**INSTITUTO TECNOLÓGICO Y DE ESTUDIOS SUPERIORES DE MONTERREY**

**Imagen que contiene Interfaz de usuario gráfica

Descripción generada automáticamente**

**Microcontrollers Laboratory**

**Practice 01, 02, 03**

**Names:**

**Marcos Rubén Flores Lara A01351360**

**Edgar Alejandro Rodríguez Gallegos A00822028**

**Date: March 29, 2022**

# Report 01 – Expansion Board Fabrication

Practice 01, 02, 03

ITESM Campus Mty

Marcos Rubén Flores Lara

A01351360

Edgar Alejandro Rodríguez Gallegos A00822028

Horario: 14:30-16:00

**horizontal line**

Instructor: [Matias Vázquez Piñón](https://experiencia21.tec.mx/courses/261838/users/136680)

Due Date: 29/03/2022

# Part I: Schematic Capture

# Part II: PCB Layout

# Part III: PCB Manufacturing

**INTRODUCTION:**

* A brief explanation of the work carried out in labs 1, 2, and 3.

Along the various practices he had during the periods that involved the first 3 laboratories que carried out the development of a PCB (Printed Circuit Board), from the mere beginning of the design to the actual involvement of dipping the board in acid to clear the ink out.

* What are PCBs used for?

PCB´s are the foundational building block of most modern electronic devices, Semiconductors, connectors, resistors, diodes, capacitors, and radio devices are mounted to, and “talk” to one another through the PCB.

* Why is important as an engineering student to know how to develop PCBs?

Since circuits are one of the most important practical subjects used in engineering, it is very important to know how to apply this in any project that engineers may have to develop across their different applications, from the medical order to the aerospace discipline. PCBs are found everywhere a complex system is required.

* What other fabrication techniques are available for PCB fabrication?

Printed electronics is an all-encompassing term for the printing method used to create electronic devices by printing on a variety of substrates. Applications where more flexibility is needed, such as wearable uses.

**PROCEDURE:**

* Explain in a series of steps, the PCB fabrication process you followed, including schematic capture, layout and fabrication.

First, we had to design the schematic for our board and to do it we used Proteus, a software that let us simulate a board and all the components we were going to use, afterwards we collocated the digital components in an array that we believed was the most practical and ordered.

Diagrama, Esquemático

Descripción generada automáticamente

Ilustración 1: Expansion Card used

Following this, we had to use the resulting expansion card so we could generate a Layout where we gave as an input the dimensions we wanted for our actual pins, that way we transformed a digital schematic into a PCB layout. Basically, selecting the components previously generated, and along with the desired dimensions the software was able to create what was the most adequate array for the card.

Interfaz de usuario gráfica

Descripción generada automáticamente

Ilustración 2: PCB Layout

Finally, we had to print this Layout in a LaserJet ink printer on a sublimate paper, and got a 4x6 in, copper board where we ironed the previous print, this process of ironing the schematic into the board took around 1:30 to 2 hours, and when we had finished we dipped the board in water so that the remaining paper detached from the board, that way we ended up with a clearer image on the board.

Imagen que contiene edificio, firmar

Descripción generada automáticamente

Ilustración 3: Clear Board

Then we dipped it into acid so that the copper washed away and afterwards applied water to the board and wash away the ink.

Imagen que contiene naranja, tabla, computadora

Descripción generada automáticamente

Ilustración 4: Board in acid

Once the board is completely clean, we proceed to perforate and weld the components that we set on the layout.

Puerta de entrada

Descripción generada automáticamente con confianza baja

Ilustración 5: Perforated board

GitHub Links:

**CONCLUSIONS:**

**Marcos:** I really enjoyed this set of practices because up to this point in the career I had jus heard about this type of procedures, whether it was from teachers or from YouTube videos that mentioned this process as fundamental for the formation of an engineer.

I´m very proud of the result, and it was definitely harder than what it appeared as a first view. All the knowledge I had gained in specific subjects was put into test during the practices and the knowledge that I acquired during the investigation part of the report I believe it to be very rich for my future projects.

**Alejandro:** I believe that the practice we developed, every single part of it had its own challenges, since I was responsible for the welding of the board I found myself struggling with real difficulties since its not a simple process to do, besides the timing between practices was something that challenged us to perform our best since both of us have to develop professional practices as interns and also had to have excellent results with this PCB fabrications .

* Conclusions
  + Each teammate writes a short paragraph (~100 words) telling the experience of fabricating a PCB from scratch. There is no need to be technical in your narrative; instead, tell what you liked the most throughout the process and how you solved the faced challenges, just as you' would tell a friend about your work.