Modification of an Introduction to Electronics Course to a Project Oriented Approach

Gregory Harstine, Samuel Mastromateo, Elizabeth Craddock, Joshua Boley, Andrew Milks,
Engineering and Science Technology Department
University of Akron
Akron, Ohio 44325
Email:gph@uakron.edu

Abstract

A year ago, the faculty of the Electrical & Electronic Engineering Technology program evaluated the two-course sequence of classes that begins with a first semester freshman class titled Introduction to Electronics & Computers. The follow up course is called Digital Circuits. A fair amount of time was being spent in Digital Circuits to repeat the topics covered in Introduction to Electronics & Computers. As a result, it was decided that it might be advantageous to simply cover digital material in Digital Circuits and revamp Introduction to Electronics & Computers to focus more time on circuit design, simulation, prototyping and the board construction processes. This material treated in Introduction to Electronics & Computers.

Beginning with the Fall 2018 semester, Introduction to Electronics & Computers was modified with the important goal of each student completing two projects resulting in the construction of complete and functional electronic circuit boards. The first project contained a three-way board level switch that switched between two LEDs connected to a 9-volt battery pigtail and the second was a BJT common emitter voltage amplifier. In addition, assignments that covered important topics such as Excel spreadsheets, graphing and important calculator operations were needed in the program and could be added to replace the relocated digital material.

The two circuit projects began by prototyping the circuits on a prototyping bread board. They were next developed and simulated in Multisim and then exported to Ultiboard. Students were provided with design specifications for guidance prior to creating Gerber files. Once the Gerber files were created the student has options pertaining to how to the board is to be manufactured. The two options we have on campus are an AMC2500 Quick Circuit 5000 circuit board router and a newly purchased Voltera V-One circuit board printer. In addition, off campus options for inexpensive and convenient manufacturer can be utilized when larger quantities were needed. Once the circuit boards are manufactured students then solder parts and troubleshoot to get the boards completely functional.

This paper will examine the results of these modifications to the Introduction to Electronics & Computers course. Results of the modified course successes and failures will be examined. A close look at the purchase of the Voltera printer and AMC Router will be examined as well as the financial merits and time considerations of producing the boards on either of these two options versus having the boards manufactured externally.

Introduction

The department offers an Associate of Applied Science and a Bachelor of Science in Electrical & Electronic Engineering Technology. As a result, we have a two hundred level and a four hundred level Project class for each respective degree. Over the past several years, faculty have observed that actual electronic circuit design and board construction skills were lacking in the students going through these courses. Further evaluation revealed that program did not have opportunities for students to develop circuit manufacturing skills.

Multisim is introduced and utilized in the first-year courses 2860:120 Circuit Fundamentals, 2860:122 AC Circuits and 2860:123 Electronic Devices & Circuits to provide an supplemental circuit analysis methodology to support pencil and paper analysis. Through application in multiple courses circuit simulation skills develop and are reinforced through this tool. However, the development of physical circuit boards and soldering abilities was only minimally covered in a lab or two. We have the companion software to Multisim called Ultiboard, to seamlessly design models for electronic boards. From Ultiboard, Gerber files can be created, which are an industry standard set of files that can be electronically distributed anywhere for actual fabrication of the printed circuit boards. Students needed earlier exposure to circuit manufacturing and construction prior to use in the associate and baccalaureate capstone projects.

In the Spring of 2018, the faculty decided to modify 2860:121 Introduction to Electronics & Computers, to give the students more opportunities to construct actual functioning electronic boards. In addition, to these skills, it was noted that it would be a good idea to cover a few other soft skills that would be beneficial to freshman students. The emphasis of this paper however, will be the coverage of the development of student board design & construction capabilities.

Topics Proposed

Modified course topics were identified to emphasize the missing student skills. The challenge was to develop basic circuit construction skills with incoming freshman, many of which did not have not electronics background. In addition, faculty in engineering technology are known for practical skills and we wanted to build practical know how into the program from the beginning.

The skills we wanted to emphasize were basic prototyping using the breadboards with neatly wired components, using meters for troubleshooting circuits on a breadboard, proper design of an electronic board using Ultiboard along with a list of specifications or constraints. Additionally, faculty wanted to provide several opportunities to utilize soldering and soldering safety as well as opportunities to improve. Finally, we wanted each student to create from start to finish a complete functioning electronic board of their own creation.

The following course outline was developed for Introduction to Electronics and Computers which indicates the board construction topics as well as soft skill topics identified for improvement, such as neatly organized homework problem presentation.

Week	Topics	
1	Intro, SI Units Unit Conversions	Units & Conversions/Calculators
2	Preparing a Proper Homework	Preparing a Proper Homework
3	Brightspace/Calculators	Navigating Brightspace
4	BreadBoarding/Prototyping	Using Meters
5	Project I MultSim	Project I MultSim
6	Project I Ultiboard	Project I Ultiboard
7	Gerber Files/Board Fab	Gerber Files/Board Fab
8	Terminals/Wiring Shop Tools-screwdrivers, wire cutters	Terminals/Wiring Shop Tools-screwdrivers, wire cutters
9	Soldering	Soldering
10	Breadboarding/Prototyping Troubleshooting	Breadboarding/Prototyping Troubleshooting
11	Project II MultSim	Project II MultSim
12	Project II Ultiboard/Gerber Files	Project II Ultiboard/Gerber Files
13	Gerber Files/Board Fab	Gerber Files/Board Fab
14	Soldering	Soldering
15	Review	**Thanksgiving**
16	Final	

Projects

Through faculty discussion, it was determined that the completion of two projects would be would allow development and immediate reinforcement of circuit manufacturing skills. The two projects where designed, by the faculty, with the same objectives for both projects:

1. Breadboard/Prototype & troubleshoot the project to a fully functioning circuit.
2. Design and simulate each circuit in Mutism.
3. Export each circuit to Ultiboard and design a board to a particular set of specifications and constraints.
4. Create Gerber files for physical construction of the board using available equipment at the University or having the boards fabricated off campus.
5. Solder all components to the board and create a fully functioning circuit.

Prototyping/Breadboarding

Typical breadboards are shown figures 1 and 2. The circuits for projects 1 and 2 are shown in fugrues 3 and 4 along with what a typical circuit looks like being constructed on a breadboard[1]. We wanted to introduce and provide opportunity for our students to properly breadboard a circuit efficiently and to learn how to use meters on a breadboard. For each project this portion was completed in two labs during the semester.

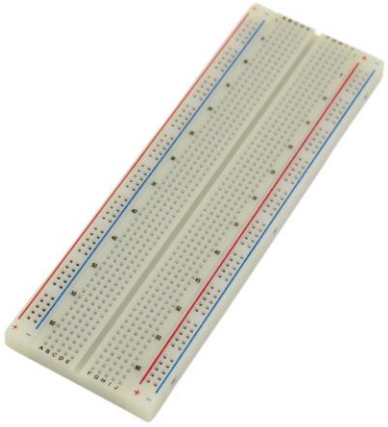


Figure 1. Breadboad

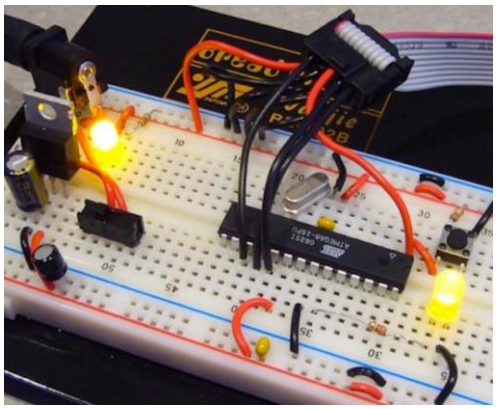


Figure 2. Operating Circuit[1]

Each student did a lab for each project to create each circuit in mutism.

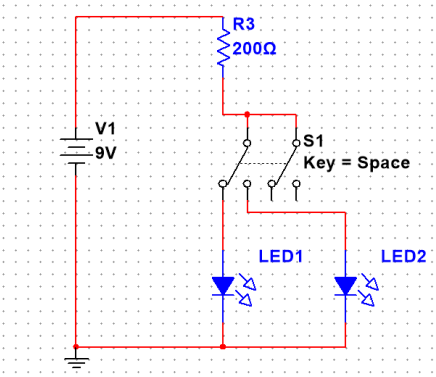


Figure 3. Project 1

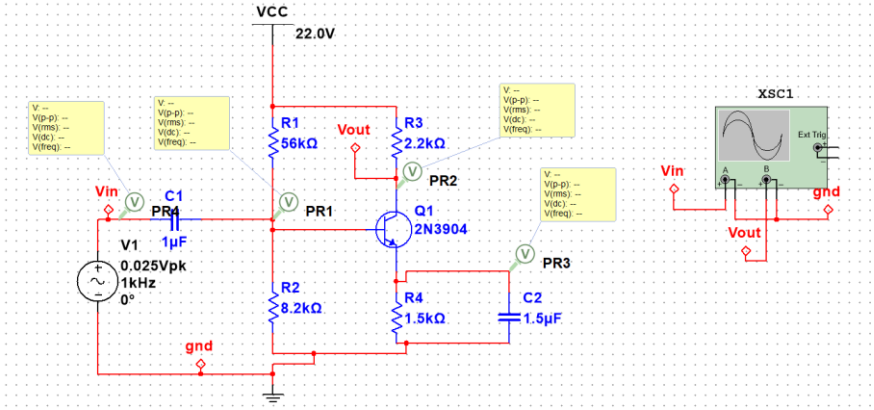


Figure 4. Project 2

The projects progress in difficulty and the students were introduced to methods to get quick measurements in mutism, the yellow boxes and a method of simplifying more complex circuits with mutism connectors, the red diamonds.

Ultiboard

From mutism, the circuit can be easily exported to another National Instruments Software package called Ultiboard for designing the physical electronics boards. The students were given the following specifications and constraints:

1. In Multisim, go to the tab View>Spreadsheet and click on the components tab on the bottom.
2. The transistor and each resistor should match the following table for values and mounting package.

RefDes	Sheet	Se...	Family	Value	Tolerance	Manufacturer	Package
Q1	amplifier...		BJT_NPN	2N3904		National Semicondu...	TO-92-3(Z03A)
R1	amplifier...		RESISTOR	56kΩ		Generic	RES0.25
R2	amplifier...		RESISTOR	8.2kΩ		Generic	RES0.25
R3	amplifier...		RESISTOR	2.2kΩ		Generic	RES0.25
R4	amplifier...		RESISTOR	1.5kΩ		Generic	RES0.25

3. And the capacitors need to match the following:

RefDes	Sheet	Se...	Family	Value	Tolerance	Manufacturer	Package
C1	amplifier...		CAPACITOR	1µF		Panasonic	Can-Radial(B9)
C2	amplifier...		CAPACITOR	1.5µF		Panasonic	Can-Radial(F13)

4. Transfer your board to ultiboard.

5. Additional Ultiboard requirements:

   a. Board Size: 1” x 1”
   b. Trace Size: 25 Mils
   c. Trace Clearance: 25 mils
   d. All Pads Square, 75 mil in size
      i. Except Q1, pin 1, 2 & 3 make round 65 mil dia
   e. Add three headers
      i. One for Vin with nets appropriately connected to C1 and gnd.
      ii. One for Vout with nets appropratiely connected to R3 and BJT collector
      iii. One for Vcc, pin2 connected to R1 & R3 net
   f. Add text titles at Vin and Vout

The design in Ultiboard will ultimately appear somewhat like these below for each Project. Figures 5 and 6 show the completed circuit design and 3D rendering for Project 1 and Figures 7 and 8 for Project 2.

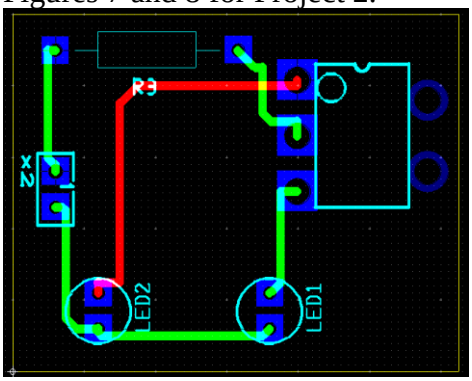


Figure 5. Project 1 Gerber Layout

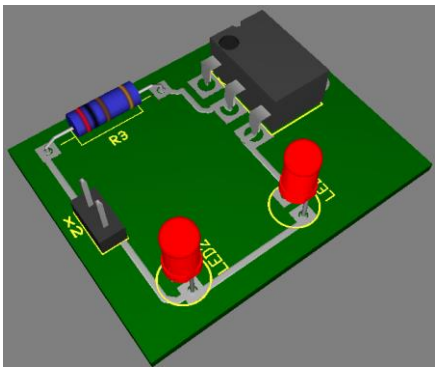


Figure 6. Project 1 3D View

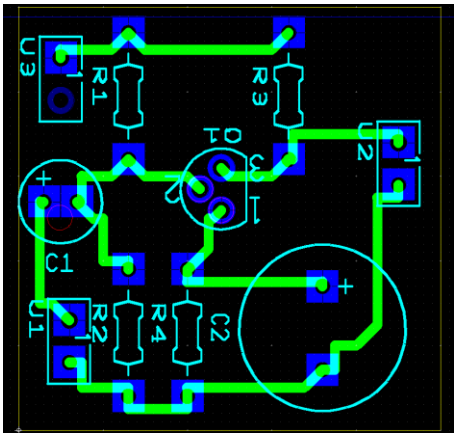


Figure 7. Project 2 Gerber Layout

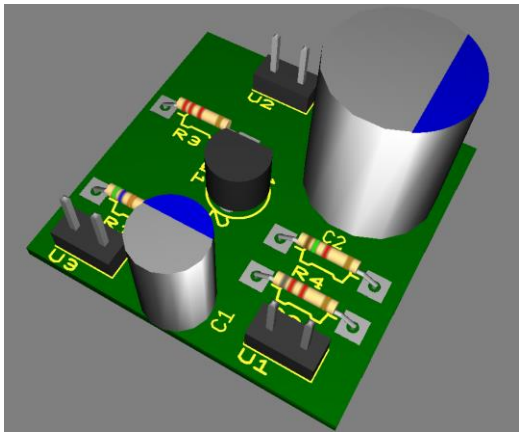


Figure 8. Project 2 3D View

From Ultiboard, Gerber files are created with the industry standard formats. Figures 9 and 10 show examples of individual Gerber layers.

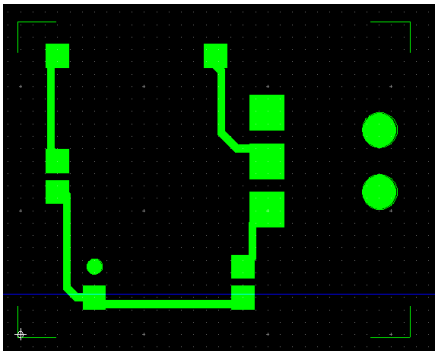


Figure 9. Project 1:Copper Top.gbr

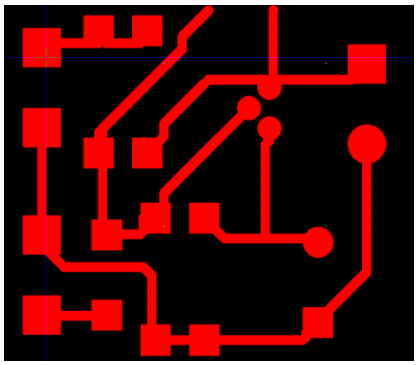


Figure 10 Project 2:Copper Bottom.gbr

Board Construction

Students have several options when having the board fabricated. We have the most experience with an AMC Quick Circuit board router shown below:

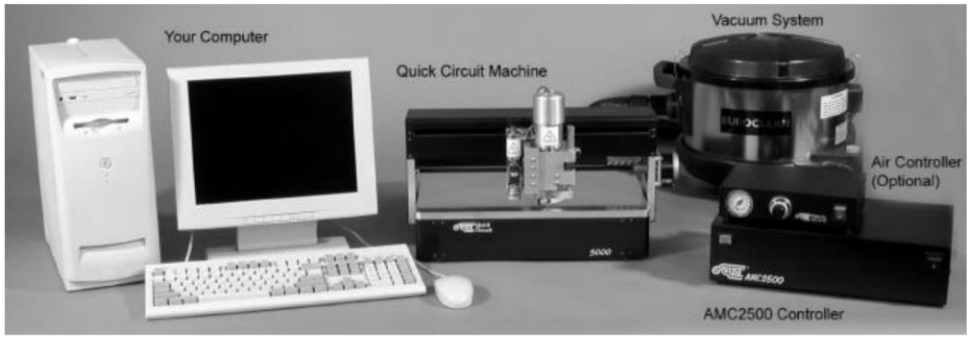
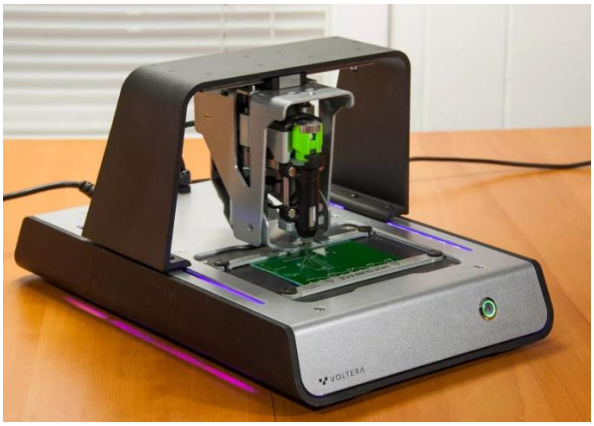


Figure 11. AMC Circuit Board Router

This machine is older and not very precise but can produce a 2” board in about 30 minutes once the Gerber files are complete. The AMC machine is a subtractive manufacturing process that starts with a complete two-sided copper boards and routes the traces and pads indicated in the Gerber files.

Recently a Voltera V1 circuit board printer was purchases. This machine arrived after at the end of the semesters and was not utilized in the fall 2018 semester but will be used in upcoming semesters. This machine uses additive manufacturing to print traces onto a blank fiberglass board. The purchase price for this machine was approximately $5000. The V1 can print traces, heat the board to cure the traces, print solder paste and reflow surface mounted parts. Through holes parts require manual soldering.



In addition, circuit manufactures such as www.oshpark.com have been utilized for when multiple copies of printed circuit boards are required. It is beneficial to have boards from each project available in the event a student was unsuccessful in completing their own physical boards to allow them to move on to the soldering stage. Figures 12 and 13 show boards made by oshpark. Delivery time for each was approximately 2 weeks.

Figures 14 and 15 show completed projects with boards cut on the AMC router and manufactured by OSH Park.

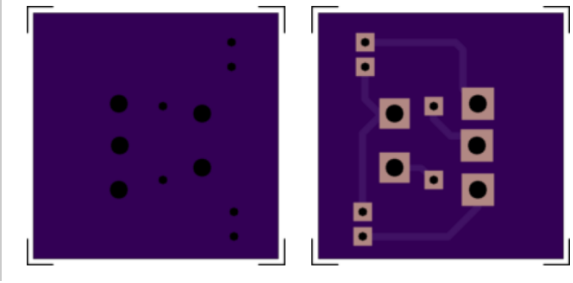


Figure 12. Project 1 – 3 boards for $5.55

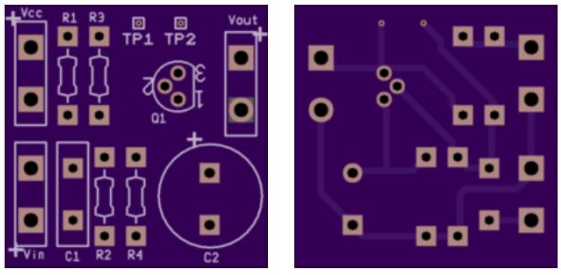


Figure 13. Project 2 – 21 boards for $35

Figure 14. Completed Project 1 (AMC Router)

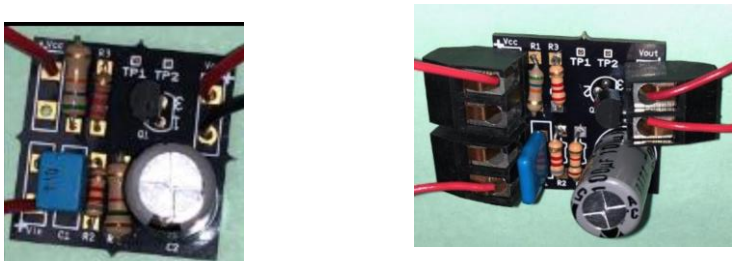


Figure 15. Completed Project 2 (OSH Park Boards)

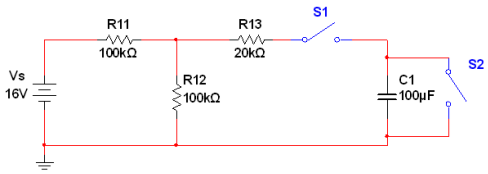
Conclusion/Observations

The best measure that we have is student performance on the final. The final was a good representation of the entire class and is provided here.

OBJECTIVES:
To complete a final project incorporating skills learned throughout the semester including excel, multisim and Ultiboard.

PROCEDURE:

1. **Multisim: Construct and simulate the following circuit in multisim. Gather data to be graphed in Microsoft Excel.**



   a. **In multisim, with S1 closed and S2 open, measure the voltage across C1 and record it in the table below.**

Time, seconds	0	5	10	15	20	25	30	35	40	45	50
Vc											

2. **Ultiboard**: Layout board and submit project files, .ewpri.
   a. **Be sure to have a connector for Vs and a connector to measure Vc.** Or you can have test points to measure Vc.
   b. **C1** is the large capacitor used in **project II,** so use that footprint. I used CAPPR500-1000X1250.
   c. **Use generic package RES0.25** for the resistors. This is again the same resistor used in project II.
   d. **For the two spst switches use footprint** SW_MA1801**.** These switches are huge! You can line them up with the edge of the circuit board, no clearance needed.
   e. **The Board outline should be 1300mils X 1250mils.** IF you can fit on 1250 X1250 I will give bonus points. Square boards are often cheaper to make. Smaller boards are always cheaper, the smaller the cheaper. But the parts must fit on the board outline. No hanging over.
   f. Trace width and trace clearance should be 25 mils.
   g. Part to part clearance should be 10mils.
   h. Pad to trace clearance should be 25 mils.
   i. Pads should be square.
   j. Drill diameter should be 35 mils for resistors and capacitors and 50 mils for switches and Vs and Vc.
   k. Pad diameter should be 50 mils for resistors and capacitors and 90 mils for switches and Vs and Vc.
   l. Annular Ring should be 20 mils for all components.

Deliverables:

1. Multsim file, .ms14, file to the dropbox.
2. An excel spreadsheet that contains:
   a. Table of data collected from the table
   b. Graph of the capacitor voltage response.
3. Ultiboard file, .ewpri
4. The following Gerber files:
   a. Solder Mask Top.gbr
   b. Solder Mask Bottom.gbr
   c. Silkscreen Top.gbr
   d. Drill-Copper Top-Copper Bottom.gbr
   e. Copper Top-Copper Bottom.drl
   f. Copper Top .gbr
   g. Copper Bottom.gbr
   h. Board Outline.gbr

Each of the deliverables was submitted through our educational website dropbox. In two sections of the course there were a total of 15 students. The grade distribution was as follows: 5

- A's, 5 – B's, 2 – C's, 2 – D's and 1 – F. Assessment of the grade distribution was tied to class attendance more than any other component. Several students weren't present when key components of the process were examined and the time constraints of the final shows that some were not prepared to complete certain tasks in a timely and efficient manner.

Faculty "Take Aways"

The course changes to Introduction to Electronics and Computers provided benefit to the students as well as our programs as a whole. We are excited to see how the improvement in student skills will distribute throughout their coursework. These 15 students will be in a capstone project class during spring 2019 at the earliest which is when we will be able to gauge the improvement of the overall student body in these skills.

Faculty knowledge and skills in these areas showed improvement as well. At the beginning of the process, we were comfortable with the AMC board router, but as we began creating more boards, we learned some things to speed up the process, such as routing only a course cut rather than a course and fine cut. We are excited to implement use of the Voltera board printer and look forward to improving our skills with this machine as well. A valuable faculty learning was to leverage the off campus board manufacturing process to free up campus resources. Having gained the experience of downloading and obtaining boards and knowing that they can be quickly obtained is a great value to the program. Further, we discovered that our soldering equipment was not adequate to develop student skills and is in the process of being upgraded with the purchase of new soldering stations and better organization of the existing.

References

1. Sparkfun, https://learn.sparkfun.com
2. OshPark, www.oshpark.com