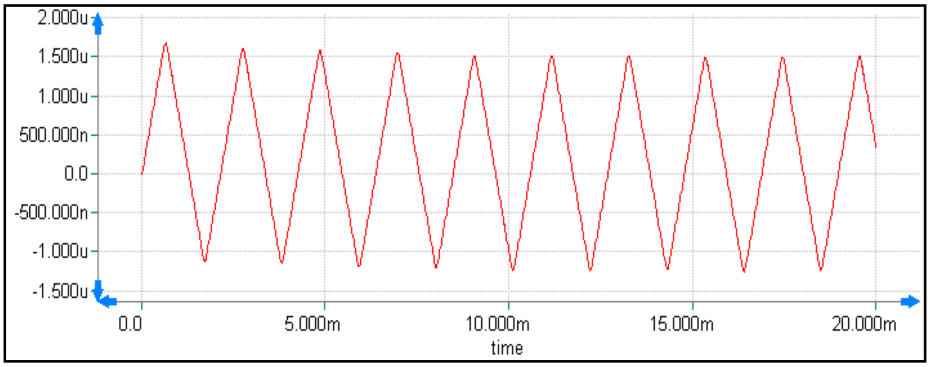

Figure 6 Output current of Triangular wave generator using B2 Spice

- Here wave starts from 0 s. At this time the potential is 0.002 V.
- At the positive half we get the sharp peak but opposite to it there is thick peak.
- At the starts Pulse goes to -Ve amplitude.
- Initially at the positive amplitude peak potential is 11.204 V and it increases up to the 12.549 V peak potential, but at the negative amplitude initial peak is at the maximum Potential i.e. -14.136 V and up to 20 ms it increases to the -14.133 V.
- Frequency is near about 475 Hz.
- The potential and currents are out of phase shown in the above graph.
- Current wave starts at 0 s.
- Minimum current is -1.45 μA.
- Maximum current is 1.700 μA.
- In this software, dead time is 0 s.
- Period of the wave is2.1 ms [13].

Table 1 Data for simulated triangular wave generator circuit in PSpice,Top Spice and B2 Spice

Software	Negative Maxima	Positive Maxima	Start Time	Starting potential
PSpice	-13.856	13.853	0.0002	8.1905
Top Spice	-14.802	14.32	0.028	-803.589
B2 Spice	-14.138	12.549	0	0

Table 2 Data for simulated triangular wave generator circuit in PSpice, Top Spice and B2 Spice.

PSPICE		TOPSPICE		B2 SPICE	
Negative Potential	Positive Potential	Negative Potential	Positive Potential	Negative Potential	Positive Potential
-13.856	10.374	-14.796	11.71	-14.136	11.204
-13.853	12.236	-14.798	12.334	-14.139	11.446
-13.853	13.582	-14.796	12.789	-14.138	11.893
-13.778	13.853	-14.797	12.97	-14.138	12.089
-12.674	13.853	-14.61	13.75	-14.133	12.367
-11.892	13.853	-14.44	14.15	-14.134	12.349
-11.344	13.853	-14.51	14.22	-14.136	12.18
-10.985	13.853	-14.26	14.3	-14.125	12.549
-10.73	13.853	-13.73	14.29	-14.133	12.375
-9.2	13.853	-13.13	14.3	-14.133	12.375

Table 3 Data for period, frequency, % output wrt theoretical value and theoretical value of frequency of simulated Triangular wave generator circuit in Pspice, TopSpice& B2 Spice.

Triangular wave generator				
Software	**Period in ms**	**Simulation frequency in Hz**	**% output wrt theoretical value**	**Theoretical value of frequency in Hz**
PSpice	2.1711	460	79.04	**582**
Top Spice	2.062	485	83.34	
B2 Spice	2.105	475	81.61	

V. RESULTS AND DISCUSSION:

This paper reports that the results obtained after simulating the triangular wave generator using astable multivibrator (same circuit) using different spice software test tools i.e. PSpice, B2 Spice and Top Spice. We observed the Positive maxima, negative maxima, starting potential and starting time for the triangular waves are different in different software's are given in Table 1. Table 2 shows us the peak to peak –Ve & +Ve potential &Table 3 Data for period, frequency, % output wrt theoretical value and theoretical value of frequency in different software's. The results obtained after performing simulation and theoretical results are not matching exactly, there is little bit variation.These software's avoids the complex, cumbersome, cyclic testing procedure for a newly developed product.

VI. REFERENCES:

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

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

[3] Walunj, A. K, A. V. Mancharkar and A. D. Shaligram. 2001. PSpiceSimulation for performance and reliability testingof sensorsignal conditioning circuits, J. Instrument. Soc. India 30 (20): 65-74.

[4] Martin O Hara1998. Modelling board- level DC-DC convertorsin SPICE (Electronic Product Design), July.

[5] Tunenga Paul W1998. SPICE: A guide to circuit simulation andanalysis using PSPICE, Englewood Cliffs N. J. USA,Prientice Hall.

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

[7] University of Pennsylvania, Department of Electrical &System Engineering2006. PSPICE – A brief primer. Jan VanderSpiegel @ 2006 jan-at-seusepenn.cdu.

[8] Rangan, C. S., G. R. Sharma and V. S. V. Mani 1997. InstrumentationDevices & System, Tata McGraw-Hill Ltd, New Delhi.
[9]Maheshwari, L. K. and M. M. S. Anand2007. LaboratoryExperiments and PSPICE simulations in AnalogElectronics Prentice – all of IndiaPrivate limited New Delhi – 110 001.
[10] Mumhad H. Rashid2006. Introduction to PSpice using orCADfor Circuits and Electronics, Prentice – Hall of India,Private Limited New Delhi – 110 001.
[11] PSpice Schematics, Evolution Version 9.1, Cadance Design Systems, www.Cadance.com.
[12] B2 Spice A/D 5.2.3 Copyright @ 2004-2007.Beige Bag Software, www.beigebag.com, info @ beigebag.com.
[13] TopSPICE/Win32 Schematic EditorDemo version 7.16c Copyright@ 1991-2008. By panzer Development. http://www.penzar.com.

Sanjay K. Tupe
Assistant Professor, Department of Physics, Kalika Devi Arts, Commerce & Science College, Shirur (k.) Dist.- Beed.