Freshman Experience Course in Electrical and Computer Engineering Technology Emphasizing Computation, Simulation, Mathematical Modeling, and Measurements

Prof. Omer Farook, Purdue University Northwest (College of Technology)

Omer Farook is a member of the faculty of Electrical and Computer Engineering Technology at Purdue University, Nothwest. Farook received the diploma of licentiate in mechanical engineering and B.S.M.E. in 1970 and 1972, respectively. He further received B.S.E.E. and M.S.E.E. in 1978 and 1983, respectively, from Illinois Institute of Technology. Farook's current interests are in the areas of embedded system design, hardware-software interfacing, digital communication, networking, image processing, and biometrics, C++, Python, PHP and Java languages. He has a keen interest in pedagogy and instruction delivery methods related to distance learning. He has a deep commitment to social justice and in achieving economic and educational equity.

Dr. Jai P. Agrawal, Purdue University Northwest

Jai P. Agrawal is a professor in electrical and computer engineering technology at Purdue University Northwest. He received his Ph.D. in electrical engineering from University of Illinois, Chicago, in 1991, dissertation in power electronics. He also received M.S. and B.S. degrees in electrical engineering from Indian Institute of Technology, Kanpur, India, in 1970 and 1968, respectively. His expertise includes analog and digital electronics design, power electronics, and optical/wireless networking systems. He has designed several models of high frequency oscilloscopes and other electronic test and measuring instruments as an entrepreneur. He has delivered invited short courses in Penang, Malaysia and Singapore. He is also the author of a textbook in power electronics, published by Prentice-Hall, Inc. His other books are, Analog and digital communication laboratory, and First course in Digital Control, published by Creatspace (Amazon). His professional career is equally divided in academia and industry. He has authored several research papers in IEEE journals and conferences. His current research is focused on renewable energy technology and wireless power transfer.

Dr. Athula Kulatunga, Purdue University Northwest

N. Athula Kulatunga, Ph.D., CEM is the Department Head of Engineering Technology at Purdue University Northwest in Hammond, Indiana, USA. Before becoming the head, he was the George W McNelly Professor in Electrical and Computer Engineering Technology at Purdue University, West Lafayette, Indiana, USA. He received a Ph.d. from Purdue University in 1995. He is the founder and director of two industry sponsored applied research labs: Power Electronics Development and Applications Lab (PEDAL) and Smart Meter Integration Lab (SMIL). He is the Principal Investigator of one of 10 Global Innovation projects funded by the US department of State, Rapid, Smart Grid Impact RSGI), partnering with DeMontfort University in Leicester, UK, and UNESP in Sao Paulo, Brazil. He has been a Certified Energy Manager (CEM) since 1998.

Prof. Ashfaq Ahmed P.E., Purdue University Northwest (College of Technology)

Ashfaq Ahmed is a professor of Electrical and Computer Engineering Technology at Purdue University Northwest. Ahmed received his bachelor's of science degree in electrical engineering from the University of Karachi in 1973 and master's of applied science degree in 1978 from University of Waterloo. He is the author of a textbook on power electronics, published by Prentice-Hall. He is a registered Professional Engineer in the state of Indiana. He is senior member of IEEE. Ahmed's current interests include embedded system design, electric vehicle, and VHDL design.

Dr. Wangling Yu, Purdue University Northwest

Dr. Wangling Yu is an assistant professor in the Electrical & Computer Engineering Technology Department of the Purdue University Northwest. He was a test engineer over 15 years, providing technical leadership in the certification, testing and evaluation of custom integrated security systems. He received his PhD degree in Electrical Engineering from the City University of New York in 1992, specializing in control theory and electronic technology.

Prof. Yoonill Lee, Purdue University Northwest

YOONILL LEE is currently an assistant professor of Electrical Engineering Technology at the Purdue University Northwest. He earned his B.S. and M.S. in Electrical Engineering from Yonsei University in Seoul, South Korea, in 1984, 1986, respectively, and M.S. in Electrical Engineering from Oklahoma State University in Stillwater, OK in 1992, and Ph.D. in Electrical Engineering from the University of South Florida in Tampa, FL in 2001. He worked at Cisco Systems as a communications engineer. His research interests include CDMA, Multi-Carrier Systems, and MIMO technology. Dr. Lee can be reached at lee2273@pnw.edu.

Mr. Hassan Abdullah Alibrahim, Purdue University Northwest

I have an associates degree with first honor in Electrical Engineering Technology from Jubail Industrial College, Saudi Arabia. Now, I am studying at Purdue University Northwest to accomplish my bachelor degree in Electrical and Computer Engineering Technology (Senior classification). An active member in the national honor society for engineering technology, Tau Alpha Pi, at Purdue University Northwest. I work as teaching assistant and supplemental instructor in my school and planing to start my Master degree next semester.

Freshman Experience Course in Electrical and Computer Engineering Technology Emphasizing Computation, Simulation, Mathematical Modeling and Measurements

Abstract

The paper expounds the challenges and rewards of revamping the freshman engineering / Engineering Technology curriculum with the notion of introducing computational analysis with the help of Matlab[1]. The paper discusses in details the five areas of 1) Network theory, 2) Simulation by the help of Multisim[2], 3) Computation and mathematical modelling by utilization of Matlab, 4) Physical implementations of the circuits and 5) A gentle introduction to microcontrollers by utilizing Arduino[4] Open source board.

The pedagogy of the course delivery is based on “Interactive Learning Model”, utilizing the methodology of Outcome Based Education. Outcome Based Education’s end result is the students’ design projects performed at the end of the course. The course is conducted in a lab or studio like settings, that integrates both lecture and laboratory work in the same settings. The paper elaborates the benefits derived through the pedagogical approaches of keeping the learner actively engaged in all aspects of discovery and design. The course interactively involves the learner in directing and defining the material under discourse.

Paper provides a road map and serves as pointer to host of design tools that are available that can be incorporated in a similar freshman course offering for Engineering Technology and Engineering programs. Paper provides the content details in terms of topics covered as well as all the labs performed during the course. The paper presents the details of pedagogical approach that was implemented in successful implementation of this course.

I. Introduction

Traditionally Electrical and Computer Engineering /Technology (ECE/T) curriculum start with a freshman experience course. In ECET program the faculty has taken the opportunity to revamp this course. The whole faculty of Electrical and Computer Engineering Technology program numbering into eight faculty members actively participated in this pedagogical task. Realizing one of the weakness of the curriculum is that mathematical modeling is utilized, without providing a formal introduction to it. A number of courses throughout the curriculum utilizes computational and modeling tools like Matlab, without a formal introductory course. The paper explains how this weakness is addressed head on in the freshman experience course. The course provides the freshmen students with the concurrent experiences in computation, simulation, mathematical modeling and actual measurements of electrical parameters with the physical implementation of the circuits.

The paper presents here the practice followed by the authors in the Department of Electrical and Computer Engineering Technology, for over two decade of successful course delivery utilizing Interactive Learning Model with Outcome Based Education methodology that translates into effective learning from the students’ perspective.

II. Course Background

The subject course ECET 10000, Introduction to Electrical and Computer Engineering Technology is taken by all the freshman students opting for the 4 years BS in Electrical and Computer Engineering program. This is

The course is offered in 2:3:3 (2 Lecture: 3 Lab 3 Credits Pattern), there is no prerequisites for the course other than freshman standing in ECET Program, and the only co-requisite is Math14800, Algebra and Trigonometry. The lecture portion of the course was recorded and delivered in two campuses concurrently, while lab portion was performed independently in an instructor directed manner.

The faculty by choice have taken upon themselves to make the changes, the essence of which is reflected in this paper. After a lot of deliberation the curriculum committee agreed to make the changes to the curriculum that essentially, prepares the student to function in the market place as the system designer. It was realized that the ECET curriculum should impart to the graduating student enough exposure to pursue a clear comprehension in the areas of theoretical computation, mathematical modeling, simulation and physical measurement. In the following pages we are going to elaborate the practice with regard to the five areas of, 1) Network theory, 2) Simulation utilizing Multisim software, 3) Computation and mathematical modelling by utilization of Matlab, and 4) Physical implementations of the circuits, and pertinent measurements utilizing power supplies, multi-meters and scopes. The course then culminates with an 5) Exposure to microcontrollers by utilizing Arduino Open source board. Freshmen in ECET starts their career-bound path with this course.

III. Network Theory /Circuit Analysis

The first nine chapters of the of Boylestad's book[5] have been slated for the course. The topics covered after a brief introduction have been, 1) Voltage and Current, 2) Resistance, 3) Ohm's Law, Power and Energy, 4) Series DC circuits, 5) Parallel DC circuits, 6)Series – Parallel Circuits, 7) Method of Analysis and 8) Network Theorems.

Hand computation in the traditional sense was always the starting point and students were encourage to verify the results arrived through simulation utilizing Multisim, Matlab computations, and physical lab measurements.

IV. Simulation Utilizing Multisim Software

In this introductory course students were very receptive in learning and using Multisim software during the computation of the lab exercises. With the progression in computer technology several electronics laboratory simulation software packages are available to academia and industry. The Multisim software developed by the Electronics Workbench and National Instruments is a popular circuit capture and simulation software that is frequently used for education and training. With power and flexibility provided by Multisim students gain the advantages of an industry-caliber, easy-to-use circuit simulator. Multisim includes powerful virtual instruments, which are simulated instruments found in the laboratory such as oscilloscopes, multi-meters, and function generators, among many others[6]. Multisim is an industry-standard, best-in-class SPICE simulation environment. It is the cornerstone of the NI circuits teaching solution to build expertise through practical application in designing, prototyping, and testing electrical circuits[7]. However, with context to this course students only utilize Multisim's ability to draw schematic diagram and use the virtual instrumentation to capture required data from the respective instruments.

V. Computation and Mathematical Modelling by Utilization of Matlab

Matlab by Mathworks has become the standard computational engine in most academic and industrial settings. As such our freshmen students are trained to be competent and efficient in its use starting with this freshman class and later continuing in as many as six different classes during their 4 years of degree program.

Solving sets of linear system equations in circuits is often tedious and something not liked by our students. This task when delineated through Matlab becomes easy and is readily accepted and used. The following examples demonstrates how Matlab has been used by the freshman class students for circuit analysis problems. Please note the schematics are drawn by the students using Multisim.

V.a. Books and Topics

The class was provided with Matlab lectures along with handouts using the two text books, 1) Getting Started with Matlab by Rudra Pratap[8] and 2) Learning Matlab Superfast by Brian L.F. Daku[9].The following topics were covered:

1. Basics of Matlab
2. General Useful Commands and Operations
3. Matlab Windows
4. Input /Outputs (Formatting)
5. File Types and Scripting
6. Introduction to Matrix and Vectors
7. Matrix Manipulations
8. Arithmetic Operations

V.b. Matrices in Electrical Engineering[10]

Use of Matlab in Circuit Analysis in solving system's Linear Equation is included here, as it is the core theme to find unknown elements' values of our electrical circuits.
We will take the following circuit for our analysis, and determine the currents flowing in each branch of the circuit. The value of the supply voltage is 100V and the values of the resistors is as per the circuit.

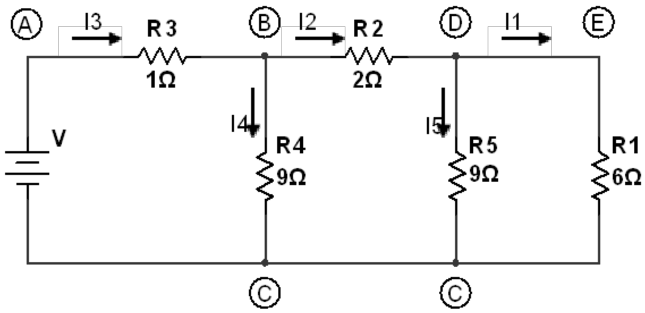

Figure 1: Schematic diagram for a series-parallel circuit

If we start with the nodes and write equations we notice the following:
The current through the R_4 is equal to $I_4 = I_3 – I_2$.
The Current through R_5 is equal to $I_5 = I_2 – I_1$.

The Loop equations:
ABC is: $-100 + 1 X I_3 + 9 X I_4 = 0$, substituting for $I_4 = I_3 – I_2$ and simplifying we have,
$10 I_3 – 9 I_2 + I_1 = 100$……………………………………(1)

BCD is: $9 X I_4 - 9 x I_5 – 2 x I_2 = 0$, substituting for $I_4 = I_3 – I_2$ and $I_5 = I_2 – I_1$ we have,
$-9 I_3 + 20 I_2 – 9 I_1 = 0$…………………………….……(2)

CDE is: $-9 I_2 + 6 I_1 = 0$, substituting for $I_5 = I_2 – I_1$ we have,
$0 I_3 – 9 I_2 + 15 I_1 = 0$………………………………………..(3)

$10 I_3 – 9 I_2 + I_1 = 100$……………………………………(1)
$-9 I_3 + 20 I_2 – 9 I_1 = 0$…………………………….……(2)
$0 I_3 – 9 I_2 + 15 I_1 = 0$…………………………….…..(3)

In the Matrix notations:

$$\begin{matrix} 10 & -9 & 0 \\ -9 & 20 & -9 \\ 0 & -9 & 15 \end{matrix} \quad \begin{matrix} I3 \\ I2 \\ I1 \end{matrix} \quad = \quad \begin{matrix} 100 \\ 0 \\ 0 \end{matrix}$$

These set of equations are of the standard form; $Ax = b$
$A^{-1} Ax = A^{-1} b$
$Ix = A^{-1} b$
$x = A^{-1} b$

V.b.1. Solving with Matlab:

miee.m

```
A = [10 -9 0 ; -9 20 -9 ; 0 -9 15];
b = [100 ; 0 ; 0];
z = A^-1 *b
```

Matlab Result:

```
>> miee

z =

   22.4615
   13.8462
    8.3077
```

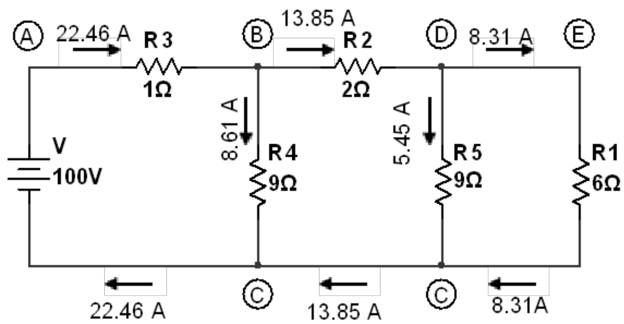

Figure 2: Schematic diagram for a series-parallel circuit with the results

V.c. Sample Student Problem 1.

Chapter 7 Problem 3, Boylestad's book.

Determine R_T for the following Networks.

1. <u>Multisim Computation:</u>

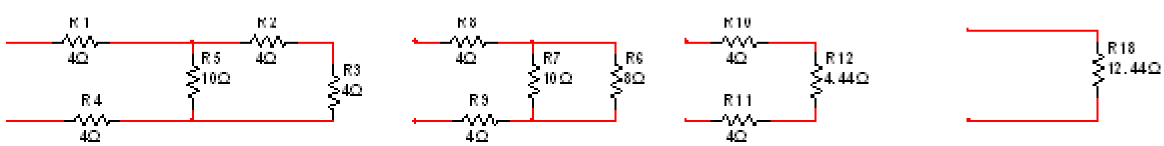

Figure 3: Schematic Diagram of Problem 7_3 (a) from text (Drawn in Multisim)

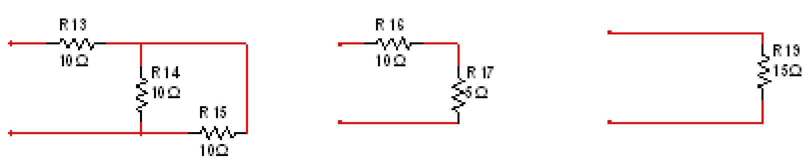

Figure 4: Schematic Diagram of Problem 7_3 (b) from text (Drawn in Multisim)

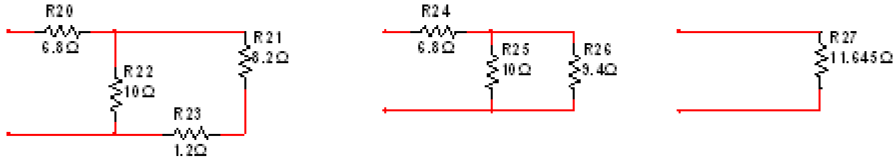

Figure 5: Schematic Diagram of Problem 7_3 (c) from text (Drawn in Multisim)

2. Matlab Computations:

This computation shows how we calculated the resultant circuit for each network. It shows also some comments explaining the technique or the law we did use.

```
%This solution below is for problem 3,a
R1=4, R2=10, R3=4, R4=4, R5=4

%these are the known variable that we are given, so we can calculate the total resistance. Based on the schematic diagram, we know that R3 and R4 are in series.
Rx= R3+R4

Rx

    8

%Then, R3 and R4 are parallel with R2.
Rz= Rx*R2/(Rx+R2)

ans

   4.4444

% Now, we can calculate the total resistance by adding up R1, 4.4444, and R5
Rt1=R1+Rz+R5

Rt1

  12.4444

%This following solution is for problem 3,b.
R1=10,R2=10,R3=10

%These are the given values for resistors, so we can calculate thetotal resistance.
%In the schematic diagram, it shows that R2 and R3 are in parallel.
Rq= R2*R3/(R2+R3)

ans

    5
```

```
%This comulative resistor is connected in series with R1.
Rt2=Rq+R1
```

```
Rt2

   15
```

```
%This last solution is for problem 3,c
R1=6.8,R2=10,R3=8.2,R4=1.2

%These are the given values for calculating the total resistance.
%We noticed that R3 and R4 are connected in series, and the comulative resistance of
these two will be connected in parallel with R2
Rp=R3+R4

ans

   9.4000

Rk=R2*Rp/(Rp+R2)

ans

   4.8454

%The total resistance would be the sum of 4.8454 and R1
Rt3=R1+Rk
```

```
Rt3

  11.6454
```

V.d. Sample Student Problem 2,

Chapter 7 problem number 11, Boylestad's book.

For the network, redrawn from diagram in book:

a) Find the voltages V_a, V_b, and V_c.
b) Find the currents I_1 and I_2.

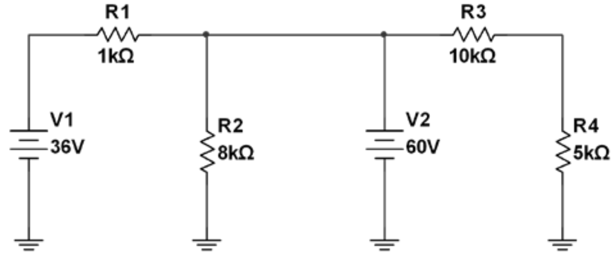

Figure 6: Schematic Diagram of Problem 7_11 from Text (Drawn in Multisim)

Equations:

$V_a = 36V$
$V_b = 60V$
$V_c = \frac{R_4}{R_4 + R_3} * V_b = \frac{5k\Omega}{5k\Omega + 10k\Omega} * 60V = 20V$
$V_{R1} = V_b - V_a = 60V - 36V = 24V,\quad I_{R1} = \frac{V_{R1}}{R_1} = \frac{24V}{1k\Omega}$ = 24mA, $\quad I_{R1} = I_1,\quad$ $I_1 = 24mA$
$I_2 = I_1 + \frac{V_b}{R_2} + \frac{V_b}{R_3+R_4} = 24mA + \frac{60V}{8k\Omega} + \frac{60V}{10k\Omega+5k\Omega} = 35.5mA$

Matlab Script File:

```
%Student: xyz
%Assignment: Solve chapter 7 problem number 11 in Matlab.
%For the network, find the voltages Va, Vb, and Vc.
%Known values:
format compact
V1=36;
V2=60;
R1=1000;
R2=8000;
R3=10000;
R4=5000;

%Analyzing the circuit:
Va=V1
Vb=V2
Vc=(R4/(R4+R3))*Vb

%Find the currents I1 and I2
VR1=V2-V1; IR1=VR1/R1; I1=IR1
I2=I1+Vb/R2+(Vb/(R3+R4))
```

Matlab Output:

```
>> Chapter_7_Assigment
Va =    36
Vb =    60
Vc =    20

I1 =    0.0240

I2 =    0.0355
```

VI. Physical Implementations of the Circuits - Labs

As the regular part of lab work, students throughout the course were engage in actual physical building of the circuits and using the instruments like power supply and Multi-meters. The disparity in physical instrument reading and computed value along with simulated results help students understand and appreciate the differences between ideal and physical components' values. Among the many methods physical implementation was most challenging and least liked by the students. Never the less the insight it provides is invaluable in their experiential learning.

VI.(a) Partial list of labs performed during this course:

1. The Electrical Laboratory Instruments
2. Introduction to Multisim
3. DC Sources and Metering
4. Resistor Color Code
5. Ohm's Law
6. Series DC Circuits
7. Parallel DC Circuits
8. Series-Parallel DC Circuits
9. Ladders and Bridges
10. Potentiometer and Rheostat

These set of labs have been performed from the textbook Laboratory Manual for DC Circuits by James M. Fiore. All these laboratories exercises were performed by a) Multisim software and then by b) Physical components and physical instruments.

VI.(b) A Sample Laboratory: Introduction to Multisim

To Start the program:
Click on Start → All Programs → National Instruments → Circuit Design Suite 14.0 → Multisim 14.0.

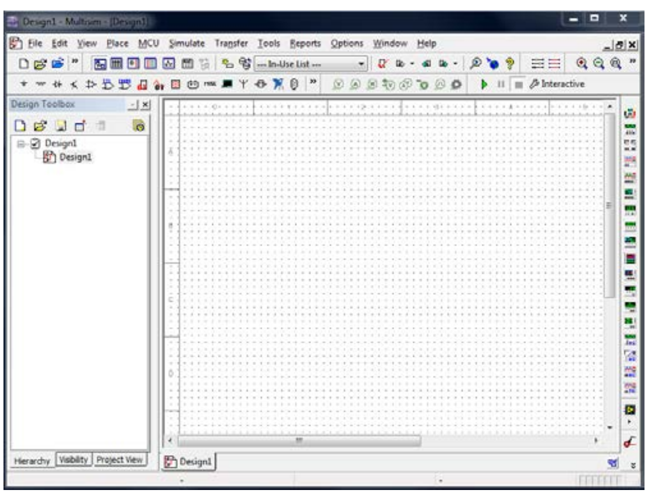

Figure 7: Blank schematic circuit

a. Open/Create Schematic

A blank schematic Circuit 1 is automatically created. To create a new schematic, click on File → New → Schematic Capture. To save the schematic click on File /Save As. To open an existing file, click on File/ Open in the toolbar.

b. **B Place Components**

To Place Components, click on **Place/Component**. On the Select Component Window click on **Group** to select the components needed for the circuit. Click OK to place the component on the schematic.

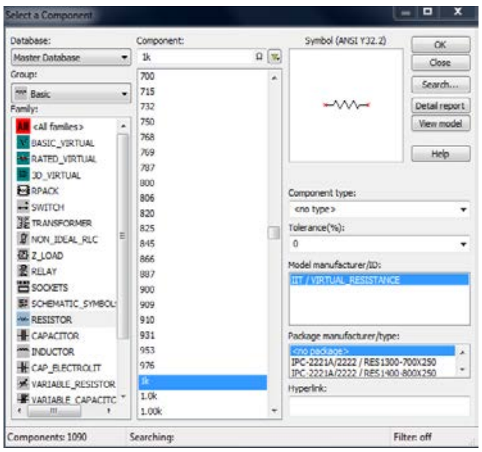

Figure 8: Select resistor

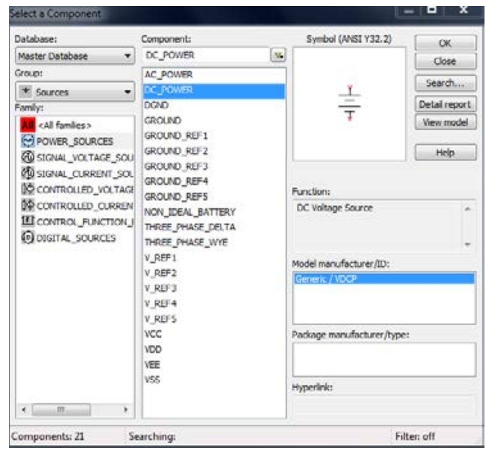

Figure 9: Select DC voltage

For example, to select resistors and the DC source shown in Figure 3 click on **Place/ Component**. In **Group** select **Basic** scroll down to Resistors and select the value of the resistor needed to construct the circuit, for this example 1k is selected. To place DC source, click on Sources in **Group** and select **DC Source**. As shown in Figure 2 and Figure 3 respectively.

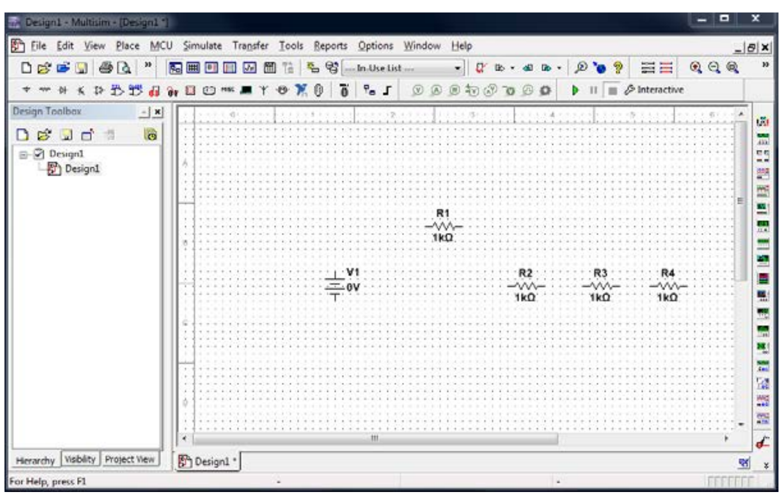

Figure 10: DC Source & Resistors

c. Rotate Components

To rotate the components right click on the Resistor to flip the component on 90 Clockwise (Ctrl + R) and 90 Counter Clockwise (Ctrl + Shift + R).

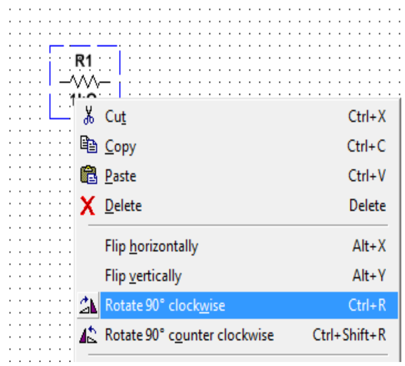

Figure 11: Rotate components

d. Place Wire/Connect Components

To connect resistors, click on **Place/Wire** drag and place the wire. Components can also be connected by clicking the mouse over the terminal edge of one component and dragging to the edge of another component. Reference Figure 6.

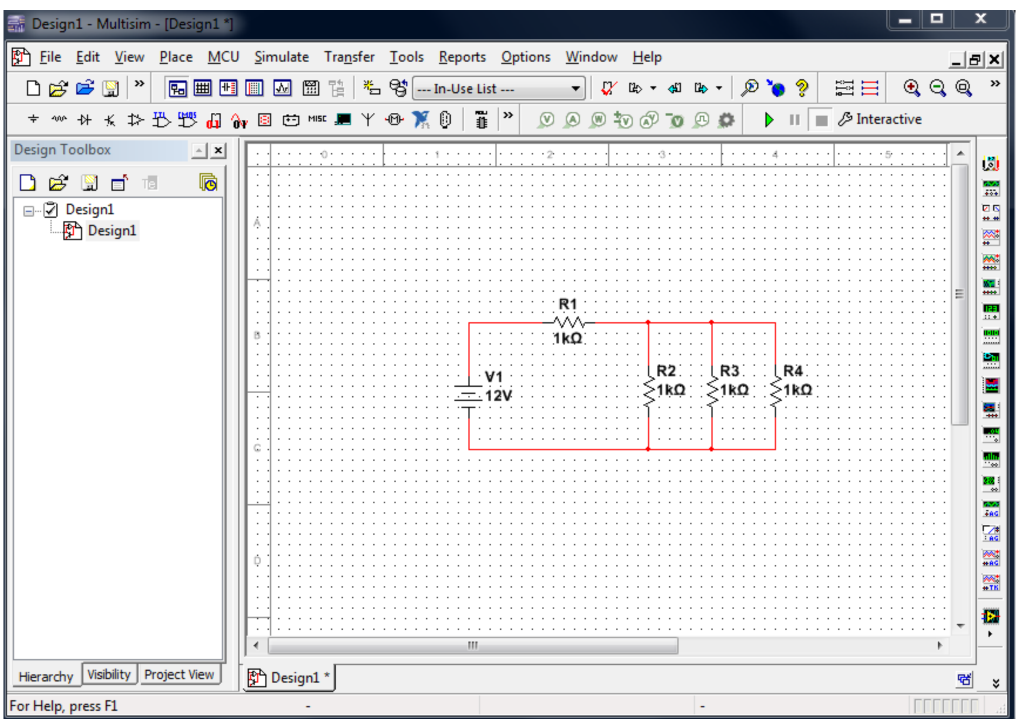

Figure 12: Place/Wire

e. Change Component Values

To change component values double click on the component, this brings up a window that display the properties of the component. Reference Figure 7. Change R_1 from 1k Ohm to 10 Ohms, R_2 to 20 Ohms, R_3 to 30 Ohms, and R_4 to 40 Ohms. Also change the DC voltage source from 12 V to 20 V. Figure 8 shows the completed circuit.

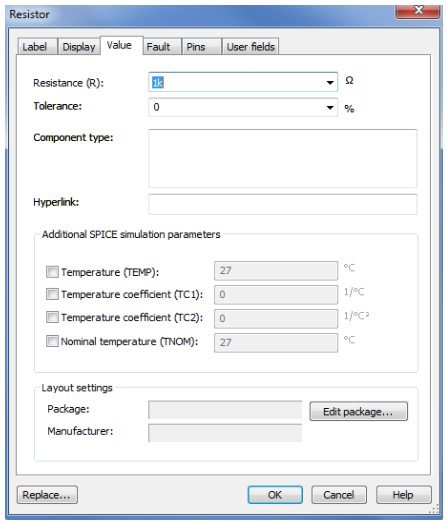

Figure 13: Change component values

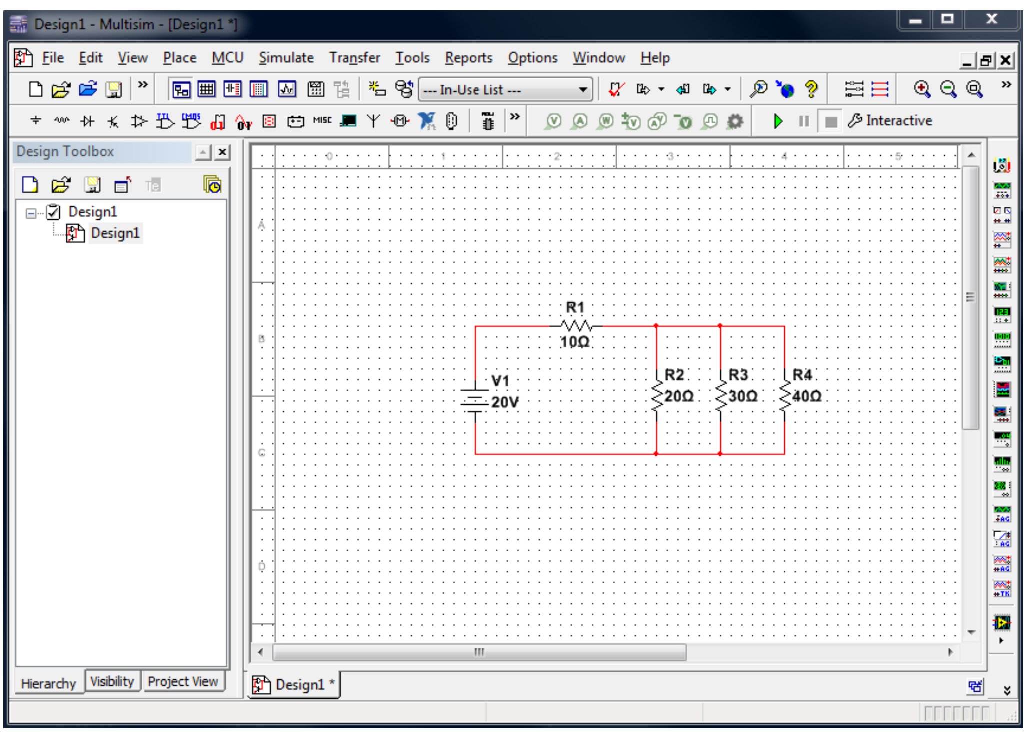

Figure 14: Completed circuit

f. Grounding

All circuits must be grounded before the circuit can be simulated. Click on Place/Source in the toolbar, then select ground the circuit. **If the circuit is not grounded, Multisim will not run the simulation.**

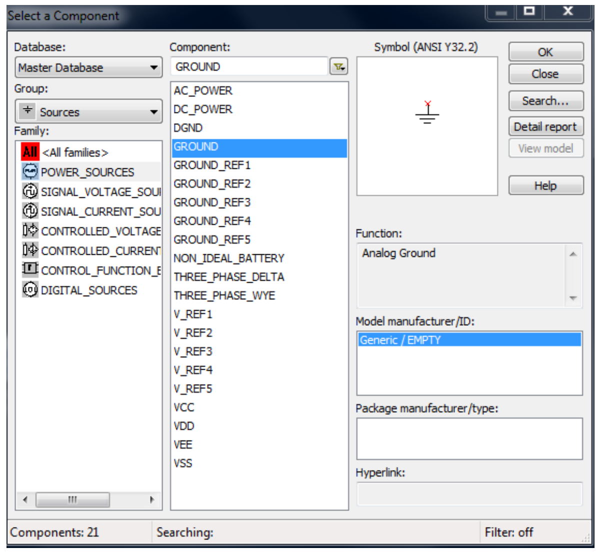

Figure 15: Grounding

g. Simulation

To simulate the completed circuit, click on **Simulate/Run** or **F5**. This feature can also be accessed from the toolbar as shown in Figure 10 below.

Figure 16: Simulation

Multisim offers multiple ways to analyze the circuit using virtual instruments. Some of the basic instruments needed for this lab are described below:

1) Multimeter

Multimeter is used to measure AC or DC voltage or current, and resistance or decibel loss between two nodes in a circuit. To use the Multimeter, click on the Multimeter button in the **Instruments toolbar** and click to place its icon on the workspace. Double-click on the icon to open the instrument window, which is used to enter settings and view measurements.

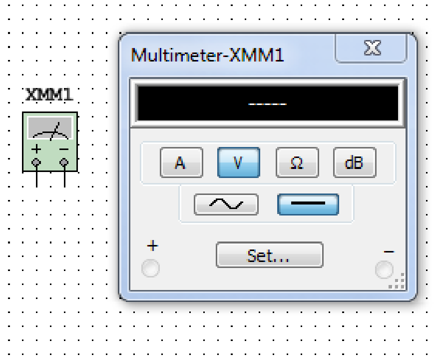

Figure 17: Multimeter

To measure Voltage, place multimeter in Parallel with the component (Resistor, Voltage etc.). To measure Current, place the multimeter in series with the component. Reference the Figure 12 and 13.

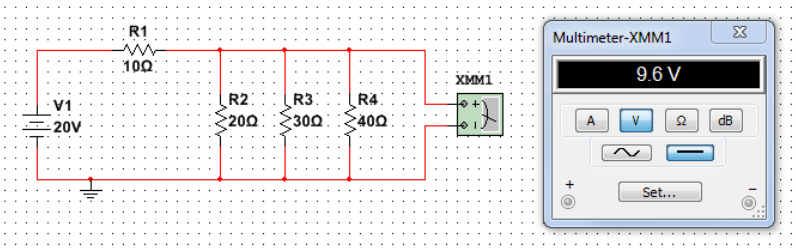

Figure 18: Measure voltage

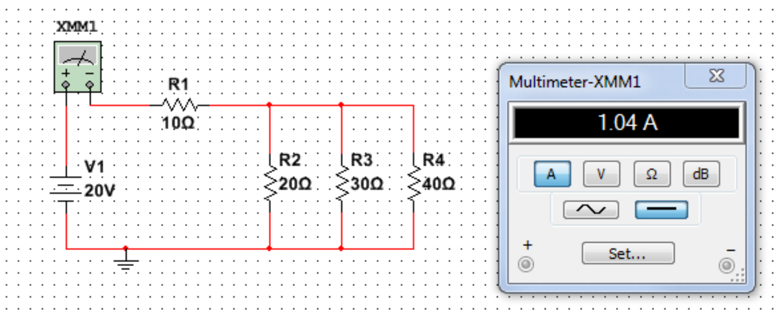

Figure 19: Measure current

2) <u>Wattmeter</u>

The wattmeter measures power. It is used to measure the magnitude of the active power, that is, the product of the voltage difference and the current flowing through the current terminals in a circuit.

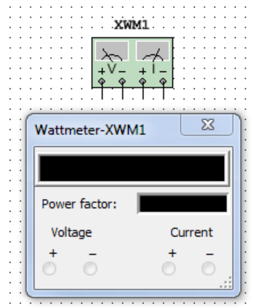

Figure 20: Wattmeter

To use the instrument, click on the Wattmeter button in the **Instruments toolbar** and click to place its icon on the workspace. The icon is used to wire the Wattmeter to the circuit. Double-click on the icon to open the instrument window, which is used to enter settings and view measurements. Reference Figure 15 for more details.

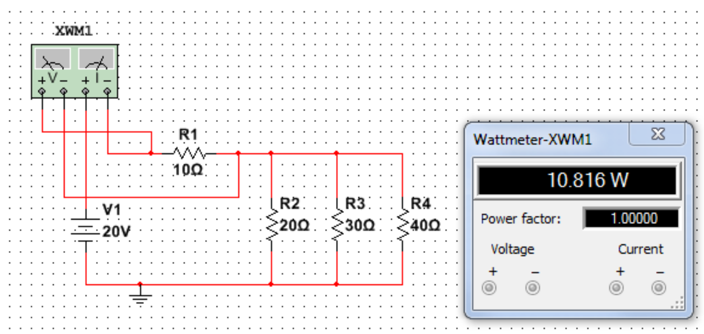

Figure 21: Wattmeter Connection

3) <u>Ammeter</u>

The ammeter offers advantages over the multimeter for measuring current in a circuit. It takes up less space in a circuit and you can rotate its terminals to suit your layout. **Always connect the ammeter in series with the load**. To place Ammeter, click on View → Toolbar → Select → Measurement Components. See Figure 17 on how to use the Ammeter.

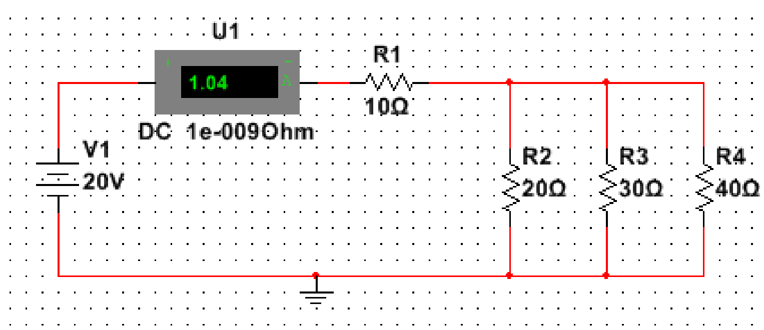

Figure 22: Ammeter

4) Voltmeter

The Voltmeter offers advantages over the multimeter for measuring voltage in a circuit. **Always connect the voltmeter in parallel with the load**. The voltmeter can be found in the measurement toolbar.

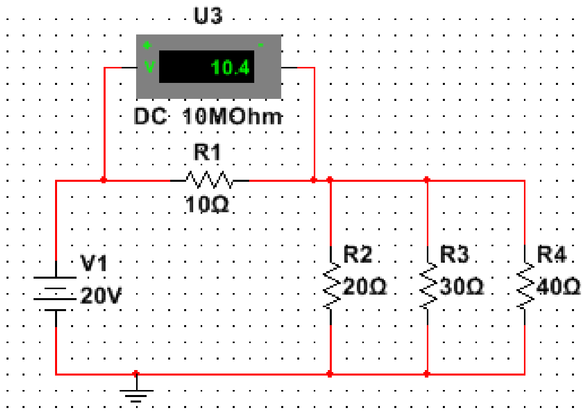

Figure 23: Voltmeter

NOTE: This tutorial offers an introduction to Multisim which includes description and examples on how to use basic instruments needed for ECET 100 labs. For more information on instruments not described in this tutorial please referred to, Multisim Instruction Manual.Pdf[12].

VII. Exposure to Microcontrollers by Utilizing Arduino

The last four weeks of the semester were devoted to Introduction to Embedded Systems. The approach we used was to introduce the subject with the help of Arduino Open source Platform which utilizes Atmel 328P processor. This choice was made due to a fast learning curve associated with Arduino and the great repertoire of technical knowhow in the public domain. The specification for the Arduino are as follows:

Microcontroller	ATmega328
Operating Voltage	5V
Input Voltage (recommended)	7-12V
Input Voltage (limits)	6-20V
Digital I/O Pins	14 (of which 6 provide PWM output)
Analog Input Pins	6
DC Current per I/O Pin	40 mA
DC Current for 3.3V Pin	50 mA
Flash Memory	32 KB (ATmega328) of which 0.5 KB used by bootloader
SRAM	2 KB (ATmega328)
EEPROM	1 KB (ATmega328)
Clock Speed	16 MHz

The class utilized Sparkfun's Inventors Kit[13]. The kit was complete with the required set of parts and Sparkfun provides a set of following 16 experiments:

Introduction: SIK RedBoard & Sparkfun Mini Inventor's Kit
Introduction: SIK Arduino Uno
Experiment 1: Blinking an LED
Experiment 2: Reading a Potentiometer
Experiment 3: Driving an RGB LED
Experiment 4: Driving Multiple LEDs
Experiment 5: Push Buttons
Experiment 6: Reading a Photoresistor
Experiment 7: Reading a Temperature Sensor
Experiment 8: Driving a Servo Motor
Experiment 9: Using a Flex Sensor
Experiment 10: Reading a Soft Potentiometer
Experiment 11: Using a Piezo Buzzer
Experiment 12: Driving a Motor
Experiment 13: Using Relays
Experiment 14: Using a Shift Register
Experiment 15: Using an LCD
Experiment 16: Simon Says

In this class we were able to perform only the first two experiments from the above set of experiments, due to time constraint. Students on their own went much beyond and performed a number of experiments outside of class allocated time. This demonstrate their intense interest in the area of Microcontrollers and Embedded Systems. Authors are of opinion that this interest in the discipline can be sustained throughout in various disciplines of Electrical Engineering, like Communications, Control theory, Motor Drives, Bio-medical Instrumentation, Automation, etc., if these disciplines are combined with embedded systems.

Student Satisfaction Survey

The following survey is a measurement of Students Satisfaction with regard to Course Learning Objectives:

ECET 10000 – Introduction to ECET Semester: Fall 2016						
		Student Evaluation				
Course Objective		**Excelent (4)**	**Good (3)**	**Average (2)**	**Need Attention (1)**	**Percentage**
1. Gain a working knowledge of Different Functionary Roles Of Different Units Of University, like ECET office, Library, Registrar's office. Financial aid office, and Dean of Students office etc.		16	7	1	1	88
2. Gain proficiency in solving problems and doing computations in Basic Concepts And Basic Circuit Analysis, like, current and voltage, electric resistance, Ohm's law, Kirchhoff's laws, series and parallel combinations of resistors, voltage and current dividers, short circuits and open circuits, and Electrical power and energy.		21	3	1		95
3. Gain proficiency in solving problems and doing computations in Dc Circuit Analysis like nodal analysis, loop analysis, maximum power transfer theorem.		13	5	7		81
4. Gain proficiency in use of Multisim software in the labs for Circuit Analysis Computation. The software is also utilized for drawing schematic diagrams.		19	3	3		91
5. Gain proficiency in solving problems and doing computations with MatLab, utilizing matlab fundamentals, plotting commands and control statements.		16	6	3		88
6. Build Arduino based simple projects to demonstrate electronic peripheral interfaces like motor control.		15	5	5		85

Figure 24: Course learning objectives survey

The following survey is a measurement of Students Satisfaction with regard to ABET[14] Criteria Satisfied with regard to a, b, d, g.:

ECET 10000 – Introduction to ECET Semester: Fall 2016 **Proposed ABET Criterion Satisfied: a,b,d,g**					
Instructors : Omer Farook / Hassan Alibrahim	**Fall 2016**				
	Student Evaluation (%)				
Criteria	**Excelent (4)**	**Good (3)**	**Average (2)**	**Need Attention (1)**	**Percentage**
a. Outcome a: The course provides the fundamentals of Basic Concepts And Basic Circuit Analysis, like, current and voltage, electric resistance, Ohm's law, Kirchhoff's laws, series and parallel combinations of resistors, voltage and current dividers, short circuits, open circuits, Electrical power and energy. My understanding can be rated as:	19	6			94
b.Outcome b: Students utilize MatLab, utilizing MatLab fundamentals, plotting commands and control statements.	17	7	1		91
d. Outcome d: As a result of this course, my ability to apply Multisim in the design and analysis of circuits can be rated as,	22	3			97
g. Outcome g: The students write a bi weekly learning report, which includes the lecture topics covered, lab performance, computation and homework assignment.	24	1			99

Figure 25: ABET Criteria Satisfied with regard to a, b, d, g

XII. Pedagogy of the Course

The pedagogy of the course is based on Outcome Based Education, and utilizes the interactive model of learning. All the students maintain an online portfolio of their work. The system designed in the laboratory to perform a specific task is the core measurement as the learning outcome of the course. The laboratory performance of the course is performed in teams of two students. This mode provides a platform for horizontal learning through active and engaged discourse and discussion. Students are empowered to charter their learning and feed their curiosity. The course culminates in a Final Project which is based on students own research from a set of selected topics of interest in the field of Electrical and Computer Engineering Technology. These projects were assessed based upon its comprehensiveness and originality. Students are required to master the soft skills of comprehensive report writing on a weekly basis and of Technical Project Report writing and project oral presentation based upon the Team's Final Project. These classroom practices and laboratory environment provides a challenging and invigorating environment that prepares them for a lifelong learning process and career path[15].

XIII. Conclusion

The paper has provided to the reader the philosophical framework and turnkey path way for Freshman Experience Course in Electrical and Computer Engineering Technology, Emphasizing Computation, Simulation, Mathematical Modeling and Measurements. The diverse subjects of 1) Circuit Analysis pursued through Matlab, 2) the use of Multisim in drawing schematic diagrams along with circuit simulation, 3) physical implementation of the circuits and use of instruments and 4) introduction to Microcontrollers, are anchoring freshmen students to go forth with a set of skillset that are so needed for their success in their chosen discipline. The authors sincerely hope that this paper is a pointer to many academicians by offering such a course in their respective curriculums for providing rigor and challenge to the freshmen students.

Bibliography

[1] https://www.mathworks.com/products/matlab.html

[2] http://www.ni.com/multisim/

[3] https://www.arduino.cc/

[4] Omer Farook, Jai P. Agrawal, Chandra R. Sekhar, Essaid Bouktache, Ashfaq Ahmed and Mohammad Zahraee “Outcome Based Education And Assessment”, Proceedings of the 2006 American Society for Engineering Education Annual Conference & Exposition June 20 -23, 2006. Chicago, IL.

[5] Introductory Circuit Analysis, 13th Edition, By: Robert L. Boylestad, Publisher Pearson.

[6] Asad Yousuf, Ayush Bhardwaj, Crystal Reeves, William Lehman, Sylvester Chukwukere, and Shinemin Lin. "Closing The Hardware Design Loop With Multisim: A Case Study". 2007 Annual Conference & Exposition, Honolulu, Hawaii, 2007, June. ASEE Conferences, 2007

[7] http://www.ni.com/multisim/what-is/

[8] Rudra Pratap, Getting Started with MATLAB, A Quick Introduction for Scientists and Engineers, Oxford University Press.

[9] Brian L. F. Daku, Learning Matlab Superfast. Publishers: John Wiley & Sons INC.

[10] Matrices in Electrical Engineering – Basic Circuit Analysis for Electrical Engineering, by L D Constantinovici and Mathew Govindsamy. Publisher Juta and Co Ltd 1999.Kenwyn South Africa

[11] Laboratory Manual for DC Electrical Circuits, by James M. Fiore is copyrighted under the terms of a Creative Commons license: contact: James Fiore, Professor, Electrical Engineering Technology, Mohawk Valley Community College 1101 Sherman Drive Utica, NY 13501 jfiore@mvcc.edu or via www.dissidents.com

[12] http://www.ni.com/pdf/manuals/374483d.pdf

[13] https://learn.sparkfun.com/tutorials/sik-experiment-guide-for-arduino---v32/all
https://cdn.sparkfun.com/datasheets/Kits/SFE03-0012-SIK.Guide-300dpi-01.pdf
https://learn.sparkfun.com/tutorials/sik-experiment-guide-for-arduino---v32/all

[14] http://www.abet.org/

[15] Embedded System Design Based on Beaglebone Black with Embedded Linux.
Farook, O., & Agrawal, J. P., & Ahmed, A., & Kulatunga, A., & Koyi, N. K., & Alibrahim, H. A., & Almenaies, M. (2016, June), Paper presented at 2016 ASEE Annual Conference & Exposition, New Orleans, Louisiana.