Chapter 6
COMPARATIVE DISCUSSION OF THEORETICAL AND SIMULATION RESULTS AND FEATURES OF DIFFERENT SOFTWARE’S
6.1 COMPARISON OF SIMULATION AND THEORETICAL RESULTS:

Table 6.1, 6.2, 6.3, 6.4, 6.5 and 6.6 shows the variations between the theoretical and simulated values for the simulated circuits which are:

1. Square wave generator.
2. Triangular wave generator.
3. Astable multivibrator.
4. Voltage regulator.
5. Wein Bridge Oscillator.
6. RC Coupled Amplifier.

The above circuits are simulated in software’s Pspice, Top Spice, B2 Spice, TINA and Circuit Maker. We have performed the transient analysis, which gives the variation in output of the same circuit with respective to time.

Table 6.1 Data for period, frequency, % output wrt theoretical value and theoretical value of frequency of simulated square wave generator circuit in Pspice, Top Spice, B2 Spice, Tina and Circuit Maker.

<table>
<thead>
<tr>
<th>Software</th>
<th>Period in ms</th>
<th>Simulation frequency in Hz</th>
<th>% output wrt theoretical value</th>
<th>Theoretical value of frequency in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>2.08</td>
<td>480</td>
<td>96</td>
<td></td>
</tr>
<tr>
<td>Top Spice</td>
<td>1.9046</td>
<td>525</td>
<td>105</td>
<td></td>
</tr>
<tr>
<td>B2 Spice</td>
<td>2.143</td>
<td>467</td>
<td>93.4</td>
<td></td>
</tr>
<tr>
<td>Tina</td>
<td>2.1428</td>
<td>467</td>
<td>93.4</td>
<td></td>
</tr>
<tr>
<td>Circuit M.</td>
<td>2.1276</td>
<td>470</td>
<td>94</td>
<td></td>
</tr>
</tbody>
</table>
Figure 6.1 Period in msec. for square wave generator circuit in different software’s

Figure 6.2 Shows simulation frequency in Hz for square wave generator circuit in different software’s
From the above table and figures it is observed that the frequency of square wave generator circuit simulated in Pspice 96% matches with respective to the theoretical frequency value. Top Spice gives us frequency 105%, in B2 Spice and TINA we get the minimum frequency which matches 93.4% with the theoretical value.

Means Pspice gives the better frequency results, but we get the better square wave in a Top Spice (figure 3.4).

Table 6.2 Data for period, frequency, % output wrt theoretical value and theoretical value of frequency of simulated Triangular wave generator circuit in Pspice, Top Spice, B2 Spice, Tina and Circuit Maker.

<table>
<thead>
<tr>
<th>Software</th>
<th>Period in ms</th>
<th>Simulation frequency in Hz</th>
<th>% output wrt theoretical value</th>
<th>Theoretical value of frequency in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>2.1711</td>
<td>460</td>
<td>79.04</td>
<td></td>
</tr>
<tr>
<td>Top Spice</td>
<td>2.062</td>
<td>485</td>
<td>83.34</td>
<td></td>
</tr>
<tr>
<td>B2 Spice</td>
<td>2.105</td>
<td>475</td>
<td>81.61</td>
<td></td>
</tr>
<tr>
<td>Tina</td>
<td>2.176</td>
<td>470</td>
<td>80.76</td>
<td></td>
</tr>
<tr>
<td>Circuit M.</td>
<td>2.027</td>
<td>494</td>
<td>84.88</td>
<td>582</td>
</tr>
</tbody>
</table>
Figure 6.4 Period in ms. for triangular wave generator circuit in different software’s

![Bar Chart: Period in mSec](chart1.png)

Figure 6.5 Simulation frequencies in Hz for triangular wave generator circuit in different software’s

![Bar Chart: Simulation frequency in Hz](chart2.png)
From the above table and figures it is observed that the frequency of triangular wave generator circuit simulated in Circuit Maker 84.88% matches to the theoretical frequency value. Top Spice, gives us frequency 105%, in B2 Spice and TINA it matches 81.61 % and 80.76 % respectively with the theoretical value.

In Pspice we observe the minimum frequency i.e.79.04 %. Top Spice gives the quality triangular wave (figure 3.17). In triangular wave it is observed that initially we can’t get constant peak to peak value.

**Table 6.3 Data for period, frequency, % output wrt theoretical value and theoretical value of frequency of simulated Astable multivibrator circuit in Pspice, Top Spice, B2 Spice, Tina and Circuit Maker.**

<table>
<thead>
<tr>
<th>Software</th>
<th>Period in µSec.</th>
<th>Simulation frequency in Hz</th>
<th>% output wrt theoretical value</th>
<th>Theoretical value of frequency in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>100</td>
<td>10000</td>
<td>97.02</td>
<td>10307</td>
</tr>
<tr>
<td>Top Spice</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>B2 Spice</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td>--</td>
</tr>
<tr>
<td>Tina</td>
<td>90</td>
<td>11112</td>
<td>107.8</td>
<td></td>
</tr>
<tr>
<td>Circuit M.</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td></td>
</tr>
</tbody>
</table>
Figure 6.7 Period in µs. for Astable multivibrator circuit in different software’s

Figure 6.8 Simulation frequencies in Hz for Astable multivibrator circuit in different software’s
From the above table and figures it is observed that the frequency of square wave generator circuit simulated in Pspice is 97.02% matches to the theoretical frequency value. Top Spice, B2 Spice and Circuit Maker we cannot calculate it properly because these give the square waves of decreasing frequency. In these software’s initially period is less and up to 500 μs it increase. In TINA we get the perfect output but have more frequency than the theoretical frequency i.e. 107.8 %.

Means for the above circuit we observed Pspice and TINA give us the proper output (figure 4.2, 4.7) but Top Spice, B2 Spice and Circuit Maker does not gives (figure 4.4, 4.5, 4.8).

Figure 6.9: % output wrt theoretical value for Astable multivibrator circuit in different software’s.
Table 6.4 Simulated value of regulated voltage and % output wrt theoretical value of simulated voltage regulated circuit in Pspice, TopSpice, B2 Spice, Tina and Circuit Maker.

<table>
<thead>
<tr>
<th>Software</th>
<th>Voltage regulator</th>
<th>Simulation value of regulated voltage</th>
<th>% output wrt theoretical value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td></td>
<td>9.995 V</td>
<td>99.5</td>
</tr>
<tr>
<td>Top Spice</td>
<td></td>
<td>13.72 V</td>
<td>136.6</td>
</tr>
<tr>
<td>B2 Spice</td>
<td></td>
<td>13.25 V</td>
<td>131.9</td>
</tr>
<tr>
<td>Tina</td>
<td></td>
<td>9.76 V</td>
<td>97.17</td>
</tr>
<tr>
<td>Circuit M.</td>
<td></td>
<td>10.23 V</td>
<td>101.85</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>Theoretical value of regulated voltage</td>
</tr>
</tbody>
</table>

Figure 6.10 Simulation value for regulated voltage for voltage regulated circuit in different software’s
Figure 6.11 % output wrt theoretical value for voltage regulated circuit in different software’s

From the above table and figures it is observed that in Pspice we get the most proper value of regulated voltage i.e. 99.5 % matches with the theoretical value. In TINA the 97.17 % output value is 97.17, In Circuit Maker it is 101.85 %. But in Top Spice and B2 Spice respectively these are 136.6 % and 131.9 %. In these software’s if we change the values of R₇ (figure 4.12) the value of the output changes

Top Spice and B2 Spice does not responds to the change in the values of the resistance R₇ (figure 4.12) but if we decrease the input value of V1 (figure 4.12) up to certain value we get the desired output. At the output of these software’s we get the negligible ac distortions (figure 4.15, 4.16).

But the Pspice, TINA and Circuit Maker gives the regulated output without ac distortions (figure 4.13, 4.18, 4.19).

Means for to design the regulated voltage power supply Pspice, TINA and Circuit Maker are better than the Top Spice and B2 Spice.
Table 6.5: Period, frequency, % output wrt theoretical value and theoretical value of frequency of simulated square Wein bridge circuit in Pspice, Top Spice, B2 Spice, Tina and Circuit Maker.

<table>
<thead>
<tr>
<th>Software</th>
<th>Period in ms</th>
<th>Simulation frequency in Hz</th>
<th>% output wrt theoretical value</th>
<th>Theoretical value of frequency in Hz</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>1.11</td>
<td>900</td>
<td>94.14</td>
<td></td>
</tr>
<tr>
<td>Top Spice</td>
<td>1.08</td>
<td>925</td>
<td>96.76</td>
<td></td>
</tr>
<tr>
<td>B2 Spice</td>
<td>1.05</td>
<td>950</td>
<td>99.37</td>
<td>956</td>
</tr>
<tr>
<td>Tina</td>
<td>--</td>
<td>--</td>
<td>--</td>
<td></td>
</tr>
<tr>
<td>Circuit M.</td>
<td>1.05</td>
<td>950</td>
<td>99.37</td>
<td></td>
</tr>
</tbody>
</table>

Figure 6.12 Shows period in ms. for Wein bridge oscillator circuit in different software’s
Figure 6.13 Shows simulation frequency in Hz for Wein bridge oscillator circuit in different software’s

Figure 6.14 Shows % output wrt theoretical value for Wein bridge oscillator circuit in different software’s

From the above table and figures it is observed that in B2 Spice and Circuit Maker we get the most proper values of Wein bridge oscillator which matches 99.37 % to
the theoretical value. In Pspice and Top Spice we also get the proper values of frequency i.e. 94.14% and 96.37%.

In TINA we can’t get the desired output (figure 5.7). It should be noted here that in Pspice, Top Spice, B2 Spice and Circuit Maker the positive and negative peak potentials are different as shown in a table 5.1. In Circuit Maker we get the output whose peak values are at the same potential i.e. 41.8 μV or 0.0418 mV (figure 5.8).

Table 6.6 Input and output voltage amplitude for RC coupled amplifier

<table>
<thead>
<tr>
<th>Software</th>
<th>Peak to peak input voltage $V_{pi}$ (mV)</th>
<th>Input voltage amplitude $V_{pi}/2$ (mV)</th>
<th>Peak to peak output voltage $V_{po}$ (mV)</th>
<th>output voltage amplitude $V_{po}/2$ (mV)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pspice</td>
<td>1.99</td>
<td>0.995</td>
<td>22.235</td>
<td>11.1275</td>
</tr>
<tr>
<td>Top Spice</td>
<td>1.982</td>
<td>0.991</td>
<td>22.013</td>
<td>11.0065</td>
</tr>
<tr>
<td>B2 Spice</td>
<td>2</td>
<td>1</td>
<td>20.412</td>
<td>10.206</td>
</tr>
<tr>
<td>TINA</td>
<td>2.002</td>
<td>1.001</td>
<td>21.16</td>
<td>10.58</td>
</tr>
<tr>
<td>Circuit M.</td>
<td>2</td>
<td>1</td>
<td>21.34</td>
<td>10.67</td>
</tr>
</tbody>
</table>

Figure 6.15 Maximum amplitude of an input signal to RC coupled amplifier.
From the above table and figures it is observed that in Pspice we get the maximum amplification of the given input signal i.e. 11.1275 mV. In Top Spice it is 11.0065 mV. Comparatively in B2 Spice, TINA and Circuit Maker give less amplification i.e. respectively 10.206 mV, 10.58 mV and 10.67 mV.

Pspice and Top Spice give slight more amplitude of the input signal than the actual provided signal amplitude. B2 Spice and Circuit Maker give the same amplitude of input signal, which is to be actually provided. TINA give slight more amplitude than the provided signal amplitude.

Overall it is observed, all software gives good response for the RC coupled amplifier circuit for amplification of given input signal.

6.2 ACTUAL OUTPUTS:

The following figure 6.17 shows the outputs of the actual circuits build in lab. 1) Square wave generator output. 2) Triangular wave generator output. 3) Astable multivibrator output. 4) Wein bridge oscillator output.
1. Square wave generator output  
2. Triangular wave generator  
3. Astable multivibrator output  
4. Wein bridge output  
5. Voltage regulator output

Figure 6.17 Actual outputs of the circuits built in the electronic lab.
5) Voltage regulator output. For to observe the output of first two circuits we used digital CRO and for the rest of three circuits analog CRO. In this work it is observed that the theoretical values, practical values and the simulated values are nearly equal. Some software does responds for one circuit but for the other circuit its response is very nice. In simulation software one can build the circuit within very short time. We can change the component or its desired value with in fraction of second. It is without any loss or extra expenditure. Due to simulation one can get the idea about the output of the circuit and values of the required components.

It should be noted here in this work the same circuits with same component values are used in all software which we have used for the comparative study. So we get the different output values depends upon the reliability of the virtual components in different software. It should be taking into account one can get the perfect desired output in each software, by changing the values of virtual components in respective software.

6.3 FEATURES OF SOFTWARE ARE USED:

6.3.1 Features of Pspice:
Schematics are a schematic capture front-end program with a direct interface to other PSpice programs and options.
You can perform the following tasks, all in one environment, using Schematics:

- Design and draw circuits.
- Simulate circuits using PSpice.
- Analyze simulation results using the PSpice waveform viewer (Probe).
- Graphically characterize simulation stimuli using the fully integrated Stimulus Editor, so stimulus definitions are automatically associated with the appropriate symbols.
Graphically characterize simulation models using the fully integrated Model Editor utility (formerly known as Parts), so model definitions are automatically associated with the appropriate symbols.

Interface to PSpice Optimizer for analog circuit performance optimization. An important prerequisite to building a schematic is the availability of the necessary devices in the form of symbols. Schematic has extensive symbol libraries and includes a fully integrated symbol editor for creating your own symbols or modifying existing symbols. The main tasks in Schematics are:

- Creating and Editing Designs.
- Creating and Editing Symbols.
- Creating and Editing Hierarchal Designs.
- Preparing your Design for Simulation.
- Preparing your Design for Board Layout [1].

6.3.2 Features Top Spice:

- Macro modeling with parameter passing.
- Automatic model libraries search.
- Supports most PSpice™ syntax extensions, and many HSPICE™ extensions and model libraries.
- User defined parameters and expressions.
- State-of-art convergence aid algorithms solve difficult convergence problems.
- Monte Carlo and worst case analyses allow with device and lot statistical distributions.
- BSIM3 v3.2 (levels 7, 8, 49, 53) and EKV v2.6 (levels 44 and 55) MOSFET models.
- "Statz", "Curtice" and Triquint GaAs FET models (levels 1, 2, 3 and 6).
- Jiles-Atherton nonlinear magnetic core model.
- HSPICE™ compatible diode level 3, and geometrical integrated resistor and capacitor models.
- Table look-up modeling of capacitors and inductors.
- Built-in advanced ferroelectric capacitor model.
- Lossy transmission line models.
- Analog behavioral modeling using arbitrary equations, logical expressions and look-up tables.
- Linear system frequency domain modeling using arbitrary Laplace transforms frequency response and s-parameter tables.
- Enhanced K element allows unlimited number of coupled inductors.
- PRINT output supports device currents, source impedances, magnetic flux and digital states.
- Binary output option saves all voltages, device currents and digital node states.
- Complex AC waveform expression support.
- FFT, histogram and Smith chart plots.
- CSDF data file support allows you to plot simulation data from other SPICE simulators [2].

6.3.3 Features of B2 spice:

- A range of powerful virtual instruments.
- Circuit Visualization to display the actual current flow through the circuit and the
- Relative voltage relationships by varying the wire's display color.
- Quick, easy, and intuitive schematic entry.
- Export to and import from Eagle, a world leader in PCB software.
- A continuously run Simulation mode.
- PCB export to make your designs a reality.
- 25,000 digital and analog parts including hundreds of REALISTIC behavioral models for such parts as resistors.
- Multiple bit ports and buses.
- Parameterized sub circuits.
- Create a part from any circuit.
- Password protected defects.
- Shared models.
- Database editor to import and manage the library of parts.
- Integrated symbol editor.
- PCB export and bill of materials.
- Improved schematics with DIN and ANSI symbols.
- RF simulations and network analysis.
- Schematic borders and title-box for professional output.
- Smith and polar plots.
- Intuitive, full featured schematic editor.
- Cut, copy and paste of selected items.
- Undo support.
- Full device rotation.
- Device mirroring.
- Parts browser.
- Repeat placement of a device.
- Easy to draw and edit wires.
- Browse-able, filterable, device libraries.
- User-defined devices and symbols.
- Complete macro device capability.
- Rubber banding of wires and devices.
- Annotation of devices.
- Quick menu selection of commonly used parts.
- Zoom in or out on an area or item with custom zoom factors.
Fit circuit to window function.
User-selectable colors.
Export circuit drawings and waveforms.
Supports all Microsoft Windows driven printers and plotters.
Built-in Symbol Editor to create custom device symbols.
Modification of devices and symbols in schematic.
Control over fonts and colors in schematic.
Export SPICE3 compatible net lists.
Generate SPICE3 sub circuits from the circuit.
Create part from the circuit.
View steady state results directly in the schematic.
View node numbers.
Color coded digital wire states.
Copy circuit picture to the clipboard for pasting into other applications.
Print circuit to any output device [3].

6.3.4 Features of TINA:

Sub circuits (Create your own components from Schematic or Spice sub circuits).
Schematic Symbol Editor (Create your own Schematic symbols).
Component Toolbar Editor (Add your new components to TINA's graphic component bar).
Parameter Extractor (create component models from measurement or catalog data).
Spice Library Manager (add more models to TINA's Spice Libraries or create your own library).
Symbolic Analysis (Results as closed form expressions).
Fourier Analysis (Fourier Spectrum, Fourier Series and distortion).
- Noise Analysis (Noise spectrum, Signal to Noise ratio and more).
- Tolerance Analysis (Monte Carlo and Worst-Case Analysis).
- Virtual Instruments (Multi-meter, Oscilloscope, Signal Generator, Signal Analyzer, Logic Analyzer).
- Real Time Test and Measurement.
- Teaching and training tools (Problem solving and troubleshooting) Interpreter.
- Components Optimization (Find predefined target and sensitivity) [4].

### 6.3.5 Features of Circuit Maker:

- Allowing you to design, simulate and output your printed circuit board designs.
- Exceptional ease of use and tight integration means you'll spend less time learning the software and more time designing... all the features of professional, high-end software at a fraction of the cost." Student Version, same as CM 6 with the following limitations: Max 50 devices per design (any type); Device library limited to 1,000 models; Symbol editor and Macro feature disabled; PCB netlist export limited to TraxMaker format; Technical support limited to fax and email only.
- Technical Support is virtually non-existent, so it's just as well that paper and on-screen Help is relatively good. I'm a user of CM 2000 myself.
- Students must test every design in CircuitMaker. CircuitMaker has virtual voltmeters, oscilloscopes, function generators that behave much like the ones they use in lab.
- Building circuits in Circuit Maker is much faster than building them in lab, and students make fewer wiring mistakes.
- The software is safe to use alone, and the components never break!
- Students quickly determine whether their circuits behave as they planned. If they do not understand something, they find out immediately.
They usually discover their own misunderstandings. Students frequently discuss the measurements reported by Circuit Maker, and work with each other to determine what is wrong with the circuit and their understanding of its underlying principles.

Occasionally the students need help from me. We nearly always discuss the student's design in front of Circuit Maker. Rather than telling the student the answer, I can prompt the student to measure voltages at key points, so that the student can often discover the problem on his or her own. The student will have learned not only more about the design of the circuit, but will also have learned more about diagnosing circuit problems [5].

REFERENCES:

1. PSpice Schematics, Evaluation Version9.1 www.cadence.com
2. Top SPICE/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. TINA™ for Windows, The Complete Electronics Lab version 6.00.008SFS.
5. CircuitMaker V6.2C Protel Technology, Inc. 5252N Edgewood Dr Ste175 Provo UT84604 USA.