Chapter 4
SPICE SOFTWARE’S APPLIED TO THE ASTABLE MULTIVIBRATOR AND VOLTAGE REGULATOR
4.1 ASTABLE MULTIVIBRATOR:

As shown in fig 4.1 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 $V_c$ across it will rise exponentially towards $V_{cc}$. However as soon as this voltage reaches $V_{UT} = (2/3)V_{cc}$, 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 $V_c$. When the capacitor voltage becomes $V_{LT} = (1/3)V_{cc}$, 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 $t = 0$ is the instant when charging of $C$ begins, we can write the voltage across the capacitor during charging as

$$V_c(t) = V_{cc} - (V_{cc} - V_{LT})e^{-t/(R_1 + R_2)C}$$

And at $t = T_H$

$$V_c(t) = (2/3)V_{cc} = V_{UT}$$

and $V_{LT} = (1/3)V_{cc}$

Therefore

$$\frac{2}{3}V_{cc} = V_{cc} - (V_{cc} - \frac{1}{3}V_{cc})e^{T_H/(R_1 + R_2)C}$$

$$T_H = (R_1 + R_2)C \ln 2 = 0.69(R_1 + R_2)C$$

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

$$V_c(t) = V_{UT} e^{-t/R_2C}$$

$t = o$ is beginning of discharging of $C$)

At $t = T_L$
Figure 4.1 Astable multivibrator

\[ V_c(t) = \frac{1}{3}V_{cc} = V_{LT} \]

Hence

\[ \frac{1}{3}V_{cc} = \frac{2}{3}V_{cc} e^{-\frac{TL}{R2C}} \]

Or

\[ T_L = R_2C \ln2 \times 0.69R2C \]

The total time period,

\[ T = T_H + T_L \]
$$T = 0.69(R_1 + 2R_2)C$$

$$f = \frac{1}{T} = \frac{1.443}{(R_1+2R_2)C}$$

The duty cycle is

$$\% \text{ duty cycle} = \frac{TH}{T} \times 100$$

In this circuit the duty cycle is always be greater than 50%. If $R_1 \ll R_2$, it approaches 50%. [1-2].

4.2 Figures of Astable multivibrator circuit output in different software’s:

![Figure 4.2 Output potential of Astable multivibrator using Pspice](image)
Output starts 3.8364 µV to 4.9836 V.
Rise time and fall time in this software is exactly equal to 598 µs.
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 49.936 µA.
Figure 4.4 Output current and potential of astable multivibrator using Top

Spice [Reference: 7]

- 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 49.62 µA.
- Rise time and fall time; initially it is less but as the time increases it also increases.
Lower level of the output is 0.045 V to 0.049 V.

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 0.05 µA.

![Figure 4.7 Output current of Astable multivibrator using TINA [Reference: 9]](image)

- 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.
The maximum output voltage at the peak is 4.950 V.
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 4.1 Data for simulated Astable multivibrator circuit using Pspice, Top Spice, B2 Spice, Tina and Circuit Maker [Reference: 11].

<table>
<thead>
<tr>
<th>Software</th>
<th>Start Time</th>
<th>Start Pot.</th>
<th>Positive Potential</th>
<th>Negative Potential</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>100</td>
<td>-13.884</td>
<td>0.005</td>
<td>4.9836</td>
</tr>
<tr>
<td>Top Spice</td>
<td>0</td>
<td>-38.98</td>
<td>0.049</td>
<td>4.962</td>
</tr>
<tr>
<td>B2 Spice</td>
<td>0</td>
<td>0</td>
<td>0.045</td>
<td>4.962</td>
</tr>
<tr>
<td>Tina</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>3.6</td>
</tr>
<tr>
<td>Circuit m.</td>
<td>0</td>
<td>45</td>
<td>0</td>
<td>4.95</td>
</tr>
</tbody>
</table>

4.3 Comparison of Astable multivibrator circuit output in different software’s:

![Figure 4.9](image_url)  
Figure 4.9 Shows start time for Astable multivibrator in different software’s
Figure 4.10 Shows starting potential for Astable multivibrator in different software’s

Figure 4.11 Shows maximum and minimum potential for Astable multivibrator in different software’s
4.4 VOLTAGE REGULATOR:

Electronic circuits and systems need a stable dc voltage for their intended operation. The required d.c. voltage is usually obtained by converting the mains ac voltage. After a suitable step-up or step-down transformation, the ac voltage is rectified and filtered resulting in a dc voltage. However, the dc voltage thus obtained does not remain constant with increasing load current, variations in mains voltage and changes in the ambient temperature. The filtered output is therefore undergoes a voltage regulation which provides a stable dc voltage at its output [3-4].

The lack of regulation of the output voltage may lead to distorted amplifier output, frequency shift in an oscillator or change in calibration in measuring instruments. Therefore a voltage regulator forms an important component in an electronic powersupply. The dc voltage regulators using op-amp will be studied in this work using different software’s.

Most voltage regulators employ a zener diode as a astable reference voltage source. A zener diode with a series resistance can be used as simple low current voltage regulator. The addition of a transistor allows a larger load current to be supplied. A vast improvement in the regulator performance results when an error amplifier is added to detect and amplify the difference between the regulated output and the reference voltage. The dc voltage regulator circuits are also available as ICs and some of these are widely employed. The regulated output voltage $V_o$ is given by

$$V_o = (1 + \frac{R_f}{R_I}) V_z$$

Where $R_f = R_5 + R_7$ and $R_I = R_6$

In the simple operational regulator a pass transistor $Q_1$ is used as shown in following figure 4.12. The load current can be increased by a factor of 100 as the op-amp will be required to supply only the base current of $Q_1$. The voltage can be adjusted by varying the potentiometer i.e. adjusting the values of $R_6$ [1].
In Transient Analysis, we perform the simulation of the circuit and analyze the output voltage with respect to time. For this maximum simulation time and the time limit are set in the corresponding parameters window. Then simulation is performed with ‘Display Waveform’ option enabled. The output of the amplifier is viewed in the Waveform viewer. The user also has the provision to simulate with print and plot outputs [5].
4.5 Figures of voltage regulator by using op-amp output in different software’s:

Figure 4.13 Output potential of voltage regulator circuit using PSpice

Figure 4.14 Output current of voltage regulator circuit using PSpice

➢ The output starts at 20 ns.
At 20 ns the output potential is 10.07007 V.
As the time increases the output potential increases.
After 25.568 µs we get the regular output 9.995 V.
20 ns to 25.698 µs current varies from 762.4 µA to 4.6769 mA.
The maximum output current is 4.6769 mA.
For this, the values of $R_7$ and $R_5$ are 137 Ohms and 1000 Ohms respectively.

![Graph showing the output potential and current of a voltage regulator circuit using PSpice.](image)

**Figure 4.15 Output potential and current of voltage regulator circuit using PSpice**

- The output starts at 20 ns.
- At 20 ns the output potential is 13.73 V.
- As the time increases the output potential decreases.
At 48.56 ns, we get the regular output 13.7245 V.

After 48.56 ns we get the regular output which varies between 13.724521 V to 13.724389 V.

We get the variation in output voltage is about 0.000132 V.

The current varies from -6.3255363 mA to -6.3256127 mA.

We get the variation in output current is about 0.0000764 mA.

For this, the values of $R_7$ and $R_5$ are 137 Ohm and 1000 Ohm respectively.

This software does not respond for the change in the values of the $R_7$ and $R_5$.

For to get the regulated 10 V output input battery voltage should be change and it is equal to 12.75 V.

**Figure 4.16** Output Potential of voltage regulator circuit using B2 Spice

**Figure 4.17** Output current of voltage regulator circuit using B2 Spice
The output starts at 0 sec.
At 0 sec the output potential is 7.02 V.
As the time increases the output potential increase.
At 57.496 µs, we get the regular output 13.309 V.
After 57.496 µs, we get the regular output, which varies between 13.326 V to 13.139 V.
We get the variation in output voltage is about 0.187 V.
The output current is 6.206 mA.
We get the variation in output current is about 0.006 mA.
For this, the values of R_7 and R_5 are 137 Ohm and 1000 Ohm respectively.
This software does not respond for the change in the values of the R_7 and R_5.
Under the same conditions in all circuits in this software, we get output 13.326 V.
For to get the regular output it takes the period 60 us.
In this circuit there is no impact of R_7 and R_5 on the output when we change the potential of the battery to 11 V, we get the regulated output which varies between 10.03 V and 10.011 V.
In this software, we get the maximum fluctuations up to 669.139 µs for the current.
We get the variation in the output up to 1.121 ms and after that we get the regulated output potential up to 13.235 V. in regular output varies between 13.253 V to 13.252 V. That is the variation in output is 0.001 V.
Figure 4.18 Output potential of voltage regulator circuit using Tina

- The output starts at 306.35 ns.
- At 306.35 ns the output potential is 2.2 V.
- As the time increases the output potential increase.
- At 8.12 µs, we get the regular output 9.78 V.
- After 8.12 µs, we get the regular output 9.76 V.
- For this, the values of $R_7$ and $R_5$ are 137 Ohm and 1000 Ohm respectively.
- This software does not respond for the change in the values of the $R_7$ and $R_5$.
- For to get the 10 v regular power supply it requires $R_f$ 1190 Ohm and $R_i$ 1000 Ohm.
- In this software, we cannot observe the variations in the output.
Figure 4.19 Output potential of voltage regulator circuit using Circuit Maker

- Under the same conditions in all circuits in this software, we get output 10.23 V.
- For to get the regular output of 10 V takes the period 1.67 μs and requires a $R_f$ 1089 Ohm.
- This software is most sensitive for the change in the values of the resistance’s.
- In this software we get the maximum fluctuations up to 0.9 μs.

Table 4.2 Data for simulated voltage regulator circuit in Pspice, Top Spice, B2 Spice, Tina and Circuit Maker.

<table>
<thead>
<tr>
<th>Software</th>
<th>Start Time</th>
<th>Max. Pot.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>20</td>
<td>9.995</td>
</tr>
<tr>
<td>Top Spice</td>
<td>48.56</td>
<td>13.72</td>
</tr>
<tr>
<td>B2 Spice</td>
<td>0</td>
<td>13.25</td>
</tr>
<tr>
<td>Tina</td>
<td>306.35</td>
<td>9.76</td>
</tr>
<tr>
<td>Circuit M.</td>
<td>0</td>
<td>10.23</td>
</tr>
</tbody>
</table>
4.6 Comparison of voltage regulator circuit output in different software’s:

Figure 4.20 Shows start time for voltage regulator in different software’s

Figure 4.21 Shows regulated output potential for voltage regulator in different software’s
REFERENCES:

8. B2 Spice A/D 5.2.3, Beige Bag Software www.beigebage.com info@beigebage.com
9. TINA™ for Windows, The Complete Electronics Lab version 6.00.008SFS.
10. CircuitMaker V6.2C Protel Technology, Inc. 5252N Edgewood Dr Ste175 Provo UT84604 USA.