Instituto Tecnológico y de Estudios Superiores de Occidente

Reconocimiento de validez oficial de estudios de nivel superior según acuerdo secretarial 15018, publicado en el Diario Oficial de la Federación del 29 de noviembre de 1976.

Departamento de Electrónica, Sistemas e Informática

Especialidad en Sistemas Embebidos

Designing a printed circuit board capable of connecting an uNicom 2 and uNicom 3 to an ICT Medalist 3050

TRABAJO RECEPCIONAL que para obtener el **GRADO** de **ESPECIALISTA EN SISTEMAS EMBEBIDOS**

Presenta: **CARLOS SANDOVAL GARRIDO**

Asesor **ENRIQUE RAUL VILLA LOUSTAUNAU**

Tlaquepaque, Jalisco. julio de 2019.

Acknowledgments

I would like to thank the company iBtest for allowing me to participate in this project and all the supporting during the Embedded Systems Specialization Program. I also thank the engineers that helped me design this project for the iBtest company. To the Director Luis Rizo for allowing me to be part of this project.

I want to thank to the Professor Enrique Raúl Villa Loustaunau for being my advisor in this project and teaching me the basic principles for PCB design. Thanks to all the professors for their support and patience. Thanks to CONACYT (scholarship number 937017)**,** for their support in order to conclude my studies. Special thanks to my family and friends for their unconditional support.

Abstract

The time required for testing preparation, testing verification, operator work, and other factors, especially in the automotive industry, can be interpreted as money per hour. However, the testing period can be reduced if a circuit board is created to connect different programming tools. The aim of this article is to design a circuit board that is able to connect the programming tools from CSM company in an ICT Medalist 3050 from Keysight company in the intention to reduce board testing time. The solution included designing two boards. The design of the first board locates the board inside the ICT Medalist 3050 and the second board is designed to connect the desired programming tools, uNicom 2 or uNicom3. The main process to design a board can be divided in the schematic stage and the layout stage. The design is a proof of concept and requires mechanical modifications of the board to be tested in the ICT Medalist 3050. For example, the footprint for the components as the MinPin78 needs to be validated. The electrical design was simulated in LTSpice, and the board design was made using the Allegro software for PCB (Printed Circuit Board). This design reduces the testing time, so the cost invested in testing boards will be reduced.

Introduction

Nowadays most cars, cellphones, airplanes, televisions, refrigerators, and other electronic devices use circuit boards to function properly by connecting the components inside the device. Programming and testing circuit boards are required by manufacturers of these industries to ensure product quality. An important element in competitive companies is quality assurance and product testing is also a key factor [1]. Furthermore, testing circuit boards guarantees conformity with the established requirements of the product design.

The testing process of circuit boards is divided in the following steps: test case description, test execution and result analysis [2]. Test case description includes the initial conditions and the steps to obtain an expected result. Test execution involves the test case description phase to obtain the expected result. Lastly, result analysis is the result from the test execution compared with the expected result described in the test case description. In order to pass the testing process, the result must be acceptable. However, one of the critical factors is the time lapse during the test execution phase, if the time lapse is increased, then a greater investment in the testing process will be required.

The equipment used to perform the test execution phase in this paper will be the ICT Medalist 3050. The ICT Medalist 3050 is a device used for testing circuit boards in the automotive industry by Keysight Technologies Company. This device can be configured for testing different types of circuit boards. However, modifying the ICT Medalist takes a considerable amount of time. The uNicom is an adapter used for circuit in test or test computer and the control unit by the CSM company. Some programming tools, such as uNicom II and uNicom 3 are used for developing, testing and validation [3]. These programmable tools can be connected to an ICT Medalist 3050 with a special board dedicated to connecting external devices. Till date, there is no board able to connect a uNicom II or a uNicom 3 at the same time. Therefore, the time lapse for testing execution will be reduced if there is a board able to perform this connection.

The time lapse required for testing preparation, testing verification, operator work, and other factors, especially in the automotive industry, can be interpreted as money per hour. The more time required for testing, the more quantity of money invested. Finding new alternatives to reduce the time in test execution while maintaining the quality assurance will add value to the product. Therefore, the aim of this project is to design a new alternative to reduce the time in test execution by creating a new board that can connect an uNicom2 and uNicom3 in the ICT Medalist 3050.

1. System design

There are two boards to accomplish the connection between the uNicoms and the ICT Medalist. The two boards in this design are called Board1 and Board2.

1.1. Components

The components selected for these boards are:

For Board 1:

- MinPin78
- DB78

For Board2:

- HE3321C0500
- CD0603-S01575
- FMMT491QTA
- DB62
- DB37
- DB78
- Arduino Nano
- Signal resistance
- Current limiter resistance

1.2. Board 1 design

Designing a PCB (Printed Circuit Board) follows a certain number of steps to accomplish a final design. The process can be divided in schematic stage and layout stage. Schematic stage consists in getting the specifications, and create the logic libraries and netlist import. The layout stage consists in creating the PCB in the program applying the constraints. For this project, the software used is PCBEditor. It is necessary to build the footprint for each component.

First, it is necessary to create the logical libraries. The logical libraries contain all the components used in a PCB, and their electrical connections in a schematic. After the creation of the logical libraries, it is necessary to create the physical libraries. The physical libraries contain all the footprints of the components used in the logical library.

The development of the logic libraries was made using Allegro Entry CIS software. The logic components for the PCB were divided into four library sections; discrete, connections, transistors and uC (microcontroller). Once the logic is obtained, it is necessary to make the connections for each component.

The connections required for Board 1 are show in Figure 1-1. For this board, the connectors MinPin78 and DB78 are need.

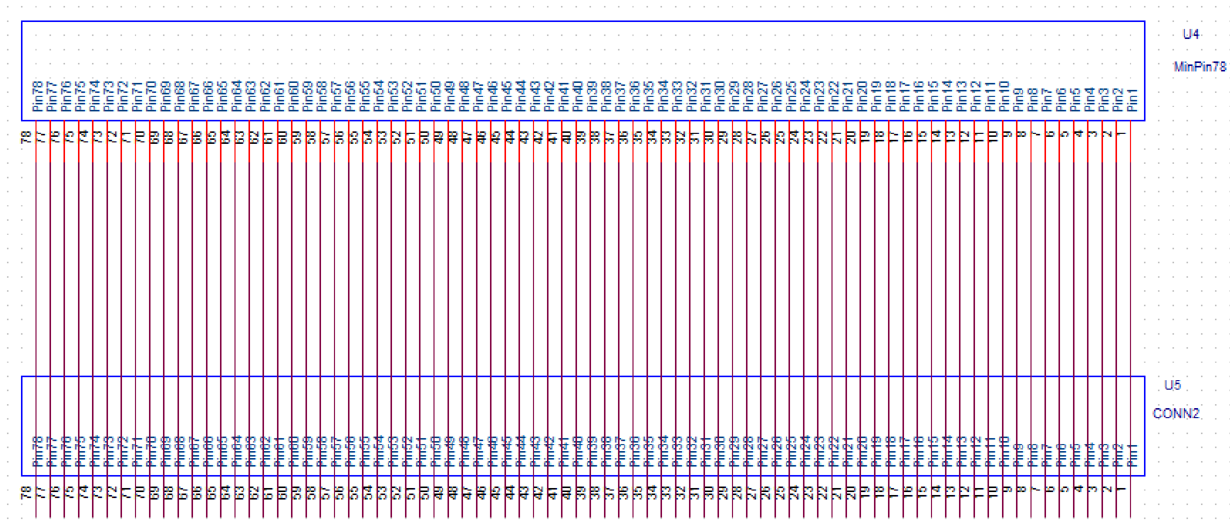

Fig. 1-1 Connection for board 1.

1.3. Board 2 design

1.3.1 Relay Specifications

Figure 1-2 shows the connections for each relay. Each relay has different connections to the MinPin78, the DB37 or the DB62. The relays are activated by transistors, in total 23 relays are used.

The relay HE3321C0500 was chosen for this project because this relay is inside the uNicom 2 and controls outputs inside the programming tool. The relay HE3321C0500 has the following specifications.

Table I. HE3321C0500 specifications [4]

Nominal coil voltage	5v
Coil resistance	125 Ohms
Release Vdc	0.5v
Maximum coil voltage	11v
Operate Vdc	3.75v

The Figure 1-2 shows a connection example between the DB37, DB62, DB78 and the relays. Each relay will have its own diode to decrease the peak of current generated by the inductor.

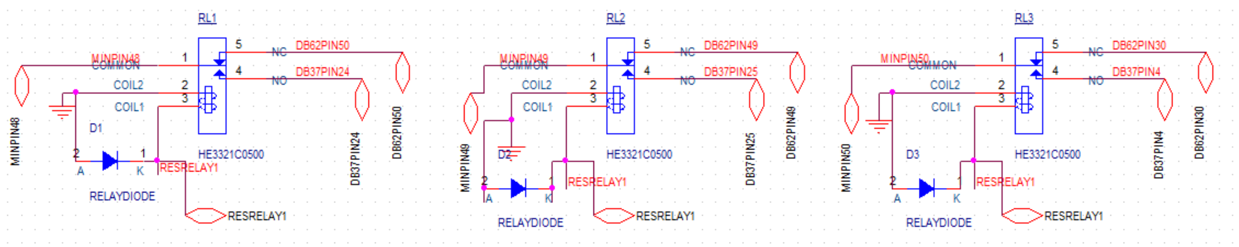

Fig. 1-2 Connection for relays.

1.3.2 Transistor calculus

In Figure 1-3, the transistor's basic configuration used in this project is shown. The power supply V3 is the signal from the microcontroller. The power supply V4 is the voltage for the relay coil, R2 is a current limiter, and R5 is the signal resistance used to limit the current from the microcontroller.

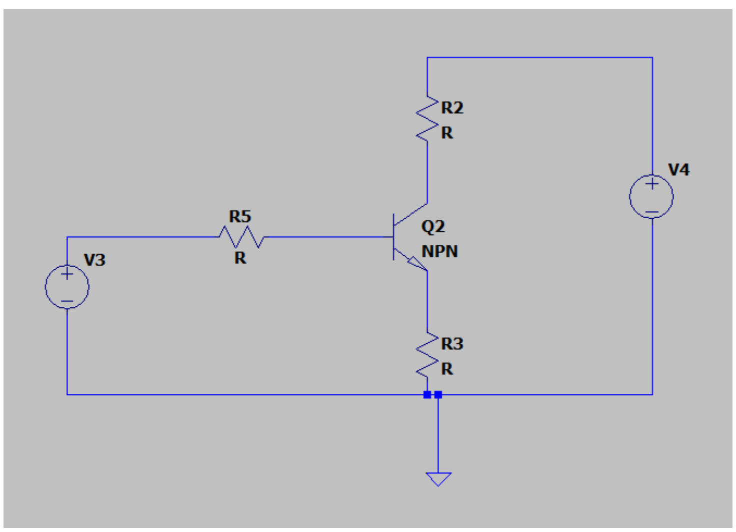

Fig.1-3 Transistor connection.

In Figure 1-4, the transistor configuration was set to accomplish the activation of each relay. The activation of each transistor was set by the microcontroller. Each transistor controls 5 relays.

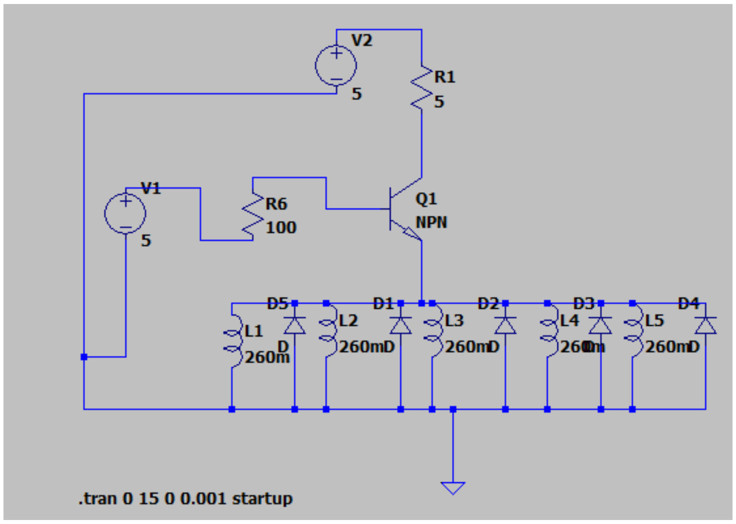

Fig. 1-4 Transistor connection to relays.

1.3.3 Simulation

The circuit was simulated using LTSpice. Figure 1-5 shows the current passing through R1.

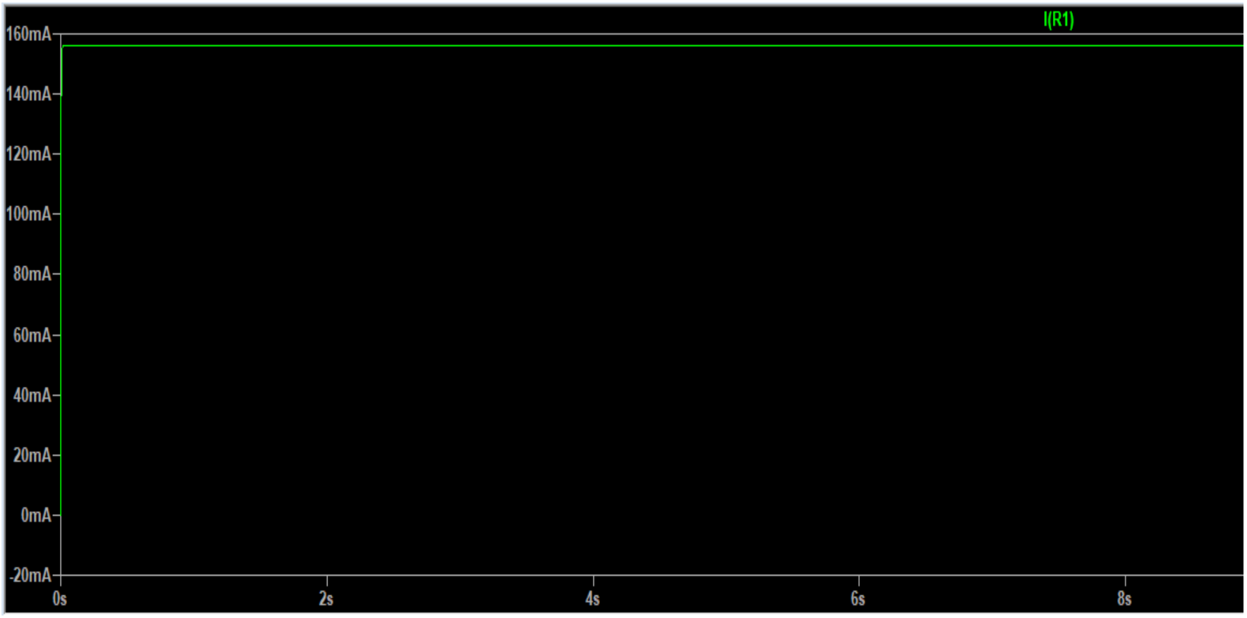

Fig. 1-5 Current for load 1.

Figure 1-6 shows the coil voltage at the relays. The voltage is near to 4v. The 4v voltage is enough to activate the relays.

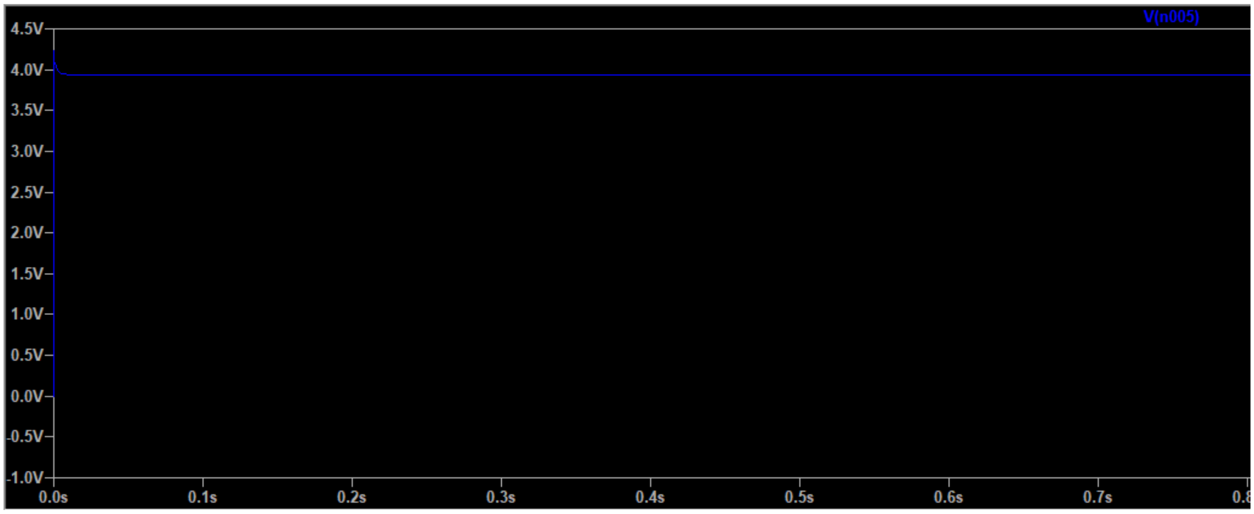

Fig. 1-6 Voltage for each coil.

Figure 1-7 shows the coil current through each relay, it is approximately 30 mA. This is enough to activate and maintain the relay current.

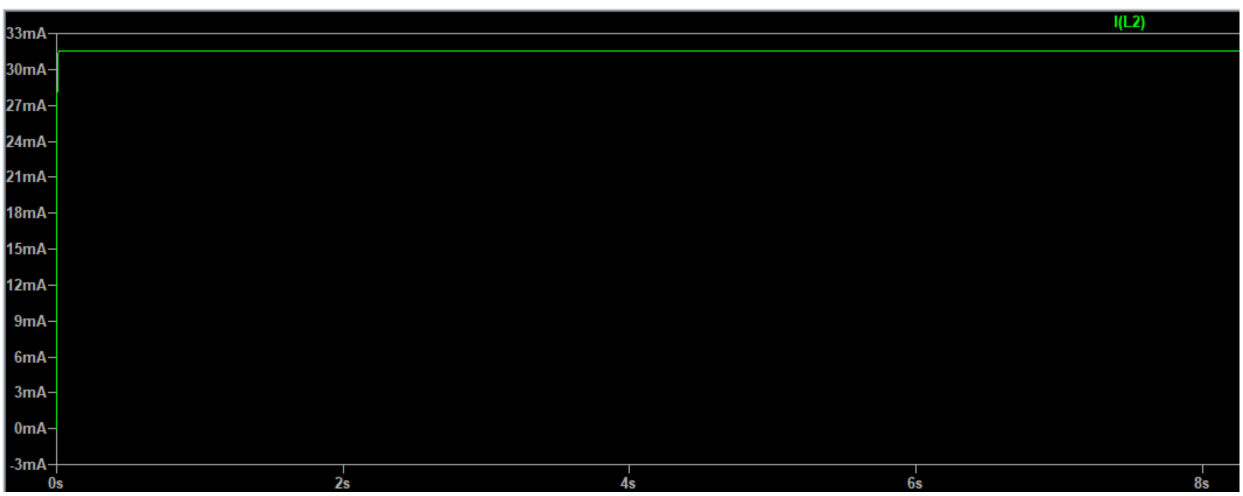

Fig. 1-7 Current for each coil.

According to the last 3 figures, it is necessary that each transistor supports the power requirement for each relay. The power consumption calculation is below:

$$Power = (Voltage)(Current) \qquad (1\text{-}1)$$

It can be assumed that each relay has 5v of voltage and 32 mA. The current is changed to 40 mA to give hardiness to the design. The power calculation afterwards is below:

$$1W = (5v)(200mA) \quad (1\text{-}2)$$

Each transistor must be able of handling 1 W without damaging. The FMMT491QTA datasheet says that the transistor can support 1A of continuous collector current. Collector-Base voltage of 80v and Collector-Emitter voltage of 60v. The FMMT491QTA is an applicable choice for this project [5]. The Figure 8 shows a schematics diagram of the connections. The signal resistor must be able to handle 1 W of power. A signal resistor with the package 2512 will be enough to hold 1W of power [6]. It is a good design practice to use components whose maximum power rating is half the calculated value.

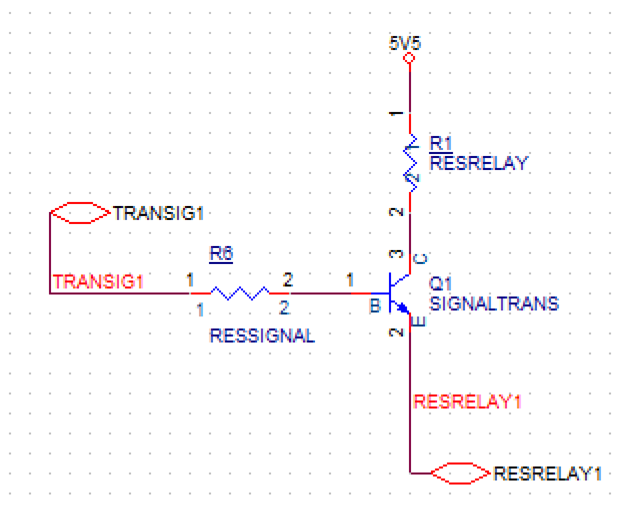

Fig. 1-8 Example for transistor connection.

1.3.4 uC schematic

The Figure 1-9 shows the connections between the microcontroller, the transistors, and the connections for input power. The microcontroller Unit will control 5 transistors. Each transistor will manage 3 relays except for the last transistor; since it will set 2 relays. Also, there is the connection for the DC input.

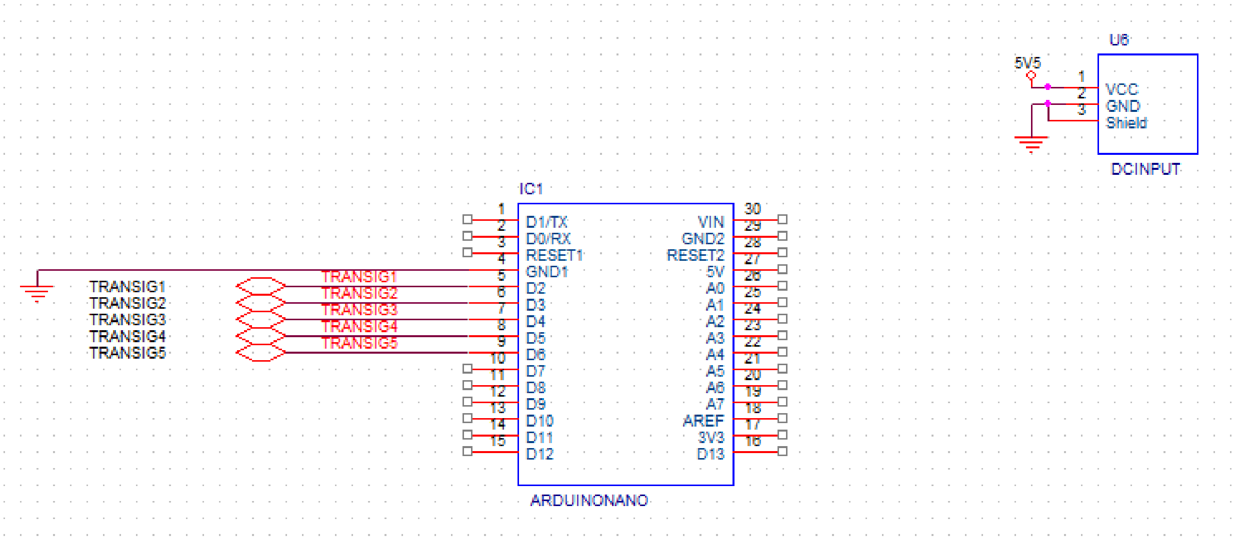

Fig. 1-9 Connection for the microcontroller and the input power supply.

1.3.5 Board 2 Schematic

Figures 1-10 and 1-11 show the connections between the DB62, DB37 and the Minpin78 used in the board.

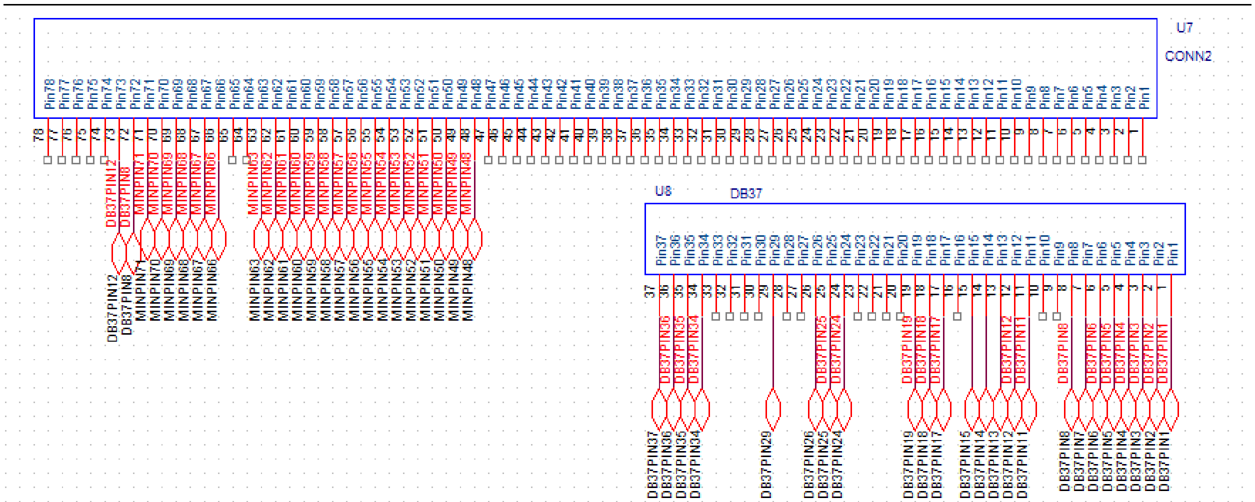

Fig. 1-10 Connections for the DB37 and Output in Board 2.

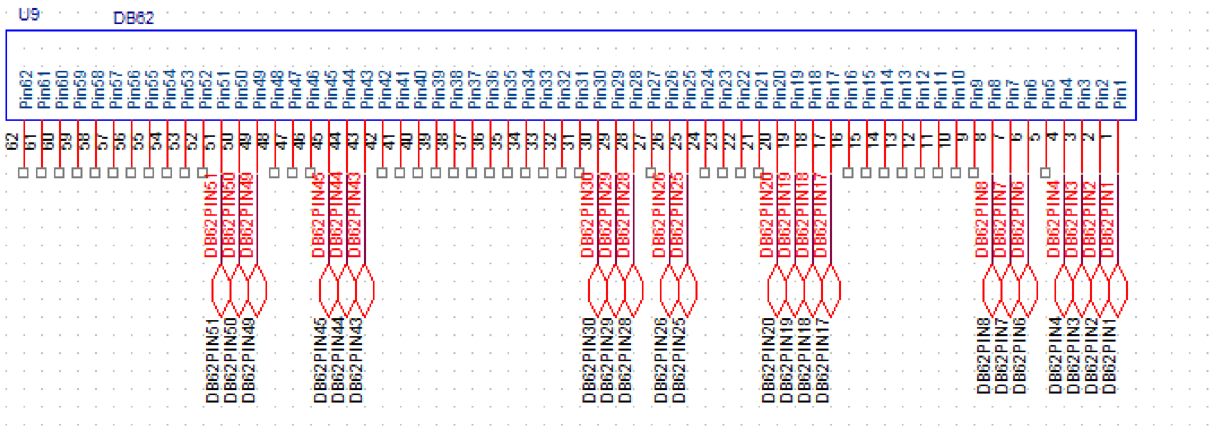

Fig. 1-11 Connections for the DB62 in Board 2.

After completing the schematic, it is necessary to build the NetList. The NetList creation is the last step of the schematic stage. Once the Nerlist is ready, it must be imported to the PCB Editor software. Mouser company have the software Library Loader to load the footprint for all its components. All the footprints were modified accor

1.3.6 Board 2 general design

The board design consists of a PCB of 4 layers. Bottom and top are used for signals and the middle layers are used to ground and power signals. Due to common signal integrity guidelines, it is necessary to have ground next the signal routes. Some routes are in layer 2 due the number of routes used in Board 2. The cooper in the inner layers and the outside layers is of 1oz size. The dielectric material used in the designed boards is a FR-4 because it is cheap, common, and meets the electrical requirements for this project.

2. Final design

2.1 Board 1 design

Figures 1-12 and 1-13 show the Board 1 final design. The routing was design between the DB78 and the MinPin78. The layers 2 and 3 are ground in order to preserve the signal in all the way across the board.

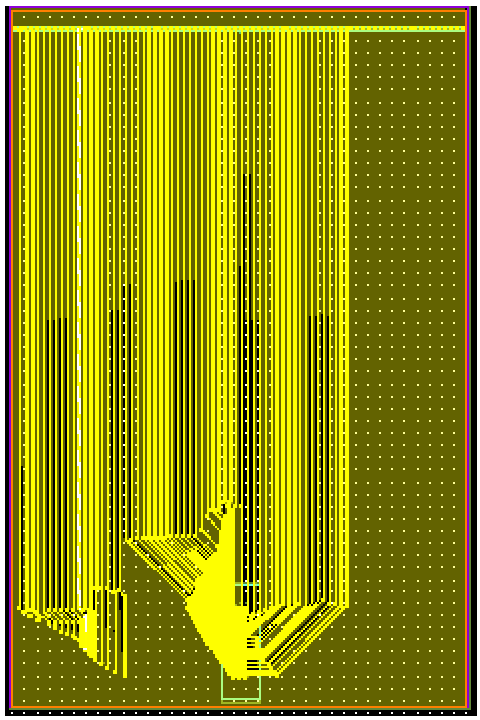

Fig. 1-12. Bottom connections for board 1.

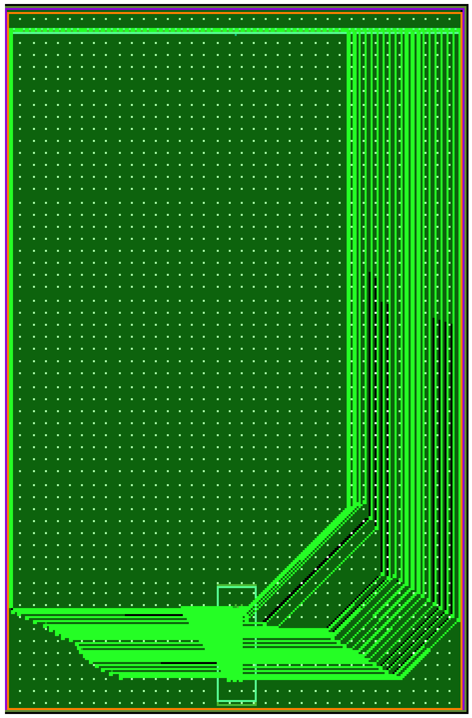

Fig. 1-13 Top connections for board 1.

2.2 Board 2 design

Figures 1-14 and 1-15 show the Board 2 final design. The main purpose of this board is to connect the DB37 and DB62 with the DB78. The microcontroller is in middle of the board. The power plug is in the right side of the board.

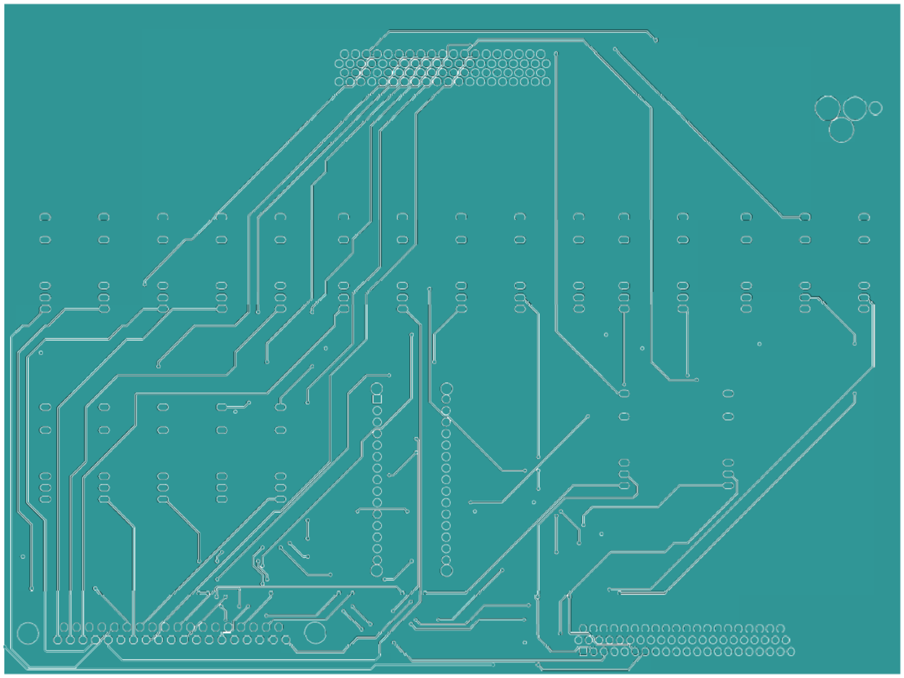

Fig. 1-14 Connections in bottom for Board 2.

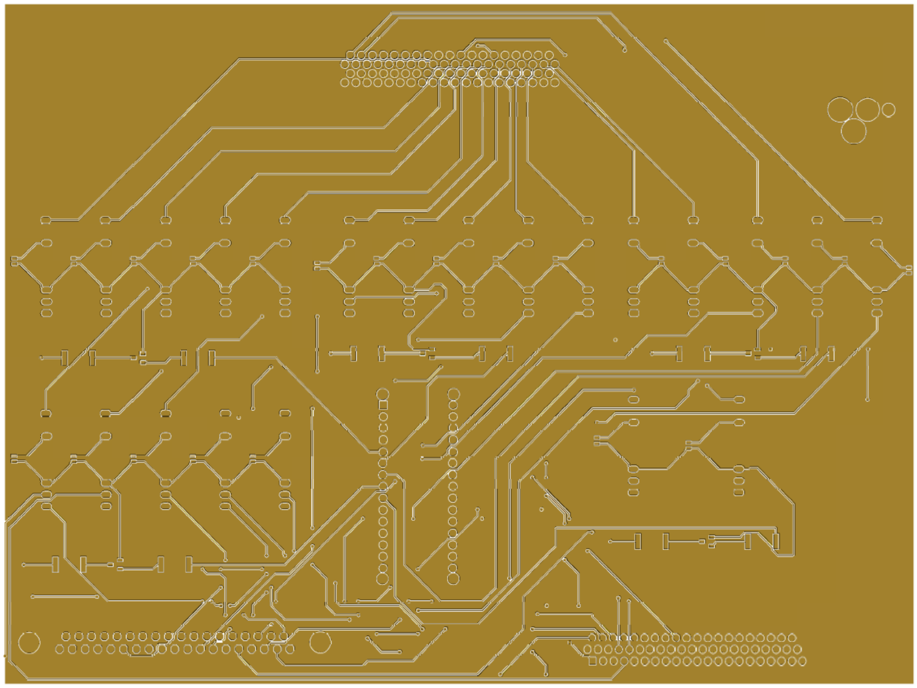

Fig. 1-15 Connections in top for Board 2.

3. Future work

These boards were designed for testing purposes only. Some improvements for these designs in the electronic industry include substituting the DB78 connector with another popular connector in the industry, using a more robust microcontroller and verifying the following points: the board size inside the ICT Medalist 3050, the connector MinPin78, DB62 and DB37 dimensions, and changing the power plug for a popular alternative used in the industry. In case Board 2 needs to be modified, it can be substituted with a new board for another implementation.

Conclusion

Boards 1 and 2 were designed as a proof of concept. The keys aspects considered to make the design functional were to simulate the electrical design, verify the mechanical concepts, such as board size, and types of connectors. Once the boards are built they can be tested to know how the signal integrity is modified by of the magnetic field generated by the inductor. The purpose of these board designs will help reduce the time lapse for testing and, in consequence, save money.

References

[1] Lamberg, K. «Model-Based Testing of Automotive Electronics». En *Proceedings of the Design Automation Test in Europe Conference*, 1:1-1, 2006. https://doi.org/10.1109/DATE.2006.243990.

[2] Kum, D., J. Son, S. Lee, y I. Wilson. «Automated Testing for Automotive Embedded Systems». En *2006 SICE-ICASE International Joint Conference*, 4414-18, 2006. https://doi.org/10.1109/SICE.2006.314687.

[3] «Programming tool UNICOM». Accedido 14 de julio de 2019. https://www.csm.de/en/products/programming-tools.

[4] «HE3321C0500 Hamlin / Littelfuse | Mouser». Mouser Electronics. Accedido 18 de julio de 2019. https://www.mouser.mx/ProductDetail/934-HE3321C0500.

[5] «FMMT491QTA Diodes Incorporated | Mouser». Mouser Electronics. Accedido 18 de julio de 2019. https://www.mouser.mx/ProductDetail/621-FMMT491QTA.

[6] «RK73HW3A2RTTE1501F KOA Speer | Mouser». Mouser Electronics. Accedido 18 de julio de 2019. https://www.mouser.mx/ProductDetail/N-A.