**Chittagong University of Engineering and Technology**

**Department of Electrical and Electronic Engineering**

# Design and Implementation of Digital Capacitance Meter

**Group No:23**

**Group Members:**

**1.Eram Khan ,1902065**

**2.Abdul Kader Nayon,1902066**

**3.Md. Mohiuddin Insun,1902067**

**4.Shafi Sami,1902103**

**Date of submission: 20/08/2023**

# Abstract

A capacitor's size is characterised by its capacitance, which is expressed in Farads and is a sort of passive electrical component. The labelled value might not correspond to the component's real value because of the tolerance range for the capacitor. The fact that our current measurement devices, like the multimeter, cannot measure capacitance at low levels is another factor in the importance of this study. Analogue capacitance metres continue to dominate the market. The goal of this study was to create the Digital Capacitance Metre, a device for measuring capacitor values in a digital lab that ranges from 1nF to 100uF.We utilised the A PIC16F628a microcontroller for our project because microcontrollers with built-in timers can measure capacitances by calculating how long it takes an RC circuit to charge or discharge.

# Contents

# List of Figures

# List of Tables

# Listings

# Chapter 1

# Introduction

Capacitors are frequently used in electronic devices. The circuit present in a power bank can be used as an example to demonstrate the use of capacitor components. We are all aware that power banks can be used to recharge various electronic gadgets, including cell phones and other portable devices. The capacitance of the component will be disclosed on the label attached to the packaging of the capacitor itself. However, the capacitance value sometimes listed on packaging differs from the actual value.Due to the tolerance of the capacitor, the value shown on the box is within the capacitor's useful range. Older packaging for capacitors could have unreadable numbers. Performance of capacitor-based circuits is impacted by this [1]. Current, resistance, and voltage are all measured using multimeters. Capacitance of capacitors is typically measured by multimeters up to 2nF. A digital capacitance measurement tool must be created to address this.

# Chapter 2

# Working and Design of Circuit

The time constant, which is a fundamental characteristic of capacitors, is necessary for a capacitance metre to function. The duration of time required for a capacitor's voltage to increase by 63.2 percent is known as the time constant. The microcontroller will receive a signal in the form of a steadily increasing voltage during the charging process through an ADC. The microcontroller's timer circuit is in charge of figuring out how long it will take to reach the desired value V. T = RC (1) can be used to express how long it takes to charge a capacitor to the value of one time constant (T). The capacitor's terminal voltage during charging is determined by the formula Vc = Vs(1-e-t/RC). Where t shows how long it has been since the voltage was applied, Vc shows the voltage of the capacitor, Vs shows the voltage of the source, and RC shows the time constant of an RC circuit. Since the resistor in the microcontroller circuit has a known value, we will use its pins to charge the capacitor through it. The peak voltage of the capacitor can be calculated using the analog-to-digital converter (ADC) of

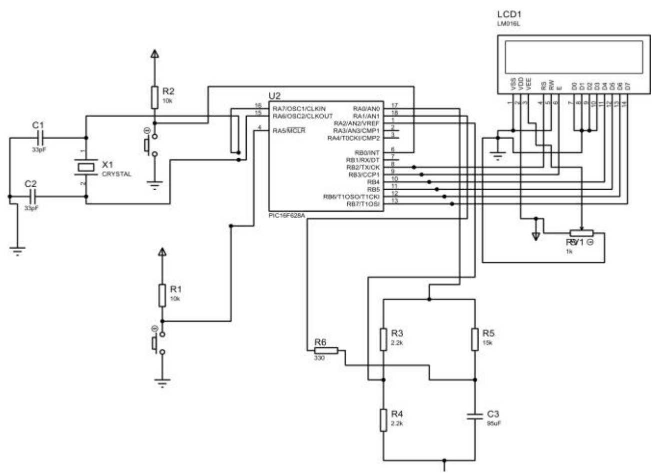


Figure 2.1: Circuit Diagram

the microcontroller.The charge is stopped when the voltage reaches 63.2% of the total charge, and the capacitance is then determined. Using the circuit's resistance, measured voltage, and duration to 63.2% of its maximum charge, we may determine the capacitance. C is determined by:

C=T/R.

## 2.1 Mamthematical Formula

The formula for the time constant (T) is : T = RC.

Vc = Vs(1-e-t/RC)

Where t shows how long it has been since the voltage was applied, Vc shows the voltage of the capacitor, Vs shows the voltage of the source, and RC shows the time constant of an RC circuit.

### 2.1.1 Program code

```
LCD module co nne ctio nss
bit LCD RS at RB2 bit ;s
bit LCD EN at RB3 bit ;s
bit LCD D4 at RB4 bit ;s
bit LCD D5 at RB5 bit ;s
bit LCD D6 at RB6 bit ;s
bit LCD D7 at RB7 bit ;
sbit LCD RS Direction at TRISB2 bit ;s
bit LCD EN Direction at TRISB3 bit ; s
bit LCD D4 Direction at TRISB4 bit ; s
bit LCD D5 Direction at TRISB5 bit ; s
bit LCD D6 Direction at TRISB6 bit ; s
bit LCD D7 Direction at TRISB7 bit ;

sbit Va at RA0 bit ;
sbit Switch at RB0 bit ;
```

```
char  message1 [] = ”Capacitance ”;

char  message2 [] = ”Meter ”;


unsigned  in t  T_Value ,  Num;f
l o a t x , cap , y ;
unsigned  sho rt i , j ,  TimerValue ,  OverRange  =  0;char
Capacitance [ ]  =  ”00.000          ” , txt [ 2 0 ] ;



void  in t e r r u p t () {
   i f ( PIR1 . TMR2IF) {
  TMR2   =   TimerValue   ;
  Num  ++;
   i f (Num  >  9999 )  OverRange  =  1;  //  Range  i s  99.99  uF
  PIR1 . TMR2IF  =0;  //  Clear  TMR0  in t e r r u p t  f l a g
  }
}

void   Display _ Cap ( unsigned   in t  n) {
Capacitance [ 0 ]  =  n/10000  + 48;
 Capacitance [ 1 ]  =  (n/1000)%10  + 48;
 Capacitance [ 3 ]  =  (n/100)%10  + 48;
 Capacitance [ 4 ]  =  (n/10)%10  + 48;
 Capacitance [ 5 ]  =  (T_Value ∗10)/153  +  48;
 Lcd_Cmd( _Lcd  _Clear ) ;
 Lcd _Out ( 1 ,  1 ,  ”C(uF ) : ” ) ;
 x=n+(T_Value  / 2 0 ) ;
 cap =(16∗x +0. 7271 ) ;
 Float To Str ( cap   ,   txt ) ;
 i f ( ! n)
 Lcd _Out ( 2 ,  1 ,  Capacitance )  ;e
 l s e
 Lcd _Out ( 2 ,1 ,  txt ) ;
```

```
}

void reset(){
 TRISA = 0 b00000100 ;
 CMCON = 7;
 RA1_bit = 0;
 Delay_ms(2000);TRISA
 = 0 b00000110 ;CMCON
 = 5;
}

void main(){

   char cap_size;TRISB
   = 0 b00000001 ;PORTB
   = 0;
   TRISA = 0 b00000110 ;
   OPTION_REG.T0CS = 0;
   INTCON.GIE = 1; //Enable global interrupt INTCON.
   PEIE = 1; //Enable peripheral interrupt

   // Configure Timer2 module
   PIE1.TMR2IE = 1;   // Enable Timer2 interrupt
   T2CON = 0;         // Prescaler 1:1, and Timer2 is off initially
   PIR1.TMR2IF =0;    // Clear int bit

   // Configure Comparator module
   CMCON = 5;         // Independent comparator between RA1 (−) and RA2(+)

   Lcd_Init();
   Lcd Cmd( Lcd Clear );
   Lcd Cmd( LCD CURSOR OFF);
```

```
Lcd Out ( 1 , 1 , message1 ) ; Lcd
Out ( 2 , 1 , message2 ) ; delay
ms ( 2000 ) ;
Lcd_Cmd( Lcd Clear ) ;

Lcd _Out ( 1 ,  1 ,  "C =  " ) ;
Lcd Out ( 1 , 5 , Capacitance ) ;Va
= 0 ;
TimerValue  =  112 ;   // 104 + 4 c lo c k delay  on branching  to  ISR and o th e rswhile
(1) {

 Num     =     0     ;
 OverRange  =0;
 Lcd_Cmd( Lcd  Clear ) ;
 Lcd Out ( 1 ,  1 ,  " Testing . " ) ;
 Lcd Out ( 2 ,  1 ,  " . . . " ) ;
 TMR2 =  TimerValue ;           // I n i t i a l value   of Timer2  f o r  30 us  delayVa
 = 1 ; // apply  vo lta g e
 T2CON.TMR2ON = 1 ; // s t a r t timer
 while (CMCON.C2OUT)  {
   i f ( OverRange )  ;
 }
 T2CON.TMR2ON =  0;  //  stop  timerT
 Value  = TMR2 − TimerValue ;
 Va  = 0 ;
//−−−−−−−−−−−−−−−−−−−−−−−−−−−−−−−−−−
if ( ! OverRange ) {
   Display  Cap (Num∗10) ;
}
else {
   OverRange = 0 ;
   Lcd Cmd( Lcd  Clear ) ;
```

```
      Lcd_Out(1, 1, "Out of Range!");
    }
    reset();
  }

}
```

- 

-

- - -

# Chapter 3

# PCB layout

The components and connections required for precise capacitance measurements are accommodated in the PCB layout produced using Proteus for the capacitance metre project. It adopts a condensed and systematic layout to maximise space usage and reduce signal interference. The PIC16F628A microcontroller, resistors, capacitors, and other supporting components are positioned appropriately in the layout. To preserve signal integrity and prevent noise-induced inaccuracies, special consideration is paid to the routing of traces. The configuration makes it simple to interface with the microcontroller by including defined locations for power supply cables and an LCD display. Overall, Proteus' PCB layout is well-organized, making it easier to construct a useful and effective capacitance metre.

The design and production of electronic circuits must include a PCB (Printed Circuit Board) layout. To build a working circuit, electronic components must be arranged and connected on a board. The PCB layout procedure is crucial for ensuring the reliability, appropriate operation, and signal integrity of the electronic system. Here is a summary of the PCB layout procedure:

1.Schematic Design: Making a circuit schematic is the initial stage in PCB layout. The electrical connections between different circuit components are shown in this schematic illustration. It acts as the layout's blueprint on the PCB.

2.Component Selection: After the schematic is finished, the right electronic parts are chosen depending on the circuit's specs and needs.

3.Board Size and Shape Definition: The PCB designer determines the size and shape of the board based on the available space in the product housing and the circuit's complexity.

4.Placement of Components: The chosen components are organised on the PCB layout in this step. Component placement should be optimised for effective signal flow, thermal considerations, and board size reduction. The important parts might go in first, and the rest might be structured around them.

5.Routing Traces: Electrical signals are transported between components using traces, which are copper connectors. The necessary electrical connections are made by the PCB designer by routing these traces. To preserve signal integrity and prevent interference, signal traces, power traces, and ground traces are all meticulously routed

6.Layer Stacking: Copper traces and planes can be arranged in a number of layers on PCBs. A 4-layer PCB also contains two internal copper layers, whereas a 2-layer PCB only has copper on one side of the board. The intricacy of the circuit and the requirement for signal integrity determine how many layers are used.
7.Ground and Power Planes: Large copper surfaces called planes serve as reliable points of reference for ground and power connections. They improve power distribution and contribute to noise reduction. Effective grounding and power plane design are essential for the PCB's overall performance.

8.Signal Integrity Analysis: Signal integrity analysis techniques are used in high-speed circuits to look for potential problems such reflections, crosstalk, and timing errors. This analysis aids in design optimisation for signal transmission that is reliable.

9.Design Rule Check (DRC): A Design Rule Check is performed on the PCB layout before it is put into production. against ensure manufacturability, this automated method compares the design against the manufacturer's capabilities and design guidelines.

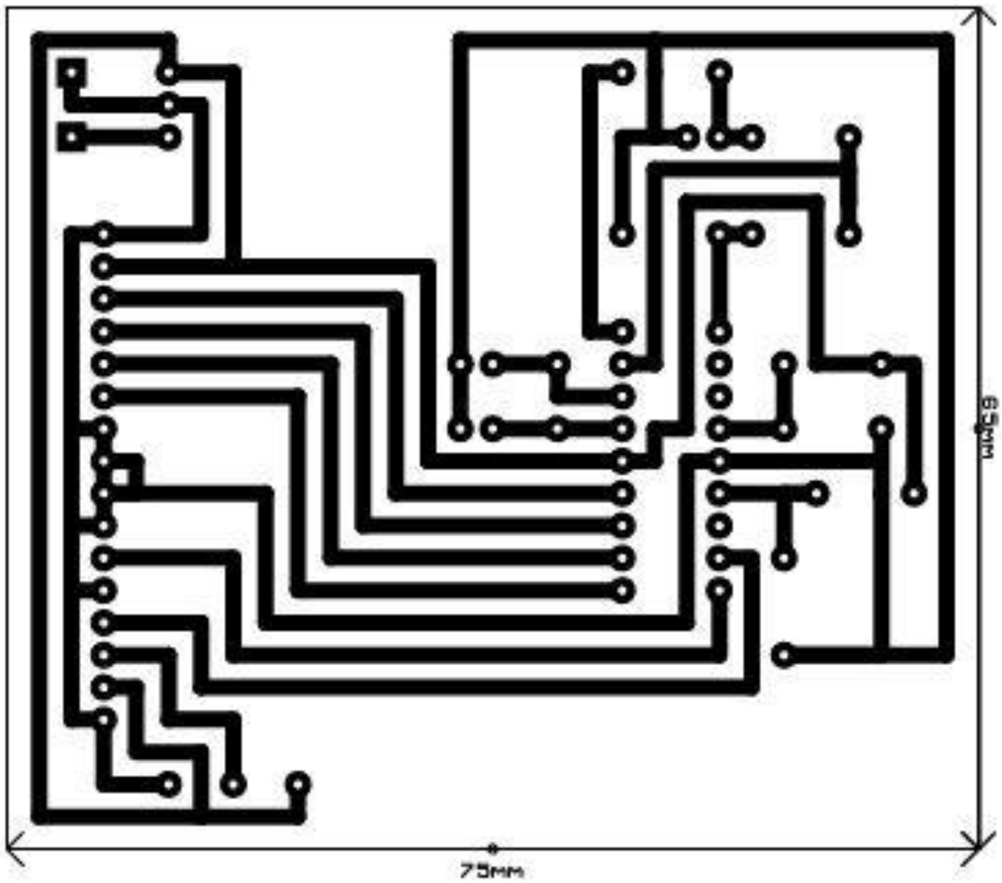


Figure 3.1: PCB Layout.

# Chapter 4

# Breadboard Implementation

Electronic circuits can be prototyped using the breadboard method, also known as breadboarding, without the use of solder. Before switching to a more long-term solution like a printed circuit board (PCB), it's an easy approach to quickly test and iterate circuit ideas. The breadboard implementation procedure is described in the following manner:

1.Select Components:

Choose the resistors, capacitors, integrated circuits (ICs), LEDs, and wires that you want to utilise in your circuit.

2.Choose a Breadboard:

Typically, breadboards come in a variety of sizes and layouts. Select a breadboard based on how sophisticated your circuit is. On breadboards, you can place parts and wires into rows and columns of connected sockets or holes.

3.Component Placement:

Insert Components: Put the parts in the sockets on the breadboard. Use the leads on through-hole components, such as LEDs and resistors, to attach them to the breadboard.

ICs:Put any integrated circuits (ICs) you're using very cautiously into the breadboard. To show their orientation, ICs frequently have a notch or dot.

4.Wiring:

Use Jumper Wires: Jumper wires are employed to establish connections between various breadboard components. They come in a variety of colours and lengths. Connect components using wires by inserting them into the proper sockets.

Power Rails: Breadboards have power rails (often labeled as "+" and "-") along the sides. These provide power and ground connections for your circuit. Insert the necessary power wires to these rails.

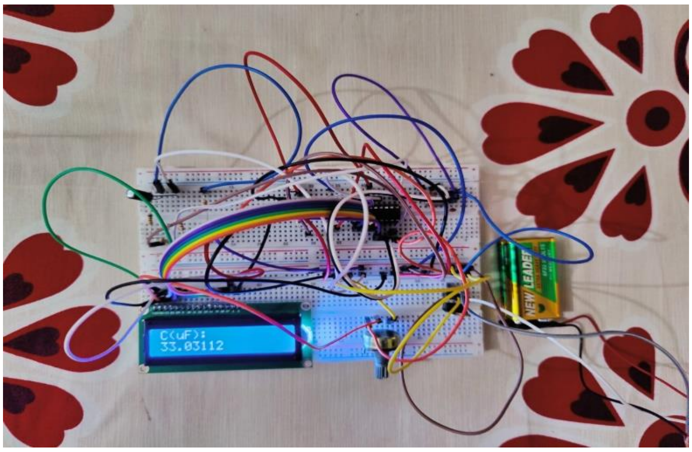


Figure 4.1: Bread Board Implementation

5.Plan the Wiring:

Plan the connections on paper or online before putting the cables. This ensures a clean layout and reduces misunderstanding.

6.Wire Routing:

To reduce the likelihood of unintentional shorts and to keep a clear perspective of your circuit, route wires logically and neatly.

Avoid Long Wires: In order to minimise signal interference and ensure signal integrity, keep cable lengths as minimal as possible.

7.Test the Circuit:

Power on the circuit after the components have been installed and connected. To supply the circuit with the necessary voltage, use a suitable power source.

Observe LED Status: If your circuit contains LEDs, they can offer visual feedback regarding the circuit's performance.

8.Iterate and Troubleshoot:

With breadboards, circuit modifications are simple. Check connections, component values, and component orientation to see if your circuit is operating as it should.

9.Documentation:

It's a good idea to record the circuit connections as you wire up and test the circuit. To achieve this, use paper, a digital platform, or even take images.

# Chapter 5

# PCB Implementation

Our capacitance metre was first prototyped on a breadboard before the concept was put into action on a PCB. Our ability to quickly and simply test and validate the circuit's functionality was made possible by the breadboard implementation. In order to troubleshoot and make any necessary changes to the circuit design, this breadboard prototype functioned as an effective working platform. Before moving forward with the more permanent PCB implementation, it provided a flexible and affordable way to confirm the viability of our capacitance metre.

The process of physically generating and constructing a printed circuit board based on the PCB layout design is known as PCB implementation, often referred to as PCB assembly or PCB manufacturing. From gathering the materials to testing the final PCB, there are various phases involved.

Here's a description of the typical PCB implementation process:
Material Preparation:

1.PCB Substrate: The first step in the process is choosing the right substrate material, often FR-4, an epoxy laminate with fibreglass reinforcement. Based on the needs of the design, the thickness and other characteristics of the substrate are selected.
2.Copper Foil: Conductive layers for traces and planes are created by laminating copper sheets onto the substrate. The current-carrying capability and other electrical factors affect the copper layer's thickness.
3.PCB Imaging:
Photoresist Application:The copper-clad substrate is covered with a layer of photoresist that is light-sensitive. The circuit pattern will be made using this layer.
PCB Exposure: Photomasks with the desired circuit pattern are made using the PCB layout design's Gerber files.

The pattern is then transferred to the photoresist layer by aligning the photomasks and exposing them to UV light.

4.Etching:

The PCB goes through a chemical etching process after being exposed. The exposed sections of the photoresist are etched away, leaving the required copper traces and patterns, while the photoresist's unexposed regions shield the copper layer.

5.Drilling:

Holes are drilled into the PCB to accommodate component leads and vias that connect different layers. These holes are plated to provide electrical continuity between the layers.

PCB Plating:

If the PCB has numerous layers, vias are created by plating copper into the drilled holes. To further improve solderability and safeguard the exposed copper surfaces, surface finish plating is also used.

6.Solder Mask Application:

The PCB surface is completely covered with a solder mask, with the exception of the locations where components will be soldered. During soldering, the solder mask helps shield the PCB from impurities and shorts.

7.Silkscreen Printing:

To help with component placement and identification, a layer of white ink containing component identifiers, reference designators, and other information is placed to the PCB surface.

8.Component Placement:

Electronic components are placed on the PCB using automated machinery or hand assembly procedures. The PCB layout design serves as the foundation for component placement.

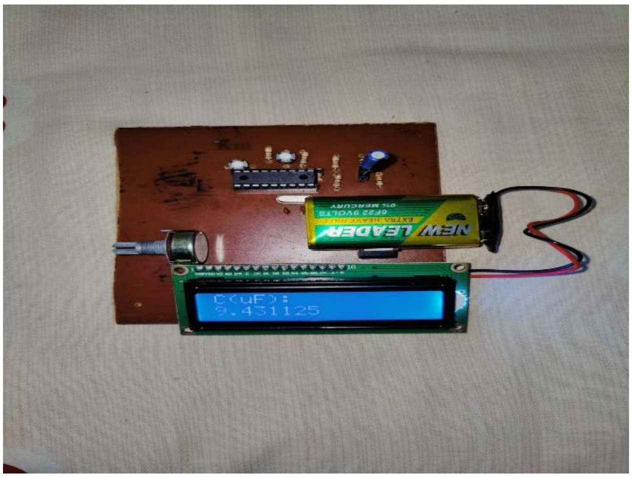


Figure 5.1: PCB Implementation

# Chapter 6

# Price Table

Table 6.1: Price of components

| Description | Amount | Price |
|---|---|---|
| PIC 16F628A | 1 | 350 |
| Breadboard | 2 | 260 |
| Capacitor-µF | 3 | 200 |
| 9V Battery | 1 | 70 |
| Resistor(300,10k,2.2,15k) | 20 | 50 |
| Crystal oscillator(4MHz) | 1 | 15 |
| Voltage Regulator(5V) | 1 | 10 |
| Jumper Wire | 50 | 100 |
| Capacitor | 10 | 10 |
| 1k Pot | 1 | 25 |
| LCD(16*2) | 1 | 200 |
| Push Switch | 2 | 10 |
| Ccb Board | 1 | 90 |
| Others | | 100 |

# Chapter 7

# Limitations and Future Improvement

1.It responds very quickly to changes in environmental variables like humidity, temperature, etc. This will have an impact on the performance.

2.It is difficult to compare capacitance and resistance measurements. Inductive sensor types are more accurate than capacitive proximity sensors.

3.Non-linear behaviour is produced. Frequency affects the output impedance.

4.The capacitance may change as a result of moisture and dust, which could result in mistakes.

Future Improvements:

First, more exact resistors and capacitors can be added to the circuit to further improve measurement precision. Measurement disparities can also be decreased by putting calibration methods into place and accounting for any potential offset errors. A temperature compensation mechanism could enhance the meter's functionality in a variety of environmental situations. Further noise reduction and improved capacitance measurements can be achieved by investigating new algorithms and filtering methods.

# Chapter 8

# Conclusion

In this project, a low-cost microcontroller-based digital capacitance metre with a 1nF to 1uF range was created. The meter's readings fell within 95 percent of what we predicted. This project is great for exact measurements as it would be simple to integrate into a laboratory environment. An Internet of Things (IoT)-based system that enables the online archiving and display of pertinent data could be useful for the capacitor industry's production sector. We also want to emphasise improving accessibility.

# Chapter 9

# Plagiarism Declaration

All information contained in this report, with the exception of any statements to the contrary, is the result of my own work. Furthermore, no portions of this material were taken verbatim from other sources. I am aware that any instances of plagiarism and/or the unauthorised use of resources from other parties will be taken seriously.

# References


[1] Samosir, A. S., Sumantri, J., No, B., Lampung, B. (2016). Implementasi Alat Ukur Kapasitansi Digital (Digital Capacitance Meter) berbasis Mikrokontroler. Electrician,10(1), 21–26. https://electrician.unila.ac.id/index.php/ojs/article/view/211

[2] Kolle, C., and P. O'Leary. 'Low-cost, high-precision measurement system for capacitive sensors.' Measurement Science and Technology 9.3 (1998): 510.

[3] W.Q. Yang, T.A. York, 'New A.C. based capacitance tomography system', IEEE Sci. Meas. Technol. 146 (1) (1999) 47

[4] Matsumoto H., Shimizu H. and Watenabe K.,' Switched-capacitor charge-balancing analogtodigital converter and its application to capacitance measurement', IEEE Trans.onInstrum. Meas., Vol. IM-36, No. 4, pp. 873-877,Dec. (1987)

[5] MA Kumar, BR Murthy, KV Madhav, 'Microcontroller based capacitance meter', International Journal of Computer Application and Engineering Technology 1 (2012) .pp.41-48

[6] Reventer F., Gasulla M. and Pallas-Areny R., 'A low cost microcontroller interface for low value capacitive sensors', Proc. IEEE IMTC, Italy, Como, May (2004) pp. 1771- 1775.