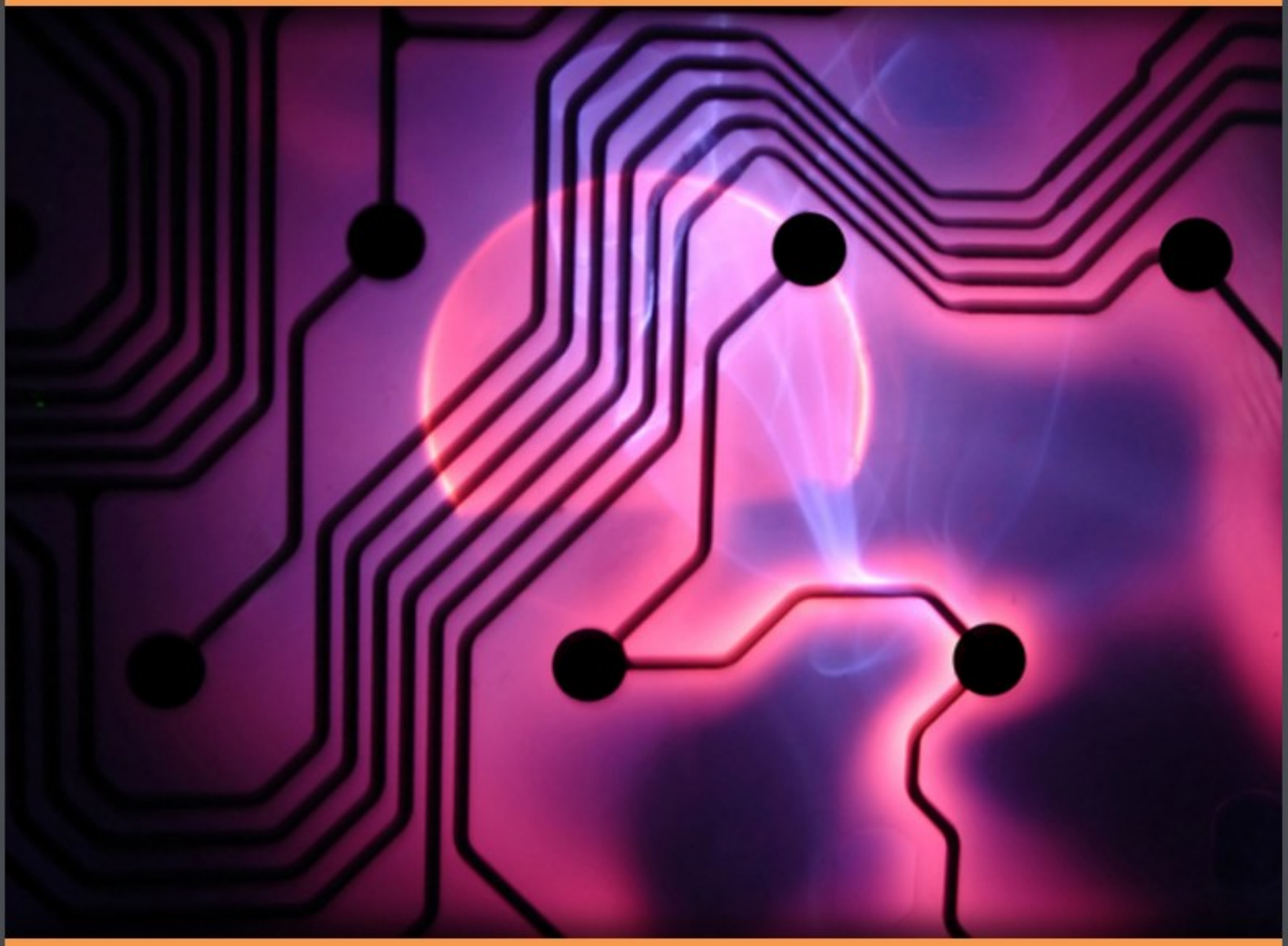
bookboon.com

Concepts in Electric Circuits

Dr. Wasif Naeem



Download free books at

bookboon.com

Dr. Wasif Naeem

Concepts in Electric Circuits

Download free ebooks at bookboon.com

Concepts in Electric Circuits
© 2009 Dr. Wasif Naeem & Ventus Publishing ApS
ISBN 978-87-7681-499-1

Download free ebooks at bookboon.com

Contents

	Preface	**8**
1	**Introduction**	**9**
1.1	Contents of the Book	9
2	**Circuit Elements and Sources**	**11**
2.1	Introduction	11
2.2	Current	11
2.3	Voltage or Potential Difference	13
2.4	Circuit Loads	13
2.5	Sign Convention	15
2.6	Passive Circuit Elements	16
2.6.1	Resistor	16
2.6.2	Capacitor	17
2.6.3	Inductor	18
2.7	DC Sources	19
2.7.1	DC Voltage Source	19

Please click the advert

Download free ebooks at bookboon.com

2.7.2 DC Current Source 22
2.8 Power 23
2.9 Energy 25

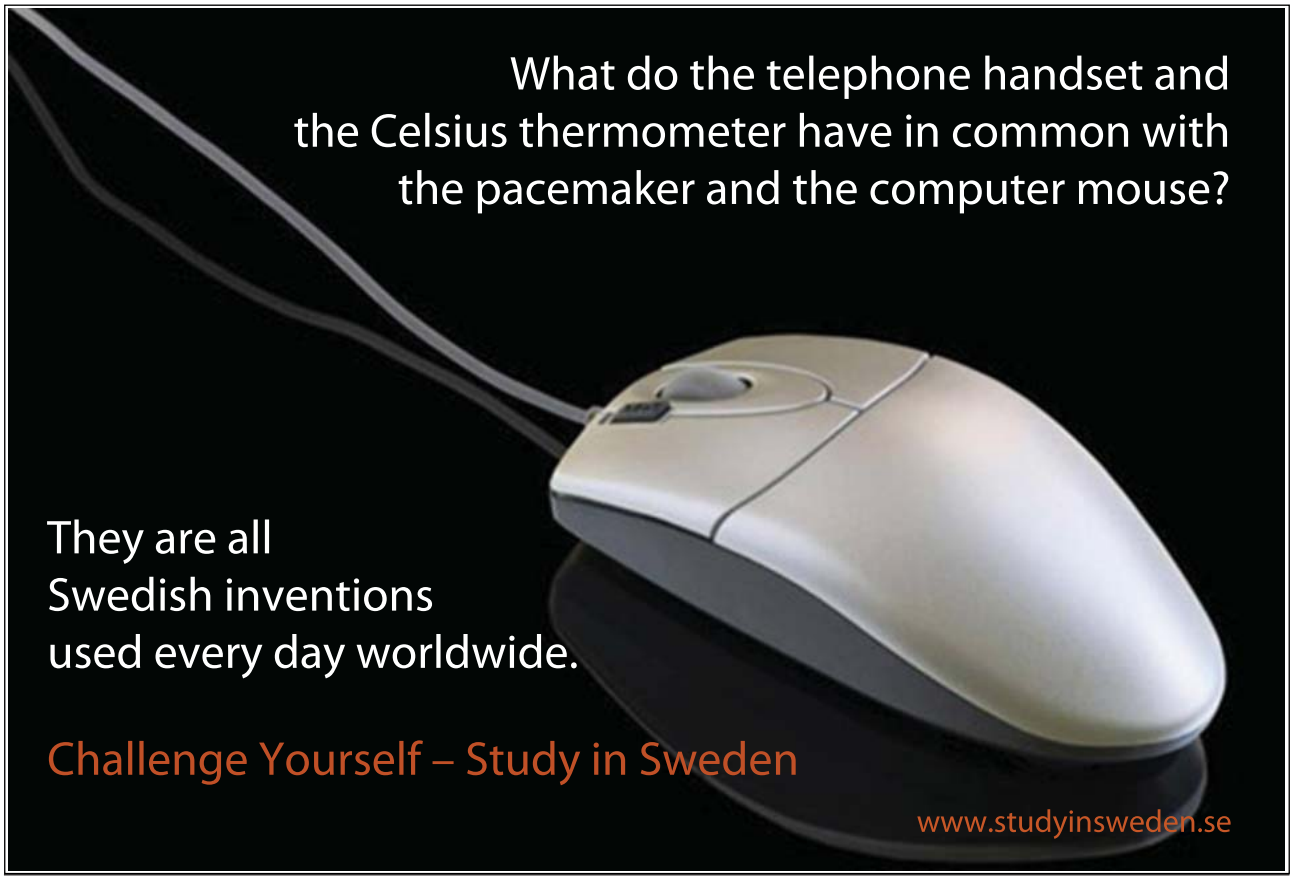
3 Circuit Theorems 27
3.1 Introduction 27
3.2 Definitions and Terminologies 27
3.3 Kirchoff's Laws 28
3.3.1 Kirchoff's Voltage Law (KVL) 28
3.3.2 Kirchoff's Current Law (KCL) 32
3.4 Electric Circuits Analysis 34
3.4.1 Mesh Analysis 34
3.4.2 Nodal Analysis 36
3.5 Superposition Theorem 40
3.6 Thévenin's Theorem 42
3.7 Norton's Theorem 45
3.8 Source Transformation 46
3.9 Maximum Power Transfer Theorem 48
3.10 Additional Common Circuit Configurations 48
3.10.1 Supernode 49

Please click the advert

Download free ebooks at bookboon.com

3.10.2	Supermesh	50
3.11	Mesh and Nodal Analysis by Inspection	51
3.11.1	Mesh Analysis	52
3.11.2	Nodal Analysis	52
4	**Sinusoids and Phasors**	**54**
4.1	Introduction	54
4.2	Sinusoids	54
4.2.1	Other Sinusoidal Parameters	56
4.3	Voltage, Current Relationships for R, L and C	58
4.4	Impedance	59
4.5	Phasors	60
4.6	Phasor Analysis of AC Circuits	65
4.7	Power in AC Circuits	68
4.8	Power Factor	70
4.8.1	Power Factor Correction	71
5	**Frequency Response**	**73**
5.1	Introduction	73
5.2	Frequency Response	74

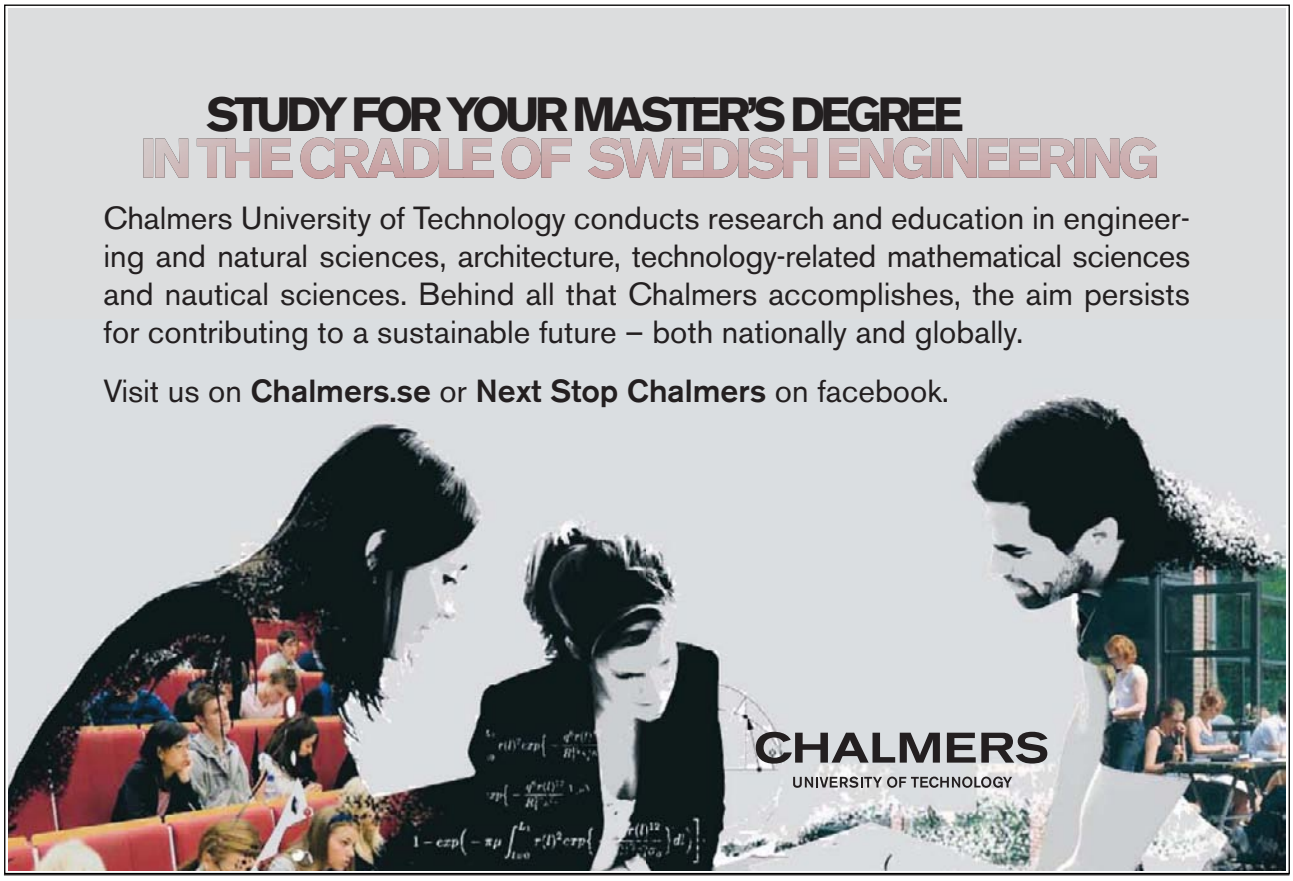
Please click the advert



Download free ebooks at bookboon.com

5.3	Filters	75
5.3.1	Low Pass Filter	75
5.3.2	High Pass Filter	78
5.3.3	Band Pass Filter	80
5.4	Bode Plots	80
5.4.1	Approximate Bode Plots	81
	Appendix A: A Cramer's Rule	**86**

Please click the advert



Download free ebooks at bookboon.com

Preface

This book on the subject of electric circuits forms part of an interesting initiative taken by Ventus Publishing. The material presented throughout the book includes rudimentary learning concepts many of which are mandatory for various engineering disciplines including chemical and mechanical. Hence there is potentially a wide range of audience who could be benefitted.

It is important to bear in mind that this book should not be considered as a replacement of a textbook. It mainly covers fundamental principles on the subject of electric circuits and should provide a solid foundation for more advanced studies. I have tried to keep everything as simple as possible given the diverse background of students. Furthermore, mathematical analysis is kept to a minimum and only provided where necessary.

I would strongly advise the students and practitioners not to carry out any experimental verification of the theoretical contents presented herein without consulting other textbooks and user manuals. Lastly, I shall be pleased to receive any form of feedback from the readers to improve the quality of future revisions.

W. Naeem
Belfast
August, 2009
w.naeem@ee.qub.ac.uk

Download free ebooks at bookboon.com

Chapter 1

Introduction

The discovery of electricity has transformed the world in every possible manner. This phenomenon, which is mostly taken as granted, has had a huge impact on people's life styles. Most, if not all modern scientific discoveries are indebted to the advent of electricity. It is of no surprise that science and engineering students from diverse disciplines such as chemical and mechanical engineering to name a few are required to take courses related to the primary subject of this book. Moreover, due to the current economical and environmental issues, it has never been so important to devise new strategies to tackle the ever increasing demands of electric power. The knowledge gained from this book thus forms the basis of more advanced techniques and hence constitute an important part of learning for engineers.

The primary purpose of this compendium is to introduce to students the very fundamental and core concepts of electricity and electrical networks. In addition to technical and engineering students, it will also assist practitioners to adopt or refresh the rudimentary know-how of analysing simple as well as complex electric circuits without actually going into details. However, it should be noted that this compendium is by no means a replacement of a textbook. It can perhaps serve as a useful tool to acquire focussed knowledge regarding a particular topic. The material presented is succinct with numerical examples covering almost every concept so a fair understanding of the subject can be gained.

1.1 Contents of the Book

There are five chapters in this book highlighting the elementary concepts of electric circuit analysis. An appendix is also included which provides the reader a mathematical tool to solve a simultaneous system of equations frequently used in this book. Chapter 2 outlines the idea of voltage and current parameters in an electric network. It also explains the voltage polarity and current direction and the technique to correctly measure these quantities in a simple manner. Moreover, the fundamental circuit elements such as a resistor, inductor and capacitor are introduced and their voltage-current relationships are provided. In the end, the concept of power and energy and their mathematical equations in terms of voltage and current are presented. All the circuit elements introduced in this chapter are explicated in the context of voltage and current parameters. For a novice reader, this is particularly helpful as it will allow the student to master the basic concepts before proceeding to the next chapter.

Download free ebooks at bookboon.com

A reader with some prior knowledge regarding the subject may want to skip this chapter although it is recommended to skim through it so a better understanding is gained without breaking the flow.

In Chapter 3, the voltage-current relationships of the circuit elements introduced in Chapter 2 are taken further and various useful laws and theorems are presented for DC[1] analysis. It is shown that these concepts can be employed to study simple as well as very large and complicated DC circuits. It is further demonstrated that a complex electrical network can be systematically scaled down to a circuit containing only a few elements. This is particularly useful as it allows to quickly observe the affect of changing the load on circuit parameters. Several examples are also supplied to show the applicability of the concepts introduced in this chapter.

Chapter 4 contains a brief overview of AC circuit analysis. In particular the concept of a sinusoidal signal is presented and the related parameters are discussed. The AC voltage-current relationships of various circuit elements presented in Chapter 2 are provided and the notion of impedance is explicated. It is demonstrated through examples that the circuit laws and theorems devised for DC circuits in Chapter 3 are all applicable to AC circuits through the use of phasors. In the end, AC power analysis is carried out including the use of power factor parameter to calculate the actual power dissipated in an electrical network.

The final chapter covers AC circuit analysis using frequency response techniques which involves the use of a time-varying signal with a range of frequencies. The various circuit elements presented in the previous chapters are employed to construct filter circuits which possess special characteristics when viewed in frequency domain. Furthermore, the chapter includes the mathematical analysis of filters as well as techniques to draw the approximate frequency response plots by inspection.

[1] A DC voltage or current refers to a constant magnitude signal whereas an AC signal varies continuously with respect to time.

Download free ebooks at bookboon.com

Chapter 2

Circuit Elements and Sources

2.1 Introduction

This chapter provides an overview of most commonly used elements in electric circuits. It also contains laws governing the current through and voltage across these components as well as the power supplied/dissipated and energy storage in this context. In addition, difference between ideal and non-ideal voltage and current sources is highlighted including a discussion on sign convention i.e. voltage polarity and current direction.

The concepts of current and voltage are first introduced as these constitutes one of the most fundamental concepts particularly in electronics and electrical engineering.

2.2 Current

Current can be defined as the motion of charge through a conducting material. The unit of current is Ampere whilst charge is measured in Coulombs.

Definition of an Ampere

> "*The quantity of total charge that passes through an arbitrary cross section of a conducting material per unit second is defined as an Ampere.*"

Mathematically,

$$I = \frac{Q}{t} \text{ or } Q = It \tag{2.1}$$

where Q is the symbol of charge measured in Coulombs (C), I is the current in amperes (A) and t is the time in seconds (s).

The current can also be defined as the rate of charge passing through a point in an electric circuit i.e.

$$i = \frac{dQ}{dt} \tag{2.2}$$

Download free ebooks at bookboon.com

A constant current (also known as direct current or DC) is denoted by the symbol I whereas a time-varying current (also known as alternating current or AC) is represented by the symbol i or $i(t)$.

Current is always measured through a circuit element.

Figure 2.1 demonstrates the use of an ampere-meter or ammeter in series with a circuit element, R, to measure the current through it.

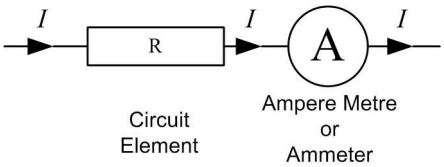


Figure 2.1: An ammeter is connected in series to measure current, I, through the element, R.

Example

Determine the current in a circuit if a charge of 80 coulombs (C) passes a given point in 20 seconds (s).

$$Q = 80 \text{ C}, t = 20 \text{ s}, I = ?$$

$$I = \frac{Q}{t} = \frac{80}{20} = 4 \text{ A}$$

Please click the advert

GCHQ
it's an interesting world

Get under the skin of it.

Graduate opportunities
Cheltenham | £24,945 + benefits

One of the UK's intelligence services, GCHQ's role is two-fold: to gather and analyse intelligence which helps shape Britain's response to global events, and, to provide technical advice for the protection of Government communication and information systems. In doing so, our specialists – in IT, internet, engineering, languages, information assurance, mathematics and intelligence – get well beneath the surface of global affairs. If you thought the world was an interesting place, you really ought to explore our world of work.

TOP 100 TOP GOVERNMENT EMPLOYER INVESTORS IN PEOPLE

www.careersinbritishintelligence.co.uk

Applicants must be British citizens. GCHQ values diversity and welcomes applicants from all sections of the community. We want our workforce to reflect the diversity of our work.

Download free ebooks at bookboon.com

2.3 Voltage or Potential Difference

Definition

> *Voltage or potential difference between two points in an electric circuit is 1 V if 1 J (Joule) of energy is expended in transferring 1 C of charge between those points.*

It is generally represented by the symbol V and measured in volts (V). Note that the symbol and the unit of voltage are both denoted by the same letter, however, it rarely causes any confusion.

The symbol V also signifies a constant voltage (DC) whereas a time-varying (AC) voltage is represented by the symbol v or $v(t)$.

Voltage is always measured across a circuit element as demonstrated in Figure 2.2.

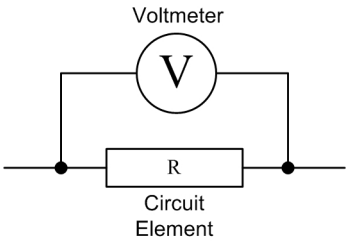


Figure 2.2: A voltmeter is connected in parallel with the circuit element, R to measure the voltage across it.

A voltage source provides the energy or emf (electromotive force) required for current flow. However, current can only exist if there is a potential difference and a physical path to flow. A potential difference of 0 V between two points implies 0 A of current flowing through them. The current I in Figure 2.3 is 0 A since the potential difference across R_2 is 0 V. In this case, a physical path exists but there is no potential difference. This is equivalent to an open circuit.

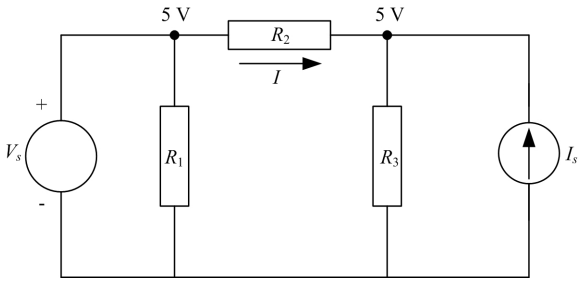


Figure 2.3: The potential difference across R_2 is 0 V, hence the current I is 0 A where V_s and I_s are the voltage and current sources respectively.

Table 2.1 summarises the fundamental electric circuit quantities, their symbols and standard units.

2.4 Circuit Loads

A load generally refers to a component or a piece of equipment connected to the output of an electric circuit. In its fundamental form, the load is represented by any one or a combination of the following

Download free ebooks at bookboon.com

Quantity	Symbol	Unit
Voltage	V	Volts (V)
Current	I	Ampere (A)
Charge	Q	Coulomb (C)
Power	P	Watts (W)
Energy	W	Joules (J)
Time	t	seconds (s)

Table 2.1: Standard quantities and their units commonly found in electric circuits.

circuit elements

1. Resistor (R)
2. Inductor (L)
3. Capacitor (C)

A load can either be of resistive, inductive or capacitive nature or a blend of them. For example, a light bulb is a purely resistive load where as a transformer is both inductive and resistive. A circuit load can also be referred to as a *sink* since it dissipates energy whereas the voltage or current supply can be termed as a *source*.

Table 2.2 shows the basic circuit elements along with their symbols and schematics used in an electric circuit. The R, L and C are all passive components i.e. they do not generate their own emf whereas the DC voltage and current sources are active elements.

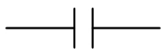
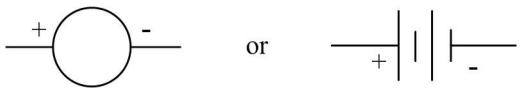
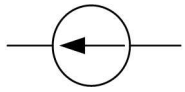
Circuit Element	Symbol	Schematic
Resistor	R	
Inductor	L	
Capacitor	C	
DC Voltage Source	V_s	
DC Current Source	I_s	

Table 2.2: Common circuit elements and their representation in an electric circuit.

Download free ebooks at bookboon.com

2.5 Sign Convention

It is common to think of current as the flow of electrons. However, the standard convention is to take the flow of protons to determine the direction of the current.

In a given circuit, the current direction depends on the polarity of the source voltage. Current always flow from positive (high potential) side to the negative (low potential) side of the source as shown in the schematic diagram of Figure 2.4(a) where V_s is the source voltage, V_L is the voltage across the load and I is the loop current flowing in the clockwise direction.

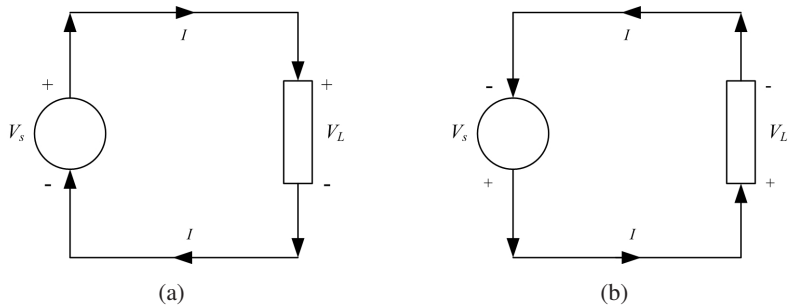


Figure 2.4: Effect of reversing the voltage polarity on current direction.

Please observe that the voltage polarity and current direction in a sink is opposite to that of the source.

In Source	current leaves from the positive terminal
In Load (Sink)	current enters from the positive terminal

A reversal in source voltage polarity changes the direction of the current flow and vice versa as depicted in Figures 2.4(a) and 2.4(b).

Please click the advert

HORIZONS UNIVERSITY

In Paris or Online

International programs taught by professors and professionals from all over the world

BBA in Global Business
MBA in International Management / International Marketing
DBA in International Business / International Management
MA in International Education
MA in Cross-Cultural Communication
MA in Foreign Languages

Innovative – Practical – Flexible – Affordable

Visit: www.HorizonsUniversity.org
Write: Admissions@horizonsuniversity.org
Call: 01.42.77.20.66

www.HorizonsUniversity.org

Download free ebooks at bookboon.com

2.6 Passive Circuit Elements

2.6.1 Resistor

To describe the resistance of a resistor and hence its characteristics, it is important to define the Ohm's law.

Ohm's Law

It is the most fundamental law used in circuit analysis. It provides a simple formula describing the voltage-current relationship in a conducting material.

Statement

> *The voltage or potential difference across a conducting material is directly proportional to the current flowing through the material.*

Mathematically

$$V \propto I$$
$$V = RI \text{ or } I = \frac{V}{R} \text{ or } R = \frac{V}{I}$$

where the constant of proportionality R is called the resistance or electrical resistance, measured in ohms (Ω). Graphically, the $V - I$ relationship for a resistor according to Ohm's law is depicted in Figure 2.5.

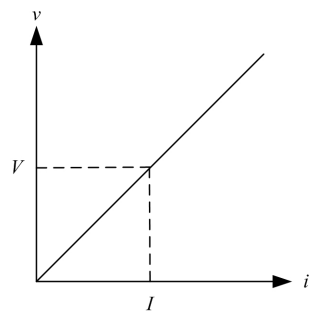


Figure 2.5: $V - I$ relationship for a resistor according to Ohm's law.

At any given point in the above graph, the ratio of voltage to current is always constant.

Example

Find R if the voltage V and current I in Figure 2.5 are equal to 10 V and 5 A respectively.

$$V = 10 \text{ V}, I = 5 \text{ A}, R = ?$$

Using Ohm's law

$$V = IR \text{ or } R = \frac{V}{I} = \frac{10}{5}$$

$$\therefore R = 2\ \Omega$$

A short circuit between two points represents a zero resistance whereas an open circuit corresponds to an infinite resistance as demonstrated in Figure 2.6.

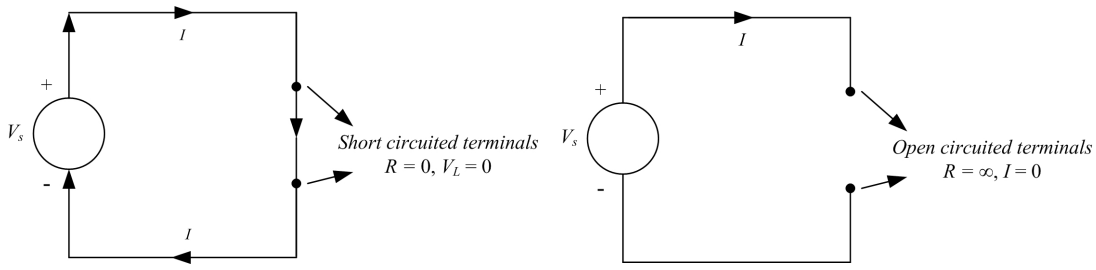


Figure 2.6: Short circuit and open circuit resistance characteristics.

Using Ohm's law,

when R = 0 (short circuit), V = 0 V

when R = ∞ (open circuit), I = 0 A

Conductance

Conductance (G) is the exact opposite of resistance. In mathematical terms,

$$G = \frac{1}{R}$$

$$\therefore I = \frac{V}{R} = VG$$

where G is measured in siemens (S) and sometimes also represented by the unit mho $(\mho)$ (upside-down omega).

2.6.2 Capacitor

A capacitor is a passive circuit element that has the capacity to store charge in an electric field. It is widely used in electric circuits in the form of a filter. The $V - I$ relationship for a capacitor is governed by the following equation

$$i = C\frac{dv}{dt} \text{ or } v = \frac{1}{C}\int_0^t idt + v(0)$$

where C is the capacitance measured in Farads (F) and $v(0)$ is the initial voltage or initial charge stored in the capacitor.

When $v = V$ (constant DC voltage), $\frac{dv}{dt} = 0$, and $i = 0$. Hence a capacitor acts as an open circuit to DC.

Download free ebooks at bookboon.com

Example

For the circuit diagram shown in Figure 2.7, determine the current, I flowing through the 5 Ω resistance.

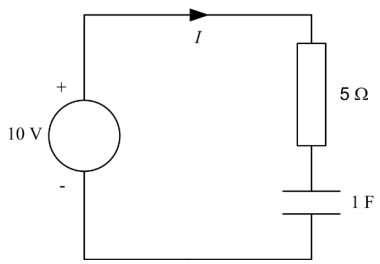


Figure 2.7

Since the supply voltage is DC, therefore the capacitor will act as an open circuit. Hence no current can flow through the circuit regardless of the values of capacitor and resistor i.e.

$$I = 0$$

2.6.3 Inductor

An inductor is a piece of conducting wire generally wrapped around a core of a ferromagnetic material. Like capacitors, they are employed as filters as well but the most well known application is their use in AC transformers or power supplies that converts AC voltage levels.

In an inductor, the $V - I$ relationship is given by the following differential equation

$$v = L\frac{di}{dt} \text{ or } i = \frac{1}{L}\int_0^t v dt + i(0)$$

where L is the inductance in Henrys (H) and $i(0)$ is the initial current stored in the magnetic field of the inductor.

When $i = I$ (constant DC current), $\frac{di}{dt} = 0$, $v = 0$. Hence an inductor acts as a short circuit to DC. An ideal inductor is just a piece of conducting material with no internal resistance or capacitance. The schematics in Figure 2.8 are equivalent when the supply voltage is DC.

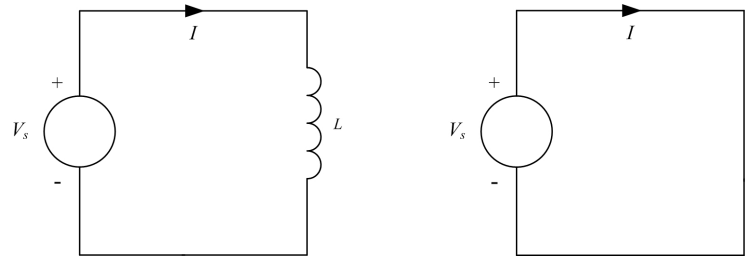


Figure 2.8: An ideal inductor can be replaced by a short circuit when the supply voltage is DC.

A summary of the $V - I$ relationships for the three passive circuit elements is provided in Table 2.3.

Circuit Element	Voltage	Current
Resistor	$V = IR$	$I = \dfrac{V}{R}$
Inductor	$v = L\dfrac{di}{dt}$, $v = 0$ for DC	$i = \dfrac{1}{L}\displaystyle\int_0^t vdt + i(0)$
Capacitor	$v = \dfrac{1}{C}\displaystyle\int_0^t idt + v(0)$	$i = C\dfrac{dv}{dt}$, $i = 0$ for DC

Table 2.3: $V - I$ relationships for a resistor, inductor and capacitor.

2.7 DC Sources

In general, there are two main types of DC sources

1. Independent (Voltage and Current) Sources
2. Dependent (Voltage and Current) Sources

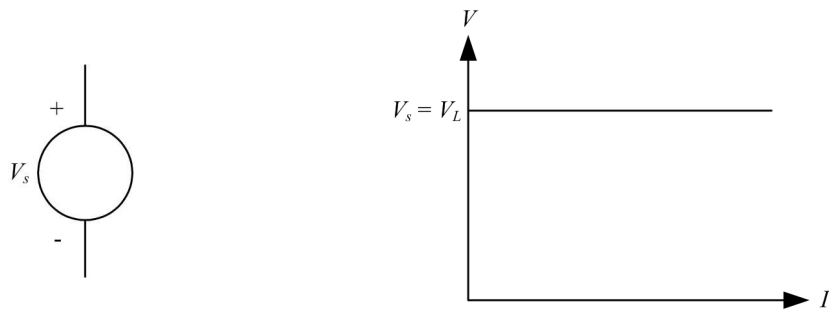
An independent source produces its own voltage and current through some chemical reaction and does not depend on any other voltage or current variable in the circuit. The output of a dependent source, on the other hand, is subject to a certain parameter (voltage or current) change in a circuit element. Herein, the discussion shall be confined to independent sources only.

2.7.1 DC Voltage Source

This can be further subcategorised into ideal and non-ideal sources.

Download free ebooks at bookboon.com

The Ideal Voltage Source An ideal voltage source, shown in Figure 2.9(a), has a terminal voltage which is independent of the variations in load. In other words, for an ideal voltage source, the supply current alters with changes in load but the terminal voltage, V_L always remains constant. This characteristic is depicted in Figure 2.9(b).



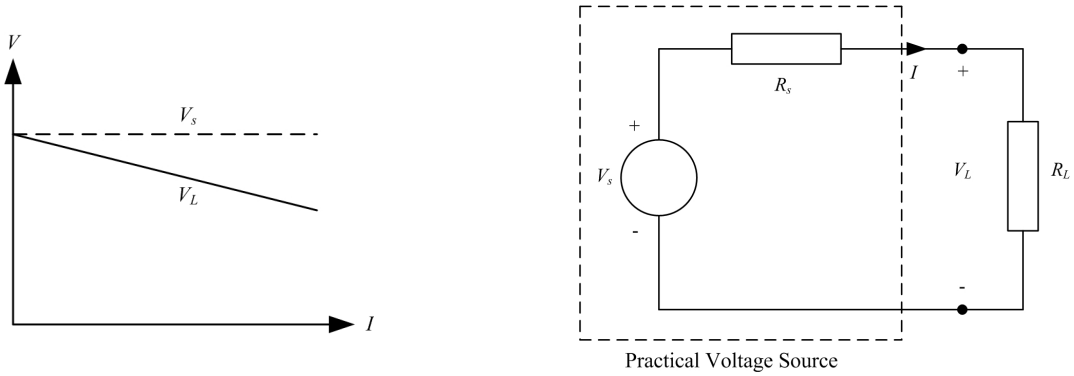
(a) An ideal voltage source.

(b) $V - I$ characteristics of an ideal voltage source.

Figure 2.9: Schematic and characteristics of an ideal voltage source

Non-Ideal or Practical Voltage Source For a practical source, the terminal voltage falls off with an increase in load current. This can be shown graphically in Figure 2.10(a).

This behaviour can be modelled by assigning an internal resistance, R_s, in series with the source as shown in Figure 2.10(b).



(a) $V - I$ characteristics of a practical voltage source

(b) A practical voltage source has an internal resistance connected in series with the source.

Figure 2.10: Characteristics and model of a practical voltage source

where R_L represents the load resistance.

The characteristic equation of the practical voltage source can be written as

$$V_L = V_s - R_s I$$

For an ideal source, $Rs = 0$ and therefore $V_L = V_s$.

Example

The terminal voltage of a battery is 14 V at no load. When the battery is supplying 100 A of current to a load, the terminal voltage drops to 12 V. Calculate the source impedance[1].

$$V_s = V_{NL} = 14.0 \text{ V when } I = 0 \text{ A (without load)}$$

$$V_L = 12.0 \text{ V when } I = 100 \text{ A (at full load)}$$

$$\because V_L = V_s - R_s I$$

$$R_s = \frac{V_s - V_L}{I} = \frac{14 - 12}{100} = \frac{2}{100}$$

$$R_s = 0.02\ \Omega$$

[1]Impedance is a more common terminology used in practice instead of resistance. However, impedance is a generic term which could include inductive and capacitive reactances. See Chapter 4 for more details

Please click the advert

Download free ebooks at bookboon.com

Voltage Regulation

Voltage regulation (VR) is an important measure of the quality of a source. It is used to measure the variation in terminal voltage between no load ($I_L = 0$, open circuit) and full load $(I_L = I_{FL})$ as shown in Figure 2.11.

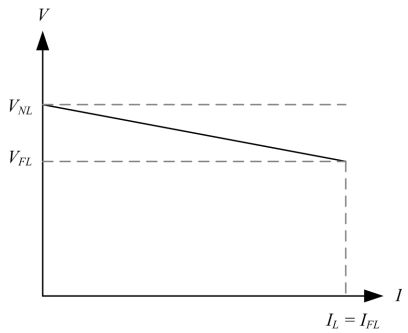


Figure 2.11: No load and full load voltages specified on a $V - I$ characteristic plot of a practical voltage source.

If V_{NL} and V_{FL} represents the no load and full load voltages, then the VR of a source is defined mathematically as

$$VR = \frac{V_{NL} - V_{FL}}{V_{FL}} \times 100\%$$

For an ideal source, there is no internal resistance and hence $V_{NL} = V_{FL}$ and

$$VR = 0\%$$

Hence, the smaller the regulation, the better the source.

In the previous example, $V_{NL} = 14.0$ V and $V_{FL} = 12.0$ V, therefore

$$VR = \frac{14 - 12}{12} \times 100 = 16.67\%$$

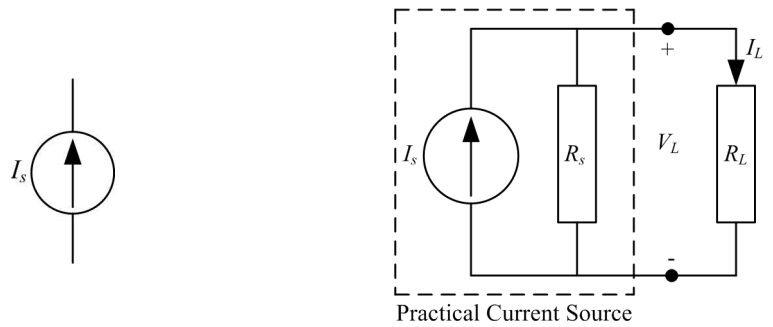
2.7.2 DC Current Source

A current source, unlike the DC voltage source, is not a physical reality. However, it is useful in deriving equivalent circuit models of semiconductor devices such as a transistor. It can also be subdivided into ideal and non-ideal categories.

The Ideal Current Source By definition, an ideal current source, depicted in Figure 2.12(a), produces a current which is independent of the variations in load. In other words the current supplied by an ideal current source does not change with the load voltage.

Non-Ideal or Practical Current Source The current delivered by a practical current source falls off with an increase in load or load voltage. This behaviour can be modelled by connecting a resistance in parallel with the ideal current source as shown in Figure 2.12(b) where R_s is the internal

Download free ebooks at bookboon.com



(a) An ideal current source

(b) A practical current source has an internal resistance connected in parallel with the source.

Figure 2.12: Ideal and non-ideal current sources.

resistance of the current source and R_L represents the load.

The characteristic equation of the practical current source can be written as

$$I_L = I_s - \frac{V_L}{R_s}$$

In an ideal current source, $R_s = \infty$ (open circuit), therefore $I_L = I_s$.

2.8 Power

Given the magnitudes of V and I, power can be evaluated as the product of the two quantities and is measured in Watts (W).

Mathematically

$$P = VI(\text{W})$$

Example

If the power dissipated in a circuit element is 100 W and a current of 10 A is flowing through it, calculate the voltage across and resistance of the element.

$$P = 100 \text{ W}, I = 10 \text{ A}, V = ?, R = ?$$

$$P = VI$$

$$V = \frac{P}{I} = \frac{100}{10} = 10 \text{ V}$$

$$\text{Also, } R = \frac{V}{I} = \frac{10}{10} = 1\ \Omega$$

Alternate Expressions for Power Using Ohm's Law

Using Ohm's law i.e. $V = IR$, two more useful expressions for the power absorbed/delivered can be derived as follows

$$P = VI = (IR)I = I^2R$$

Also, $I = \frac{V}{R}$

$$\therefore P = VI = V\frac{V}{R} = \frac{V^2}{R}$$

Example

A light bulb draws 0.5 A current at an input voltage of 230 V. Determine the resistance of the filament and also the power dissipated.

From Ohm's law

$$R = \frac{V}{I} = \frac{230}{0.5} = 460\ \Omega$$

Since a bulb is a purely resistive load, therefore all the power is dissipated in the form of heat. This can be calculated using any of the three power relationships shown above

$$\begin{aligned} P &= VI = 230 \times 0.5 = 115 \text{ W} \\ P &= I^2R = (0.5)^2 \times 460 = 115 \text{ W} \\ P &= \frac{V^2}{R} = \frac{(230)^2}{460} = 115 \text{ W} \end{aligned}$$

Please click the advert

Download free ebooks at bookboon.com

2.9 Energy

Energy is defined as the capacity of a physical system to perform work. In the context of electric circuits, energy (w) is related to power by the following relationship

$$p = vi = \frac{dw}{dt}$$

i.e. power is the rate of change of energy.

Using Equation 2.2, voltage can also be written in terms of energy as the work done or energy supplied per unit charge (q) i.e.

$$v = \frac{dw}{dq}$$

In terms of the three passive circuit elements, R, L and C, the energy relationships can be derived as follows

Resistor

$$p = vi = i^2R = \frac{v^2}{R}$$

Electrical power or energy supplied to a resistor is completely dissipated as heat. This action is irreversible and is also commonly termed as i^2R losses.

Inductor

$$p = vi = Li\frac{di}{dt}$$

Total energy supplied from 0 to t is

$$w = \int_0^t p\,dt = L\int_0^t i\frac{di}{dt}dt = L\int i\,di$$

$$\therefore \boxed{W = \frac{1}{2}LI^2}$$

This energy is stored in the magnetic field of the inductor which can be supplied back to the circuit when the actual source is removed.

Download free ebooks at bookboon.com

Capacitor

$$p = vi = Cv\frac{dv}{dt}$$

Total energy supplied from 0 to t is

$$w = \int_0^t p\,dt = C\int_0^t v\frac{dv}{dt}dt = C\int v\,dv$$

$$\therefore \boxed{W = \frac{1}{2}CV^2}$$

This energy is stored in the electric field of the capacitor which is supplied back to the circuit when the actual source is removed.

Please click the advert

Destination MMU

Manchester Metropolitan University

MMU is proud to be one of the most popular universities in the UK. Some 34,000 students from all parts of the globe select from its curricula of over 1,000 courses and qualifications.

We are based in the dynamic yet conveniently compact city of Manchester, located at the heart of a sophisticated transport network including a major international airport on the outskirts. Parts of the campus are acclaimed for their architectural style and date back over 150 years, in direct contrast to our teaching style which is thoroughly modern, innovative and forward-thinking.

MMU offers undergraduate and postgraduate courses in the following subject areas:

- Art, Design & Performance
- Computing, Engineering & Technology
- Business & Management
- Science, Environmental Studies & Geography
- Law, Education & Psychology
- Food, Hospitality, Tourism & Leisure Studies
- Humanities & Social Science

For more details or an application form please contact MMU International.
email: international@mmu.ac.uk
telephone: +44 (0)161 247 1022
www.mmu.ac.uk/international

Download free ebooks at bookboon.com

Chapter 3

Circuit Theorems

3.1 Introduction

This chapter outlines the most commonly used laws and theorems that are required to analyse and solve electric circuits. Relationships between various laws and equation writing techniques by inspection are also provided. Several examples are shown demonstrating various aspects of the laws. In addition, situations are presented where it is not possible to directly apply the concepts and potential remedies are provided.

3.2 Definitions and Terminologies

In the following, various definitions and terminologies frequently used in circuit analysis are outlined. The reader will regularly encounter these terminologies and hence it is important to comprehend those at this stage.

- ***Electric Network*** A connection of various circuit elements can be termed as an *electric network*. The circuit diagram shown in Figure 3.1 is an electric network.
- ***Electric Circuit*** A connection of various circuit elements of an electric network forming a closed path is called an *electric circuit*. The closed path is commonly termed as either *loop* or *mesh*.

  In Figure 3.1, meshes BDEB, ABCA and BCDB are electric circuits because they form a closed path. In general, all circuits are networks but not all networks are circuits.
- ***Node*** A connection point of several circuit elements is termed as a *node*. For instance, A, B, C, D and E are five nodes in the electric network of Figure 3.1. Please note that there is no element connected between nodes A and C and therefore can be regarded as a single node.
- ***Branch*** The path in an electric network between two nodes is called a *branch*. AB, BE, BD, BC, CD and DE are six branches in the network of Figure 3.1.

Download free ebooks at bookboon.com

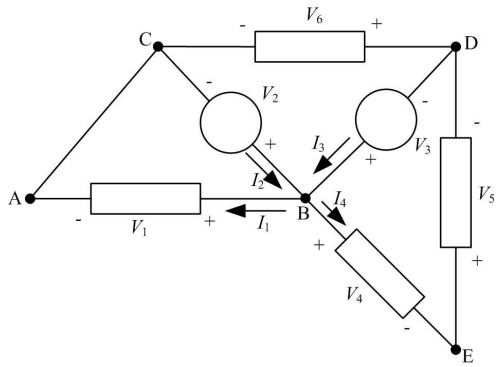


Figure 3.1: An electric network showing nodes, branches, elements and loops.

3.3 Kirchoff's Laws

Arguably the most common and useful set of laws for solving electric circuits are the Kirchoff's voltage and current laws. Several other useful relationships can be derived based on these laws.

3.3.1 Kirchoff's Voltage Law (KVL)

"*The sum of all the voltages (rises and drops) around a closed loop is equal to zero.*"

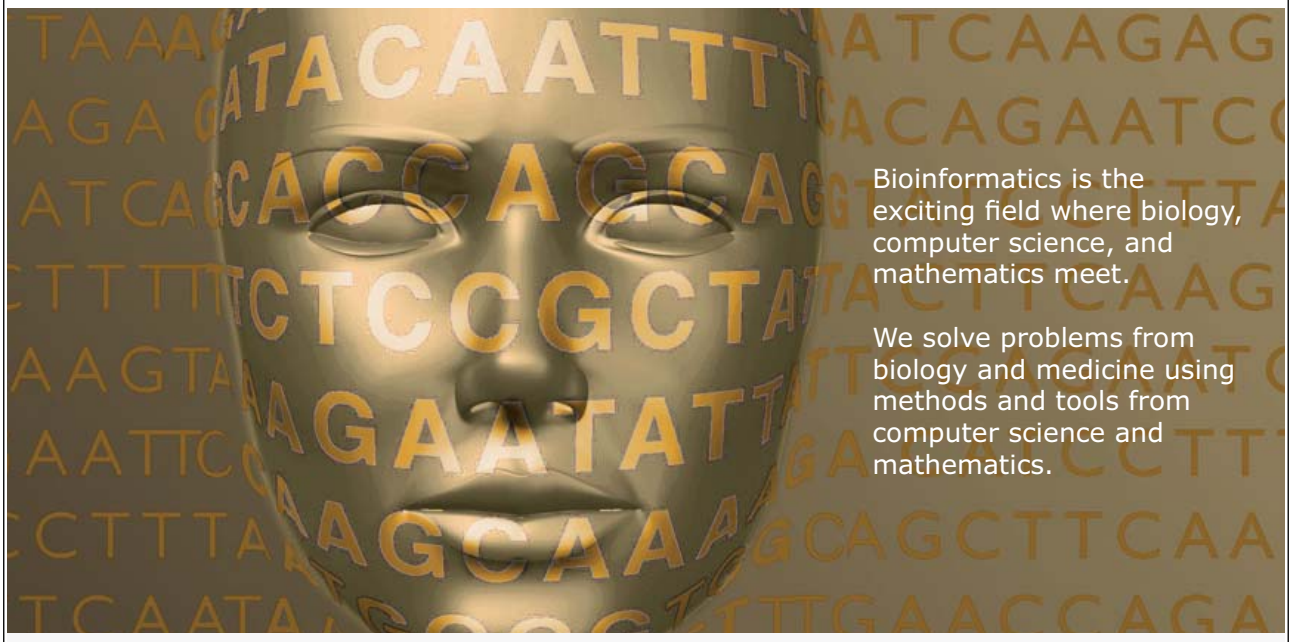
In other words, the algebraic sum of all voltage rises is equal to the algebraic sum of all the voltage drops around a closed loop. In Figure 3.1, consider mesh BEDB, then according to KVL

$$V_3 = V_4 + V_5$$

Please click the advert

UPPSALA UNIVERSITET

Develop the tools we need for Life Science
Masters Degree in Bioinformatics

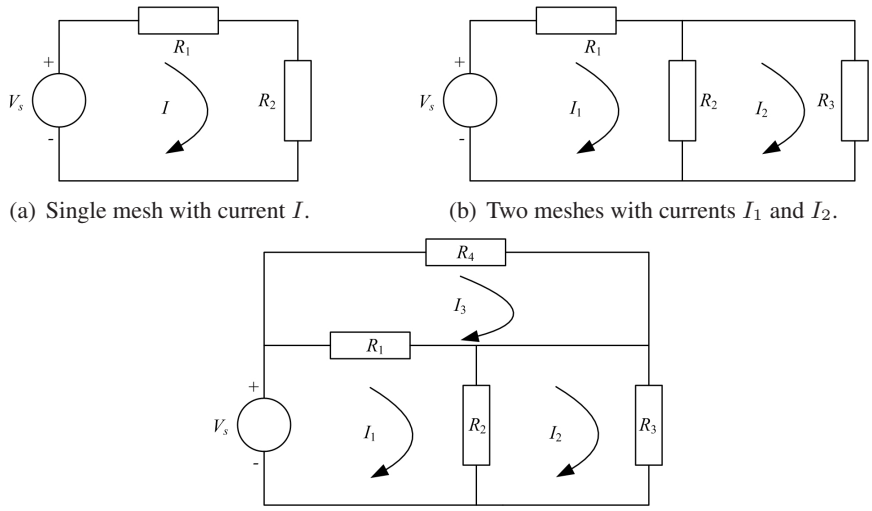


Read more about this and our other international masters degree programmes at www.uu.se/master

Download free ebooks at bookboon.com

Example

In each of the circuit diagrams in Figure 3.2, write the mesh equations using KVL.



(a) Single mesh with current I.

(b) Two meshes with currents I_1 and I_2.

(c) Three meshes with currents I_1, I_2 and I_3.

Figure 3.2: Circuit diagrams to demonstrate the application of KVL in the above example.

Figure 3.2(a) contains a single loop hence a single current, I is flowing around it. Therefore a single equation will result as given below

$$V_s = IR_1 + IR_2 \tag{3.1}$$

If V_s, R_1 and R_2 are known, then I can be found.

Figure 3.2(b) contains two meshes with currents I_1 and I_2 hence there will be two equations as shown below. Note that the branch containing R_2 is common to both meshes with currents I_1 and I_2 flowing in opposite directions.

Left loop

$$\begin{aligned} V_s &= I_1R_1 + (I_1 - I_2)R_2 \\ V_s &= (R_1 + R_2)I_1 - R_2I_2 \end{aligned} \tag{3.2}$$

Right loop

$$\begin{aligned} 0 &= (I_2 - I_1)R_2 + I_2R_3 \\ 0 &= -R_2I_1 + (R_2 + R_3)I_2 \end{aligned} \tag{3.3}$$

Given V_s, R_1, R_2 & R_3, Equations 3.2 and 3.3 can be solved simultaneously to evaluate I_1 and I_2.

For the circuit diagram of Figure 3.2(c), three equations need to be written as follows. Also note that there is no circuit element shared between loops 2 and 3 hence I_2 and I_3 are independent of each other.

Download free ebooks at bookboon.com

Left bottom loop

$$\begin{aligned} V_s &= (I_1 - I_3)R_1 + (I_1 - I_2)R_2 \\ V_s &= (R_1 + R_2)I_1 - R_2 I_2 - R_1 I_3 \end{aligned} \tag{3.4}$$

Right bottom loop

$$\begin{aligned} 0 &= (I_2 - I_1)R_2 + I_2 R_3 \\ 0 &= -R_2 I_1 + (R_2 + R_3)I_2 \end{aligned} \tag{3.5}$$

Upper loop

$$\begin{aligned} 0 &= (I_3 - I_1)R_1 + I_3 R_4 \\ 0 &= -R_1 I_1 + (R_1 + R_4)I_3 \end{aligned} \tag{3.6}$$

If V_s and resistors' values are known, the mesh currents can be evaluated by solving[1] Equations 3.4, 3.5 and 3.6 simultaneously.

Resistors in Series

Consider Figure 3.3 with one voltage source and two resistors connected in series to form a single mesh with current I.

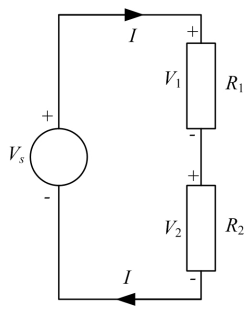


Figure 3.3: Series combination of two resistors.

According to KVL

$$V_s = V_1 + V_2$$

Using Ohm's law $(V = IR)$ from Chapter 2,

$$\begin{aligned} IR_{eq} &= IR_1 + IR_2 \\ R_{eq} &= R_1 + R_2 \end{aligned} \tag{3.7}$$

[1]When there are three or more unknown variables, it may be convenient to use matrix method or Cramer's rule. See Appendix A for a description of Cramer's rule.

Download free ebooks at bookboon.com

where R_{eq} is the combined or equivalent resistance of the series network. Hence the equivalent resistance of two or more resistors connected in series is given by the algebraic sum of all the resistances. In general, for n number of serial resistors, R_{eq} is given by

$$\boxed{R_{eq} = R_1 + R_2 + R_3 + \cdots + R_n} \tag{3.8}$$

Voltage Divider Rule (VDR)

Voltage divider rule provides a useful formula to determine the voltage across any resistor when two or more resistors are connected in series with a voltage source. In Figure 3.3, the voltage across the individual resistors can be given in terms of the supply voltage and the magnitude of individual resistances as follows

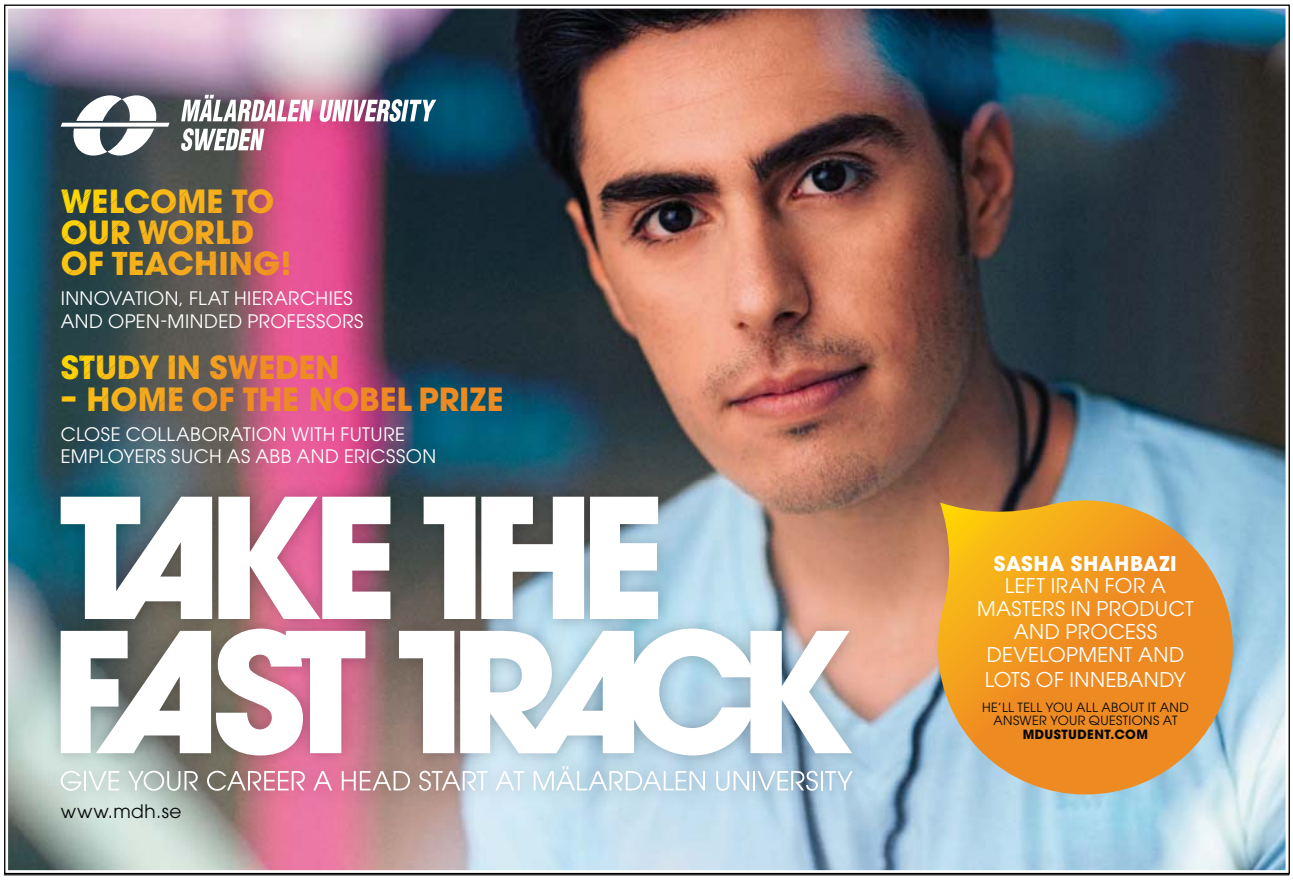
$$V_1 = V_s \frac{R_1}{R_1 + R_2} \tag{3.9}$$

$$V_2 = V_s \frac{R_2}{R_1 + R_2} \tag{3.10}$$

In general, for n number of resistors connected in series, the voltage across the ith resistor can be specified as

$$\boxed{V_i = V_s \frac{R_i}{R_1 + R_2 + \cdots + R_i + \cdots + R_n}} \tag{3.11}$$

Please click the advert



Download free ebooks at bookboon.com

3.3.2 Kirchoff's Current Law (KCL)

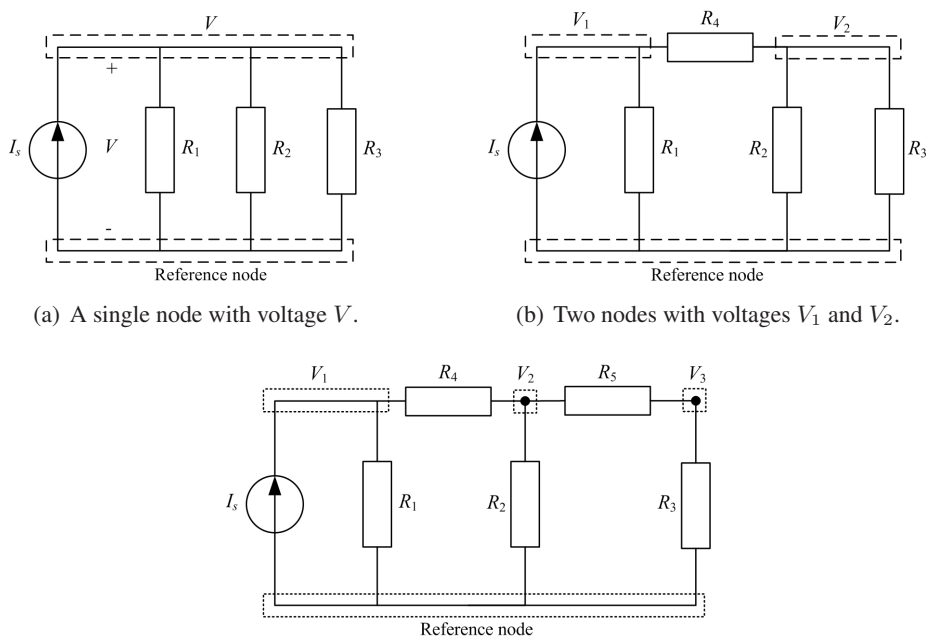
> "*The algebraic sum of all the currents entering or leaving a node in an electric circuit is equal to zero.*"

In other words, the sum of currents entering is equal to the sum of currents leaving the node in an electric circuit. Consider node B in Figure 3.1, then according to KCL

$$I_1 + I_4 = I_2 + I_3$$

Example

For the circuit diagrams depicted in Figure 3.4, write the nodal equations.



(a) A single node with voltage V.

(b) Two nodes with voltages V_1 and V_2.

(c) Three nodes with voltages V_1, V_2 and V_3.

Figure 3.4: Circuit diagrams to demonstrate the application of KCL in the above example.

Figure 3.4(a) contains just one node excluding the reference, hence one equation is required.

$$I_s = \frac{V}{R_1} + \frac{V}{R_2} + \frac{V}{R_3} \tag{3.12}$$

If I_s, R_1, R_2 and R_3 are know in Equation 3.12, V can be determined.

In Figure 3.4(b), two equations are written for the two nodes labelled V_1 and V_2.

Left node

$$\begin{aligned} I_s &= \frac{V_1}{R_1} + \frac{V_1 - V_2}{R_4} \\ I_s &= \left(\frac{1}{R_1} + \frac{1}{R_4}\right) V_1 - \frac{1}{R_4} V_2 \end{aligned} \tag{3.13}$$

Right node

$$\begin{aligned} 0 &= \frac{V_2 - V_1}{R_4} + \frac{V_2}{R_2} + \frac{V_2}{R_3} \\ 0 &= -\frac{1}{R_4} V_1 + \left(\frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4}\right) V_2 \end{aligned} \tag{3.14}$$

Equations 3.13 and 3.14 can be simultaneously solved to determine the node voltages provided the resistors' values and I_s are known.

Figure 3.4(c) contains 3 nodes hence three equations are required to solve for node voltages V_1, V_2 and V_3.

Left node

$$\begin{aligned} I_s &= \frac{V_1}{R_1} + \frac{V_1 - V_2}{R_4} \\ I_s &= \left(\frac{1}{R_1} + \frac{1}{R_4}\right) V_1 - \frac{1}{R_4} V_2 \end{aligned} \tag{3.15}$$

Middle node

$$\begin{aligned} 0 &= \frac{V_2 - V_1}{R_4} + \frac{V_2}{R_2} + \frac{V_2 - V_3}{R_5} \\ 0 &= -\frac{1}{R_4} V_1 + \left(\frac{1}{R_2} + \frac{1}{R_4} + \frac{1}{R_5}\right) V_2 - \frac{1}{R_5} V_3 \end{aligned} \tag{3.16}$$

Right node

$$\begin{aligned} 0 &= \frac{V_3 - V_2}{R_5} + \frac{V_3}{R_3} \\ 0 &= -\frac{1}{R_5} V_2 + \left(\frac{1}{R_3} + \frac{1}{R_5}\right) V_3 \end{aligned} \tag{3.17}$$

Node voltages V_1, V_2 and V_3 can be evaluated by simultaneously solving Equations 3.15, 3.16 and 3.17 using Cramer's rule.

Resistors in Parallel

Consider Figure 3.5 with a single current source and two resistors connected in parallel. All parallel circuit elements have the same voltage, V across them i.e. $V_1 = V_2 = V$

Download free ebooks at bookboon.com

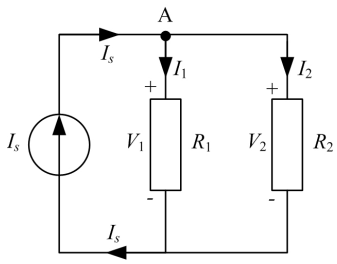


Figure 3.5: Parallel connection of resistors.

Applying KCL at node A

$$I_s = I_1 + I_2$$

Using Ohm's law $(I = \frac{V}{R})$

$$\frac{V}{R_{eq}} = \frac{V}{R_1} + \frac{V}{R_2}$$

$$\boxed{\frac{1}{R_{eq}} = \frac{1}{R_1} + \frac{1}{R_2}} \tag{3.18}$$

where R_{eq} is the equivalent resistance of the parallel[2] network of resistors. For n number of resistors connected in parallel, the combined resistance is given by

$$\boxed{\frac{1}{R_{eq}} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \cdots + \frac{1}{R_n}} \tag{3.19}$$

or in terms of conductance, G

$$\boxed{G_{eq} = G_1 + G_2 + G_3 + \cdots + G_n} \tag{3.20}$$

[2]To illustrate a parallel relationship between two or more resistors, the symbol $'||'$ is occasionally used.

Current Divider Rule (CDR)

Current divider rule provides a useful relationship for determining the current through individual circuit elements that are connected in *parallel*. For the circuit depicted in Figure 3.5, the current through each resistor can be evaluated using the following formulae

$$I_1 = I_s \frac{R_2}{R_1 + R_2} \tag{3.21}$$

$$I_2 = I_s \frac{R_1}{R_1 + R_2} \tag{3.22}$$

Notice the difference between VDR and CDR (for two resistors) in terms of the resistor value in the numerators. In order to generalise CDR for n number of resistors, the conductance parameter is used. Hence to find the current through ith of n resistors connected in parallel, the following relationship can be used

$$I_i = I_s \frac{G_i}{G_1 + G_2 + \cdots + G_i + \cdots + G_n} \tag{3.23}$$

The above equation has the same form as the generalised VDR with R replaced by G and voltages replaced by current variables.

3.4 Electric Circuits Analysis

The KVL, KCL and Ohm's law are the primary tools to analyse DC electric circuits. The term *nodal analysis* is generally used when analysing an electric circuit with KCL whereas loop or *mesh analysis* is designated for problem solving using KVL.

The mesh and nodal analysis methods outlined below are quite systematic and usually provides the solution to a given problem. However, they are fairly computational and an alternative straightforward solution may exist using circuit reduction techniques such as series/parallel combination of resistors and/or VDR/CDR methods.

3.4.1 Mesh Analysis

The mesh analysis technique consists of the following steps

1. Transform all currents sources to voltage sources, if possible (see Section 3.8).
2. Identify and assign a current to each mesh of the network (preferably in the same direction).
3. Write mesh equations using KVL and simplify them.
4. Solve the simultaneous system of equations.
5. Number of equations is equal to number of meshes in the network.

Download free ebooks at bookboon.com

3.4.2 Nodal Analysis

The following steps describe the nodal analysis method

1. Transform all voltage sources to current sources, if possible (see Section 3.8).
2. Identify and assign an arbitrary voltage to each node including the reference node in the network assuming all currents leaving the node. The reference node is normally assumed to be at zero potential.
3. Write nodal equations using KCL and simplify them.
4. Solve the simultaneous system of equations.
5. Number of equations is $N - 1$ where N is the number of nodes in the network including the reference node.

Example

Calculate the current supplied by the 30 V source and the current through each resistor in the circuit diagram shown in Figure 3.6 using (1) nodal analysis, (2) mesh analysis and (3) circuit reduction techniques, where $R_1 = R_2 = R_3 = R_4 = 10\ \Omega$.

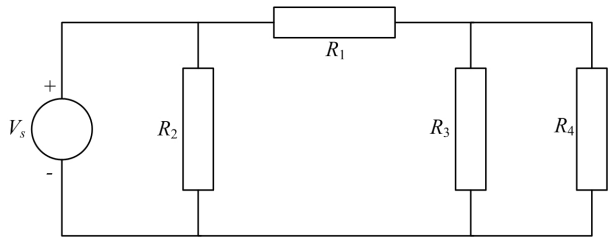


Figure 3.6

Solution 1. Nodal Analysis

Source transformation is not possible for this circuit since it requires a resistor in series with the voltage source (see Section 3.8). Three nodes are identified in the above circuit diagram and labelled as 0, 1 and 2 as illustrated in Figure 3.7 where 0 is the reference node. Specify V_1 and V_2 as voltages of nodes 1 and 2 respectively.

The voltage source, V_s and R_2 are in parallel therefore $V_1 = V_s = 30$ V is known by inspection.

Applying KCL at node 2 assuming all currents are leaving the node, therefore

$$\frac{V_2 - V_1}{R_1} + \frac{V_2}{R_3} + \frac{V_2}{R_4} = 0$$
$$\frac{V_2 - 30}{10} + \frac{V_2}{10} + \frac{V_2}{10} = 0$$
$$V_2 - 30 + V_2 + V_2 = 0$$
$$V_2 = 10 \text{ V}$$

The required currents can now be found using Ohm's law.

$$
\begin{aligned}
I_{R_1} &= \frac{V_1 - V_2}{R_1} = \frac{30-10}{10} = 2 \text{ A} \\
I_{R_2} &= \frac{V_1}{R_2} = \frac{30}{10} = 3 \text{ A} \\
I_{R_3} &= \frac{V_2}{R_3} = \frac{10}{10} = 1 \text{ A} \\
I_{R_4} &= \frac{V_2}{R_4} = \frac{10}{10} = 1 \text{ A}
\end{aligned}
$$

The current supplied by the voltage source, I_s is given by the sum of currents I_{R_1} and I_{R_2} i.e.

$$I_s = I_{R_1} + I_{R_2} = 2 + 3 = 5 \text{ A}$$

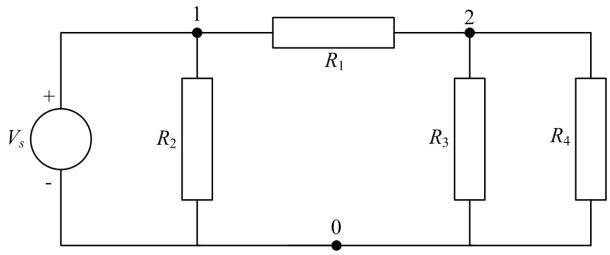
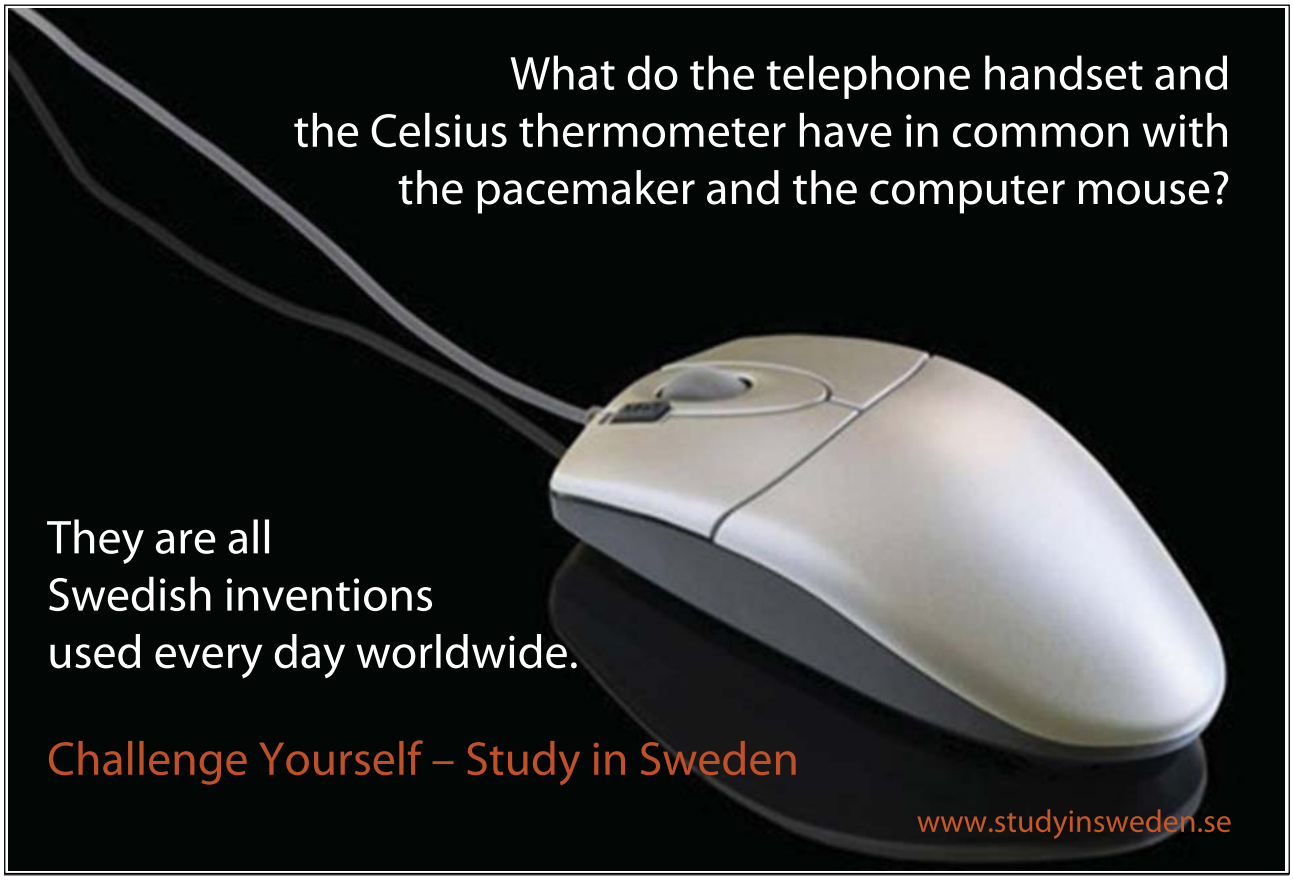


Figure 3.7: Circuit diagram showing three nodes labelled as 0, 1, 2 where 0 is the reference node.

Please click the advert



Download free ebooks at bookboon.com

Solution 2. Mesh Analysis

Three meshes can be identified with currents I_1, I_2 and I_3 as illustrated in Figure 3.8 where all the currents are assumed to be flowing in the clockwise direction.

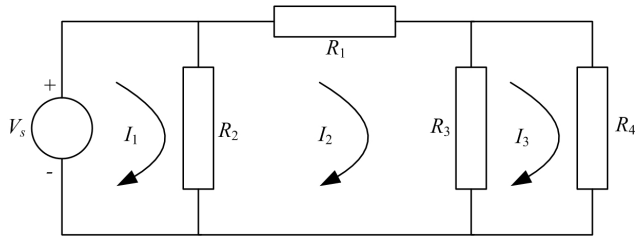


Figure 3.8: For mesh analysis, three loops are highlighted with currents I_1, I_2 and I_3.

Observe the common (shared) circuit elements between the meshes such as R_2 (between meshes 1 and 2) and R_3 (between meshes 2 and 3). The current through R_2, for instance, will be $I_1 - I_2$ when considering mesh 1 whilst it will be $I_2 - I_1$ for mesh 2.

The KVL equations can now be written as follows

$$\begin{aligned}(I_1 - I_2)R_2 &= V_s \\ (I_2 - I_1)R_2 + I_2R_1 + (I_2 - I_3)R_3 &= 0 \\ (I_3 - I_2)R_3 + I_3R_4 &= 0\end{aligned}$$

Substitute the given values of the resistors and voltage source,

$$I_1 - I_2 = 3 \tag{3.24}$$

$$I_1 - 3I_2 + I_3 = 0 \tag{3.25}$$

$$I_2 - 2I_3 = 0 \tag{3.26}$$

The solution of the above simultaneous equations can be obtained either by substitution or by Cramer's rule or matrix method. Herein, the Cramer's rule explained in Appendix A is employed.

$$I_1 = \frac{\begin{vmatrix} 3 & -1 & 0 \\ 0 & -3 & 1 \\ 0 & 1 & -2 \end{vmatrix}}{\begin{vmatrix} 1 & -1 & 0 \\ 1 & -3 & 1 \\ 0 & 1 & -2 \end{vmatrix}} = 5 \text{ A}, \qquad I_2 = \frac{\begin{vmatrix} 1 & 3 & 0 \\ 1 & 0 & 1 \\ 0 & 0 & -2 \end{vmatrix}}{\begin{vmatrix} 1 & -1 & 0 \\ 1 & -3 & 1 \\ 0 & 1 & -2 \end{vmatrix}} = 2 \text{ A}, \qquad I_3 = \frac{\begin{vmatrix} 1 & -1 & 3 \\ 1 & -3 & 0 \\ 0 & 1 & 0 \end{vmatrix}}{\begin{vmatrix} 1 & -1 & 0 \\ 1 & -3 & 1 \\ 0 & 1 & -2 \end{vmatrix}} = 1 \text{ A}$$

Download free ebooks at bookboon.com

$$\begin{aligned}
\therefore I_s = I_1 &= 5 \text{ A}, \\
I_{R_1} = I_2 &= 2 \text{ A}, \\
I_{R_4} = I_3 &= 1 \text{ A}, \\
I_{R_2} = I_1 - I_2 &= 3 \text{ A and} \\
I_{R_3} = I_2 - I_3 &= 1 \text{ A}
\end{aligned}$$

Solution 3. Circuit Reduction Techniques

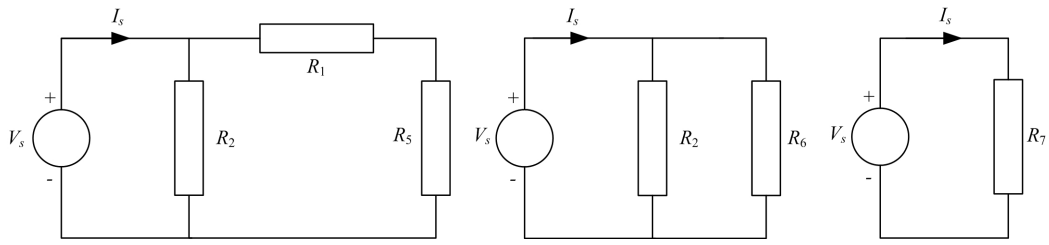
Circuit reduction techniques include series/parallel combination and VDR/CDR formulae. From the schematic diagram in Figure 3.6, it is clear that R_3 and R_4 are in parallel, therefore they can be combined and reduced to a single resistor, R_5 using Equation 3.18. Hence

$$R_5 = \frac{R_3 R_4}{R_3 + R_4} = \frac{10 \times 10}{10 + 10} = 5\ \Omega$$

R_5 is in series with R_1, therefore the combined resistance R_6 can be calculated by Equation 3.7 which is a simple algebraic sum i.e.

$$R_6 = R_1 + R_5 = 10 + 5 = 15\ \Omega$$

The equivalent resistance, R_6, is now in parallel with R_2 giving a single resistance, R_7 of 6 Ω. The process at each step is depicted in Figure 3.9.



(a) $R_5 = R_3||R_4$ (b) $R_6 = R_1 + R_5$ (c) $R_7 = R_2||R_6$

Figure 3.9: Circuit reduction using series/parallel combination of resistors.

From Figure 3.9(c), the current, I_s, supplied by the voltage source can now be calculated using Ohm's law

$$I_s = \frac{V_s}{R_7} = \frac{30}{6} = 5 \text{ A}$$

In Figure 3.9(b), current through R_2 can be found by applying CDR between R_2 and R_6 i.e.

$$\begin{aligned}
I_{R_2} &= I_s \frac{R_6}{R_2 + R_6} = 5 \frac{15}{10 + 15} = 3 \text{ A} \\
\therefore I_{R_6} &= I_s - I_{R_2} = 2 \text{ A}
\end{aligned}$$

Since R_6 is a series combination of R_1 and R_5, therefore

$$I_{R_1} = I_{R_5} = 2 \text{ A}$$

The current I_{R_5} is the sum of currents through the parallel combination of R_3 and R_4. Hence CDR can be applied to determine the currents through each resistor. Since $R_3 = R_4$, therefore the currents I_{R_3} and I_{R_4} are

$$I_{R_3} = I_{R_4} = 1 \text{ A}$$

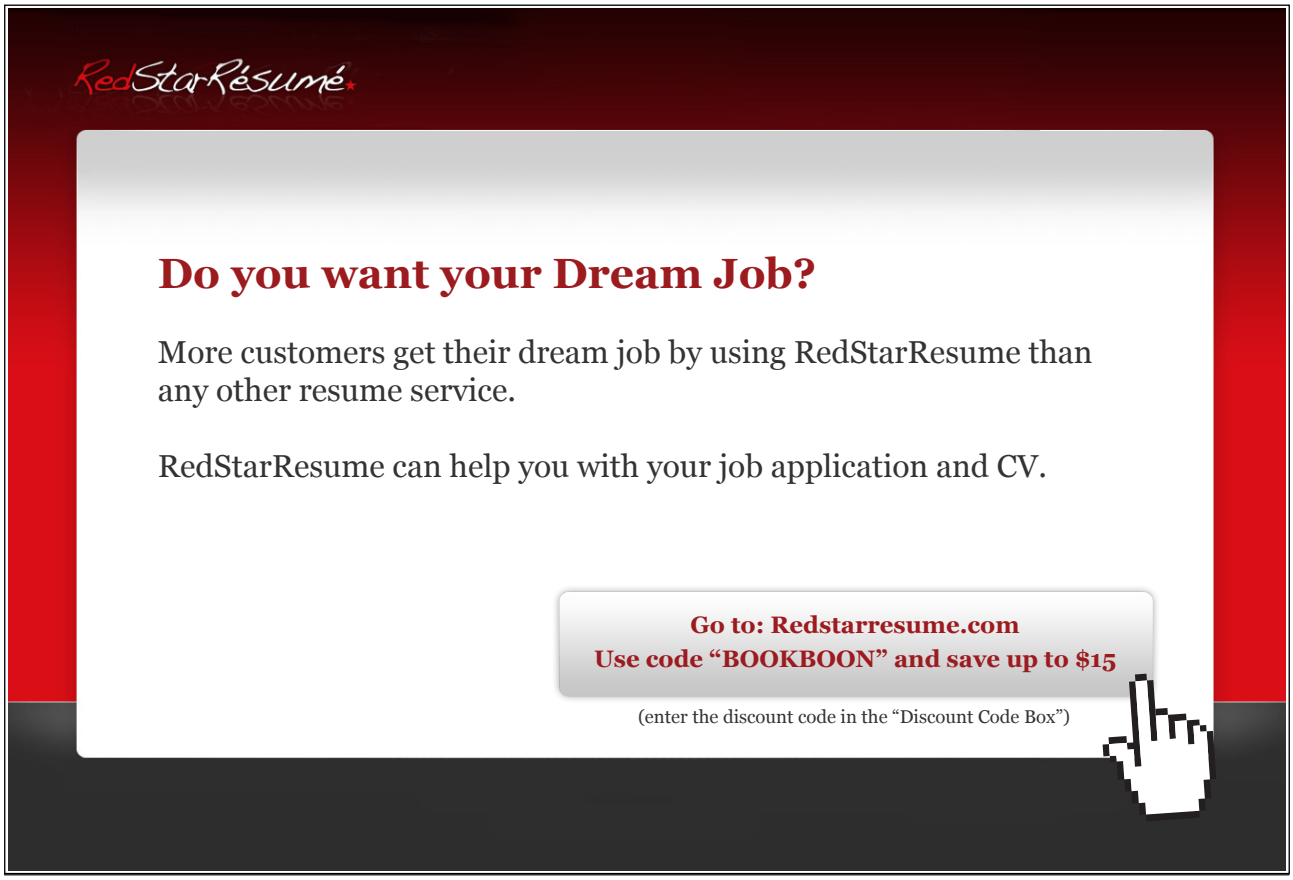
3.5 Superposition Theorem

Superposition theorem is extremely useful for analysing electric circuits that contains two or more active sources. In such cases, the theorem considers each source separately to evaluate the current through or voltage across a component. The resultant is given by the algebraic sum of all currents or voltages caused by each source acting independently. Superposition theorem can be formally stated as follows

> "*The current through or voltage across any element in a linear circuit containing several sources is the algebraic sum of the currents or voltages due to each source acting alone, all other sources being removed at that time.*"

Linearity is a necessary condition for the theorem to apply. Fortunately, the v, i relationship for R, L and C are all linear.

Please click the advert



Download free ebooks at bookboon.com

The sources can be removed using the following methodology,

1. Ideal voltage sources are short-circuited
2. Ideal current sources are open-circuited

In general, practical sources are replaced by their internal resistances.

Example

Find the voltage V_L using Superposition theorem in the circuit diagram of Figure 3.10.

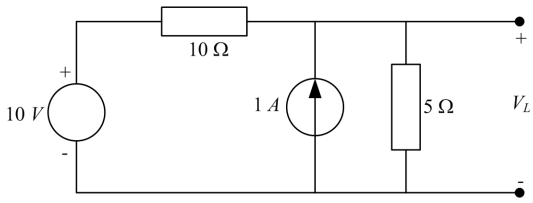
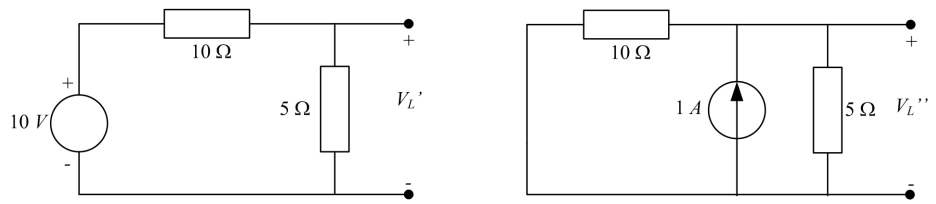


Figure 3.10: An electric circuit containing multiple sources.

Step 1: Suppressing the 1 A current source by replacing it with an open circuit

This will result in the circuit diagram of Figure 3.11(a). The output voltage, now called $V_L^{'}$, is the voltage across the 5 Ω resistance. Since both 10 Ω and 5 Ω are in series, VDR can be applied here

$$\therefore V_L^{'} = 10\frac{5}{10+5} = 3.33 \text{ V}$$



(a) Superposition (step 1) - suppressing the current source.

(b) Superposition (step 2) - suppressing the voltage source.

Figure 3.11: Application of superposition theorem to Figure 3.10.

Step 2: Suppressing the 10 V voltage source by replacing it with a short circuit

The resulting circuit diagram is shown in Figure 3.11(b). The output voltage is now represented by $V_L^{''}$ and has the same polarity as V_L. It is obvious that both the resistors are in parallel and therefore CDR can be applied to determine the current flowing through 5 Ω resistor.

$$\begin{aligned} I_{5\Omega} &= 1\frac{10}{10+5} &= 0.67 \text{ A} \\ \therefore V_L^{''} &= I_{5\Omega} \times 5 &= 3.33 \text{ V} \end{aligned}$$

Download free ebooks at bookboon.com

The total voltage, according to the superposition theorem is then given by the sum of $V_L^{'}$ and $V_L^{''}$ i.e.

$$V_L = V_L^{'} + V_L^{''} = 3.33 + 3.33 = 6.66 \text{ V}$$

Superposition Theorem - Power Calculation

Although the superposition theorem can be used to find the voltage across and current through a circuit element by adding the responses due to each source acting alone, the power (as a nonlinear relationship) cannot be evaluated until the net voltage or current is found. This is because

$$\begin{aligned} P &= (I_1 + I_2)^2 R \neq I_1^2 R + I_2^2 R \\ \text{or} \quad P &= \frac{(V_1 + V_2)^2}{R} \neq \frac{V_1^2}{R} + \frac{V_2^2}{R} \end{aligned}$$

3.6 Thévenin's Theorem

Thévenin's theorem provides a useful tool when solving complex and large electric circuits by reducing them to a single voltage source in series with a resistor. It is particularly advantageous where a single resistor or load in a circuit is subject to change.

Formally, the Thévenin's theorem can be stated as

> "*Any two-terminal linear electric circuit consisting of resistors and sources, can be replaced by an equivalent circuit containing a single voltage source in series with a resistor connected across the load.*"

In the circuit diagrams shown in Figure 3.12, the current I_L through the load resistance R_L is the same. Hence the circuits are equivalent as far as the load resistor R_L is concerned.

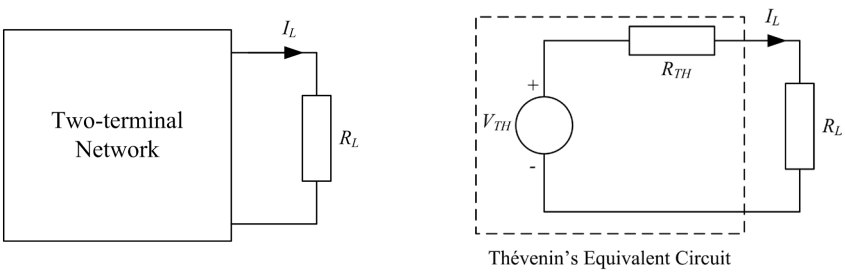


Figure 3.12: Illustration of Thévenin's theorem.

The following steps outline the procedure to simplify an electric circuit using Thévenin's theorem where V_{TH} and R_{TH} are the Thévenin's voltage and Thévenin's resistance respectively.

1. Remove the load resistance R_L.
2. V_{TH} is the open circuit (OC) voltage across the load terminals and

3. R_{TH} is the resistance across the load terminals with all sources replaced by their internal resistances.

Alternatively, measure the OC voltage across, and the short circuit (SC) current through the load terminals. Then

$$V_{TH} = V_{oc} \text{ and } R_{TH} = \frac{V_{oc}}{I_{sc}} \tag{3.27}$$

Example

Use Thévenins theorem to find the current through the 5 Ω resistance in the circuit diagram of Figure 3.13.

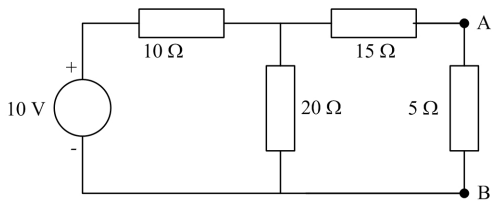
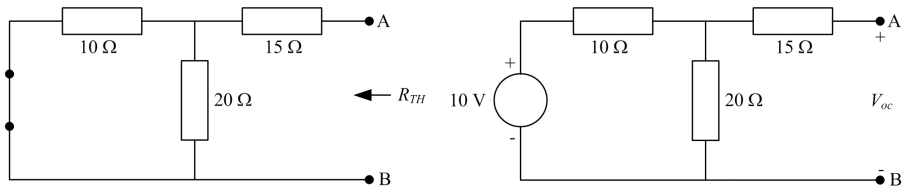


Figure 3.13

To evaluate R_{TH}, remove the load resistance of 5 Ω and replace the 10 V voltage source by its internal resistance as depicted in Figure 3.14(a).



(a) Step 1: Determining the Thévenin's resistance.

(b) Thévenin's voltage calculation.

Figure 3.14: Determination of Thévenin's voltage, V_{TH} and Thévenin's resistance, R_{TH}.

R_{TH} is then given by

$$R_{TH} = (10||20) + 15 = \frac{10 \times 20}{10 + 20} + 15 = 21.67\ \Omega$$

To determine V_{TH} or V_{oc} across the OC load terminals, note in Figure 3.14(b) that the voltage across the 20 Ω resistance is the same as V_{oc} since the right loop is open and no voltage drop occurs across the 15 Ω resistance ($I_{15\Omega} = 0$).

The 10 Ω and 20 Ω resistors are in series therefore VDR can be employed to determine V_{oc}

$$V_{TH} = V_{oc} = 10\frac{20}{10 + 20} = 6.67 \text{ V}$$

Download free ebooks at bookboon.com

The Thévenin's equivalent circuit can now be drawn as shown in Figure 3.15. The load resistance of 5 Ω is inserted back between the terminals A and B and the load current can be found as follows

$$I_{5\Omega} = \frac{V_{TH}}{R_{TH} + R_L} = \frac{6.67}{21.67 + 5} = 0.25 \text{ A}$$

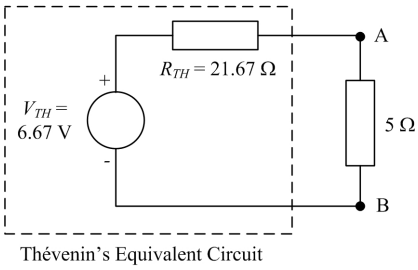


Figure 3.15: Thévenin's equivalent circuit for Figure 3.13 across the 5 Ω load resistance.

Please click the advert

Download free ebooks at bookboon.com

3.7 Norton's Theorem

Thévenin's equivalent circuit is a practical voltage source. In contrast, Norton's equivalent circuit is a practical current source. This can be formally stated as

> "*Any two-terminal, linear circuit, of resistors and sources, can be replaced by a single current source in parallel with a resistor.*"

To determine Norton's equivalent circuit, Norton current, I_N, and Norton resistance, R_N, are required. The following steps outline the procedure required.

1. Remove the load resistance, R_L.
2. I_N is the SC current through the load terminals and
3. R_N is the resistance across the load terminals with all sources replaced by their internal resistances. Clearly $R_N = R_{TH}$.

Example

For the circuit diagram depicted in Figure 3.13, use Norton's theorem to determine the current through the $5\ \Omega$ resistance.

As mentioned above, The Norton's resistance is the same as the Thévenin's resistance, i.e.

$$R_N = 21.67\ \Omega$$

To find Norton's current, the terminals A and B in Figure 3.14(b) are short circuited. Then the current through the short circuited terminals is the Norton current, I_N or I_{sc}.

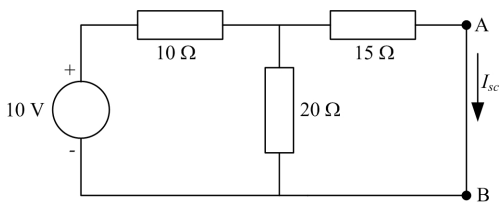


Figure 3.16

There are a number of ways to solve the above circuit such as KVL, KCL, however, circuit reduction techniques have been chosen here.

Firstly, the total current supplied by the voltage source can be found by adding the resistances using series/parallel combination.

$$R_{eq} = (15||20) + 10 = \frac{15 \times 20}{15 + 20} + 10 = 18.57\ \Omega$$

$$\therefore I_s = \frac{V_s}{R_{eq}} = \frac{10}{18.57} = 0.54\ \text{A}$$

Download free ebooks at bookboon.com

Since $I_s = I_{10\Omega},\ I_{sc}$ can be found by employing CDR between the 15 Ω and 20 Ω resistances i.e.

$$I_{sc} = I_s \frac{20}{15+20} = 0.54\frac{20}{35} = 0.31 \text{ A}$$

The Norton's equivalent circuit can be drawn as shown in Figure 3.17.

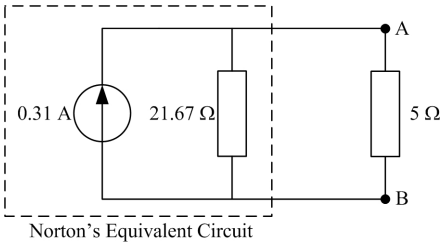


Figure 3.17: Norton's equivalent circuit for Figure 3.13 across the 5 Ω load.

The current through the 5 Ω resistance can be determined by the application of CDR between R_L and R_N, thus

$$\boxed{I_{5\Omega} = 0.31\frac{21.67}{21.67+5} = 0.25 \text{ A}}$$

which also tallies with the result when Thévenin's theorem was employed.

3.8 Source Transformation

In an electric circuit, it is often convenient to have a voltage source rather than a current source (e.g. in mesh analysis) or vice versa. This is made possible using source transformations. It should be noted that only practical voltage and current sources can be transformed. In other words, a Thévenin's equivalent circuit is transformed into a Norton's one or vice versa. The parameters used in the source transformation are given as follows.

Thévenin parameters: $V_{TH}, R_{TH} \implies R_N = R_{TH}, I_N = \frac{V_{TH}}{R_{TH}}$

Norton parameters: $I_N, R_N \implies R_{TH} = R_N, V_{TH} = R_N I_N$

Any load resistance, R_L will have the same voltage across, and current through it when connected across the terminals of either source.

Example

In Figure 3.13, use repeated source transformation to determine the current through the 5 Ω resistance.

Noting that the voltage source and the 10 Ω resistor are in series, transform the combination into a current source in parallel with the 10 Ω resistor. The magnitude of the current source is calculated as follows

$$I = \frac{V}{R} = \frac{10}{10} = 1 \text{ A}$$

The resulting schematic is depicted in Figure 3.18(a). Now the 10 Ω and 20 Ω resistors are in parallel which can be combined together to form a single resistance of 6.67 Ω (see Figure 3.18(b)). This resistance will be in parallel with the 1 A current source that can be transformed back to a voltage source in series with the 6.67 Ω resistance resulting in single loop as shown in Figure 3.18(c). KVL can be applied to determine the loop current which is the same current flowing through the 5 Ω resistor. Hence

$$I_{5\Omega} = \frac{6.67}{6.67 + 15 + 5} = 0.25 \text{ A}$$

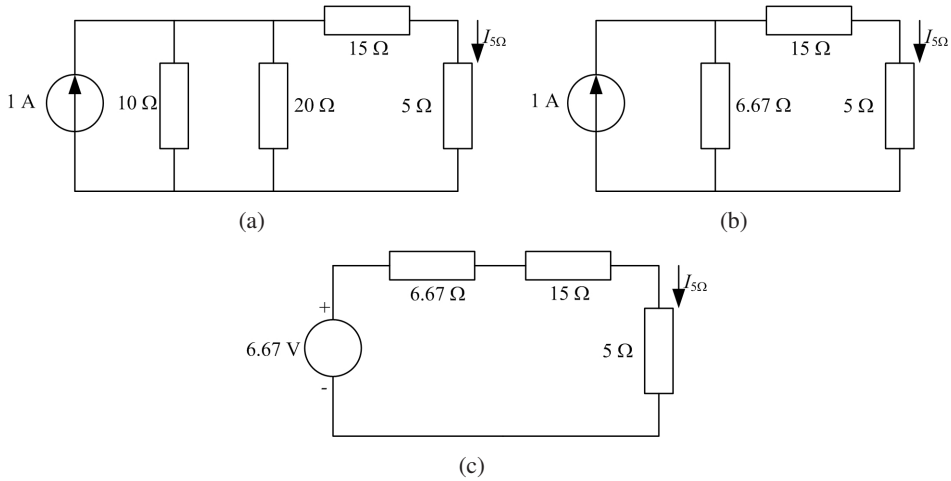


Figure 3.18: Repeated source transformation of Figure 3.18

Please click the advert

Trust and responsibility

NNE and Pharmaplan have joined forces to create NNE Pharmaplan, the world's leading engineering and consultancy company focused entirely on the pharma and biotech industries.

Inés Aréizaga Esteva (Spain), 25 years old
Education: Chemical Engineer

– You have to be proactive and open-minded as a newcomer and make it clear to your colleagues what you are able to cope. The pharmaceutical field is new to me. But busy as they are, most of my colleagues find the time to teach me, and they also trust me. Even though it was a bit hard at first, I can feel over time that I am beginning to be taken seriously and that my contribution is appreciated.

NNE Pharmaplan is the world's leading engineering and consultancy company focused entirely on the pharma and biotech industries. We employ more than 1500 people worldwide and offer global reach and local knowledge along with our all-encompassing list of services. **nnepharmaplan.com**

nne pharmaplan®

Download free ebooks at bookboon.com

3.9 Maximum Power Transfer Theorem

As discussed in the section on Thévenin's theorem, any DC network of sources and resistances can be replaced by a single voltage source in series with a resistance connected across the load (see Figure 3.12). The maximum power transfer theorem states that the power delivered to the load is maximum when the load resistance, R_L is equal to the internal (source) resistance, R_s of the DC power supply. In other words, it can be said that the load resistance must match the Thévenin's resistance for maximum power transfer to take place i.e.,

$$(R_s = R_{TH}) = R_L$$

When this occurs, the voltage across the load resistance will be $\frac{V_s}{2}$ and the power delivered to the load is given by

$$\begin{aligned} P &= V_L I_L \\ &= I_L^2 R_L \\ P &= \frac{V_s^2}{(R_s + R_L)^2} R_L \end{aligned}$$

The above equation is plotted in Figure 3.19 which clearly demonstrates maximum power delivered when $R_s = R_L$. Under this condition, the maximum power will be

$$\boxed{P_{max} = \frac{V_s^2}{4R_s}} \tag{3.28}$$

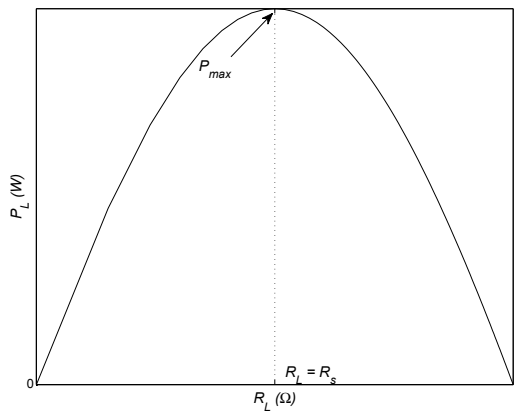


Figure 3.19: Illustration of maximum power transfer theorem.

3.10 Additional Common Circuit Configurations

In addition to the examples described in the previous sections, there are certain situations where direct application of the circuit rules may not be possible. In this section, two such configurations and their solutions are described.

Download free ebooks at bookboon.com

3.10.1 Supernode

A *supernode* exists when an ideal voltage source appears between any two nodes of an electric circuit. The usual way to solve this is to write KCL equations for both nodes and simply add them together into one equation ignoring the voltage source in question. However, this would mean one less equation than the number of variables (node voltages) present in the circuit. A contsraint equation can be easily specified given by the magnitude of the ideal voltage source present between the nodes and the respective node voltages. The following example will help clarify this scenario.

Example

Determine the voltage across the 1 A current source in the circuit diagram of Figure 3.20 using nodal analysis.

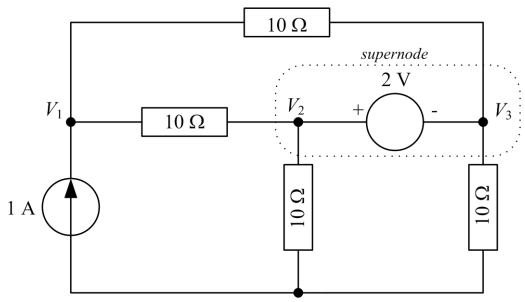


Figure 3.20: Illustration of a supernode.

A simple inspection of the circuit diagram reveals the existence of three nodes excluding the reference and are labelled as V_1, V_2 and V_3 as their respective node voltages. The presence of an ideal voltage source between nodes 2 and 3 creates a *supernode* and a constraint equation is necessary.

As mentioned, nodal analysis is carried out as normal and the nodal equations of nodes 2 and 3 will be added together.

Node 1

$$1 = \frac{V_1 - V_2}{10} + \frac{V_1 - V_3}{10}$$

$$10 = 2V_1 - V_2 - V_3 \tag{3.29}$$

Supernode

$$0 = \left(\frac{V_2 - V_1}{10} + \frac{V_2}{10}\right) + \left(\frac{V_3}{10} + \frac{V_3 - V_1}{10}\right)$$

$$0 = -V_1 + V_2 + V_3 \tag{3.30}$$

Since there are three variables and two equations, therefore another equation is needed. This can be written as the constraint equation between nodes 2 and 3 as follows

Download free ebooks at bookboon.com

Constraint Equation

$$V_2 - V_3 = 2 \tag{3.31}$$

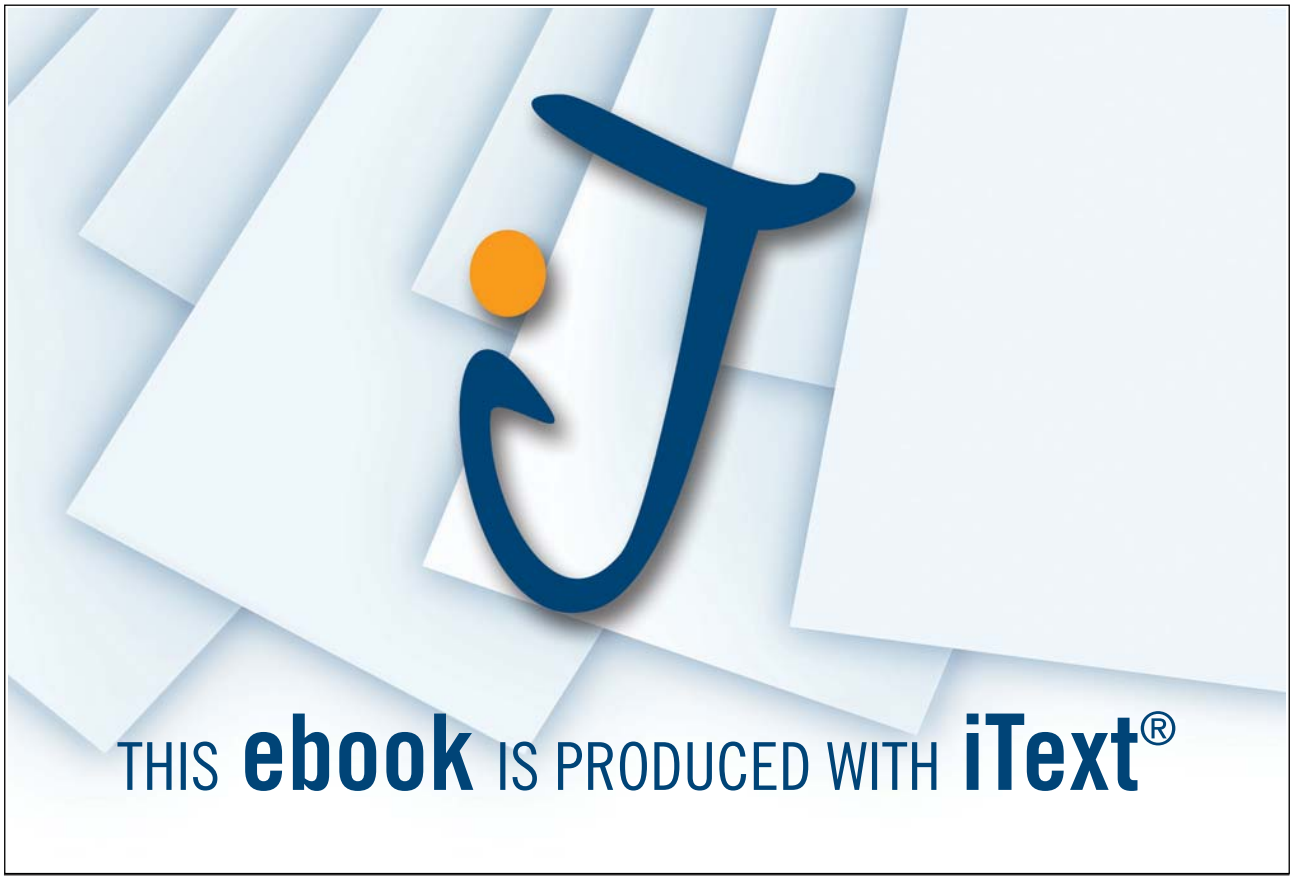
Equations 3.29, 3.30 and 3.31 can be solved simultaneously to determine the node voltages V_1, V_2 and V_3. Herein, the voltage, V_1, across the current source is required only. Hence by using Cramer's rule

$$V_1 = \frac{\Delta_1}{\Delta} = \frac{\begin{vmatrix} 10 & -1 & -1 \\ 0 & 1 & 1 \\ 2 & 1 & -1 \end{vmatrix}}{\begin{vmatrix} 2 & -1 & -1 \\ -1 & 1 & 1 \\ 0 & 1 & -1 \end{vmatrix}} = 10 \text{ V},$$

3.10.2 Supermesh

A *supermesh* exists when an ideal current source appears between two meshes of an electric circuit. In such a situation, like supernode, mesh equations are written for the meshes involved and added giving a single equation. Again, there would be one less equation than the number of variables (mesh currents) and hence a constraint equation is needed. This would be based on the magnitude of the ideal current source present between the two meshes and their mesh currents. The following example will help clarify this scenario.

Please click the advert



Download free ebooks at bookboon.com

Example

Determine the current supplied by the 10 V voltage source in the circuit diagram of Figure 3.21.

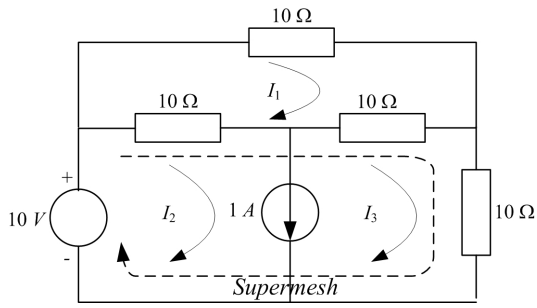


Figure 3.21: Illustration of a supermesh.

Mesh 1

$$\begin{aligned} 10I_1 + 10(I_1 - I_2) + 10(I_1 - I_3) &= 0 \\ 3I_1 - I_2 - I_3 &= 0 \end{aligned} \tag{3.32}$$

Supermesh

$$\begin{aligned} [10(I_2 - I_1)] + [10(I_3 - I_1) + 10I_3] &= 10 \\ -2I_1 + I_2 + 2I_3 &= 1 \end{aligned} \tag{3.33}$$

Constraint Equation

$$I_2 - I_3 = 1 \tag{3.34}$$

The current supplied by the 10 V source is I_2, thus according to Cramer's rule

$$I_2 = \frac{\Delta_2}{\Delta} = \frac{\begin{vmatrix} 3 & 0 & -1 \\ -2 & 1 & 2 \\ 0 & 1 & -1 \end{vmatrix}}{\begin{vmatrix} 3 & -1 & -1 \\ -2 & 1 & 2 \\ 0 & 1 & -1 \end{vmatrix}} = 1 \text{ A}$$

3.11 Mesh and Nodal Analysis by Inspection

It may seem rather cumbersome and demanding (especially for the beginners) to write correct nodal and mesh equations using the methods outlined in Sections 3.4.1 and 3.4.2. Although it is vital that students have a clear understanding of the underlying concepts, nonetheless there are methods devised to write nodal and mesh equations by inspection using *ad hoc* relationships. This provides a useful way to quickly analyse a given circuit to determine a particular current or voltage.

Download free ebooks at bookboon.com

3.11.1 Mesh Analysis

To demonstrate writing mesh equations by inspection, consider Figure 3.2(c) which consists of three meshes with currents denoted as I_1, I_2 and I_3. The mesh equations have been derived earlier in Equations 3.4, 3.5 and 3.6 and reproduced below for convenience.

$$V_s = (R_1 + R_2)I_1 - R_2 I_2 - R_1 I_3 \tag{3.35}$$
$$0 = -R_2 I_1 + (R_2 + R_3)I_2 \tag{3.36}$$
$$0 = -R_1 I_1 + (R_1 + R_4)I_3 \tag{3.37}$$

In matrix form, the above equations can be written as

$$\begin{bmatrix} V_s \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} R_1 + R_2 & -R_2 & -R_1 \\ -R_2 & R_2 + R_3 & 0 \\ -R_1 & 0 & R_1 + R_4 \end{bmatrix} \begin{bmatrix} I_1 \\ I_2 \\ I_3 \end{bmatrix} \tag{3.38}$$

The order of the resistance matrix ($\mathbf{R}$) is the same as number of loops in the circuit. Additionally it has a general structure as outlined below which is independent of the number of equations or meshes.

R_{ii} = i^{th} diagonal entry
= sum of all resistances in the ith mesh.

R_{ij} = off-diagonal entry
$-$[sum of resistances common to meshes i and j].

$\mathbf{R} = \mathbf{R}^T$ i.e. $\mathbf{R}$ is a symmetric matrix.

3.11.2 Nodal Analysis

The circuit diagram in Figure 3.4(c) can be used to demonstrate the technique to write nodal equations by inspection. The equations for this circuit have been derived earlier and reproduced below for reader's convenience.

$$\begin{aligned} I_s &= \left(\frac{1}{R_1} + \frac{1}{R_4}\right) V_1 - \frac{1}{R_4} V_2 \\ 0 &= -\frac{1}{R_4} V_1 + \left(\frac{1}{R_2} + \frac{1}{R_4} + \frac{1}{R_5}\right) V_2 - \frac{1}{R_5} V_3 \\ 0 &= -\frac{1}{R_5} V_2 + \left(\frac{1}{R_3} + \frac{1}{R_5}\right) V_3 \end{aligned}$$

Download free ebooks at bookboon.com

Noting that $\frac{1}{R} = G$ (conductance), the above equations can be rewritten as follows

$$I_s = (G_1 + G_4)\, V_1 - G_4 V_2 \tag{3.39}$$

$$0 = -G_4 V_1 + (G_2 + G_4 + G_5)\, V_2 - G_5 V_3 \tag{3.40}$$

$$0 = -G_5 V_2 + (G_3 + G_5)\, V_3 \tag{3.41}$$

In matrix form

$$\begin{bmatrix} I_s \\ 0 \\ 0 \end{bmatrix} = \begin{bmatrix} G_1 + G_4 & -G_4 & 0 \\ -G_4 & G_2 + G_4 + G_5 & -G_5 \\ 0 & -G_5 & G_3 + G_5 \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \end{bmatrix} \tag{3.42}$$

The conductance matrix $(\mathbf{G})$ has the same order as the number of nodes excluding the reference node. Moreover, it has a general structure as outline below which is independent of the number of nodes in the network.

G_{ii} = i^{th} diagonal entry
= sum of all conductances directly connected to i^{th} node.

G_{ij} = off-diagonal entry
$-$[sum of conductances connected between nodes i and j].

$\mathbf{G} = \mathbf{G}^T$ i.e. $\mathbf{G}$ is a symmetric matrix.

Please click the advert

Download free ebooks at bookboon.com

Chapter 4

Sinusoids and Phasors

4.1 Introduction

In Chapter 3, circuit analysis was carried out with DC excitation voltage or current. Herein, the response to same type of RLC circuits is analysed using AC sources. Phasors are introduced as well as the concept of impedance of an electric circuit. The chapter concludes with power relationships in AC circuits and its related theory.

The concept of sinusoidal current and voltage is first introduced as this forms the basis of AC circuit analysis.

4.2 Sinusoids

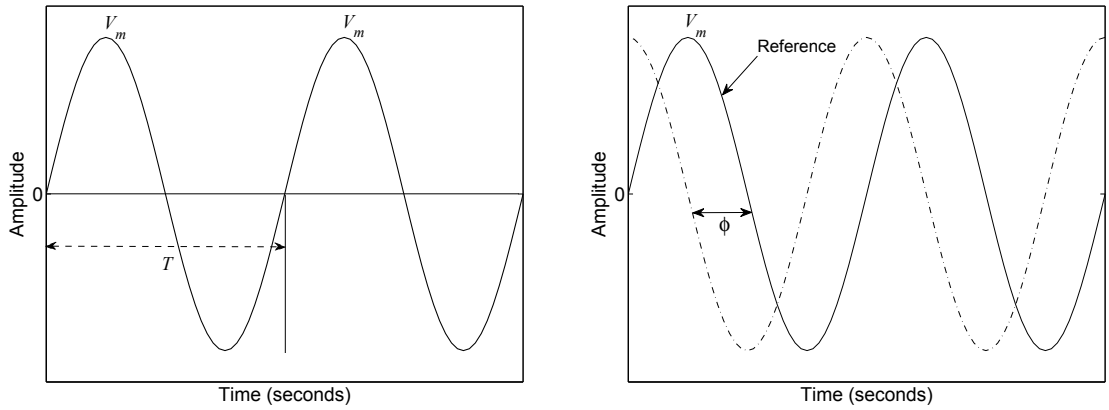
In circuit analysis, the term AC generally refers to a sinusoidal current or voltage signal. Sinusoids or more commonly sine waves are arguably the most important class of waveforms in electrical engineering. A common example is the mains voltage which is sinusoidal in nature. In general, any periodic waveform can be generated using a combination of sinusoidal signals of varying frequencies and amplitudes. The two most important parameters of a sine wave are its amplitude, V_m and frequency, f or time period, T.

Figure 4.1(a) shows a sinusoid and related parameters V_m and T. The frequency is then given by the reciprocal of the time period i.e. $f = \frac{1}{T}$. Note that two complete cycles (time periods) are plotted which clearly shows the periodic nature of a sine wave.

Peak to peak amplitude is also occasionally used given as two times the peak amplitude. Another parameter that is vital for AC analysis is the phase, ϕ of the sine wave. This is normally measured with respect to a reference waveform as shown in Figure 4.1(b). It is important to point out that sinusoidal generally refers to either sine or cosine waveform where the cosine wave has the same shape but is 90^0 out of phase with respect to the sine wave.

Mathematically, a sinusoidal signal is given by the general form

Download free ebooks at bookboon.com



(a) Sinusoidal waveform and related parameters. (b) Phase shift between two sine waves.

Figure 4.1: Parameters of a sine wave (a) Amplitude and time period, (b) Phase

$$v(t) = V_m \sin(\omega t + \phi) \tag{4.1}$$

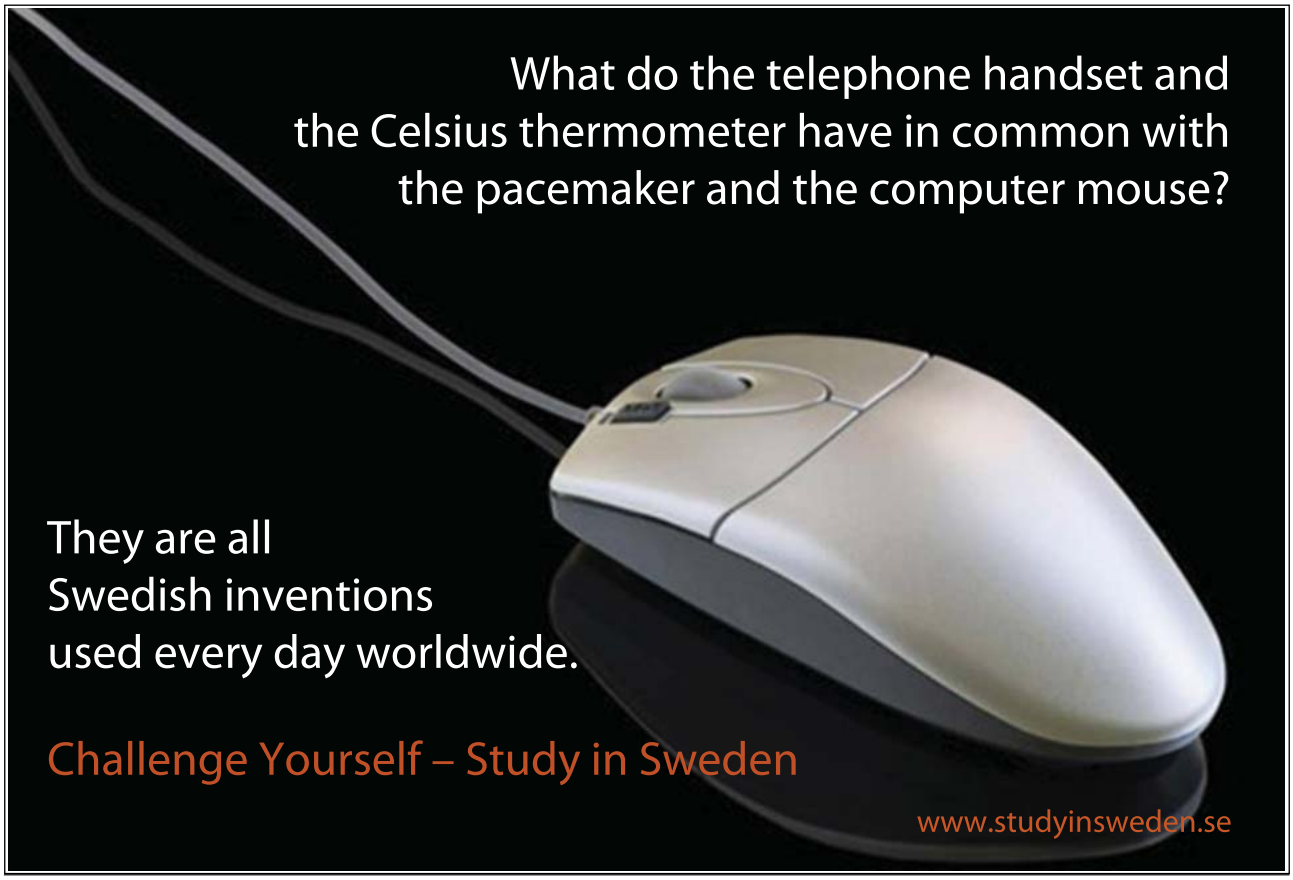
where

$$\begin{aligned} &V_m && \rightarrow \text{ amplitude or peak value} \\ &\omega = 2\pi f = \frac{2\pi}{T} = && \rightarrow \text{ angular frequency in rad/s} \\ &\phi && \rightarrow \text{ phase angle in degrees or radians} \end{aligned}$$

when $\phi = 90^0$, $\sin(\omega t + 90^0) = \cos(\omega t)$[1].

[1]use the trigonometric identity $\sin(\alpha + \beta) = \sin\alpha\cos\beta + \cos\alpha\sin\beta$

Please click the advert



Download free ebooks at bookboon.com

4.2.1 Other Sinusoidal Parameters

Mean or Average Value

Given a discrete sequence of signals such as $1, 2, 3, 4, 5$ the average value is calculated as follows

$$\text{Average Value} = \frac{1+2+3+4+5}{5} = 3$$

For a continuous periodic waveform such as a sinusoid, the mean value can be found by averaging all the instantaneous values during one cycle. This is given by

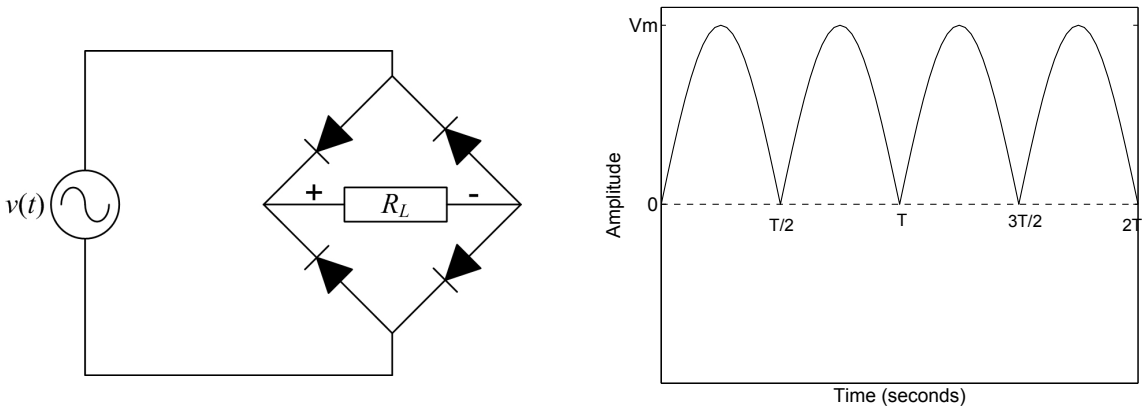
$$V_{av} = \frac{1}{T}\int_0^T v(t)d(t) \tag{4.2}$$

Clearly, the average value of a complete sine wave is 0 because of equal positive and negative half cycles. This is regardless of the peak amplitude.

Average value of a fully rectified sine wave By taking the absolute value of a sine wave, a fully rectified waveform can be generated. In practise, this can be achieved by employing the bridge circuit shown in Figure 4.2(a) with sine wave as the excitation signal. The output waveform across the load, R_L is depicted in Figure 4.2(b) with a time period of $T/2$ where T is the time period of a normal sine wave. Thus

$$V_{av} = \frac{2}{T}\int_0^{T/2} v(t)d(t)$$

$$\boxed{V_{av} = \frac{2V_m}{\pi}} \tag{4.3}$$



(a) Full wave rectification circuit (commonly known as a bridge rectifier).

(b) Fully rectified sine waveform.

Figure 4.2: A bridge rectifier and the resulting fully rectified sine waveform

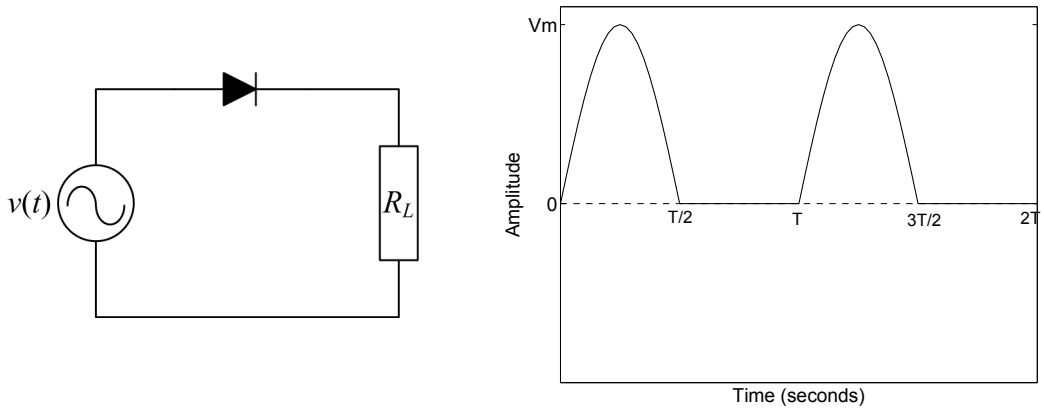
A bridge rectifier circuit is common in DC power supplies and is mainly employed to convert an AC input signal to a DC output.

Download free ebooks at bookboon.com

Average value of a half rectified sine wave A half wave rectification circuit is depicted in Figure 4.3(a) and the resulting waveform is also shown in Figure 4.3(b). In this case, the negative half cycle of the sine wave is clipped to zero and hence the time period T of the waveform remains the same. Therefore,

$$V_{av} = \frac{1}{T}\int_0^{T/2} v(t)d(t) + \frac{1}{T}\int_{T/2}^{T} 0 \cdot dt$$

$$\boxed{V_{av} = \frac{V_m}{\pi}} \tag{4.4}$$



(a) Half wave rectification circuit using a single diode.

(b) Half wave rectified sine waveform.

Figure 4.3: A half wave rectification circuit and the resulting output.

A half wave rectifier is also commonly employed in DC power supplies for AC to DC conversion. This requires just one diode as opposed to four in a bridge rectifier. However, the average value is half of the full wave as given by Equation 4.4.

Effective or RMS Value

The effective or root mean square (RMS) value of a periodic signal is equal to the magnitude of a DC signal which produces the same heating effect as the periodic signal when applied across a load resistance.

Consider a periodic signal, $v(t)$, then

$$\begin{aligned} \text{Mean} &= \frac{1}{T}\int_0^T v(t)\,dt \\ \text{Mean Square} &= \frac{1}{T}\int_0^T v^2(t)\,dt \\ \text{Root Mean Square} &= \sqrt{\frac{1}{T}\int_0^T v^2(t)\,dt} \end{aligned} \tag{4.5}$$

Download free ebooks at bookboon.com

The RMS value of a sine wave is found out to be

$$\boxed{V_{rms} = \frac{V_m}{\sqrt{2}}} \tag{4.6}$$

The mains voltage of $230\ V$ *in the UK is its RMS value. A multimeter measures RMS voltages whereas an oscilloscope measures peak amplitudes. Hence the mains voltage when displayed on an oscilloscope will read* $230 \times \sqrt{2} = 325.27\ \ V$

All the above expressions are independent of the phase angle ϕ.

4.3 Voltage, Current Relationships for R, L and C

The AC voltage-current relationships for a resistor, inductor and capacitor in an electric circuit are given by

$$R \rightarrow v_R(t) = R\, i(t) \tag{4.7}$$

$$L \rightarrow v_L(t) = L\, \frac{di(t)}{dt} \tag{4.8}$$

$$C \rightarrow v_C(t) = \frac{1}{C} \int i(t)\, dt \tag{4.9}$$

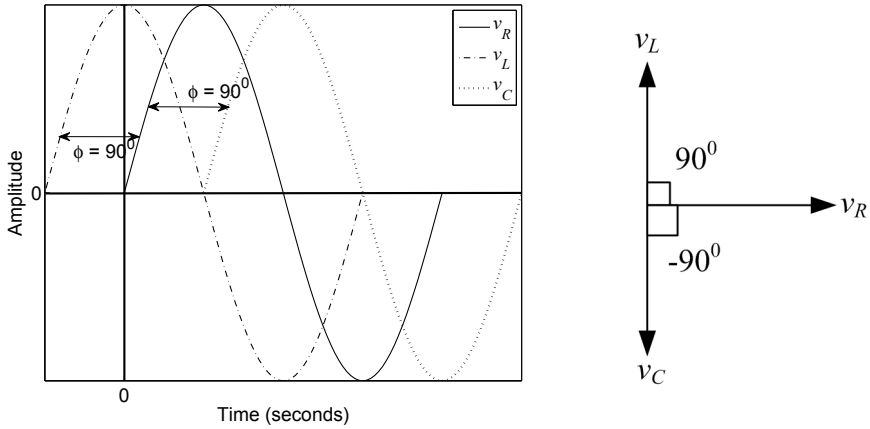
if $i(t) = I_m \sin(\omega t)$, then

$$v_R(t) = RI_m \sin(\omega t) \tag{4.10}$$

$$v_L(t) = L\frac{d}{dt} I_m \sin(\omega t) = \omega L I_m \cos(\omega t) = \omega L I_m \sin(\omega t + 90^0) \tag{4.11}$$

$$v_C(t) = \frac{1}{C} \int I_m \sin(\omega t) dt = \frac{-I_m}{\omega C} \cos(\omega t) = \frac{I_m}{\omega C} \sin(\omega t - 90^0) \tag{4.12}$$

It is clear that with AC excitation, all voltages and currents retain the basic sine wave shape as depicted in Figure 4.4(a).



(a) Voltage waveforms across R, L and C showing the phase difference.

(b) Vector diagram showing $V - I$ phase lag/lead concept in RLC.

Figure 4.4: Phase difference between voltages across R, L and C.

However, a phase difference of 90^0 (lead) and -90^0 (lag) with respect to the input current is observed in the inductor and capacitor voltages respectively. This is demonstrated in the vector diagram of Figure 4.4(b). A simple way to remember this concept is to consider a CIVIL relationship where C, L, V and I represents capacitor, inductor, voltage and current respectively.

This reads as

In a Capacitor ($\underline{C}$), current, $\underline{I}$ leads voltage, $\underline{V}$ whereas V leads $\underline{I}$ in an inductor ($\underline{L}$).

4.4 Impedance

From Equations 4.10, 4.11 and 4.12, the magnitudes of the ratio of voltage to current across the three circuit elements can be written as

$$\left|\frac{V_R}{I_m}\right| = R \tag{4.13}$$

$$\left|\frac{V_L}{I_m}\right| = \omega L \tag{4.14}$$

$$\left|\frac{V_C}{I_m}\right| = \frac{1}{\omega C} \tag{4.15}$$

where ωL and $\frac{1}{\omega C}$ have the dimensions of resistance (Ω) and are termed as inductive reactance and capacitive reactance respectively i.e.

$$X_L = \omega L \quad \rightarrow \quad \text{inductive reactance}$$
$$X_C = \frac{1}{\omega C} \quad \rightarrow \quad \text{capacitive reactance}$$

Download free ebooks at bookboon.com

The impedance, Z can now be defined by the following relationship

$$\begin{aligned} \text{Impedance} &= \text{Resistance} \pm j\ \text{Reactance} \\ \text{or } Z &= R + j\,X \qquad \text{where } j \text{ represents a } 90^0 \text{ phase shift} \end{aligned}$$

X is positive for inductance and negative for capacitance. The magnitude and phase of the impedance can be calculated as follows

$$|Z| = \sqrt{R^2 + X^2} \tag{4.16}$$

$$\phi = \arctan \pm \frac{X}{R} \tag{4.17}$$

Admittance The reciprocal of impedance is called admittance (Y) and is measured in $\mho$. Mathematically

$$Y = \frac{1}{Z} = \frac{I}{V}$$

Also $Y = G \pm jB$, where G and B represents conductance and susceptance respectively. For a purely resistive or purely inductive/capacitive circuit

$$G = \frac{1}{R} \text{ and } B = \frac{1}{X}$$

For a combination of resistance and reactance

$$Y = \frac{1}{Z} = \frac{1}{R + jX}$$

multiply and divide by the complex conjugate

$$Y = \frac{R - jX}{R^2 + X^2}$$

$$Y = \frac{R}{R^2 + X^2} - j\frac{X}{R^2 + X^2}$$

$$\therefore G = \frac{R}{R^2 + X^2} \ \ \& \ \ B = \frac{X}{R^2 + X^2}$$

4.5 Phasors

Addition of two *out-of-phase* sinusoidal signals is rather complicated in the time domain. An example could be the sum of voltages across a series connection of a resistor and an inductor.

Phasors simplify this analysis by considering only the amplitude and phase components of the sine wave. Moreover, they can be solved using complex algebra or treated vectorially using a vector diagram.

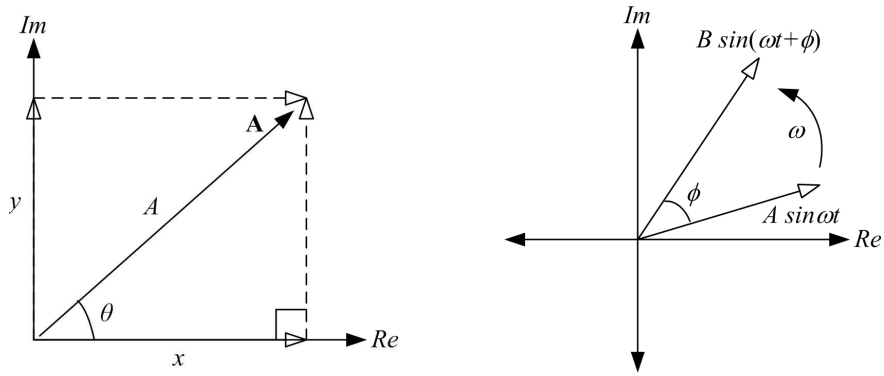
Download free ebooks at bookboon.com

Consider a vector quantity $\mathbf{A}$ in the complex plane as shown in Figure 4.5(a). Then

$$\mathbf{A} = x + j\,y \tag{4.18}$$

where $x = A\cos\theta$ and $y = A\sin\theta$, therefore

$$\begin{aligned}
\mathbf{A} &= A\cos\theta + jA\sin\theta \\
&= A\left(\cos\theta + j\sin\theta\right) \\
\because e^{j\theta} &= \cos\theta + j\sin\theta \;\; \text{(Euler's formula)} && (4.19)\\
\therefore \mathbf{A} &= Ae^{j\theta} && (4.20)
\end{aligned}$$



(a) Phasor as a vector (b) Relative motion of two phasors

Figure 4.5: Vector representation of a phasor.

Please click the advert

FOSS

Sharp Minds - Bright Ideas!

Employees at FOSS Analytical A/S are living proof of the company value - First - using new inventions to make dedicated solutions for our customers. With sharp minds and cross functional teamwork, we constantly strive to develop new unique products - Would you like to join our team?

FOSS works diligently with innovation and development as basis for its growth. It is reflected in the fact that more than 200 of the 1200 employees in FOSS work with Research & Development in Scandinavia and USA. Engineers at FOSS work in production, development and marketing, within a wide range of different fields, i.e. Chemistry, Electronics, Mechanics, Software, Optics, Microbiology, Chemometrics.

We offer
A challenging job in an international and innovative company that is leading in its field. You will get the opportunity to work with the most advanced technology together with highly skilled colleagues.

Read more about FOSS at www.foss.dk - or go directly to our student site www.foss.dk/sharpminds where you can learn more about your possibilities of working together with us on projects, your thesis etc.

The Family owned FOSS group is the world leader as supplier of dedicated, high-tech analytical solutions which measure and control the quality and production of agricultural, food, pharmaceutical and chemical products. Main activities are initiated from Denmark, Sweden and USA with headquarters domiciled in Hillerød, DK. The products are marketed globally by 23 sales companies and an extensive net of distributors. In line with the corevalue to be 'First', the company intends to expand its market position.

Dedicated Analytical Solutions
FOSS
Slangerupgade 69
3400 Hillerød
Tel. +45 70103370
www.foss.dk

Download free ebooks at bookboon.com

Equation 4.20 is the phasor notation of the vector $\mathbf{A}$ where A is the length of the vector or amplitude of the signal and θ is the angle which $\mathbf{A}$ makes with a reference phasor.

Rotating Vector Concept Let $\theta = \omega t$, therefore $\mathbf{A} = Ae^{j\omega t}$ where ω is the angular velocity and t is the time. Then $\mathbf{A}$ can be regarded as a vector rotating with an angular velocity ω. Now if two phasors are rotating with the same velocity as illustrated in Figure 4.5(b), then their relative positions are unchanged with respect to time and therefore can be added.

> *Two or more sinusoidal signals can be added mathematically using phasors if they all have the same angular velocities.*

Polar form is also commonly used to represent a phasor and is given by the magnitude (modulus) and phase (argument) of the signal i.e.,

$$\mathbf{A} = A\angle\theta$$

It may be convenient to transform the polar form into cartesian form (Equation 4.18) and vice versa when adding two sinusoidal signals.

Example

Express the following as phasors and write the corresponding polar and cartesian forms.

1. $5\sin(\omega t + 45^0)$
2. $2\cos(\omega t)$
3. $10\sin(\omega t)$
4. $3\cos(\omega t + 30^0)$

Solution

1. $5e^{j45^0}$ or $5e^{j\pi/4}$ (phasor notation)
   (a) $5\angle 45^0$ (polar form)
   (b) $5\cos 45^0 + j5\sin 45^0 = 3.54 + j3.54$ (cartesian form)
2. $2\sin(\omega t + 90^0) = 2e^{j90^0}$ (phasor notation)
   (a) $2\angle 90^0$ (polar form)
   (b) $2\cos 90^0 + j2\sin 90^0 = 0 + j2$ (cartesian form)
3. $10e^{j0^0}$ (phasor notation)
   (a) $10\angle 0^0$ (polar form)
   (b) $10\cos 0^0 + j10\sin 0^0 = 10 + j0$ (cartesian form)
4. $3\sin(\omega t + 30^0 + 90^0) = 3\sin(\omega t + 120^0) = 3e^{j120^0}$ (phasor notation)

Download free ebooks at bookboon.com

(a) $3\angle 120^0$ (polar form)

(b) $3\cos 120^0 + j3\sin 120^0 = -1.5 + j2.6$ (cartesian form)

In the above examples and all other instances in this book, $\sin\omega t$ is used as a reference for phasor conversion. This means that a cosine signal is converted to a sine wave before writing the phasor notation. It is also valid to use $\cos\omega t$ as reference, however, consistency is required in a given problem.

Example

Express the following as sinusoids.

1. $2e^{j\pi/2}$
2. $5e^{j\pi/4}$
3. $5 + 6j$
4. $3e^{j120^0}$

Solution

1. $2\sin(\omega t + \pi/2) = 2\cos\omega t$
2. $5\sin(\omega t + \pi/4)$
3. $\sqrt{5^2 + 6^2} = 7.81,\ \tan^{-1}\frac{6}{5} = 50.2^0 \Rightarrow 7.81\angle 50.2^0 = 7.81\sin(\omega t + 50.2^0)$
4. $3\sin(\omega t + 120^0)$ or $3\sin(\omega t + 30^0 + 90^0) = 3\cos(\omega t + 30^0)$

Example

Add the following two sinusoidal signals using (a) complex algebra (b) vector diagram.

$$\begin{aligned} v_1(t) &= 2\sin(5t + 30^0) \\ v_2(t) &= \sin(5t + 60^0) \end{aligned}$$

Solution (a) Since $\omega = 5$ rad/s for both sinusoids, hence phasors can be employed to add these signals.

The polar form representation for v_1 and v_2 is

$$\mathbf{V_1} = 2\angle 30^0,\ \ \mathbf{V_2} = 1\angle 60^0$$

The sum of the phasors is given by

$$\mathbf{V} = \mathbf{V_1} + \mathbf{V_2}$$

Download free ebooks at bookboon.com

$$
\begin{aligned}
\mathbf{V} &= 2\angle 30^0 + 1\angle 60^0 \\
& \text{converting polar to cartesian coordinates} \\
&= 2\cos 30^0 + j2\sin 30^0 + \cos 60^0 + j\sin 60^0 \\
&= 1.732 + j + 0.5 + j0.866 \\
&= 2.232 + j1.866 \\
& \text{converting back to polar form} \\
\mathbf{V} &= 2.91\angle 39.9^0
\end{aligned}
$$

$$\therefore v(t) = 2.91\sin(5t + 39.9^0)$$

Solution (b) See vector diagram in Figure 4.6 for the solution.

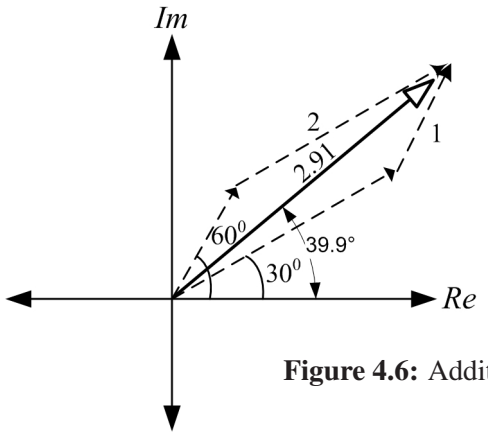
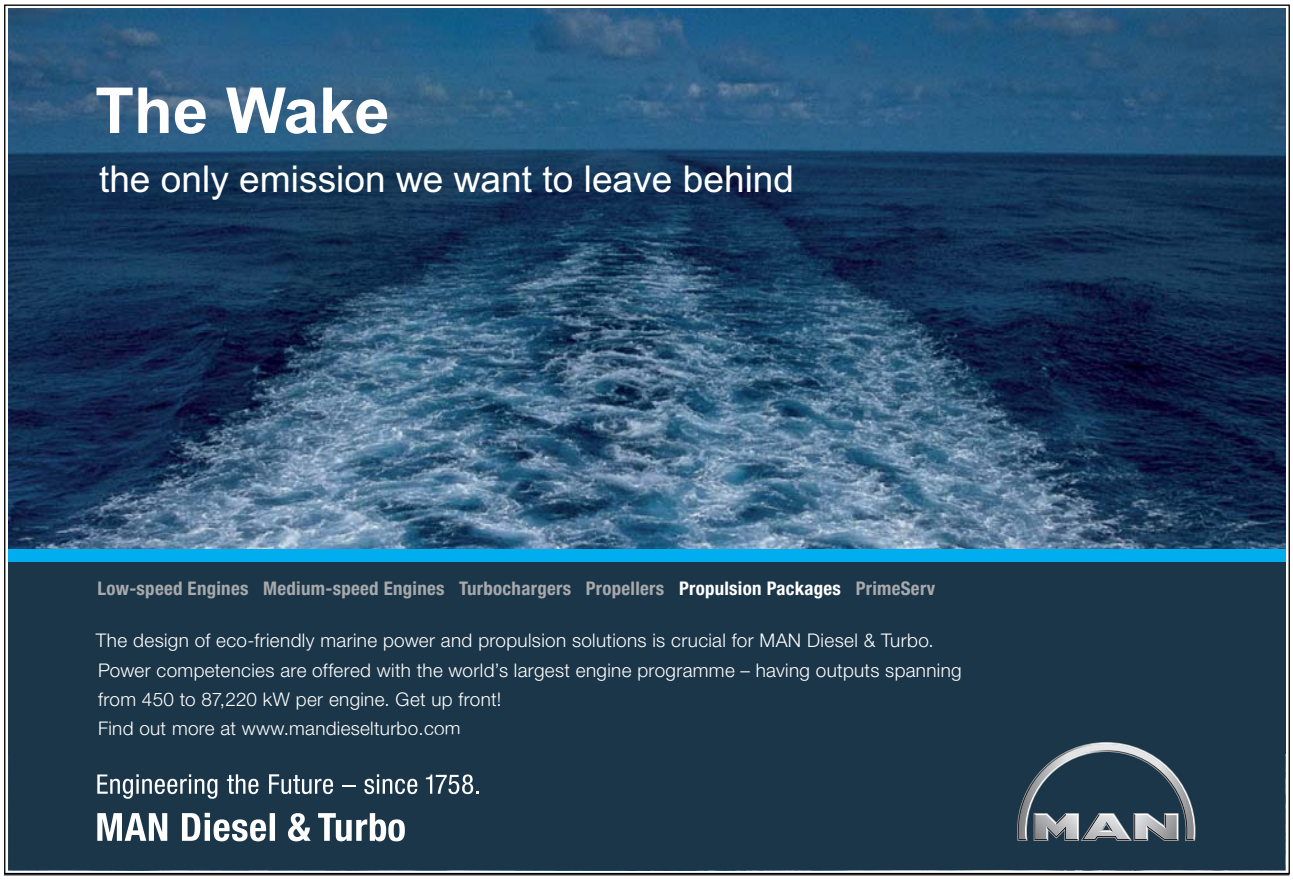


Figure 4.6: Addition of phasors using a vector diagram.

Please click the advert



Download free ebooks at bookboon.com

4.6 Phasor Analysis of AC Circuits

Fortunately, the techniques employed for DC circuit analysis can be reused for AC circuits by following the procedure outlined below:

1. Replace sinusoidal voltages and currents, v and i by their respective phasors.
2. Replace the linear circuit elements R, L and C by their respective impedances R, $j\omega L$ and $-j/(\omega C)$.
3. Use DC circuit laws redefined for AC circuits below to evaluate the required voltage and current phasors. Complex algebra will have to be employed to simplify the equations.

   a. Ohms law $V = ZI$

   b. KVL $\sum V = 0$ (mesh voltages)

   c. KCL $\sum I = 0$ (nodal currents)

   d. Impedances in series: $Z = Z_1 + Z_2 + \cdots$

   e. Impedances in parallel: $\dfrac{1}{Z} = \dfrac{1}{Z_1} + \dfrac{1}{Z_2} + \cdots$ or $Y = Y_1 + Y_2 + \cdots$

   f. VDR (impedances in series) $V_i = V \cdot \dfrac{Z_i}{\sum Z}$

   g. CDR (impedances in parallel) $I_i = I \cdot \dfrac{Y_i}{\sum Y}$

4. Convert the voltage and current phasors back to sinusoidal form using the techniques learned in the previous section.

Example

Use phasor analysis to determine the voltage across the terminals a and b in the circuit diagram of Figure 4.7.

In phasor form, the supply voltage can be written as

$$\mathbf{V} = 10\angle 0^0 \text{ where } \omega = 2\text{rad/s}$$

The 1 Ω resistance and 1 H inductance can be combined into a single impedance Z_1. Similarly the 2 Ω resistance and 1 F capacitor can be added to form Z_2 as shown in Figure 4.7.

$$\therefore Z_1 = R + j\omega L = 1 + j2 \times 1 = 1 + j2\ \Omega$$

Also

$$Z_2 = R + 1/(j\omega C) = 2 + 1/(j2 \times 1) = 2 - j0.5\ \Omega$$

Download free ebooks at bookboon.com

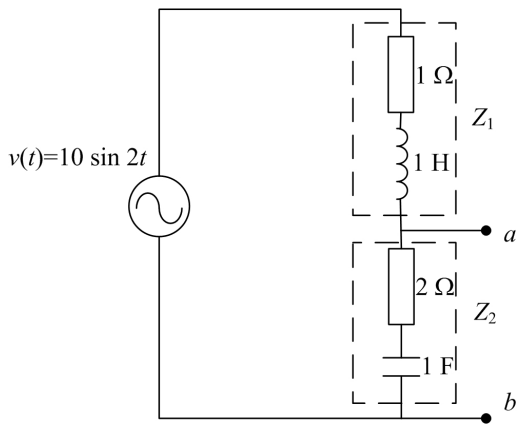


Figure 4.7

Steps 1 and 2 of AC circuit analysis have now been completed whereas step 3 entails the use of one of the circuit laws. Since both Z_1 and Z_2 are in series with the voltage source, therefore VDR can be applied to determine the voltage across Z_2 which is also the voltage across the terminals a and b.

$$\begin{aligned}
\mathbf{V_{ab}} &= V\frac{Z_2}{Z_1+Z_2} \\
&= 10\angle 0^0 \frac{2-j0.5}{1+j2+2-j0.5} \\
&= 10\angle 0^0 \cdot \frac{2.06\angle -14.04^0}{3.35\angle 26.56^0} \\
&= \frac{10\times 2.06}{3.35}\angle\left(0^0-14.04^0-26.56^0\right) \\
\mathbf{V_{ab}} &= 6.15\angle -40.6^0 \\
\therefore v_{ab}(t) &= 6.15\sin(2t-40.6^0)\text{ V}
\end{aligned}$$

Example

For the circuit diagram shown in Figure 4.8, determine the voltage V across the 5 Ω impedance and specify it in time domain when ω = 1000 rad/s.

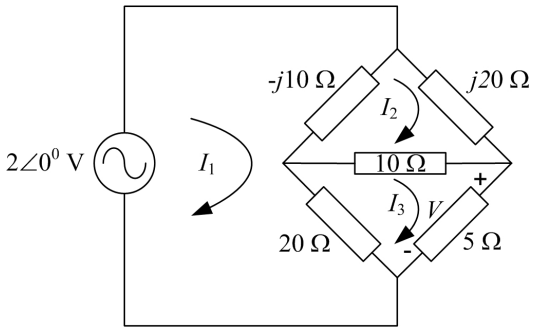


Figure 4.8

The circuit for this example is an AC bridge circuit which is similar to the bridge rectifier circuit in Figure 4.2(a). This can be solved using mesh analysis with mesh currents $I_1,\ I_2$ and I_3 as shown.

Mesh 1

$$\begin{aligned}(I_1 - I_2)(-j10) + (I_1 - I_3)(20) &= 2 \\ (20 - j10)I_1 + j10I_2 - 20I_3 &= 2 \end{aligned} \tag{4.21}$$

Mesh 2

$$\begin{aligned}(I_2 - I_1)(-j10) + j20I_2 + (I_2 - I_3)(10) &= 0 \\ j10I_1 + (10 + j10)I_2 - 10I_3 &= 0 \end{aligned} \tag{4.22}$$

Mesh 3

$$\begin{aligned}(I_3 - I_1)(20) + (I_3 - I_2)(10) + I_3(5) &= 0 \\ -20I_1 - 10I_2 + 35I_3 &= 0 \end{aligned} \tag{4.23}$$

Equations 4.21-4.23 can be solved simultaneously to determine the unknown mesh currents using Cramer's rule as outlined in Appendix A. In this case, since the voltage across the 5 Ω resistance is required, therefore only the current I_3 is evaluated using the following relationship.

$$I_3 = \frac{\Delta_3}{\Delta}$$

Please click the advert

Download free ebooks at bookboon.com

where

$$\Delta_3 = \begin{vmatrix} 20-j10 & j10 & 2 \\ j10 & 10+j10 & 0 \\ -20 & -10 & 0 \end{vmatrix} \text{ and } \Delta = \begin{vmatrix} 20-j10 & j10 & -20 \\ j10 & 10+j10 & -10 \\ -20 & -10 & 35 \end{vmatrix}$$

$$I_3 = 0.05 - j0.0024\,(A) = 0.05\angle{-2.8^0}\text{ A}$$

$$\therefore \mathbf{V} = 5 \times I_3 = 5 \times 0.05\angle{-2.8^0} = 0.25\angle{-2.8^0}\text{ V}$$

In the time domain

$$v(t) = 0.25\sin(1000t - 2.8^0)\text{ V}$$

The mesh and nodal equations for AC circuits can also be directly written in matrix form by inspection using the same method as outlined in Section 3.11 for DC circuits. However, the R and G parameters are replaced by Z and Y respectively and complex algebra is employed for all complex numbers computations.

4.7 Power in AC Circuits

From DC circuit analysis in Chapter 3, power is given by the following relationship

$$P = VI \text{ Watts}$$

Therefore the instantaneous power is given by $p(t) = v(t) \cdot i(t)$. Let $i(t) = I_m \sin(\omega t + \beta)$ be the current flowing through an impedance, Z, then the voltage across the impedance will be $v(t) = V_m \sin(\omega t + \alpha)$. Therefore

$$p(t) = V_m I_m \sin(\omega t + \alpha)\sin(\omega t + \beta)$$

Using the trigonometric identity

$$\sin A \sin B = \frac{1}{2}\left[\cos(A - B) - \cos(A + B)\right]$$

$$p(t) = -\frac{1}{2}V_m I_m \cos(2\omega t + \alpha + \beta) + \frac{1}{2}V_m I_m \cos(\alpha - \beta) \tag{4.24}$$

The first term is a time-varying sinusoidal waveform with twice the frequency of $v(t)$ or $i(t)$ and an average value of zero. On the other hand, the second term is a constant quantity and is called the DC level or average value of the power signal, $p(t)$ or average power delivered to the load i.e.,

$$\begin{aligned} P_{av} &= \frac{1}{2}V_m I_m \cos(\alpha - \beta) \\ P_{av} &= \frac{1}{2}V_m I_m \cos(\phi) \end{aligned} \tag{4.25}$$

Download free ebooks at bookboon.com

where $\phi = \alpha - \beta$ is the phase angle between $v(t)$ and $i(t)$.

The waveform $p(t)$ is plotted in Figure 4.9 for an arbitrary value of ϕ where the negative shaded portion of the waveform is the power returned (not absorbed by the load) to the source.

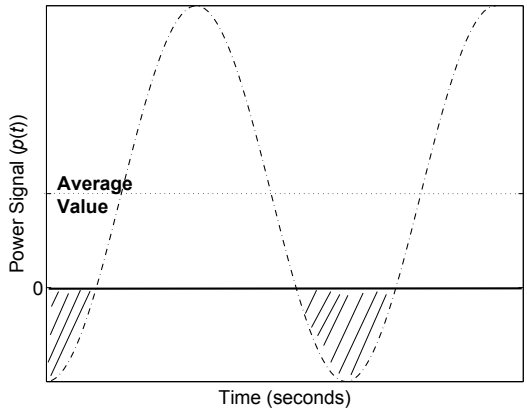
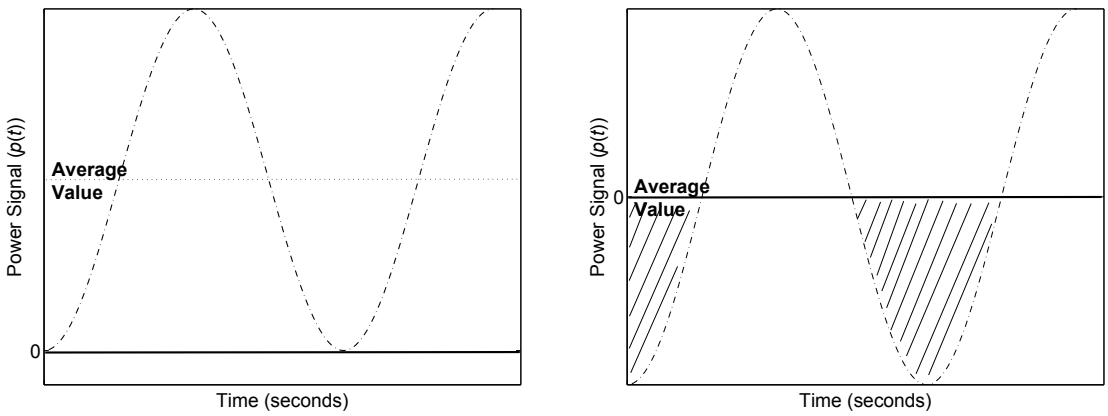


Figure 4.9: Power supplied to a generic load impedance.

For a purely resistive load ($\phi = 0^0$), Equation 4.25 becomes

$$P_{av} = \frac{1}{2} V_m I_m$$

and the resulting power signal, $p(t)$ is depicted in Figure 4.10(a). In this case, there is no negative portion of the waveform and hence all the power is dissipated in the load.



(a) Purely resistive load.

(b) Purely reactive load.

Figure 4.10: Power supplied to purely resistive and reactive loads.

For a purely reactive load, i.e., $Z = \pm jX$, $\phi = 90^0$ and $P_{av} = 0$. The power signal waveform is drawn in Figure 4.10(b) showing that the power oscillates between the source and electric/magnetic field of the load. Thus the power dissipated in the load is zero for a purely reactive load.

In terms of RMS voltage and current, the average power is given by

Download free ebooks at bookboon.com

$$P_{rms} = V_{rms} I_{rms} \cos\phi \tag{4.26}$$

4.8 Power Factor

The term $\cos\phi$ in Equation 4.25 is called the *power factor* and is an important parameter in determining the amount of actual power dissipated in the load. In practise, power factor is used to specify the characteristics of a load.

For a purely resistive load $\phi = 0^0$, hence *Unity* Power Factor

For a capacitive type load I leads V, hence *Leading* power factor

For an inductive type load I lags V, hence *Lagging* power factor

From Equation 4.26, the current can be specified as

$$I_{rms} = \frac{P_{rms}}{V_{rms} \cos\phi}$$

Clearly, for a fixed amount of demanded power, P, at a constant load voltage, V, a higher power factor draws less amount of current and hence low I^2R losses in the transmission lines. A purely reactive load where $\phi \to 90^0$ and $\cos\phi \to 0$ will draw an excessively large amount of current and a power factor correction is required as discussed in the next section.

Please click the advert

Download free ebooks at bookboon.com

Real and Apparent Power It is important to highlight that in AC circuits, the product of voltage and current yields the apparent power which is measured in volt-amperes[2] or VA. The real AC power is given by Equation 4.26 which is measured in Watts.

Example

An AC generator is rated at 900 kVA (450 V /2000 A). This is the apparent power and represents the highest current and voltage magnitudes the machine can output without temperature exceeding the recommended value. The load voltage, however, could be 230 V with a supplied current depending on the load impedance. The phase angle between the current and voltage determines the power factor and the real power can be calculated using Equation 4.26.

4.8.1 Power Factor Correction

As mentioned above, to minimise I^2R losses in AC networks, it may be necessary to manipulate the power factor. This can be achieved by connecting a purely reactive impedance, Z' in parallel with the load impedance as depicted in Figure 4.11.

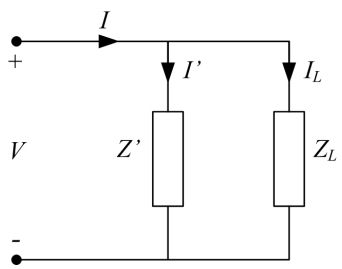


Figure 4.11: Power factor correction by introducing a pure reactance in parallel with the load impedance.

By doing so, the phase angle between the line voltage and line current can be adjusted whereas the power consumption remains the same since a pure reactance does not dissipate any power (see Figure 4.10(b)). A carefully chosen reactance will decrease ϕ and thus increases the power factor which in turn reduces the line current.

Example

Given that the RMS line voltage and line current are $\mathbf{V} = 100\angle 0^0$ V and $\mathbf{I} = 50\angle -36.9^0$ A respectively, determine the impedance which will result in a unity power factor.

Taking voltage as the reference phasor, the phase angle between the line voltage and line current is -36.9^0

$$\therefore \text{Power factor} = \cos\phi = \cos(-36.9^0) = 0.8$$

$$\mathbf{I} = \mathbf{I_L} = 50\angle -36.9^0 = 40 - j30 \text{ A}$$

[2]In DC circuits, the product of voltage and current is the real power measured in Watts.

Download free ebooks at bookboon.com

$$\therefore \text{ real power } P = |V||I|\cos\phi = 100 \times 50 \times 0.8 = 4 \text{ kW}$$

To improve the power factor, an impedance is inserted in parallel with the load as in Figure 4.11 resulting in the following nodal equation

$$I = I_L + I'$$

It is clear that for unity power factor, the imaginary component of the line current must be zero. This is accomplished by assigning a current of $j30$ A through the pure reactance resulting in the following line current

$$I = (40 - j30) + (j30) = 40 \text{ A}$$

Thus $I' = j30$ A and the magnitude of the impedance required can be calculated as follows

$$Z' = \frac{V}{I'} = \frac{100}{j30} = -j3.33\ \Omega$$

The power consumed by the load remains the same i.e.

$$P = |V||I|\cos\phi = 100 \times 40 \times 1 = 4 \text{ kW}$$

Thus to achieve unity power factor, a pure reactance of $-j3.33\ \Omega$ is connected in parallel with the load.
The vector diagram of the above example is shown in Figure 4.12.

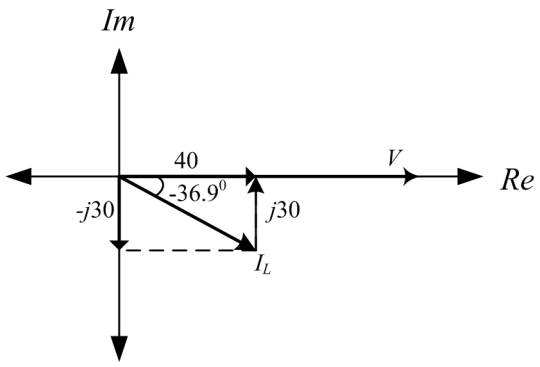


Figure 4.12: Vector diagram to illustrate power factor correction.

Chapter 5

Frequency Response

5.1 Introduction

The AC circuits concepts presented so far assumes sinusoidal excitation at a fixed frequency. It was mentioned that two or more signals can be combined using phasors provided they all have the same frequency. In this chapter, the response of a linear AC circuit is examined when excited with an AC signal of constant amplitude but varying frequency hence the term frequency response. These signals are common in everyday application areas such in a radio, television and telephone and can be studied using frequency response analysis. In addition, filter circuits are introduced and their output response is observed. Techniques to draw approximate frequency response plots by inspection is described in the end.

Please click the advert

Download free ebooks at bookboon.com

5.2 Frequency Response

To understand the concept of frequency response, consider a linear circuit with input and output voltage signals represented by V_i and V_o respectively as shown in Figure 5.1.

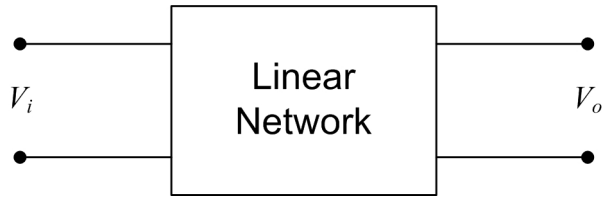


Figure 5.1: A two port linear network showing the input and output voltage signals.

Now if the input amplitude V_i is kept constant whilst its frequency ω (rad/s) is varied, then it is observed that the amplitude and phase of the output signal V_o will also change. However, the input and output frequencies remain unaltered. This means that V_o and ϕ become functions of the frequency ω represented by $V_o(\omega)$ and $\phi(\omega)$ with V_i being the reference signal.

The ratio of the output to input voltage signal is denoted by $H(j\omega)$ which is a complex function and can be written mathematically as

$$H(j\omega) = \frac{V_o}{V_i} \tag{5.1}$$

In polar form

$$\mathbf{H}(\mathbf{j}\omega) = \frac{|V_o(\omega)|\angle\theta^0(\omega)}{|V_i|\angle 0^0} \tag{5.2}$$

$$\text{or} \quad \mathbf{H}(\mathbf{j}\omega) = A\angle\phi \tag{5.3}$$

where the voltage gain $A = |H(j\omega)|$ and $\phi = \arg H(j\omega)$ are both functions of frequency, ω.

Amplitude Response The variations recorded in the amplitude gain, A with respect to ω is called the amplitude response of the network.

Phase Response The variations observed in the phase, ϕ of the network with respect to ω is known as the phase response.

The amplitude and phase outputs together define the frequency response of a network.

In the next section, filters are introduced and their response to an input signal with varying frequency is observed. This will help further to clarify the concepts of frequency response analysis.

Download free ebooks at bookboon.com

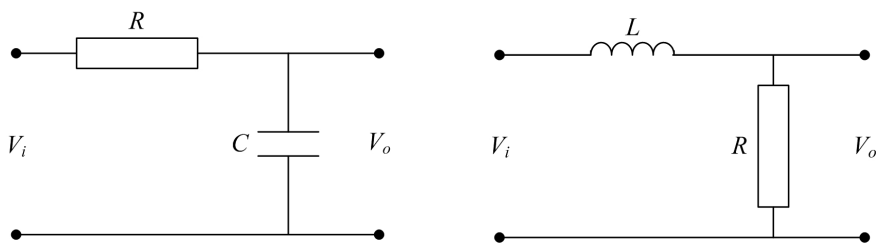
5.3 Filters

Filters form a vital part in electrical networks especially where a particular frequency range is of prime concern. For instance, a radio station is broadcasting a transmission at a frequency of 100 MHz. This means that it is required to design a receiving filter which allows only 100 MHz frequency to pass through whilst other frequencies are filtered out. An ideal filter will attenuate all signals with frequencies less than and greater than 100 MHz thus providing the best channel sound quality without any distortion.

The following sections outline the most commonly used filters found in electric circuits and their respective frequency responses. This includes low pass, high pass and band pass filters.

5.3.1 Low Pass Filter

A low pass filter *allows* low frequencies to pass through the circuit whereas high frequencies are severely attenuated or blocked. Figure 5.2(a) depicts a low pass filter constructed using a simple RC network. The output voltage, V_o is taken across the capacitor.



(a) An RC low pass filter.

(b) Alternate circuit of a high pass filter using RL elements.

Figure 5.2: Two different circuits used to construct a low pass filter.

Note that it is also possible to construct a low pass filter using an RL combination as shown in Figure 5.2(b). Herein, the analysis is restricted to RC circuit only whereas the RL circuit can be observed in a similar manner.

The reactance of the capacitor is $\frac{1}{j\omega C}$ which will be used in the mathematical analysis of the low pass filter. To determine V_o, VDR (Chapter 3) can be applied since R and C are in series hence share the same current. Therefore

$$\begin{aligned} V_o &= V_i \frac{\frac{1}{j\omega C}}{R + \frac{1}{j\omega C}} \\ V_o &= V_i \frac{1}{1 + j\omega RC} \end{aligned}$$

Let $T = RC$ be the time constant, then

$$H(j\omega) = \frac{V_o}{V_i} = \frac{1}{1 + j\omega T} \tag{5.4}$$

Download free ebooks at bookboon.com

The output response of Equation 5.4 to a change in input signal frequency is now investigated.

At low frequencies, $\omega \approx 0, \therefore V_o \approx V_i$, i.e. most of V_i appears at the output across the capacitor. When $\omega \rightarrow \infty$, $V_o \rightarrow 0$, hence output voltage amplitude is severely attenuated i.e. very little of V_i appear at V_o at high frequencies. Therefore this circuit acts as a low pass filter. The frequency response can also be visually inspected by plotting the amplitude and phase responses versus frequency. From Equation 5.4, the amplitude response equation of the low pass filter can be derived as

$$A = |H(j\omega)| = \frac{1}{\sqrt{1 + (\omega T)^2}}$$

and the phase is given by

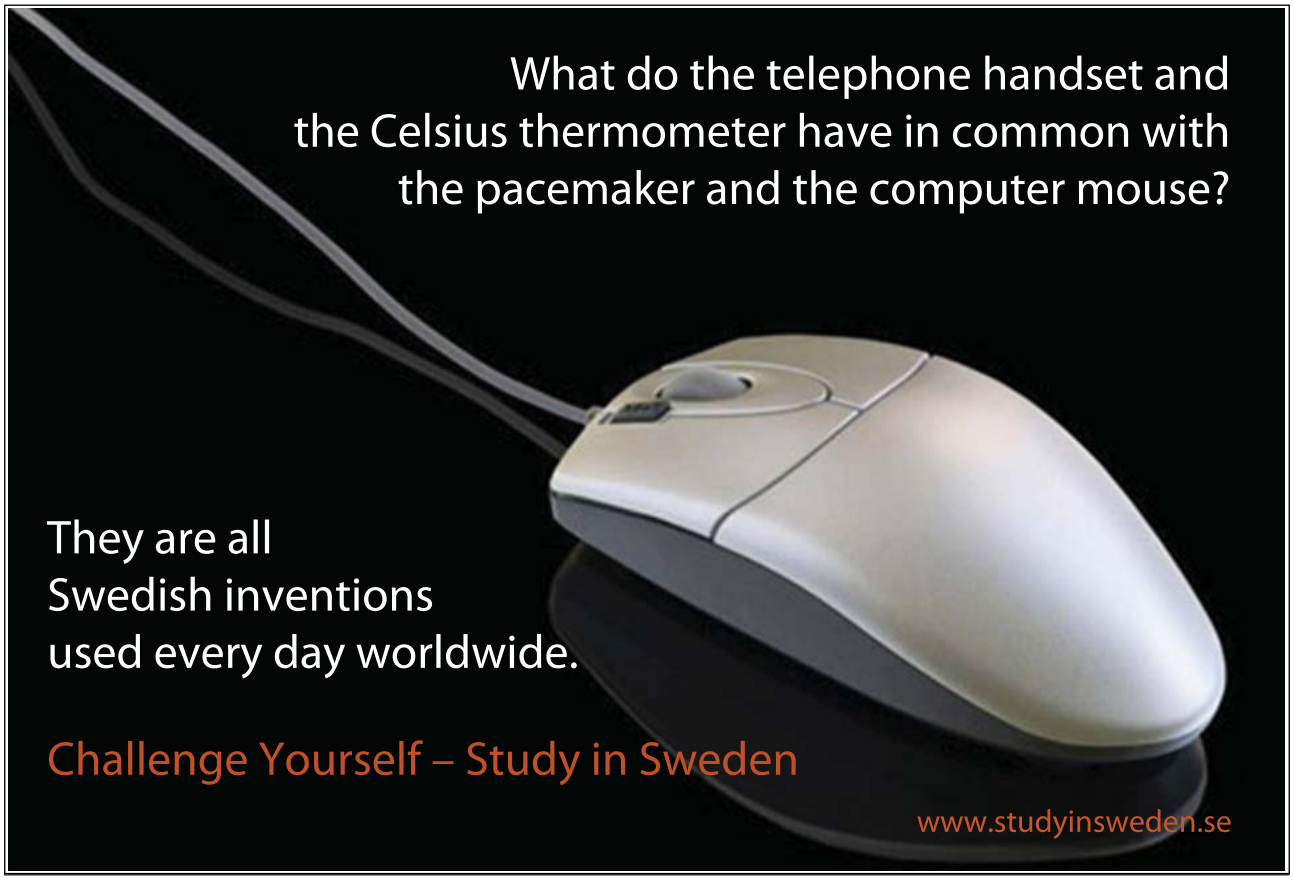
$$\phi = \arg(H(j\omega)) = \tan^{-1}(-\omega T)$$

For $\omega \approx 0, A \approx 1$ and $\phi \approx 0$

For $\omega \rightarrow \infty, A \rightarrow 0$ and $\phi \rightarrow -\pi/2$

For $\omega = \frac{1}{T}, A = \frac{1}{\sqrt{2}} = 0.7071$ and $\phi = -\pi/4 = -45^0$

Please click the advert



Download free ebooks at bookboon.com

The response plots are shown in Figure 5.3.

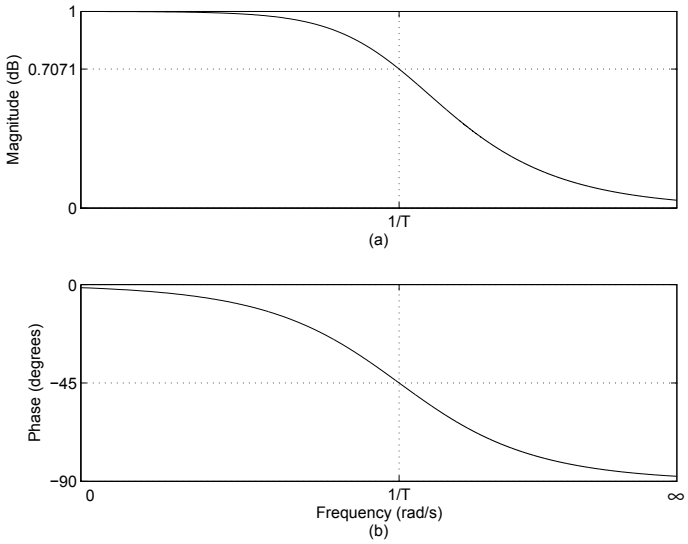


Figure 5.3: Frequency response of a low pass filter where (a) Amplitude response (b) Phase response

Bandwidth The frequency $\omega = 1/T$ is the cutoff frequency or bandwidth of the low pass filter. It is defined as the frequency range for which the amplitude $A \geq \frac{1}{\sqrt{2}}$. Mathematically, the bandwidth of an RC low pass filter is given by

$$\text{Bandwidth } = \frac{1}{T} = \frac{1}{RC}$$

Clearly the bandwidth is user and problem dependent and can be adjusted simply by tuning the R and C parameters.

It is more common to express the gain, A of a circuit in Decibels (dB) units given by

$$A_{dB} = 20 \log_{10} A$$

which was originally developed as the logarithmic unit of power ratio i.e.

$$\text{dB Power Gain } = 10 \log_{10} \frac{P_o}{P_i}$$

The amplitude plot in Figure 5.3 is redrawn in Figure 5.4 with gain converted to dB versus logarithm of frequency. The phase response remains unaltered.

Some vital observations can be made on comparing Figures 5.3 and 5.4.

- Zero dB corresponds to a unity gain i.e. $A = 1$
- Negative dB corresponds to attenuation i.e. $A < 1$
- Positive dB corresponds to amplification i.e. $A > 1$

Download free ebooks at bookboon.com

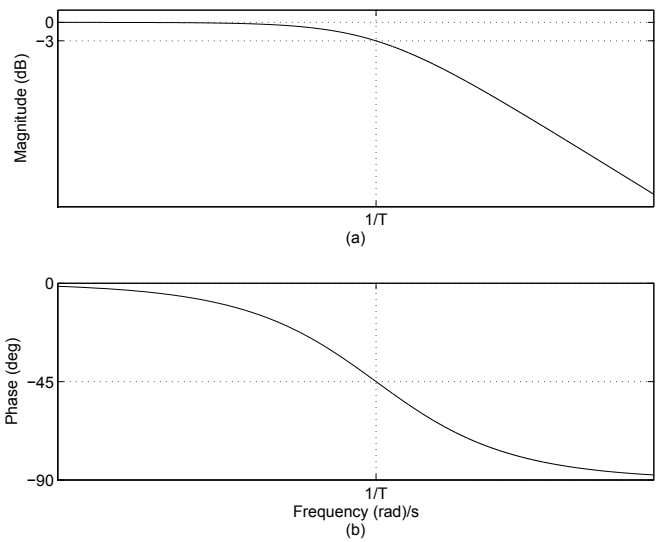


Figure 5.4: Frequency response of a low pass filter where (a) Amplitude response in dB (b) Phase response

- The bandwidth of a low pass filter is the frequency range for which the gain, $A \geq$ -3 dB

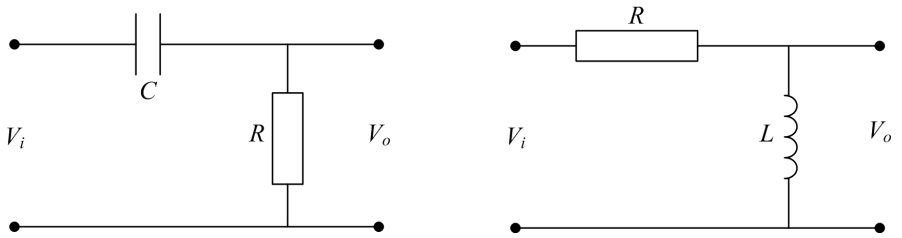
The -3 dB point is also called the *half power point* since

$$10\log_{10}\frac{P_o}{P_i} = 10\log_{10}\frac{V_o^2}{V_i^2} = 10\log_{10}(A)^2 = 10\log_{10}\left(\frac{1}{\sqrt{2}}\right)^2 = 10\log_{10}\frac{1}{2} = -3 \text{ dB}$$

5.3.2 High Pass Filter

A high pass filter, as the name suggests, *allows* high frequencies to pass through the circuit whilst low frequencies are attenuated or blocked. The cut-off point or bandwidth concept is the same as in the low pass filter.

In practise, the same circuit used for the low pass filter can be adopted as a high pass filter with the output taken across the resistor this time. The resulting schematic is shown in Figure 5.5(a).



(a) An RC high pass filter.

(b) Alternate circuit of a low pass filter using RL combination.

Figure 5.5: Two alternate circuit configurations used to construct a high pass filter.

As before, VDR can be applied to determine the output voltage across the resistor in Figure 5.5(a).

$$\begin{aligned} V_o &= V_i \frac{R}{R + \frac{1}{j\omega C}} \\ \frac{V_o}{V_i} &= \frac{1}{1 + \frac{1}{j\omega RC}} \end{aligned}$$

Clearly, when $\omega \approx 0, V_o \approx 0$, and for $\omega \to \infty, V_o \to V_i$. Hence at high frequencies, most of the input voltage appears at the output making this circuit as a high pass filter.

The frequency response diagrams are depicted in Figure 5.6 showing low gain at low frequencies with gain approaching 1 for high frequencies. The phase plot varies from 90^0 to 0^0 with the cutoff frequency, $\omega = 1/T$, crossing at 45^0.

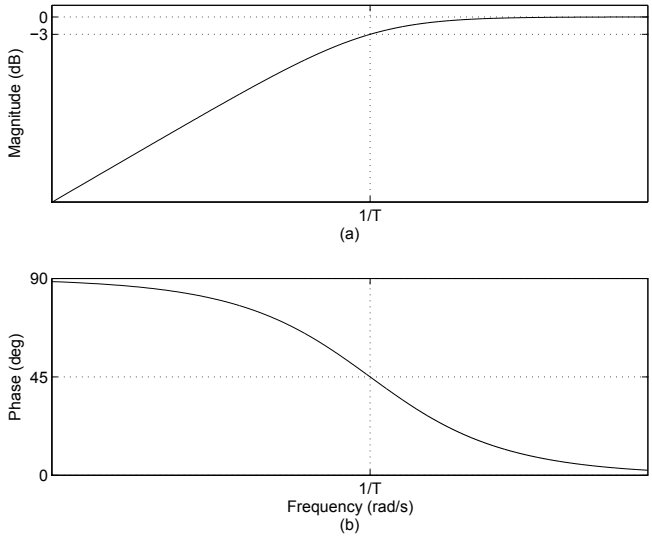


Figure 5.6: Frequency response of a high pass filter where (a) Amplitude response (b) Phase response.

5.3.3 Band Pass Filter

A band pass filter permits a certain *band* of frequencies to pass through the network which is adjusted by the designer. It is simply an amalgamation of a low pass and a high pass filter. If Figures 5.5(a) and 5.2(a) are joined in cascade, a high pass filter can be constructed as shown in Figure 5.7. In this case, there are two resistors and two capacitors whose values can be altered to tune the desired band.

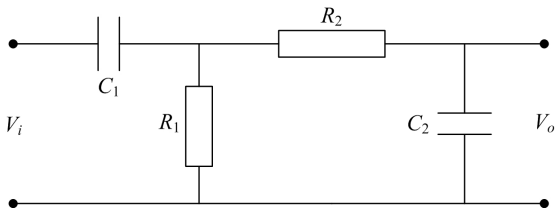


Figure 5.7: Schematic of a band pass filter.

The amplitude and phase response of a band pass filter is shown in Figure 5.8 highlighting the two cutoff frequencies $\omega 1$ and $\omega 2$. All frequencies within this range in an input signal will be allowed to pass unaltered through the circuit whilst any frequency outside these limits will be blocked or severely attenuated.

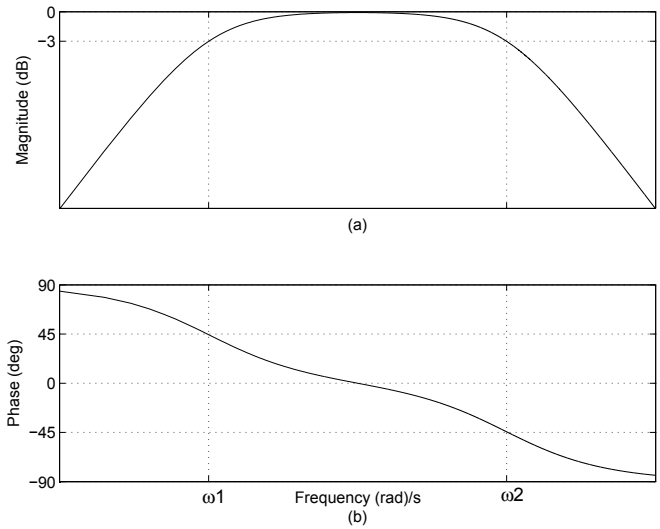


Figure 5.8: Frequency response of a band pass filter

5.4 Bode Plots

Bode plots are a graphical way to display the behaviour of a circuit over a wide range of frequencies. By plotting the amplitude and phase versus the logarithm of frequency, each unit of change on the ω axis is equal to a factor of 10 also called a *decade* of frequency. Also, there may be a wide distribution in the amplitude response over a specified range of frequencies. The usual way is to plot the amplitude in dB and phase in degrees or radians versus the logarithm of frequency. The frequency response diagrams depicted in Figures 5.4, 5.6 and 5.8 are all Bode plots.

Download free ebooks at bookboon.com

5.4.1 Approximate Bode Plots

Another advantage of using Bode plots is the easiness through which they can be sketched using a simple graphical technique which is particularly very useful for complicated transfer functions. In this method, the numerator and denominator polynomials are factorised which are then treated individually. Asymptotic plots are drawn for each first and second order factors by following some simple rules. The resultant Bode plot is obtained by simply adding all the individual plots. This is another rationale of employing logarithmic units since $\log(AB) = \log A + \log B$.

Consider the voltage transfer function of an electric circuit in factored form as follows

$$H(j\omega) = \frac{K(1 + j\omega T_1)}{(j\omega)^2(1 + j\omega T_2)(1 + \omega T_2)}$$

where K is a constant gain and T_1, T_2 and T_3 are the three time constants. The factor in the numerator is a *zero* whereas the two denominator factors are called the *poles*. The term $j\omega$ in the denominator is a second order integrator.

Let $A_1\angle\phi_1, A_2\angle\phi_2$ and $A_3\angle\phi_3$ represents the magnitudes and phases of the single zero and two poles of $H(j\omega)$ respectively. Then the combined amplitude and phase can be written in polar form as

$$\mathbf{H}(j\omega) = A\angle\phi$$

where

$$A = \frac{KA_1}{\omega^2 A_2 A_3} \text{ and } \phi = \angle\big(\phi_1 - \phi_2 - \phi_3 - 180^0\big) \tag{5.5}$$

In dB units, the gain can be specified as

$$A_{dB} = 20\log_{10} K + 20\log_{10} A_1 - 20\log_{10} \omega^2 - 20\log_{10} A_2 - 20\log_{10} A_3$$

Hence the individual plots can be simply added to find the resultant amplitude response. In the following, some simple rules are outlined to draw the approximate or asymptotic Bode amplitude and phase plots.

Amplitude Response The amplitude response curve is the magntiude of the given transfer function in dB versus the logarithm of frequency, ω in rad/s.

- For a constant gain, K, the amplitude is a horizontal line of magnitue $20\log_{10} K$ (Figure 5.9(a)).
- For a simple zero of order 1, the gain is 0 in the interval, $w \leq 1/T$ and increases at a rate of +20 dB/decade afterwards (Figure 5.10(a)).
- For a simple zero of order n, the gain is 0 in the interval, $w \leq 1/T$ and increases at a rate of $+20 \times n$ dB/decade afterwards.

Download free ebooks at bookboon.com

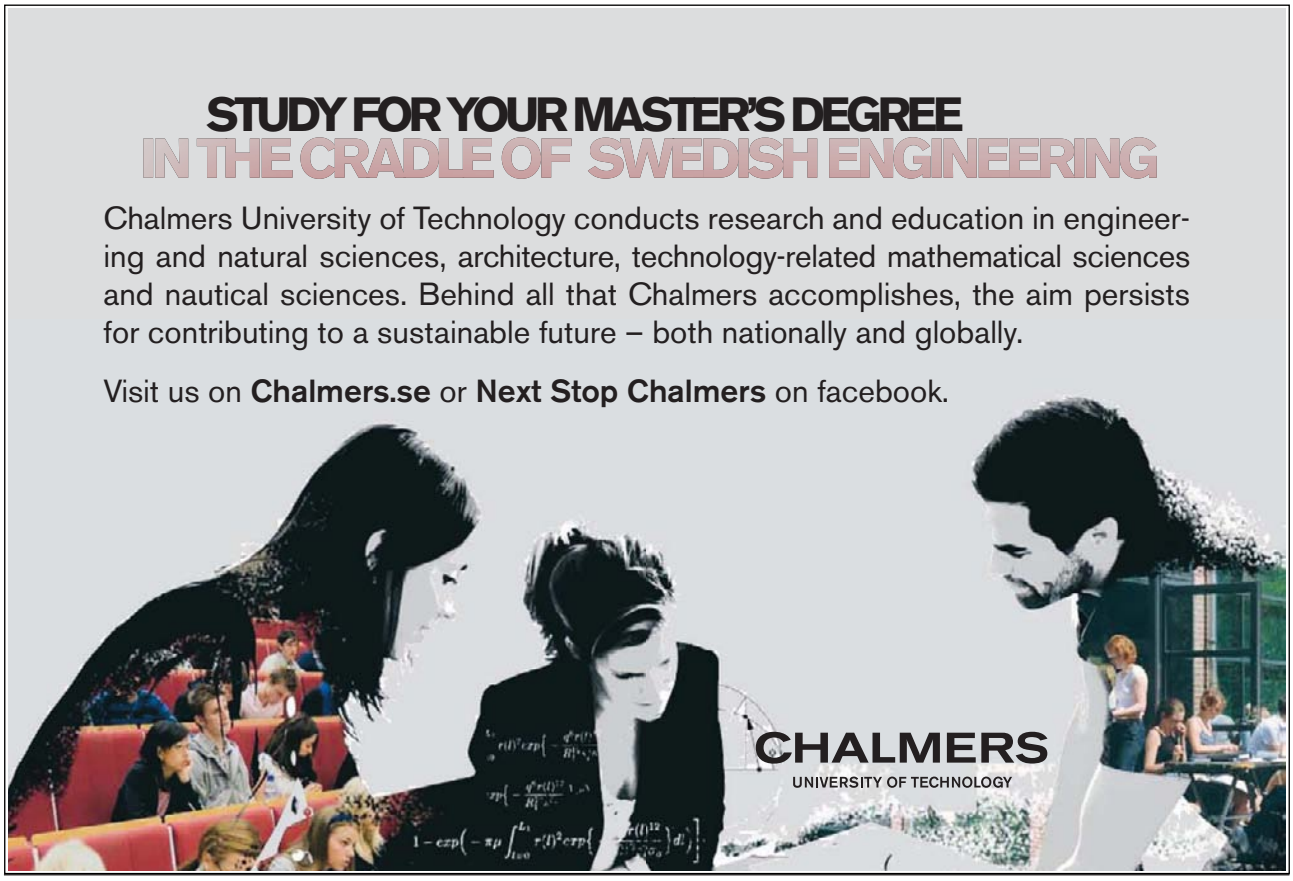
- For a simple pole of order 1, the gain is 0 in the interval, $w \leq 1/T$ and then decreases at a rate of -20 dB/decade (Figure 5.11(a)).
- For a simple pole of order n, the gain is 0 in the interval, $w \leq 1/T$ and then decreases at a rate of $-20 \times n$ dB/decade.
- For a differentiator of order n i.e. $H(j\omega) = (j\omega T)^n$, the magnitude is a constant ramp with a slope of $+20 \times n$ dB/decade crossing the 0 dB line at a break frequency of $w = 1/T$ rad/s (see Figure 5.12(a) for a first order differentiator).
- For an integrator of order n, the magnitude is a constant ramp with a slope of slope $-20 \times n$ dB/decade crossing the 0 dB line at a break frequency of $w = 1/T$ rad/s (see Figure 5.13(a) for a first order integrator).

Note that all amplitude response curves have either a slope of zero or $\pm 20 \times n$ dB/decade where n is the order of numerator or denominator factors.

Phase Response The phase response plot is the phase of the given transfer function in degrees or radians versus the logarithm of frequency, ω in rad/s.

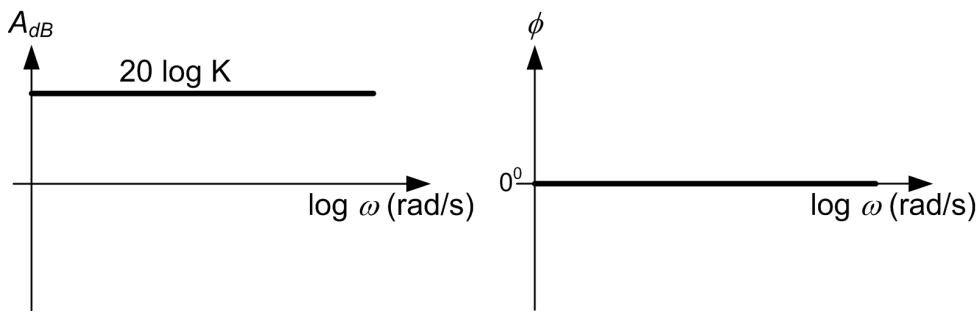
- For a constant gain, K, the phase response is a constant 0^0 horizontal line (Figure 5.9(b)).
- For a simple zero of order 1, the phase angle is 0^0 until $0.1/T$, then rises at the rate of 45^0/decade and settles at 90^0 for $\omega \geq 10/T$ (Figure 5.10(b)).

Please click the advert



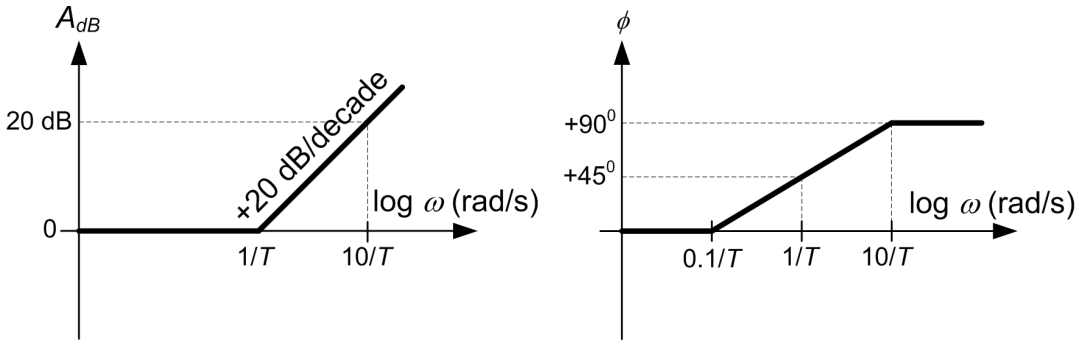
Download free ebooks at bookboon.com

- For a simple zero of order n, the phase is 0^0 until $0.1/T$, then rises at the rate of $45^0 \times n$/decade and settles at $90^0 \times n$ for $\omega \geq 10/T$.
- For a simple pole of order 1, the phase is 0^0 until $0.1/T$, then falls at the rate of -45^0 /decade and settles at -90^0 for $\omega \geq 10/T$ (Figure 5.11(b)).
- For a simple pole of order n, the phase is 0^0 until $0.1/T$, then drop-off at the rate of $-45^0 \times n$ /decade and settles at $-90^0 \times n$ for $\omega \geq 10/T$.
- For a differentiator of order n i.e. $H(j\omega) = (j\omega T)^n$, the phase plot is a constant line of $90^0 \times n$ (see Figure 5.12(b) for a first order differentiator).
- For an integrator of order n, the phase response is a horizontal line of $-90^0 \times n$ (see Figure 5.13(b) for a first order integrator). In Equation 5.5, the 180^0 term in the phase angle is due to the 2nd order integrator.



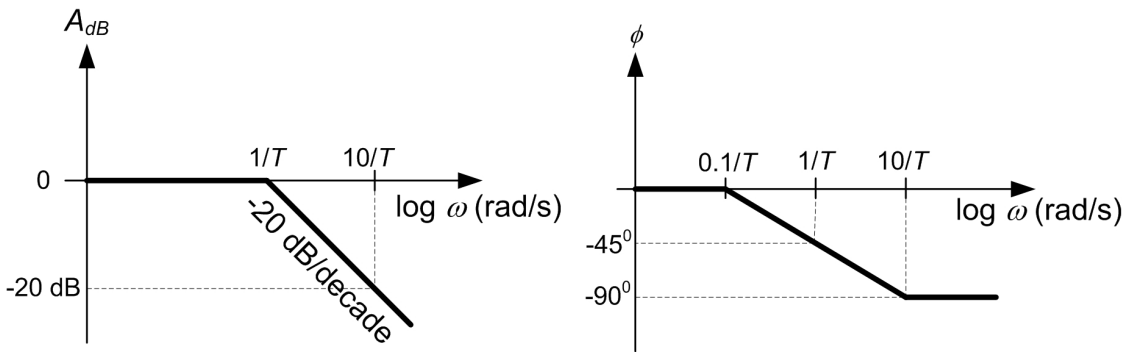
(a) Magnitude plot. (b) Phase plot.

Figure 5.9: Asymptotic Bode plots of a constant gain.



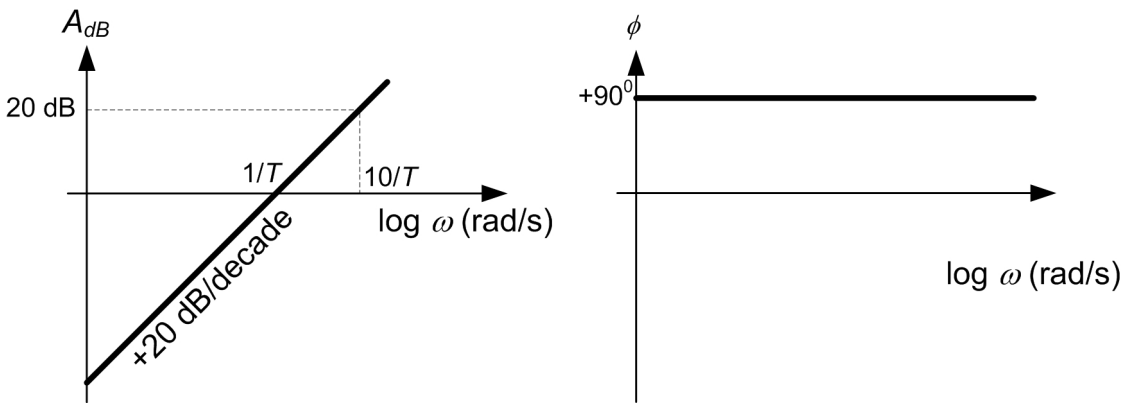
(a) Magnitude plot. (b) Phase plot.

Figure 5.10: Asymptotic Bode plots of a simple zero of order 1.



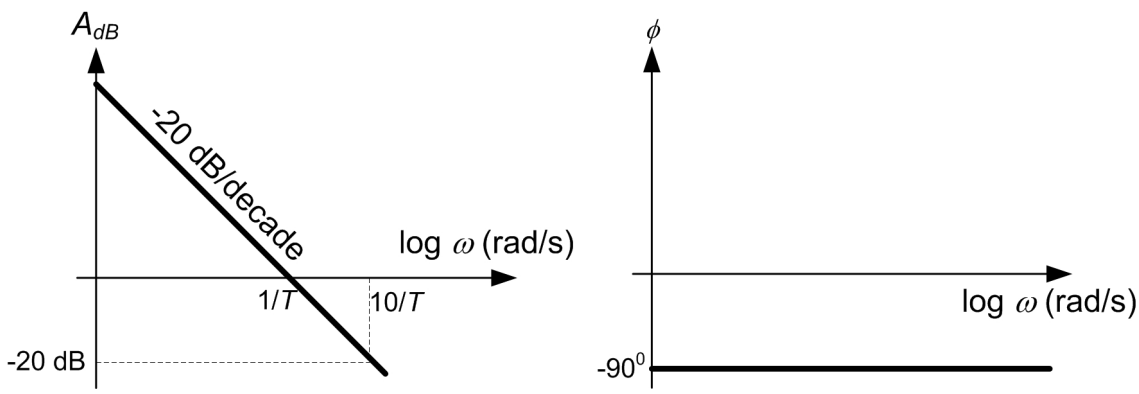
(a) Magnitude plot. (b) Phase plot.

Figure 5.11: Asymptotic Bode plots of a simple pole of order 1.



(a) Magnitude plot. (b) Phase plot.

Figure 5.12: Asymptotic Bode plots of a first order differentiator.



(a) Magnitude plot. (b) Phase plot.

Figure 5.13: Asymptotic Bode plots of a first order integrator.

Download free ebooks at bookboon.com

Example

Draw the approximate Bode plots of the following transfer function

$$H(j\omega) = \frac{10(1 + j\omega 2)}{(j\omega 5)(1 + j\omega 10)^2} \tag{5.6}$$

The above transfer function consists of a simple zero, a second order pole and an integrator. This can be rearranged in the standard form as below

$$H(j\omega) = \frac{(1 + j\omega/0.5)}{(j\omega/2)(1 + j\omega/0.1)^2}$$

where 0.5, 0.1 and 2 rad/s are the break frequencies of the simple zero, double pole and integrator respectively. From the rules outlined above, Bode plot can be drawn by inspection as demonstrated in Figure 5.14.

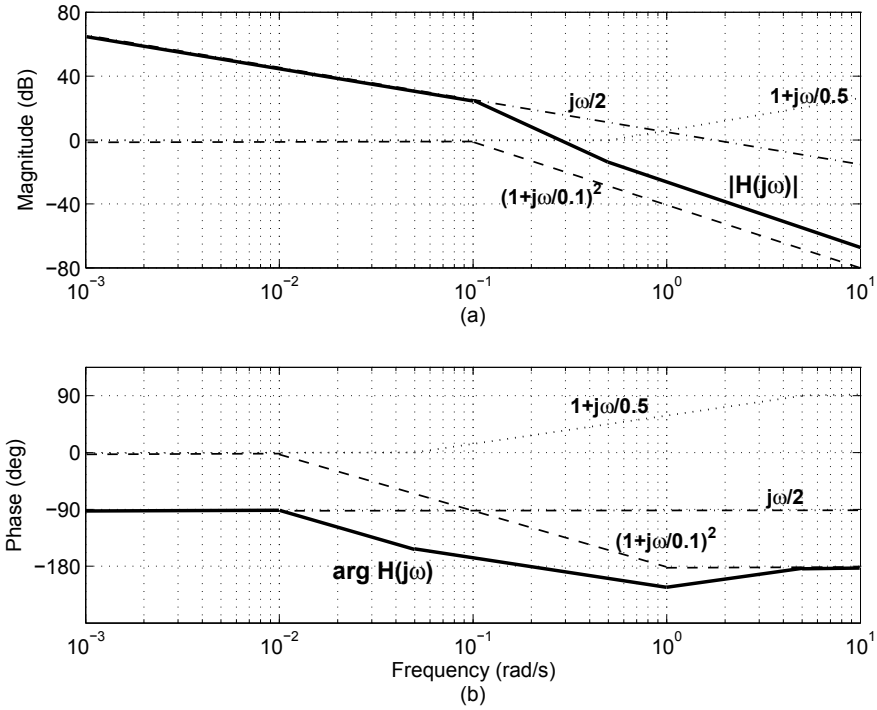


Figure 5.14: Approximate Bode plot for the transfer function in Equation 5.6 where (a) Amplitude response and (b) Phase response.

Appendix A

Cramer's Rule

Given two or more simultaneous system of equations, a convenient way to solve for the unknown variables is the application of Cramer's rule as explained below.

Consider three equations with unknown variables x_1, x_2 and x_3 as follows:

$$a_{11}x_1 + a_{12}x_2 + a_{13}x_3 = b_{11} \tag{A.1}$$

$$a_{21}x_1 + a_{22}x_2 + a_{23}x_3 = b_{21} \tag{A.2}$$

$$a_{31}x_1 + a_{32}x_2 + a_{33}x_3 = b_{31} \tag{A.3}$$

where a_{ij} and b_{i1} are constants and $i, j = 1, 2, 3$.

Please click the advert

GCHQ
it's an interesting world

Get under the skin of it.

Graduate opportunities
Cheltenham | £24,945 + benefits

One of the UK's intelligence services, GCHQ's role is two-fold: to gather and analyse intelligence which helps shape Britain's response to global events, and, to provide technical advice for the protection of Government communication and information systems. In doing so, our specialists – in IT, internet, engineering, languages, information assurance, mathematics and intelligence – get well beneath the surface of global affairs. If you thought the world was an interesting place, you really ought to explore our world of work.

TOP 100 · TOP GOVERNMENT EMPLOYER · INVESTORS IN PEOPLE

www.careersinbritishintelligence.co.uk

Applicants must be British citizens. GCHQ values diversity and welcomes applicants from all sections of the community. We want our workforce to reflect the diversity of our work.

Download free ebooks at bookboon.com

In matrix form, the above equations can be written as

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} b_{11} \\ b_{21} \\ b_{31} \end{bmatrix}$$

To determine x_1, x_2 and x_3, the following determinants are evaluated

$$\Delta = \begin{vmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{vmatrix}$$

$$\Delta_1 = \begin{vmatrix} b_{11} & a_{12} & a_{13} \\ b_{21} & a_{22} & a_{23} \\ b_{31} & a_{32} & a_{33} \end{vmatrix}$$

$$\Delta_2 = \begin{vmatrix} a_{11} & b_{11} & a_{13} \\ a_{21} & b_{21} & a_{23} \\ a_{31} & b_{31} & a_{33} \end{vmatrix}$$

$$\Delta_3 = \begin{vmatrix} a_{11} & a_{12} & b_{11} \\ a_{21} & a_{22} & b_{21} \\ a_{31} & a_{32} & b_{31} \end{vmatrix}$$

Then x_1, x_2 and x_3 can be found according to the following equations

$$\begin{aligned} x_1 &= \frac{\Delta_1}{\Delta} \\ x_2 &= \frac{\Delta_2}{\Delta} \\ x_3 &= \frac{\Delta_3}{\Delta} \end{aligned}$$