BANDGAP VOLTAGE REFERENCE ANALYSIS USING LTSPICEXVII

Jaycee Ranes Jalos
Sarah Jane Evangelista Delgado
Master of Science in Electronics Engineering
Batangas State University
Batangas City, Batangas, Philippines
jaycee_jalos@yahoo.com
sarah.jane.delgado@dlsl.edu.ph

***Abstract*—There are different electronics circuit simulator available in the market as well as in the internet that can be purchased and downloaded. Those simulators are used to design, analyze, and simulate electronics circuit design. Designing an electronic circuit in a software is much cheaper than doing it in an actual experiment, testing and analyzing its function and behaviour with respect to temperature, power supply variations, and other parameters that may affect the circuit. One of those softwares is LTSpiceXVII. LTSpice is an open source computer software that is capable of designing, analyzing, and simulating electronic circuit design. It is developed by Linear Technology which is now part of Analog Devices.**

For the purpose of the tutorial, bandgap voltage reference circuit is chosen to demonstrate the use of LTSpiceXVII in analyzing the circuit. A bandgap voltage reference is a temperature independent circuit, meaning its design output voltage is stable eventhough there are variations in temperature. It is very useful in Analog to Digital Converter circuit that needs a stable voltage reference in order to operate properly. Using the LTSpiceXVII, we are able to measure current and voltages at different nodes, measuring points, and components in the circuit by graphing those parameters.

Using LTSpiceXVII, we can verify the measured currents, voltages and other parameters by using some formulas of the transistors, such as Saturation Current, Vbe, and other parameters of the circuit.

***Index Terms*—Bandgap, LTSpiceXVII.**

I. Introduction

Many electronics circuits or devices require a voltage reference that is as precise as possible. An ideal bandgap voltage reference circuit must produce a constant amount of voltage which is independent of manufacturing process variation, power supply, temperature changes and circuit loading from a device. It commonly has an output voltage of around 1.25V which is close to the theoretical 1.22eV energy gap or band gap of Silicon at absolute temperature. A Bandgap Voltage Reference circuit using LTSpiceXVII is shown in Figure 1.

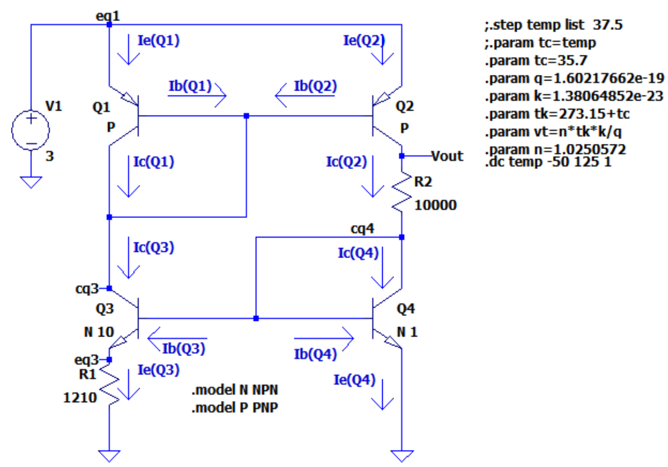

Fig. 1. Bandgap Voltage Reference Circuit

Bandgap Reference circuit must be able to cancel out two opposing variations which is caused by temperature. Thus, if we have two reference voltages which have the opposite temperature coefficients and they will be added, a temperature-independent circuit can be produced. The circuit in Fig. 1 shows a bandgap voltage reference circuit which utilizes a Bipolar Junction Transistor because it can provide both the Negative Temperature Coefficient (NTC) and one Positive Temperature Coefficient (PTC) voltages.

A. Bipolar Junction Transistor

The BJT collector current is defined as:

$$I_C = I_{sat} e^{\frac{V_{be}}{V_T}} \tag{1}$$

Where V_{be} is the base-emitter voltage and I_{sat} is the saturation current which is dependent with the process and temperature variations. The thermal voltage, V_T is calculated using the following equation.

$$V_T = \frac{T_K k}{q} n \tag{2}$$

Where k is the Boltzmann's constant, q is the charge of an electron, n is an ideality factor, which is a function of operating conditions and physical construction of the transistor and T_K is the temperature in Kelvin. The saturation current can also be written using the following equation:

$$I_{sat} = I_0 e^{-\frac{V_{G0}}{V_T}} \tag{3}$$

$$I_0 = I_{sat} e^{\frac{V_{G0}}{V_T}} \tag{4}$$

Where I_0 is a device current density rating and V_{G0} is the bandgap or energy gap voltage of silicon, which is the energy required to free an electron form the outermost shell of a Silicon atom. The bandgap itself is temperature-dependent, so V_{G0} is $V_G(T_K)$ extrapolated from 300K to 0K. In substituting equation (3) to equation (1), we get:

$$I_C = I_0 e^{\frac{1}{V_T}(-V_{G0}+V_{be})} \tag{5}$$

B. Negative Temperature Coefficient

The NTC voltage can be produced by the PN junction created between the base and emitter of the transistor. Using this equation (5), we can derive V_{be}.

$$e^{\frac{1}{V_T}(V_{G0}-V_{be})} = \frac{I_0}{I_C} \tag{6}$$

$$V_{be} = -\frac{T_K k}{q} \log\left(\frac{I_0}{I_C}\right) + V_{G0} \tag{7}$$

Assuming that I_C does not change with temperature, the change in V_{be} with respect to temperature will be:

$$\partial\left(\frac{V_{be}}{T_K}\right) = -\frac{k}{q} \log\left(\frac{I_0}{I_C}\right) \tag{8}$$

Normaly, the temperature coefficient is approximately equal to -2mV/C.

C. Positive Temperature Coefficient

The positive temperature coefficient can be produced by the thermal voltage V_T. Let us first take the difference of the two base-emitter voltages of the two transistors in the bandgap circuit.

$$V_{BE3} - V_{BE4} = V_T \log\left(\frac{I_{C3}}{I_{sat3}N_3}\right) - V_T \log\left(\frac{I_{C4}}{I_{sat4}N_4}\right) \tag{9}$$

Given that all the transistors are of the same model, it is assumed that their saturation currents are equal. Equation 9 will become:

$$\Delta(V_{be}) = V_T \log\left(\frac{I_{C3}N_4}{I_{C4}N_3}\right) \tag{10}$$

This shows that the difference in V_{be} between the two resistors is proportional to the thermal voltage, V_T.

II. Review of Related Literature

Anaconda is a package manager wherein it has a collection of different open source packages. It is a free and open source distribution of the Python and R programming languages for data science and achine learning applications which includes large-scale data processing, predictive analytics and scientific computing. The Python(x,y) and Anaconda contained the editor Spyder[12] that feature single line code run and interactive coding response. Spyder[12] is an open-source cross-platform Itegrated Development Environment (IDE) for scientific programming in the Python Language. Anaconda was prefered because it is based on Python 3.7 version while Python(x,y) was based on Python 2.7 version. The advantage of Anaconda is the Sympy symbolic math library that includes latex printing facilities. Hence, the interface to TexStudio[2][10] became straight forward. Block diagram could be manually drawn using MicroSoft Paint or Scilab XCOS. LTSpice is a circuit simulation tool which was designed by Linear Technology If you are an IC Design Engineer, SPICE simulator is one of the ways on checking your designed circuit prior to integration onto a chip. It also allows voltage and current measurements. Using this tool, the theoritical values of different parameters can be validated. The real-time responses of different circuits can be also examined using this. After learning how to use the tools above, the equation that was written using the editor Spyder[12], which has been tested and calculated also, can be validated through the use of the circuit simulator LTSpice. According to Brokaw (2011), Bandgap Voltage Reference Circuits can be effective voltage reference if it produces a constant amount of voltage which is not affected by the variation of manufacturing process and supply and temperature. Bandgap reference circuit must be able to cancel out two opposing variations which is caused by temperature. Thus, designing a bandgap circuit must utilize a device wherein it can provide both Proportional to Absolute Temperature (PTAT) and Complimentary to Absolute Temperature (CTAT). Brokaw (2011) showed the relationship between current densities and base-to-emitter volatge differences which practically displayed that the difference in base-to-emitter voltages was affected by the thermal voltage as well as the collector currents and emitter area of the transistors.

III. Methodology

Bandgap voltage reference should have different current densities between the two transistors to be utilized. Like in Figure 2 current density for Q3 and Q4 are N_3=10 and N_4=1 respectively, having 10:1 ratio.

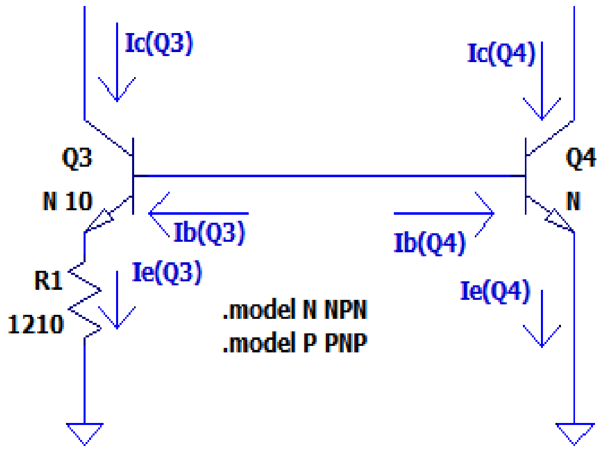

Fig. 2. Current Density of Q3=10 and Q4=1

Some of the known parameters are enumerated below.

$$I_{sat1} = I_{sat2} = I_{sat4} = I_{sat} = 7.6122742 \cdot 10^{-16} \tag{11}$$

$$I_{sat3} = 10 I_{sat} = 7.6122742 \cdot 10^{-15} \tag{12}$$

We'll approximate I_{sat} equal to $8e-16$.

$$T_C = 37.5 \tag{13}$$

$$T_K = T_C + 273.15 = 310.65 \tag{14}$$

$$V_T = \frac{T_K k}{q} n = 0.0274405113525198 \tag{15}$$

$$H_{fe1} = H_{fe2} = H_{fe3} = H_{fe4} = H_{fe} = 100 \tag{16}$$

where in Equation 12, it is stated that I_{sat3} is ten times of I_{sat} because it has to have different current densities between Q3 and Q4.
The difference between the V_{be} of Q3 and Q4 will determine the value of the PTC voltage. Saturation Current for all the transistors in the Bandgap circuit should be equal. Knowing this fact, we can equate the saturation current of Q3 and Q4 to find the difference between the V_{be} of Q3 and Q4.

$$I_{sat3} = \frac{I_{C3}}{N_3} e^{-\frac{V_{BE3} q}{T_K k}} \tag{17}$$

$$I_{sat4} = \frac{I_{C4}}{N_4} e^{-\frac{V_{BE4} q}{T_K k}} \tag{18}$$

Equating Equation 17 and Equation 18 wherein I_{sat3} and I_{sat4} are equal, we get.

$$\frac{I_{C4}}{N_4} e^{-\frac{V_{BE4} q}{T_K k}} = \frac{I_{C3}}{N_3} e^{-\frac{V_{BE3} q}{T_K k}} \tag{19}$$

Simplifying Equation 19.

$$e^{\frac{V_{BE3} q}{T_K k}} e^{-\frac{V_{BE4} q}{T_K k}} = \frac{I_{C3} N_4}{I_{C4} N_3} \tag{20}$$

$$\Delta(V_{be}) = V_{BE3} - V_{BE4} = \frac{T_K k}{q} n \log\left(\frac{N_4}{N_3}\right) \tag{21}$$

After getting the value for $\Delta(V_{be})$, we can now compute for the value I_{C3} using the formula below.

$$I_{R1} = \frac{1}{R_1} \Delta(V_{be}) \tag{22}$$

We have to establish the relationship among the currents in the circuit.

We can relate I_{C3} to I_{R1} using the current formula of a transistor.

$$I_{R1} = I_{B3} + I_{C3} \tag{23}$$

$$I_{R1} = I_{C3} + \frac{I_{C3}}{H_{fe}} \tag{24}$$

$$I_{R1} = \frac{I_{C3}}{H_{fe}} (H_{fe} + 1) \tag{25}$$

$$I_{C3} = \frac{H_{fe} I_{R1}}{H_{fe} + 1} \tag{26}$$

Figure 3 is a current mirror circuit used in our Bandgap Circuit in Figure 1. This current mirror circuit keeps the current between Q1 and Q2 equal, which is I_{C1} and I_{C2}. I_{C1} and I_{C2} is equal eventhough the temperature increases or decreases and regardess of loading.

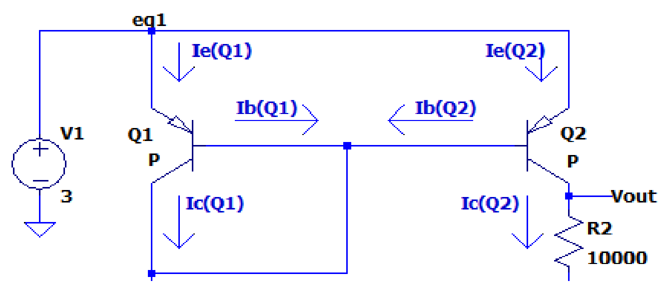

Fig. 3. Current Mirror Circuit

Using Kirchoffs Current Law we can say:

$$I_{C3} = I_{B1} + I_{B2} + I_{C1} \tag{27}$$

Considering I_{B1} and I_{B2} are equal and simplifying Equation 27, we can derived the formula for I_{C1}.

$$I_{C3} = \frac{I_{C1}}{H_{fe}} (H_{fe} + 2) \tag{28}$$

Since we have two equations to solve for I_{C3}. Now we can establish the relationship between I_{C1} and I_{R1} by equating Equation 26 and 28.

$$\frac{H_{fe} I_{R1}}{H_{fe} + 1} = \frac{I_{C1}}{H_{fe}} (H_{fe} + 2) \tag{29}$$

Using Equation 29 and simplifying it we can solve now the I_{C1}.

$$I_{C1} = \frac{H_{fe}^2 I_{R1}}{(H_{fe}+1)(H_{fe}+2)} \tag{30}$$

After finding the value of I_{C1} we can determine the value of I_{C3} by using either Equation 26 or 28.

Then V_{BE3} can be evaluated using the computed value of I_{C3} using Equation 31 below.

$$V_{BE3} = V_T \log\left(\frac{I_{C3}}{10 I_{sat}}\right) \tag{31}$$

Then, using the formula for $\Delta(V_{be})$ in Equation 21 we can compute for V_{BE4}.

$$\Delta(V_{be}) = V_{BE3} - V_{BE4} \tag{32}$$

Finally, computing for our V_{out}.

$$V_{out} = I_{C2}R_2 + V_{BE4} \tag{33}$$

IV. Result and Discussion

Considering the Bandgap Circuit in Figure 1 and applying the methods discussed above, the values of the circuit parameters can be obtained.

A. $\Delta(V_{be})$

Determining the value of $\Delta(V_{be})$ using Equation 21

$$V_{BE3} - V_{BE4} = -0.0605687523135148 \tag{34}$$

After verifying the value computed in Equation 22 using LTSpiceXVII we can see that it has nearly the same value. See Figure 3a.

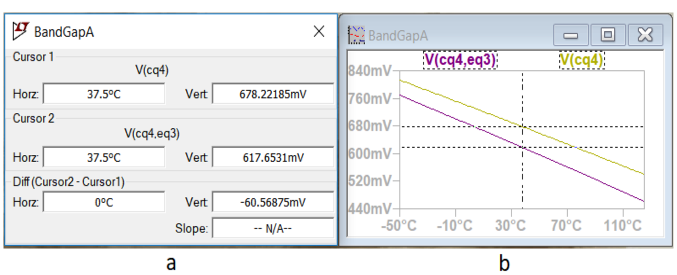

Fig. 4. a The difference between Vbe, b The graph for Vbe3 and Vbe4

This value of V_{be} is actually the value of our Positive Temperature Coefficient and also the voltage that we are going to use in determining the value of our resistor. But, since we are using the pre-designed circuit of LTSpiceXVII such has already the value of resistor (R_1) which is 1210 Ω. See Figure 1.

With this information we can verify in LTSpiceXVII that the voltage across R1 at 37.5 C. Figure 5 shows the computed change in $\Delta(V_{be})$ between V_{BE3} and V_{BE4} are nearly equal.

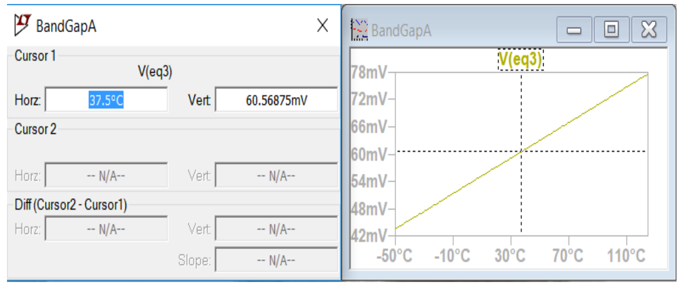

Fig. 5. The voltage drop across R1

B. Current at R_1

We get the value of our $\Delta(V_{be})$ by assuming that I_{C3} and I_{C4} are equal. Now let us try to solve for the value of I_{R1} using the value of R_1 in Figure 1 using Equation 22.

$$I_{R1} = 5.00568200938139 \cdot 10^{-5} \tag{35}$$

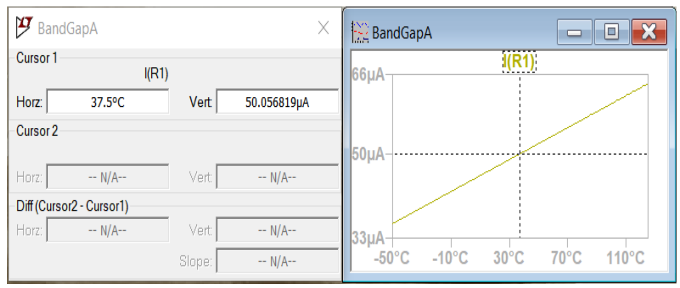

Fig. 6. The value of I_{R1} using LTSpiceXVII

By using the value we got for I_{R1}, we can compute for the value of I_{C1} using Equation 30.

$$I_{C1} = 4.8589419621252 \cdot 10^{-5} \tag{36}$$

We know that I_{C1} and I_{C2} are equal as this is the concept of a current mirror circuit. So we can state the equality of the two collector current of Q1 and Q2 as:

$$I_{C2} = I_{C1} = 4.8589419621252 \cdot 10^{-5} \tag{37}$$

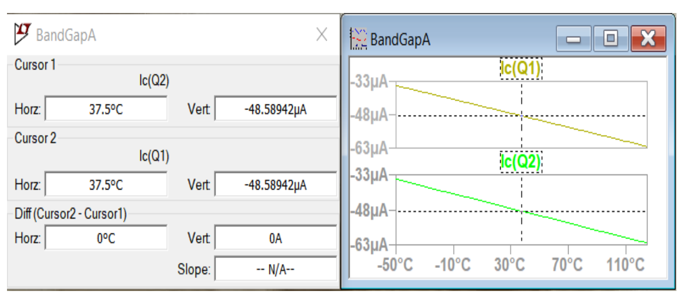

Fig. 7. The equal current of I_{C1} and I_{C2}

Find the value for I_{C3} by using Equation 26.

$$I_{C3} = 4.95612080136771 \cdot 10^{-5} \tag{38}$$

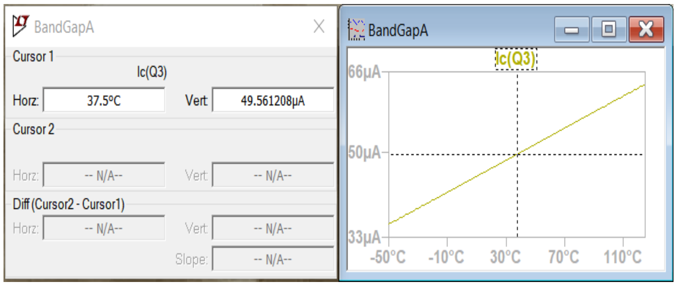

Fig. 8. The value of I_{C3} in LTSpiceXVII

We can now solve for V_{BE3} using Equation 31.

$$V_{BE3} = 0.618702107497214 \tag{39}$$

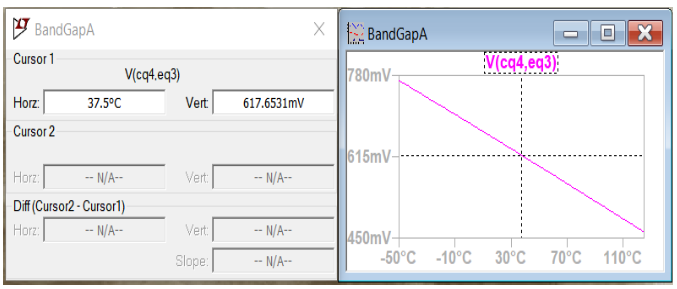

Fig. 9. The value of V_{BE3} in LTSpiceXVII

Then, solving for the V_{BE4}.

$$V_{BE4} = V_{BE3} - \Delta(V_{be}) \tag{40}$$

$$V_{BE4} = 0.679270859810729 \tag{41}$$

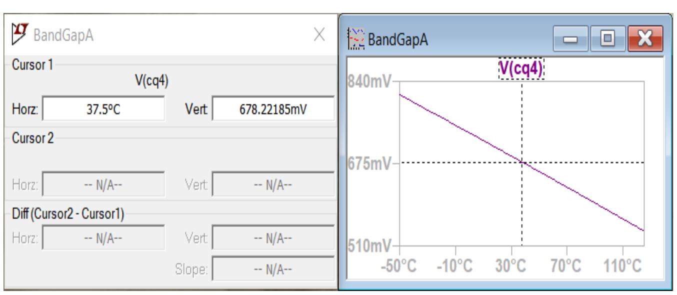

Fig. 10. The value of V_{BE4} in LTSpiceXVII

Then finally, the value of our V_{out} with its corresponding value using LTSpiceXVII.

$$V_{out} = 1.16516505602325 \tag{42}$$

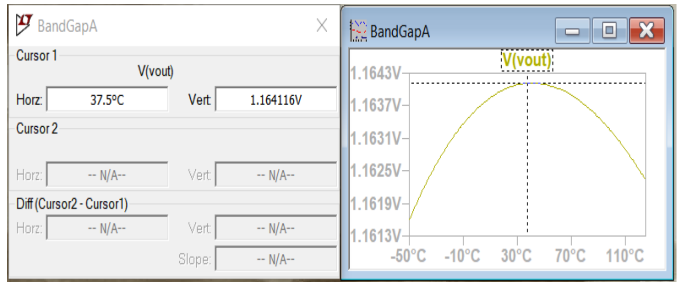

Fig. 11. The value of V_{out} in LTSpiceXVII

V. Conclusion

The Bandgap voltage reference circuit had been analyzed and the results were validated using the circuit simulator LTSpice. In designing a Bandgap circuit, a lot of parameters must be considered. Therefore, the current density and the change in the base-to- emitter voltages greatly affect each other. Every particular devices has unique current densities in which the emitter area affects the amount of collector current. Collector current densities can also be used to define the saturation currents of each transistor, which is assumed to be equal. Through this relationship, the difference between base-to-emitter voltages and two transistors was calculated. The voltage drop at the resistor R_1 is equal to the change in V_{be}. Using this relationship the current flowing through it can be calculated using Ohm's Law. Upon measuring the current at resistor R_1 using LTSpice, it is concluded that the voltage drop across it, is indeed the change in V_{be} of transistors Q3 and Q4. Using the current I_{R1}, the value of the collector current I_{C1} can be obtained. Since I_{C1} is also equal to the collector current I_{C2}, the output voltage V_{out}, is determined by getting the total voltages across R_1 and V_{ce4} of transistor Q4. The value of the output voltage was also validated using LTSpice which is equal to 1.16V, closely enough to the ideal value of 1.25V.

VI. Recommendation

The used of computing software changes the way we do analyzation, designing, simulation, and testing. LTSpiceXVII is one of the recommended software tools to do the job. Using LTSpiceXVII we can prove and gather data of such circuit design, even our own design. Verifying solutions for mathematical formulas using LTSpiceXVII. TextStudio[2] is also one of the recommended softwares in doing your paper in Latex format. This software is easy to use with regards to the used of mathematical symbols and other related field of science. In designing bandgap circuits, the most critical components, wherein the value must be precise, are the resistors R_1 and R_2. Also, the current density ratio of transistor Q3 and Q4 must be properly identified. The concepts discussed above can also be utilized in designing an effective bandgap circuit, especially if you already have a target output voltage. The methodology mentioned above can also be accustomed in circuit analysis for further circuit improvements, debugging and troubleshooting. The methodology aims to understand the effect of the value of R_1 to the value of the output voltage V_{out}. It also exposes that the current densities of the transistor are necessary to be well defined, for it affects the change in V_{be} between the transistors Q3 and Q4.

VII. Acknowledgment

The authors acknowledge Batangas State University, Association of Semiconductor and Electronics Manufacturing Engineers of the Philippines (ASEMEP), CBK Power Company Ltd., De La Salle Lipa Inc. for the support in promoting the development of knowledge and skills of the people in the industry and academic community. Special thanks to Dr. Celso B. Co, for the continous advancement and learnings that he provided for the authors to write this paper. The authors appreciate the emotional support and constant inspiration from their family. And above all, to the Creator who bestows the authors wisdom and knowledge.

VIII. Reference

References

[1] Anaconda, *Anaconda 5.3 For Windows Installer*, Copyright ©2018, Anaconda, Inc. All Rights Reserved, https://www.anaconda.com/download/ accessed on Nov 5, 2018, ,

[2] Benito van der Zander, Jan Sundermeyer, Daniel Braun, Tim Hoffmann (TeXstudio); Pascal Brachet (Texmaker); Luc Buant (QCodeEdit); Joel Amblard (html conversion), *TextStudio 2.12.10*, Copyright ©2018, Hosted by Source Forge https://sourceforge.net/projects/texstudio/ , https://www.texstudio.org/ accessed on Nov 5, 2018, ,

[3] Burovski, Evgeni, Matthew Brett, pv (avatar), Ralf Gommers, rkern, Travis Oliphant, Tyler reddy , *Scipy 1.1.0*, Copyright ©2018, Scipy Developrs. Created using Sphinx 1.6.6 http://www.sphinx-doc.org/en/master/, https://pypi.org/project/scipy/, https://pypi.org/project/scipy/#files accessed on Nov 6, 2018, ,

[4] Engelhardt, Mike, *LTspice XVII(x64)*, Copyright ©2018, Linear Technology Corporation is now part of Analog Devices, https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html accessed no Nov 5, 2018, ,

[5] Enthought, *Enthought Python Distribution*, Copyright ©2018, Enthought Inc. All Rights Reserved, https://www.enthought.com/product/canopy/ https://www.enthought.com/product/enthought-python-distribution/ accessed on Nov 6, 2018 , ,

[6] Haldane, Allan, cdavid, certik, Charles R. Harris, Jarrod Milman, jtaylor, Mathew Brett, Matti Picus, Ralf Gommers, rkern, Travis Oliphant, *Numpy 1.15.4*, Copyright ©2018, Numpy Developrs. Created using Sphinx 1.6.6 http://www.sphinx-doc.org/en/master/, https://pypi.org/project/numpy/#files accessed on Nov 6, 2018, ,

[7] Hunter, John, Darren Dale, Eric Firing, Michael Droettboom , *MatPlotLib*, Copyright ©2002-2012, 2012-2018, MatPlotLIb Development Team, https://matplotlib.org/# accessed on Nov 6, 2018, ,

[8] Knuth, Donald, *MikTex*, Copyright ©2018, Chrisstian Schenk, https://miktex.org/ accessed on Nov 5, 2018, ,

[9] Python Packaging Authority , *Python Package Index*, Copyright ©2014, Python Software Foundation, , https://www.pypa.io/en/latest/ https://pypi.org/ accessed on Nov 6, 2018 , , The Python Packaging Authority (PyPA) is a working group that maintains many of the relevant projects in Python packaging.

[10] Raybaut, Pierre, *Spyder 3.2.8, The Sicentific PYthon Development EnviRonment*, Copyright ©2018, The Spyder Project Contributors, Licensed under the terms of the MIT License, https://github.com/spyder-ide/spyder accessed on Nov 5, 2018, ,

[11] Raybaut, Pierre, Gabi Dvar (Maintainer), *Python(x,y)-2.7/10.0.exe*, Copyright ©2008,2015, GitHub, https://github.com/ https://python-xy.github.io/ accessed on Nov 6, 2018 , , History:Python(x, y) was concieved, developed and maintained by Pierre Raybaut since 2008 with the above goals. Gabi Davar joined the project as a maintainer since 2011. Pierre moved to work on other projects since 2013 leaving Gabi as the primary maintainer.

[12] Rossum, Guido van, *Python 3.6.5*, Copyright ©2018, Python Software Foundation, https://www.python.org/ accessed on Nov 5, 2018, ,

[13] Raphael Auphan, Puslishing Director, Scilab Team , *Scilab-6.0.1*, Copyright ©2018, ESI Group, , http://scilab.io/legal-notice/ http://scilab.io/ accessed on Nov 7, 2018, , Scilab is a free and open source software for engineers & scientists, with a long history (first release in 1994) and a growing community (100 000 downloads every months worldwide). http://scilab.io/about/open-source/

[14] Sympy Development Team, *SymPy 1.3*, Copyright ©2018, Github, https://github.com/sympy/sympy accessed on Nov 6, 2018, ,

[15] A. Paul Brokaw, *How to Make a Bandgap Voltage Reference in One Easy Lesson*, Copyright ©2011, Integrated Device Technology, https://www.idt.com/document/whp/how-make-bandgap-voltage-reference-one-easy-lesson-paul-brokaw accessed on Nov 10, 2018 , ,

IX. About the Authors

Engr. Jaycee R. Jalos, ECE He earned his Bachelors Degree in Laguna State Polytechnic University in 2009. After he graduated, he took the ECE Licensure Exam in 2009 and passed the said exam. He is currently employed in CBK Power Company Limited as a Plant Automation and Communications Engineer II since 2010.

He is currently taking his Masters Degree in Electronics Engineering in Batangas State University, Batangas City, Batangas.

Engr. Sarah Jane E. Delgado, ECE She got her degree in Bachelor of Science in Electronics Engineering from Batangas State University in April 2013. She passed the ECE Licensure Examination in 2014 and is currently employed as a college instructor Lecturer I in De La Salle Lipa.

She's been an academe since 2015 under the Electronics Engineering Department of College of Information Technology and Engineering. She is currently taking up her Master's Degree in Electronics and Communications Engineering in Batangas State University.