MEMRISTOR SPICE MODEL FOR DESIGNING INTEGRATED CIRCUITS

This thesis is presented in partial fulfillment for the award of the
Bachelor of Engineering (Hons.) in Electronic Engineering
UNIVERSITI TEKNOLOGI MARA MALAYSIA

AHMAD FUAD BIN ADZMI

FACULTY OF ELECTRICAL ENGINEERING
UNIVERSITI TEKNOLOGI MARA MALAYSIA
40450 SHAH ALAM, SELANGOR, MALAYSIA

JANUARY 2013

ACKNOWLEDGEMENT

I would like to express my gratitude to Allah S.W.T. for giving me the opportunity to complete this Final Year Project. I am deeply indebted to individuals who, directly or indirectly, are responsible for this project.

This paper has been presented in the IEEE Student Conference Research & Development (SCORED 2012) and IEEE International Conference on Control System (ICCSCE 2012). The author would like to thank Universiti Teknologi MARA for funding the research work through the Excellence Fund Grant 600-RMI/ST/DANA 5/3/RIF(360/2012).

I am most grateful to the most kind heartedness supervisor Dr. Wan Fazlida Hanim for her guidance, advice, ideas and opinions in this project.Not forget to the panel of seminar presentation, Pn. Tuan Nor Jihan and Cik Raudah with their superior guidance, information and ideas for this project become abundance. Not forget to Dalibor Biolek, member of the CAS/COM Czech National Group of IEEE, and the president of Commission C of the URSI National Committee for Czech Republic, for answering my queries through email and helping me with the simulations. May Almighty Allah bless and give reward to all of them for their generosity.

A part from that I would like to thanks to my parents who have been so tolerant and supports me all these years. Thanks for their encouragement, love and supports that they had given to me. Last but not least, to my best friends Afiq Azman, Zubir Zulkifly, Faiz Omar and all those who supported me in any aspect and lending a hand during the completion of this project.

ABSTRACT

A SPICE model of the memristor would be a useful tool to analyze circuit behavior to help in developing applications of this memristor via simulation. In this paper, we incorporate a memristor SPICE model for designing memristor circuit which is focused on non-linear model and implementation with analog circuits. To describe the real device operation in a circuit simulation, a SPICE model is required. We incorporate the memristor with various window functions that have been proposed in non linear ion drift memristor devices. In investigating and characterizing the behavioral properties of memristor devices, the circuit analyses of the proposed memristor models are then studied. Then, a simple analog circuit is constructed using the model specifically a simple integrator and differentiator op-amp circuit and comparison is made between memristor implemented circuit and normal integrated circuit. The research shows the difference of memristor model and the possibilities of implementing the memristor in analog circuit. The memristor model has been simulated to give a good approximation to the real memristor. Memristor model based on Strukov window function gives minimum power dissipation with 120μW. While memristor model based on Joglekar window function give the fastest time of change of the resistance for each memristor model with 320ms. It is 49.2% faster than memristor model based on Strukov window function but in term of controlling the device, memristor model based on Prodromakis window function give a better result. Also, memristor model based on Prodromakis window function only suffer 0.11% slower speed than the memristor model based on Strukov window function. It give much better result with great extend of scalability and flexibility to the model. It also found that, changing the frequency and control parameter p will also affect the memristance behavior. In implementation of memristor with analog circuit, it shows that the existence of the memristor give better functionality to the application.

Keywords— memristor, SPICE model, non linear, window functions, analog circuit.

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

In 2008, a research team at HP announced that they had discovered a new circuit element which called the memristor [1]. This discovery was arguably as important as that of the resistor, capacitor, or inductor. Since the HP announcement, researchers and engineering professionals have been trying to understand the new technology and to develop potential applications. It is useful to have a computer model of the memristor as a tool for speeding-up the analysis of the behavior and developing applications of this interesting circuit element via simulation experiments [2]. To do this, they need robust circuit simulation software that allows them to explore how memristors work alongside existing components. With regard to the fact that memristor is specified as the fourth, newly discovered passive circuit element, supplementing the well-known R, C, L triplet (resistor, capacitor, inductor), it is logical to suggest extending the model libraries of SPICE-family simulating programs just with the model of the memristor.