

Development of a D.C Circuit Analysis Software Using Microsoft Visual C#.Net

¹B.O. Omijeh And ²M. Ehikhamenle

^{1,2} Department of Electronic and Computer Engineering, University of Port Harcourt, Rivers State, Nigeria.

Abstract: In this paper, the development of D.C circuit simulation software, using Microsoft visual C#.net, has been achieved. This paper aims at (i) analysing a purely resistive planar circuit, (ii) displaying current direction as they flow through components and (iii) having a user-friendly Graphic User Interface (GUI). The methodology employed in this work includes: (i) equations, (ii) pseudo code, (iii) algorithm, (iv) flowchart and (v) Microsoft C#.Net codes. The solutions obtained from the software correlate with those generated manually. The software has an efficiency of 99.3396% which makes it very reliable for circuit analysis and simulation.

Keywords: D.C circuit, Circuit simulation software, Microsoft Visual C#.net, CiRSiS

I. Introduction

A circuit basically, is a collection of interconnected components. A Direct Current circuit (D.C circuit) is an electrical circuit that consists of any combination of constant voltage sources, constant current sources, and resistors (Malvino ,1993). A Circuit can be analyzed through various techniques, which include: Mesh analysis, Nodal analysis, Thevenin's theorem, Superposition theorem, etc. Computer programming is the process of instructing the computer on tasks to perform via a programming language which involves various sets of codes that the computer understands. The DC circuit simulation software is developed using Microsoft Visual C#.net. The Objective of this project is to design a D.C circuit simulation software using Microsoft Visual C#.net (Watson et al,2010). It will display a current-voltage-power table of each component and the direction of current flow through each component of a purely resistive D.C planar circuit. The circuits are within the scope of a ladder circuit (with a maximum of 4 loops), a bridge circuit, and an Operational Amplifier circuit. Some basic circuit theorems (Mesh analysis and Nodal analysis) were applied in the modelling of this software (Omijeh, 2009).

Over the years, simulation software has been developed. This section contains some of the various simulation software used both commercially and in the classroom for academic purpose. Some of these simulation software are briefly introduced (Butow ,2002; Bart ,2011).

SPICE (Simulated Program with Integrated Circuit Emphasis) was one of the first simulation softwares to evolve. SPICE was developed at the Electronics Research Laboratory of the University of California, Berkeley by Laurence Nagel with direction from his research advisor, Prof. Donald Pederson (Stroud ,2003 ;Wikipedia.,2014),.

Another simulation software of note is PSPICE. PSPICE is a SPICE analog circuit and digital logic simulation program for Microsoft Windows (www.cadencedesignsystem.com, 2014). Amongst these softwares is also TINA. TINA is a powerful yet affordable circuit simulation and PCB design software package for analyzing, designing, and real time testing of analog, digital, HDL, MCU, and mixed electronic circuits and their PCB layouts (www.tina.com, 2008). Proteus is a software for microprocessor simulation, schematic capture, and printed circuit board (PCB) design. It is developed by Labcenter Electronics. It was founded in 1988. Proteus professional software is a PCB design software integrated with the simulation of the circuit you design. It is integrated with real time simulation of the electronic circuit and test whether your designed circuit is working properly or not.

II. Theoretical Background

For this work, circuits are analysed using Mesh analysis and Nodal analysis as earlier stated. These analysis are briefly discussed (Sadiku, 2004; Mehta,2006)

MESH ANALYSIS

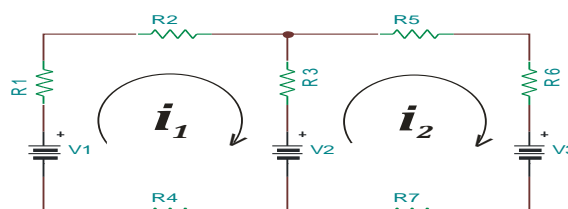


Fig. 1: A 2-loop system of ladder circuit

Applying Kirchhoff's voltage law to fig. 1, we have,

From mesh 1, $(R_1 + R_2 + R_3 + R_4)i_1 - R_3 i_2 = V_1 - V_2$

From mesh 2, $-R_3 i_1 + (R_3 + R_5 + R_6 + R_7)i_2 = V_2 - V_3$

$$\begin{pmatrix} R_1 + R_2 + R_3 + R_4 & -R_3 \\ -R_3 & R_3 + R_5 + R_6 + R_7 \end{pmatrix} \begin{pmatrix} i_1 \\ i_2 \end{pmatrix} = \begin{pmatrix} V_1 - V_2 \\ V_2 - V_3 \end{pmatrix}$$

From Ohm's law, $V = IR$... (1)

$I = R^{-1}V$... (2)

Where **I**, **R** and **V** stand for the current, resistance and voltage matrices.

Or from **Cramer's rule**, we obtain the determinants

$$\Delta = \begin{vmatrix} R_1 + R_2 + R_3 + R_4 & -R_3 \\ -R_3 & R_3 + R_5 + R_6 + R_7 \end{vmatrix} \Delta$$

$$= (R_1 + R_2 + R_3 + R_4)(R_3 + R_5 + R_6 + R_7) - R_3 R_3$$

$$\Delta_1 = \begin{vmatrix} V_1 - V_2 & -R_3 \\ V_2 - V_3 & R_3 + R_5 + R_6 + R_7 \end{vmatrix}$$

$$\Delta_1 = (V_1 - V_2)(R_3 + R_5 + R_6 + R_7) - \{-R_3(V_2 - V_3)\}$$

$$i_1 = \frac{\Delta_1}{\Delta}; i_2 = \frac{\Delta_2}{\Delta}$$

Current equations: $I_{R1} = i_1; I_{R2} = i_1; I_{R3} = i_1 - i_2; I_{R4} = i_1; I_{R5} = i_2; I_{R6} = i_2; I_{R7} = i_2$

Voltage equations: From Ohm's law, $V_{Rn} = I_{Rn} R_n$... (3)

$V_{R1} = I_{R1} R_1; V_{R2} = I_{R2} R_2; V_{R3} = I_{R3} R_3; V_{R4} = I_{R4} R_4; V_{R5} = I_{R5} R_5; V_{R6} = I_{R6} R_6; V_{R7} = I_{R7} R_7$

Power equations: $P = IV; P_{Rn} = I_{Rn} V_{Rn}$... (4)

$P_{R1} = I_{R1} V_{R1}; P_{R2} = I_{R2} V_{R2}; P_{R3} = I_{R3} V_{R3}; P_{R4} = I_{R4} V_{R4}; P_{R5} = I_{R5} V_{R5}; P_{R6} = I_{R6} V_{R6}; P_{R7} = I_{R7} V_{R7}$

Bridge Analysis

This system of network also incorporates mesh analysis in finding its solutions.

Consider the circuit below-

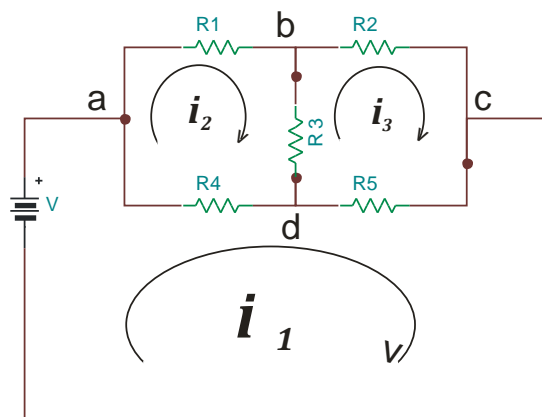


Fig 2: A bridge circuit

From mesh adca, $(R_4 + R_5)i_1 - R_4 i_2 - R_5 i_3 = V$

From mesh abda, $-R_4 i_1 + (R_1 + R_3 + R_4)i_2 - R_3 i_3 = 0$

From mesh bcd, $-R_5 i_1 - R_3 i_2 + (R_2 + R_3 + R_5)i_3 = 0$ From ohm's law, $I = R^{-1}V$, we obtain our currents.

Thus,

Current equations : $I_{R1} = i_2; I_{R2} = i_3; I_{R3} = i_2 - i_3; I_{R4} = i_1 - i_2; I_{R5} = i_1 - i_3$

Voltage equations: The same as equation 3; **Power equations :** The same as equation 4

III. Design Methodology

The methodology employed in this work is shown in the block diagram below.

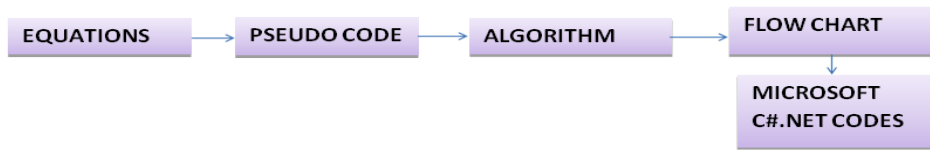


Fig 3. Organogram showing the different sections of the software

Equations for Design Consideration:

The loop current can be determined from equation 3.2. The currents across each resistor (I_{R1} - I_{R7}) gotten from these loop currents are given section 3.1.1. The voltages across each resistor (V_{R1} - V_{R7}) are given in eqn (3.3). The Power dissipated by each resistor (P_{R1} - P_{R7}) are given in eqn (3.4).

Pseudocode for Design:

- Start by selecting the Two Loop Subsection
- Input values for different components if not deleted.
- Compute the current flowing through each Resistor.
- Compute the Voltage passing through each Resistor.
- Compute the Power dissipated by each Resistor.
- Print “Current flowing through each Resistor”, “Voltage across each Resistor”, “Power dissipated by each Resistor”.
- Stop

Algorithm for Design:

Step 1: Start

Step 2: Input $R_1, R_2, R_3, R_4, R_5, R_6, R_7, V_1, V_2, V_3$

Step 3: Compute $I_{R1}, I_{R2}, I_{R3}, I_{R4}, I_{R5}, I_{R6}, I_{R7}$.

Step 4: Compute $V_{R1}, V_{R2}, V_{R3}, V_{R4}, V_{R5}, V_{R6}, V_{R7}$

Step 5: Compute $P_{R1}, P_{R2}, P_{R3}, P_{R4}, P_{R5}, P_{R6}, P_{R7}$

Step 6: Print “ $I_{R1}, I_{R2}, I_{R3}, I_{R4}, I_{R5}, I_{R6}, I_{R7}$ ”; “ $V_{R1}, V_{R2}, V_{R3}, V_{R4}, V_{R5}, V_{R6}, V_{R7}$ ”; “ $P_{R1}, P_{R2}, P_{R3}, P_{R4}, P_{R5}, P_{R6}, P_{R7}$ ”.

Step 7: Stop

Flowchart For Design:

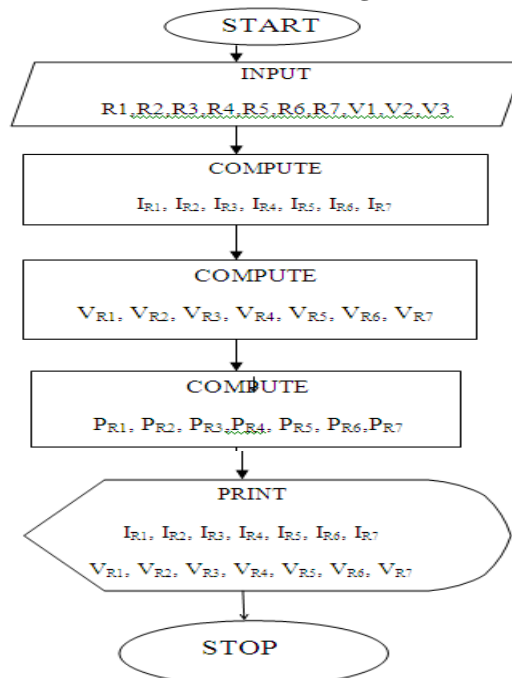


Fig.4: Flowchart of the Two Loop Circuit Subsection.

IV. Results And Discussion

As stated earlier, this software is able to simulate ladder, bridge and op amp circuits. Some examples of these circuits shall be analysed mathematically and then simulated using this software; and then their results are compared.

Ladder circuit example (2 Loop Circuit)

Consider the circuit below,

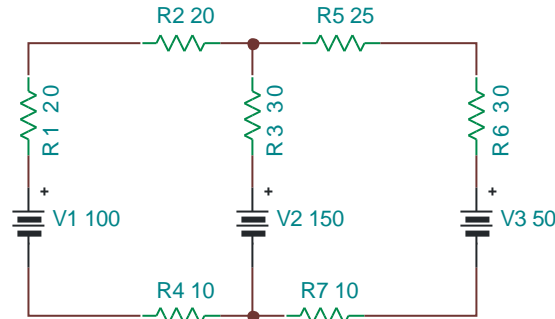


Fig. 5: A simple 2-loop ladder circuit

We are required to find the current flowing through each resistor, the voltage across each resistor and the power dissipated by each of these resistors. We start by introducing the loop currents inside the loops. Thus we have-

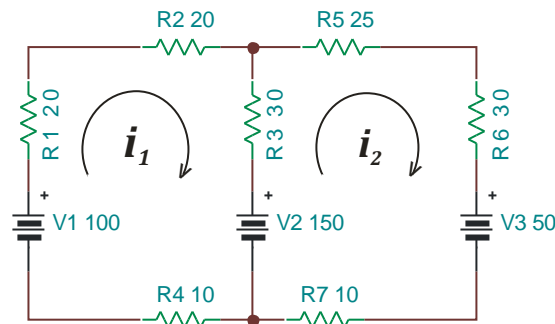


Fig. 6:A 2-loop circuit with loop currents

From mesh 1, $80i_1 - 30i_2 = -50$

From mesh 2, $-30i_1 + 95i_2 = 100$

By solving the last two equations simultaneously, we obtain

$i_1 = -0.26119A$ and $i_2 = 0.9701A$. Recall from section 3.1.1

$I_{R1} = i_1; I_{R2} = i_1; I_{R3} = i_1 - i_2; I_{R4} = i_1; I_{R5} = i_2; I_{R6} = i_2; I_{R7} = i_2$

So, $I_{R1} = I_{R2} = I_{R4} = -0.26119A; I_{R3} = -1.23129A; I_{R5} = I_{R6} = I_{R7} = 0.9701A$

To find the voltages across the resistors, recall eqn (3.3), i.e. $V_{Rn} = I_{Rn}R_n$

$V_{R1} = -5.2238V; V_{R2} = -5.2238V; V_{R3} = -36.9387V; V_{R4} = -2.6119V; V_{R5} = 24.2525V; V_{R6} = 29.103V; V_{R7} = 9.701V$

To obtain the power dissipated by each resistor, we recall from eqn (3.4) that $P_{Rn} = I_{Rn}V_{Rn}$ and so, we have that

$P_{R1} = 1.3644W; P_{R2} = 1.3644W; P_{R3} = ; P_{R4} = ; P_{R5} = ; P_{R6} = ; P_{R7} =$

Below are figures showing how the problem given in fig. 5. is solved using CiRSiS

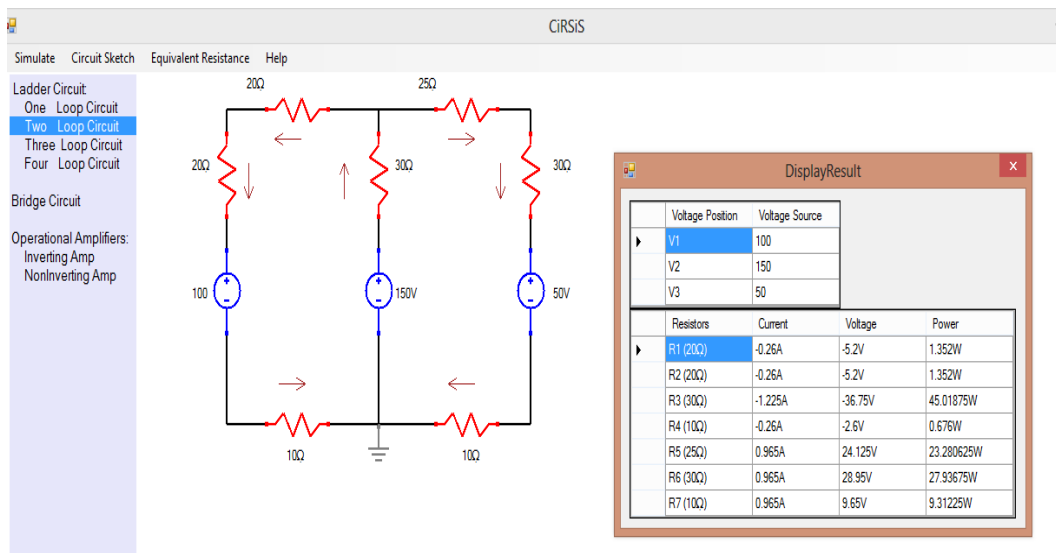


Fig. 7: A solution to fig.5 in CiRSiS showing current direction

Fig.7above has a circuit diagram on the left and a tabled result on the right. The simulated 2 loop circuit with the direction of the current flow in each component is displayed. Table in Fig.7 shows a current-voltage-power tabled result of each component. It is expanded below.

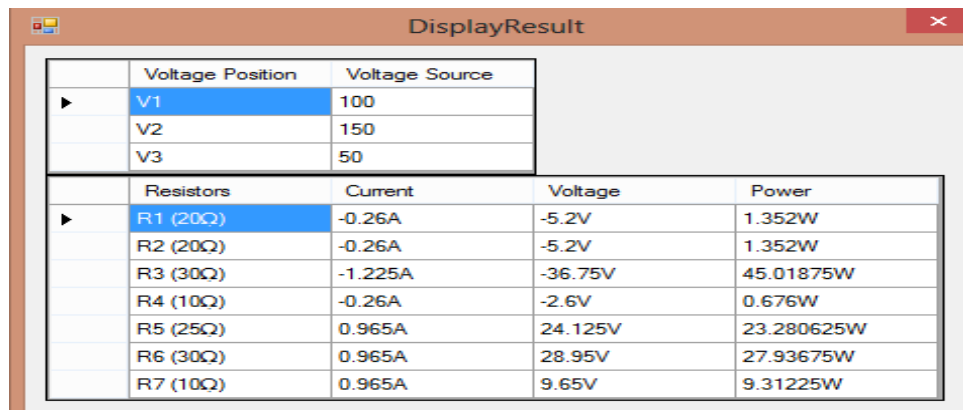


Fig. 8: An expanded form of table in Fig .7

As can be seen, the manually calculated values and the simulated values correlate. In order to ascertain the efficiency of the software, a percentage error analysis shall be performed comparing the manually calculated results with the simulated result of one of the examples taken so far. This is shown in the table below. In order to ascertain the efficiency of the software, a percentage error analysis is performed comparing the manually calculated results with the simulated result of one of the examples taken so far. This is shown in the table below. The subscript “C” represents the calculated values; the subscript “S” represents the simulated values; the subscript “P” represents the percentage error while the subscript “eff” represents the efficiency.

Table 1 Validation of software using the 2 Loop circuit example

components	CURRENT VALUES				VOLTAGE VALUES				POWER VALUES					
	I _C (A)	I _S (A)	I _P (%)	I _{EFF}	V _C (V)	V _S (V)	V _P (%)	V _{EFF}	P _C (W)	P _S (W)	P _P (%)	P _{EFF}		
R1	-0.261	-0.260	0.421	99.579	-5.223	-5.200	0.455	99.545	1.364	1.352	0.908	99.092		
R2	-0.261	-0.260	0.421	99.579	-5.223	-5.200	0.455	99.545	1.364	1.352	0.908	99.092		
R3	-1.231	-1.225	0.503	99.497	-36.938	-36.75	0.510	99.490	45.482	45.018	1.019	98.981		
R4	-0.261	-0.260	0.421	99.579	-2.611	-2.600	0.455	99.545	0.682	0.676	0.908	99.092		
R5	0.970	0.965	0.525	99.475	24.252	24.125	0.529	99.471	23.527	23.280	1.048	98.952		
R6	0.970	0.965	0.525	99.475	29.103	28.950	0.525	99.475	28.282	27.936	1.223	98.777		
R7	0.970	0.965	0.525	99.475	9.701	9.65	0.525	99.475	9.410	9.312	1.048	98.952		
AVERAGE I _{EFF} (%)				99.522	AVERAGE V _{EFF} (%)				99.506	AVERAGE P _{EFF} (%)				98.990
AVERAGE EFFICIENCY(%)								99.339						

The percentage error is gotten using the formula:

$$\frac{((\text{Calculated value} - \text{Simulated value})/\text{Calculated value}) \times 100\%}{}$$

The efficiency is given as: $100 - \text{Percentage Error}$

From the table, the average efficiency is given as:

$$\text{Total efficiency} = (99.522+99.5059+98.990)/3 = 99.339\%$$

As proven above with the 2 loop example, the efficiency of CiRSiS is 99.339% which is very high and thus is suitable for simulating circuits.

V. Conclusion

From the objectives of this work, the analyses of a purely resistive D.C planar circuit has been made easy, without one having to go through the tedious mathematics involved in solving more than two(2) linear equation manually. Understanding of current behaviour is also enhanced, haven made provision for a display of current direction for any circuit within the scope.

CiRSiS has a very user friendly environment and thus can be properly used for academic purposes. Also the expenses involved in carrying out trial and error during the design and construction of D.C circuits can be avoided since trial and error can be done with CiRSiS, spending less time and money. Therefore, the software has not only made D.C circuit analyses easy by providing a user friendly environment and display of result table (containing values of current through, voltage across and power dissipated by various circuit components) and direction of current flow, but is also of economic value.

References

- [1]. Bart D.S. (2011): C# 4.0 Unleashed, Sams, Indianapolis, Indiana, ISBN:13-978-0-672-33079-7
- [2]. Butow .E and Ryan .T (2002): C#. Your visual blueprint for building .net applications, Maran graphics and Hungry minds, New-York, ISBN: 0-7645-3601-X
- [3]. Malvino (1993): Electronic Principles 5th edition, Macmillan/McGraw-Hill, ISBN 0-02-800846-6.
- [4]. Mehta V. K. and Mehta R. (2006): Basic Electrical Engineering, S. Chand & Company Ltd, New Belhi, India, ISBN 81-219-0871-X.
- [5]. Mitchell E. S. (2007): Basic Electronics 10th edition, McGraw-Hill, ISBN-13 978-0-07-297475-1
- [6]. Omijeh B.O. (2009): Circuit Analysis & Simulation (with MATLAB and TINA applications), Joint Heirs publishers Nig. Ltd, ISBN 978-978-48383-4-7.
- [7]. Sadiku M.N.O. and Charles K.A. (2004): Fundamentals of Electric Circuits, McGraw Hill, ISBN 0-07-304718-X.
- [8]. Solis D. (2008): Illustrated C# 2008, A press, USA, ISBN-13:978-1-59059-954-9
- [9]. Stroud K.A. and Dexter J. Booth (2003): Advanced Engineering Mathematics 4th edition, Palgrave Macmillian, Great Britain, ISBN-13:978-1-4039-0312-9
- [10]. Watson K., Nagel C., Pedersen J.H., Reid J.D., Skinner M. (2010): Beginning visual C# 2010, Wrox, ISBN 978-0-470-50226-6
- [11]. Wikipedia, 2014:SPICE from <http://en.wikipedia.org/wiki/SPICE>, February 14th,2014.

Authors' Biography

Bourdillon .O. Omijeh holds a B.Eng degree in Electrical/Electronic Engineering, M.Eng and Ph.D



.Degrees in Electronics/Telecommunications Engineering from the University of Port Harcourt & Ambrose Alli University (A.A.U), Ekpoma respectively. His research areas include: Artificial Intelligence, Robotics, Embedded Systems Design, Modeling and Simulation of Dynamic systems, Intelligent Metering Systems, Automated Controls, Telecommunications and ICT. He has over thirty (30) technical papers & publications in reputable International learned Journals and also, has developed over ten(10) application Software. He is a member, Institute of Electronics and Electrical Engineers (MIEEE), Member, Nigeria Society of Engineers; and also, a Registered Engineer (COREN). He is currently a Senior Lecturer & pioneer HOD, Department of Electronic and Computer Engineering, University of Port Harcourt, Nigeria; and also, a consultant to companies & Institutions. He is happily married with Children

Ehikhamenle M. graduated with BEng (Second class upper honors) Electrical/Electronics Engineering from



Ambrose Alli University Ekpoma in 2007.He obtained his Master degree in Electrical/Electronics Engineering from University of Benin (UNIBEN), specializing in Electronics and Telecommunications in 2010. He was Head of Department Essential services, Works and Services Directorate Ambrose Alli University Ekpoma in 2012. He is presently a lecturer in Electronic and Computer Engineering Department University of Port-Harcourt (UNIPORT).He has taught courses in electronics and telecommunications Engineering. His research interest include: Electronic, telecommunication, power electronic CAD, ICT and Control Systems.