# CONTENTS

## CHAPTER 1: INTRODUCTION  

<table>
<thead>
<tr>
<th>Section Description</th>
<th>Page No.</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1 Background</td>
<td>1</td>
</tr>
<tr>
<td>1.2 Electron</td>
<td>1</td>
</tr>
<tr>
<td>1.3 Vacuum tubes</td>
<td>3</td>
</tr>
<tr>
<td>1.4 Semiconductor</td>
<td>5</td>
</tr>
<tr>
<td>1.5 Diode</td>
<td>7</td>
</tr>
<tr>
<td>1.5.1 Zener diode</td>
<td>9</td>
</tr>
<tr>
<td>1.5.2 Light emitting diode (led)</td>
<td>9</td>
</tr>
<tr>
<td>1.5.3 Photo diode</td>
<td>9</td>
</tr>
<tr>
<td>1.5.4 Tunnel diode</td>
<td>9</td>
</tr>
<tr>
<td>1.5.5 Varactor diode</td>
<td>10</td>
</tr>
<tr>
<td>1.5.6 Shockley diode</td>
<td>10</td>
</tr>
<tr>
<td>1.6 Transistors</td>
<td>11</td>
</tr>
<tr>
<td>1.7 Integrated circuit</td>
<td>16</td>
</tr>
<tr>
<td>1.7.1 Introduction</td>
<td>16</td>
</tr>
<tr>
<td>1.7.2 Invention</td>
<td>17</td>
</tr>
<tr>
<td>1.7.3 Generations: SSI, MSI AND LSI</td>
<td>18</td>
</tr>
<tr>
<td>1.7.4 VLSI</td>
<td>18</td>
</tr>
<tr>
<td>1.7.5 ULSI, WSI, SOC &amp; 3D-IC</td>
<td>20</td>
</tr>
<tr>
<td>1.7.6 Advances in integrated circuits</td>
<td>20</td>
</tr>
<tr>
<td>1.7.7 Popularity of ICs</td>
<td>21</td>
</tr>
<tr>
<td>1.7.8 Classification</td>
<td>21</td>
</tr>
<tr>
<td>1.7.9 Fabrication</td>
<td>22</td>
</tr>
<tr>
<td>1.7.10 Packaging</td>
<td>25</td>
</tr>
<tr>
<td>1.7.11 Legal protection of semiconductor chip layouts</td>
<td>26</td>
</tr>
<tr>
<td>1.7.12 Other developments</td>
<td>26</td>
</tr>
</tbody>
</table>
1.7.13 Silicon graffiti 27
1.7.14 Notable ICs 27
1.8 Microprocessor 27
  1.8.1 History First types 28
  1.8.2 General purpose 31
  1.8.3 Notable 8-bit designs 31
  1.8.4 16-bit designs 33
  1.8.5 32-bit designs 34
  1.8.6 64-bit designs in personal computers 36
  1.8.7 Multicore designs 37
  1.8.8 RISC 38
  1.8.9 Special-purpose designs 39
  1.8.10 Market statistics 40
  1.8.11 Memory Chips 40
1.9 What is spice? 42
  1.9.1 Why use spice? 42
1.10 Motivation of present study 44
1.11 Organization of the thesis 46
References 47

CHAPTER 2: SPICE SOFTWARE'S IN DETAILS 50-79
2.1 Introduction 50
2.2 Spice:
  2.2.1 Descriptions of spice 52
2.3 Analysis Tests 54
2.4 Virtual instruments 57
2.5 Circuit simulation procedure 60
  2.5.1 Drawing the circuit 60
2.6 Different spice software’s

2.6.1 Pspice

2.6.1.1 Probe pspice

2.6.1.2 Output variables

2.6.2 Top Spice

2.6.2.1 Program overview

2.6.2.2 Graphical post-processor with waveform analysis in top spice

2.6.3 B2 spice

2.6.3.1 Schematic view tools and menus

2.6.3.2 Overview

2.6.3.3 Highly accurate simulations

2.6.3.4 Short review

2.6.4 TINA

2.6.4.1 Fault management

2.6.4.2 To insert a fault

2.6.4.3 To restore a faulty component to its original parameters

2.6.4.4 Training / examination with TINA

2.6.4.5 T&M Open Testcard

2.6.4.6 T&M Close Testcard

2.6.4.7 Virtual instruments in TINA

2.6.4.8 Features

2.6.5 Circuit Maker

2.6.5.1 Analog/mixed-signal simulation

2.6.5.2 Circuit Maker’s simulation mode

2.6.5.3 Circuit Maker probe tool

References
CHAPTER 3:
SPICE SOFTWARE'S APPLIED TO THE SQUARE WAVE GENERATOR AND TRIANGULAR WAVE GENERATOR 80-105

3.1 Introduction 80
3.2 Square wave generator using astable multivibrator 80
3.3 Figures of square wave generator output in different software's 83
3.4 Comparison of square wave generator circuit output in different software's 88
3.5 Triangular wave form generator circuit 90
3.6 Figures of triangular wave generator circuit output in different software's 92
3.7 Comparison of triangular wave generator circuit output in different software's 103
References 105

CHAPTER 4:
SPICE SOFTWARE'S APPLIED TO THE ASTABLE MULTIVIBRATOR AND VOLTAGE REGULATOR 106-125

4.1 Astable Multivibrator 106
4.2 Figures of Astable Multivibrator output in different software 108
4.3 Comparison of Astable Multivibrator circuit output in different software's 114
4.4 Voltage Regulator 116
4.5 Figures of Voltage Regulator by using op-amp output in different software's 118
4.6 Comparison of Voltage Regulator circuit output in different software's 124
References 125

CHAPTER 5:
SPICE SOFTWARE'S APPLIED TO THE WEIN- BRIDGEOSCILLATOR AND RC COUPLED AMPLIFIER 126-146

5.1 Wein-bridge oscillator 126
5.2 Figures of Wein Bridge oscillator output in different software's 129
5.3 Comparison of Wine bridge oscillator output in different software's 134
5.4 RC coupled amplifier 136
5.5 Figures of RC coupled amplifier output in different software’s 139
5.6 Comparison of RC coupled amplifier circuit output in different software’s 144
   References 146

CHAPTER 6:
COMPARATIVE DISCUSSION OF THEORETICAL AND SIMULATION
RESULTS AND FEATURES OF DIFFERENT SOFTWARE’S 147-167

6.1 Comparison of simulation and theoretical results 147
6.2 Actual outputs 159
6.3 features of software are used 161
   6.3.1 Features of Pspice 161
   6.3.2 Features Top Spice 162
   6.3.3 Features of B2 spice 163
   6.3.4 Features of TINA 165
   6.3.5 Features of Circuit Maker 166
   References 167

CHAPTER 7:
SUMMARY AND FUTURE SCOPE 168-176

7.1 Summary 168
7.2 Applications 173
7.3 Future Scope 174
   References 176

PUBLICATIONS 177