International Journal of Scientific Research in Science, Engineering and Technology
Print ISSN: 2395-1990 | Online ISSN : 2394-4099 (www.ijsrset.com)

Study of Voltage Regulator Using B2 Spice, Tina & Circuit Maker

Dr. Sanjay K. Tupe[1]

[1]Department of Physics, Kalikadevi Arts, Commerce & Science College, Shirur (K.), Dist. Beed-413249, Maharashtra, India

ABSTRACT

Article Info
Volume 9, Issue 5
Page Number: 157-160

Publication Issue :
July-August-2021

Article History
Accepted : 02July2021
Published:25 July, 2021

This paper addresses the study of voltage regulator using PSPICE and TOP SPICE. Spice stands for "The Simulation Programme for Integrated Circuit Emphasis". Traditionally electronic circuit design was verified by building prototypes, subjecting the circuits to various stimuli and measuring its response using appropriate laboratory equipment's. It is time consuming, instead of this if we use various SPICE software's, we can perform the number of analyses of the same circuit virtually. Virtual results are very close to the actually build circuits. It gives new ideas that could lead to improve the circuit performance. As well gives idea about the configuration of electronic components. It saves the time for to develop the proper circuit design for the special purpose.

Keywords: - Prototype, Spice, Virtual design

I. INTRODUCTION

SPICE is a great tool to learn a lot in a short time. Also, busy lives and limited budgets can make experimenting with real parts and expensive equipment nearly impossible. What may take you an hour to wire up in the lab to get a minor concept could be covered in a few minutes with SPICE. For example, how does an amplifier's gain vary with bandwidth? Before the circuit parts were even collected, you can get hands on experience with the gain-bandwidth trade off. While text and equations tell you the story, a simulation can clarify the concept and drive it home.

Electronic circuit and systems need a stable dc voltage for their intended operation. The required dc voltage is usually obtained by converting the mains ac voltage into a dc voltage. After a sutitable step-up or step-down transformation, the ac voltage is rectified and filtered resulting in to a dc voltage. However, the dc voltage thus obtained d-oes not remain constant with increasing load current, variations in mains voltageand changes in the ambient tempurature. The filtered output is therefore applied a voltage regulator which provides a stable dc voltage at its output.

The lack of regulation of the dc output voltage may lead to distorted output, frequency shift in an oscillator or change of calibrationin measuring instrumnts. Therefore voltage regulator forms an important component in an electronic powersupply. The DC voltage regulator using op-amp is studed with fallowing circuit diagram.

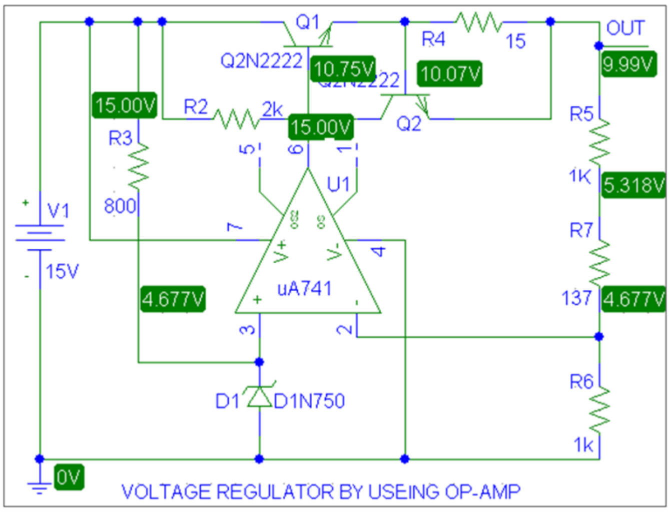
II. EXPERIMENTAL WORK

In this study we have choose the one circuit of voltage regulator. Same circuit is built in the worksheet or windows of PSPICE and TOP SPICE with same configuration of electronic components as well the inputs and observed the outputs by simulating these. The regulated output voltage Vo is given by

$$Vo \quad (1 + \frac{Rf}{RI})\ Vz$$

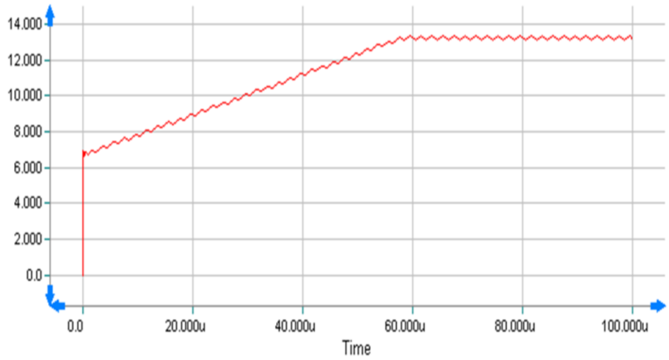
Where R_f = R5 + R7 and RI = R6

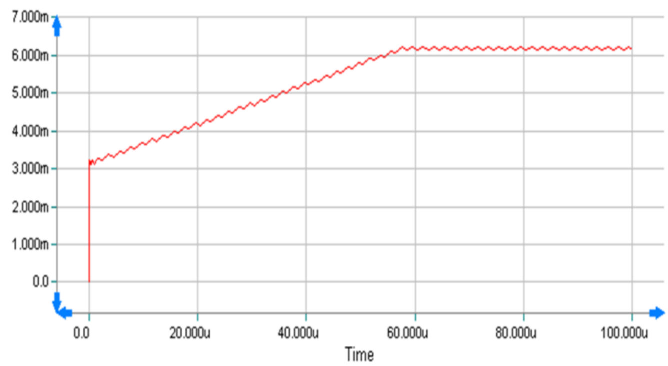
In the simple operational regulator by using a pass transistor Q1 as shown in fallowing figure, the load current can be increased by a factr of 100 as the op-amp will required supply only the base current of Q1. The voltage can be adjusted by varying the potentiometer i.e. adjusting the values of R_f



A]. **The Following Graphs Shows the Output of Voltage Regulator inB2 SPICETINA &CIRCUIT MAKERSoftware:**

- The output starts at 0 Sec.
- At 0 Sec the output potential is 7.02V.
- As the time increases the output potential increase.
- At 57.496µsec, we get the regular output 13.309V.
- After 57.496µsec, we get the regular output, which varies between 13.326V to 13.139V.
- We get the variation in output voltage is about 0.187V.
- The output current is 6.206mA.
- We get the variation in output current is about 0.006mA.
- For this, the values of R7 and R5 are 137 Ohm and 1000 Ohm respectively.
- This software does not respond for the change in the values of the R7 and R5.
- Under the same conditions in all circuits in this software, we get out put 13.326 V.
- For to get the regular output it takes the period 60 uS.
- in this circuit there is no maximum impact of R7 and R5on the output when we change the potential of the battery to 11 V, we get the regulated output varies between 10.03 V and 10.011V.
- In this software, we get the maximum flucations up to 669.139 µSec for the current.
- We get the variation in the output up to1.121msec and after that we get the regulated output potential up to 13.235V. in regular output varies between 13.253V to 13.252V. That is the variation in output is 0.001V.

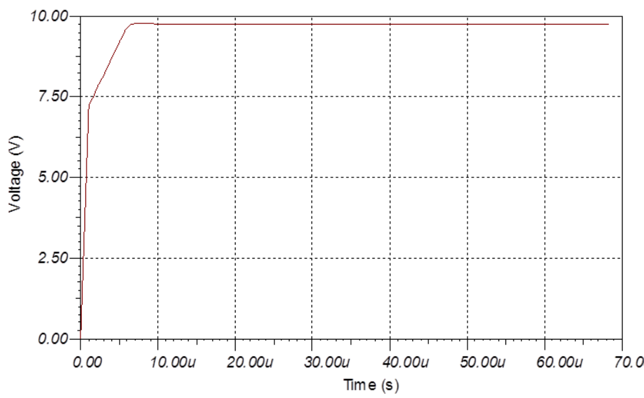




The above graph is the output of voltage regulator usingB2 SPICE

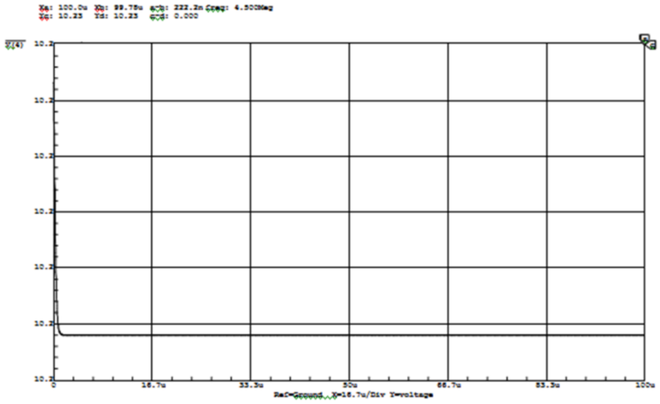
B]. The Following Graphs Shows the Output of Voltage Regulator in TINA & CIRCUIT MAKER Software:

- The output starts at 306.35nSec.
- At 306.35nSec the output potential is 2.2V.
- As the time increases the output potential increase.
- At 8.12 µsec, we get the regular output 9.78V.
- After 8.12 µsec, we get the regular output9.76V.
- For this, the values of R7 and R5 are 137 Ohm and 1000 Ohm respectively.
- This software does not respond for the change in the values of the R7 and R5.
- For to get the 10V regular power supply it requires Rf 1190 Ohm and Ri 1000 Ohm.
- 15 V 1190 Ohm 10 V
- 15.5V 1190 Ohm 10.03V
- 16.5 V 1190 Ohm 10.09 V
- In this software, we cannot observe the variations in the output.



The above graph is the output of voltage regulator using TINA

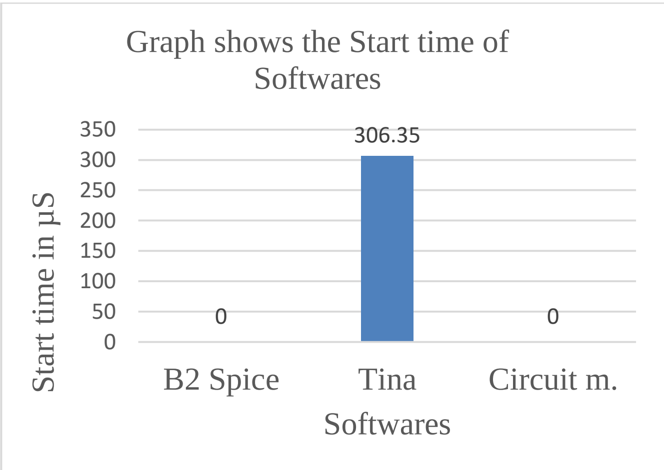
C]. The Following Graphs Shows the Output of Voltage Regulator in CIRCUIT MAKER Software:



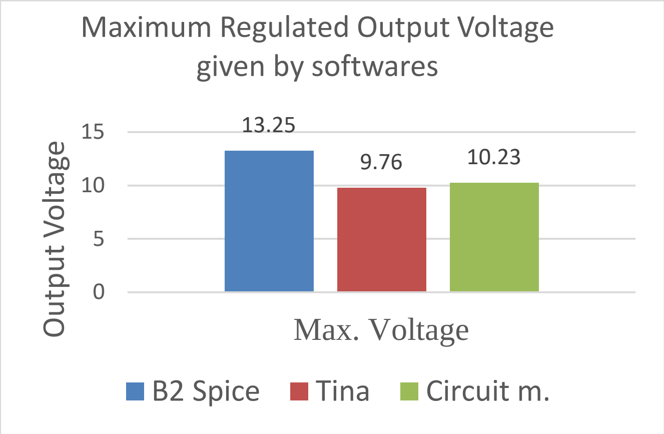
The above graph is the output of voltage regulator using CIRCUIT MAKER

- Under the same conditions in all circuits in this software, we get out put 10.23 V.
- For to get the regular output of 10 V takes the period 1.67uS.and requires a Rf 1089 ohm.
- this software is most sensitive for the change in the values of the resistances.
- In this software we get the maximum fluctuations up to .9 µS.

Software	Start Time in µS
B2 Spice	0
Tina	306.35
Circuit m.	0



Software	Max. Voltage
B2 Spice	13.25
Tina	9.76
Circuit m.	10.23



III. CONCLUSION

The output regulated voltage given by B2 Software is maximum ie 13.25 volt. It is maximum voltage output but there are negligible variations. The tina gives the 9.76 volt non fluctuated output. The Circuit Maker gives the 10.23 volt non fluctuated output.

IV. 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]. "Comparative study of different spice software's using astable multivibrator in different spice software's", Sanjay .K. Tupe, Sayyad S.B. and S.H. Behere. International Journal of Recent Trends in Engineering, CEE NOV 2009.

[8]. 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).

[9]. http://www.ecircuitcenter.com/About SPICE.htm.

[10]. Operational Amplifier - G. B. Clayton

[11]. Operational Amplifier and Linear Integrated Circuits - R. A. Gaikwad

[12]. Principles of Electronics - V. K. Mehta

[13]. Electronic Principles - A. P. Malvino

[14]. 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.

[15]. "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.