Electrical Circuit Computation Laboratory Manual

3rd Sem, B.Tech. (Electrical Engineering)

Department of Electrical Engineering

Veer Surendra Sai University of Technology Burla

VISION

To be recognized as a centre of excellence in education and research in the field of Electrical Engineering by producing innovative, creative and ethical Electrical Engineering professionals for socio-economic development of society in order to meet the global challenges.

MISSION

Electrical Engineering Department of VSSUT Burla strives to impart quality education to the students with enhancement of their skills to make them globally competitive through:

M1. Maintaining state of the art research facilities to provide enabling environment to create, analyze, apply and disseminate knowledge.

M2. Fortifying collaboration with world class R&D organizations, educational institutions, industry and alumni for excellence in teaching, research and consultancy practices to fulfil 'Make in India' policy of the Government.

M3. Providing the students with academic environment of excellence, leadership, ethical guidelines and lifelong learning needed for a long productive career.

PROGRAM EDUCATIONAL OBJECTIVES of B.Tech. (EE)

The program educational objectives of B.Tech. in Electrical Engineering program of VSSUT Burla are to prepare its graduates:

1. To have basic and advanced knowledge in Electrical Engineering with specialized knowledge in design and commissioning of electrical systems/renewable energy systems comprising of generation, transmission and distribution to become eminent, excellent and skillful engineers.
2. To succeed in getting engineering position with electrical design, manufacturing industries or in software and hardware industries, in private or government sectors, at Indian and in Multinational organizations.
3. To have a well-rounded education that includes excellent communication skills, working effectively on team-based projects, ethical and social responsibility.
4. To have the ability to pursue study in specific area of interest and be able to become successful entrepreneur.
5. To have broad knowledge serving as foundation for lifelong learning in multidisciplinary areas to enable career and professional growth in top academic, industrial and government/corporate organizations.

List of experiments electrical circuit computation lab

1. Power measurement of AC system using MATLAB.
2. Time response of a first order system using Laplace Transform.
3. Time response of a second order system using Laplace Transform.
4. DC analysis for R-L and R-C circuits using MATLAB.
5. DC analysis for R-L-C circuits using MATLAB.
6. AC analysis for R-L and R-C circuits using MATLAB.
7. AC analysis for R-L-C circuits using MATLAB.
8. Simulation of series resonance circuit.
9. Simulation of parallel resonance circuit.
10. Design Simulink model of half wave (HW) diode rectifier.
11. Design Simulink model of full wave (FW) diode bridge rectifier.
12. Numerical analysis: Non-linear equations and optimization, Differential equations.

Course Outcomes:

After completion of this laboratory course the students will be able to

CO1	Know about the MATLAB software and its application in DC, single phase and three phase electric circuit to analyze.
CO2	Extend understanding for solving other electrical problems using the software.
CO3	The students can interpret and summarize from the response the type of the system.
CO4	Discover how to apply the different numerical techniques for analysis of electrical systems and its implementation with MATLAB.
CO5	Modify circuit simulation in different ways by both programming and Simulink blocks in MATLAB.

Experiment no.: 01

1.1 Aim of the Experiment:

Power measurement of AC system using MATLAB

a) Measure active power
b) Measure reactive power
c) Measure apparent power
d) Show variation of instantaneous power with time
e)

1.2 Software/tools required:

MATLAB/Simulink

1.3 Theory:

(1) Real Power: (P)

Alternative words used for Real Power (Actual Power, True Power, Watt-full Power, Useful Power, Real Power, and Active Power). In a DC Circuit, power supply to the DC load is simply the product of Voltage across the load V and Current I flowing through it i.e., P = V I. because in DC Circuits, there is no concept of phase angle between current and voltage. In other words, there is no Power factor in DC Circuits. But the situation in Sinusoidal or AC Circuits is more complex because of phase difference between Current and Voltage. Therefore average value of power (Real Power) is P = VI Cosθ is in fact supplied to the load. In AC circuits, When circuit is pure resistive, then the same formula used for power as used in DC as P = V I.

Real Power formulas:

P = V I (In DC circuits)

P = VI Cosθ (in Single phase AC Circuits)

Where, Cosθ=Power factor

$P = \sqrt{3}\ V_L\ I_L\ \cos\theta$ or (in Three Phase AC Circuits)

$P = 3\ V_{Ph}\ I_{Ph}\ \cos\theta$

$P = \sqrt{(S^2 - Q^2)}$ or

$P = \sqrt{(VA^2 - VAR^2)}$ or

Real or True power = $\sqrt{(\text{Apparent Power}^2 - \text{Reactive Power}^2)}$ or

$kW = \sqrt{(kVA^2 - kVAR^2)}$

(2) Reactive Power: (Q)

Also known as (Use-less Power, Watt less Power). The powers that continuously bounce back and forth between source and load is known as reactive Power (Q). Power merely absorbed and returned by load due to its reactive properties is referred to as reactive power.

Reactive power represent that the energy is first stored and then released in the form of magnetic field or electrostatic field in case of inductor and capacitor respectively. Reactive power is given by Q = V I Sinθ which can be positive (+ve) for inductive, negative (-ve) for capacitive load. The unit of reactive power is Volt-Ampere reactive (VAR).

In more simple words, in Inductor or Capacitor, how much magnetic or electric field made by 1A x 1V is called the unit of reactive power.

Reactive power formulas:

$Q = V I \sin\theta$

Reactive Power=$\sqrt{}$ (Apparent Power^2– True power^2)

VAR =$\sqrt{}$ ($VA^2 - P^2$)

kVAR = $\sqrt{}$ ($kVA^2 - kW^2$)

(3) Apparent Power: (S)

The product of voltage and current if and only if the phase angle differences between current and voltage are ignored. Total power in an AC circuit, both dissipated and absorbed/returned is referred to as apparent power. The combination of reactive power and true power is called apparent power. In an AC circuit, the product of the r.m.s voltage and the r.m.s current is called apparent power. It is the product of Voltage and Current without phase angle

The unit of Apparent power (S) is VA.

When the circuit is pure resistive, then apparent power is equal to real or true power, but in inductive or capacitive circuit, (when Reactances exist) then apparent power is greater than real or true power.

Apparent power formulas:

$S = V I$

Apparent Power = $\sqrt{}$ (True power^2 + Reactive Power^2)

kVA = $\sqrt{(kW^2 + kVAR^2)}$

1.4 Procedure:

1. Assume any value of voltage, current and phase angle (θ) for single phase 50 Hz system.
2. Write down MATLAB code for power measurement.
3. Plot wave form of instantaneous power, voltage and current with proper legend and axis label.
4. Plot active, reactive and apparent power.

1.5 Simulation results/Conclusion:

Write conclusion and attach program and simulation results.

1.6 References:

1. A. Chakrabarti, “Circuit Theory: Analysis and Synthesis”, Dhanpat Rai Publications.
2. B. L. Theraja and A. K. Theraja, “A Text Book of Electrical Technology”, Vol. I, S. Chand publication.
3. M. L. Soni, J. C. Gupta, “A Course in Electrical Circuits and Analysis”, Dhanpat Rai Publications.

Experiment no.: 02

2.1 Aim of the Experiment:

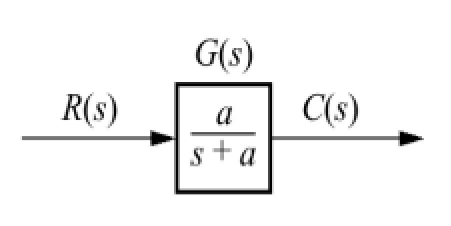
Time response of a first/ second order system using Laplace Transform.

2.2 Software/tools required:

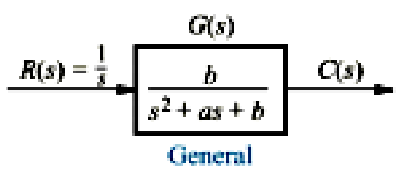
MATLAB/Simulink

2.3 Theory:

The **time response** represents how the state of a dynamic system changes in time when subjected to a particular input. A first-order without zeros can be described by the transfer function given below.



The order of a control system is determined by the power of s in the denominator of its transfer function. If the power of s in the denominator of transfer function of a control system is 2, then the system is said to be second-order control system. The general expression of transfer function of a second order control system is given as,



The standard input test signals are :

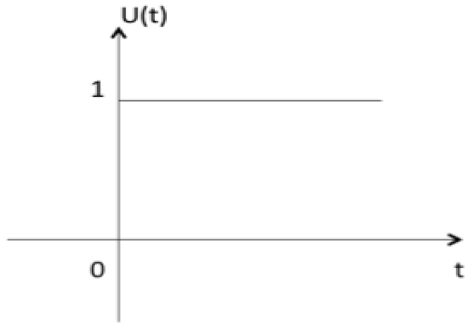
1. Unit Step Signal
2. Ramp Signal

2.4 Procedure:

1. Develop Laplace transform of first and second order system model using MATLAB code/ Simulink.
2. Study the response for the given test signals.

Unit Step Function

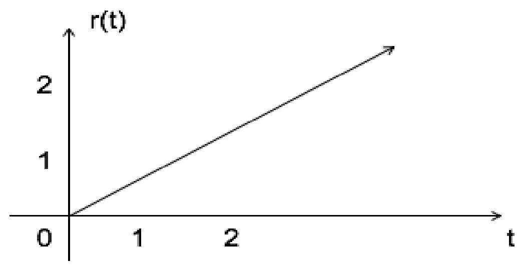
Unit step function is denoted by u(t). It is defined as u(t) = $\begin{cases} 1 & t \geqslant 0 \\ 0 & t < 0 \end{cases}$



- It is used as best test signal.
- Area under unit step function is unity.

Ramp Signal

Ramp signal is denoted by r(t), and it is defined as r(t) = $\begin{cases} t & t \geqslant 0 \\ 0 & t < 0 \end{cases}$



$$\int u(t) = \int 1 = t = r(t)$$

$$u(t) = \frac{dr(t)}{dt}$$

Area under unit ramp is unity.

2.5 Simulation results/Conclusion:

Write conclusion and attach program and simulation results.

2.6 References:

1. A. Chakrabarti, "Circuit Theory: Analysis and Synthesis", Dhanpat Rai Publications.
2. B. L. Theraja and A. K. Theraja, "A Text Book of Electrical Technology", Vol. I, S. Chand publication.
3. M. L. Soni, J. C. Gupta, "A Course in Electrical Circuits and Analysis", Dhanpat Rai Publications.

Experiment no.: 03

3.1 Aim of the Experiment:

DC analysis for R-L, R-C and R-L-C circuits using MATLAB

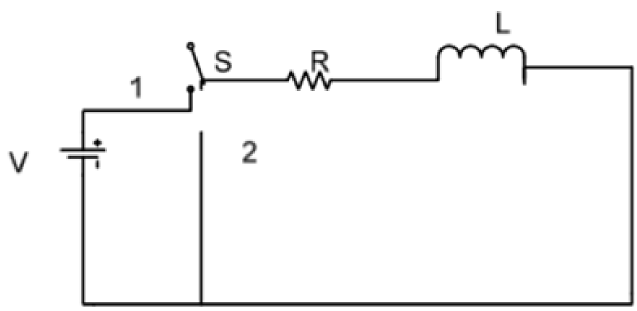
3.2 Software/tools required:

MATLAB/Simulink

3.3 Theory:

In general transient disturbances are produced when,

- A circuit is suddenly connected to or disconnected from the supply.
- Circuit is shorted.
- There is a sudden change in applied voltage from one finite value to another.



Series R-L Circuit:

Charging/ Rise case current (switch at 1):

$$i = \frac{V}{R}\left(1 - e^{\frac{-R}{L}t}\right)$$

Discharging/ Decay case current (switch at 2):

$$i = \frac{V}{R}\, e^{\frac{-R}{L}t}$$

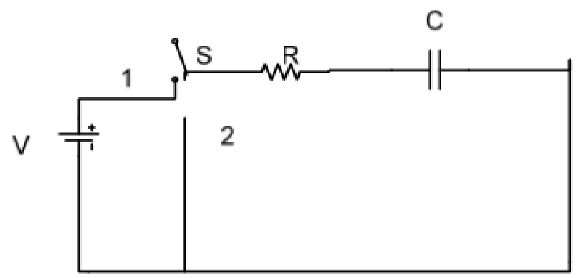
Series R-C Circuit:

Charging case current (switch at 1):

$$i = I_0 e^{-t/RC}$$

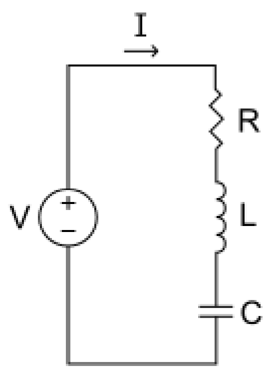
Discharging case current (switch at 2):

$$i = -I_0 e^{-t/RC}$$



Series R-L-C circuit

A Series R-L-C circuit is shown in figure with a DC voltage source V, resistance R, inductance L and capacitance C.



From KVL in general for series circuit,

$$v_R + v_L + v_C = v(t)$$

Where v_R , v_L *and* v_C are the voltages across R, L and C respectively and *v(t)* is the time varying voltage from the source that is constant DC here. But, this produces transient in current *i(t)*. Putting the values and differentiating leads to the second order differential equation:

$$\frac{d^2 i(t)}{dt^2} + \frac{R}{L}\frac{di(t)}{dt} + \frac{1}{LC} i(t) = 0$$

The general solution of the differential equation is an exponential in either root or a linear superposition of both,

$$i(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}$$

The coefficients A_1 and A_2 are determined by the boundary conditions of the specific problem being analysed.

In the case of the series RLC circuit, the damping factor is given by,

$$\zeta = \frac{R}{2}\sqrt{\frac{C}{L}}$$

The value of the damping factor determines the type of transient that the circuit will exhibit.

Overdamped response

$$i(t) = A_1 e^{-\omega_0\left(\zeta + \sqrt{\zeta^2 - 1}\right)t} + A_2 e^{-\omega_0\left(\zeta - \sqrt{\zeta^2 - 1}\right)t}$$

The overdamped response is a decay of the transient current without oscillation.

Underdamped response

$$i(t) = B_1 e^{-\alpha t}\cos(\omega_d t) + B_2 e^{-\alpha t}\sin(\omega_d t)$$

By applying trigonometric identities the two trigonometric functions may be expressed as a single sinusoid with phase shift.

$$i(t) = B_3 e^{-\alpha t}\sin(\omega_d t + \varphi)$$

The underdamped response is a decaying oscillation at frequency ω_d. The oscillation decays at a rate determined by the attenuation α. The exponential term describes the envelope of the oscillation. B_1 and B_2 (or B_3 and the phase shift φ in the second form) are arbitrary constants determined by boundary conditions.

This ω_d is called the damped resonance frequency or the damped natural frequency given by:

$\omega_d = \omega_0\sqrt{(1 - \zeta^2)}$

Critically damped response

The critically damped response is,

$$i(t) = D_1 t e^{-\alpha t} + D_2 e^{-\alpha t}$$

The critically damped response represents the circuit response that decays in the fastest possible time without oscillation. This is important in control systems where it is required to reach the desired state as quickly as possible without overshooting. D_1 and D_2 are constants determined by boundary conditions.

3.4 Procedure:

1. Assume any value of dc source, R, L and C.
2. Write down MATLAB code for transient study.
3. Plot variation of voltage and current across L/C during charging/discharging case.

Procedure (Series RLC Circuit):

1. Assume any value of AC source, choose R, L and C such that for the series circuit you generate overdamped, underdamped and critically damped circuit.
2. Write down MATLAB code for v and i.
3. Plot variation of current for the three circuits with different ζ.

3.5 Simulation results/Conclusion:

Write comparative conclusion and attach simulation results.

3.6 References:

4. A. Chakrabarti, “Circuit Theory: Analysis and Synthesis”, Dhanpat Rai Publications.
5. B. L. Theraja and A. K. Theraja, “A Text Book of Electrical Technology”, Vol. I, S. Chand publication.
6. M. L. Soni, J. C. Gupta, “A Course in Electrical Circuits and Analysis”, Dhanpat Rai Publications.

Experiment no.: 04

4.1 Aim of the Experiment:

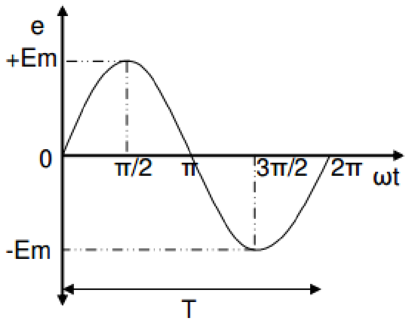
AC analysis for R-L, R-C and R-L-C circuits using MATLAB

4.2 Software/tools required:

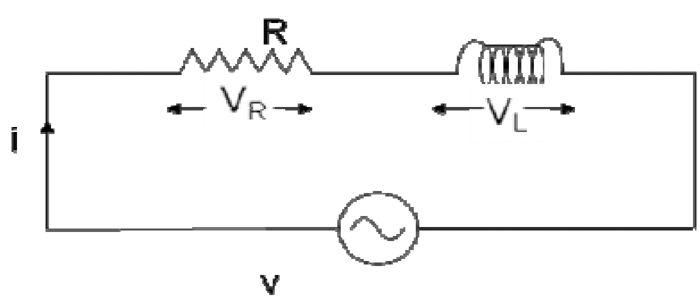
MATLAB/Simulink

4.3 Theory:

An alternating quantity changes continuously in magnitude and alternates in direction at regular intervals of time. Important terms associated with an alternating quantity are defined below.



R-L Series circuit



Consider an AC circuit with a resistance R and an inductance L connected in series as shown in the figure. The alternating voltage v is given by

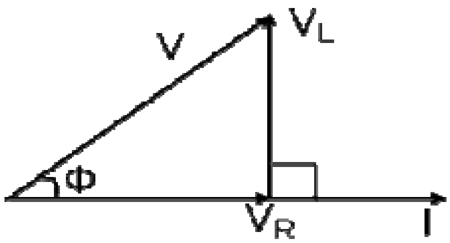
$$v = V_m \sin \omega t$$

The current flowing in the circuit is i. The voltage across the resistor is V_R and that across the inductor is V_L.

V_R=IR is in phase with I

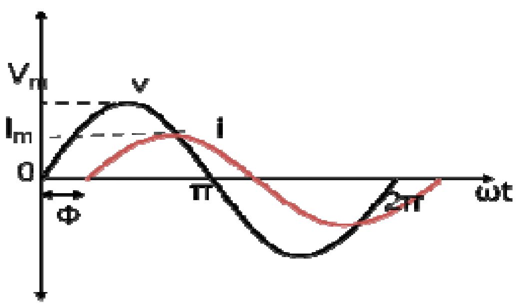
V_L=IX_L leads current by 90 degrees

With the above information, the phasor diagram can be drawn as shown.



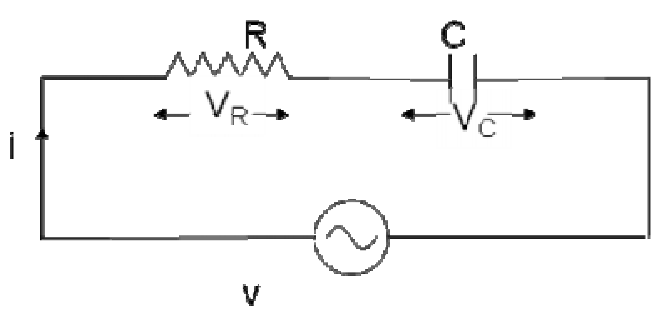
The current I is taken as the reference phasor. The voltage V_R is in phase with I and the voltage V_L leads the current by 90°. The resultant voltage V can be drawn as shown in the figure. From the phasor diagram we observe that the voltage leads the current by an angle Φ or in other words the current lags behind the voltage by an angle Φ.

The waveform and equations for an RL series circuit can be drawn as below.



$$V = V_m \sin \omega t$$
$$I = I_m \sin(\omega t - \Phi)$$

R-C Series circuit



Consider an AC circuit with a resistance R and a capacitance C connected in series as shown in the figure. The alternating voltage v is given by

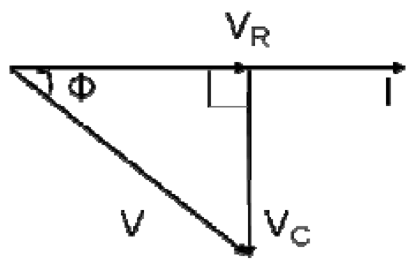
$$v = V_m \sin \omega t$$

The current flowing in the circuit is i. The voltage across the resistor is V_R and that across the capacitor is V_C.

V_R=IR is in phase with I

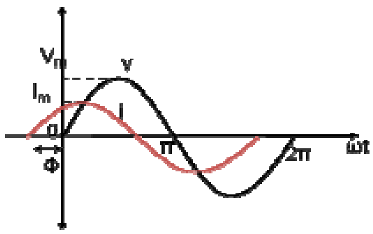
V_C=IX_C lags behind the current by 90 degrees

With the above information, the phasor diagram can be drawn as shown.



The current I is taken as the reference phasor. The voltage V_R is in phase with I and the voltage V_C lags behind the current by 90^0. The resultant voltage V can be drawn as shown in the figure. From the phasor diagram we observe that the voltage lags behind the current by an angle Φ or in other words the current leads the voltage by an angle Φ.

The waveform and equations for an RC series circuit can be drawn as below.



$$V = V_m \sin \omega t$$

$$I = I_m \sin(\omega t + \Phi)$$

4.4 Procedure:

1. Assume any value of AC source, R, L and C.
2. Write down MATLAB code for v and i.
3. Plot variation of voltage and current.

4.5 Simulation results/Conclusion:

Write conclusion and attach simulation results.

4.6 References:

1. A. Chakrabarti, "Circuit Theory: Analysis and Synthesis", Dhanpat Rai Publications.
2. B. L. Theraja and A. K. Theraja, "A Text Book of Electrical Technology", Vol. I, S. Chand publication.
3. M. L. Soni, J. C. Gupta, "A Course in Electrical Circuits and Analysis", Dhanpat Rai Publications.

Experiment no.: 05

5.1 Aim of the Experiment:

Series & parallel resonance circuit simulation

5.2 Software/tools required:

MATLAB/Simulink

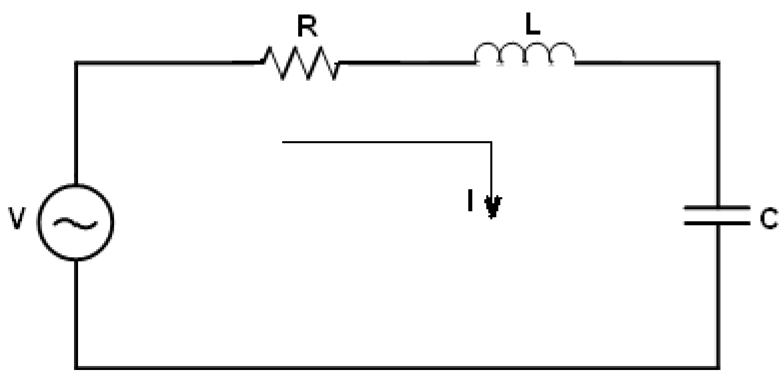
5.3 Theory:

A **RLC circuit** is an electrical circuit consisting of a resistor (R), an inductor (L), and a capacitor (C), connected in series or in parallel. Resonance is characterized by the input voltage and current being in phase. The impedance (or admittance) is completely real when this condition exists. Basically ,there are two types of resonant circuits :

(a) Series resonance, and

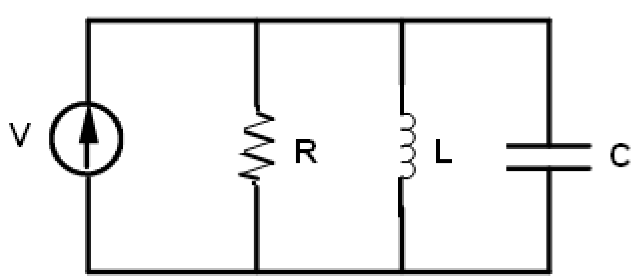
(b) Parallel resonance.

***Series Resonance*:**



As f increases, the XL increases and the XC decreases till at a frequency fr where the two reactances become equal. With further increase in f, → XL > XC. At fr the net reactance of the circuit = 0. The impedance of the circuit z= R and the current in the circuit = V/R. fr is known as resonance frequency and the circuit , is said to be in resonance.

Parallel Resonance:



Variation of inductive and capacitive reactance as the frequency f of the source is varied: Therefore resonance occurs when,

$$wL = \frac{1}{wC}$$

$$f_r = \frac{\omega_r}{2\pi} = \frac{1}{2\pi\sqrt{LC}}$$

This is an important equation to remember. It applies to both series and parallel resonant circuit.

5.4 Procedure:

1. Write down MATLAB code.
2. Plot Z, XL, XC, power factor, voltage and current varying f.

5.5 Simulation results/Conclusion:

Write conclusion and attach simulation results. Mention:
Why series RLC AC resonance circuit is called an acceptor circuit?
Why parallel RLC AC resonance circuit is called an acceptor circuit?

5.6 References:

1. A. Chakrabarti, “Circuit Theory: Analysis and Synthesis”, Dhanpat Rai Publications.
2. B. L. Theraja and A. K. Theraja, “A Text Book of Electrical Technology”, Vol. I, S. Chand publication.
3. M. L. Soni, J. C. Gupta, “A Course in Electrical Circuits and Analysis”, Dhanpat Rai Publications.

Experiment no.: 06

6.1 Aim of the Experiment:

Design Simulink model of half wave (HW) diode rectifier

6.2 Software/tools required:

MATLAB Simulink

6.3 Theory:

A rectifier is a circuit which converts the *Alternating Current* (AC) input power into a *Direct Current* (DC) output power. The input power supply may be either a single-phase or a multi-phase supply with the simplest of all the rectifier circuits being that of the Half Wave Rectifier.

The power diode in a half wave rectifier circuit passes just one half of each complete sine wave of the AC supply in order to convert it into a DC supply. Then this type of circuit is called a "half-wave" rectifier because it passes only half of the incoming AC power supply as shown below.

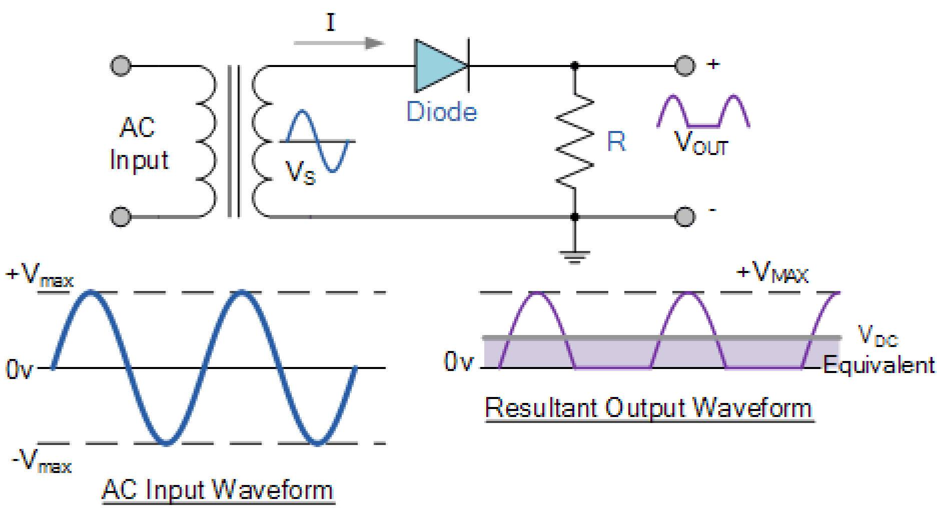


Fig. Half wave diode rectifier circuit and its input and output waveforms

During each "positive" half cycle of the AC sine wave, the diode is forward biased as the anode is positive with respect to the cathode resulting in current flowing through the diode.
Since the DC load is resistive (resistor, R), the current flowing in the load resistor is therefore proportional to the voltage (Ohm´s Law), and the voltage across the load resistor will therefore be the same as the supply voltage, Vs (minus Vf), that is the "DC" voltage across the load is sinusoidal for the first half cycle only so Vout = Vs.

During each "negative" half cycle of the AC sinusoidal input waveform, the diode is reverse biased as the anode is negative with respect to the cathode. Therefore, no current flows through the diode or circuit. Then in the negative half cycle of the supply, no current flows in the load resistor as no voltage appears across it so therefore, Vout = 0.

The current on the DC side of the circuit flows in one direction only making the circuit Unidirectional. As the load resistor receives from the diode a positive half of the waveform, zero volts, a positive half of the waveform, zero volts, etc, the value of this irregular voltage would be equal in value to an equivalent DC voltage of 0.318 x Vmax of the input sinusoidal waveform or 0.45 x Vrms of the input sinusoidal waveform.

6.4 Procedure:

1. Draw the Simulink model with different blocks.
2. Analyze and initialize different blocks of the model.
3. Run the model and see the output.

6.5 Simulation results/Conclusion:

Write conclusion and attach simulation results.

6.6 References:

1. "Power Electronics" by M. H. Rashid, Pearson Education.
2. "Power Electronics" by P. S. Bimra, Khanna Publisher.

Experiment no.: 07

7.1 Aim of the Experiment:

Design Simulink model of full wave (FW) diode bridge rectifier

7.2 Software/tools required:

MATLAB/Simulink

7.3 Theory:

In a Full Wave Rectifier circuit two diodes are now used, one for each half of the cycle. A multiple winding transformer is used whose secondary winding is split equally into two halves with a common centre tapped connection, (C). This configuration results in each diode conducting in turn when its anode terminal is positive with respect to the transformer centre point C producing an output during both half-cycles, twice that for the half wave rectifier so it is 100% efficient as shown below.

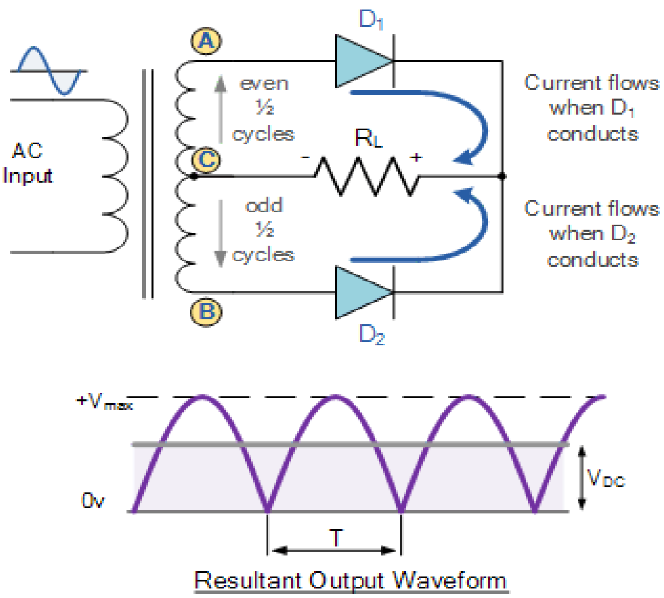


Fig.1 Full-wave centre-tapped rectifier circuit and results

Another type of circuit that produces the same output waveform as the full wave rectifier circuit above is that of the Full Wave Bridge Rectifier. This type of single phase rectifier uses four individual rectifying diodes connected in a closed loop “bridge” configuration to produce the desired output. The main advantage of this bridge circuit is that it does not require a special centre tapped transformer, thereby reducing its size and cost. The single secondary winding is connected to one side of the diode bridge network and the load to the other side as shown below.

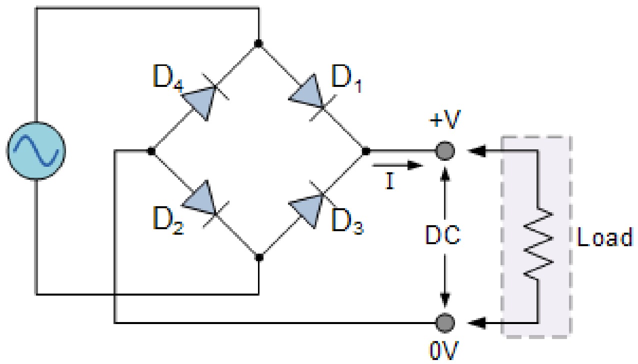


Fig.2 Full-wave bridge rectifier circuit

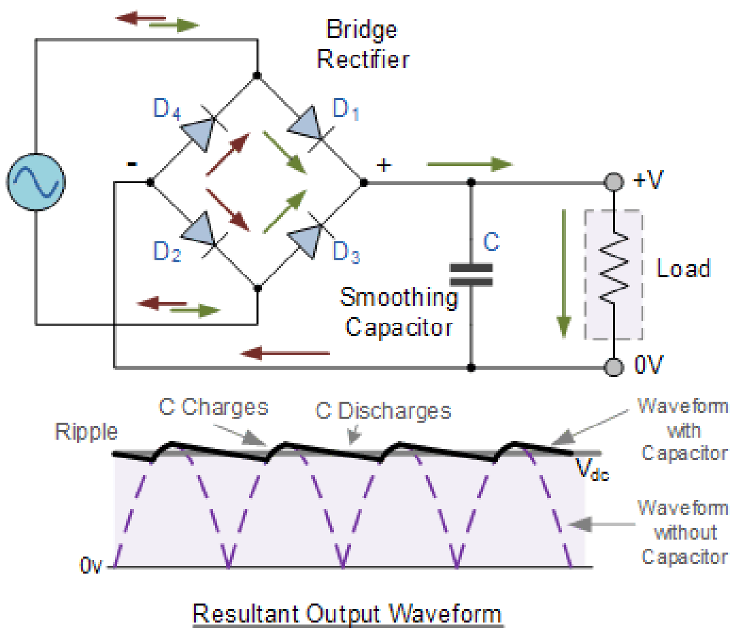


Fig.3 Full-wave bridge rectifier circuit with smoothing capacitors and output voltage waveforms

7.4 Procedure:

1. Draw the Simulink model with different blocks.
2. Analyze and initialize different blocks of the model.
3. Run the model and see the output.

7.5 Simulation results/Conclusion:

Write conclusion and attach simulation results.

7.6 References:

3. "Power Electronics" by M. H. Rashid, Pearson Education.
4. "Power Electronics" by P. S. Bimra, Khanna Publisher.

Experiment no.: 08

8.1 Aim of the Experiment:

Numerical analysis: Non-linear equations and optimization, Differential equations

8.2 Software/tools required:

MATLAB

8.3 Theory:

In mathematics, an ordinary differential equation or ODE is a differential equation containing a function or functions of one independent variable and its derivatives. ODEs that are linear differential equations have exact closed-form solutions that can be added and multiplied by coefficients. By contrast, ODEs that lack additive solutions are nonlinear, and solving them is far more intricate, as one can rarely represent them by elementary functions in closed form. Instead, exact and analytic solutions of ODEs are in series or integral form. Graphical and numerical methods applied by hand or by computer, may approximate solutions of ODEs and perhaps yield useful information, often sufficing in the absence of exact, analytic solutions.

8.4 Procedure:

Solve in MATLAB and plot y(t) or u(t) with t

a. First-Order Linear ODE
   Solve the equation y'(t) = t*y
b. First-Order Linear ODE with initial condition
   What is the actual solution to the following IVP?
   $$2t\,y' + 4y = 3 \qquad y(1) = -4$$
c. Second-Order ODE with Initial Conditions
   Find y solution of the second order nonlinear equation
   $y'' = -2t\,(y')^2$ with initial conditions $y(0) = 2$, $y'(0) = -1$.
d. Nonlinear ODE
   $$\frac{dy}{dx} = 6y^2x \qquad y(1) = \frac{1}{25}$$
e. Third-Order ODE
   Solve this third-order ordinary differential equation
   $$\frac{d^3u}{dx^3} = u$$
   $$u(0)=1,\ u'(0)=-1,\ u''(0)=\pi,$$

f. Find maxima of
   f(x) = x^3 - 12x + 2
   plot (f(x),x)

8.5 Simulation results/Conclusion:

Write conclusion and attach simulation results.

8.6 References:

1. A. Chakrabarti, “Circuit Theory: Analysis and Synthesis”.
2. B. L. Theraja and A. K. Theraja, “A Text Book of Electrical Technology”, Vol. I.
3. M. L. Soni, J. C. Gupta, “A Course in Electrical Circuits and Analysis”.