ATME College of Engineering

DEPARTMENT OF ELECTRICAL & ELECTRONICS ENGINEERING

LABORATORY MANUAL

Scilab for Electrical and Electronic Measurements – BEEL456B

ACADEMIC YEAR 2024-25

SEMESTER: IV

Prepared by:

Dr. Sathish K R and Mrs Kavyashree S

Verified by:

Approved by:

INSTITUTIONAL VISION AND MISSION

VISION:

Development of academically excellent, culturally vibrant, socially responsible and globally competent human resources.

MISSION:

- To keep pace with advancements in knowledge and make the students competitive and capable at the global level.
- To create an environment for the students to acquire the right physical, intellectual, emotional and moral foundations and shine as torchbearers of tomorrow's society.
- To strive to attain ever-higher benchmarks of educational excellence

DEPARTMENT VISION AND MISSION

VISION:

To create Electrical and Electronics Engineers who excel to be technically competent and fulfill the cultural and social aspirations of the society.

MISSION:

- To provide knowledge to students that builds a strong foundation in the basic principles of electrical engineering, problem solving abilities, analytical skills, soft skills and communication skills for their overall development.
- To offer outcome based technical education.
- To encourage faculty in training & development and to offer consultancy through research & industry interaction.

PROGRAMME OUTCOMES:

Engineering Graduates will be able to:

PO1. **Engineering knowledge**: Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.

PO2. **Problem analysis**: Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

PO3. **Design/development of solutions**: Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.

PO4. **Conduct investigations of complex problems**: Use research-based knowledge and research methods including design of EXPERIMENTs, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.

PO5. **Modern tool usage**: Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.

PO6. **The engineer and society**: Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.

PO7. **Environment and sustainability**: Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.

PO8. **Ethics**: Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.

PO9. **Individual and team work**: Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.

PO10. **Communication**: Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.

PO11. **Project management and finance**: Demonstrate knowledge and understanding of the engineering and management principles and apply these to one's own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.

PO12. Life-long learning: Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

Program Specific Outcomes (PSOs)

At the end of graduation, the student will be able,

PSO1: Apply the concepts of Electrical & Electronics Engineering to evaluate the performance of power systems and also to control Industrial drives using power electronics.

PSO2: Demonstrate the concepts of process control for Industrial Automation, design models for environmental and social concerns and also exhibit continuous self- learning.

Program Educational Objectives (PEOs)

PEO1: To produce competent and Ethical Electrical and Electronics Engineers who will exhibit the necessary technical and managerial skills to perform their duties in society.

PEO2: To make graduates continuously acquire and enhance their technical and socio-economic skills.

PEO3: To aspire graduates on R&D activities leading to offering solutions and excel in various career paths.

PEO4: To produce quality engineers who have the capability to work in teams and contribute to real time projects.

Scilab for Electrical and Electronic Measurements

BEEL456B

ATME COLLEGE OF ENGINEERING
13th Kilometer, Mysore-Kanakapura-Bangalore Road, Mysore – 570 028 P : 0821-2593335 F: 0821-2593328
Email: info@atme.in, Web : www.atme.in

Ability Enhancement Course
Scilab / MATLAB for Electrical and Electronic Measurements (BEEL456B)

Cycle of Experiments

Cycle-1

1. Design and Analysis of measurement of Resistance using Wheatstone and Kelvins double bridge.
2. Design and Analysis of measurement of Capacitance using Schering and De-Sauty's Bridges.
3. Design and Analysis of measurement of Inductance using Maxwells and Anderson Bridges.
4. Design and Analysis of measurement of Frequency using Wien's Bridge.
5. Design and Analysis of measurement of Real Power, Reactive and Power Factor in Three Phase Circuits.
6. Design and Analysis of measurement of Energy in Three Phase Circuits.

Cycle-2

1. Design and Analysis of measurement of Flux and Flux density.
2. Testing and Analysis of Current Transformer using Silsbees Deflection Method.
3. Testing and Analysis of Voltage Transformer using Silsbees Deflection Method.
4. Design and Analysis of True RMS Reading Volt Meters.
5. Design and Analysis of Integrating and Successive approximation type Digital Volt Meters.
6. Design and Analysis of Q Meter.

EXPERIMENT NO: 01 **DATE:**

Design and Analysis of Measurement of Resistanceusing Wheatstone and Kelvin`s Double Bridge

Aim: Design and analysis of measurement of resistance using Wheatstone and Kelvin`s Double Bridge

1(a) Wheatstone Bridge

Theory:

Wheatstone Bridge is an instrument designed to measure unknown resistance in electrical circuits. It calculates the unknown resistance by balancing the two legs of the bridge circuit where one leg contains both known resistors and the other leg contains one known (variable) and one unknown resistor. Since it estimates unknown resistance in an electric circuit, it is also known as a resistance bridge. Wheatstone bridge is a very reliable instrument as it measures the resistance very precisely.

Wheatstone Bridge works on the principle of null deflection i.e., there is no current flowing through the galvanometer, and its needle shows no deflection, hence the name null deflection. In the unbalanced state of the Wheatstone bridge i.e., when the potential across the galvanometer is different, the galvanometer shows the deflection, and as the bridge becomes balanced by changing the variable resistor, the potential difference across the galvanometer becomes zero i.e., the equilibrium state of Wheatstone bridge.

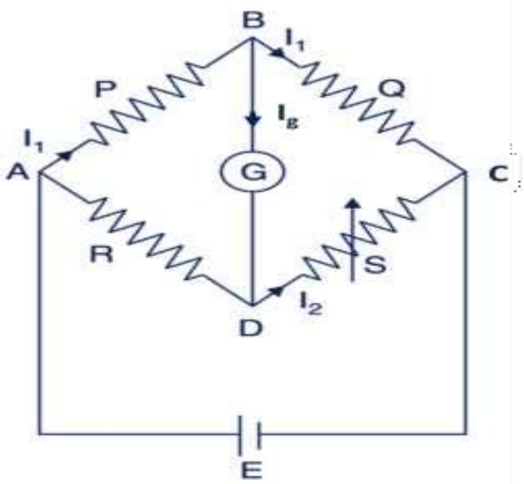


Fig. 1(a) Wheatstone Bridge

Formula used to calculate the unknown resistance "R":

$$R = \frac{P}{Q} S$$

Problem Statement:

A Wheatstone bridge having ratio arms of P=1000 Ω & Q=10 Ω is used for the measurement of high resistance. The adjustable arm has a maximum value of S=5 kΩ. Determine the maximum value of resistance that can be measured with the given arrangement.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Wheatstone Bridge
clc
clear
P=1000;//in ohm
Q=10;//in ohm
S=5000;//in ohm
R=(P*S)/Q;
mprintf("the unknown resistance value is %.2f ohm",R);
```

Result:

1(b) Kelvin`s Double Bridge

Theory:

A kelvin bridge or kelvin double bridge is a modified version of the Wheatstone bridge, which can measure resistance values in the range between 1 to 0.00001 ohms with high accuracy. It is named because it uses another set of ratio arms and a galvanometer to measure the unknown resistance value. The basic operation of the Kelvin double bridge can be understood from the basic construction and operation of the kelvin bridge.

A Wheatstone bridge is used to measure resistance equal to or greater than 1 – ohm, but if we want to measure the resistance below 1 – ohm, it becomes difficult because the leads which are connected to the galvanometer adds up the resistance of the device along with the resistance of leads leading to variation in the measurement of the actual value of resistance. Hence in order to overcome this problem, we can use a modified bridge called kelvin bridge.

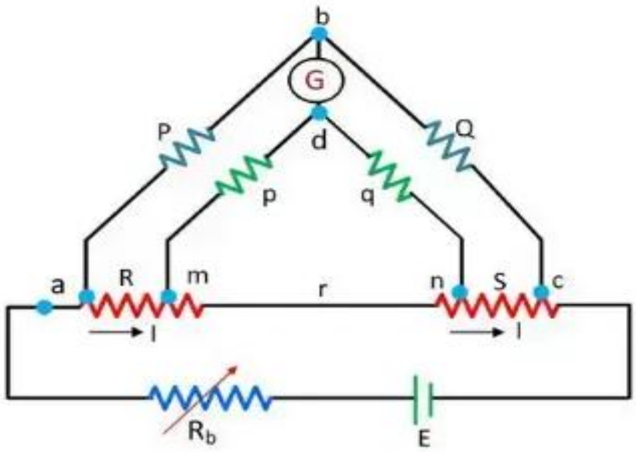


Fig. 1(b) Kelvin`s Double Bridge

Formula used to calculate the unknown resistance "X":

$$X = \frac{P}{Q} S + \frac{qr}{p+q} \left[\frac{P}{Q} - \frac{p}{q} \right]$$

Problem Statement:

A four terminal resistance of approximately 50 μΩ was measured by Kelvin's double bridge. The bridge has following component resistances.

Standard resistance S=100.03 μΩ

Inner ratio arms p=200.62 Ω & q=400 Ω

Outer ratio arms P=200.48 Ω & Q=400 Ω

The resistance of the link connecting the standard & unknown resistance r=700 μΩ. Calculate the unknown resistance.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Kelvin's Double Bridge
clc
clear
p=200.62;//in ohm
q=400;//in ohm
P=200.48;//in ohm
Q=400;//in ohm
S=100.03*10^(-6);//in ohm
r=700*10^(-6);//in ohm
X1=(P*S)/Q;
X2=(q*r)/(p+q);
X3=(P/Q)-(p/q);
```

```
X=X1+(X2*X3);
mprintf("The unknown resistance value is %f ohm",X);
```

Result:

EXPERIMENT NO: 02 **DATE:**

Design and Analysis of Measurement of Capacitance using Schering and De-Sauty's Bridges

Aim: Design and Analysis of measurement of Capacitance using Schering and De-Sauty's Bridges.

2(a) Schering Bridge

Theory:

A Schering Bridge is a bridge circuit used for measuring an unknown electrical capacitance and its dissipation factor. The dissipation factor of a capacitor is the ratio of its resistance to its capacitive reactance. The Schering Bridge is basically a four-arm alternating-current (AC) bridge circuit whose measurement depends on balancing the loads on its arms

In AC bridges, the power lines are used as a source of excitation at low frequencies, oscillators are used as a source at high-frequency measurements. The frequency range of an oscillator is 40 Hz to 125 Hz. The AC bridges not only measure the resistance, capacitance, and inductance but also measure the power factor.

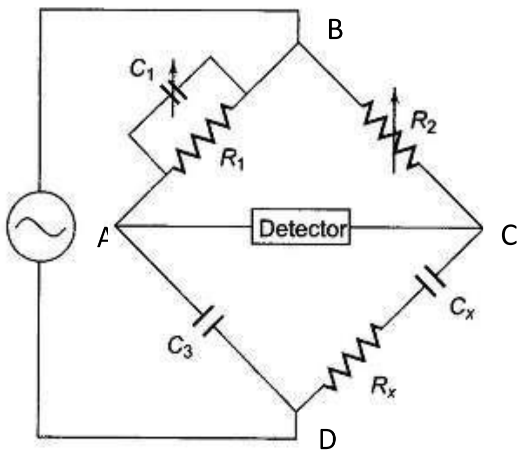


Fig. 2(a) Schering Bridge

Formula used to calculate the unknown values R_x & C_x:

$$R_x = R_2 \left[\frac{C_1}{C_3}\right]$$

$$C_x = C_3 \left[\frac{R_1}{R_2}\right]$$

Formula used to calculate the Dissipation Factor D:

$$D = \omega * C_x * R_x = 2\pi f * C_x * R_x$$

Problem Statement:

The Schering bridge has the following constants.

Arm AB: Capacitor of C1=0.4 µF in parallel with R1=1.5 kΩ resistance

Arm BC: Resistance of R2=3 kΩ

Arm CD: Unknown capacitance Cx & Rx in series

Arm DA: Capacitor of C3=0.4µF

Frequency=1 kHz

Determine i) Rx & Cx ii) Dissipation factor

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the “Scilab Console” window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Schering Bridge
clc
clear
f=1*10^3;
R1=1.5*10^3;
R2=3*10^3;
C1=0.4*10^(-6);
C3=0.4*10^(-6);
Cx=((R1/R2)*C3)*10^6;//Unknown Capacitance in micro farad
Rx=R2*(C1/C3);//Loss component of capacitor
Df=(2*%pi*f)*Cx*10^-6*Rx;//Dissipation factor of capacitor
mprintf("The unknown capacitance is %f micro farad\n",Cx);
```

```
mprintf("The loss component is %f ohm\n",Rx);
mprintf("The dissipation factor is %f",Df);
```

Result:

2(b) De Sauty's Bridge

Theory:

De Sauty Bridge is an instrument used to measure the capacitance value in a circuit. It is an Alternating Current bridge. It is based on the principle of null deflection. De Sauty Bridge is the simplest method for comparing two capacitances. It consists of two variable resistors, one standard capacitance, and one unknown capacitance. It finds the value of unknown capacitance in terms of standard capacitance. The balance point in the bridge can be obtained by varying the variable resistors. However, to measure capacitors with dielectric loss, a modified version of the De Sauty Bridge should be used.

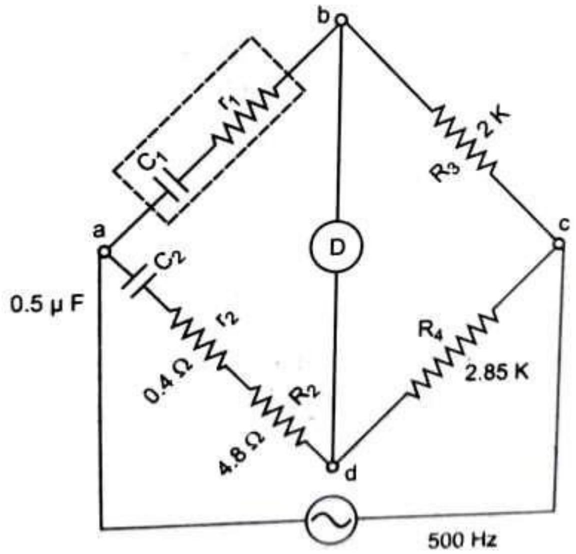


Fig. 2(b) De Sauty`s Bridge

Formula used to calculate the unknown values C1 & R1:

$$C_1 = C_2 * \left(\frac{R_4}{R_3}\right)$$

$$r_1 = \frac{R_3(R_2 + r_2)}{R_4}$$

Formula used to calculate the Dissipation Factor D:

$$D = \omega * C_1 * r_1 = 2\pi f * C_1 * r_1$$

Problem Statement:

The four arms of the bridge are,

Arm ab: An imperfect capacitor C1 with an equivalent series resistance r1

Arm bc: A non-inductive resistance R3= 2 kΩ

Arm cd: A non-inductive resistance R4= 2.85 kΩ

Arm da: An imperfect capacitor C2= 0.5 μF with an equivalent series resistance of r2= 0.4 Ω in series with resistance R2= 4.8 Ω

A supply of 500 Hz is given between the terminals a & c and the detector is connected between b & d. Calculate the value of C1, r1 & also dissipation factor of this capacitor.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the “Scilab Console” window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program Code:

```
//Measurement of Capacitance using De-Sauty's Bridge
clc
clear
f=500;
R2=4.8;
R3=2000;
R4=2850;
C2=0.5*10^(-6)
r2=0.4;
C1=(C2*(R4/R3))*(10^6);//Unknown Capacitance in micro farad
r1=((R3*(R2+r2)/R4));//Loss component of capacitor
```

```
Df=(2*%pi*f)*C1*10^-6*r1;//Dissipation factor of capacitor
mprintf("The unknown capacitance is %f micro farad \n",C1);
mprintf("The loss component is %f ohm\n",r1);
mprintf("The dissipation factor is %f",Df);
```

Result:

EXPTERIMENT NO: 03 **DATE:**

Design and Analysis of Measurement of Inductanceusing Maxwell and Anderson Bridges

Aim: Design and Analysis of measurement of Inductance using Maxwell and Anderson Bridges

3(a) Maxwell Bridge

Theory:

Maxwell's bridge is also known as Maxwell's Wein bridge or modified form of Wheatstone bridge or Maxwell's inductance capacitance bridge, consists of four arms used to measure unknown inductances in terms of calibrated capacitances and resistances. It can be used to measure unknown inductance value and compares it with the standard value. It works on the principle of comparison of known and unknown inductance values.

It uses the null deflection method to calculate inductance with a parallel calibrated resistor and capacitor. The Maxwell's bridge circuit is said to be in resonance if the positive phase angle of an inductive impedance is compensated with the negative phase angle of the capacitive impedance (connected in the opposite arm). Hence there will be no current flowing through the circuit and no potential across the null detector.

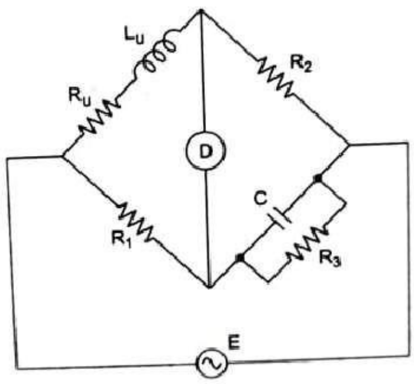


Fig. 3(a) Maxwell Bridge

Formula used to calculate the unknown values Ru & Lu:

$$R_u = \frac{R_1 R_2}{R_3}$$

$$L_u = C R_1 R_2$$

Formula used to calculate the Quality Factor Q:

$$Q = \omega C R_3 = 2\pi f * C R_3$$

Problem Statement:

In a Maxwell bridge, the fixed value bridge components have the values, $R_3 = 5\ \Omega$ &C = 1 mF. Calculate the value of unknown impedance (Lu & Ru) if $R_1 = 159\ \Omega$ & $R_2 = 10\ \Omega$ at balance. Also calculate the Q-factor of the unknown impedance at supply frequency of 50 Hz.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the “Scilab Console” window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program Code:

```
//Measurement of Inductance using Maxwell's Bridge
clc
clear
R3=5;
C=1*10^(-3);
R1=159;
R2=10;
R=1.36;
f=50;
Ru=(R1*R2)/R3;//Resistance of the coil
Lu=C*R1*R2;//Inductnce of the coil
Q=2*%pi*f*C*R3;
mprintf("The resistance of the coil is %f ohm\n",Ru);
mprintf("The inductance of the coil is %f henry\n",Lu);
mprintf("Q factor of the coil is %f",Q);
```

Result:

3(b) Anderson Bridge

Theory:

The Anderson's bridge gives the accurate measurement of self-inductance of the circuit. The bridge is the advanced form of Maxwell's inductance capacitance bridge. In Anderson bridge, the unknown inductance is compared with the standard fixed capacitance which is connected between the two arms of the bridge.

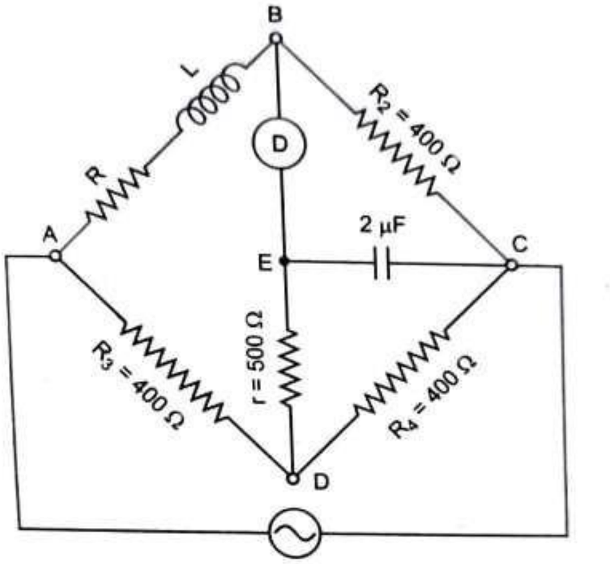


Fig. 3(b) Anderson Bridge

Formula used to calculate the unknown values R & L:

$$R = \frac{R_2\ R_3}{R_4}$$

$$L = \frac{CR_2}{R_4}[r(R_3 + R_4) + R_3 R_4]$$

Problem Statement:

In an anderson bridge, branch AB is an inductive resistor, branches BC & ED are variable resistors, branches CD & DA are non-reactive resistors of 400 Ω each & branch CE is a condenser of 2 μF capacitance. The supply is connected to A & C and the detector to B & E. Balance is obtained when the resistance of BC is 400 Ω & that of ED is 500 Ω. Determine the resistance & inductance of AB.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.

4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the “Scilab Console” window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Measurement of Inductance using Anderson's Bridge
clc
clear
R2=400;
R3=400;
R4=400;
r=500;
C=2*10^(-6);
R1=(R2*R3)/R4;//Resistance of the coil
L1=((C*R2)/R4)*(r*(R3+R4)+(R3+R4)); //Inductace of the coil
mprintf("The resistance of the coil is %f ohm\n",R1);
mprintf("The inductance of the coil is %f henry",L1);
```

Result:

EXPERIMENT NO: 04 **DATE:**

Design and Analysis of Measurement of Frequency using Wien's Bridge

Aim: Design and analysis of measurement of frequency using wien's bridge

Theory:

The Wien's bridge use in AC circuits for determining the value of unknown frequency. The bridge measures the frequencies from 100Hz to 100 kHz. The accuracy of the bridges lies between 0.1 to 0.5 percent. The bridge is used for various other applications like capacitance measurement, harmonic distortion analyzer and in the HF frequency oscillator. The bridge is used for measuring frequency in the audio range. Resistances R_1 and R_3 can be ganged together to have identical values. Capacitors C_1 and C_3 are normally of fixed values. The audio range is normally divided into 20 – 200 – 2 k – 20 kHz ranges. In this case, the resistances can be used for range changing and capacitors C_1 and C_3 for fine frequency control within the range. The Wien Bridge Circuit Diagram can also be used for measuring capacitances. In that case, the frequency of operation must be known. The Wien's bridge is frequency sensitive. Thereby, it is difficult to obtain the balance point in it. The input supply voltage is not purely sinusoidal, and they have some harmonics. The harmonics of the supply voltage disturbs the balance condition of the bridge. To overcome this problem the filter is used in the bridge. The filter connects in series with the null detector.

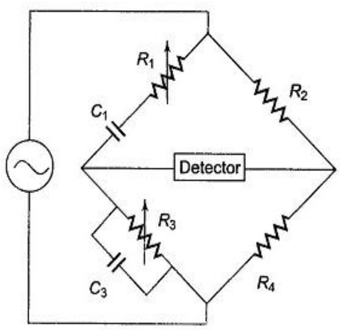


Fig. 4: Wien's Bridge

Formula used to calculate the unknown values:

$$f = \frac{1}{2\pi\sqrt{C_1 R_1 C_3 R_3}}$$

$$R_3 = \frac{R_4}{R_2}\left(R_1 + \frac{1}{\omega^2 R_1 C_1^2}\right)$$

$C_3 = \frac{R_2}{R_4}(\frac{C_1}{1+\omega^2 R_1^2 C_1^2})$ OR $C_3 = \frac{1}{\omega^2 C_1 R_1 R_3}$

Problem Statement:

Find the equivalent parallel resistance and capacitance that causes a wien bridge to null with the following component values.

$R_1 = 3.1k\Omega$

$C_1 = 5.2\mu F$

$R_2 = 25k\Omega$

f = 2.5kHz

$R_4 = 100k\Omega$

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
clc;
clear all;
// Given data
R1=3.1; // k Ohms
C1=5.2; // micro farads
R2=25; // k Ohms
f=2.5; // kHz
R4=100;// k Ohms
// Solution
w=2*%pi*f; // Angular frequency
```

```
R3=R4/R2 *(R1+1/(w^2*R1*C1^2));
C3=1/(w^2*C1*R1*R3);
printf(' The parallel resistance of %.1f K ohms and capacitance of %.1f pf \n     causes a Wien bridge to null with values of given component values. \n',R3,C3*10^6);
```

Result:

EXPTERIMENT NO: 05 **DATE:**

Design and Analysis of Measurement of Real, ReactivePower & Power Factor in Three Phase Circuits

Aim: Design and Analysis of measurement of Real, Reactive Power & Power Factor in Three Phase Circuits

Theory:

Real power (watts) is the energy consumed by the resistive part of an AC circuit. The true power of a circuit that contains inductive and/or capacitive reactance in addition to resistance is measured with a wattmeter

Reactive power (VARs) is power supplied to a reactive load. Almost all AC circuits include reactive power in the form of inductive reactance and/or capacitive reactance. Inductive reactance is by far the most common since all motors, transformers, solenoids, and coils have inductive reactance.

Power factor (PF) is the ratio of true power (watts) to apparent power (VA) and is expressed as a percentage that does not go above 100 %. Power factor measures how far the current and voltage are out of phase with respect to each other. Unity power factor (1 or 100%) can only occur if the AC circuit supplies resistive loads or when capacitive reactance (X_C) is equal to inductive reactance (X_L). When the power factor is less than 100 percent, the circuit is less efficient and has a higher operating cost because not all current performs useful work.

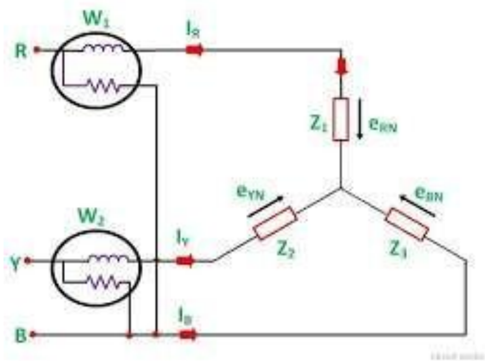


Fig. 5: Two wattmeter connected in a 3-phase circuit

Formula used to calculate the unknown values:

Real Power $P = W_1 + W_2$

Power Factor $PF = \cos\left[\tan^{-1}\sqrt{3}\ \frac{W_1 - W_2}{W_1 + W_2}\right]$

Reactive Power $= \sqrt{3} * (W_1 - W_2)$

Problem Statement:

Two watt-meters connected to measure the power in a 3-phase circuit read 5kW & 1kW, the latter being read after reversing the potential coil ends. Calculate the real power, power factor & reactive power of the circuit.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Measurement of power and power factor in three phase circuit
clc
clear
W1=5*10^3;//Wattmeter-1 reading
W2=-1*10^3;//Wattmeter-2 reading
P=(W1+W2)/10^3;//Input power to the 3phase circuit in kW
PFA=atan(((W1-W2)/(W1+W2))*sqrt(3));
PF=cos(PFA);
RP=(sqrt(3)*(W1-W2))/10^3;
mprintf("The input power is %4.2f kW \n",P);
mprintf("The reactive power is %4.2f kVAR \n",RP);
mprintf("The power factor is %1.2f (lagging)",PF);
```

Result:

EXPERIMENT NO 06 **DATE:**

Design and Analysis of Measurement of Energy inThree Phase Circuits

Aim: Design and Analysis of Measurement of Energy in Three Phase Circuits

Theory:

A three phase energy meter is a device used to measure the amount of electrical energy consumed in a three phase electrical system. It is an essential tool for monitoring and managing electricity usage in industrial and commercial settings, where three phase power is commonly used.

The three phase energy meter is designed to accurately measure the energy consumption of three phase loads, which are typically found in large-scale electrical systems. It is capable of measuring the total energy consumption as well as the energy consumption for each individual phase, providing a comprehensive overview of electrical usage.

The three phase energy meter operates by measuring the voltage and current in each phase of the electrical system and calculating the total energy consumption based on these measurements. It is equipped with sophisticated electronic components and sensors that enable it to accurately capture and record electrical data, ensuring precise and reliable measurements.

Formula used to calculate the unknown values:

Actual energy consumption during the test period

$$= \sqrt{3} \times \text{ratio of PT} \times \text{ratio of CT} \times V_s I_s \cos\Phi \times t \times 10^{-3} \ldots\ldots \text{kWh}$$

Energy recorded by meter during the test period $= \frac{\text{Number of revolutions made}}{\text{Meter constant in rev/kwh}}$ …………kWh

Percentage error $= \frac{\text{Energy Recorded} - \text{Actual Energy Consumption}}{\text{Actual Energy Consumption}} * 100$

Problem Statement:

A 3-phase, 2-element energy meter has a constant of 0.2 revolution of disc per kWh. The meter is being used with a PT of ratio 22kV/220 V and a CT of ratio 100/5 A. If the line voltage is 220 V, current is 10 A, time to complete 10 revolutions is 30 seconds on unity power factor; determine the error expressed as a percentage of the correct reading.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Energy measurement in 3 phase circuit
clc
clear
rPT=22000/220;
rCT=100/5;
Vs=220;
Is=10;
PF=1;
t=30/3600;//time in hours
N=10;//no. of revolutions
m=0.2;//meter constant
Actual=sqrt(3)*rPT*rCT*Vs*Is*PF*t*10^(-3);
Recorded=N/m;
%error=(Actual-Recorded)*100/Recorded;
mprintf("The actual energy consumption is %f kWh \n",Actual);
mprintf("The energy recorded by meter is %f kWh \n",Recorded);
mprintf("The percentage error value is -21.3 percent");
```

Result:

EXPERIMENT NO 07 **DATE:**

Design and Analysis of Measurement of Flux & Flux Density

Aim: Design and Analysis of measurement of Flux & Flux Density

Theory:

Magnetic flux is defined as the number of magnetic field lines passing through a given closed surface. It provides the measurement of the total magnetic field that passes through a given surface area. Here, the area under consideration can be of any size and under any orientation with respect to the direction of the magnetic field.

Magnetic flux density (B) is defined as the force acting per unit current per unit length on a wire placed at right angles to the magnetic field.

Formula used to calculate the unknown values:

Length of conductor $l = \pi d$

Reluctance $R = \frac{1}{\mu_o \mu_r a}$

μ_o = permeability of vaccum $= 4\pi * 10^{-7}$

μ_r = relative permeability

Flux $\Phi = \frac{NI}{R}$

Flux density $B = \frac{\Phi}{a}$

Problem Statement:

A coil of 1000 turns is wound on a silicon steel ring having relative permeability of 1200. The ring has a mean diameter of 10 cm & cross-sectional area of 12 sq. cm. When a current of 4 A flows through the coil, find the flux & flux density in the core.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the “Scilab Console” window. If there is any error in the program, the line where correction in the syntax will be displayed.

6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Measurement of Flux and Flux density

clc
clear
N=1000;
MUo=4*%pi*10^(-7);//Permeability of vaccum
MUr=1200;//Relative permeability
d=10*10^(-2);//diameter in meter
a=12*10^(-4);//area in sq. meter
I=4;//Current flowing through the coil
l=%pi*d;//length of the conductor
R=1/(MUo*MUr*a);//reluctance
F=(N*I)/R;//flux
FD=F/a;//Flux density
mprintf("The flux value is %f weber\n",F);
mprintf("The flux density value is %f tesla",FD);
```

Result:

EXPERIMENT NO 08 **DATE:**

Testing and Analysis of Current Transformer using Silsbee`s Deflection Method

Aim: Testing and Analysis of Current Transformer using Silsbee`s Deflection Method.

Theory:

Silsbee’s method is a comparison method used for Current Transformers testing. There are two types of Silsbee’s methods; deflectional and null. Only the deflectional method is described here. The arrangement for this method is shown schematically in below figure 8. Here the ratio and phase angle of the test transformer X are determined, in terms of that of a standard transformer S having the same nominal ratio.

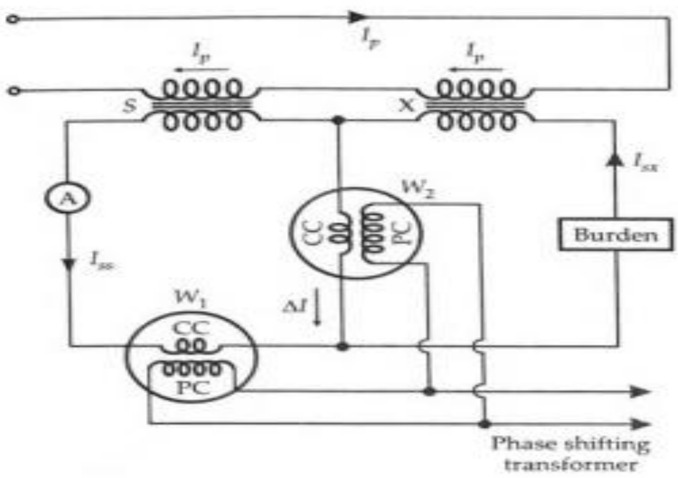


Fig. 8: Silsbee’s Deflection Method

Here the ratio and phase angle of the test transformer X are determined, in terms of that of a standard transformer S having the same nominal ratio. The two transformers are connected with their primaries in series. An adjustable burden is put in the secondary circuit of the transformer under test. An ammeter is included in the secondary circuit of the standard transformer so that the current may be set to the desired value. W2 is a wattmeter whose current coil is connected to carry the secondary current of the standard transformer. The current coil of wattmeter W2 carries a current ΔI which is the difference between the secondary currents of the standard and test transformers. The voltage circuits of the wattmeters (i.e., their pressure coils) are supplied in parallel from a phase shifting transformer at a constant voltage V.

Problem Statement:

Two current transformers of the same nominal ratio 500/5 A, are tested by Silsbee's method. With the current in the secondary of the transformer adjusted at its rated value, the content in the middle conductor I = $0.05e^{-j126.9^0}$A expressed with respect to current in the secondary of standard transformer as the reference. It is known that standard transformer has a ratio correction factor (RCF) of 1.0015. Find RCF of transformer under test.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Testing of CT using Silsbees Method
clc
clear
NR=500/5;//Nominal ratio
Iss=5;//Since Iss is the reference Iss=5+j0
Isx=5.03;//secondary current of test transformer
RCFs=1.0015;//Ratio correction factor of standard transformer
Rs=RCFs*NR;//Actual ratio of standard transformer
Ip=Rs*Iss;//primary current
Rx=Ip/Isx;//Actual ratio of transformer under test
RCFx=Rx/NR;//Ratio correction factor of test transformer
mprintf("Nominal ratio of test transformer is %f \n",NR);
mprintf("Actual ratio of test transformer is %f \n",Rx);
mprintf("Ratio correction factor of test transformer is %f",RCFx);
```

EXPERIMENT NO 09 **DATE:**

Testing and Analysis of Voltage Transformer using Silsbee`s Deflection Method

Aim: Testing and analysis of voltage transformer using Silsbee’s deflection method

Theory:

Comparison Method using wattmeter. The arrangement is shown in Fig. 9. The ratio and phase angle errors of a test transformer are determined in terms of those of a standard transformer S having the same nominal ratio.

The two transformers are connected with their primaries in parallel. A burden is put in the secondary circuit of test transformer. W1 is a wattmeter whose potential coil is connected across the secondary of standard transformer. The pressure coil of wattmeter W2 is so connected that a voltage ΔV which is the difference between secondary voltages of standard and test transformers, is impressed across it. The current coils of the two wattmeters are connected in series and are supplied from a phase shifting transformer. They carry a constant current I.

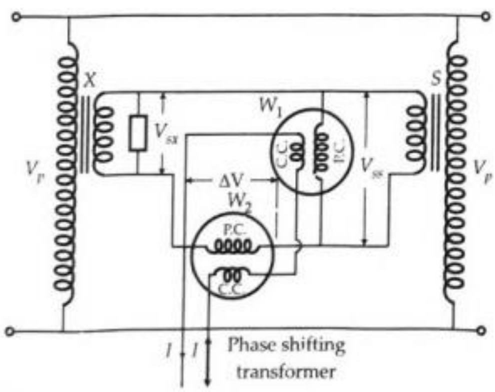


Fig. 9 Silsbee’s Method

Problem Statement:

An instrument potential transformer of nominal ratio 24000/120 V is tested by comparison with a calibrated transformer of the same nominal ratio. The standard transformer is known to have an RCF of 0.9985 and a phase angle error of -12'. The secondaries are connected in opposition and the resultant voltage ΔV (terminal voltage of test transformer - terminal voltage of standard transformer) is 0.5 ∟ 216.9° volt with respect to secondary voltage of standard transformer when the transformers are fed from rated voltage supply. Compute the RCF and phase angle of test transformer.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
clc
clear
NR=24000/120;
RCFs=0.9985;
P_Error_s=-12
Vp=24000;
dVr=cos(3.7856);
dVi=sin(3.7856);
e=complex(dVr,dVi);
dV=0.5*e;
disp(dV)
Actual_ratio_s=RCFs*NR;
disp(Actual_ratio_s)
Vss=Vp/Actual_ratio_s;
disp(Vss);
Vsx=dV+Vss;
disp(Vsx)
Vsx_mag=abs(Vsx);
disp(Vsx_mag)
Actual_ratio_x=Vp/Vsx_mag;
disp(Actual_ratio_x)
```

```
RCF_x=Actual_ratio_x/NR;
disp(RCF_x)
theta=-12+((imag(Vsx)/real(Vsx))*(60*180/%pi));
printf("RCF of test transformer=%f",RCF_x);
printf("\n phase angle of test transformer=%f",theta);
```

Result:

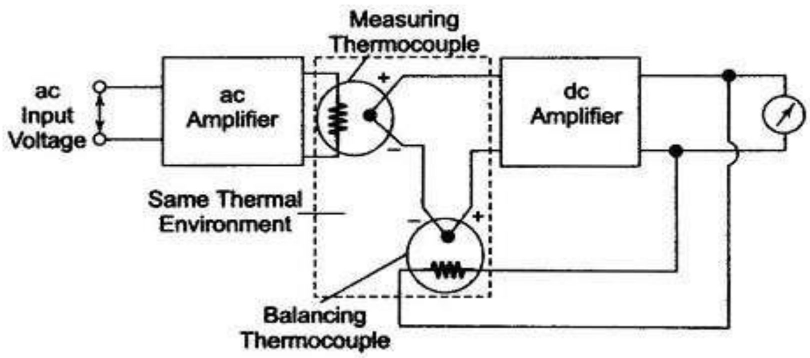
EXPERIMENT NO 10 **DATE:**

Design and Analysis of True RMS Reading Volt Meters

Aim: Design and Analysis of True RMS Reading Volt Meters.

Theory:

The average reading voltmeter can be converted into an RMS reading voltmeter by calibrating the meter scale in terms of the RMS value of a sine wave. Complex waveforms can be accurately measured by RMS reading voltmeter. This type of voltmeter provides an output by sensing the heating power of the waveform. Since the heating power of the waveform is proportional to the square of the RMS value of the voltage, the scale of the instrument can be calibrated to read the RMS directly.



True RMS Voltmeter (Block Diagram)

Formula used to calculate the unknown values:

RMS value of pulse waveform = $\sqrt{\frac{1}{T}\int_0^{dT} V^2 dt}$ …….V

Problem Statement:

Calculate the error when a symmetrical square wave voltage is applied to an average reading AC voltmeter with scale calibrated in terms of rms value of a sinusoidal wave.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be

displayed.

6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//True RMS Reading Voltmeter
clc;
clear;
close;
//given data :
Kf_sin=1.11;//Form factor of sine wave
kf=1; // from interation Erms=Eav
R=Kf_sin/kf; // ratio of the two form factors
Pe=(R-1/1)*100;
disp(Pe,"the percentage error,Pe(%) = ")
```

Result:

EXPERIMENT NO 11 **DATE:**

Design and Analysis of Integrating and Successive Approximation Type Digital Volt Meters

Aim: To design and analyze Integrating and Successive approximation type Digital Volt meters

11(a) Integrating Type Digital Voltmeter

Theory:

: A digital voltmeter (DVM) displays the value of a.c. or d.c. voltage being measured directly as discrete numerals in the decimal number system. The use of digital voltmeters increases the speed with which readings can be taken. The increasing popularity of DVMs has brought forth a wide number of types employing different circuits. The various types of DVMs in general use are:

i. Ramp type DVM

ii. Integrating type DVM

iii. Potentiometric type DVM

iv. Successive approximation type DVM

v. Continuous balance type DVM

Integrating Type Digital Voltmeter:

This voltmeter measures the true average value of the input voltage over a fixed measuring period. This voltmeter employs an integration technique which uses a voltage to frequency conversion. The voltage to frequency (V/F) converter functions as a feedback control system which governs the rate of pulse generation in proportion to the magnitude of input voltage. While using the voltage to frequency conversion techniques, a train of pulses, whose frequency depends upon the voltage being measured, is generated. Then the number of pulses appearing in a definite interval of time is counted. Since the frequency of these pulses is a function of unknown voltage, the number of pulses counted in that period of time is an indication of the input (unknown) voltage. The heart of this technique is the operational amplifier acting as an Integrator.

In V/F conversion technique, a brain of pulses whose frequency depends upon the voltage being measured is generated. Then the number of pulses appearing in a definite interval of time is measured. The operational integrator consists of an amplifier with a capacitor C connected between output and input. If V_i is the input voltage, then V_o, of the operational integration is given by:

$V_O = \frac{1}{RC}\int V_i \, dt$

If the input voltage is constant, we have :

$V_o = \frac{V_i}{RC}.t$

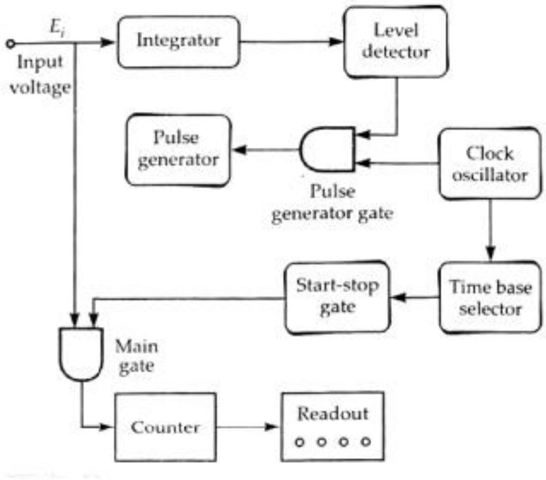


Fig. 12(a) Block diagram of integrating type DVM

Problem statement:

1) An integrator contains a 100kΩ and 1μF capacitor. If the voltage applied to the integrator input is 1V, what voltage will be present at the output of the integrator after 1second.
2) Now if a reference voltage is applied to the integrator of the above example at time t_1 is 5V in amplitude, what is the time interval of t_2?

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).

8. Change the values of known parameters and analyze the bridge.

Program code:

1)

```
clc;
clear all;
// Given data
R=100; //  in k Ohms
C=1;  //in micro farads
ei=1;// Applied voltage to integrator(V)
t1=1; // time in Sec
eo=ei*t1/(R*10^3*C*10^-6); // Output voltage from integrator
printf(' The voltage at output of integrator after 1 sec is = %d Volts \n',eo);
```

2)

```
clc;
clear all;
// Given data
ei=1; // Applied input voltage to integrator(V)
t1=1; // sec
// Given data
er=5;// Reference voltage applied at time t1 to integrator(V)
//  Solution
t2=ei/er * t1;// Time interval t2(sec)
printf(' The time interval of t2 is = %.1f sec \n',t2);
```

Results:

11(b) Successive Approximation Type Digital Voltmeter

Theory:

A digital to analog converter is used to provide the estimates. The "equal to or greater than" or "less than" decision is made by the comparator. The D/A converter provides the estimate and is compared to the input signal. A special shift register called a "successive approximation register (SAR)" is used to control the D/A converter and consequently the estimates. At the beginning of the conversion all the outputs from the SAR are at logic zero. If the estimate is greater than the input, the comparator output is high and the first SAR output reverses state and the second output changes to logic "one". If the comparator output is low, indicating that the estimate is lower than the input signal the first output remains in the logic one state and the second output assumes the logic one state. This continues to all the states until the conversion is complete. The above sequence of events are performed electronically. For an N-bit conversion after N clocks, the actual value of the input is known. The least significant bit is the state of the comparator. In some systems an additional clock is employed to store the last bit in the SAR and thus N+1 clocks are required for the conversion.

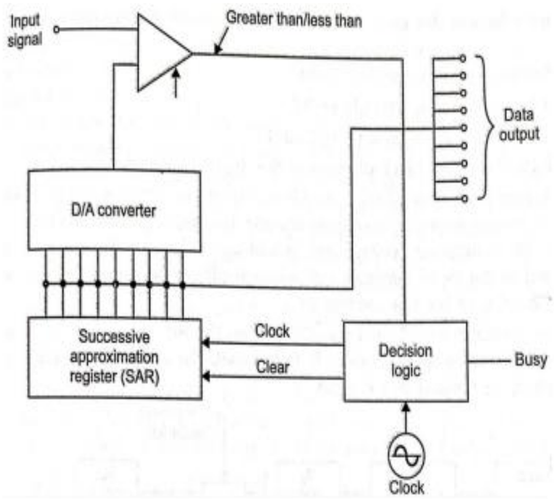


Fig.11(b) Successive Approximation type Digital Voltmeter

Problem Statement:

The successive approximation type DVM has the maximum range of 255 volts, determine the time it will take to read the unknown voltage V_x = 180V, using a clock oscillator of 10 kHz.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.

4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
clc;
clear;
close;
// Given data
Vmax= 255;// in volt
Vx= 180;// in volt
f=10;// in kHz
f=f*10^3;// in Hz
t= (Vmax-Vx)/(2*%pi*f*Vmax);// time taken to read the unknown voltage in second
t=t*10^6;// in micro second
disp(t,"Time taken to read the unknown voltage in micro second");
```

Result:

EXPERIMENT NO 12 **DATE:**

Design and Analysis of Q Meter

Aim: Design and Analysis of Q meter

Theory:

Q meter is an instrument designed to measure some electrical properties of coils and capacitors. The principle of the Q meter is based on series resonance the voltage drop across the coil or capacitor is Q times the applied voltage. If a fixed voltage is applied to the circuit, a voltmeter across the capacitor can be calibrated to read Q directly. The overall efficiency of the coils and capacitors intended for radio frequency applications is best evaluated using the Q value.

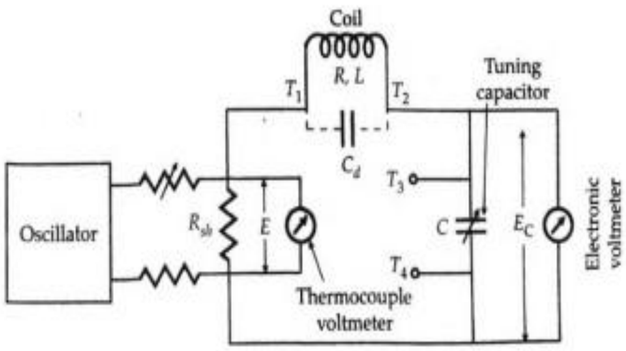


Fig. 12 Circuit of a Q Meter

Formula used to calculate the unknown values:

$$Q = \frac{X_C}{R} = \frac{1}{\omega CR}$$

$$Q_{actual} = \frac{1}{\omega CR}$$

$$Q_{measured} = \frac{1}{\omega C(R + R_{sh})}$$

$$\% \text{ error} = \frac{Q_{actual} - Q_{measured}}{Q_{actual}} * 100$$

Problem Statement:

A coil with a resistance of 10 Ω is connected in the direct connection in a Q meter.Resonance occurs when the oscillator frequency is 1MHz & the resonating capacitor is set to65 pF. Calculate Q value & % error introduced in the calculated value of Q by 0.02 Ω insertion resistance.

Procedure for writing program in Scilab:

1. Open the Scilab software and click on Launch Scinotes. A new scinotes file will be opened.
2. Write a suitable program to solve the given problem statement.
3. Save the program by giving a proper file name.
4. Click on Execute to run the program and check the output.
5. If the program is correct, the result will be displayed in the "Scilab Console" window. If there is any error in the program, the line where correction in the syntax will be displayed.
6. Referring to hint given, the correction need to be done in the program, save it again and click on Execute. The result will be displayed in Scilab Console.
7. If still the error persists, repeat the procedure (5) & (6).
8. Change the values of known parameters and analyze the bridge.

Program code:

```
//Q Meter
clc
clear
f=1*10^6;//frequency
C=65*10^(-12);//capacitance
R=10;//Resistance
Rsh=0.02;
w=2*%pi*f;
Qa=1/(w*C*R);//Actual value
Qm=1/(w*C*(R+Rsh));//Measured value
%error=(Qa-Qm)*100/Qa;//%Error calculation
mprintf("The Q value calculated is %f  \n",Qa);
mprintf("The Q value measured is %f  \n",Qm);
mprintf("The percentage error value is %3.2f",%error);
```

Result: