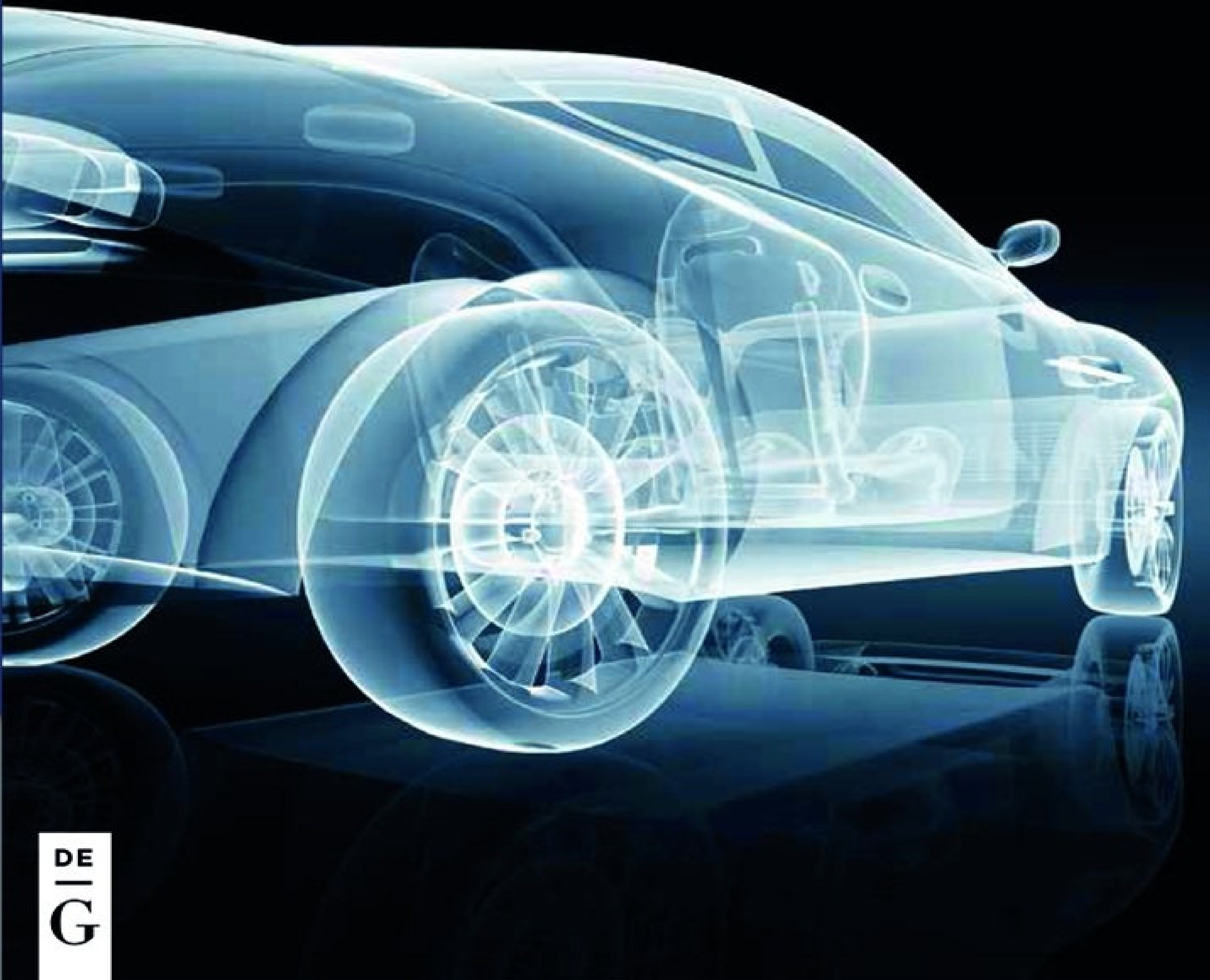
DE GRUYTER
OLDENBOURG

STUDIUM

Felix Hüning

FUNDAMENTALS OF ELECTRICAL ENGINEERING FOR MECHATRONICS



Felix Hüning

The Fundamentals of Electrical Engineering

Felix Hüning

The Fundamentals of Electrical Engineering

for Mechatronics

DE GRUYTER

Author
Prof. Dr. rer. nat. Felix Hüning
FH Aachen, University of Applied Sciences
Faculty for Electrical Engineering and Information Technology
Eupener Strasse 70
52066 Aachen
huening@fh-aachen.de

ISBN 978-3-11034991-7
e-ISBN (EPUB) 978-3-11-030840-2

e-ISBN (PDF) 978-3-11-034990-0

Library of Congress Cataloging-in-Publication Data A CIP catalog record for this book has been applied for at the Library of Congress.

Bibliographic information published by the German National Library
The German National Library lists this publication in the Deutsche Nationalbibliografie; detailed bibliographic data are available on the Internet at http://dnb.dnb.de.

Rosenheimer Straße 143, 81671 München, Germany
www.degruyter.com
Part of De Gruyter
Editor: Gerhard Pappert
Production editor: Tina Bonertz
Cover picture: Oksana Kostyushko/iStock/Thinkstock

This paper is resistant to aging (DIN/ISO 9706).

Epub-production: Jouve, www.jouve.com

Preface

Mechatronics is a multidisciplinary field and synergistic combination of electronic engineering, mechanical engineering and software engineering. Therefore knowledge of these disciplines is a must for mechatronic engineers as well as for mechanical and software engineers. They should have a basic understanding of electrical engineering to be able to work on mechatronic systems.

You can find mechatronic systems in nearly every area of modern life, from rather simple white goods to much more complex systems such as are found in satellites. I have used the practical applications found in modern vehicles as examples of complex mechatronic systems. Mechanical, electrical and software components are needed to realise complex functionalities like hybrid and electrical powertrains in hybrid and electric vehicles (HEV/EV), electric power steering or advanced driver assistant systems. Automotive applications are therefore ideal for demonstrating the application of the electrical topics introduced in this textbook.

The aim of this textbook is the introduction of basic concepts and laws of electrical engineering with an emphasis on mechatronic systems. It is based on a one-semester introduction course in the “Fundamentals of electrical engineering for mechatronics” at the University of Applied Sciences in Aachen, Germany.

The target group are students of either mechatronics, or other engineering topics who require a brief introduction to electrical engineering. In addition, non-electrical engineers working with mechatronic, or electrical systems can use this textbook as a quick start to understanding electrical engineering. The focus of this book is to help students understand electrical circuits and to learn different methods of how to analyse them.

The book starts with an introduction to the basic laws and concepts involved in solid state physics and electric fields. Then basic electric circuit concepts and components like resistors and sources are introduced. The main part of this book focuses on circuit analysis techniques. For example, DC circuits are analysed in chapter four using Kirchhoff's laws. DC analysis is extended in chapter five using structured methods of analysing circuits using nodal, or mesh analysis. In chapter six the operational amplifier is introduced as the first more complex electrical device and circuits with operational amplifiers are analysed.

After DC analysis, time-dependent circuits are analysed. Chapter seven starts with an introduction to capacitors and inductors as energy storage elements. These elements are used to build circuits with transient behaviour. Afterwards AC circuits are analysed using complex AC analysis.

In chapter eight simple circuits are combined into more complex building blocks. The analysis of complex circuits can be reduced to the analysis of simpler sub-circuits using the concept of building blocks. Chapter nine deals with AC power and in chapter 10 oscillating circuits are introduced.

More complex circuit elements are part of chapter 11. It includes semiconductor devices like diodes, biopolar junction transistors and MOSFETs. The textbook finishes with a short introduction to the important field of circuit simulation.

In addition to this theoretical introduction using a textbook, exercises are very important to gain a deeper understanding of the subjects. Exercises and solutions to each of the chapters can be found online under www.degruyter.com.

Last but not least, I would like to thank Prof. Dr. Martin Ossmann for discussing the technical details and for his very helpful feedback as well as to Gary Evans for editing the text and to Caroline Huertgen for her support with the images.

Table of Contents

1 The fundamentals of solid-state physics

1.1 Charge carriers, crystal structure and conductivity

You should be familiar with some very basic concepts of matter and electrical quantities such as charge, current and voltage in order to describe the operation of electronic circuits. Matter consists of atoms made of a nucleus which is orbited by negatively charged electrons. The nucleus itself is made up of positively charged protons and neutrons without an electrical charge. Electric charge is measured in coulombs (C). The smallest fundamental unit of electrical charge is a physical constant called the elementary positive charge:

$$e = 1.602 \cdot 10^{-19} C$$

All charges are an integer multiple of this elementary charge. The proton has a charge of +e while the electron has a charge of –e.
As a consequence of quantum mechanics, the electrons of a single isolated atom (not interacting with other atoms) can occupy only discrete energy levels (atomic orbitals), see Fig. 1.1.

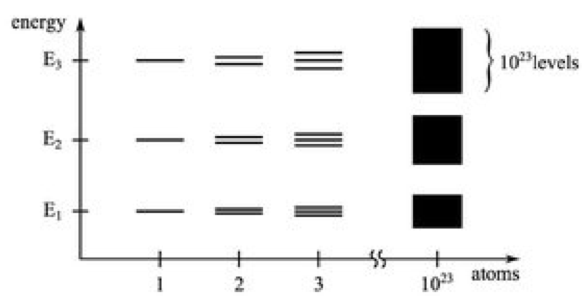


Fig. 1.1: Splitting of discrete energy levels into energy bands for increasing number of

atoms.

If several atoms form a molecule by chemical bonding the atomic orbitals split into separate molecular orbitals with different energy levels. Fig. 1.1 shows this splitting of energy levels: For an H_2 molecule with two atoms the energy levels of the single atoms split into two energy levels with the lower energy level being occupied by two electrons. In general the outermost electrons (valence electrons) can participate in the formation of chemical bonds with other atoms to form molecules, in solid, liquid or gaseous states.
When a large number of atoms form a solid the number of orbitals (proportional to the number of valence electrons) becomes very large and the difference in energy between these orbitals becomes very small. In consequence solids (with about 10^{23} or more atoms) show continuous energy bands rather than discrete energy levels. The energy bands can overlap or are separated by intervals of energy without orbitals (electrons within the solid cannot have these energies). These forbidden energy intervals are called band gaps and the total of bands and band gaps is called the band structure.
Fig. 1.2 shows a simplified diagram of the electronic band structure of crystalline solids on the right. The shape of the band structure depends on the atoms forming the solid and its crystal structure.

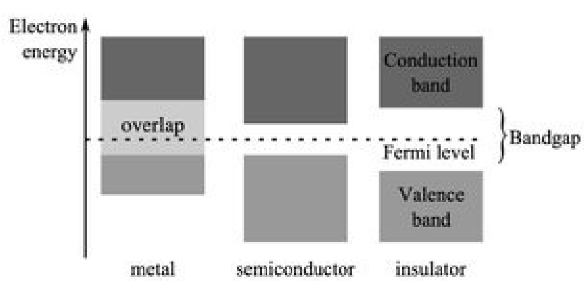


Fig. 1.2: The electronic band structure of solids: metal (left); semiconductor (center); insulator (right).

The electric properties of a solid are mainly determined by the band structure around the so-called Fermi level (see Fig. 1.2) where the Fermi level in a simplified image divides the band structure into a region at a low energy level that is occupied by electrons and a region at a higher energy level that is empty. The highest (almost fully) occupied band is called the valence band and the lowest (almost) empty band is called conduction band. Conductivity occurs if the conduction band is partly filled by electrons or electrons are missing in the valence band.

1.2 The electrical properties of solids

Electrical properties can be distinguished depending on the presence and size of a band gap at the Fermi level.

Metals

The valence and conduction band overlap (the Fermi level lies within this overlapping band) or the Fermi level lies within the conduction band (not shown) and therefore the band is partly filled with electrons regardless of temperature. These electrons form a “sea” of practically free electrons moving in the background of the positively charged crystal structure (Fig. 1.3). The electron density is of the same magnitude as the density of the atoms. For example the density of free electrons in copper is about $8 \cdot 10^{22}$ electrons per cm^{-3}. The conductivity is very high as the electrons can easily absorb energy. Conductivity decreases with increasing temperature. Classic examples are silver, copper and iron (see Tab. 1.1).

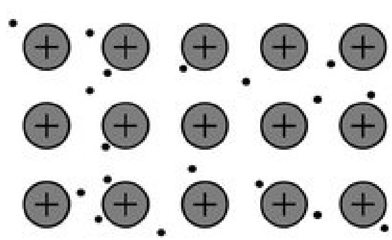


Fig. 1.3: The crystal structure of a metal: positively charged atomic cores surrounded by delocalized free electrons.

Insulators

In an insulator the Fermi level is located within a large band gap. The valence band (at absolute zero) is fully occupied and the conduction band is empty, resulting in no conductivity. At higher temperatures electrons can be excited to the conduction band due to thermal energy (leaving a hole in the valence band), but at reasonable temperatures the number of excited electrons is negligible and there is no conductivity.
Examples of insulators are glass or plastic materials, see Tab. 1.1.

Semiconductors
Like for insulators the Fermi level for pure semiconductors, (also known as intrinsic semiconductors), is within a band gap of width ΔE, but this time the band gap is smaller. Semiconductors are isolators at absolute zero. Thanks to the smaller band gap, electrons can be excited to the conduction band more easily due to thermal energy. As the excitation of electrons is a thermal effect the number of intrinsic electrons in the conduction band n_i is strongly temperature dependent:

$$n_i \sim T^{\frac{3}{2}} e^{-\frac{\Delta E}{2k_B T}}$$

Here k_B=1.38…·10^{-23} J/K which is the Boltzmann constant.
When an electron is excited to the conduction band this electron is missing in the valence band. This missing electron is called an electron hole, or defect electron and also contributes to conductivity. The density of holes is the same as the density of excited electrons. The conductivity of semiconductors is higher than that of insulators, but still significantly lower than that of metals.

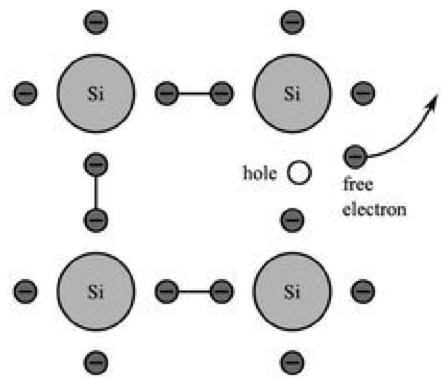


Fig. 1.4: The crystal structure of silicon.

As every excited electron generates a hole the density of electrons n_0 and holes p_0 is the same and corresponds to the intrinsic charge carrier density n_i:

$$n_0 = p_0 = n_i$$

The opposite of the generation of electron-hole pairs is called recombination. The recombination rate depends on the carrier density of electrons and holes. The rate of generation and recombination of electron-hole pairs is temperature dependent. In thermal equilibrium both rates are the same and the number of free electron-hole pairs is constant at the given temperature. The equality of the two rates leads to the mass action law:

$$n_0 \cdot p_0 = n_i^2$$

The product of the charge density of the free electrons and holes equals the square of the intrinsic charge carrier concentration. Mass action law also holds true for doped semiconductors.

The most important semiconductor used in semiconductor devices is silicon.

The band gap of silicon:

> ~1.1 eV (where 1 eV = $1.602\ldots\cdot 10^{-19}$ J is a measure of small scale energy)

Atom density:

$\sim 5 \cdot 10^{22}$ atoms per cm^{-3}

The number of intrinsic charge carriers due to thermal activation at room temperature (293 K):

$\sim 1.5 \cdot 10^{10}$ electrons per cm^{-3}
$\sim 1.5 \cdot 10^{10}$ holes per cm^{-3}

This example shows that the carrier density of intrinsic semiconductors like silicon is significantly lower than of metals. Therefore intrinsic semiconductors are rather poor electrical conductors.
Other examples of intrinsic semiconductors are germanium (band gap about 0.7 eV), SiC (band gap about 2.3 eV) or GaN (band gap about 3.2 eV).

Doped semiconductors
The electron density and conductivity of semiconductors can be varied by so called doping. By doping small amounts of silicon atoms (with four valence electrons per atom) are replaced by other atoms with a different number of valence electrons in a very controlled manner. In p-type semiconductors the number of valence electrons is less than 4 (e.g. boron, three valence electrons) and holes are easily generated as majority carriers. In n-type semiconductors the number of valence electrons is more than four (e.g. arsenic, five valence electrons) and electrons as majority carriers are easily generated. In this way the electron (or hole) density of the semiconductor can be changed to a desired number.
Fig. 1.5 shows the crystal structure of silicon doped with arsenic (left side) as a donor. The binding of the additional electron to the arsenic atom is rather weak and, as shown in the band

structure in Fig. 1.5 (right), the energy levels of this electron are within the band gap, just 0.049 eV below the conduction band. This electron can easily be excited to the conduction band even at low temperatures and can increase conductivity. Note that no hole is generated in the valence band.

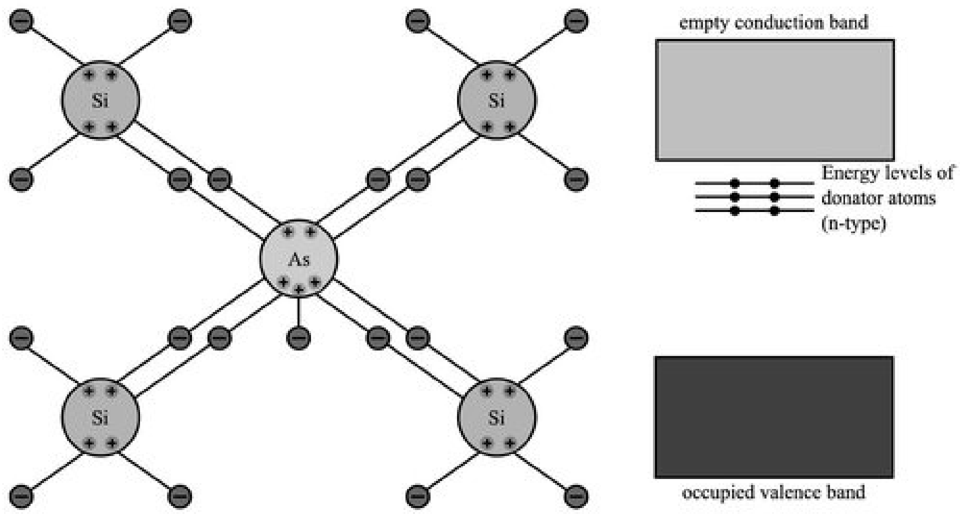


Fig. 1.5: The crystal structure of arsenic-doped silicon, n-type semiconductor (left) and the band structure of a n-type semiconductor showing the donator’s energy levels within the band gap (right).

The temperature dependence of a doped semiconductor is depicted in Fig. 1.6. Even at low temperatures (some 10 K) all impurity atoms are ionized generating electrons in the conduction band (n-type semiconductors) or holes in the valence band (p-type semiconductors). In a wide temperature range around room temperature the carrier density is almost constant n_0 and equals the density of impurity atoms N_D. At higher temperatures intrinsic carriers are increasingly generated.

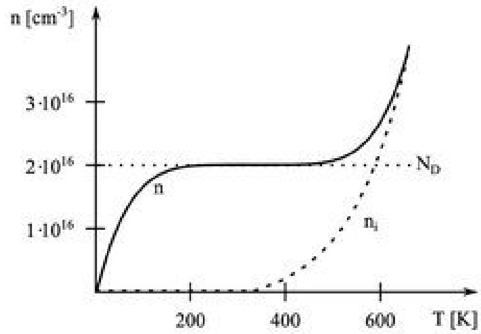


Fig. 1.6: The carrier density n of an n-type semiconductor as a function of temperature: solid line = total carrier density n, dotted line = intrinsic carrier density n_i; N_D is the density of impurity atoms.

As the mass action law is also valid in doped semiconductors the density of minority charge carriers can be calculated, e.g. for n-type semiconductors:

$$p_0 = \frac{n_i^2}{n_0} = \frac{n_i^2}{N}$$

Tab. 1.1 lists some examples for metals, semiconductors and insulators and corresponding conductivity values.

Tab. 1.1: The conductivity values and electrical classification for certain materials.

Classification	**Material**	**Specific conductivity σ [S/m]**
Metal	Silver	$61 \cdot 10^6$
	Copper	$58 \cdot 10^6$
	Iron	$10 \cdot 10^6$
Semiconductor	Germanium	1.45
	Silicon (pure)	$252 \cdot 10^{-6}$
Insulator	Plastic material	$< 10^{-9}$
	Glas	$< 10^{-9}$
	Diamond	$< 10^{-9}$

2 Fundamental electrical laws

2.1 The basics of electric field theory

Any electric charge is responsible for an electric field that surrounds the charge. Forces are exerted on each other by charges due to this electric field. Charges with same sign repel whereas charges with an opposite sign attract. The force of charge Q_2 on another charge Q_1 can be expressed by Coulomb's law:

$$\vec{F}_{21} = \frac{Q_1 \cdot Q_2}{4\pi\varepsilon R^2}\vec{a}_{21} = \frac{Q_1 \cdot Q_2}{4\pi\varepsilon_0\varepsilon_r R^2}\vec{a}_{21}$$

As shown in Fig. 2.1 R is the separation of the two point charges Q_1 and Q_2. $\vec{a}_{21}$ is the unit vector along the line joining the charges. The parameter $\varepsilon = \varepsilon_0 \cdot \varepsilon_r$ is the permittivity of the surrounding material between Q_1 and Q_2. Constant ε_0 is the permittivity of free-space ($\varepsilon_0 = 8.854\ldots\cdot 10^{-12}$ As/Vm) and ε_r is the relative permittivity of the material. For a vacuum the relative permittivity is equal to 1.

Of course Q_1 exerts a force of the same magnitude, but opposite direction on Q_2. The force is directly proportional to the product of the charges and inversely proportional to the square of the distance.

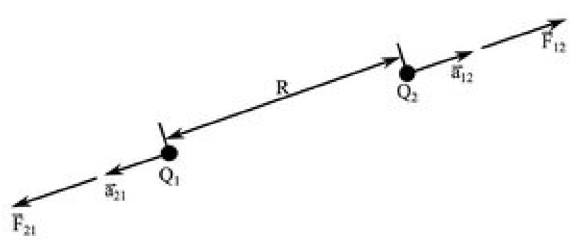


Fig. 2.1: An illustration of Coulomb's law

The electric field strength $\vec{E}$ describes the force of the electric field onto charged particles. Consider a very small charge Q_1 in the electric field of Q_2. The force of Q_2 on small charge Q_1 is given by Coulomb's law and the electric field strength is:

$$\vec{E} = \frac{\vec{F}_{21}}{Q_1} = \frac{Q_2}{4\pi\varepsilon R^2}\vec{a}_{21}$$

The unit of the electric field strength is N/C = V/m.

The electric field strength is parallel to the force for positive charge Q_1 and antiparallel for negative charge Q_1. The electric field strength can be depicted by electric field lines as shown in Fig. 2.2. The field lines normally start at the positive and end at the negative charge, the direction is indicated by arrows. The strength of the field is given qualitatively by the separation of the field lines: the closer the lines the higher the strength.

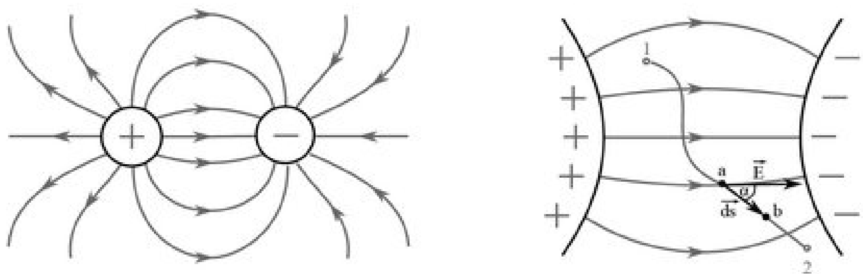


Fig. 2.2: Electric field lines for two charges (left) and the voltage in an inhomogeneous electric field (right)

2.2 Electric potential and voltage

When electrical forces act on a particle with charge Q_1, it will possess potential energy. To move the particle in an electric field of strength $\vec{E}$ along a distance $d\vec{s}$ you have to expend (or you gain) the energy dE:

$$d\vec{E} = \vec{F} \cdot d\vec{s} = Q_1 \cdot \vec{E} \cdot d\vec{s}$$

Do not mix up electrical field strength $\vec{E}$ (it is a vector field) with the electrical energy E which is a scalar. If a positive charge is moved in the direction of the electric field it gains energy (dE > 0), if it is moved in opposite direction you have to expend energy (dE < 0). If moved perpendicular to the field on the lines of same potential the energy is zero. These lines of same potential are called equipotential lines.
The electrical voltage is now defined as the energy needed to move the particle in an electric field from point A to B divided by the charge Q_1:

$$U_{AB} = \frac{E_{AB}}{Q_1} = \int_A^B \vec{E} \cdot d\vec{s}$$

The electrical voltage is a scalar quantity and the unit for the voltage is the volt (V).
In a homogeneous electrostatic field (the electric field strength has the same size and direction everywhere) the voltage between A and B with distance d is simply:

$$U_{AB} = E \cdot d$$

The voltage in case of electro static fields is independent of the path the particle takes within the field when moving from point A to B (path P_1 or P_2):

$$\int_A^B \vec{E} \cdot d\vec{s}\Big|_{P1} = \int_A^B \vec{E} \cdot d\vec{s}\Big|_{P2} = -\int_B^A \vec{E} \cdot d\vec{s}\Big|_{P2}$$

Or in other words: The line integral along any closed path (loop) is always zero:

$$\int_A^B \vec{E} \cdot d\vec{s} + \int_B^A \vec{E} \cdot d\vec{s} = \oint \vec{E} \cdot d\vec{s} = 0$$

This equation is one of Maxwell’s equations for the electrostatic field in integral form. It can be rewritten in differential using Stoke's integral theorem.

Excursus: Stoke's integral theorem

> For an arbitrary vector field $\vec{a}$ the integral along a closed loop around any arbitrary surface $\vec{A}$ is:

$$\oint \vec{a} \cdot d\vec{s} = \int_{\vec{A}} rot \left(\vec{a} \cdot d\vec{A}\right)$$

Applying Stoke's integral theorem to the Maxwell equation given above yields:

$$\oint \vec{E} \cdot d\vec{s} = \int_{\vec{A}} rot \left(\vec{E} \cdot d\vec{A}\right) = 0$$

As this equation holds true for any surface $\vec{A}$ the second integrand has to be zero:

$$rot\, \vec{E} = 0$$

This differential form of Maxwell's equation states that the electrostatic field has no curls. The electric potential is the voltage at any point referred to a fixed reference point. It is a scalar field. As the voltage is always referred to two locations (A and B) the voltage can be written in terms of the electric potential as:

$$U_{AB} = \int_A^B \vec{E} \cdot d\vec{s} = \Phi_A - \Phi_B = -\int_A^B d\Phi$$

This equation is true for any path between to arbitrary locations and therefore yields the correlation of electric field and electric potential:

$$\vec{E} = -grad\Phi$$

The electric field is given by the gradient of the electric potential.

2.3 Displacement field and electric flux

As given above Coulomb's law and the electric field of a charge is strongly dependent on the surrounding material: by changing the surrounding material the electric field is changed. This dependence is given by the relative permittivity ε_r. It is a material parameter without a unit. In the simplest case ε_r is just a constant scalar, but in general it can also be non-linear (ε_r depends on $\vec{E}$ or $\vec{D}$ like in ferroelectric materials such as $BaTiO_3$) or anisotripic (ε_r is a tensor and $\vec{E}$ and $\vec{D}$ are not parallel). These three cases are depicted in Fig. 2.3.

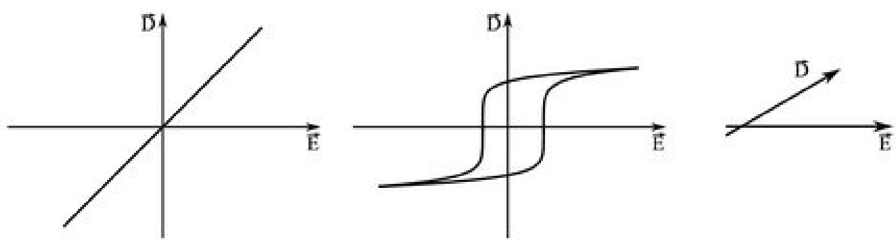


Fig. 2.3: The relationship between electric and displacement field : linear with constant ε_r (left); non-linear with hysteresis shape (center); anisotropic (right).

If the material is an electrical insulator the material is called a dielectric. An external electric field causes a shift of electric charges inside the dielectric as shown in a simplified picture in Fig. 2.4. The origin of the dielectric polarization can be a displacement polarization of the atoms, or an orientation polarization by the alignment of permanent dipoles with respect to the external field. In both cases the polarization field $\vec{P}$ has an opposite direction compared with the external field and the external field is weakened.

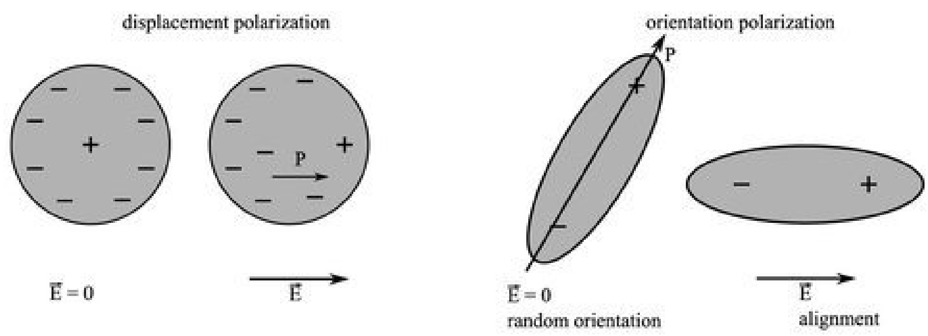


Fig. 2.4: The displacement polarization (left) and orientation polarization (right).

To get rid of the material dependence the electric displacement field can be defined by the superposition of the external electric field and the polarization field:

$$\vec{D} = \varepsilon\vec{E} = \varepsilon_r\varepsilon_0\vec{E} = \varepsilon_0\vec{E} + \vec{P}$$

The displacement field or electric flux density describes the density of field line independent of the surrounding material. The unit for the displacement field is C/m^2.
For a point charge q in the origin of the coordinate system the displacement field at location $\vec{r}$ yields:

$$\vec{D}(\vec{r}) = \frac{q}{4\pi r^2}\vec{a}_{21} = \frac{q}{4\pi r^3}\vec{r}$$

As the displacement field describes the density of field lines an electric flux is associated with it. The electric flux ψ is given by the integral of the displacement field perpendicular to any arbitrary surface $\vec{A}$:

$$\Psi = \iint_{\vec{A}} \vec{D}\cdot d\vec{A}$$

The unit of the electrical flux is C. If the displacement field is homogeneous and the surface $\vec{A}$ is perpendicular to the field the electric flux is just:

$$\Psi = D\cdot A$$

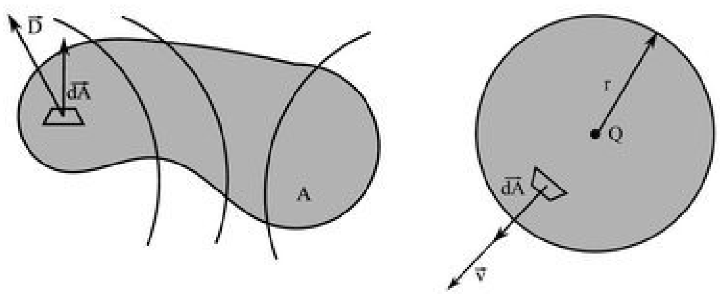


Fig. 2.5: Electric flux $\vec{D}$ through a surface $\vec{A}$ with surface element d $\vec{A}$

The displacement field of a point charge is given above and the electric flux for any arbitrary surface is:

$$\Psi = \iint_{\vec{A}} \frac{q}{4\pi r^3} \vec{r} \cdot d\vec{A}$$

Integrating over a closed surface with the point charge inside gives the total electric flux of the point charge. This closed surface integral is in particularly simple for a sphere (with radius r, see Fig. 2.5) as the displacement field everywhere is perpendicular to the sphere.

$$\Psi = \oiint_{sphere} \vec{D} \cdot d\vec{A} = \frac{q}{4\pi r^2} \cdot \oiint_{sphere} dA = \frac{q}{4\pi r^2} \cdot 4\pi r^2 = q$$

The total electric flux through a closed surface like the sphere is equal to the point charge inside the surface. This result is valid for any surface. In addition any arbitrary charge distribution can be built by point charges and the total charge Q within the closed surface is given by the charge density $\rho(\vec{r})$:

$$Q = \iiint_{volume} \rho(\vec{r}) dV$$

Second Maxwell's equation:
For any arbitrary charge distribution within a volume V the total electric flux through the closed surface $\vec{A}$ around the volume is equal to the total charge:

$$\Psi = \oiint_{sphere} \vec{D} \cdot d\vec{A} = Q = \iiint_{volume} \rho(\vec{r}) dV$$

Excursus: Gauss's integral theorem

For an arbitrary vector field $\vec{a}$ the integral over a closed surface around an infinitesimal volume V is:

$$div\ \vec{a}(\vec{r}) = \lim_{V \to 0} \frac{\iint_A \vec{a}(\vec{r}) dA}{V}$$

The flux through a closed surface of an infinitesimal volume equals the divergence of the original vector field $\vec{a}(\vec{r})$.

Applying Gauss's theorem to the second Maxwell's equation yields its differential form:

$$div\ \vec{D}(\vec{r}) = \rho(\vec{r})$$

Charges are the sources of the displacement field.

2.4 Electric current and current density

The rate of movement of net positive charge dq(t) per unit of time dt through a cross section of a conductor is known as current:

$$i(t) = \frac{dq(t)}{dt}$$

For time-dependent values lowercase letters are used, for time-independent values capital letters. The SI basic unit for current is the Ampere (A) where 1 Ampere corresponds to the charge flow of 1 C within 1 second:

$$1A = 1\frac{C}{s}$$

As we have seen in chapter 1.2 free electrons are the exclusive charge carriers in most metallic conductors such as copper wires. Since the charge of electrons is negative and since the direction designated for the current, as given above, is that of the net positive charge, the charges in the wire thus move in opposite direction to the current designation, see Fig. 2.6.

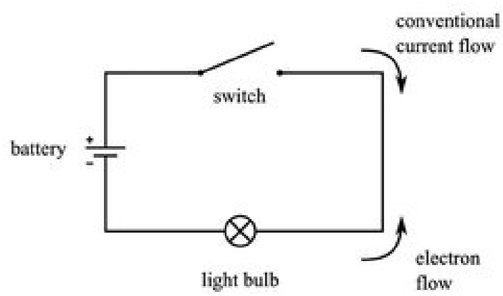


Fig. 2.6: The definition of conventional (technical) current flow (flow of positive charge carriers) and the direction of electron flow.

If a current I is uniformly distributed across a cross-section of a conductor A (like a wire) the current density J is:

$$J = \frac{I}{A}$$

The unit for the current density is A/m^2 or (more realistic) A/mm^2.
To calculate the flow speed or drift velocity v of electrons we need the time t an electron needs to move for a distance Δl. In other words t is the time it takes for the charges in the grey volume of Fig. 2.7 to pass through cross-section A. The charge Q in the grey volume inside the blue volume is:

$$Q = e \cdot n \cdot A \cdot \Delta l$$

Here e is the elementary charge and n is the electron density of the conductor (e.g. $8.5 \cdot 10^{19}$ mm^{-3} for copper). Corresponding current is:

$$I = \frac{Q}{t} = e \cdot n \cdot A \cdot \frac{\Delta l}{t} = e \cdot n \cdot A \cdot v$$

Finally the drift velocity yields:

$$v = \frac{I}{e \cdot n \cdot A}$$

Drift velocity is rather small (~1 mm/s), depending on the current, cross-section and electron density. Nevertheless the cause of the current propagates with about the speed of light.

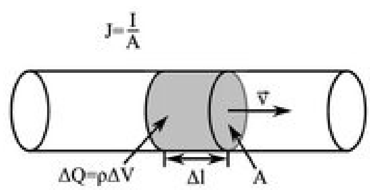


Fig. 2.7: The current density J for a current flowing in a wire with cross-section A

3 Fundamental circuit elements

3.1 Electric circuits

A collection of interconnected electronic components such as voltage and current sources, resistors, capacitors and other active and passive elements that has at least one closed path in which current may flow is called an electric circuit or network (Fig. 3.1). The behavior of these networks can be determined using so-called circuit analysis, either time-independent or time-dependent.

To make circuit analysis easier the so called lumped element model is used. This model simplifies a circuit in a way that the properties of the circuit, like resistance, inductance and sources, are concentrated into idealized electrical components. These idealized components (resistors, capacitors, inductors, etc.) are connected by perfectly conducting wires. In addition, the dimensions of the circuit have to be much smaller than the circuit's operating wavelength, or, in other words, the time it takes for signals to propagate around the circuit can be ignored.

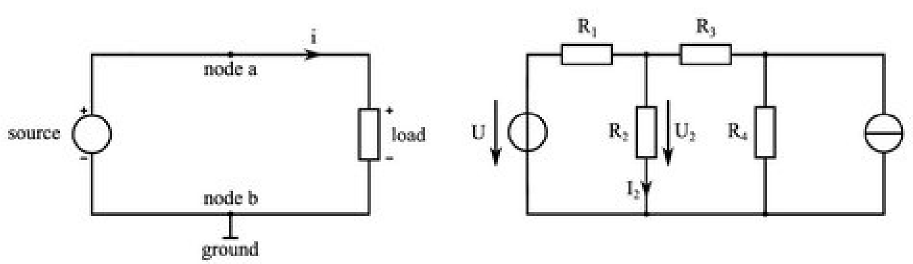


Fig. 3.1: Simple electric circuits or networks.

There are two types of elements in a circuit: sources and loads. A source usually supplies energy to the circuit. It is a force that drives the current through the circuit, like a battery or a generator. When the current flows out of the positive terminal of an electric source, it implies that nonelectrical energy has been

transformed into electrical energy. In contrast a load absorbs the energy supplied by a source. The current delivered by the source passes through the load. When current flows in the direction of a voltage drop, it implies that electrical energy is transformed into nonelectrical energy.
In a circuit, electrical sources and loads may usually be easily distinguished by a comparison of their current direction and voltage drop polarities:

- – Electrical source: voltage polarity opposite to technical current flow (voltage rise)
- – Electrical load: voltage drop parallel to technical current flow (voltage drop)

A source-load combination is depicted in Fig. 3.1. A node in a circuit is a point where two or more components or devices are connected together. A branch is a part of a circuit containing only one component between two nodes. A loop or mesh is a closed path through a circuit in which no electric element or node is encountered more than once. A mesh that contains no other meshes is called an essential mesh. Both nodes and meshes play a major role in circuit analysis.

3.2 Consumer and generator system

Both current and voltage have an orientation in electric circuits. As seen above, the current flows in the direction of the positive charge carriers from the positive terminal of a voltage source through the circuit to the negative. Voltages usually point from the positive to the negative terminal of a source as shown in Fig. 3.2.

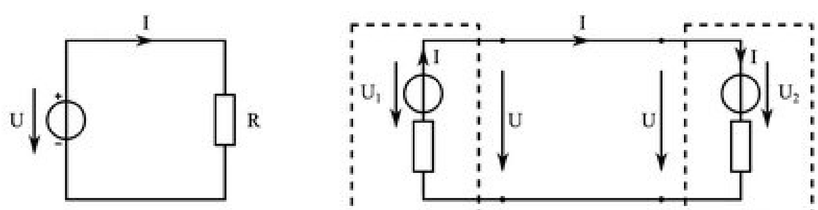


Fig. 3.2: The direction of current and voltage.

Consider now the circuit on the right of Fig. 3.2. The direction of current depends on voltages of the two sources and the direction indication can be chosen arbitrarily at the beginning. Consider voltage U_1 is higher than U_2, then the current flow is as shown. For the right part of the circuit current and voltage are parallel and the power $P = U \cdot I$ is dissipated. For this reason it is called a consumer system when current and voltage are parallel. In the left part current and voltage are antiparallel. This behavior is called generator system as this part provides power $P = -U \cdot I$.
During circuit analysis it is common to use the consumer system for resistances to be able to use Ohm's law in the way given below without a minus sign.
Before starting circuit analysis, basic elements like current and voltage sources and resistors are introduced.

3.3 Voltage sources

Any electric circuit needs at least one point where charge carriers are driven by a force. This point is called the source. The driving force of voltage sources is the electrical voltage. Voltage sources can be constant or may be a function of time as depicted in Fig. 3.3.

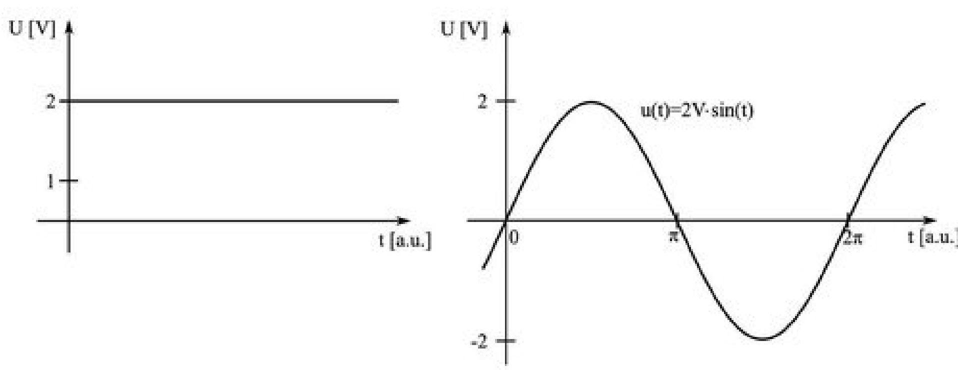


Fig. 3.3: Constant (left) and time-dependent (right) voltage source.

Voltage sources can be based on different physical and chemical principles, for example electrochemical voltage sources (e.g. batteries) and electro-mechanical voltage sources (e.g. generators).

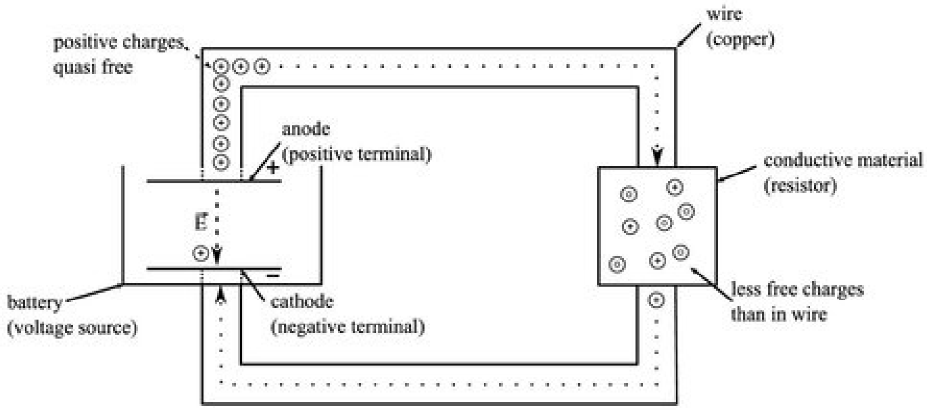


Fig. 3.4: A simple electric circuit with an ideal voltage source.

The electromagnetic force within the sources causes current to flow as soon as the loop is closed. The current flows in a closed loop from one terminal of the source to the other. Therefore current lines do not have a starting or an ending point. The positive terminal is called the anode whereas the negative terminal is called the cathode. In case there are both positive and negative charges inside a conductor these different charge carriers flow in opposite directions. As already mentioned above we need a definition for the direction of the current flow:

> The direction of a current is defined to be same as the direction of the positive charges and opposite to the direction of negative charges.

This definition implies that flow of electrons in metallic conductors is opposite to the direction of the current.

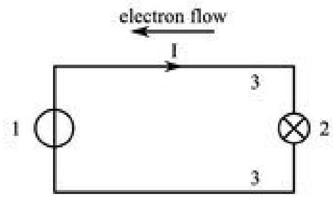


Fig. 3.5: A simple electric circuit indicating the direction of current flow I and the direction of electrons.

Ideal voltage sources

For an ideal voltage source the voltage at the terminals is independent of the load connected to the terminals and the current. It is called a DC source if the voltage is time-independent as the current is a direct current in this case. If the voltage of the source is a function of time (like a sinusoidal voltage source) it is called an AC source as it produces an alternating current.

Instead of this physical model the following graphical symbols are used for ideal voltage sources in electronic schematics:

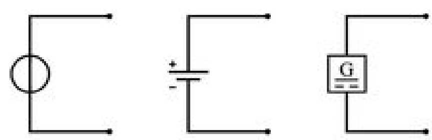


Fig. 3.6: A simple notation of voltage sources: general symbol (left), electrochemical symbol (battery, center), DC generator (right).

The definition of an ideal voltage source (voltage independent of load connected to the source) implies that it is not permitted to connect two (or more) ideal voltage sources in parallel to the same terminal.

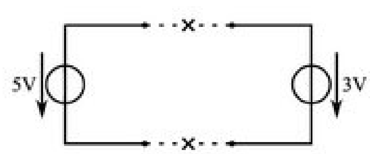


Fig. 3.7: Do not connect ideal voltage sources in parallel.

A series connection of ideal voltage sources is possible and for the resulting voltage at the terminals the voltages of the single sources are simply added.

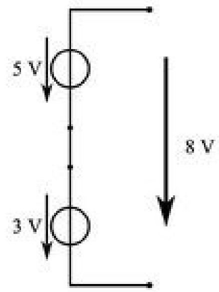


Fig. 3.8: The permitted connection of two ideal voltage sources in series..

Automotive application

The number of electrical systems in modern cars is steadily increasing. The control of these systems is often achieved using electrical control units (ECUs). Even in conventional cars with a combustion engine the number of electrical systems and ECUs can be up to 100, or even higher. ECUs can be found nearly everywhere in the car: for lighting, motor control and transmission as well as for convenience applications like seat heating, window winders or multimedia systems. The electrical systems are supplied by a 12 V power circuit and lead acid batteries are commonly used as electrical energy storage elements. To reach the power circuit voltage of 12 V 6 lead acid cells with a nominal voltage of about 2 V each are connected in series.

In electric vehicles (EV) with an electric traction motor (and to some extent also for hybrid electric vehicles (HEV) combining a combustion and electric motor) an additional power circuit with higher voltages is introduced to provide sufficient power to the electric traction motor. The voltage level is up to 350–400 V and lithium ion batteries are used for the high voltage power circuit. Again the battery is built up of a series connection of single cells. For lithium ion batteries each cell has a voltage of about 3.24 V depending on the technology. E.g. $LiFePO_4$ cells have an end-of-charge voltage of about 3.6 V and 100 cells are concatenated in series to reach a voltage level of 360 V.

3.4 Ideal current sources

An ideal current source drives a current I or i(t) regardless of the load connected to the terminals but there has to be a load connected to the terminals. Without the load the current source cannot produce the current as the circuit has to be closed. Like for ideal voltage sources the current will be in general a function of time. Fig. 3.9 shows models of ideal current sources and waveforms.

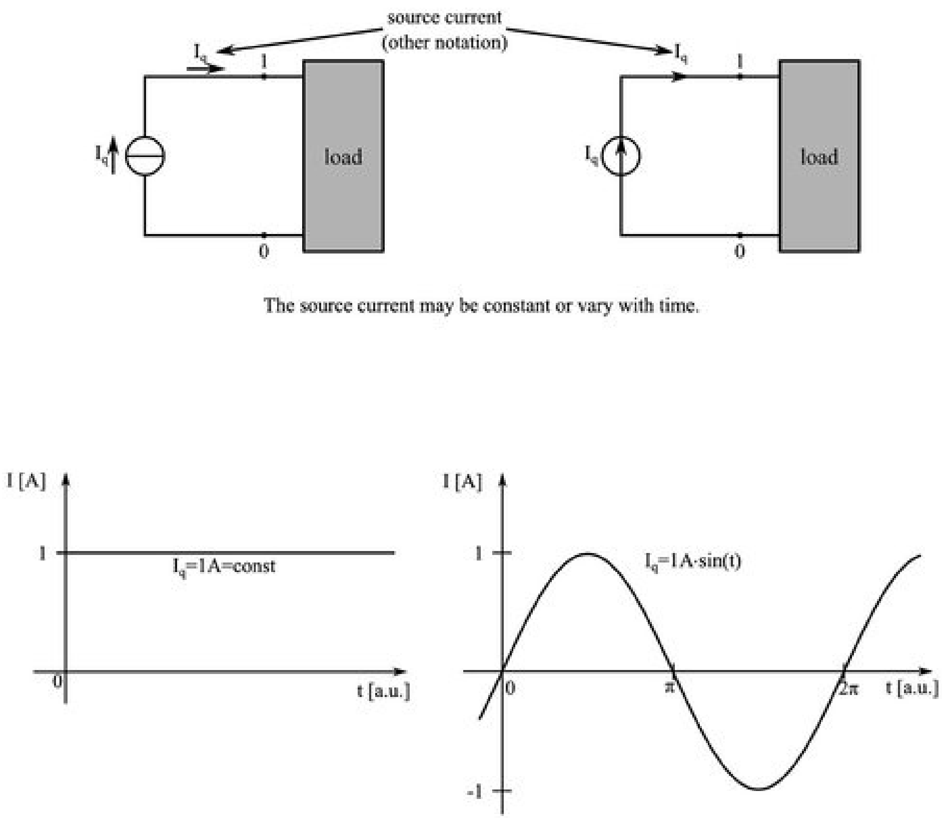


Fig. 3.9: Models of ideal current sources (top) and current waveforms (bottom).

For ideal current sources it is not permitted to connect two or more in series. But a parallel connection of ideal current source is possible and the currents of the two sources are simply added:

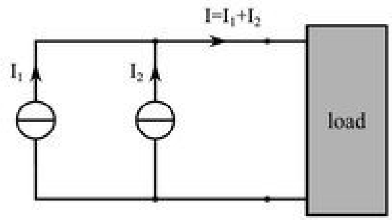


Fig. 3.10: Two ideal current sources connected in parallel.

3.5 Resistance, resistors and Ohm's law

The ratio of the voltage across a material u(t) and the current through it i(t) is called the resistance of the material. If this ratio is constant for the material independent of current or voltage, is it called a linear resistor (short: resistor) and it's resistance is:

$$R = \frac{u(t)}{i(t)}$$

In case of DC current and voltage sources Ohm's law yields:

$$R = \frac{U}{I}$$

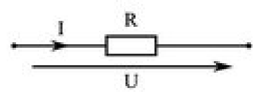


Fig. 3.11: Current I and voltage U for a resistor R.

This law is called Ohm's law and the SI unit for the resistance is the Ohm or Ω.
The resistor is a simple component, usually considered as linear, concentrated (lumped model) and is a constant. Symbols for resistors are:

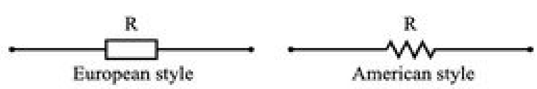


Fig. 3.12: Different models for resistors used in electric circuits: European style (left) and American style (right).

The reciprocal of the resistance R is the conductance G (SI unit: S=1/Ω, siemens):

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

The value of the resistance of a component is mainly determined by the physical dimensions of the component and the specific resistivity of the material of which the resistor is composed. For a bar of resistive material of length l and cross-section A the resistance R is given by

$$R = \frac{l \cdot \rho(T)}{A}$$

Here ρ(T) is the specific resistivity of the material in Ω·m. The reciprocal is called the specific conductivity, given in S/m. Tab. 1.1 lists the specific conductivity values for some materials.
A copper wire of 1 m length and a diameter of 2 mm has a resistance of about 5.5 mΩ at room temperature.
The (specific) resistivity of conductor metals is temperature dependent and varies approximately linearly over (normal operating) temperature (see Fig. 3.13). The resistivity at temperature T can be calculated based on the resistivity at a given temperature (e.g. room temperature, R(293 K) and a material dependent constant τ:

$$R(T) = R(293K) \cdot \frac{T - \tau}{293K - \tau} = R(293K) \cdot (1 + \alpha \cdot (T - 293K))$$

The abbreviation

$$\alpha = \frac{1}{293K - \tau}$$

is called the temperature coefficient of the resistance. It depends on the material and the given temperature (here: 293 K). Copper for example has a constant τ = 38 K and a temperature coefficient at room temperature of $\alpha_{293\ K} = 3.9 \cdot 10^{-}$

^{3} K^{-1}. In other words, the resistance of copper increases by ~0.4 % for every degree Kelvin or doubles when heated up to 463 K. For the 1 m copper wire of 2 mm diameter the resistance at 125 °C (the ambient temperature within the engine compartment) increases by 40 %.
The temperature dependence of metals has a positive slope and metals are typical examples for materials with a positive temperature coefficient (PTC): the higher the temperature the higher the resistance. The opposite of PTC elements are NTC elements (negative temperature coefficient). For these materials (e.g. semiconductors) the resistance decreases with increasing temperature.

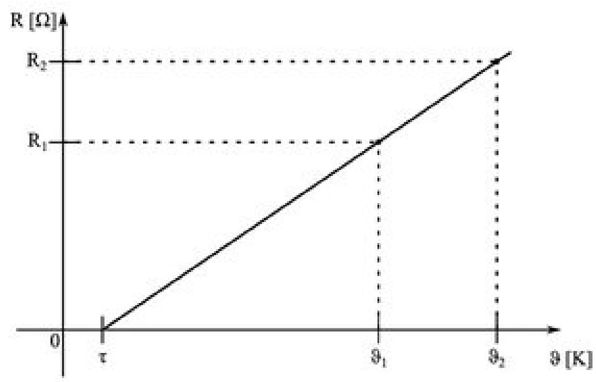


Fig. 3.13: Temperature dependence of the resistance of a conductor.

Automotive application
Everything is resistive in any electrical application and countless resistors are used in all kinds of electrical systems. Besides resistors in electric circuit resistive devices can also be used as sensors. In this application they make use of the geometric and temperature dependence of the resistance for example.
Resistors with a well known temperature dependence are used as sensors to measure e.g. air temperature, water or oil temperature. Very often NTC materials with a dedicated temperature dependence are used as temperature sensors. A small measurement current flowing through the NTC causes a voltage drop across this element. According to Ohm’s law this voltage drop together with the measurement current corresponds to a resistance value. This resistance is in the end a

measure of the temperature. In Fig. 3.14 the typical characteristics of a NTC temperature sensor is depicted. The resistance varies in a wide range and makes a temperature measurement with high resolution possible.

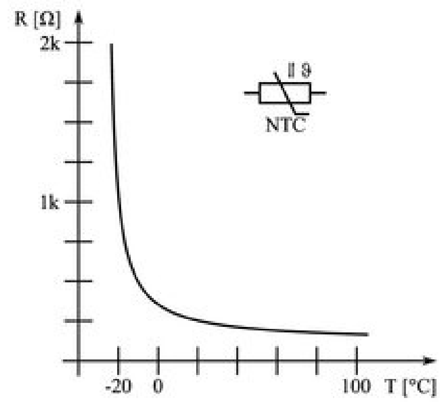


Fig. 3.14: Typical characteristic of a NTC and circuit symbol.

The geometric dependence of the resistance is used by resistance strain gauges to measure force, pressure or torque. If a bar of resistive material like silicon is compressed or lengthened, the geometry of the bar changes as shown in Fig. 3.15 on the left side. Both the length and the diameter of the bar change slightly if strain is applied to it. Due to the very small geometric change the change of resistance is rather small. To increase the geometric effect dedicated structures like a meander are used (Fig. 3.15, right side).

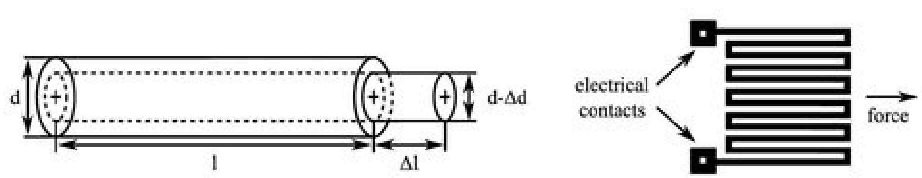


Fig. 3.15: Geometric changes of a bar of material in case of lengthening (left); resistance strain gauge with a meander like structure.

As the resistance changes are still small the measurement is usually done using four strain gauges configured in a Wheatstone bridge configuration (Fig. 3.16, left). A Wheatstone bridge consists of two legs with two resistors in each leg. Both legs build a voltage divider. Depending on theses four resistors

a voltage difference U_a between the middle nodes of the legs can be measured. In case that all resistors have the same resistance the voltage difference is zero. If one resistor changes its resistance the voltage will be non-zero and a measure for the change of the resistance.
For the measurement with strain gauges four elements are used. These elements are mounted in a way that the change in resistance of the elements amplifies the voltage difference U_a. Two of the strain gauges are compressed (e.g. the top left and bottom right element) and the other two are lengthened to increase the voltage difference between the two half bridges of the Wheatstone bridge.
A pressure sensor as shown in Fig. 3.16 uses four strain gauges to determine the differential pressure between p_1 and p_2. The four strain gauges are implemented onto a silicon membrane fabricated using microsystems technology. One strain gauge element each is located at region “a” at the edge of the membrane and the other two are located at “b” in the middle of the membrane. Without a differential pressure each strain gauge has a resistance R. Due to a differential pressure the membrane is deformed and the mechanical stress is opposite for the two strain gauges in the middle compared to the elements at the edge. Therefore the resistance of two strain gauges is reduced by ΔR and for the other two it is increased by ΔR. The voltage difference between the two half bridges is a measure for the resistance change and hence for the pressure.

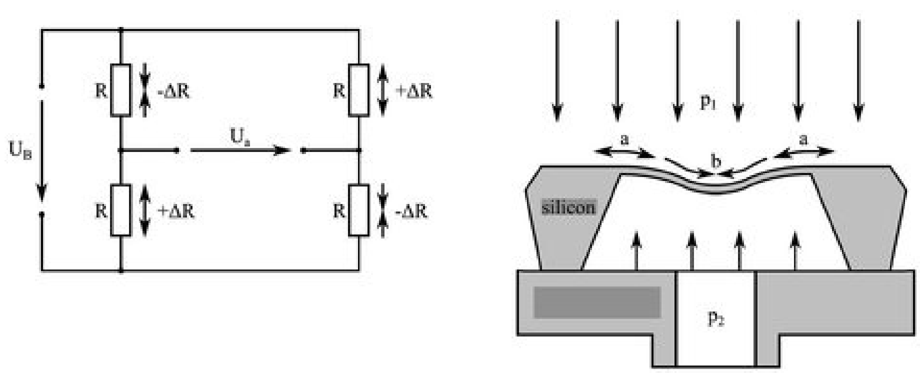


Fig. 3.16: A Wheatstone bridge (left) with four strain gauges; pressure sensor for

differential pressure measurement (right).

3.5.1 Real resistors

Of course resistors can be made in any value and any shape that is needed. But in reality resistors are manufactured in standard values and a number of different shapes. Examples are the E series of resistors with standardized resistance values. Types of resistors include composition type, wire-wound type and metal-film type. The most common construction technique for resistors is the composition type, which uses carbon or graphite and is molded into a cylindrical shape. As the shape of the cylinder is the same the value of the resistance and its tolerance is color-coded in bands as shown in Fig. 3.17 and Tab. 3.1.
Using these 4 bands the resistance can be calculated using

$$R = \left(10 \cdot b_1 + b_2\right) \cdot 10^{b_3}\,\Omega$$

Sometimes a fifth band indicates the reliability of the device.

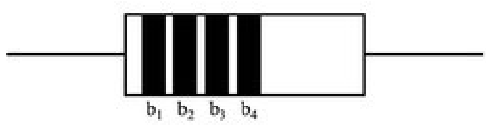


Fig. 3.17: Color-coding of resistors.

Tab. 3.1: Color coding of resistors.

Color of the band	Value of the band b_1, b_2	Multiplier b_3	Tolerance value [%]
Black	0	10^0	
Brown	1	10^1	1
Red	2	10^2	2
Orange	3	10^3	
Yellow	4	10^4	
Green	5	10^5	0.5
Blue	6	10^6	0.25
Violet	7	10^7	0.1
Grey	8		0.05
White	9		–
Gold		10^{-1}	5
Silver		10^{-2}	10
Black/no color			

Standard resistance values for the E24 series in the range from 1 to 9.1Ω are listed in Tab. 3.2. Other available values can be obtained by multiplying these values by factors of 10 ranging from 10 Ω to about $22 \cdot 10^6 \Omega$.

Tab. 3.2: Resistance values in Ω for the E24 series of resistors.

1.0	1.8	3.3	5.6
1.1	2.0	3.6	6.2
1.2	2.2	3.9	6.8
1.3	2.4	4.3	7.5
1.5	2.7	4.7	8.2
1.6	3.0	5.1	9.1

Besides the resistance value and the shape of a resistor also its power capability has to be taken into account when selecting a resistor for an application. Electrical power that is dissipated within a resistor is converted into heat. As excessive heating of the resistor may destroy the device the heat has to be conducted away from the resistor by providing a good thermal path.

3.5.2 Short circuit and open load

After an introduction to resistance and resistors we will analyze two extremes. Consider an electric circuit as shown in Fig. 3.18 with a load resistance R. What happens for R = 0 Ω and R = ∞ Ω?

The first case is called a short circuit. Keeping Ohm's law in mind we see that voltage drop is zero for finite currents: a zero-Ohm resistor is equivalent to an ideal voltage source with zero volts. In other words: if you connect an ideal voltage source to a zero-Ohm resistor, the current will rise to infinity. As a conclusion, never place a short circuit, neither intentionally nor unintentionally, across a voltage source to avoid excessive currents.

The latter with R = ∞ Ω is called an open circuit. Again looking at Ohm's law it is obvious that current will tend towards 0 A (as long as the voltage has finite value) in this case. This behavior is equivalent to a circuit with an opening and no current is flowing. In other words, as already stated at the beginning of chapter 3.1, it needs a closed loop for currents to flow.

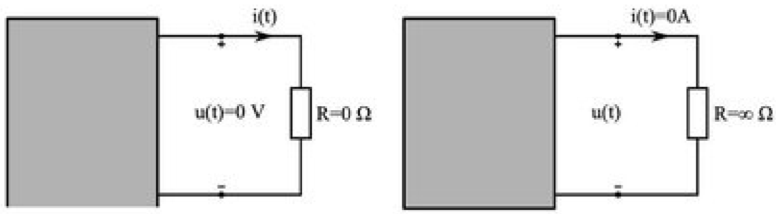


Fig. 3.18: Simple electric circuit: short circuit (left) and open load (right).

Automotive application

Both short circuit and unwanted open load are severe fault conditions in automotive applications. Consider a lighting application like a headlight as depicted in a simplified circuit in Fig. 3.19. A voltage source with internal resistance R_i drives the bulb and a current I of some amperes flows through the circuit. For a given voltage (e.g. 12 V vehicle electrical system) the current is determined by the internal resistance and the resistance of the bulb. If a short circuit occurs that shorts the bulb the current will just flow through the short circuit path and it will only be limited by the internal resistance. Hence the current will be much higher. This excessive current will rapidly discharge the battery, or even severely damage the circuit and the battery until total destruction of the system and maybe the vehicle occurs. Therefore a short circuit has to be detected to prevent damage to the system. In the simplest case a fuse in the circuit separates the battery from the rest of the circuit if the current gets too high. Alternatively the current is measured and a switch is triggered to open the circuit in case of excessive currents without using the fuse.

An open load situation can happen if the filament of the bulb is broken, no current flows through the bulb and the bulb does not shine anymore. This malfunction of the lighting has to be detected at least in cases where the lighting is used for a safety critical application like headlights. A defect headlight is critical for the recognizability and visibility of the vehicle and hence is a traffic hazard.

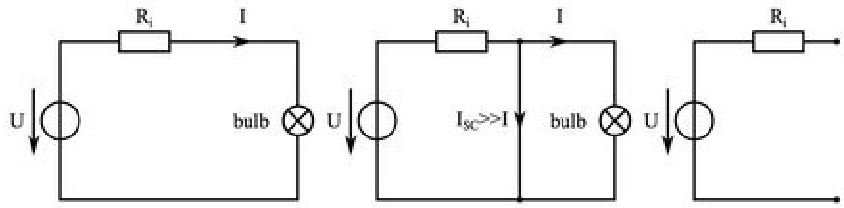


Fig. 3.19: A simple circuit with voltage source and internal resistance to drive a bulb (left); short circuit (center); open load by broken filament of the bulb (right).

3.5.3 Real voltage sources

An ideal voltage source produces a voltage at the terminals regardless of what is connected to it (independent of the load and the current). In reality the voltage at the terminals drops if a load is connected to the source. This voltage drop can be modeled by an internal resistance. In reality this internal resistance is unavoidable as every voltage source contains internal resistive elements like wires. Fig. 3.20 shows a simple drawing of a real voltage source with an internal resistance R_i.

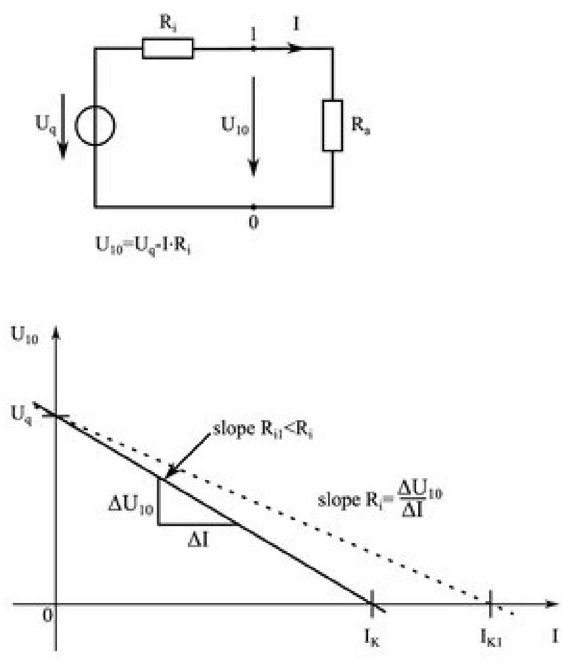


Fig. 3.20: A simple schematic for a real voltage source with internal resistance R_i and voltage vs current characteristic.

The voltage source shown in this simple schematic is divided into an ideal voltage source (U_q) and an internal resistance R_i, connected in series. R_a is the external load resistance. The voltage U_{10} at the terminals of the real voltage source is then:

$$U_{10} = U_q - I \cdot R_i$$

The two parameters U_q and R_i are in general independent of the load current I. As voltage U_{10} is a linear function of the load current I. This type of voltage source is called a linear source.
In the case of an open circuit (no external load R_a) the voltage at the terminals is called the open-circuit voltage (OCV). If R_i is zero the real voltage source turns again into an ideal voltage source with a constant voltage at the terminals (constant voltage source).
In the case of a short-circuit condition (U_{10} = 0 V) the current is limited by the internal resistance R_i according to:

$$I_{SC} = \frac{U_q}{R_i}$$

In reality an internal resistance R_i as small as possible is often required to come close to an ideal voltage source (for example for batteries). In this case the short-circuit current I_{SC} might become very high – be careful of creating short-circuit conditions in your applications.
Even though you deal with real voltage sources in reality we will use ideal voltage sources during our analysis of electric circuits if not otherwise stated.

Automotive application
A combustion engine needs an electrical starter motor for initial starting. This electrical starter motor requires rather high currents of some hundred amperes from the lead acid battery to generate the torque to start the engine. To provide these high currents during starting the internal resistance of the battery

should be very low. But the internal resistance depends on many parameters: it increases over lifetime due to corrosion for example and it is higher at low temperatures, e.g. in winter.
Consider a starter motor with a resistance of 30 mΩ that requires at least 180 A to start the engine. A new 12 V lead acid battery has an internal resistance of 30 mΩ. During starting the starter motor is in series with the battery's internal resistance and a current of 200 A flows through the starter motor. The terminal voltage of the battery drops down to 6 V (and all electrical systems supplied by the battery have to keep on operating). For an old battery the internal resistance at low temperatures might increase up to 60 mΩ. Now the maximum current through the starter is just 133 A and the starter motor cannot generate sufficient torque to start the engine.

3.5.4 Real current sources

An ideal current source produces a current regardless of what is connected to it (independent of the load and the voltage), it is a constant current source. Like for the real voltage source we model the real current source with an ideal element and an internal resistance, but this time the ideal element and R_i are connected in parallel as shown in Fig. 3.21.

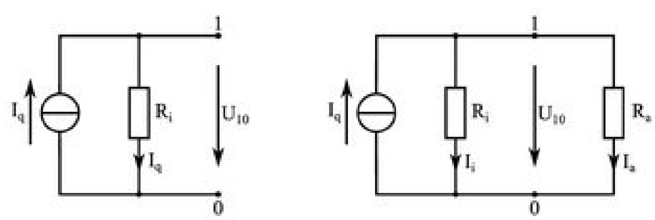


Fig. 3.21: A real current source without (left) and with load (right).

For an open circuit as shown on the left side of Fig. 3.21 the voltage U_{10} at the terminals is:

$$U_{10} = I_q \cdot R_i = \frac{I_q}{G_i}$$

Connecting a load to the real current source this constant current is divided into two parts flowing through R_i and R_a:

$$I_q = I_a + I_i$$

$$I_a = I_q - \frac{U_{10}}{R_i} = I_q - U_{10} \cdot G_i$$

The voltage at the terminals now depends on the external load resistance and is:

$$U_{10}(R_a) = I_a(R_a) \cdot R_a$$

According to the current divider rule (based on Kirchhoff's current law, see below), this voltage can be calculated as:

$$U_{10}(R_a) = I_q \cdot \frac{R_i \cdot R_a}{R_i + R_a}$$

Again the voltage is a linear function of the load current as shown in Fig. 3.22.

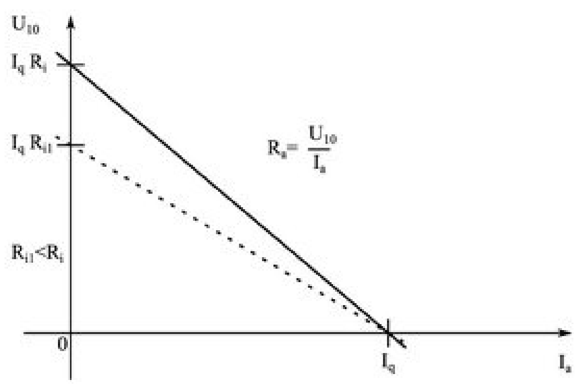


Fig. 3.22: Voltage as a function of load current for different external loads.

Linear voltage (see Fig. 3.20) and current (see Fig. 3.21) sources are equivalent and can be transformed into each other. At the end of the introduction of real voltage and current sources the following images show some examples of sources.

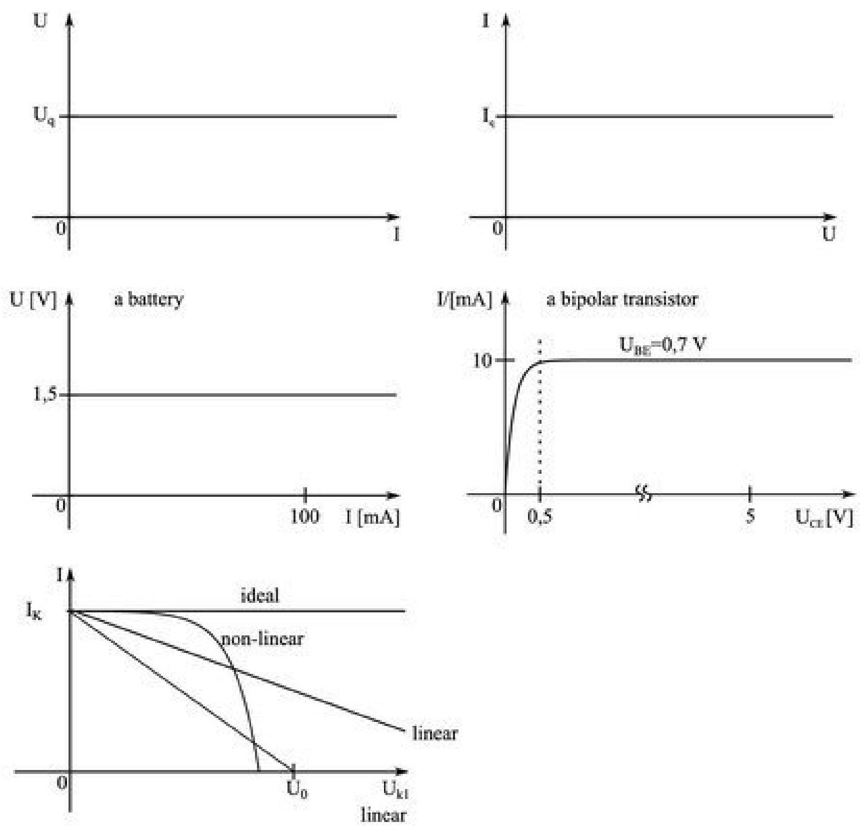


Fig. 3.23: Examples of sources:An ideal voltage source (top left); an ideal current source (top right); a battery (center left); a bipolar transistor (center right); ideal, linear and non-linear current source of a solar cell (bottom).

3.5.5 Transformation of sources

Consider real voltage and current sources as shown in Fig. 3.20 and Fig. 3.21. If we want to replace the former (given U_q and R_i) by the latter we have to determine the parameters I_q and R_i (or G_i) of the real current source.
In case of a short circuit ($R_a = 0\ \Omega$) we find for the real voltage source:

$$U = \frac{U_q}{R_i}$$

And for the corresponding current source:

$$I = I_q$$

As these currents have to be the same we conclude for the current source:

$$I_q = \frac{U_q}{R_i}$$

On the other hand for open load (infinite R_a) of the real voltage source we get:

$$U = U_q$$

And for the corresponding current source:

$$U = \frac{I_q}{G_i}$$

Finally:

$$G_i = \frac{1}{R_i}$$

The internal resistances of the corresponding sources are the same and the relation between I_q and U_q is as given above.

4 Fundamental electrical circuit laws

4.1 Kirchhoff's laws

As we have already seen in the previous chapter about real sources, electric circuits in general are built up of several different parts. To analyze more complex circuits two basic laws are fundamental: Kirchhoff's current law (KCL) and Kirchhoff's voltage law (KVL). These laws describe the correlations of currents and voltages in an electric circuit.

4.1.1 Kirchhoff's current law

Remember that a node is a connection of two, or more elements of a circuit. The first of Kirchhoff's laws, the current law describes the currents at any node of the circuit and is based on the law of conservation of electric charge:

– At any node of a circuit, the currents algebraically sum to zero at any instant of time.

Here currents flowing into the node are considered to be positive and currents directed out of the node are negative. In other words, the sum of the currents into the node is equal to the sum of the currents out of the node. Refer to Fig. 4.1 to see examples for nodes with several currents flowing into and out of the node.

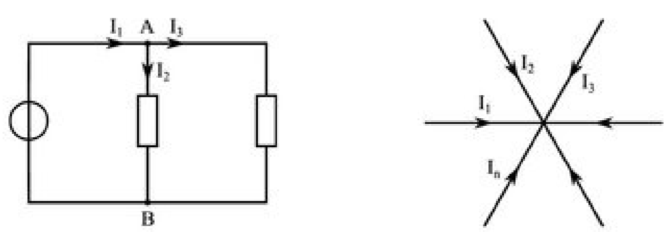


Fig. 4.1: An electric circuit with one voltage source, two resistors and corresponding current vectors (left); one node as part of a circuit with six elements connected to the

node and corresponding currents (right).

Kirchhoff's current law can now be written as (see Fig. 4.1):

$$\sum_{k=1}^{n} I_k = 0$$

Example: The circuit on the left side of Fig. 4.1. Here current I_1 flows into the node and therefore is positive whereas I_2 and I_3 are directed out of the node and are counted negative:

$$I_1 - I_2 - I_3 = 0$$

In a more general way Kirchhoff's current law is not only applicable for nodes but also for any closed region of a circuit (see Fig. 4.2). Here KCL of course applies for node 1, but also for the closed region marked by the dotted line. The algebraic sum of all currents flowing into and out of the closed region has to be zero. In this case:

$$\sum_{k=1}^{n} I_k = I_{10} + I_{11} + I_4 - I_5 + I_6 + I_7 + I_8 + I_9 = 0$$

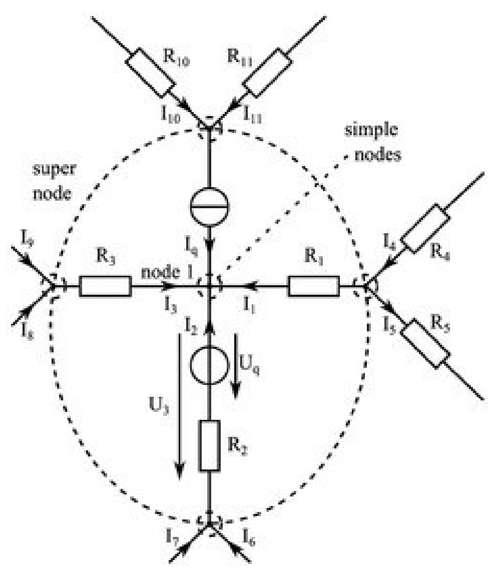


Fig. 4.2: A more complex part of a circuit with a closed region (dotted line) for which KCL also applies.

Application of KCL: Resistors connected in parallel

Consider two resistors connected in parallel to a voltage source as depicted in Fig. 4.3 on the left side. Two (or more) parts are connected in parallel if they are connected to the same pair of nodes. We would like to find the equivalent resistor R (see right side of Fig. 4.3) to replace the two parallel resistors. How is R related to R_1 and R_2?

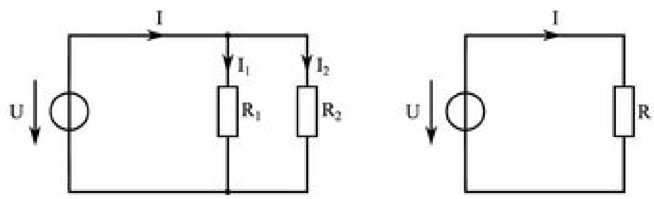


Fig. 4.3: Parallel connection of two resistors (left) and equivalent circuit with one equivalent resistor (right).

According to KCL the current I splits into I_1 and I_2:

$$I = I_1 + I_2$$

As the voltage across each resistor R_1 and R_2 is U Ohm's law for each resistor is:

$$I_1 = \frac{U}{R_1}$$

$$I_2 = \frac{U}{R_2}$$

And for the equivalent circuit:

$$I = \frac{U}{R}$$

Using the three equations from Ohm's law in KCL results in:

$$\frac{U}{R} = \frac{U}{R_1} + \frac{U}{R_2}$$

Thus, we see that the parallel connection of two resistors is equivalent to a single resistor provided that:

$$\frac{1}{R} = \frac{1}{R_1} + \frac{1}{R_2}$$

$$R = \frac{R_1 \cdot R_2}{R_1 + R_2}$$

This rule for parallel resistors can be generalized to any number n of parallel resistors:

$$\frac{1}{R} = \sum_{k=1}^{n} \frac{1}{R_k}$$

Or, using the conductance G and G_k:

$$G = \sum_{k=1}^{n} G_k$$

The resulting conductance of n parallel resistors is the sum of all single conductances. Coming back to the easy example of two resistors connected in parallel: how is the current I divided by the resistors? From the equations above we conclude that the voltage U is:

$$U = R \cdot I = \frac{R_1 \cdot R_2}{R_1 + R_2} \cdot I$$

Replacing the voltage U in Ohm’s laws for the two resistors gives the current for both resistors:

$$I_1 = \frac{U}{R_1} = \frac{R_2}{R_1 + R_2} \cdot I$$

$$I_2 = \frac{U}{R_2} = \frac{R_1}{R_1 + R_2} \cdot$$

These two formulas describe how the current I is divided into two parts through the resistors. This circuit is often referred to as a current divider. The ratio of the two currents is:

$$\frac{I_1}{I_2} = \frac{R_2}{R_1}$$

From these equations it is obvious that the currents are reciprocally proportional to the resistances. In other words: the smaller the resistance (compared to the other resistance), the higher the current through this resistance. The current tends to take the path of least resistance.

4.1.2 Kirchhoff's voltage law

Remember that a loop is a closed path through a circuit in which no electric element, or node is encountered more than once. Kirchhoff's second law, the voltage law, describes the voltages within loops and is based on the physical law of the conservation of energy:

- Around any loop in a circuit, the voltages algebraically sum to zero.

In other words: in traversing any loop in any circuit, at every instance of time, the sum of voltages having one polarity equals the sum of the voltages having the opposite polarity. KVL is valid for all loops of a circuit, even for open loops (loops with an open circuit) and loops that do not follow a physical branch in the circuit. But you never encounter any other node twice except the starting point.
Two loops, I and II, are marked in the circuit shown in Fig. 4.4. In fact these loops are even meshes. First of all the direction of traversing has to be defined. For both loops in Fig. 4.4 this direction is defined arbitrarily as counterclockwise. All voltages pointing in the same direction are counted positive. Voltages pointing in opposite direction are counted negative. Hence we find for the two loops of Fig. 4.4:

- Loop I:

$$U - U_2 - U_1 = 0$$

– Loop II:

$$U_2 - U_4 - U_3 = 0$$

As KVL is valid for all loops, not only meshes like I and II, we can also write for the outer loop:

– Outer Loop:

$$U - U_4 - U_3 - U_1 = 0$$

In general we can write for any loop of a circuit:

$$\sum_{k=1}^{n} U_k = 0$$

Here n is the number of voltages within the loop.

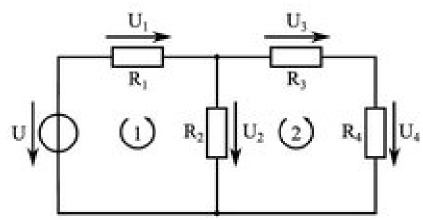


Fig. 4.4: A simple circuit with a voltage source and 4 resistors, two loops (meshes) are marked with I and II.

Application of KVL: Resistors connected in series

Consider two resistors connected in series to a voltage source as depicted in Fig. 4.5 on the left side. Two elements are connected in series if they have a node in common and no other element is connected to this common node. As a consequence of this definition, the same current I flows through elements connected in series. We would like to find the equivalent resistor R to replace the two resistors in series. How is R related to R_1 and R_2?

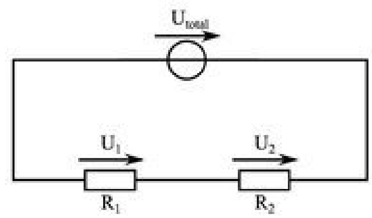


Fig. 4.5: Series connection of two resistors.

Applying KVL yields:

$$\sum_{k=1}^{n} U_k = U_{total} - U_1 - U_2 = 0$$

With Ohm's law for R_1, R_2 and R this results in:

$$R \cdot I = R_1 \cdot I + R_2 \cdot I$$

$$\Rightarrow R = R_1 + R_2$$

The resistance R of a single resistor equivalent to a series connection of resistors is just the sum of the resistors connected in series. In general for n resistors in series:

$$R = \sum_{k=1}^{n} R_k$$

Looking again at Fig. 4.5 shows that the total voltage U_{total} is divided by the two resistors into two parts, U_1 and U_2. Two voltage divider rules describe how the voltage is divided between the resistors and two resistors connected in series are therefore often called a voltage divider:

$$U_1 = \frac{R_1}{R_1 + R_2} \cdot U$$

$$U_2 = \frac{R_2}{R_1 + R_2} \cdot U$$

The larger voltage drop will be across the larger resistor.

4.2 Operating point

Consider an easy example for an electric circuit as shown in Fig. 4.6. It consists of two meshes, the source and the load mesh. Both meshes are closed via the terminal with voltage U_{10}.

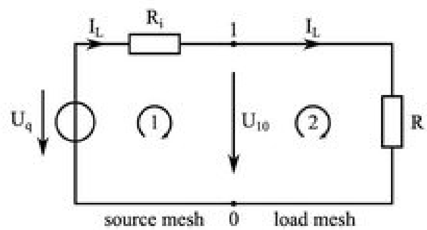


Fig. 4.6: A simple electric circuit with a source and a load mesh.

For mesh I we can write the following equation according to KVL:

$$-U_q + R_i \cdot I_L + U_{10} = 0$$

For mesh II we can write:

$$-U_{10} + R_L \cdot I_L = 0$$

Thus we have two unknown variables (I_L, U_{10}) and two equations and this linear equation system is algebraically solvable:

$$I_L = \frac{U_q}{R_i + R_L}$$

$$U_{10} = U_q \cdot \frac{R_L}{R_i + R_L}$$

These two values define the operating, or working point of this circuit. For given parameters (like R_L, R_i and U_q) the operating point (or working point, WP) defines the steady state of the system. In practical problems often small parameter changes around the working point are considered: small voltage, temperature or resistance variations for example.

Instead of solving these equations algebraically they can also be solved graphically. For this purpose the equations for the two meshes as given above are transformed to show the dependence of load current I_L as a function of terminal voltage U_{10}:

$$I_L = \frac{U_q}{R_i} - \frac{U_{10}}{R_i}$$

$$I_L = \frac{U_{10}}{R_L}$$

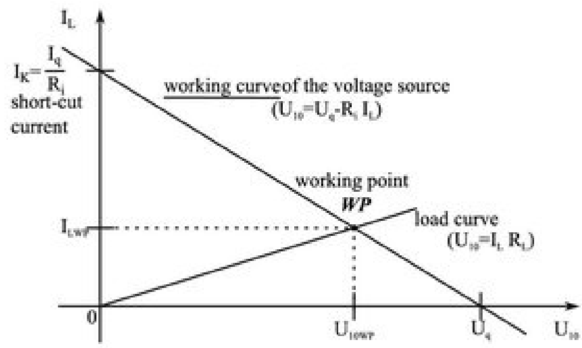


Fig. 4.7: Characteristic curves of the voltage source and the load. WP indicates the working point of the circuit.

Fig. 4.7 shows the characteristic curves for both the voltage source and the load. The characteristic curve of the voltage source is a falling line that intersects the I_L-axis at the short circuit current I_{SC} (U_{10} = 0 V) and the U_{10}-axis at the value of the source voltage. The straight load line rises according to Ohm’s law. Since any point on the source curve satisfies the source equation and any point on the load curve satisfies the load equation, the intersection of both plots satisfies both equations simultaneously and the point of intersection is the operating, or working point (WP).

For linear equations as shown above the graphical solution seems to be inappropriate. But consider a circuit with a non-linear load where the load current is a non-linear function of the applied voltage. Semiconductor components like diodes or transistors are examples for such non-linear components. For these systems with non-linear components the techniques used for linear, algebraic simultaneous equations cannot be employed and the equation system has no analytical solution in general. Fig. 4.8 shows an example of a circuit with a non-linear component, a diode.

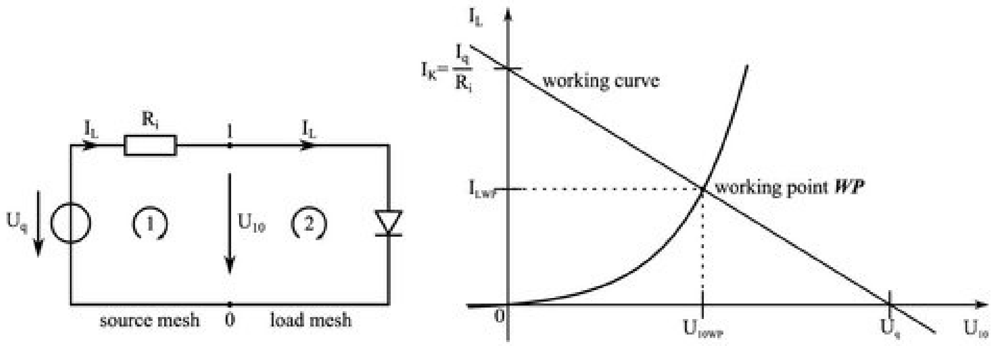


Fig. 4.8: A simple circuit with a diode as load (left); Characteristic curves for the source and the diode; diode current is a non-linear function of the voltage (right).

The current of the diode is a non-linear function of the voltage as shown on the right side of Fig. 4.8, e.g.

$$I_L = I_S \cdot \left(e^{\frac{U_{10}}{C}} - 1 \right)$$

Here I_S (inverse current) and C are constants. As it is not possible to solve the source and load equation simultaneously in this case, the WP is determined by the graphical solution as shown on the right side of Fig. 4.8.
In general for electronic systems the operating conditions have to be set properly to operate the components and devices in the required functionality. The method for setting proper operating points, voltages or currents, is also called biasing.

4.3 Wye-Delta transformation

We have seen that certain circuit configurations, serial and parallel connections of resistors, can be simplified by a single resistor to make circuit analysis easier. Refer to Fig. 4.9 as an example of simplification:

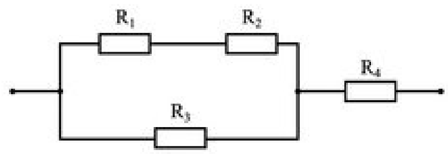


Fig. 4.9: A circuit with series and parallel connections for the demonstration of simplification.

First of all replace resistors R_1 and R_2, connected in series, by equivalent resistor $R_{12} = R_1 + R_2$. This equivalent resistor is parallel to R_3 and these two resistors can be replaced by R_{123}:

$$R_{123} = \frac{(R_1 + R_2) \cdot R_3}{R_1 + R_2 + R_3}$$

R_{123} now is connected in series with R_4 and the final resistor R, replacing resistors R_1-R_4 is:

$$R = \frac{(R_1 + R_2) \cdot R_3}{R_1 + R_2 + R_3} + R_4$$

But not all configurations can be simplified by these simple laws. In some of such cases a special transformation, a Wye-Delta transformation can be used to replace three resistors in Wye configuration by three resistors in Delta configuration or vice versa, so that the circuits are equivalent as far as the terminals are concerned. Refer to Fig. 4.10 for the two configurations. Both configurations are equivalent if the resistances measured between two of the terminals 1, 2, 3 are the same.

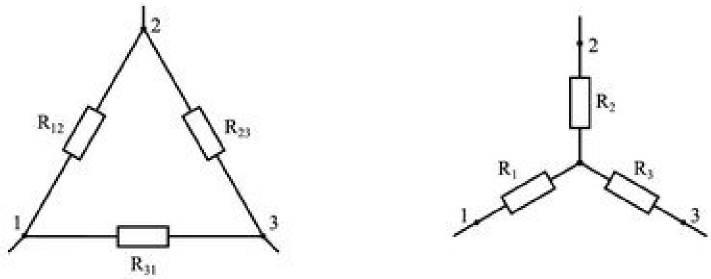


Fig. 4.10: Delta configuration (left) and Wye configuration (right).

Delta configuration to Wye configuration

Starting with the Delta configuration with given resistors R_{12}, R_{23} and R_{31} we are looking for the equivalent Wye configuration with resistors R_1, R_2, R_3. For the resistance between terminal 1 and 2 to be the same for both configurations it follows that:

$$R_1 + R_2 = \frac{R_{12} \cdot (R_{23} + R_{31})}{R_{12} + R_{23} + R_{31}}$$

For the resistance between terminal 1 and 3 and terminal 2 and 3 similar equations can be obtained:

$$R_2 + R_3 = \frac{R_{23} \cdot (R_{31} + R_{12})}{R_{12} + R_{23} + R_{31}}$$

$$R_3 + R_1 = \frac{R_{31} \cdot (R_{12} + R_{23})}{R_{12} + R_{23} + R_{31}}$$

Using these three equations we can calculate the three resistors of the Wye-configuration as follows:

$$R_1 = \frac{R_{12} \cdot R_{31}}{R_{12} + R_{23} + R_{31}}$$

$$R_2 = \frac{R_{23} \cdot R_{12}}{R_{12} + R_{23} + R_{31}}$$

$$R_3 = \frac{R_{31} \cdot R_{23}}{R_{12} + R_{23} + R_{31}}$$

Each resistor of the Wye configuration is given by the product of the two adjacent resistors of the corresponding Delta configuration divided by the sum of the three resistors of the Delta configuration.
For a symmetric delta configuration with

$$R_{12} = R_{23} = R_{31} = R_{\Delta}$$

each resistance of the corresponding Wye configuration is just:

$$R_{wye} = \frac{R_{\Delta}^2}{3 \cdot R_{\Delta}} = \frac{R_{\Delta}}{3}$$

Wye configuration to Delta configuration
Going the opposite direction from given resistors in Wye configuration the resistors of the corresponding Delta configuration can be calculated. The final result for the Delta configuration's resistors is:

$$R_{12} = R_1 + R_2 + \frac{R_1 \cdot R_2}{R_3}$$

$$R_{23} = R_2 + R_3 + \frac{R_2 \cdot R_3}{R_1}$$

$$R_{31} = R_3 + R_1 + \frac{R_3 \cdot R_1}{R_2}$$

4.4 Meters and measurements

The measurement of parameters like current and voltage in reality is a broad field and cannot be handled here in detail. However a few basic principles will be introduced to get an idea of how to measure electrical parameters. Real measurements are made by real instruments. In general the measurement disturbs the operation of an electric circuit to some extent and therefore care has to be taken to avoid useless and wrong measurements.

Voltmeter
Voltage is measured between two terminals, or nodes of a circuit. For this voltage measurement a voltmeter is connected across these two points as shown in Fig. 4.11. Without looking at the details of how the measurement is done the voltmeter can be modeled as a parallel combination of an ideal voltmeter (without current flow) and an internal resistor R_V. The shunt resistor is therefore also parallel to the voltage (resistance) to be measured. As the current is divided by these parallel combinations of resistances according to KCL the value of the shunt resistance hast to be very high to avoid disturbance of the measurement as much as possible. In practice it is of the order of several million Ohms.
If the voltage is measured across a well known resistor, the current flowing through this resistor can be calculated using Ohm's law.

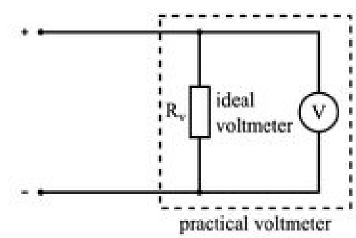


Fig. 4.11: The connection of a voltmeter to measure the voltage across the shunt resistor R_V.

Ampmeter
In contrast to the voltmeter the ampmeter is connected in series to measure the current through a line, or wire of a circuit. Therefore the circuit has to be broken to measure its current (whereas the circuit doesn't need to be broken for a voltage measurement, see above). The ampmeter can be modeled as a series combination of an ideal ampmeter and an internal resistance R_I. According to KVL the internal resistance has to be as small as possible to keep the disturbance of the circuit as small as possible.

An indirect way of measuring the current without breaking the circuit is to use a current probe or measuring caliper. These measuring instruments enclose the wire and make use of the magnetic properties of the current flowing through the wires.

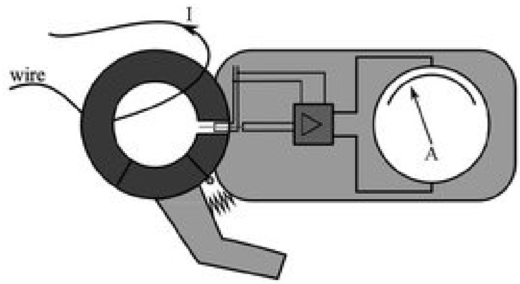


Fig. 4.12: A current probe or measuring caliper.

Oscilloscope

An oscilloscope is used to measure time-varying signals, both voltages and currents. An oscilloscope samples the time-varying signal at fixed instances of time (e.g. every 10ms) and displays a graph of the measured parameter as a function of time. This operating mode makes it possible to observe the general behavior of the voltage as a function of time.

Automotive applications:

Voltage and current are frequently measured by automotive systems and this measurement has to be done by the system itself. The voltage is usually measured using an analog to digital converter (ADC). This ADC can be a separate device or it is part of a microcontroller. In general the ADC has a maximum voltage range (e.g. 0–5 V) that can be measured. If the voltage to be measured is higher than the measurement range of the ADC can be divided using a voltage divider to fit to the ADC input requirements.

One indirect way of measuring currents in automotive systems is to use a shunt resistor. This shunt resistor is designed into the branch of the current flow. Due to the current there will be a voltage drop across the shunt resistor and this voltage drop can be measured as described above using an ADC. With the knowledge of the exact value of the shunt resistor the current can be calculated from the voltage. This simple method has the disadvantage that power is dissipated in the shunt resistor. Another current measurement method utilizes the Hall effect. Hall effect sensors measure the magnetic field of the current carrying wire. The output of the Hall sensor is the Hall voltage that can be measured by an ADC.

4.5 Power and energy

When a current i(t) flows through a resistor (voltage drop u(t)) energy is dissipated inside the resistor and electrical energy is converted into heat as (positive charged) current goes from a higher potential to lower potential. Indication arrows for current and voltage are parallel. The electrical energy E during time t_1-t_2 is given by:

$$e(t) = u(t) \cdot q(t) = \int_{t_1}^{t_2} u(t) \cdot i(t) dt$$

In the case of DC currents and voltages the energy is just (starting at t_1 = 0 s):

$$E = U \cdot I \cdot t$$

The SI unit is the Joule (J) and 1 J = 1 Vas = 1 Ws. For convenience it is usual to calculate with the unit kWh where 1 kWh = $3.6 \cdot 10^6$ Ws = 3.6 MJ. 1kWh equals to

- Working for 50 hours on a notebook (20 W power consumption)
- Heating about 10 liters of water from room temperature to 100 °C
- Driving an electric vehicle (EV) about 6–7km (for an EV with 15 kWh / 100 km)

Based on the electrical energy given above the instantaneous electrical power in Watts (W) is given by:

$$p(t) = \frac{de(t)}{dt} = u(t) \cdot i(t)$$

For DC currents power is time-independent:

$$P = U \cdot I = I^2 \cdot R = \frac{U^2}{R}$$

So a current of 1 A and a voltage drop of 1 V results in 1 W power dissipated in the resistor.

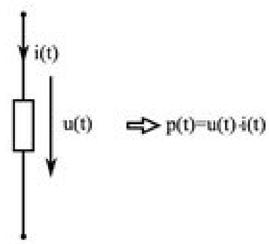


Fig. 4.13: Power at a resistor with current i(t) and voltage drop u(t).

Efficiency

When talking about the transformation of energy (in a source or load) efficiency is a key parameter. It is defined as the ratio of transformed power P_2 to spent power P_1:

$$\eta = \frac{P_2}{P_1} = \frac{E_2}{E_1}$$

Efficiency is always < 1 (or < 100 %) as not all the power can be transformed. The difference P_1-P_2 is the power loss. Target is to reduce power loss as much as possible and to get close to 1 for the efficiency. For example a generator (which does not actually generate) transforms mechanical power (P_1) into electrical power (P_2) and can reach an efficiency of up to 99.5 % (whereas an automotive combustion engine has something like < 45 % which is not very good…).

4.6 Maximum power transfer

As already stated above power is provided by a source and consumed by a load such as a resistor. Of course power is also consumed by the internal resistor of a real source. But what is the maximum power transfer between a real source (ideal source plus internal resistor) and a load resistor? What is the maximum power a source can provide? In order to investigate the power transfer in more detail, consider Fig. 4.14 in which a real voltage source is connected with a variable load resistor (R_a). Two extremes for the variable resistor were already considered above, short circuit ($R_a = 0\ \Omega$) and open load ($R_a = \infty\ \Omega$).

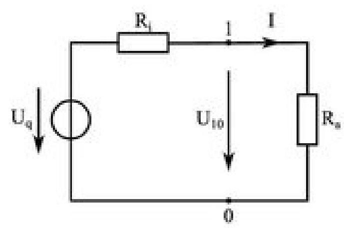


Fig. 4.14: A simple circuit for the investigation of power transfer, R_i is the internal series resistor of the voltage source, R_a the load resistor.

The current is given by:

$$I = \frac{U_q}{R_i + R_a}$$

We obtain for the power in the resistor:

$$P = I^2 \cdot R_a = \frac{U_q^2}{(R_i + R_a)^2} \cdot R_a$$

For fixed values of U_q and R_i the value of R_a that maximizes the power absorbed by the load can be found by setting the first derivative equal to zero:

$$\frac{dP}{dR_a} = \frac{U_q^2 \cdot (R_i + R_a)^2 - 2U_q^2 \cdot R_a \cdot (R_a + R_i)}{(R_a + R_i)^4} = 0$$

$$\Rightarrow (R_i + R_a)^2 - 2R_a \cdot (R_a + R_i) = 0$$

$$R_a = R_i$$

So maximum power is transferred to the load if the load resistance matches the source resistance or, in other words, both resistances have to be equal to each other. The maximum power in this case is:

$$P_{L,\max} = \frac{U_q^2}{4R_i}$$

Fig. 4.15 shows the power transfer to the load (P_L) compared to the total power provided by the source (P_{ges}) and the efficiency of the source η. In power electronics the efficiency is often maximized, in signal processing it is often the power.

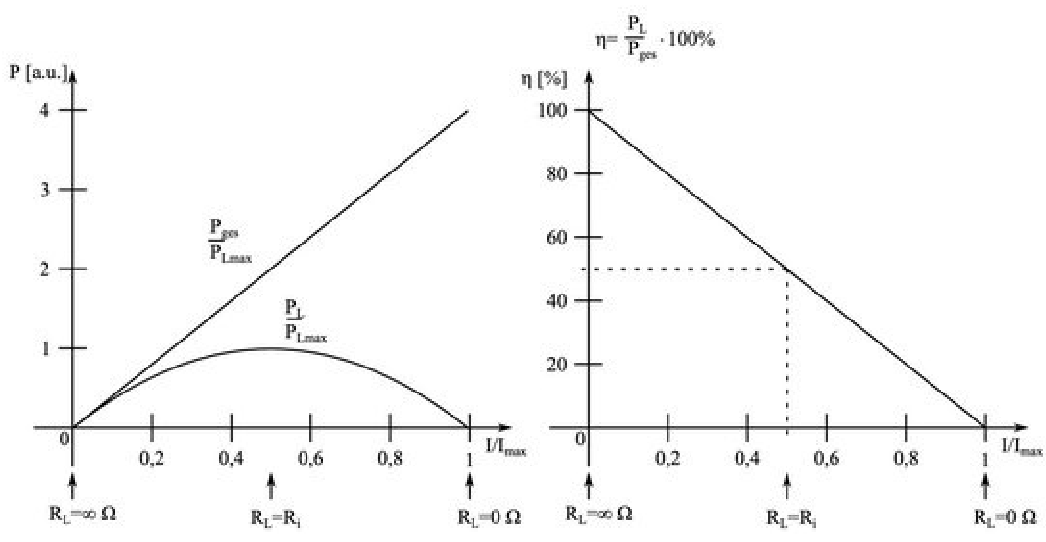


Fig. 4.15: Power transfer to the load (P_L) and total power provided by the source (P_{ges}) in units of the maximum power transfer (P_{Lmax}) (left); efficiency (right).

4.7 Dependent and independent sources

So far we have dealt with sources whose values (current or voltage) are in general time dependent. In more detail these sources were independent sources, i.e. the behavior of the sources (the current, or voltage they applied to the circuit) was independent of the behavior of the circuit to which the source belonged. No matter what happens within the circuit, the independent source supplies a fixed (but time-dependent) value.

On the other hand, for a dependent, or controlled source (current or voltage) the value depends on some variable (usually voltage or current) in the circuit to which the source belongs. In electronic circuits dependent sources are represented by the symbols shown in Fig. 4.16.

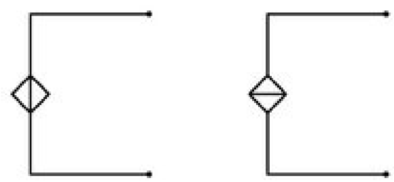


Fig. 4.16: Symbols of a dependent voltage source (left) and a dependent current source (right).

In general both voltage and current sources can be controlled by a current or a voltage. Fig. 4.17 shows the four types of dependent sources.

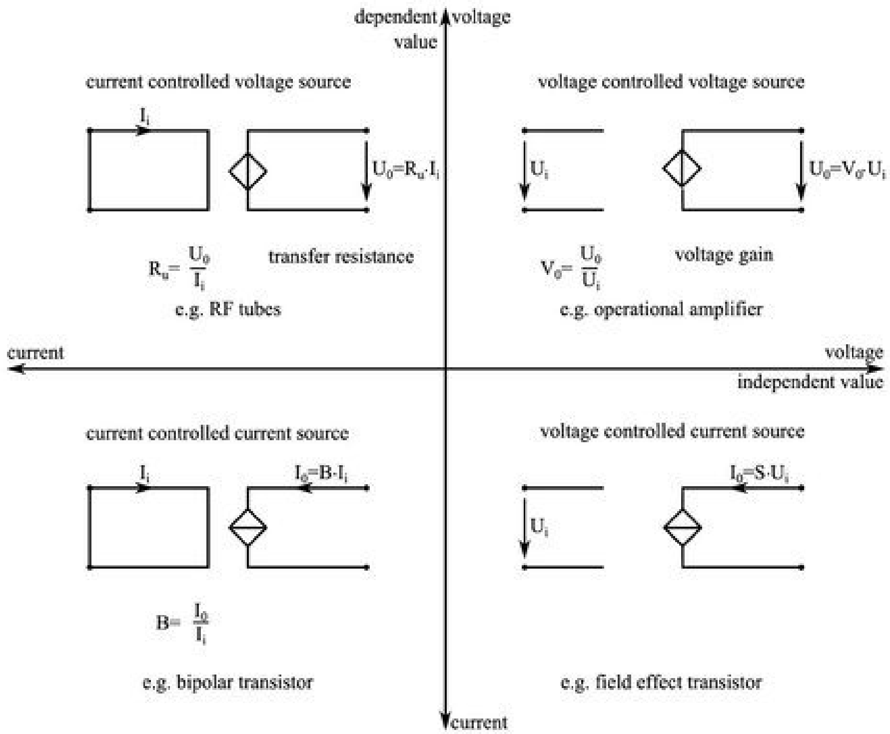


Fig. 4.17: Examples for dependent sources.

Example 1: Current controlled current source

The example as depicted in Fig. 4.18 contains a current source that depends on the value of the current I_1 through another branch of the circuit (like for example a bipolar transistor). Given parameters are the current of the independent source on the left, the two resistors and the dependence of the current controlled current source. What will the resulting currents I_1 and I_3 be?

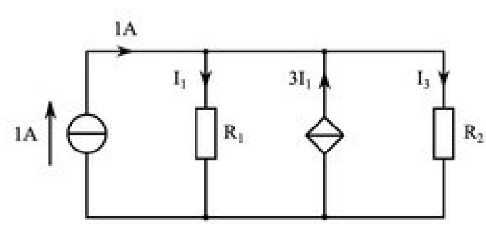


Fig. 4.18: An example of a circuit with a current controlled current source.

The node rule (KCL) provides:

$$1A + 3 \cdot I_1 - I_1 - I_3 = 0$$

KVL in combination of Ohm's law for the two resistors provides:

$$I_1 \cdot R_1 = I_3 \cdot R_2$$

Substituting I_3 into the node rule:

$$1A + 3 \cdot I_1 = I_1 + I_1 \cdot \frac{R_1}{R_2}$$

$$\Rightarrow I_1 = 1A \cdot \frac{R_2}{R_1 - 2R_2}$$

Using R_1 = 3 Ω and R_2 = 1 Ω yields I_1 = 1 A and I_3 = 3 A.

Example 2: Bipolar transistor as current controlled current source

A bipolar transistor can be regarded as a current dependent current source as the collector current is controlled by the base current: the collector current I_C (through the transistor) is given by the base current I_B multiplied by DC current gain factor B. Consider a circuit as shown in Fig. 4.19 on the left side where the bipolar transistor with a DC gain of 100 is used to make the 21 W-bulb to be operated at a voltage of 12 V. Voltage source is U_q = 14 V. On the right side of Fig. 4.19 the equivalent circuit with a current controlled current source is shown. What about the resistors R_1 and R_2?

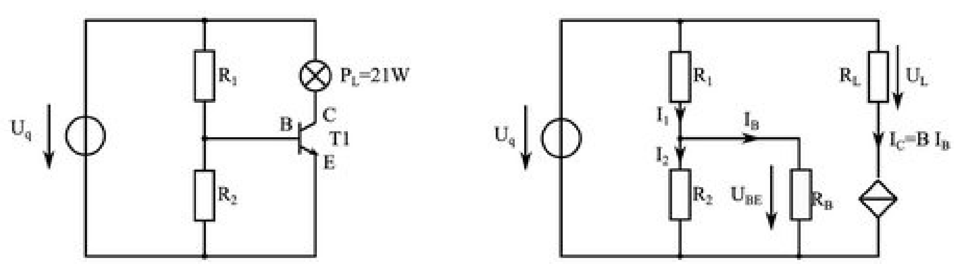


Fig. 4.19: A circuit with a biploar transistor acting as a current controlled current source (left); an equivalent circuit showing the current controlled current source (right).

From the given parameters of the bulb we can calculate the resistance of the bulb and the current that has to flow through the bulb and that equals the collector current I_C:

$$P_L = U_L \cdot I_C = \frac{U_L^2}{R_L}$$

$$\Rightarrow R_L = 6.86\Omega$$

$$\Rightarrow I_C = 1.75A$$

As the transistor works as a current controlled current source with a DC gain of B = 100 the base current has to be:

$$I_B = \frac{I_C}{B} = 17.5mA$$

The voltage drop U_{BE} for the bipolar transistor is about 0.7 V and thus the base resistance is:

$$R_B = \frac{U_{BE}}{I_B} = 40\Omega$$

To avoid excessive currents through the two resistances R_1 and R_2 we choose R_2 to be in the range of R_B, e.g. 100 Ω. Using KCL for node B ($I_1 = I_2 + I_3$) gives:

$$\frac{U_q - U_{BE}}{R_1} = \frac{U_{BE}}{R_2} + I_B$$

Hence resistor R_1 can be calculated as R_1 = 543 Ω and the current I_1 is 24 mA.

5 Circuit analysis

In the previous chapter I presented basic electric circuit concepts like KCL, KVL or Wye-Delta transformation. Now I want to introduce some more sophisticated circuit analysis techniques for practically and efficiently solving problems associated with circuit operations. We will start with two basic analysis techniques, nodal and mesh analysis. These two techniques are based on KCL and KVL and make use of two fundamental facts about electric circuits:

1. In any electric network with n nodes (n–1) independent equations for the nodes can be found
2. In any electric network with m meshes m independent equations can be found

5.1 Nodal analysis

Nodal analysis can be used for any electric circuit and in particular for circuits with few nodes (but rather many loops) as the number of equations will be small. It is based on the definition of voltage: Voltage is the difference between two electrical potentials.

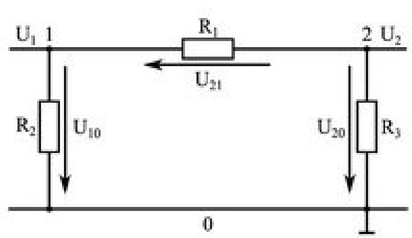


Fig. 5.1: Voltages (including a reference potential U_0) in an electric circuit.

It is common to define a reference potential and to refer the other voltages to this reference potential. The reference potential, or ground potential is marked with a special sign and is defined to have a voltage of U_0 = 0 V. For a circuit with n nodes there are (n–1) nodal voltages referring to ground potential. With nodal analysis the voltages of all nodes referring to the reference potential can be calculated. The procedure of nodal analysis will be introduced by an example before the general approach is presented.

An example of a nodal analysis

Refer to Fig. 5.2 for the first example of nodal analysis. The circuit contains 4 nodes, 0–3, and node 0 is defined as ground potential. The direction of the current vectors can be chosen arbitrarily (at the end of the calculation the sign of the current will show whether the current flows in the chosen, or in opposite direction). By using KCL and Ohm's law the three nodal voltages U_{10}, U_{20} and U_{30} can be calculated. For simplification of the calculation each resistance R_i is substituted by corresponding conductance G_i, i.e. $G_i = 1 / R_i$.

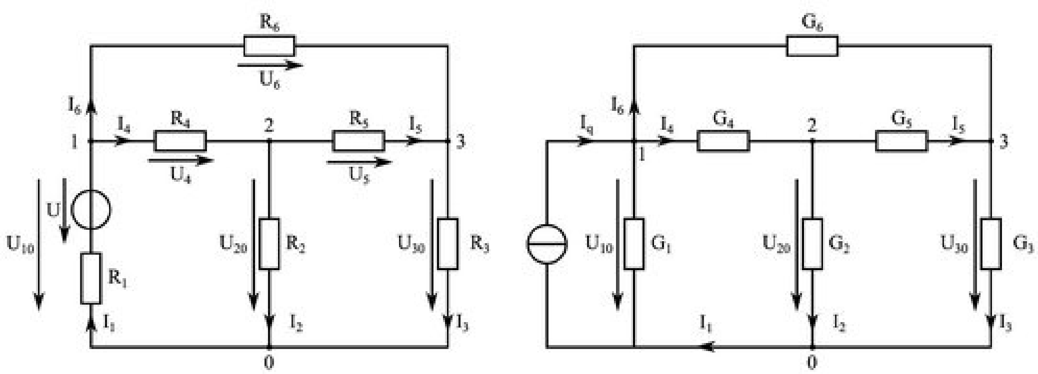


Fig. 5.2: An electric circuit with corresponding currents and voltages as an example of nodal analysis (left); node 0 is defined as ground potential; equivalent circuit with the voltage source and resistance R_1 transformed into a current source (right).

For node 1-3 in Fig. 5.2 KCL results in:

– node 1:

$$I_1 - I_4 - I_6 = 0$$

– node 2:

$$I_4 - I_2 - I_5 = 0$$

– node 3:

$$I_5 + I_6 - I_3 = 0$$

Using Ohm's law (I = G · U) for the currents give:

$$I_1 = G_1 \cdot (U - U_{10}) = G_1 \cdot U - G_1 \cdot U_{10}$$

$$I_2 = G_2 \cdot U_{20}$$

$$I_3 = G_3 \cdot U_{30}$$

$$I_4 = G_4 \cdot (U_{10} - U_{20})$$

$$I_5 = G_5 \cdot (U_{20} U_{30})$$

$$I_6 = G_6 \cdot (U_{10} - U_{30})$$

Hence for the KCL of node 1–3:

$$G_1 \cdot (U - U_{10}) - G_4 \cdot (U_{10} - U_{20}) - G_6 \cdot (U_{10} - U_{30}) = 0$$

$$G_4 \cdot (U_{10} - U_{20}) - G_2 \cdot U_{20} - G_5 \cdot (U_{20} - U_{30}) = 0$$

$$G_5 \cdot (U_{20} - U_{30}) - G_6 \cdot (U_{10} - U_{30}) - G_3 \cdot U_{30} = 0$$

So we have 3 linear independent equations for the three unknown variables U_{10}, U_{20} and U_{30}. After these voltages have been calculated the currents I_1 - I_6 can be determined using the equations for Ohm's law above.
Sorted by the unknown voltages these equations look like:

$$(G_1 + G_4 + G_6) \cdot U_{10} - G_4 \cdot U_{20} - G_6 \cdot U_{30} = G_1 \cdot U$$

$$-G_4 \cdot U_{10} + (G_2 + G_4 + G_5) \cdot U_{20} - G_5 \cdot U_{30} = 0$$

$$-G_6 \cdot U_{10} - G_5 \cdot U_{20} + (G_3 + G_5 + G_6) \cdot U_{30} = 0$$

These equations can also be written in matrix multiplication form:

$$\begin{pmatrix} G_1+G_4+G_6 & -G_4 & -G_6 \\ -G_4 & G_2+G_4+G_5 & -G_5 \\ -G_6 & -G_5 & G_3+G_5+G_6 \end{pmatrix} \cdot \begin{pmatrix} U_{10} \\ U_{20} \\ U_{30} \end{pmatrix} = \begin{pmatrix} G_1 \cdot U \\ 0 \\ 0 \end{pmatrix} = \begin{pmatrix} I_q \\ 0 \\ 0 \end{pmatrix}$$

For the last step we make use of the transformation of a voltage source into an equivalent current source: The voltage source U with resistance R_1 in series can be transformed into an equivalent current source with a parallel conductance G_1 with:

$$G_1 = \frac{1}{R_1}$$

$$I_q = \frac{U}{R_1} = G_1 \cdot U$$

Algorithm of nodal analysis

As we have seen in the example, by applying KCL and Ohm’s law we are able to set up (n–1) equations for (n–1) unknown nodal voltages of a circuit with n nodes. The following algorithm is one of several slightly different algorithms and it can be used for circuits with n nodes:

1. The circuit has n nodes. Select one node as reference potential (voltage 0 V).
2. Label the remaining (n–1) nodes with their voltages referring to the reference node (e.g. U_1, U_2).
3. Transform all voltage sources (together with the resistors in series) into equivalent current source with parallel conductance.
4. Draw the current arrows in the circuit and label the currents. The direction of the arrows can be chosen arbitrarily.
5. Derive the (n–1) equations by applying KCL to each node.
6. Substitute the unknown currents by the voltage drop across the resistance in the corresponding branch.

7. Solve the n–1 equations for the node voltages.
8. Calculate the branch currents using Ohm's law.

The application of algorithm for nodal analysis

The following example is used to apply the presented algorithm to an electric circuit as shown in Fig. 5.3:

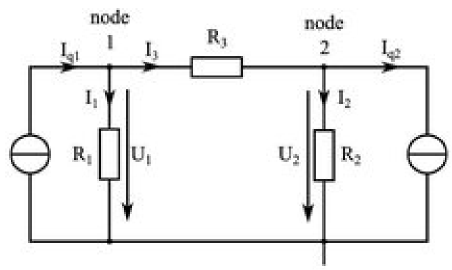


Fig. 5.3: Electric circuit with two ideal current sources and three nodes.

1. The circuit has n nodes. Select one node as reference potential (voltage 0 V).

   – See node with 0 V in Fig. 5.3

2. Label the remaining (n–1) nodes with their voltages referring to the reference node (e.g. U_1, U_2).

   – See voltages U_1, U_2 in Fig. 5.3

3. Transform all voltage sources (together with the resistors in series) into equivalent current source with parallel conductance.

   – No voltage source

4. Draw the current arrows in the circuit and label the currents. The direction of the arrows can be chosen arbitrarily.

   – See currents I_1-I_3 in Fig. 5.3

5. Derive the (n-1) equations by applying KCL to each node.

   – Two nodes and therefore two equations:
   – Node 1:

$$I_{q1} = I_1 + I_2$$

– Node 2:

$$I_{q2} = I_2 - I_3$$

6. Substitute the unknown currents by the voltage drop across the resistance in the corresponding branch.

– Node 1:

$$I_{q1} = \frac{U_1}{R_1} + \frac{U_1 - U_2}{R_2} = U_1 \cdot \left(\frac{R_1 + R_2}{R_1 \cdot R_2} \right) - \frac{U_2}{R_2}$$

– Node 2:

$$I_{q2} = \frac{U_1 - U_2}{R_2} - \frac{U_2}{R_3} = \frac{U_1}{R_2} - U_2 \cdot \left(\frac{R_3 + R_2}{R_2 \cdot R_3} \right)$$

7. Solve the (n–1) equations for the node voltages.

– We have two equations for the two unknown node voltages: U_1, U_2 and we can calculate these node voltages:

– From node 2:

$$U_2 = \left(\frac{U_1}{R_2} - I_{q2} \right) \cdot \frac{R_2 \cdot R_3}{R_2 + R_3}$$

– Substituting U_2 into the equation for node 1 gives:

$$I_{q1} = U_1 \cdot \left(\frac{R_1 + R_2}{R_1 \cdot R_2} \right) - \left(\frac{U_1}{R_2} - I_{q2} \right) \cdot \frac{R_3}{R_2 + R_3}$$

– And finally the node voltage U_1:

$$U_1 = \left(I_{q1} - \frac{I_{q2} \cdot R_3}{R_2 + R_3} \right) \cdot \left(\frac{R_1 + R_2}{R_1 \cdot R_2} - \frac{R_3}{R_2 \cdot |R_2 + R_3|} \right)^{-1}$$

– Node voltage U_2 can be calculated afterwards.

8. Calculate the branch currents using Ohm’s law.

$$I_1 = \frac{U_1}{R_1}$$

$$I_2 = \frac{(U_1 - U_2)}{R_2}$$

$$I_3 = \frac{U_2}{R_3}$$

For given values of I_{q1} = 6 A, I_{q2} = 12 A, ,R_1 = 1 Ω, R_2 = 3 Ω, R_3 = 2 Ω the voltages and currents yield U_1 = 1 V, U_2 = -14 V and I_1 = 1 A, I_2 = 5 A, I_3 = -7 A.

An example of nodal analysis having a voltage source between two non-reference nodes

So far we have dealt with circuits that contain sources between any node and the reference node. Voltage source with series were transformed into current sources with parallel conductance. In general voltage sources can also be present between non-reference nodes and without series resistance. To overcome the difficulty of transforming the ideal voltage source an extension of nodal analysis can be used. This modified nodal analysis (MNA) is for example also used in circuit simulation programs like PSPICE.

Consider the circuit given in Fig. 5.4. We want to determine the node voltages for the 4 nodes. It contains two ideal voltage sources between two nodes (U_{q1}, U_{q2}) and an ideal current source. We select one of the nodes of U_{q1} as reference point. The other three nodes and the currents are labeled as shown in Fig. 5.4.

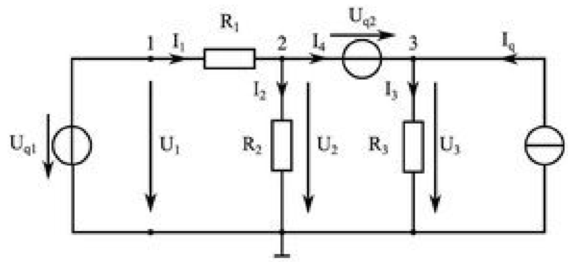


Fig. 5.4: Circuit for nodal analysis with an ideal voltage source between two reference nodes.

Now we derive the (n–1) = 3 equations by applying KCL and Ohm's law:

Node 1:

$$U_1 = U_{q1}$$

Node 2:

$$I_1 = I_2 + I_4$$

$$\frac{U_{q1} - U_2}{R_1} = \frac{U_2}{R_2} + I_4$$

We cannot immediately derive a requirement from the voltage source U_{q1} that determines the current I_4.

Node 3:

$$I_4 = I_3 - I_q = \frac{U_3}{R_3} - I_q$$

From node 1 the voltage U_1 is immediately given. From the other two nodes we have two equations but still three unknown variables as the current I_4 cannot be determined directly from voltage source U_{q2}. However we have the branch voltage for the branch containing U_{q2} that provides another equation:

$$U_2 - U_3 = U_{q2}$$

From node 2 and 3 we obtain:

$$\frac{U_{q1}-U_2}{R_1}-\frac{U_2}{R_2}=I_4=\frac{U_3}{R_3}-I_q$$

$$\Rightarrow \frac{U_{q1}}{R_1}-U_2\left(\frac{1}{R_1}+\frac{1}{R_2}\right)=\frac{U_3}{R_3}-I_q$$

Substituting the branch equation gives:

$$\frac{U_{q1}}{R_1}+I_q=\frac{U_3}{R_3}+\left(U_{q2}+U_3\right)\cdot\left(\frac{R_1+R_2}{R_1\cdot R_2}\right)$$
$$=U_3\cdot\left(\frac{1}{R_3}+\frac{R_1+R_2}{R_1\cdot R_2}\right)+U_{q2}\cdot\left(\frac{R_1+R_2}{R_1\cdot R_2}\right)$$

$$U_3=\left(\frac{U_{q1}}{R_1}-U_{q2}\cdot\frac{R_1+R_2}{R_1\cdot R_2}+I_1\right)\cdot\left(\frac{R_1\cdot R_2+R_1\cdot R_3+R_2\cdot R_3}{R_1\cdot R_2\cdot R_3}\right)^{-1}$$

Starting from this equation and with given values for U_{q1}, U_{q2}, I_q and the resistors we can calculate the values for the node voltages and the branch currents.

Determinants and Cramer's rule

The algorithm presented above derives the linear equation system from KCL and Ohm's law. Using the matrix multiplication form also presented above there is another way of determining the nodal voltages using determinants and Cramer's rule.
Determinants are special functions that associate a scalar value to a square matrix $\underline{A}$. The determinant can be used to check whether linear equation systems have a unique solution and this solution can be calculated using Cramer's rule.
Determinants for small square matrices (1 × 1, 2 × 2 and 3 × 3) are easy to calculate:

1 × 1 matrix:

$$\det \underline{\underline{A}}=\det\left(a_{11}\right)=a_{11}$$

2 × 2 matrix:

$$\det \underline{\underline{A}} = \det\begin{pmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{pmatrix} = a_{11} \cdot a_{22} - a_{12} \cdot a_{21}$$

3 × 3 matrix:

$$\det \underline{\underline{A}} = \det\begin{pmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{pmatrix} = a_{11} \cdot a_{22} \cdot a_{33} + a_{12} \cdot a_{23} \cdot a_{31} + a_{13} \cdot a_{21} \cdot a_{32}$$

$$- a3 \cdot a_{22} \cdot a_{31} - a_{12} \cdot a_{21} \cdot a_{33} - a_{11} \cdot a_{23} \cdot a_{32}$$

Consider the equation system of the first example for nodal analysis:

$$(G_1 + G_4 + G_6) \cdot U_{10} - G_4 \cdot U_{20} - G_6 \cdot U_{30} = G_1 \cdot U$$

$$-G_4 \cdot U_{10} + (G_2 + G_4 + G_5) \cdot U_{20} - G_5 \cdot U_{30} = 0$$

$$-G_6 \cdot U_{10} - G_5 \cdot U_{20} + (G_3 + G_5 + G_6) \cdot U_{30} = 0$$

This equation system can also be written in matrix multiplication form:

$$\begin{pmatrix} G_1 + G_4 + G_6 & -G_4 & -G_6 \\ -G_4 & G_2 + G_4 + G_5 & -G_5 \\ -G_6 & -G_5 & G_3 + G_5 + G_6 \end{pmatrix} \cdot \begin{pmatrix} U_{10} \\ U_{20} \\ U_{30} \end{pmatrix} = \begin{pmatrix} G_1 \cdot U \\ 0 \\ 0 \end{pmatrix} = \begin{pmatrix} I_q \\ 0 \\ 0 \end{pmatrix}$$

$$\underline{\underline{G}} \cdot \vec{U} = \vec{I}$$

Here the conductance values are known coefficients listed in conductance matrix $\underline{G}$. The (source) currents on the right side are also known. The unknown parameters we are looking for are the nodal voltages. Cramer's rule states that these unknown voltages U_i (here U_{10}, U_{20}, U_{30}) can be calculated by the determinates as follows (in fact Cramer's rule does not care about what is calculated, but is valid for linear equations with as many equations as unknowns in general):

$$U_i = \frac{\det(\underline{\underline{G_i}})}{\det(\underline{\underline{G}})}$$

$\underline{\underline{G}}$ is the conductance matrix and $\underline{\underline{G_i}}$ is constructed by replacing column i in the conductance matrix by the current vector of the right side, e.g. for U_{10}:

$$\underline{\underline{G_1}} = \begin{pmatrix} I_q & -G_4 & -G_6 \\ 0 & G_2+G_4+G_5 & -G_5 \\ 0 & -G_5 & G_3+G_5+G_6 \end{pmatrix}$$

Remembering how the determinate of a matrix looks like gives for U_{10}:

$$\det\left(\underline{\underline{G_1}}\right) = I_q \cdot (G_2+G_4+G_5)\cdot(G_3+G_5+G_6) - I_q \cdot G_5^2$$

$$\det\left(\underline{\underline{G}}\right) = (G_1+G_4+G_6)\cdot(G_2+G_4+G_5)\cdot(G_3+G_5+G_6) - 2\cdot G_4\cdot G_5\cdot G_6 - (G_2+G_4+G_5)\cdot G_6^2 - (G_1+G_4+G_6)\cdot G_5^2 - (G_3+G_5+G_6)\cdot G_4^2$$

Example of Cramer's rule

The circuit such as that shown in Fig. 5.5 can be analyzed using Cramer's rule. The circuit has four nodes. As node 1 is connected to an ideal voltage source to the reference node, the node voltage 1 is equal to U_{q1}.

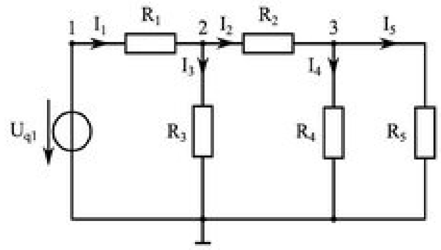


Fig. 5.5: Circuit with three nodes and the reference node for the application of Cramer's rule.

For nodes 2 and 3 we can apply KCL:

$$I_1 = I_2 + I_3$$

$$I_2 = I_4 + I_5$$

Converting the resistances into corresponding conductance values, using Ohm's law and sorting by U_2 and U_3 gives:

$$(G_1 + G_2 + G_3) \cdot U_2 - G_2 \cdot U_3 = U_{q1} \cdot G_1$$

$$-G_2 \cdot U_2 + (G_2 + G_4 + G_5) \cdot U_3 = 0$$

Or in matrix form:

$$\begin{pmatrix} G_1 + G_2 + G_3 & -G_2 \\ -G_2 & G_2 + G_4 + G_5 \end{pmatrix} \cdot \begin{pmatrix} U_2 \\ U_3 \end{pmatrix} = \begin{pmatrix} U_{q1} \cdot G_1 \\ 0 \end{pmatrix}$$

Determinants used for application of Cramer's rule are:

$$\det(\underline{\underline{G}}) = \det\begin{pmatrix} G_1 + G_2 + G_3 & -G_2 \\ -G_2 & G_2 + G_4 + G_5 \end{pmatrix} = (G_1 + G_2 + G_3) \cdot (G_2 + G_4 + G_5) - G_2^2$$

$$\det(\underline{\underline{G_2}}) = \det\begin{pmatrix} U_{q1} \cdot G_1 & -G_2 \\ 0 & G_2 + G_4 + G_5 \end{pmatrix} = (U_{q1} \cdot G_1) \cdot (G_2 + G_4 + G_5)$$

$$\det(\underline{\underline{G_3}}) = \det\begin{pmatrix} G_1 + G_2 + G_3 & U_{q1} \cdot G_1 \\ -G_2 & 0 \end{pmatrix} = (U_{q1} \cdot G_1) \cdot G_2$$

Node voltages can be calculated by these determinants as:

$$U_2 = \frac{(U_{q1} \cdot G_1) \cdot (G_2 + G_4 + G_5)}{(G_1 + G_2 + G_3) \cdot (G_2 + G_4 + G_5) - G_2^2}$$

$$U_3 = \frac{(U_{q1} \cdot G_1) \cdot G_2}{(G_1 + G_2 + G_3) \cdot (G_2 + G_4 + G_5) - G_2^2}$$

Automotive application

Since the introduction of electric lighting in vehicles the number of electric and electronic components has been steadily increasing. The nominal voltage of a typical automotive electric system is 12 V. An alternator connected to the internal combustion engine is used to generate the current needed by the large number of electronic systems. A battery, either a lead-acid storage battery or a lithium ion battery, is used as a storage element for the electrical energy if the motor and therefore the alternator stop. In this example the battery is placed in the motor compartment near to the alternator. Therefore the battery can be charged very well as the voltage drop between alternator and battery can be rather small. On the other hand the motor compartment is a very harsh environment with respect to temperature, vibration, dirt. This harsh environment generates lot of stress for the battery.

Several loads are connected to the battery via cables. Here the loads are all electronic systems of the vehicle. The loads are placed in the motor compartment (e.g. electric power steering, motor control), in the interior (e.g. dashboard, electric window lifter, seat heating) or the trunk (e.g. rear lighting).

Fig. 5.6 shows this electrical system and Fig. 5.7 shows the corresponding electric circuit. To transfer the electrical system into the corresponding electric circuit the elements are modeled: The generator is modeled by a current source and the battery by a voltage source with internal resistance. According to the lumped element method the cables between the elements are modeled by resistors R_{Wx}. As long as the details of the loads are not relevant the loads are also summarized as far as possible and modeled by resistors R_{Lx}.

By applying nodal analysis the currents and the voltages can be calculated.

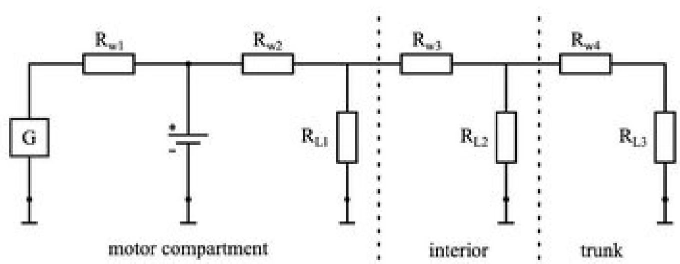


Fig. 5.6: Automotive electrical system with the battery in the engine compartment.

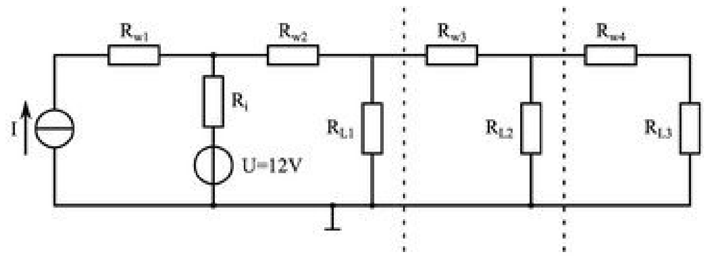


Fig. 5.7: The corresponding electric circuit, alternator modeled by a current source.

5.2 Mesh analysis

In mesh analysis the meshes of a circuit are the starting point for the calculation. In simple and small circuits the essential meshes can be used. The circuit in Fig. 5.8 consists of two essential meshes and these can be used for mesh analysis. After the meshes are found a virtual mesh current and the direction of this virtual mesh current is defined for each mesh. The labeling of the mesh currents and their direction can be chosen arbitrarily. The real branch currents are composed of the mesh currents flowing through this branch.
The KVL can be applied within the meshes. This means we get an equation for each mesh. In these equations the voltages are substituted by using Ohm's law – the corresponding voltage drop across each component expressed by the mesh current. Then the mesh currents are determined. Once the mesh currents have been found the voltages and branch currents can be calculated.

Algorithm of mesh analysis

1. Identify the meshes and draw a corresponding mesh current in each mesh (the direction is arbitrary)
2. Label the voltage drop across each component with an arrow
3. Apply KVL to each of the meshes
4. Apply Ohm's law to each voltage drop across a component and substitute the voltage by the corresponding mesh currents

5. Solve the equation system for the mesh currents
6. Calculate the branch currents
7. Determine the voltages across the elements

An example for the application of the algorithm for mesh analysis

Consider an easy example for the presented algorithm as shown in Fig. 5.8. Two meshes build the circuit and the mesh currents I_1, I_2 are identified as depicted in a clockwise manner. Based on the mesh currents the current through resistor R_1 is I_1, through R_2 it is I_2 and through R_3 it is I_1 - I_2.

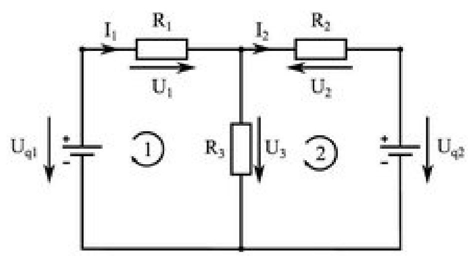


Fig. 5.8: An example for mesh analysis.

1. Identify the meshes and draw a corresponding mesh current in each mesh (the direction is arbitrary)

   – See mesh currents I_1, I_2 in Fig. 5.8

2. Label the voltage drop across each component with an arrow

   – See voltages U_1, U_2, U_3 in Fig. 5.8

3. Apply KVL to each of the meshes

   – Mesh 1:

$$U_1 + U_3 = U_{q1}$$

   – Mesh 2:

$$U_2 + U_3 = U_{q2}$$

4. Apply Ohm's law to each voltage drop across a

component and substitute the voltages by the corresponding mesh currents

– Mesh 1:

$$R_1 \cdot I_1 + R_3 \cdot (I_1 - I_2) = U_{q1}$$

– Mesh 2:

$$-R_2 \cdot I_2 + R_3 \cdot (I_1 - I_2) = U_{q2}$$

5. Solve the equation system for the mesh currents

$$\begin{pmatrix} R_1 + R_3 & -R_3 \\ -R_3 & R_2 + R_3 \end{pmatrix} \cdot \begin{pmatrix} I_1 \\ I_2 \end{pmatrix} = \begin{pmatrix} U_{q1} \\ U_{q2} \end{pmatrix}$$

$$I_1 = \frac{\det\left(\underline{\underline{R_1}}\right)}{\det\left(\underline{\underline{R}}\right)} = \frac{U_{q1} \cdot (R_2 + R_3) - U_{q2} \cdot R_3}{(R_1 + R_3) \cdot (R_2 + R_3) - R_3^2}$$

$$I_2 = \frac{\det\left(\underline{\underline{R_2}}\right)}{\det\left(\underline{\underline{R}}\right)} = \frac{-U_{q2} \cdot (R_1 + R_3) + U_{q1} \cdot R_3}{(R_1 + R_3) \cdot (R_2 + R_3) - R_3^2}$$

6. Calculate the branch currents

$$I_{R1} = I_1$$

$$I_{R2} = -I_2$$

$$I_{R3} = I_1 - I_2$$

7. Determine the voltages across the elements

$$U_1 = R_1 \cdot I_1$$

$$U_2 = -R_2 \cdot I_2$$

$$U_3 = R_3 \cdot (I_1 - I_2)$$

For given values of R_1 = 1 Ω, R_2 = 1 Ω, R_3 = 2 Ω and U_{q1} = 5 V, U_{q2} = 10 V the voltages and currents yield I_1 = -1 A, I_2 = -4 A,

U_1 = -1 V, U_2 = 4 V, U_3 = 6 V. Current $I_3 = (I_1 - I_2)$ = 3 A through resistor R_3 flows parallel to the voltage drop U_3.

An example for mesh analysis with an ideal current source

The circuit to be analyzed using the algorithm of mesh analysis is given in Fig. 5.9.

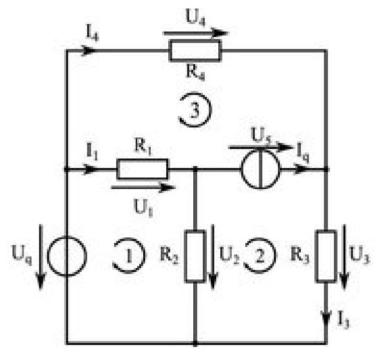


Fig. 5.9: Circuit for mesh analysis, mesh currents indicated clockwise by arrows.

Three meshes are identified and the corresponding mesh currents in clockwise direction are labeled I_1, I_2, I_3. Voltage drops are labeled accordingly. Applying KVL and Ohm's law to the three meshes results in:

– Mesh 1:

$$U_1 + U_2 = U_q$$

– Using Ohm's law for $U_1 = R_1 \cdot (I_1 - I_3)$ and $U_2 = R_2 \cdot (I_1 - I_2)$ we get:

$$U_q = I_1 \cdot (R_1 + R_2) - I_2 \cdot R_2 - I_3 \cdot R_1$$

– Mesh 2:

$$U_5 + U_3 - U_2 = 0$$

– U_5 is unknown for the moment as the voltage drop across an ideal current source is unknown. With $U_3 = R_3 \cdot I_2$ we can express U_5 by the mesh currents and the resistances:

$$U_5 + I_2 \cdot R_3 - R_2 \cdot (I_1 - I_2) = 0$$

$$\Rightarrow U_5 = I_1 \cdot R_2 - I_2 \cdot (R_2 + R_3)$$

– Mesh 3:

$$U_5 + U_1 - U_4 = 0$$

– With $U_4 = R_4 \cdot I_3$ we get for U_5:

$$U_5 = -I_1 \cdot R_1 + I_3 \cdot (R_4 + R_1)$$

– Using the two equations from mesh 2 and 3 for U_5 gives:

$$I_1 \cdot (R_2 + R_1) - I_2 \cdot (R_2 + R_3) - I_3 \cdot (R_1 + R_4)$$

– Up to now we have had two equations for the three mesh currents and we need a third one to solve for the three currents. This third equation can be derived from the ideal current source I_q:

$$I_q = I_2 - I_3$$

– This leads us to the equation system for the three currents:

$$I_q = I_2 - I_3$$

$$I_1 \cdot (R_2 + R_1) - I_2 \cdot (R_2 + R_3) - I_3 \cdot (R_1 + R_4) = 0$$

$$U_q = I_1 \cdot (R_1 + R_2) - I_2 \cdot R_2 - I_3 \cdot R_1$$

For given values of U_q = 6 V, I_q = 7 A, R_1 = 3 Ω, R_2 = 2 Ω, R_3 =

4 Ω, R_4 = 7 Ω the matrix looks like

$$\begin{pmatrix} 0 & 1 & -1 \\ 5\Omega & -6\Omega & -10\Omega \\ 5\Omega & -2\Omega & -3\Omega \end{pmatrix} \cdot \begin{pmatrix} I_1 \\ I_2 \\ I_3 \end{pmatrix} = \begin{pmatrix} I_q \\ 0 \\ U_q \end{pmatrix}$$

Using Cramer’s rule this yields for the currents I_1 = 2 A, I_2 = 5 A, I_3 = –2 A.

Independent meshes: complete tree of a circuit

So far rather simple circuits have been analyzed using essential meshes. In general the independent meshes for mesh analysis of a complex and big circuit can be found using a complete tree of the circuit. A tree of a circuit is a line along the branches of the circuit connecting all nodes without a loop. In general several trees exist for a given circuit and any of these trees can be used to find the independent meshes. Fig. 5.10 shows a circuit with four nodes. In addition two complete trees are depicted exemplarily. Both trees connect all four nodes without forming a loop. After a tree is defined, the branches that are not part of the tree (links) are used to close the meshes step by step. A mesh must not contain more than one link and the currents in the links are the mesh currents for the analysis.

In the first example (mid of Fig. 5.10) mesh M2 is closed via the link containing the voltage source. Meshes M1 and M3 are closed using the links with R_1 and R_5 respectively. In fact these are the essential meshes again. In the second example (right side of Fig. 5.10) another tree is chosen. M1 is again closed using the link with R_1. M2 is closed using the link via R_3. For M3 the mesh is closed via the voltage source. This mesh is not an essential mesh. In both examples all meshes contain just one link and the currents in these links are the corresponding mesh currents.

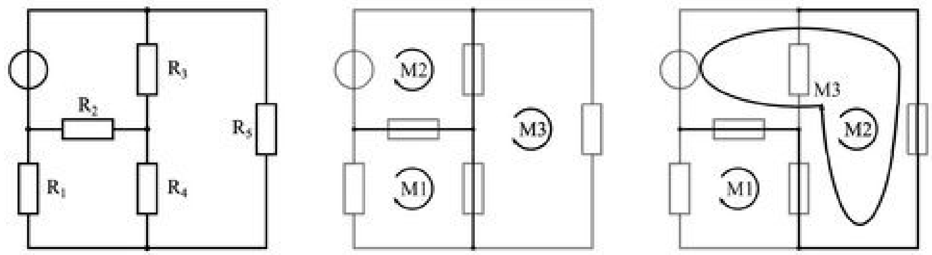


Fig. 5.10: Electric circuit with examples of two complete trees for the definition of meshes M1-M3.

Independent meshes of complex circuits can be found by using this method. After the meshed are defined mesh analysis can be done as described before.

Automotive application

We have again a look at the electrical system of a vehicle. However this time the battery is placed inside the trunk to reduce the environmental stress. But as can be seen in Fig. 5.11 the resistance between battery and alternator is now higher and charging the battery is worse as the voltage drop from alternator to battery is higher. Whether this voltage drop is acceptable should be calculated, e.g. by applying mesh analysis to determine the currents and the voltages. Of course also by using nodal analysis…

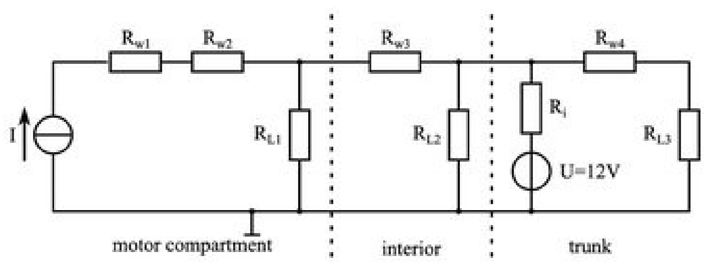


Fig. 5.11: Electric circuit with the battery located inside the trunk.

5.3 Linearity and Superposition

Mathematically a function is said to be linear if it satisfies two properties: homogeneity (scaling) and additivity (superposition). For an arbitrary function f(x) homogeneity is given by:

$$f(Kx) = Kf(x)$$

K is a constant scalar value. Additivity is given by:

$$f(x_1 + x_2) = f(x_1) + f(x_2) = y_1 + y_2$$

For a linear circuit (or system) in which excitations x_1 and x_2 produce responses y_1 and y_2, respectively, the application of K_1x_1 and K_2x_2 together (K_1 and K_2 being constants) results in a response of $K_1y_1 + K_2y_2$.

A circuit consisting of independent sources, linear dependent sources and linear elements (like resistors) is said to be a linear circuit. For a linear circuit consisting of several independent sources the net response in any element, according to the principle of superposition, is the algebraic sum of the individual responses produced by each of the independent sources acting only by itself. While each independent source acting on the circuit is considered separately, the other independent sources are suppressed. The effect of any dependent source, however, must be included in evaluating the response due to each of the independent sources. In brief:

In linear networks we can determine the results of different sources by analyzing the behavior of the circuit independently for each source and the superposition of the results for the total number of sources. But how are independent sources suppressed, which were not considered during analysis of another source?

Voltage sources are replaced by short circuits (insignificant internal resistance of the voltage source, ideal voltage source). Current sources are replaced by open circuits (infinite high internal resistance).

An example for superposition

An example for the application of principle of superposition is depicted in Fig. 5.12. Both voltage sources as well as the three resistors are known and we are looking for the current I_1

through resistor R_1.

We start analyzing the circuit by suppressing all sources but one, e.g. U_1. So U_2 is suppressed and replaced by a short circuit as shown in the middle of Fig. 5.12. The branch currents are now labeled with an extra suffix '1' to indicate that these are the first partial currents. Now the parallel connected resistors R_2 and R_3 are in series with resistor R_1 and the current I_{11} through resistor R_1 is:

$$I_{11} = \frac{U_1}{R_1 + \frac{R_2 \cdot R_3}{R_2 + R_3}} = U_1 \cdot \frac{R_2 + R_3}{R_1 \cdot R_2 + R_2 \cdot R_3 + R_3 \cdot R_1}$$

The second step is to analyze the circuit using U_2 and suppressing U_1, refer to right side of Fig. 5.12. This time the parallel connected resistors R_1 and R_3 are in series with resistor R_2 and the current I_{22} through resistor R_2 is

$$I_{22} = \frac{U_2}{R_2 + \frac{R_1 \cdot R_3}{R_1 + R_3}} = U_2 \cdot \frac{R_1 + R_3}{R_1 \cdot R_2 + R_2 \cdot R_3 + R_3 \cdot R_1}$$

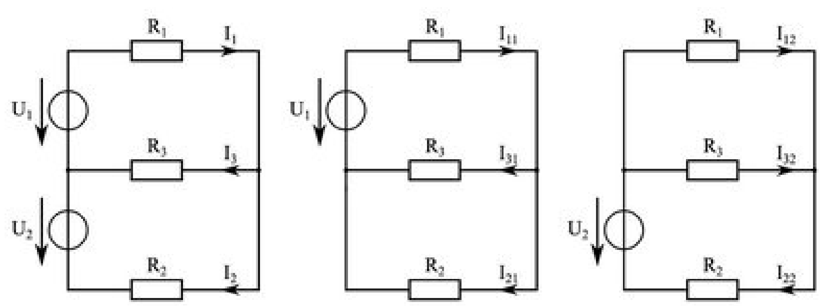


Fig. 5.12: An example for the application of superposition.

Using the current divider rule for the parallel resistors results in:

$$I_{12} = I_{22} \cdot \frac{R_3}{R_1 + R_3} = U_2 \cdot \frac{R_3}{R_1 \cdot R_2 + R_2 \cdot R_3 + R_3 \cdot R_1}$$

In the end the total current through resistor R_1 sums up from the two partial currents:

$$I_1 = I_{11} + I_{12} = U_1 \cdot \frac{R_2 + R_3}{R_1 \cdot R_2 + R_2 \cdot R_3 + R_3 \cdot R_1} + U_2 \cdot \frac{R_3}{R_1 \cdot R_2 + R_2 \cdot R_3 + R_3 \cdot R_1}$$

An example for superposition with linear dependent sources

How can we deal with linear dependent sources after introducing superposition with independent sources? The effect of any dependent source must be included in evaluating the response due to each of the independent sources. The following example illustrates the analysis. The circuit contains an independent voltage source, one independent current source and one linear voltage controlled current source. The current of the dependent source is given by the voltage drop across resistor R_4 multiplied with a factor of B.

Determine the voltage across the resistor R_4 of the circuit depicted in Fig. 5.13 by the application of superposition.

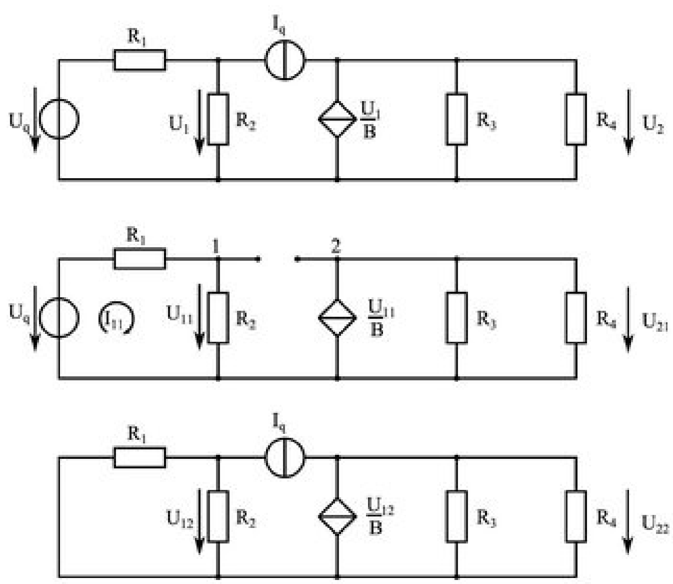


Fig. 5.13: An example for superposition for a circuit with a dependent current source; complete circuit (top); circuit with suppressed current source (open circuit, middle); circuit with suppressed voltage source (short circuit, bottom).

For superposition we have to suppress the independent sources in turn.

– Step 1: Replace the independent current source by an open circuit as shown in middle of Fig. 5.13. For node 1 we get by KCL:

$$\left(\frac{1}{R_3}+\frac{1}{R_4}\right)\cdot U_{21}=\frac{U_{11}}{B}$$

$$U_{21}=\frac{U_{11}}{B}\cdot\frac{R_3\cdot R_4}{R_3+R_4}$$

– Applying KVL to the left mesh gives:

$$(R_1+R_2)\cdot I_{11}=U_q$$

$$\Rightarrow I_{11}=\frac{U_q}{R_1+R_2}$$

$$\Rightarrow U_{11}=I_{11}\cdot R_2$$

– Therefore the voltage across resistor R_4 from this first part of the solution is:

$$U_{21}=\frac{U_{11}}{B}\cdot\frac{R_3\cdot R_4}{R_3+R_4}$$

– Step 2: Replace the independent voltage source with a short circuit (s. bottom of Fig. 5.13).
– At node 1, KCL gives

$$I_q=\left(\frac{1}{R_1}+\frac{1}{R_2}\right)\cdot U_{12}$$

$$U_{12}=I_q\cdot\frac{R_1\cdot R_2}{R_1+R_2}$$

– At node 2, KCL gives

$$\left(\frac{1}{R_3}+\frac{1}{R_4}\right)\cdot U_{22}=\frac{U_{12}}{B}-I_q$$

$$U_{22}=\left(\frac{U_{12}}{B}-I_q\right)\cdot\left(\frac{R_3\cdot R_4}{R_3+R_4}\right)$$

Finally the total net response for the voltage drop across resistor R_4 by superposition is:

$$U_2=U_{21}+U_{22}=\frac{U_{11}}{B}\cdot\frac{R_3\cdot R_4}{R_3+R_4}+\left(\frac{U_{12}}{B}-I_q\right)\cdot\left(\frac{R_3\cdot R_4}{R_3+R_4}\right)$$

For given values of U_q = 18 V, I_q = 6 A, R_1 = 6 Ω, R_2 = 12 Ω, R_3 = 80 Ω, R_4 = 20 Ω and B=3Ω the voltage across resistor R_4 is U_{22} = 96 V.

Automotive application
Superposition can of course also be used to analyze the circuits of the electric system as given in Fig. 5.7 and Fig. 5.8.

An example where the method of superposition fails
The method of superposition is only valid for linear networks. If non-linear elements are part of the circuit it fails due to the non-linear behavior. Fig. 5.14 shows an example circuit with a Zener diode (special kind of diode with rather non-linear behavior). First we determine the current I_{Z1} for U_{q1} and U_{q2} = 0 V. Afterwards we determine the current I_{Z2} for U_{q2} and U_{q1} = 0 V. Now we add the voltages U_{Z1} and U_{Z2} and compare the resulting current value with I_{Z1} + I_{Z2}. As can be seen directly in the diagram the method yields the wrong results. Don't use superposition for non-linear components.

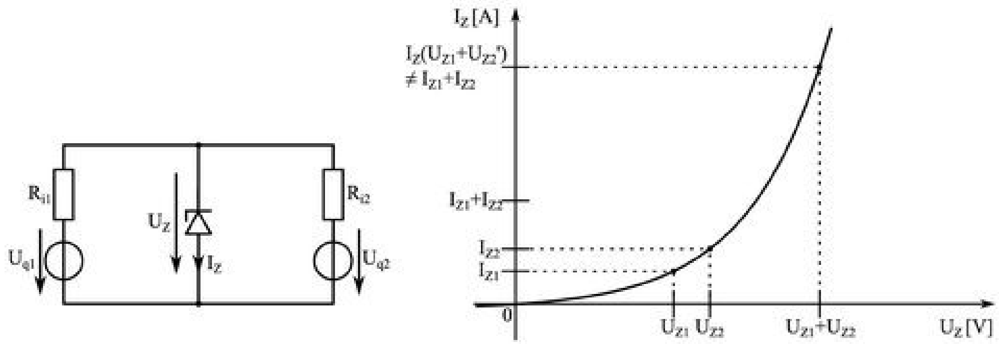


Fig. 5.14: An example for a non-linear network with a Zener diode; the method of superpopsition fails in this case.

5.4 Two-terminal circuit and Thévenin’s theorem

Any electric circuit with just two external terminals is called a two-terminal circuit. Without any current or voltage source inside it is a passive two-terminal circuit. Therefore a passive two-terminal circuit is an arbitrary configuration of resistors such as depicted in Fig. 5.15.

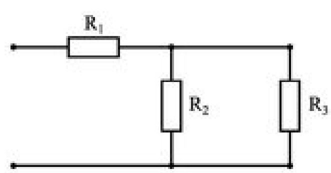


Fig. 5.15: A passive two-terminal circuit composed of resistors.

This arbitrary configuration of resistors can be replaced by a single, equivalent resistor. In the example the equivalent resistor R_{eq} of the passive two-terminal circuit is:

$$R_{eq} = R_1 + \frac{R_2 \cdot R_3}{R_2 + R_3}$$

In contrast to the passive two-terminal circuit an active two-terminal circuit does contain sources. Like any resistor, the configuration of a passive two-terminal circuit can be replaced by an equivalent resistor, any active two-terminal circuit can be replaced by a voltage source in series with a resistor. This combination of voltage source and resistor is the equivalent voltage source, or the Thèvenin equivalent circuit of the original circuit. Fig. 5.16 shows a variable load resistor connected to an active two-terminal circuit with arbitrary internal configuration of linear elements. Consider we don't care about the details of the internal configuration of the two-terminal circuit but we want to know the behavior of the load circuit, e.g. to know the maximum power transfer to the load resistor. Changing the load resistor results in a linear dependency of the terminal voltage U of the load current I:

$$U = U_q - I \cdot R_i$$

Here U_q and R_i are constants and this equation is the same as the equation for a voltage source U_q with an internal resistor R_i connected in series.

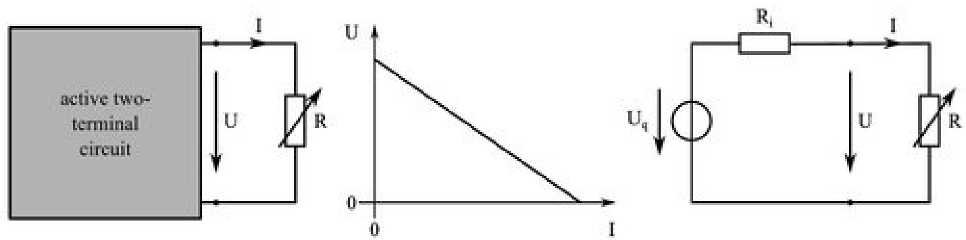


Fig. 5.16: Active two-terminal circuit with arbitrary internal configuration (left); corresponding voltage vs current diagram (middle); Thevenin equivalent circuit (right).

Thévenin's theorem says that an arbitrary linear two-terminal circuit (network of linear dependent and independent sources and elements) can be substituted by a real voltage source (ideal voltage source and internal resistor in series) if just the behavior at the terminals is regarded. In terms of this theorem, the circuit of voltage source and internal resistor on the right side of Fig. 5.16 is Thévenin's equivalent to the active two-terminal circuit on the left side. If we are just interested in the load circuit (here a single resistor) it can simplify the analysis if we use this theorem.

Example: We want to determine the maximum power transfer to

the load resistor. Using the original two-terminal circuit connected to the load resistor we have to calculate the power transfer for changing load resistor values. Depending on complexity of the two-terminal circuit this might be difficult. Using Thévenin equivalent immediately reveals the solution: maximum power is transferred to the load resistor if it is equal to the internal resistor of the Thévenin equivalent.
How can we (easily) determine the two parameters (U_q and R_i) of the Thévenin equivalent?

Algorithm to determine the Thévenin equivalent

1. The load network must not contain dependencies of the supply network
2. Determine the open loop voltage at the terminals of the supply network (i.e. load resistor $R_L = \infty\ \Omega$). This yields $U_q = U_{rep} = U$
3. Determine the inner resistance if the supply circuit $R_i = R_{rep}$. Two possibilities:

   – First possibility:

   - Determine the short-circuit current I_{sc} (i.e. $R_L = 0\ \Omega$) with

$$R_{rep} = \frac{U_{rep}}{I_{SC}}$$

   – Second possibility:

   - Short-circuit of all ideal voltage sources
   - Remove all ideal current sources (open load)
   - Leave depending sources as is
   - Look from the outside into the modified arbitrary network and determine the resistance that you “see from the outside”

- This resulting resistor will be the replacement resistor of the Thévenin equivalent

An example for Thévenin's theorem

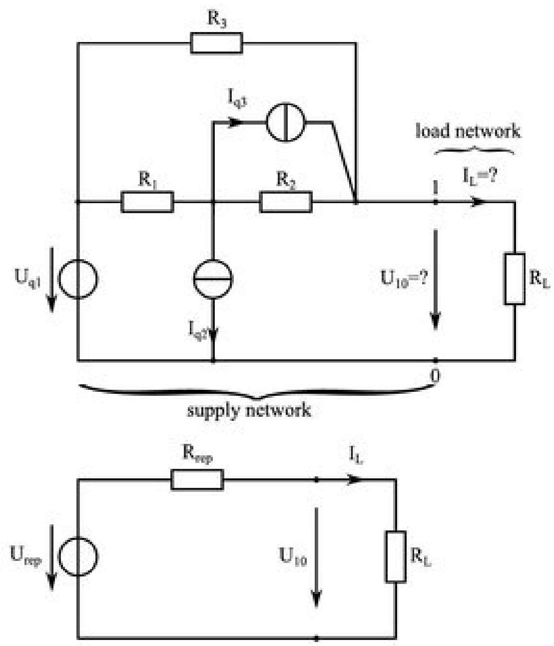


Fig. 5.17: An example for a circuit of a supply network and a simple load network (top); supply network is to be replaced by the Thévenin equivalent (bottom).

In this example (see Fig. 5.17) we are not interested in the internals of the supply network, but just in the behavior of the terminals 1 and 0 and we want to know the value of the load resistor for maximum power transfer from the source. Therefore we simplify the supply network by applying Thévenin's theorem. First we regard the supply network without the load as depicted in Fig. 5.18:

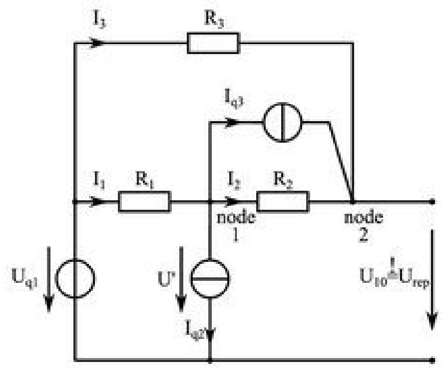


Fig. 5.18: Supply network of the example, currents and voltages are depicted in the figure.

The currents can be expressed by the nodal voltages using Ohm’s law:

$$I_1 = \frac{U_{q1} - U'}{R_1}$$

$$I_2 = \frac{U' - U_{10}}{R_2}$$

$$I_3 = \frac{U_{q1} - U_{10}}{R_3}$$

– Node 1:

$$-I_1 + I_2 + I_{q2} + I_{q3} = 0$$

$$\Rightarrow \frac{U' - U_{q1}}{R_1} + \frac{U' - U_{10}}{R_2} = -I_{q2} - I_{q3}$$

– Node 2:

$$-I_2 - I_3 - I_{q3} = 0$$

$$\Rightarrow \frac{U' - U_{10}}{R_2} + \frac{U_{q1} - U_{10}}{R_3} = -I_{q3}$$

Two equations to determine the two node voltages $U_{10} = U_{rep}$ and U′. Resorting of the two equations yields the equation system in matrix form:

$$\begin{pmatrix} R_1 + R_2 & -R_1 \\ R_3 & -(R_2 + R_3) \end{pmatrix} \cdot \begin{pmatrix} U' \\ U_{rep} \end{pmatrix} = \begin{pmatrix} R_2 \cdot U_{q1} - R_1 \cdot R_2 \cdot (I_{q2} + I_{q3}) \\ -R_2 \cdot R_3 \cdot I_{q3} - R_2 \cdot U_{q1} \end{pmatrix}$$

The following values are used to calculate U_{rep} and U′:
R_1 = 1 Ω, R_2 = 2 Ω, R_3 = 6 Ω and Uq_1 = 20 V, I_{q2} = 15 A, I_{q3} = 15 A.

$$\underline{\underline{R}} = \begin{pmatrix} 3\Omega & -1\Omega \\ 6\Omega & -8\Omega \end{pmatrix}$$

$$\det\left(\underline{\underline{R}}\right) = -18\Omega^2$$

$$\underline{\underline{R_2}} = \begin{pmatrix} 3\Omega & -20V^2/A \\ 6\Omega & -220V^2/A \end{pmatrix}$$

$$\det\left(\underline{\underline{R_2}}\right) = -540\Omega^2 V$$

$$U_{rep} = 30V$$

After determination of U_{rep} the inner resistor R_{rep} of the Thévenin equivalent is calculated by removing the sources from the supply network (replace voltage sources by short-cut and current sources by open circuit), see Fig. 5.19:

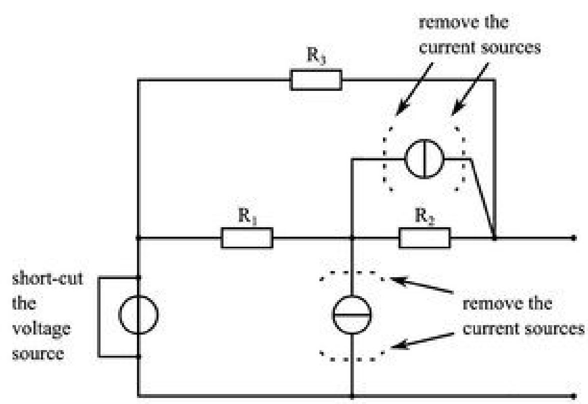


Fig. 5.19: Supply network of the example for determination of R_{rep}: removal of sources.

The resulting circuit after removal of the sources is just a series combination of two resistors (R_1 and R_2) in parallel to a third resistor R_3.

$$R_{rep} = \frac{(R_1 + R_2) \cdot R_3}{R_1 + R_2 + R_3} = 2\Omega$$

Finally we can draw the Thévenin equivalent circuit in Fig. 5.20.

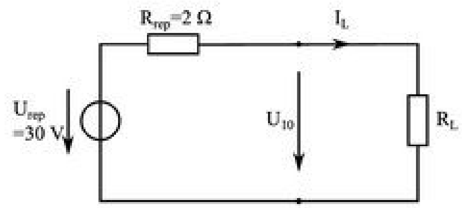


Fig. 5.20: The Thévenin equivalent circuit with values for U_{rep} and R_{rep}.

Maximum power P_{Lmax} is transferred to the load if the load resistor equals the internal resistor of the voltage source (according to the rule of maximum power transfer):

$$R_L = R_{rep} = 2\Omega$$

$$U_{10} = U_{rep} \cdot \frac{R_L}{R_L + R_{rep}} = 15V$$

$$I_L = \frac{U_{10}}{R_L} = 7.5A$$

Automotive application

Consider again the electric system as given in Fig. 5.7. But this time we are just interested in the analysis of the electrical loads R_{Lx} and are not interested in the internal details of the alternator/battery system. Therefore we would like to replace the alternator/battery system with a simple real voltage source. By application of Thévenin’s theorem, a replacement can be achieved and the Thévenin equivalent can be used for the analysis of the load system.

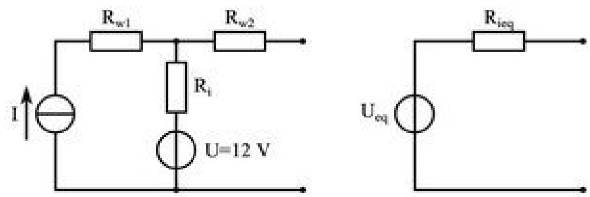


Fig. 5.21: A sub-circuit with alternator, battery and resistances (left) and corresponding Thévenin equivalent (right).

5.5 Norton's theorem

Norton's theorem is the complement to Thévenin's theorem. While Thévenin's theorem converts a linear network into a real voltage source, Norton's theorem converts an arbitrary linear network into a current source:

An arbitrary network of linear elements can be substituted by the Norton equivalent.

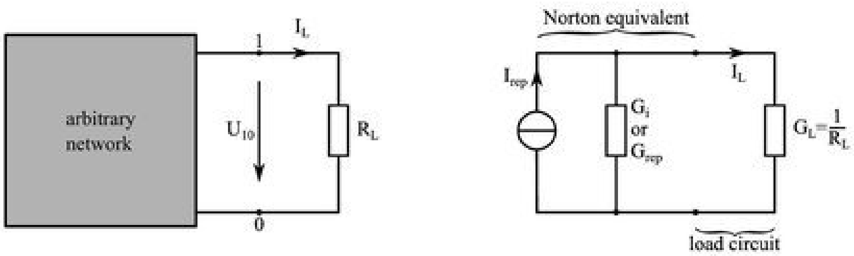


Fig. 5.22: Arbitrary (supply) network (left) and Norton's equivalent (real current source, right).

Algorithm to derive Norton's equivalent

1. Remove the load resistor in the original network
2. Replace the removed resistance by an electric short-circuit
3. Determine the short-cut current (in the "load" branch): $I_{sc} = I_{rep}$
4. Determine the replacement conductance G_{rep}
5. Construct Norton's equivalent with I_{rep} and G_{rep}
6. Determine the load current I_L with Norton's equivalent as the supply circuit

The circuit already analyzed by Thévenin's theorem is used as an example for Norton's theorem.

An example for Norton's theorem

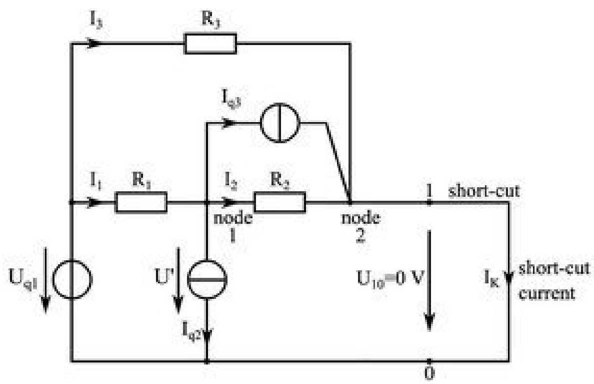


Fig. 5.23: Supply circuit for application of Norton's theorem, load circuit is short-cut for determination of I_{rep}.

As the load branch is short-cut the terminal voltage $U_{10} = 0$ V. The branch currents are:

$$I_1 = \frac{U_{q1} - U'}{R_1}$$

$$I_2 = \frac{U'}{R_2}$$

$$I_3 = \frac{U_{q1}}{R_3}$$

– Node 1:

$$I_1 - I_2 - I_{q2} - I_{q3} = 0$$

$$\Rightarrow \frac{U_{q1} - U'}{R_1} - \frac{U'}{R_2} = I_{q2} + I_{q3}$$

$$U' = \left(\frac{U_{q1}}{R_1} - I_{q2} - I_{q3} \right) \cdot \left(\frac{1}{R_1} + \frac{1}{R_2} \right)^{-1}$$

– Using the values from Thévenin's example and converting the resistor values into corresponding conductance values the node voltage U′ is:

$$U' = -\frac{20}{3}V = -6.67V$$

– Node 2:

$$I_{SC} = I_{rep} = I_2 + I_3 + I_{q3} = \frac{U_{q1}}{R_3} + \frac{U'}{R_2} + I_{q3} = 15A$$

– Determination of the replacement conductance G_{rep}:

$$G_{rep} = G_3 + \frac{G_1 \cdot G_2}{G_1 + G_2} = 0.5S$$

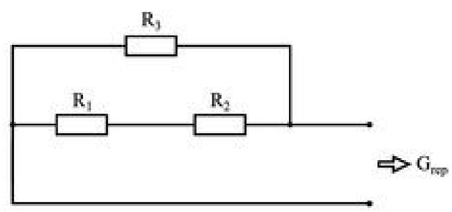


Fig. 5.24: Replacement circuit of supply network for determination of G_{rep}.

Finally Norton’s equivalent looks like depicted in Fig. 5.25:

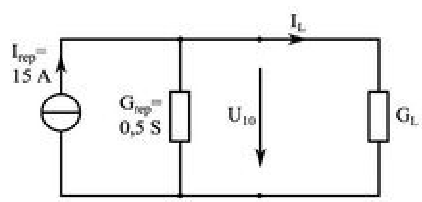


Fig. 5.25: Norton’s equivalent with I_{rep} and G_{rep}.

The current of the source is split into a current through the internal resistance and through the external load. Maximum power is transferred to the load if the internal and external resistances are equal, just like for Thévenin’s equivalent.
For a load resistor of 2 Ω the current is split between the internal and the load resistor according to the current divider rule:

$$I_L = I_{rep} \cdot \frac{G_L}{G_L + G_{rep}} = 7.5A$$

6 Operational amplifier

6.1 Operational amplifier

The voltage controlled voltage source was introduced in the chapter 4.7. Let’s recall this circuit and add two resistors to the circuit like shown in Fig. 6.1. The voltage controlled voltage source amplifies the voltage different $U' = U_1 - U_2$ by amplification factor –A. Between U_1 and U_2 there is an open load ($R = \infty\ \Omega$). There is one resistor R_1 in one of the input terminal paths and a feedback resistor R_2 from one output terminal to the same input terminal. For simplicity we set U_2 as the reference point (0 V) and also refer the output voltage U_a to this reference point.

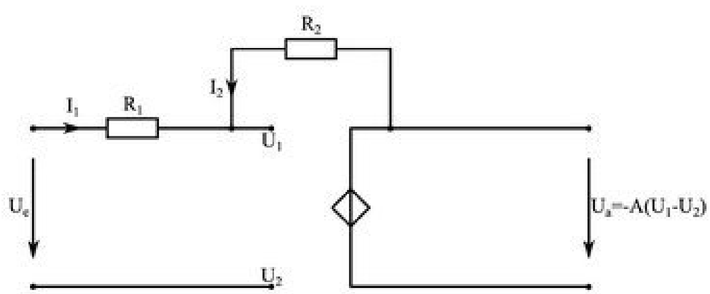


Fig. 6.1: Voltage controlled voltage source with feedback loop via resistor R_2.

What about the output voltage U_a? Is it affected by the resistors? How and why?

According to KCL the currents I_1 and I_2 sum to zero at input terminal U_1. With Ohm’s law KCL can be written as:

$$I_1 + I_2 = \frac{U_e - U'}{R_1} + \frac{U_a - U'}{R_2} = 0$$

Using the amplification of the voltage source, $U_a = -A \cdot U'$ yields:

$$R_2 \cdot \left(U_e + \frac{U_a}{A}\right) + R_1 \cdot \left(U_a + \frac{U_a}{A}\right) = 0$$

$$\Rightarrow -R_2 \cdot U_e = U_a \cdot \left(\frac{R_1 + R_2}{A} + R_1\right)$$

$$\Rightarrow U_a = -U_e \cdot R_2 \cdot \left(\frac{R_1 + R_2}{A} + R_1\right)^{-1}$$

Due to the feedback loop the output voltage depends not only on the amplification factor A but also from the resistors R_1 and R_2. This voltage controlled voltage source with an infinite input resistance R_e and open loop gain A (where $U_a = A \cdot (U_2 - U_1)$, A positive or negative) is called ideal voltage amplifier. As R_e is infinite no current will enter the input terminals, and as the output is an ideal voltage source, U_a is driven by the amplifier regardless of load connected to the output. Terminal U_2 (labeled with '+') is called the non-inverting input and terminal U_1 (labeled with '–') is called the inverting input.

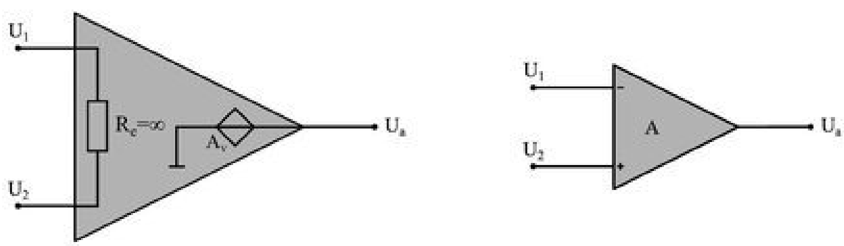


Fig. 6.2: Circuit and model of an ideal voltage amplifier with gain A.

Considering a very high amplification factor (A → ∞) the output voltage becomes independent from A and is just determined by the ratio of the resistors:

$$\lim_{A \to \infty} U_a = -U_e \cdot \frac{R_2}{R_1}$$

The output voltage is inverted compared to the input voltage and amplified by the ratio of the two resistors. The ideal amplifier can be used to obtain a defined amplification due to the external resistors R_1 and R_2. In case of $A \rightarrow \infty$ this feedback loop forces the output to stay finite (just determined by the ratio of the resistors) even though the gain is infinite. This is due to the fact that the voltage at the internal (infinite) resistor tends to zero:

$$\lim_{A \to \infty} U' = -U_a \cdot \frac{1}{A} = 0$$

As the ideal voltage amplifier with infinite gain is very important, it has its own name, the operational amplifier (or OpAmp). The major property of an OpAmp is the amplification of an input voltage that can be measured at the output of the device. The symbol that is used in electric circuits is depicted in Fig. 6.3:

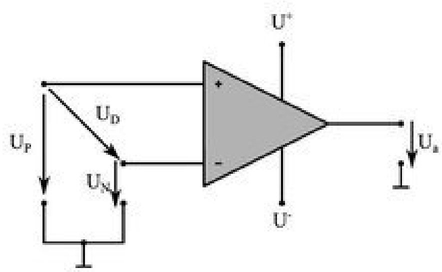


Fig. 6.3: Symbol of an OpAmp; U_N: voltage at inverting input; U_P: voltage at non-inverting input; U_D: differential input voltage; U_a: voltage at output; U^+ and U^-: power supply terminals.

U^+ and U^- are the power supply terminals for the operational amplifier, e.g. +15 V and -15 V. The input voltages must not exceed the supply voltage, otherwise the OpAmp can be destroyed:

$$U^- < U_P < U^+$$

$$U^- < U_N < U^+$$

Under normal conditions the usable output voltage is limited by the power supply voltages and the range usually is something like:

$$U^- + 1.5V < U_a < U^+ - 1.5V$$

The ideal OpAmp is characterized by an infinite open loop gain A (hence the output has to be limited somehow by a feedback resistor in most cases), infinite input resistance, zero output resistance and frequency independent amplification. Real OpAmps differ from this ideal version as shown in Tab. 6.1.

Tab. 6.1: DC characteristics of ideal and real OpAmps.

	Ideal OpAmp	**Real OpAmp**
Open loop gain	$A = \infty$	$A \approx 10^4\text{-}10^7$
Input resistance	$R_e = \infty\ \Omega$	$R_e > 1\ M\Omega$
Output resistance	$R_a = 0\ \Omega$	$R_a \approx 1\text{–}100\ \Omega$

The input-output characteristic ($U_a = U_a(U_D)$) of an OpAmp is depicted exemplarily in the following diagram (see Fig. 6.4). Notice that due to the very high open loop gain A (or V_0) the output of the OpAmp is already in saturation for rather low input voltage differences in µV range as shown in this example.

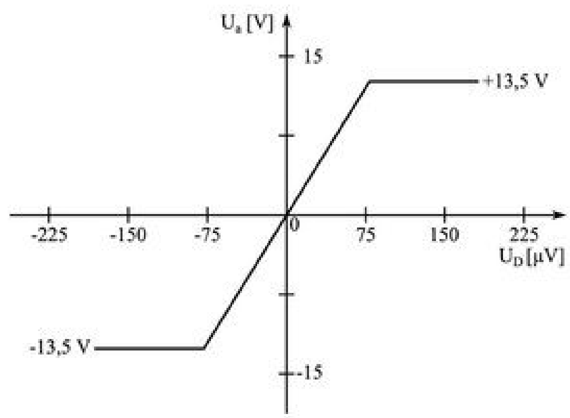


Fig. 6.4: Output voltage of a real OpAmp as a function of the differential input.

Real OpAmps are electronic semiconductor devices composed of several transistors, capacitors and diodes. Fig. 6.5 shows the internal circuit of a µA741 OpAmp from STMicroelec-tronics.

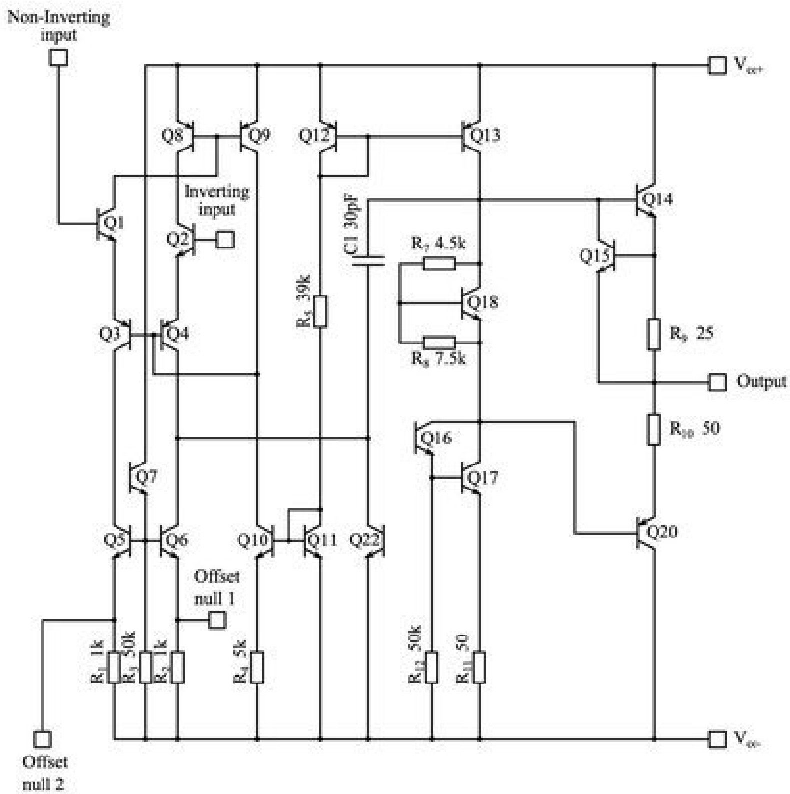


Fig. 6.5: Circuit of the µA741 OpAmp from STMicorelectronics (µA741 datasheet).

Due to its properties the OpAmp is a very well known device for all kind of applications, from simple amplification to complex analog calculations.

Several standardized packages are available for the packaging of the silicon dies of an OpAmp. For example the µA741 is housed in a through hole DIP-8 package with 8 pins (see Fig. 6.6). The package dimensions are 9.5 mm by 7.8 mm and the height of the package is about 4 mm. The pins have a length of 3.2 mm. A smaller surface mount package type is SOP-8 with a size of 4.2 mm by 5 mm and a height of 1.5 mm. The pins are just about 1 mm.

Fig. 6.6: Examples of packages for operational amplifier: DIP-8 (package of μA741 OpAmp, left); SOP-8 (right). Package drawings by Infineon Technologies AG.

6.2 Operational amplifier circuits

Comparator

If the OpAmp is used without a feedback loop its function is rather simple: Two voltages are supplied to the inverting and non-inverting input respectively and the output is, due to infinite (or at least very high) open loop gain just the maximum positive or negative voltage, depending on which input voltage is higher: $U_a = A \cdot (U_P - U_N) = A \cdot U_D$. The characteristics of a comparator is depicted in Fig. 6.7.

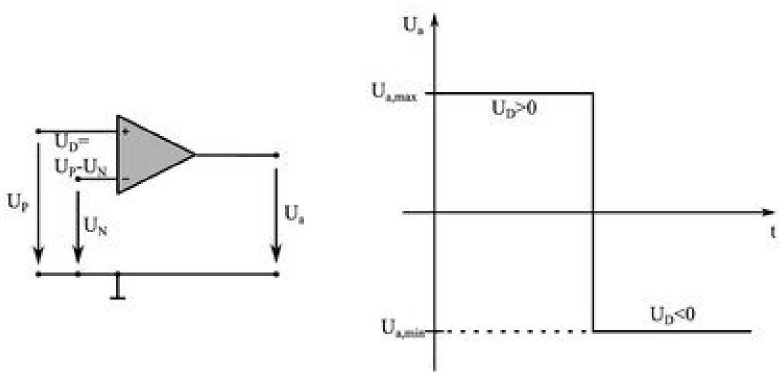


Fig. 6.7: Simple comparator circuit (left) and output voltage as a function of U_D (right).

Inverting amplifier

One of the easiest circuits with an ideal OpAmp and a feedback loop is given below (also refer to first example of this chapter with the voltage controlled voltage source):

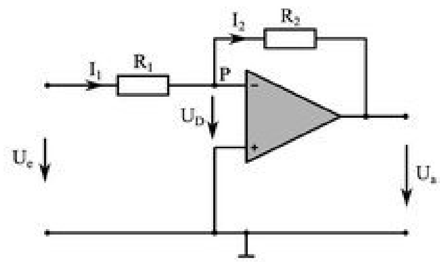


Fig. 6.8: An inverting amplifier.

As the input resistance is infinite for an ideal OpAmp, KCL at the inverting input P yields (notice that direction of I_2 is opposite to the direction of I_2 in the examples of the voltage controlled voltage source):

$$I_1 - I_2 = 0$$

Expressing the currents by the voltage drops across the resistors gives:

$$\frac{U_e - U_D}{R_1} = \frac{U_D - U_a}{R_2}$$

With the open loop gain $U_a = A \cdot U_D$:

$$\frac{U_e - U_D}{R_1} = \frac{U_D - A \cdot U_D}{R_2}$$

For limit of $A \to \infty$: $U_D \to 0$ V. Finally:

$$\Rightarrow \frac{U_a}{U_e} = -\frac{R_2}{R_1}$$

Thus this circuit inverts and amplifies the input voltage and is called inverting amplifier. Due to the feedback loop of the output to the inverting input the voltage difference U_D is zero and the voltage at the inverting input equals the voltage at the non-inverting input. As the non-inverting input is connected to ground the voltage at the inverting input is the same. This voltage is called a virtual ground as it corresponds to the ground voltage without being directly connected to the ground. Unlike the real ground there is no net current flow to the virtual ground.

Inverting amplifier with a real OpAmp

The calculations were done so far using an ideal OpAmp. What does this result look like for a real OpAmp with finite gain and finite input resistance?

As we have seen a real OpAmp has a finite input resistance and gain (see Fig. 6.9). Input voltage $u_1(t)$ (lower case if we consider it to be time-dependent) is transferred to output voltage $u_2(t)$. Resistors R_1 and R_2 are given as well as the open loop gain V. Five values are unknown and have to be determined to describe the complete behavior of the real OpAmp: $u_2(t)$, $u_i(t)$, $i_i(t)$, $i_1(t)$, $i_2(t)$. Therefore we have to find five equations:

– Functionality of OpApm

$$u_2(t) = -V \cdot u_i(t)$$

– Voltage drop across input resistance

$$u_i(t) = R_i \cdot i_i(t)$$

– KCL at inverting input

$$i_i(t) = i_1(t) + i_2(t)$$

– Mesh equation M1

$$u_1(t) = R_1 \cdot i_1(t) + u_i(t)$$

– Mesh equation M2

$$u_2(t) = R_2 \cdot i_2(t) + u_i(t)$$

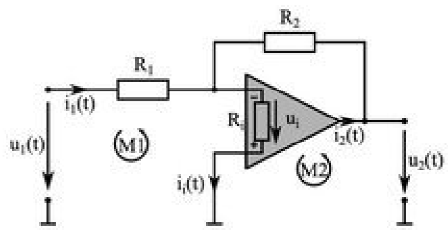


Fig. 6.9: Real OpAmp with input resistance and finite gain V.

We can solve this equation system to get the closed loop gain $u_2(t)/u_1(t)$. First we replace $i_i(t)$ from the third equation in all other equations:

$$u_2(t) = -V \cdot u_i(t)$$

$$u_i = R_i \cdot \left(i_1(t) + i_2(t)\right)$$

$$u_1(t) = R_1 \cdot i_1(t) + u_i(t)$$

$$u_2(t) = R_2 \cdot i_2(t) + u_i(t)$$

Afterwards we substitute $u_i(t)$ from the second equation:

$$u_2(t) = -V \cdot R_1 \cdot \left(i_1(t) + i_2(t)\right)$$

$$u_1(t) = R_1 \cdot i_1(t) + R_i \cdot \left(i_1(t) + i_2(t)\right)$$

$$u_2(t) = R_2 \cdot i_2(t) + R_i \cdot \left(i_1(t) + i_2(t)\right)$$

From the first equation we get the unknown $i_1(t)$ for the other two equations:

$$u_1(t) = -R_1 \frac{u_2(t)}{V \cdot R_i} - R_1 \cdot i_2(t) - \frac{u_2(t)}{V}$$

$$u_2(t) + \frac{u_2(t)}{V} = R_2 \cdot i_2(t)$$

$$i_2(t) = \frac{u_2(t)}{R_2} \cdot \left(\frac{V+1}{V}\right)$$

Using the fifth equation yields for the closed loop gain:

$$-\frac{u_1(t)}{u_2(t)} = \frac{1}{V} \cdot \left(\frac{R_1}{R_i} + 1\right) + \frac{R_1}{R_2} \cdot \left(\frac{V+1}{V}\right)$$

To check the difference to the closed loop gain of the ideal OpAmp let's consider following values:
R_1 = 10 kΩ, R_2 = 30 kΩ, V = 100000, R_i = 1 MΩ

$$-\frac{u_1(t)}{u_2(t)} = \frac{1}{100000} \cdot \left(\frac{0.01M\Omega}{1M\Omega} + 1\right) + \frac{10k\Omega}{30k\Omega} \cdot \left(\frac{100001}{100000}\right) \approx \frac{1}{3}$$

So the result for a real OpAmp with realistic values is very close to the result of the ideal OpAmp (= 1/3) and we can use the behavior of an ideal OpAmp for most purposes. Similar calculations can be done to show that a non-zero output resistance R_a changes the behavior of the real OpAmp just slightly compared to the ideal OpAmp.

Non-inverting amplifier

For the inverting amplifier the input signal is connected to the inverting input. To avoid the inversion the input signal can be connected to the non-inverting input, keeping the feedback loop to the inverting input:

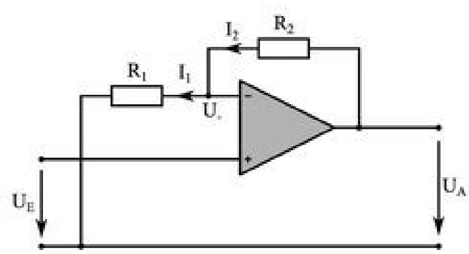


Fig. 6.10: A non-inverting amplifier.

The differential voltage at the input terminals + (non-inverting) and – (inverting) of the OpAmp is zero due to the feedback loop to the inverting input. According to voltage divider rule we get for the inverting input:

$$U_E = U_+ = U_- = \frac{R_1}{R_1 + R_2} \cdot U_A$$

Hence the closed loop gain of the circuit is

$$v = \frac{U_A}{U_E} = 1 + \frac{R_2}{R_1}$$

Automotive application

Many sensors are used all over modern vehicles for all kinds of measurements, such as in the motor compartment (e.g. for rotational speed of the cam shaft, oil pressure, motor temperature) as well as in the interior (e.g. temperature, light intensity) or on the chassis (e.g. speed, damping). The sensor output signals are transferred to the corresponding electronic control unit (ECU, e.g. motor control system). Inside the ECU a microcontroller (µC) uses these data for the algorithms of the control system.

One way of transferring the measured data to an ECU is to use a simple analog voltage. This voltage can be read by an analog-to-digital-converter (ADC) of a microcontroller. Unfortunately for some sensors the output voltage is rather small (maybe just a few mV). On the way to the ECU this small analog signal might be disturbed by the electromagnetic influence of other electronic systems. A wrong value is then read by the ADC and the control algorithms do not work correctly any more.

Fig. 6.11 shows the connection from an analog sensor via the non-inverting amplifier to the ADC input of the microcontroller. Depending on the maximum value of the output voltage of the sensor, R_1 and R_2 can be calculated to amplify the voltage to a range that fits to the input characteristics of the ADC (e.g. 5V maximum).

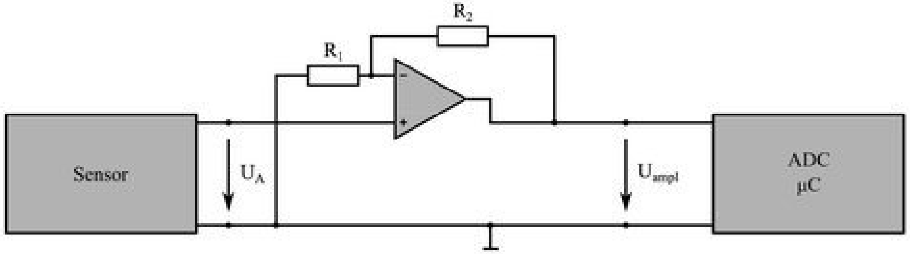


Fig. 6.11: Amplification of an analog sensor signal by a non-inverting amplifier, measurement by ADC of microcontroller (µC).

Unity gain buffer

A special case of the non-inverting amplifier is the unity gain buffer. Here the output of the OpAmp is connected directly to the inverting input, i.e. R_2 = 0 kΩ (Fig. 6.12). If the resistor R_1 is greater than zero (infinite in limit case) the closed loop gain of the unity gain buffer is

$$v = 1 + \frac{R_2}{R_1} = 1$$

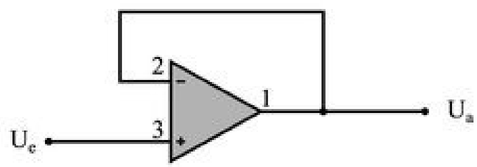


Fig. 6.12: A unity gain buffer.

The output voltage of the unity gain buffer is equal to the input voltage. The purpose of this OpAmp circuit is to make use of some basic properties of the OpAmp to convert the impedance: the input impedance is very high (infinite for the ideal OpAmp) and the output impedance is very small (zero for the ideal OpAmp). The OpAmp acts like a nearly ideal voltage source of U_e with very small internal resistance. This eliminates any feedback from the load connected to the output to the driving circuit (input voltage) as can be seen in a simple example.

An example for a circuit with a unity gain buffer

A variable load is connected to a voltage source U_q with internal resistance R_i like depicted in Fig. 6.13. Depending on the value of the load resistance the terminal voltage U (equal to the voltage across the load resistor) of the source changes according to the voltage divider rule. If the load resistance is very high compared to the internal resistance the terminal voltage will be about U_q and the current will be very small. If the load resistance is equal to the internal resistance the terminal voltage will be just half of U_q and the current will be $U_q/2R_i$. If

finally the load resistor is very small the terminal voltage will be roughly zero and the current will have its highest value of about U_q/R_i. Thus the load has a major impact on the behavior of the voltage source and the total circuit. To avoid this feedback from the load to the source a unity gain buffer can be used. The symbol for the unity gain buffer is a triangle with a 1 as shown in Fig. 6.13.
After insertion of a unity gain buffer to terminate the voltage source the load voltage will be independent of the load (see Fig. 6.13, right). The input impedance of the converter is very high and therefore the input voltage is equal to U_q. Due to its very small output impedance the OpAmp acts like a nearly ideal voltage source and thus the load voltage is constant and equal to U_q (at least as long as the load resistor is higher than the output impedance). In general a unity gain buffer is used to separate parts of circuits to avoid feedback to other parts.

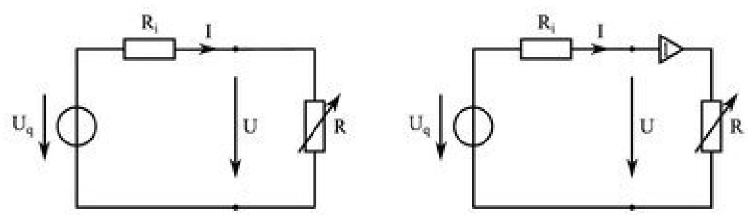


Fig. 6.13: Voltage source with internal resistance and variable load (left); same circuit like on the left side but with a unity gain buffer to separate the load from the source circuit.

7 Time domain circuit analysis

In previous chapters some concepts for the analysis of electric circuits like mesh or nodal analysis were introduced. So far only DC circuits have been considered, i.e. circuits with time-independent sources (DC sources) and after initial disturbances (e.g. switching and transients) were settled. Even the few examples were sources were time-dependent transient behavior was not taken into account. If time-dependent parameters like current and voltage are considered, lower case symbols are used to describe these parameters, e.g. u(t), i(t).
Time domain circuit analysis will be split into two parts:

1. Transient effects (switching events)
2. AC circuits

We will start with the introduction of two new elements in electrical circuits: capacitors and inductors.

7.1 Capacitor

A capacitor is an electric element that is able to store electrical energy. In a simplified image an ideal capacitor is built of two plates (electrodes). The electrodes are separated by a non-conducting space (dielectric) and each electrode is connected to one terminal of the capacitor. A current through a capacitor means that positive charges are accumulated inside the capacitor on one electrode and negative charges on the other electrode.

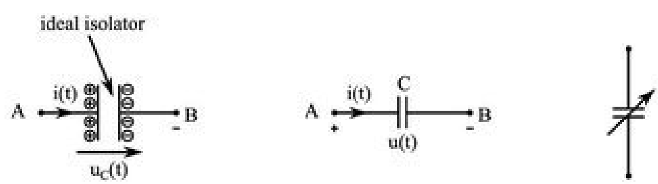


Fig. 7.1: A simple image of a capacitor: current i(t) causes positive charges to

Fig. 7.1: A simple image of a capacitor; current i(t) causes positive charges to accumulate on one electrode and negative on the other; circuit symbol of a capacitor (center) and adjustable capacitor (right).

A separation of charges means there is an electric field generated inside the capacitor storing electrical energy. The difference of potentials due to the electric field can be measured as voltage u(t) at the terminals. The ratio of accumulated charges q(t) to created voltage u(t) is called the capacitance of a capacitor:

$$C = \frac{q(t)}{u(t)}$$

The unit of capacity C is Farad:

$$[C] = 1\frac{As}{V} = 1F$$

The capacity of 1 Farad of a capacitor means that a stored charge of 1 Coulomb creates a voltage of 1 Volt at the terminals.
The capacitance C is a constant for a given capacitor depending on the geometric configuration and the dielectric of the capacitor. An ideal capacitor just has a capacitance C, with no resistance R.
Taking the definition of the voltage and Maxwell’s second equation into account for electrostatic cases the capacitance can be expressed in terms of fields:

$$C = \frac{Q}{U} = \frac{\oiint_A \vec{D} \cdot d\vec{A}}{\int_A^B \vec{E} \cdot d\vec{s}}$$

The calculation of the capacitance for arbitrary geometry is in general complex. But for simple geometries and the neglect of edge effects it can be calculated rather simply, e.g. for a plate capacitor. As depicted in Fig. 7.2 the capacitor consists of two plates with surface A, distance d and a dielectric ε. Charges +Q and -Q (same amount, opposite polarity) are accumulated on both plates respectively. The displacement field is homogeneous between the plates and zero outside the plates (good approximation if the plates are much bigger than the distance between the two plates).

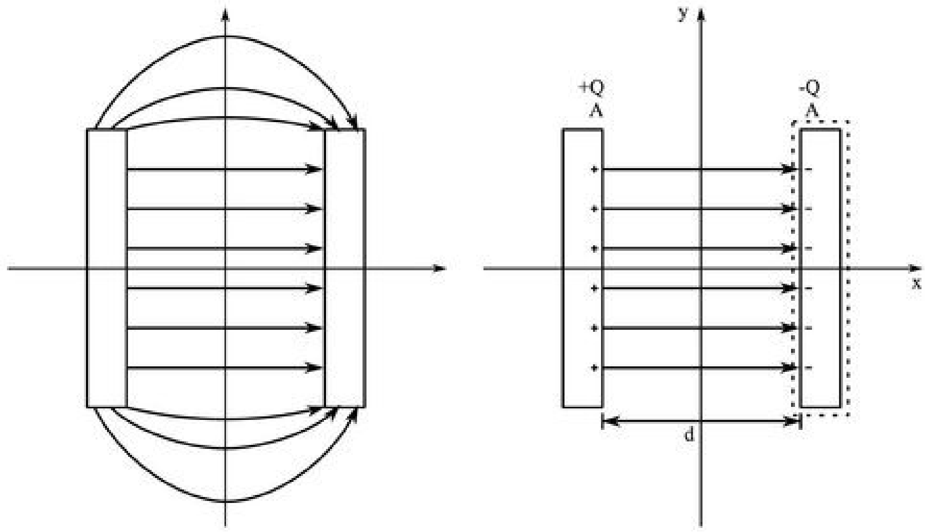


Fig. 7.2: Plate capacitor with charges +Q and -Q on both plates respectively; left: stray field outside the capacitor; right: simplification: displacement field just inside the capacitor.

The integration to calculate the charge is achieved using the closed surface shown on the right side of Fig. 7.2. Outside the capacitor the displacement field is zero. Between the plates it is in x-direction and parallel to the normal of the surface. Thus the charge yields:

$$Q = \oiint_A \vec{D} \cdot d\vec{A} = D \cdot A$$

The voltage between the plates is calculated using the electric field and the integration is done from the left plate (at A = –d/2) to the right plate (at B = d/2). As electric field and integration path are parallel the voltage is given by:

$$U = \int_A^B \vec{E} \cdot d\vec{s} = \int_{-d/2}^{d/2} E \cdot ds = \int_{-d/2}^{d/2} \frac{D}{\varepsilon} \cdot ds = \frac{Q \cdot d}{\varepsilon \cdot A}$$

Finally the capacitance of a plate capacitor is:

$$C = \frac{Q}{U} = \frac{\varepsilon \cdot A}{d}$$

It is directly proportional to the area of the plates and the dielectric between the plates and inversely proportional to the distance between the plates.
Recall the definition of electric current:

$$i(t) = \frac{dq(t)}{dt} = C \cdot \frac{du(t)}{dt}$$

Thus the current entering a capacitor is equal to the rate of buildup of charge on the plate attached to the terminal and proportional to the buildup of the voltage between the plates. Integration of the current equation above yields the integral form:

$$u(t) = \frac{1}{C} \cdot \int_{-\infty}^{t} i(\tau) d\tau = \frac{1}{C} \cdot \int_{0}^{t} i(\tau) d\tau + u(0)$$

Here u(0) is the initial capacitor voltage at t = 0 s.
To calculate the energy stored in the electric field of a capacitor we start with the power p(t) delivered to the capacitor:

$$p(t) = u(t) \cdot i(t) = C \cdot u(t) \cdot \frac{du(t)}{dt}$$

The energy e(t) stored in the capacitor is obtained by integrating:

$$e(t) = \int_{-\infty}^{t} p(\tau) d\tau = C \cdot \int_{-\infty}^{t} u(\tau) \cdot \frac{du(\tau)}{d\tau} d\tau = \frac{1}{2} C \cdot u^2(t) - \frac{1}{2} C \cdot u^2(-\infty)$$

Assuming the capacitor voltage to be zero at t = -∞ s the energy stored in a capacitor at time t represents the energy of the electric field between the plates due to the separation of charges and just depends on the voltage at that time

$$e(t) = \frac{1}{2} C \cdot u^2$$

Some properties of capacitors based on the equations above:

- In the special case that the voltage across the capacitor is constant there is no current flow through the capacitor any more. In the case of DC (after any switching effects, s. below) therefore the capacitor behaves like an open load (in fact it is an open load due to the dielectric between the plates).
- If a capacitor is charged and disconnected afterwards, the current will be zero and the voltage across the capacitor will stay constant (energy storage element)
- Energy in general cannot be changed instantaneously (this would need infinite high power). Consequently the voltage u(t) across the capacitor cannot change instantaneously. By contrast the current i(t) can change instantaneously.

Series and parallel connection of capacitors

Like resistors capacitors can of course be connected in series and parallel as depicted in Fig. 7.3. For the series connection the voltage drop u(t) across the terminals A-C is split into the voltages across the capacitors, A-B and B-C:

$$u(t) = u_{AB}(t) + u_{BC}(t)$$

$$\frac{du(t)}{dt} = \frac{du_{AB}(t)}{dt} + \frac{du_{BC}(t)}{dt}$$

As the same current i(t) is flowing through both capacitors we get:

$$\frac{i(t)}{C_1} + \frac{i(t)}{C_2} = \frac{i(t)}{C_{eq}}$$

Here C_{eq} is the equivalent capacitance if we replace the two capacitors by a single one. In a more general manner we can find the equivalent capacitance C_{eq} for a series connection of n capacitors with capacitance C_i by:

$$\frac{1}{C_{eq}} = \sum_{i=1}^{n} \frac{1}{C_i}$$

Regarding the parallel connection of capacitors (refer to Fig. 7.3) the voltage drop across both capacitors is the same and the current i(t) is split into two parts through both capacitors respectively, $i_1(t)$ and $i_2(t)$. According to KCL at node B:

$$i(t) = i_1(t) + i_2(t) = C_1 \cdot \frac{du(t)}{dt} + C_2 \cdot \frac{du(t)}{dt} = C_{eq} \cdot \frac{du(t)}{dt}$$

Again C_{eq} is the equivalent capacitance if we replace the two capacitors by a single one. For n capacitors in parallel we can write:

$$C_{eq} = \sum_{i=1}^{n} C_i$$

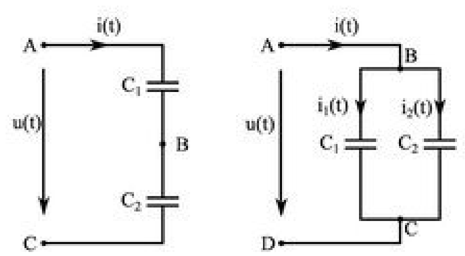


Fig. 7.3: Series (left) and parallel (right) connection of capacitors.

Capacitors and OpAmps

Capacitors can be used also in combination with OpAmps. In this example the feedback loop of an ideal OpAmp is built up of a resistor R, the inverting input is connected to the input voltage $u_1(t)$ via a capacitor C, see Fig. 7.4.

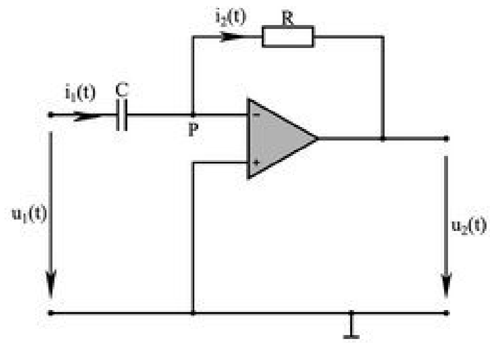


Fig. 7.4: OpAmp circuit with a capacitor in the input line. The circuit acts as a differentiator.

As the OpAmp is ideal the voltage at the inverting input is 0 V (equal to non-inverting input) and therefore KCL yields:

$$i_1(t) = i(t) = i$$

Using

$$i_1(t) = i(t) = C \cdot \frac{du_1(t)}{dt}$$

$$u_2(t) = -R \cdot i_2(t) = -R \cdot i(t)$$

we get:

$$u_2(t) = -R \cdot C \cdot \frac{du_1(t)}{dt}$$

Hence the output voltage is proportional to the negative derivative of the input voltage and the circuit realizes a differentiator. The term τ = R·C is the time constant of the differentiator.

Real capacitors

Besides the simple capacitor built out of two plane plates (s. above) there are many other geometric forms for capacitors like cylinder-type shapes (Fig. 7.5). Without derivation the capacitance for these cylinder-types is given by:

$$C_{cylinder} = \frac{2\pi \cdot \varepsilon_0 \cdot \varepsilon_r \cdot l}{\ln\left(\frac{R}{r}\right)}$$

Here the parameters are:

- l: length of cylinder
- R: radius of outer electrode
- r: radius of inner electrode

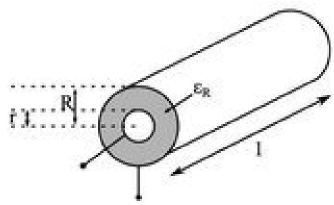


Fig. 7.5: A cylinder-type capacitor.

Capacitors are often made of tightly rolled sheets of metal film with a dielectric material (e.g. paper or nylon) in between in order to increase the capacitance for a given size. Based on geometry, dielectric and fabrication process values for the capacitance can range from a few pico Farads up to the Farad region. Refer to Tab. 7.1 for a list of different types of dielectric material. The working voltage is the maximum voltage that can safely be applied to the terminals of a capacitor. This value is in general given by the manufacturer. Exceeding this limit may result in the breakdown of the dielectric (due to the small distance between the electrodes the electric field between the electrodes reaches very high values) and the formation of an electric path between the capacitor’s plates. Values for the working voltage can range from a few volts to some thousands volts.

Tab. 7.1: Characteristics of capacitors with different dielectric.

Material	**Capacitance range**	**Maximum voltage range [V]**
Mica	1 pF-0.1 µF	50–600
Ceramic	10 pF-1 µF	50–1600

Ceramic	10 pF-1 µF	50–1600
Paper	10 pF-50 µF	50–400
Electrolytic	0.1 µF–0.2 F	3–600

Due to connections, terminals and internal configuration real capacitors have additional resistive elements in addition to the capacitive behavior. The resistive effect of these parts can be modeled by a resistor in series (ESR, equivalent series resistance) with an ideal capacitor. The ESR depends on the capacitor's type and assembly and is usually in the range of mΩ to Ω and strongly frequency dependent.
As the dielectric between the electrodes are not perfect isolators there will be a (very small) amount of current through the capacitor called leakage current. This effect can be modeled by an ideal capacitor in parallel with a parasitic resistor (Fig. 7.6). These resistors create losses and the different technical types of capacitors can be distinguished by the amount of losses.

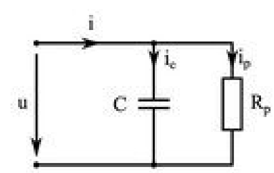


Fig. 7.6: Model of capacitor with parasitic resistor in parallel to the capacitance due to imperfect dielectric.

There are many different types of capacitors, all with specific pros and cons. Important types are:

- Ceramic capacitor: the dielectric is a ceramic material (e.g. TiO_2, $BaTiO_3$), capacitance values are in the range of 0.5 pF-100 µF and more. Applications are highfrequency applications as well as storage elements;
- Film capacitor: a dielectric film (e.g. polyester, metalized paper, Teflon) is sandwiched between the metal layers, the complete sandwich is wound into a

tight roll. It’s the most common capacitor type with many different forms, capacitance values ranging from few pF up to 100 µF. Often used in high power applications;

- Electrolytic capacitor: This type uses an electrolyte (ionic conducting liquid) as one of the electrodes (cathode). The dielectric is formed by a very thin oxide film on the anode (anode material e.g. Al, Ta or Nb). Due to the very thin dielectric the distance between the electrodes is very small and due to a coarse surface of the anode the surface is rather large. These two geometric parameters result in a rather high capacitance of 1 µF up to 47 mF. Used in all applications needing high capacitance values, e.g. DC power supplies. As this type of capacitor is in most cases polarized the terminals have to be connected with the correct polarity (positive to +terminal, negative to –terminal, otherwise the capacitor will be destroyed (explode);
- Double layer capacitor (supercap): A special kind of electrolytic capacitor where the distance between the electrodes is in the nm range. Therefore the capacitance is very high, up to several hundred Farad or even above. This type of capacitor is used as storage element e.g. in electric vehicles for short term storage and charging/discharging.

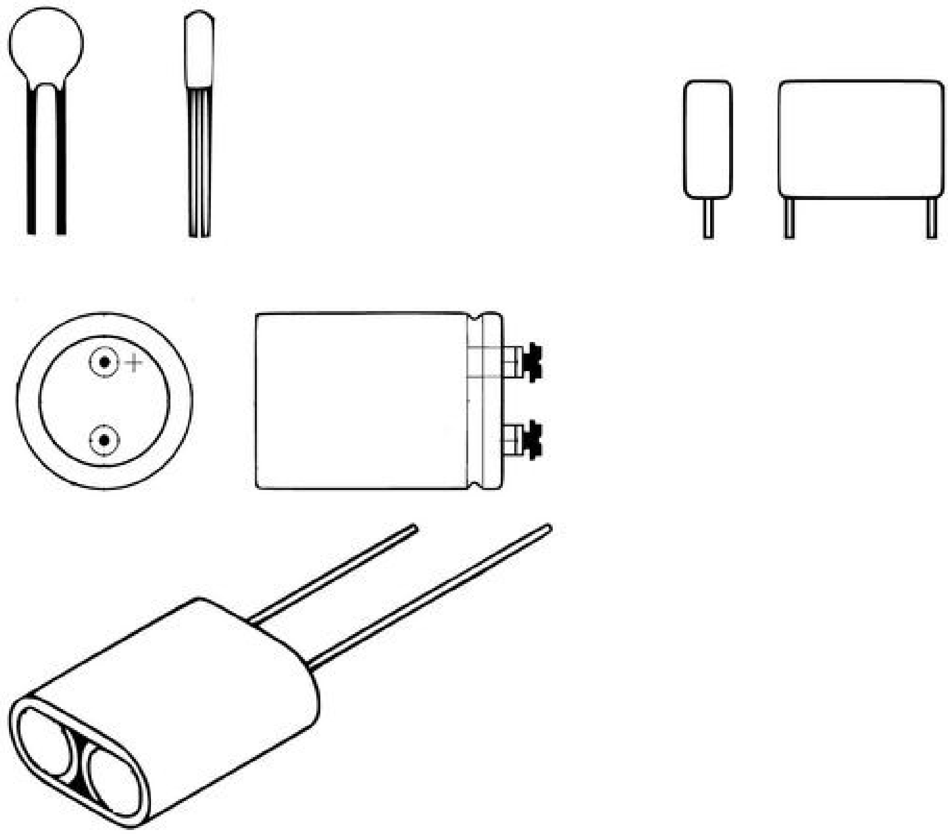


Fig. 7.7: Typical packages for capacitors: ceramic (top left); film (top right); electrolytic, positive terminal marked with + (center); supercap, positive terminal marked by longer pin (bottom).

Automotive application

Numerous capacitors can be found in almost every electronic system and ECU of modern vehicles. For example they can be used for filter applications. Or they are used as blocking capacitors for voltage stabilization as they can provide or absorb high currents in the short term. This application makes use of the capability of capacitors to store electrical energy. The combination of energy storage and high current capability makes supercaps very interesting in particular for HEV/EV applications. During braking the electrical motor of HEV/EV is used as a generator to convert mechanical energy into electrical energy. This recuperation of braking energy results in high currents. As the battery is not able to cope with the high currents supercaps can be used as high power storage element as they can absorb high currents.

7.2 Inductors

Like the capacitor the inductor is an energy-storage circuit element. However, it is not based on the electric field, but rather the magnetic field effect: a current flow in a conductor produces a magnetic field around this conductor. Winding a conductor into a coil (N windings) increases the magnetic field. This magnetic field is described by the magnetic flux N · Φ(t) that is directly proportional to the current i(t):

$$N \cdot \Phi(t) = L \cdot i(t)$$

The constant L is the inductance of the element.

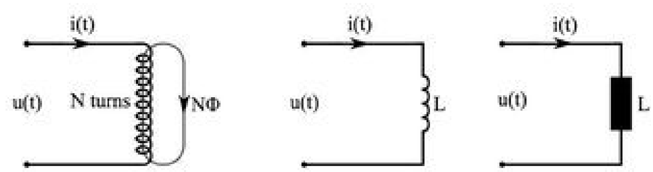


Fig. 7.8: A single inductive coil with N windings (left); American circuit symbol of an inductor (mid) and European symbol (right).

According to Faraday’s law of induction the voltage across the inductor is proportional to the change of the total magnetic flux N · Φ(t) and hence to the change of current through the inductor:

$$u(t) = N \cdot \frac{d\Phi(t)}{dt} = L \cdot \frac{di(t)}{dt}$$

The unit for inductance is Vs/A = H (= Henry):
An inductor has the inductance of 1 Henry if the induced voltage at the terminals is 1 A as a reaction to a current change rate of 1 A/s. For the special case of DC current the voltage across an inductor is zero and an ideal inductor acts like a short circuit.
Integration of the voltage equation above yields the integral form:

$$i(t) = \frac{1}{L} \cdot \int_{-\infty}^{t} u(\tau) d\tau = \frac{1}{L} \cdot \int_{0}^{t} u(\tau) d\tau + i(0)$$

Here i(0) is the initial inductor current at t = 0 s.
To calculate the energy stored in the magnetic field of an inductor we start with the power p(t) delivered to the inductor:

$$p(t) = u(t) \cdot i(t) = L \cdot i(t) \cdot \frac{di(t)}{dt}$$

The energy e(t) stored in the inductor is obtained by integrating:

$$e(t) = \int_{-\infty}^{t} p(\tau) d\tau = L \cdot \int_{-\infty}^{t} i(\tau) \cdot \frac{di(\tau)}{d\tau} d\tau = \frac{1}{2} L \cdot i^2(t) - \frac{1}{2} L \cdot i^2(-\infty)$$

Assuming the inductor current to be zero at t = -∞ s, the stored energy in the inductor at time t only depends on the current at that time and the inductance of the element and is given by:

$$e(t) = \frac{1}{2} L \cdot i^2(t)$$

Like for the capacitor (and always in physics) the energy cannot change instantaneously and according to the relation between energy and current also the current through an inductor cannot change instantaneously in a step function (but the voltage can). As with the capacitor the step of a current through an inductor would need an infinitely high voltage at the terminals which cannot be generated.
In other words: A high voltage is induced if a current is switched off very fast. Be careful with switching off a current through an inductor as this high voltage may damage other components of the circuit.

Series and parallel connection of inductors

Like resistors and capacitors inductors can of course be connected in series and parallel as depicted in Fig. 7.9.
For the series connection the voltage drop u across the terminals A-C is split into the voltages across the inductors, A-B and B-C and same current i(t) flows through the inductors:

$$u = u_{AB} + u_{BC}$$

$$L_{eq} \cdot \frac{di(t)}{dt} = L_1 \cdot \frac{di(t)}{dt} + L_2 \cdot \frac{di(t)}{dt}$$

$$\Rightarrow L_{eq} = L_1 + L_2$$

Here L_{eq} is the equivalent inductor if we replace the two inductors by a single one. In a more general manner we can find the equivalent inductance L_{eq} for a series connection of n inductors with inductance L_i by:

$$L_{eq} = \sum_{i=1}^{n} L_i$$

Regarding the parallel connection of inductors (refer to Fig. 7.9) the voltage drop u(t) across both capacitors is the same and the current i(t) is split into two parts through both inductors respectively, $i_1(t)$ and $i_2(t)$. According to KCL at node B:

$$i(t) = i_1(t) + i_2(t)$$

$$\frac{di(t)}{dt} = \frac{di_1(t)}{dt} + \frac{di_2(t)}{dt}$$

$$\Rightarrow \frac{u(t)}{L_{eq}} = \frac{u(t)}{L_1} + \frac{u(t)}{L_2}$$

$$\Rightarrow \frac{1}{L_{eq}} = \frac{1}{L_1} + \frac{1}{L_2}$$

Again L_{eq} is the equivalent inductance if we replace the two inductors by a single one. For n inductors in parallel we can write

$$\frac{1}{L_{eq}} = \sum_{i=1}^{n} \frac{1}{L_i}$$

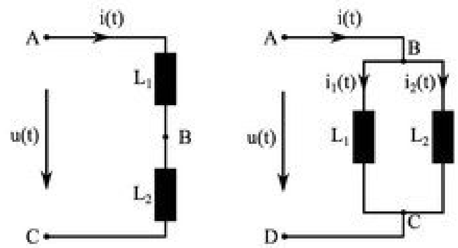


Fig. 7.9: Series (left) and parallel (right) connection of inductors.

Real inductors

Ideal inductors have just the inductance and no resistance or capacitance. Real inductors will have some associated resistance and capacitance: the wiring of the inductor has some (small but non-zero) resistance, and sizable capacitances may exist between adjacent turns. A possible model for a real inductor could be a combination of ideal elements: a combination of resistance and inductance in series, with a capacitance in parallel. The parasitic resistance can range from a few Ohms up to several hundred Ohms.
Real inductor (also called coil or choke) values range from about 0.1 µH to several hundred mH or even several H. Due to the construction of the coils and the storage of energy in the magnetic field, inductors, in particular for big inductance values, can hardly be miniaturized, they are rather bulky and expensive. The standardization of inductors is not done to the same degree as for resistors and capacitors. Some examples of inductor packages are depicted in Fig. 7.10.

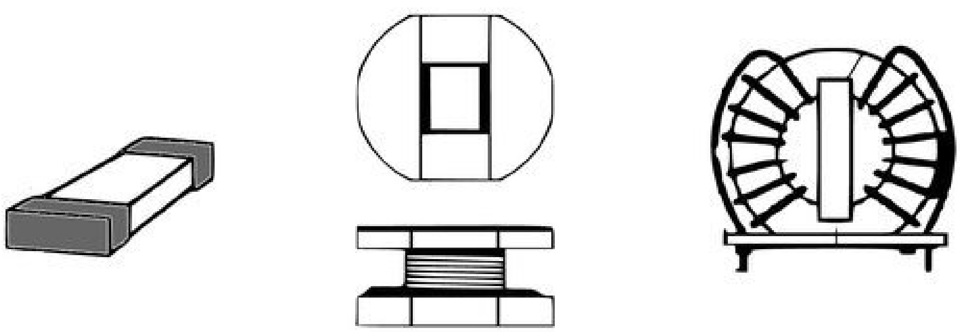


Fig. 7.10: Examples for inductor packages: shielded SMD (surface mount device, left); unshielded SMD (center); unshielded THD (through hole device, right).

Automotive application

Inductors are often used as chokes to filter out higher frequency AC currents due to the frequency dependence of their impedance. Or they are used as short term energy storage element, e.g. in DC/DC converters to convert one DC voltage to another DC voltage. Applications like an electrical relays make use of the magnetic properties of an inductor (see Fig. 7.11).
The circuit is split into two parts, a control circuit and a load circuit. Target is to switch the load without a direct electrical contact to the control circuit. By applying a current to the inductor of the relays a magnetic field is generated. This field is used to close a magnetic switch in the load circuit which. Due to the closed magnetic switch the load circuit is electrically closed and the load is switched on. As soon as the control current is switched off, the magnetic field fades away and the relays opens the load circuit again. The load is switched off. Notice that there is no direct electrical contact between control and load circuit. Both circuits are galvanically isolated.

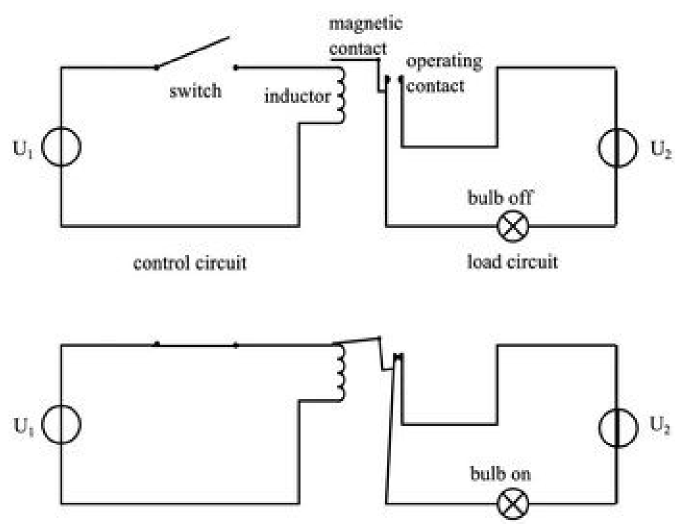


Fig. 7.11: Circuit of an electrical relais to switch a load circuit.

7.3 Transient effects and switching

By now we have just regarded DC voltages and currents in our circuit analysis. The circuits themselves were pure resistive.

When starting with time-dependent analysis of circuits with inductors, capacitors and resistors we have to analyze the behavior of inductors and capacitors as a function of time. However, laws as introduced in previous chapters (like KCL, KCL) are still valid.

7.3.1 First order circuit – the natural response

We will start with the analysis of switching events and transient effects in first order circuits. First order circuits contain a single capacitor or inductor and a network of DC sources, resistors and switches. Consider the circuit as shown in Fig. 7.12. The switch is closed for times t < 0. At t = 0 the switch is opened. What about the voltage, the current and the energy stored in the capacitor for t < 0 s, t = 0 s and t > 0 s?

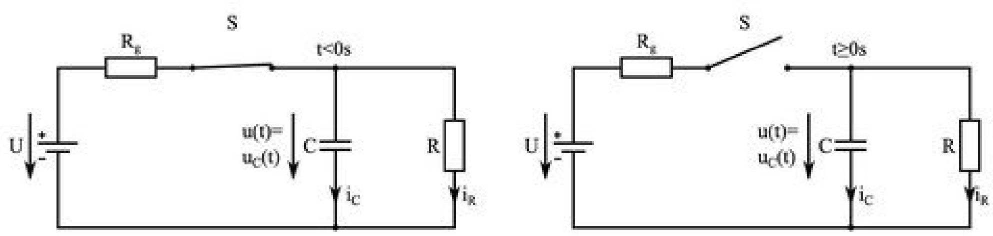


Fig. 7.12: A simple circuit with a switch.

t < 0 s

For t < 0 s the circuit is a DC circuit and the capacitor behaves like an open circuit. The current is flowing through the resistor R. According to the voltage divider rule for the two resistors in series the capacitor voltage is

$$u(t) = u_C(t) = \frac{R \cdot U}{R + R_g}$$

The current and energy in the capacitor equals:

$$i(t) = \frac{u(t)}{R} = \frac{U}{R + R_g}$$

$$e(t) = \frac{1}{2} C \cdot u^2(t) = \frac{1}{2} C \cdot U^2$$

t = 0 s
For t = 0 s the switch is opened instantaneously and the voltage source (as well as the resistor R_g) are disconnected from the resistor and capacitor connected in parallel. The circuit we are looking at (the mesh containing the parallel capacitor and resistor) contains no sources anymore and the result will be called the natural response. As the voltage across a capacitor and its energy cannot change instantaneously they stay at the value of $t < 0$ s:

$$u(0) = u_C(0) = \frac{R \cdot U}{R + R_g}$$

$$e(0) = \frac{1}{2} C \cdot u^2(0)$$

These values will be the initial conditions for the behavior and solution of times $t > 0$ s.

t > 0 s
After the switch opened at t = 0 s it stays open for $t > 0$ s. As the charged electrodes of the capacitor are connected via resistor R in a mesh, the capacitor will be discharged via the resistor. With the currents given as indicated in Fig. 7.12 we can write KCL as:

$$i_C(t) = -i_R(t)$$

Using Ohm’s law for the resistor yields:

$$i_R(t) = \frac{u(t)}{R} = \frac{u_C(t)}{R}$$

The current-voltage equation for the capacitor is:

$$i_C(t) = C \cdot \frac{du_C(t)}{dt}$$

Finally we get:

$$C \cdot \frac{du_C(t)}{dt} + \frac{u_C(t)}{R} = 0$$

Thus we have a homogenous, first-order, linear differential equation (ordinary differential equation, ODE) for the voltage across the capacitor (and the resistor) that is to be solved. Solution:

Rewriting the ODE gives:

$$\frac{du_C(t)}{dt} + \frac{1}{RC} \cdot u_C(t) = 0$$

Separation of variables and substituting t by τ:

$$\frac{du_C(\tau)}{u_C(\tau)} = -\frac{1}{RC} d\tau$$

Integration from τ = 0 s to τ = t:

$$\int_{\tau=0}^{t} \frac{1}{u_C(\tau)} du_C(\tau) = \int_{u_C(0)}^{u_C(t)} \frac{1}{u_C} du_C = -\frac{1}{RC} \int_{\tau=0}^{t} d\tau$$

$$\Rightarrow \ln u_C(t) - \ln u_C(0) = -\frac{1}{RC} \cdot t$$

$$\Rightarrow \ln \frac{u_C(t)}{u_C(0)} = -\frac{t}{RC}$$

$$\Rightarrow u_C(t) = u_C(0) \cdot e^{-\frac{t}{RC}}$$

So we have the solution for the homogenous ODE with a (so far unknown) constant $u_C(0)$. This constant is determined by the constraints of the initial condition, i.e. the voltage across the capacitor at time t = 0 s (remember the voltage across a capacitor cannot change instantaneously at t = 0 s). This initial value was already determined above (see t = 0 s) and therefore the final solution for this homogenous ODE is:

$$\Rightarrow u_C(t) = u(0)\cdot e^{-\frac{t}{RC}} = \frac{R\cdot U}{R+R_g}\cdot e^{-\frac{t}{RC}}$$

The function of $u_C(t)$ is depicted in Fig. 7.13: The voltage is at $u(0)=u_C(0)$ at t = 0 s and decreases exponentially with time, so it will never be equal to zero for finite times. The constant R·C in the denominator of the exponential function is called the time constant (unit of R·C is Ω·F = Ω·s/Ω = s). If depicted in terms of R·C it can be seen that $u_C(t)$ decreases to defined values (e.g. 36.8% of the initial value for t = R·C). So the product R·C gives a direct measure for the speed of the voltage decrease. The higher R·C is the longer it takes for the voltage to decrease.

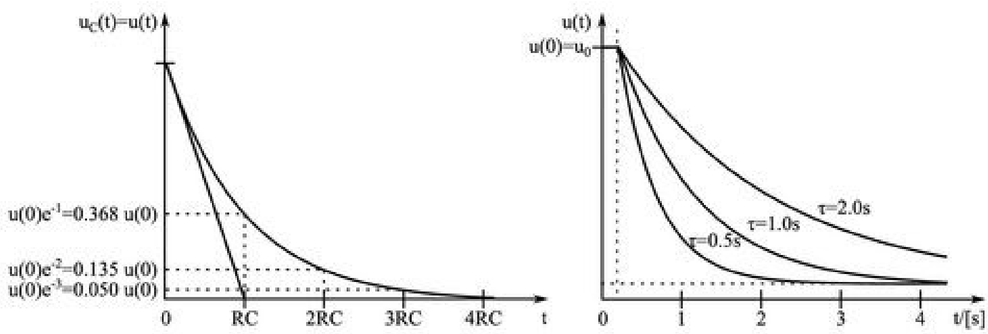


Fig. 7.13: Voltage across the capacitor (and the resistor), time constant τ=R·C determines how quickly the voltages decreases and settles to its final value..

After determination of the voltage the currents through the resistor and the capacitor are:

$$i_R(t) = \frac{u_C(t)}{R} = \frac{u(0)}{R}\cdot e^{-\frac{t}{RC}} = \frac{U}{R+R_g}\cdot e^{-\frac{t}{RC}}$$

$$i_C(t) = -\frac{u(0)}{R}\cdot e^{-\frac{t}{RC}} = -\frac{U}{R+R_g}\cdot e^{-\frac{t}{RC}}$$

The energy in the capacitor will also decrease:

$$e_C(t) = \frac{1}{2}C\cdot u_C^2(t) = \frac{1}{2}C\cdot\left(u(0)\cdot e^{-\frac{t}{RC}}\right)^2 = \frac{1}{2}C\cdot u^2(0)\cdot e^{-\frac{2t}{RC}}$$

After a very long time the capacitor will be completely discharged (never completely but nearly…) and hence the energy will be zero. As energy cannot vanish and as the resistor is the only element in the circuit besides the capacitor it is obvious that the energy is dissipated (converted to heat) in the resistor. The power absorbed by the resistor is

$$p_R(t) = \frac{u_R^2(t)}{R} = \frac{u^2(0)}{R} \cdot e^{-\frac{2t}{RC}}$$

With this power dissipation at time t the total energy absorbed by the resistor from t = 0 s until t yields:

$$e_R(t) = \int_0^t p_R(\tau) d\tau = \frac{u^2(0)}{R} \cdot \int_0^t e^{-\frac{2\tau}{RC}} d\tau = \frac{u^2(0)}{R} \cdot \frac{-R \cdot C}{2} \cdot \left(e^{-\frac{2t}{RC}} - 1 \right)$$

$$\Rightarrow e_R(t) = \frac{1}{2} C \cdot u^2(0) \cdot \left(e^{-\frac{2t}{RC}} - 1 \right)$$

Thus the total energy from the beginning is conserved and transfers from the capacitor to the resistor where it is absorbed and dissipated:

$$e_{total} = e_C(0) = e_R(t) + e_C(t) = \frac{1}{2} C \cdot u^2(0)$$

Summarizing

At t = 0 s the capacitor was charged to $u_C(0)$ ($q(0) = C \cdot u_C(0)$) and it discharges exponentially after the switch was opened. Until t = 0 s the current through the resistor is driven by the voltage supply. After disconnection of the voltage supply the current is maintained by the capacitor at t = 0 s and drops exponentially. The rate at which the voltage decreases is measured by the time constant $\tau = R \cdot C$. In 5 time constants the voltage is within 1 % of its final value (steady state value). This behavior of the R · C circuit with no external source of excitation is called the natural response. The capacitor takes the role of a voltage supply with decreasing voltage.

7.3.2 First order circuit – complete response

After the study of the natural response (no source after the switching event) circuits with a source as excitation after the switching event such as depicted in Fig. 7.14 are considered. This time the resistor and the capacitor are in series to the switch and the time-independent voltage source. The switch is open for $t < 0$ s, closes at $t = 0$ s and stays closed for $t > 0$ s.

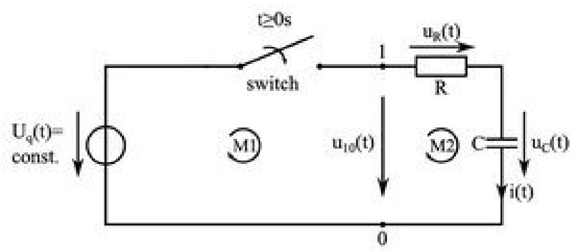


Fig. 7.14: A first order circuit with a RC combination and a voltage source as excitation.

Before we analyze the circuit in detail let’s try to figure out qualitatively what will happen, based on our experience from the natural response. With the switch open no current flows and the capacitor is uncharged. Closing the switch will make a current flow, hence charging the capacitor. By charging the capacitor the voltage across it will increase until it reaches the final value of U_q. At that time the current flow will stop and the circuit behaves like an open circuit.
The detailed analysis looks like:

t < 0 s
As the switch is open the voltages $u_{10}(t)$, $u_R(t)$ and $u_C(t)$ are zero and no current flows through the resistor and the capacitor.

t = 0 s
The switch is closed instantaneously. According to KVL for mesh M1 voltage $u_{10}(0)$ equals to $U_q(0)$. For mesh M2 the voltage across the capacitor cannot change instantaneously and thus the voltage drop across the resistor equals $u_{10}(0)$. Writing KVL for mesh M2 yields

$$u_{10}(0) = U_q(0) = u_R(0) + u_C(0) = u_R(0)$$

And the current i(0) is (remember that the current through a capacitor can change instantaneously unlike the voltage):

$$i(0) = \frac{u_R(0)}{R} = \frac{U_q(0)}{R}$$

t > 0 s

The switch stays closed and according to KVL for mesh M1 the voltage will be constant:

$$u_{10}(t) = U_q(t)$$

The current i(t) (through resistor and capacitor) will charge the capacitor

$$i(t) = C \cdot \frac{du_C(t)}{dt}$$

Writing KVL for mesh M2 yields:

$$U_q(t) = u_R(t) + u_C(t) = R \cdot i(t) + u_C(t) = R \cdot C \cdot \frac{du_C(t)}{dt} + u_C(t)$$

Thus in the case of a source in the circuit after the switching event we get a first order inhomogeneous ODE:

$$\frac{du_C(t)}{dt} + \frac{1}{RC} \cdot u_C(t) = \frac{1}{RC} \cdot U_q(t)$$

1/RC is a constant coefficient and the right side of the equation is a function f, which is in general time-dependent, f(t).

Excursus: solution of first order inhomogeneous ODE

> The general form of a first order inhomogeneous ODE with constant coefficients looks like:

$$\frac{dx(t)}{dt} + a \cdot x(t) = f(t)$$

To find the solution of this ODE we multiply this equation with e^{at}:

$$e^{at} \cdot f(t) = e^{at} \cdot \frac{dx(t)}{dt} + e^{at} \cdot a \cdot x(t) = \frac{d}{dt}\left(e^{at} \cdot x(t)\right)$$

Integration of both sides yields:

$$\int_{\tau=0}^{t} \frac{d}{d\tau}\left(e^{a\tau} \cdot x(\tau)\right) d\tau = \int_{\tau=0}^{t} e^{at} \cdot f(\tau) d\tau$$

$$\Rightarrow e^{at} \cdot x(t) - e^{a0} \cdot x(0) = \int_{\tau=0}^{t} e^{a\tau} \cdot f(\tau) d\tau$$

Multiplying with e^{-at}:

$$\Rightarrow x(t) = e^{-at} \cdot \int_{\tau=0}^{t} e^{a\tau} \cdot f(\tau) d\tau + x(0) \cdot e^{-at}$$

This formula for the solution of a first order ODE with x(0) being determined by the initial conditions is called the complete response. It consists of two parts that will be discussed in terms of our problem of transients.

Let's have a closer look at the two terms of the complete response for the switching of the RC circuit with constant voltage source U_q.

The ODE is:

$$\frac{du_C(t)}{dt} + \frac{1}{RC} \cdot u_C(t) = \frac{1}{RC} \cdot U_q$$

With $x(t) = u_C(t)$, $a = 1/(R \cdot C)$ and $f(t) = U_q/(R \cdot C)$ the complete response is:

$$\Rightarrow u_C(t) = e^{-\frac{t}{RC}} \cdot \int_{\tau=0}^{t} e^{\frac{\tau}{RC}} \cdot \frac{U_q}{RC} d\tau + u_{Ch}(0) \cdot e^{-\frac{t}{RC}}$$

Thus the complete response of this ODE is split into two parts, a solution for the homogeneous and a solution for the particular ODE:

$$u_C(t) = u_{Cp}(t) + u_{Ch}(t)$$

The solution to the homogeneous ODE (f(t) is set to zero)

$$\frac{du_{Ch}(t)}{dt} + \frac{1}{RC} \cdot u_{Ch}(t) = 0$$

was already determined previously and corresponds to the second term of the solution:

$$u_{Ch}(t) = u_{Ch}(0) \cdot e^{-\frac{t}{RC}}$$

It is the natural or transient response with an exponential behavior of the capacitor's voltage. Constant $u_{Ch}(0)$ is determined by the initial conditions of the complete system.
The first term of the solution is determined by the function f(t) which describes the excitation of the circuit (by the voltage source U_q). It is the solution of the particular ODE and describes the steady-state behavior of the circuit (t → ∞ s) forced by the excitation (forced response). For a constant forcing function U_q the steady-state response yields for t → ∞ s:

$$\Rightarrow \lim_{t\to\infty} u_{Cp}(t) = \lim_{t\to\infty} e^{-\frac{t}{RC}} \cdot \int_{\tau=0}^{t} e^{\frac{\tau}{RC}} \cdot \frac{U_q}{R \cdot C} d\tau = \lim_{t\to\infty} \frac{U_q}{R \cdot C} \cdot R \cdot C \cdot e^{-\frac{t}{RC}} \cdot \left(e^{\frac{t}{RC}} - 1 \right) = U_q$$

Looking at the circuit it is obvious that the steady-state response will be just the voltage of the voltage source (excitation voltage) as in the steady-state the circuit is a DC circuit without any current flowing and the capacitor's voltage corresponds to the voltage of the source. The complete response is thus:

$$u_C(t) = u_{Ch}(t) + u_{Cp}(t) = U_q + u_{Ch}(0) \cdot e^{-\frac{t}{RC}}$$

$u_{Ch}(0)$ is to be determined by the initial conditions. In our example the voltage across the capacitor $u_C(0)$ is zero at t = 0 s. Therefore the equation above yields:

$$u_C(0) = 0V = u_{Ch}(0) + u_{Cp}(0) = U_q + u_{Ch}(0)$$

$$\Rightarrow u_{Ch}(0) = -U_q$$

$$u_C(t) = U_q - U_q \cdot e^{-\frac{t}{RC}} = U_q \cdot \left(1 - e^{-\frac{t}{RC}}\right)$$

The complete response of the inhomogeneous ODE is depicted in with the time scaled by the time constant τ = R·C is depicted in Fig. 7.15.

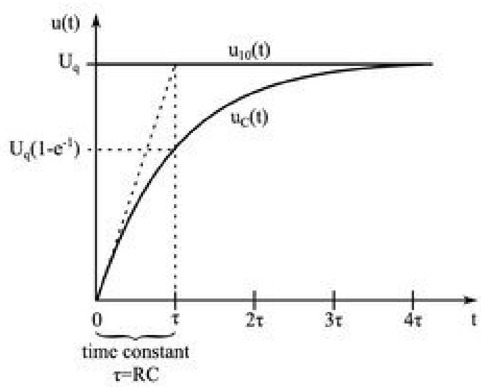


Fig. 7.15: The complete response of inhomogeneous first order ODE for an RC circuit.

Using the capacitor's voltage $u_C(t)$ the voltage across the resistor $u_R(t)$ and the current i(t) can be calculated:

$$i(t) = C \cdot \frac{du_C(t)}{dt} = \frac{U_q}{R} \cdot e^{-\frac{t}{RC}}$$

$$u_R(t) = i(t) \cdot R = U_q \cdot e^{-\frac{t}{RC}} = U_q - u_C(t)$$

$$e_C(t) = \frac{1}{2} C \cdot u_C^2(t) = \frac{1}{2} C \cdot U_q^2 \cdot \left(1 - e^{-\frac{t}{RC}}\right)^2$$

Example of an RC circuit with a charged capacitor

In the discussion of the RC circuit above the capacitor was uncharged at the beginning and the initial condition was $u_C(0) = 0$ V. Consider the same circuit like before (see Fig. 7.14), but this time the capacitor is charged for t < 0 s to a value of u_{C0}. The charging of the capacitor for t < 0 does not change the differential equation of the system:

$$\frac{du_C(t)}{dt} + \frac{1}{RC} \cdot u_C(t) = \frac{1}{RC} \cdot U_q(t)$$

The steady-state response of the system and the solution of the homogeneous differential equation are the same like before:

$$\lim_{t \to \infty} u_{Cp}(t) = U_q$$

$$u_{Ch}(t) = u_{Ch}(0) \cdot e^{-\frac{t}{RC}}$$

But the pre-charging of the capacitor changes the initial conditions of the system at t = 0 s:

$$U_q = u_R(0) + u_C(0) = u_R(0) + u_{C0}$$

Hence $u_{Ch}(0)$ can be calculated to be:

$$u_C(0) = u_{C0} = u_{Ch}(0) + u_{Cp}(0) = U_q + u_{Ch}(0)$$

$$\Rightarrow u_{Ch}(0) = u_{C0} - U_q$$

Finally the solution yields:

$$u_C(t) = U_q + (u_{C0} - U_q) \cdot e^{-\frac{t}{RC}}$$

The capacitor voltage starts at u_{C0} and rises up to the steady-state value of U_q.
For given values of C = 75 nF, u_{C0} = 25 V, U_q = 200 V and R = 10 kΩ the solution is:

$$u_C(t) = 200V - 175V \cdot e^{-\frac{t}{\tau}}$$

The time constant is:

$$\tau = R \cdot C = 750 \mu s$$

Example of an RL circuit
Consider an RL circuit like depicted in Fig. 7.16. The switch is open for t < 0 s and it is closed at t = 0 s. For t < 0 s the circuit is not closed and no current flows. At t = 0 s the current cannot change instantaneously and the voltage across the inductor equals the voltage of the source:

$$i(0) = 0A$$

$$u_L(0) = U_q$$

For t > 0 s the current rises to its steady-state value at t → ∞ s. In steady-state the inductor acts like a short-circuit and the current is given by Ohm’s law

$$i(\infty) = i_p = \frac{U_q}{R}$$

The steady-state current corresponds to the particular solution of the first order ODE.

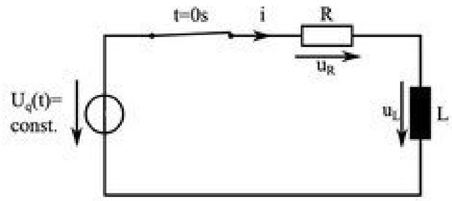


Fig. 7.16: RL circuit with a switch. Switch closes at t = 0 s.

The differential equation of the circuit for t > 0 s can be obtained by KVL and using Ohm’s law and the inductor relation:

$$U_q = u_R(t) + u_L(t) = R \cdot i(t) + L \cdot \frac{di(t)}{dt}$$

$$\frac{di(t)}{dt} + \frac{R}{L} \cdot i(t) = \frac{U_q}{L}$$

The general solution of this ODE for the current is:

$$i(t) = i_p + i_h(t) = \frac{U_q}{R} + i_h(0) \cdot e^{-\frac{R}{L}t}$$

The constant $i_h(0)$ can be calculated using the initial condition and the final result is:

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

The time constant τ for the RL circuit in series connection is given by:

$$\tau = \frac{L}{R}$$

For given values of L = 100 mH, U_q = 200 V and R = 20 Ω the solution is:

$$i(t) = 20A \cdot \left(1 - e^{-\frac{t}{5\,ms}}\right)$$

The time constant is τ = 5 ms and it takes the circuit about 5·τ ≈ 25 ms to be within 1 % of the steady-state value.

7.3.3 Second order circuit – the natural response

In the previous chapter the circuits contained one energy storing element, a capacitor or an inductor (besides resistors that do not store but dissipate energy). As a consequence the resulting equations were first order ODEs.
If two energy storing elements are part of a circuit under investigation this circuit is called a second order circuit as this kind of circuit will be described by linear second order ODEs.
An example for a second order circuit is depicted in Fig. 7.17. No source is part of this series connection of an inductor, a capacitor and resistor. The voltage across the capacitor is denoted with $u_C(t)$ and the current with i(t). The switch is open for t < 0 s and is closed at t = 0 s, initial conditions are $u_C(0)$ and i(0) respectively (For example, the capacitor was charged by an external voltage source $-U_q = u_C(0)$). Due to the open switch the current is zero for t < 0 s and due to the inductor it stays zero at t = 0 s.

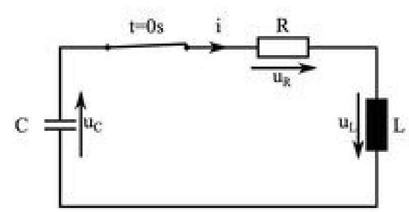


Fig. 7.17: Series connection of resistor, inductor and capacitor as an easy example for a second order circuit.

Applying KVL to the circuit yields for t > 0 s:

$$u_C(t)+u_R(t)+u_L(t)=0$$

In the next step the voltages across the inductor and the resistor are expressed in terms of the capacitor's voltage $u_C(t)$ using Ohm's law and the relation between the capacitor's voltage and the current:

$$i(t) = C \cdot \frac{du_C(t)}{dt}$$

$$u_R(t) = R \cdot i(t) = R \cdot C \cdot \frac{du_C(t)}{dt}$$

$$u_L(t) = L \cdot \frac{di(t)}{dt} = L \cdot C \cdot \frac{d^2 u_C(t)}{dt^2}$$

Finally we get this for the capacitor's voltage:

$$u_C(t) + R \cdot C \cdot \frac{du_C(t)}{dt} + L \cdot C \cdot \frac{d^2 u_C(t)}{dt^2}$$

$$\Rightarrow \frac{d^2 u_C(t)}{dt^2} + \frac{R}{L} \cdot \frac{du_C(t)}{dt} + \frac{1}{LC} \cdot u_C(t) = 0$$

This is a homogeneous second order ODE for the capacitor's voltage.

Excursus into mechanics: damped harmonic oscillator

A system which exhibits mathematically identical behavior to that of a similar, but physically different system is analogous to this system. In the case of the RLC-circuit the homogeneous second order ODE has the same structure (same but the constants) like the ODE for a damped harmonic oscillator such as given in Fig. 7.18. A mass m is connected to a spring and the movement x(t) is damped by friction. Balance of forces yields:

$$m \cdot \frac{d^2 x(t)}{dt^2} + d \cdot \frac{dx(t)}{dt} + c \cdot x(t) = 0$$

Here the force for the spring (proportional to the position of the mass) was used:

$$F_{spring} = c \cdot x(t)$$

Also the damping force (proportional to the velocity v(t) = dx(t)/dt) was used:

$$F_{damp} = d \cdot \frac{dx(t)}{dt} = d \cdot v(t)$$

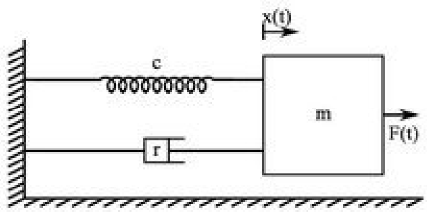


Fig. 7.18: A mechanical analog to the electrical RLC circuit.

As the mathematical behavior of both mechanical and electrical systems is identical, these systems are analog. Force causes velocity just as voltage causes current. A damper dissipates mechanical energy into heat just like a resistor dissipates electrical energy into heat. Springs and masses store energy in two different ways (potential energy and kinetic energy respectively) just as capacitors and inductors store energy in two different forms (electric and magnetic field respectively).
Analog quantities are listed in Tab. 7.2:

Tab. 7.2: Analog quantities of mechanical and electrical systems.

Mechanical	**Electrical**
Force	Voltage
Velocity	Current
Displacement	Charge
Damper (f(t) = d·v(t))	Resistor (u(t) = R·i(t))
Spring (f(t) = c·x(t)=c·∫v(t)dt)	Capacitor (u(t) = 1/C·∫i(t)dt

c·x(t)=c·∫v(t)dt)

Mass (f(t) = m·dv(t)/dt)	Inductor (u(t) = L·di(t)/dt

Due to the analogy to the mechanical system we can expect the behavior of the RLC-CIRCUIT to be the same as known from the damped oscillator: some kind of damped oscillations with different solutions depending on the values of m, c and d.

After we found a mechanical analogy to the RLC circuit (and we can expect what the solution might look like) we have to solve the second order homogeneous ODE for our electrical oscillating circuit:

$$\frac{d^2u_C(t)}{dt^2}+\frac{R}{L}\cdot\frac{du_C(t)}{dt}+\frac{1}{LC}\cdot u_C(t)=0$$

Excursus: Solution of linear homogeneous second order ODE

A linear homogeneous second order ODE

$$\frac{d^2x(t)}{dt^2}+2\alpha\cdot\frac{dx(t)}{dt}+\omega_n^2\cdot x(t)=0$$

can be solved by the following approach:

$$x(t)=e^{st}$$

Using this approach for the ODE yields

$$e^{st}\cdot\left(s^2+2\alpha\cdot s+\omega_n^2\right)=0$$

$$\left(s^2+2\alpha\cdot s+\omega_n^2\right)=0$$

This polynomial is called the characteristic polynomial of the corresponding ODE. This quadratic equation has two solutions:

$$s_1 = -\alpha - \sqrt{\alpha^2 - \omega_n^2}$$

$$s_2 = -\alpha + \sqrt{\alpha^2 - \omega_n^2}$$

All functions using the given approach and the roots (nulls) s_1 and s_2 of the characteristic polynomial are solutions to the given ODE:

$$x_i(t) = e^{s_i t}$$

Consequently all linear combinations of these basic solutions are also solutions to the ODE and the general solution for the ODE is:

$$x(t) = A_1 \cdot e^{s_1 t} + A_2 \cdot e^{s_2 t}$$

A_1 and A_2 are constant and are determined by the initial conditions. Depending on the values of α and ω_n three different cases have to be distinguished: $\alpha > \omega_n$, $\alpha < \omega_n$ and $\alpha = \omega_n$.
These three cases will be discussed during the analysis of the RLC circuit.

Coming back to our original problem of the RLC circuit:

$$\frac{d^2 u_C(t)}{dt^2} + \frac{R}{L} \cdot \frac{du_C(t)}{dt} + \frac{1}{LC} \cdot u_C(t) = 0$$

We can determine the solution using:

$$\alpha = \frac{R}{2L}$$

$$\omega_n^2 = \frac{1}{LC}$$

Depending on the values for R, L and C we have to distinguish three cases.

The overdamped case (aperiodic case): $\alpha > \omega_n$

If $\alpha > \omega_n$ R, L and C have to fulfill:

$$\frac{R}{2L} > \sqrt{\frac{1}{LC}}$$

In this case the roots of the characteristic polynomial s_1 and s_2 are real and negative:

$$s_1 = -\frac{R}{2L} - \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}}$$

$$s_2 = -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}}$$

Using these values of s_1 and s_2 the solution for the capacitor's voltage is:

$$u_C(t) = A_1 \cdot e^{s_1 t} + A_2 \cdot e^{s_2 t}$$

Constants A_1 and A_2 have to be determined by the initial conditions of the system. Initial conditions are $u_C(0)$ and $i(0)$ respectively (e.g. the capacitor was charged by an external voltage source $U_q = -u_C(0)$). Due to the open switch the current is zero for $t < 0$ s and due to the inductor it stays zero at $t = 0$ s. So there are two initial conditions to determine the two constants A_1 and A_2.

For t = 0 s we get:

$$u_C(0) = -U_q = A_1 \cdot e^{s_1 0} + A_2 \cdot e^{s_2 0} = A_1 + A_2$$

Due to the current being zero at t = 0 s the voltage change across the capacitor is also zero:

$$i(0) = C \cdot \left.\frac{du_C(t)}{dt}\right|_{t=0}$$

$$\frac{du_C(t)}{dt} = A_1 \cdot s_1 \cdot e^{s_1 t} + A_2 \cdot s_2 \cdot e^{s_2 t}$$

At t = 0 s this equation yields:

$$A_1 \cdot s_1 \cdot e^{s_1 0} + A_2 \cdot s_2 \cdot e^{s_2 0} = A_1 \cdot s_1 + A_2 \cdot s_2 = 0$$

With these two equations A_1 and A_2 can be determined:

$$A_1 = -U_q - A_2$$

$$A_2 \cdot (s_2 - s_1) = s_1 \cdot U_q$$

$$\Rightarrow A_2 = \frac{s_1 \cdot U_q}{s_2 - s_1}$$

$$\Rightarrow A_1 = -U_q - \frac{s_1 \cdot U_q}{s_2 - s_1} = -\frac{s_2 \cdot U_q}{s_2 - s_1}$$

$$u_C(t) = \left(-\frac{s_2 \cdot U_q}{s_2 - s_1}\right) \cdot e^{s_1 t} + \left(\frac{s_1 \cdot U_q}{s_2 - s_1}\right) \cdot e^{s_2 t}$$

The voltage across the capacitor decreases according to two exponential functions with time constants $1/s_1$ and $1/ s_2$ without any oscillation. This case is called the overdamped or aperiodic case and is depicted in Fig. 7.19. With knowledge of the capacitor’s voltage the current i(t) can easily be calculated by:

$$i(t) = C \cdot \frac{du_C(t)}{dt}$$

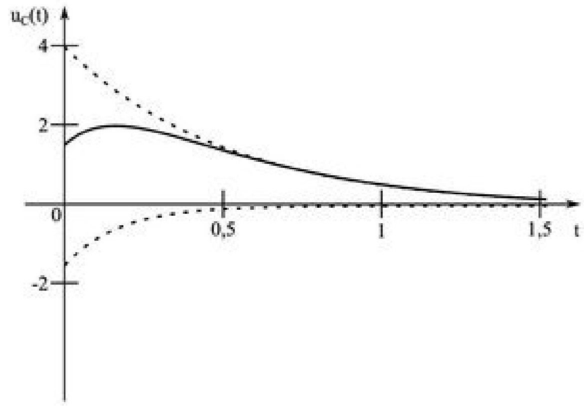


Fig. 7.19: A capacitor's voltage as a function of time for the overdamped case.

The underdamped case (periodic case): $\alpha < \omega_n$

If $\alpha < \omega_n$ then the roots of the characteristic polynomial are conjugate complex numbers (j is the imaginary unit, $j^2 = -1$)):

$$s_1 = -\frac{R}{2L} - \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha - j\sqrt{\omega_n^2 - \alpha^2} = -\alpha - j\omega_d$$

$$s_2 = -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha + j\sqrt{\omega_n^2 - \alpha^2} = -\alpha + j\omega_d$$

Here the ω_d is:

$$\omega_d = \sqrt{\omega_n^2 - \alpha^2}$$

Using these two complex number yields for the capacitor's voltage:

$$u_C(t) = A_1 \cdot e^{(-\alpha - j\omega_d)t} + A_2 \cdot e^{(-\alpha + j\omega_d)t} \text{ u}$$

$$u_C(t) = e^{-\alpha t} \cdot \left(A_1 \cdot e^{-j\omega_d t} + A_2 \cdot e^{j\omega_d t}\right)$$

The first factor again describes the damping with a time constant of $1/\alpha$. What about the term in brackets? As the voltage has to be a real number, A_1 and A_2 have to be complex conjugates of each other. We can rewrite A_1 and A_2 using new constants a and b:

$$A_1 = a - jb$$

$$A_2 = a + jb$$

Using the new constants for A_1 and A_2 yields for the voltage:

$$u_C(t) = e^{-\alpha t} \cdot \left(a \cdot \left(e^{j\omega_d t} + e^{-j\omega_d t} \right) - b \cdot \frac{e^{j\omega_d t} - e^{-j\omega_d t}}{j} \right)$$

With Euler's formular:

$$e^{j\Theta} = \cos(\Theta) + j\sin(\Theta)$$

The terms in brackets can be rewritten as:

$$e^{j\omega_d t} + e^{-j\omega_d t} = 2\cos(\omega_d \cdot t)$$

$$\frac{e^{j\omega_d t} - e^{-j\omega_d t}}{j} = 2\sin(\omega_d \cdot t)$$

$$\begin{aligned} \Rightarrow u_C(t) &= e^{-\alpha t} \cdot \left(2a \cdot \cos(\omega_d \cdot t) - 2b \cdot \sin(\omega_d \cdot t)\right) \\ &= e^{-\alpha t} \cdot \left(C_1 \cdot \cos(\omega_d \cdot t) + C_2 \cdot \sin(\omega_d \cdot t)\right) \end{aligned}$$

This expression can be further combined into a single sinusoidal function:

$$u_C(t) = e^{-\alpha t} \cdot C_0 \cdot \cos(\omega_d \cdot t - \psi)$$

To obtain this last equation the following equations for the new (and last...) constants have to hold true:

$$C_0 = \sqrt{C_1^2 + C_2^2}$$

$$\psi = \arctan \frac{C_2}{C_1}$$

Finally the following equation is the general solution to the homogeneous linear second order ODE given above:

$$u_C(t) = e^{-\alpha t} \cdot C_0 \cdot \cos(\omega_d \cdot t - \psi)$$

The constants C_0 and ψ have to be determined by the initial conditions.
Looking at Fig. 7.20 the parameters can be interpreted as follows:
The capacitor's voltage and current oscillate with the frequency ω_d (damped frequency) around the steady-state value of zero (both current and voltage). The peak value of the oscillations decreases exponentially with a damping factor α. During the oscillation energy is transferred from the capacitor to the inductor and vice versa, during the transfer energy is dissipated in the resistor. α depends linearly on the resistance value R. In the ideal case of R = 0 Ω there is no damping (α = 0) of the oscillation and the damped frequency ω_d equals the natural frequency ω_n that is just determined by the inductor and capacitor. ψ is the phase angle that describes the shift of the zero crossing of the oscillation with respect to t = 0 s.

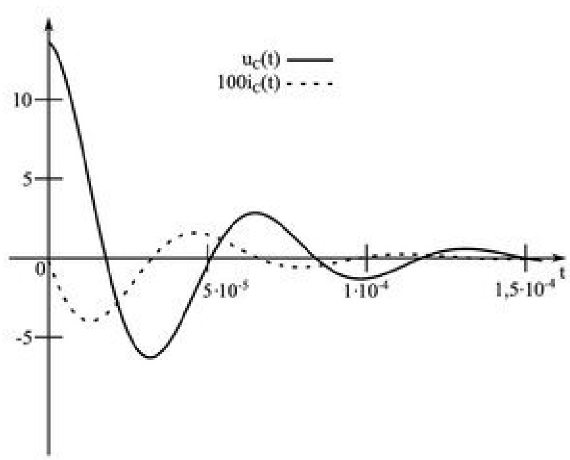


Fig. 7.20: A Capacitor's voltage and current for the underdamped case.

The critically damped case: $\alpha = \omega_n$

In case of $\alpha = \omega_n$ the roots of the characteristic polynomial are equal:

$$s_1 = s_2 = -\alpha$$

In this case the second order ODE looks like:

$$\frac{d^2u_C(t)}{dt^2} + 2\alpha \cdot \frac{du_C(t)}{dt} + \alpha^2 \cdot u_C(t) = 0$$

The general solution is:

$$u_C(t) = (A_1 \cdot t + A_2)e^{-\alpha t}$$

A_1 and A_2 are again constants to be determined by the initial conditions. The capacitor's voltage decreases exponentially and the decay is faster compared to all other overdamped cases.

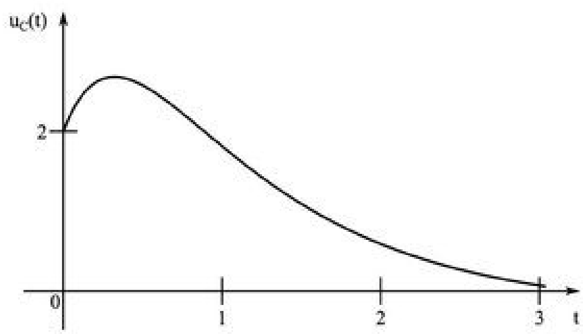


Fig. 7.21: A capacitor's voltage for the critically damped case.

7.3.4 Second order circuit – the complete response

After the study of the natural response (no source) circuits with a source as excitation such as depicted in Fig. 7.22 are considered. A resistor R, an inductor L and a capacitor C are in series to a switch. At t = 0 s the switch is closed and a voltage source is connected in series.

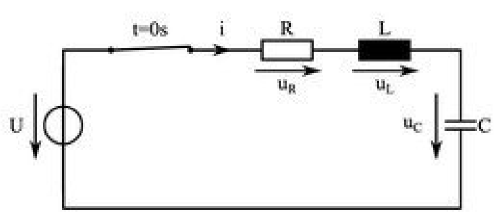


Fig. 7.22: RLC series connection, voltage source connected at t = 0 s.

Before we analyze the circuit in detail let's try to figure out what will happen qualitatively, based on our experience from the natural response. With the switch open no current flows (i(0) =

0 A) and the capacitor is uncharged ($u_C(0) = 0$ V). No energy is stored in the energy storing elements L and C. Closing the switch will make a current flow, charging the capacitor and the inductor. For a constant source U the capacitor will in the end block any current flow. At that time the current flow will stop, the capacitor's voltage $u_C(t)$ will be equal to the source voltage U and the energy will be stored in the capacitor. As no current will be flowing no energy will be stored in the inductor in the end.
The detailed analysis looks like:

t < 0 s
As the switch is open, all voltages $u_L(t)$, $u_R(t)$ and $u_C(t)$ are zero and no current flows through the circuit.

t = 0 s
The switch is closed instantaneously. As the voltage across the capacitor cannot change instantaneously it will be zero, $u_C(0) = 0$ V. As the current through the inductor cannot change instantaneously, it will also be zero, $i(0) = 0$ A. These two equations define our initial conditions.

t > 0 s
The switch stays closed and according to KVL we get:

$$u_C(t) + u_R(t) + u_L(t) = U$$

In the next step the voltages across the inductor and the resistor are expressed in terms of the capacitor's voltage $u_C(t)$ using Ohm's law and the relation between the capacitor's voltage and the current:

$$i(t) = C \cdot \frac{du_C(t)}{dt}$$

$$u_R(t) = R \cdot i(t) = R \cdot C \cdot \frac{du_C(t)}{dt}$$

$$u_L(t) = L \cdot \frac{di(t)}{dt} = L \cdot C \cdot \frac{d^2 u_C(t)}{dt^2}$$

Finally we get for the capacitor's voltage:

$$\frac{d^2 u_C(t)}{dt^2} + \frac{R}{L} \cdot \frac{du_C(t)}{dt} + \frac{1}{LC} \cdot u_C(t) = \frac{1}{LC} \cdot U$$

This is an inhomogeneous second order ODE for the capacitor's voltage. As for the inhomogeneous first order ODE we will split the general solution into two parts, a solution to the homogeneous ODE and a particular solution:

$$u_C(t) = u_{Cp}(t) + u_{Ch}(t)$$

The homogeneous solution $u_{Ch}(t)$ was already determined in the previous section. The roots of the characteristic polynomial of the ODE are:

$$s_1 = -\frac{R}{2L} - \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha - \sqrt{\alpha^2 - \omega_n^2}$$

$$s_2 = -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha + \sqrt{\alpha^2 - \omega_n^2}$$

Using the abbreviations:

$$\alpha = \frac{R}{2L}$$

$$\omega_n^2 = \frac{1}{LC}$$

The homogeneous solution is:

$$u_{Ch}(t) = A_1 \cdot e^{s_1 t} + A_2 \cdot e^{s_2 t} = A_1 \cdot e^{\left(-\alpha-\sqrt{\alpha^2-\omega_n^2}\right)t} + A_2 \cdot e^{\left(-\alpha+\sqrt{\alpha^2-\omega_n^2}\right)t}$$

The particular solution $u_{Cp}(t)$ is determined by the excitation of the circuit (by the voltage source U). It is the solution of the particular ODE and describes the steady-state behavior of the circuit (t → ∞ s) forced by the excitation (forced response). For a constant forcing function U the steady-state response yields for t → ∞ s:

$$u_{Cp}(t \to \infty) = U$$

Looking at the circuit it is obvious that the steady-state response will be just the voltage of the voltage source (excitation voltage) as in the steady-state the circuit is a DC circuit without any current flowing and the capacitor's voltage corresponds to the voltage of the source. The complete response is thus:

$$u_C(t) = U + A_1 \cdot e^{\left(-\alpha-\sqrt{\alpha^2-\omega_n^2}\right)t} + A_2 \cdot e^{\left(-\alpha+\sqrt{\alpha^2-\omega_n^2}\right)t}$$

Based on the roots of the characteristic polynomial s_1 and s_2 three cases can be distinguished (like of the homogeneous second order ODE) depending on the values of R, L and C (and therefore α and ω_n): the overdamped, the underdamped and the critically damped case. In all three cases the capacitor's voltage will tend towards the steady-state value U. Constants A_1 and A_2 are determined by the initial conditions.
Based on $u_C(t)$ all other values can be calculated:

$$i(t) = C \cdot \frac{du_C(t)}{dt} = C \cdot \left(A_1 \cdot s_1 \cdot e^{s_1 \cdot t} + A_2 \cdot s_2 \cdot e^{s_2 \cdot t}\right)$$

$$u_R(t) = R \cdot i(t) = R \cdot C \cdot \left(A_1 \cdot s_1 \cdot e^{s_1 \cdot t} + A_2 \cdot s_2 \cdot e^{s_2 \cdot t}\right)$$

$$u_L(t) = L \cdot \frac{di(t)}{dt} = L \cdot C \cdot \left(A_1 \cdot s_1^2 \cdot e^{s_1 \cdot t} + A_2 \cdot s_2^2 \cdot e^{s_2 \cdot t}\right)$$

The overdamped case (aperiodic case): α > ω$_n$ For α > ω_n, R, L and C have to fulfill:

$$\frac{R}{2L} > \sqrt{\frac{1}{LC}}$$

In this case the roots of the characteristic polynomial s_1 and s_2 are real and negative, the solution is a function that changes by two exponential functions tending towards the steady-state value U:

$$u_C(t) = U + A_1 \cdot e^{\left(-\alpha - \sqrt{\alpha^2 - \omega_n^2}\right) \cdot t} + A_2 \cdot e^{\left(-\alpha + \sqrt{\alpha^2 - \omega_n^2}\right) \cdot t}$$

The behavior is depicted in Fig. 7.23.

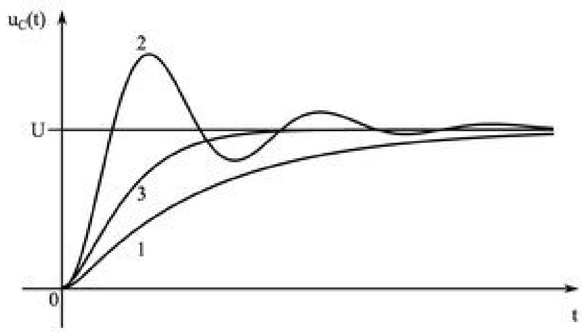


Fig. 7.23: A capacitor's voltage as a function of time for an RLC series connection, voltage source U connected at t = 0 s : 1: overdamped; 2: underdamped; 3: critically damped.

The underdamped case (periodic case): α < ω$_n$

If α < ω_n than the roots of the characteristic polynomial are conjugate complex numbers:

$$s_1 = -\frac{R}{2L} - \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha - j\sqrt{\omega_n^2 - \alpha^2} = -\alpha - j\omega_d$$

$$s_2 = -\frac{R}{2L} + \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}} = -\alpha + j\sqrt{\omega_n^2 - \alpha^2} = -\alpha + j\omega_d$$

The general solution is:

$$u_C(t) = U + e^{-\alpha t} \cdot C_0 \cdot \cos(\omega_d \cdot t - \psi)$$

As shown in Fig. 7.23 the capacitor's voltage oscillates with an angular frequency ω_d around the steady-state value U. The peak value decreases with an exponential function with time constant $1/\alpha$.

The critically damped case: $\alpha = \omega_n$

In case of $\alpha = \omega_n$ the roots of the characteristic polynomial are same:

$$s_1 = s_2 = -\alpha$$

The solution is:

$$u_C(t) = U + (A_1 \cdot t + A_2)e^{-\alpha t}$$

In this case the capacitor's voltage tends fastest towards the final steady-state value U as depicted in Fig. 7.23.

Example of an RLC series circuit

The RLC circuit of Fig. 7.22 should be operated critically damped. The values U, L and C are given: U = 100 V, C = 80 nF, L = 40 mH. What about the value of the resistor for the critically damped case?

In critically damped case $\alpha = \omega_n$:

$$\frac{R}{2L} = \sqrt{\frac{1}{LC}}$$

$$\Rightarrow R = 2 \cdot \sqrt{\frac{L}{C}} = 1.41 k\Omega$$

For a resistor of R = 1.41 kΩ the circuit operates in critically damped mode and reaches the steady-state value of $u_C(\infty) = U = 100$ V in minimum time. If the value of the resistor is smaller the circuit starts to oscillate (underdamped case), if it is higher it takes more time to reach the steady-state value.

Automotive application

Switching is frequently required in automotive applications. Either single switching events, e.g. by the driver or frequent and continuous switching within the electronic system. In general the switching circuit consists of inductive, capacitive and resistive elements (taking parasitic effects into account even always). Depending on the size of these elements and the frequency a detailed analysis of the switching behavior has to be done to avoid an unwanted behavior, e.g. an oscillation or an overdamped case. Consider a switching event from 0 V to 5 V that has to be detected by a microcontroller (e.g. with an interrupt input pin or even with an ADC). In case of an oscillation the overshoot of the voltage (see curve 2 in Fig. 7.23) can disturb or even destroy the microcontroller input pin (and hence the microcontroller) as the maximum input value of the microcontroller is exceeded. In case of an overdamped case it may take long time to reach the final value. So the switching from 0 V to 5 V is maybe recognized to late by the microcontroller.

7.4 AC Analysis

During analysis of the oscillating circuits (like RC, RL and RLC circuits) the currents and voltages turned out not to be constant but time-dependent, either some kind of exponentially damped dependence or an oscillating behavior. But the sources have been (more or less) time independent so far. During the following AC analysis just steady state systems will be studied. All transient effects such as those previously discussed are settled and the system is in a steady state.
When using AC (alternating current) analysis we will make use of some findings from DC analysis:

– All events happen at the same time independent of the location within the circuit;

– Kirchhoff’s laws are valid for all instances of time;
– Superposition is still valid for linear elements like resistors, inductors, capacitors (these elements have linear dependencies (direct linear or derivative) between electrical properties like voltage and current);

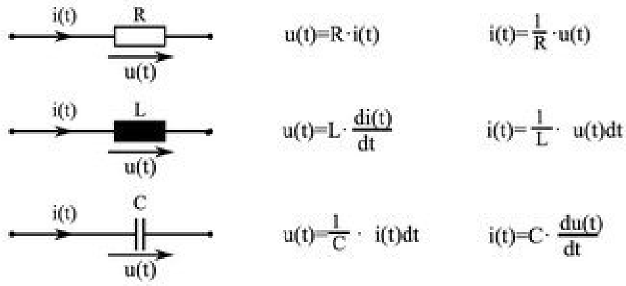


Fig. 7.24: Linear elements and their current-voltage relation.

In this section of AC analysis we will now deal with time-dependent sources (and of course voltages and currents), in particular with periodically time-dependent elements. Periodically time-dependent means that the shape u(t) of the time-dependence is repeated periodically after a time called the period T (k is some arbitrary integer constant):

$$u(t+kT)=u(t)$$

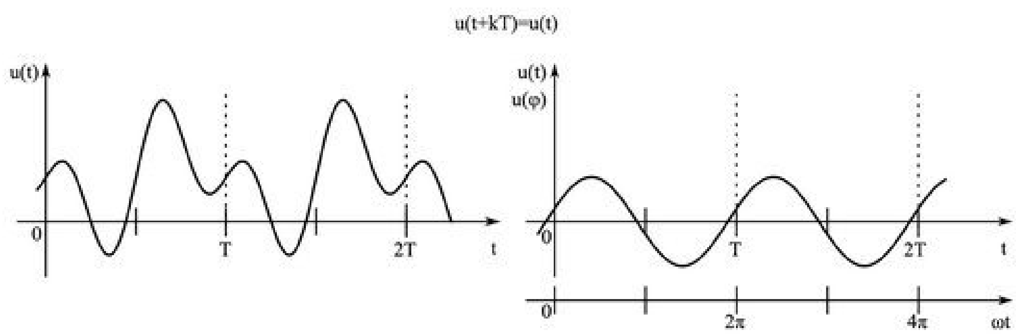


Fig. 7.25: Periodical functions u(t): arbitrary shape (left) and sinusoidal shape (right).

The dedicated value at a time t is the instantaneous value. When considering currents or voltages the arithmetical mean

$$\bar{u}=\frac{1}{T}\int_{t=t_0}^{t_0+T}u(t)dt=\frac{1}{2\pi}\int_{\omega t=\varphi_0}^{\varphi_0+2\pi}u(\omega t)d(\omega t)$$

of an alternating current or voltage is zero. In particular sinusoidal functions like shown in Fig. 7.25 on the right side have an arithmetic mean of zero:

$$u(t) = \hat{u} \cdot \sin(\omega t + \varphi)$$

$$\overline{u} = 0$$

The parameter û of the sinusoidal function is called the peak value and the angular frequency ω is related to the period T and the frequency f according to

$$\omega = \frac{2\pi}{T} = 2\pi f$$

Unit for the angular frequency ω is 1/s (whereas for frequency it is Hz). The starting point of the oscillation is in general not at t = 0 s but shifted for some time indicated by the phase angle ϕ. The difference of maximum and minimum value is called the peak-to-peak value. For sinusoidal shape the peak-to-peak value is 2·û.
Sinusoidal functions play a major role in AC circuit analysis:
The sinusoidal shape stays the same (for same frequency) for addition of sinusoidal functions and also for differentiation. This is important when using superposition and circuit analysis techniques like KCL and KVL.
In addition, by using Fourier analysis every periodic function may be represented by a sum of sinusoidal functions. Therefore the analysis of arbitrary shaped functions can be reduced to analysis of the sinusoidal functions.
The sinusoidal time-dependence of a current and a voltage looks like:

$$u(t) = \hat{u} \cdot \sin(\omega t + \varphi_u)$$

$$i(t) = \hat{i} \cdot \sin(\omega t + \varphi_i)$$

û and î are the peak values of the voltage and the current respectively. The frequency is in this case the same for both, but the phase angle is different. The phase angle is counted positive if pointing to the right and negative if otherwise. The phase difference between voltage and current is

$$\varphi = \varphi_u - \varphi_i$$

As shown in Fig. 7.26 the zero-crossing of the voltage (shifted by ϕ_u to the left) is earlier than the zero-crossing of the current (shifted by ϕ_i to the right): the voltage leads the current. In the opposite case (current earlier than voltage) the phase difference is negative and the current leads the voltage. In Fig. 7.26 both current and voltage are depicted in one single diagram even though these two have different values and units. On the y-axis it is denoted that both current and voltage are used. Even though this labeling of the y-axis will be omitted in following figures (which is rather common in AC analysis) it should be clear that voltage and current differ in size and unit.

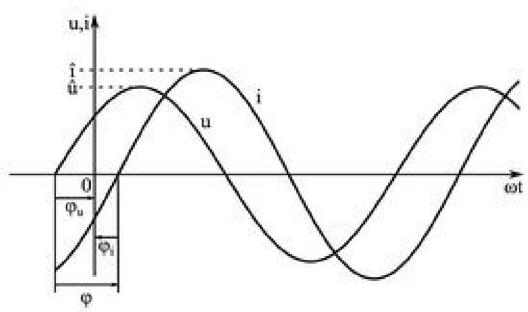


Fig. 7.26: Sinusoidal voltage and current with different phase angle and same frequency.

The arithmetic mean of an AC current, or voltage is zero. For certain applications another value, the rectified value, is used to describe the average effect of current or voltage:

$$\overline{|i|} = \frac{1}{T}\int_0^T |i(t)|dt$$

$$\overline{|u|} = \frac{1}{T}\int_0^T |u(t)|dt$$

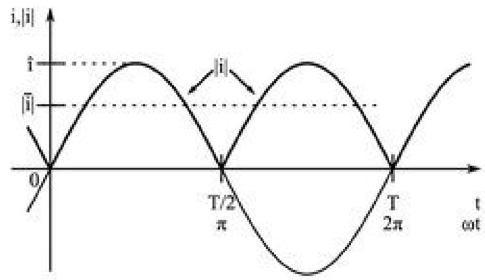


Fig. 7.27: Sinusoidal current, absolute value and rectified value.

For sinusoidal shape (e.g. current $\hat{i}\cdot\sin(\omega t+\phi_i)$) the rectified value is:

$$\overline{|i|} = \frac{2}{\pi}\cdot\hat{i}$$

Besides the rectified value the root-mean-square value (RMS) is more important, in case of current and voltage:

$$I_{eff} = I = \sqrt{\frac{1}{T}\int_0^T i^2(t)dt}$$

$$U_{eff} = U = \sqrt{\frac{1}{T}\int_0^T u^2(t)dt}$$

By definition:

> The RMS value of an AC current is defined as the DC current that leads to the same power dissipation in a resistor.

For sinusoidal shape (e.g. current $\hat{i}\cdot\sin(\omega t+\phi_i)$) the RMS is:

$$I_{eff} = I = \frac{\hat{i}}{\sqrt{2}}$$

The RMS of a sinusoidal shape is just the peak value divided by $\sqrt{2}$.
Simple circuits with just one element can easily be calculated

using the well known correlations of current and voltage for resistors, capacitors and inductors.

Resistor

Consider a resistor connected to a sinusoidal voltage source:

$$u(t) = \hat{u} \cdot \sin(\omega t)$$

The current through the resistor is given by Ohm's law:

$$i(t) = \frac{u(t)}{R} = \frac{\hat{u}}{R} \cdot \sin(\omega t) = \hat{i} \cdot \sin(\omega t)$$

Current and voltage are in phase (no phase difference between current and voltage). In Fig. 7.28 the line diagram shows both voltage and current. As no scaling for any of the two is given the size of the curves is unimportant. And both have of course different units. This kind of line diagram just serves to show the phase difference between voltage and current (no phase difference for the resistor).

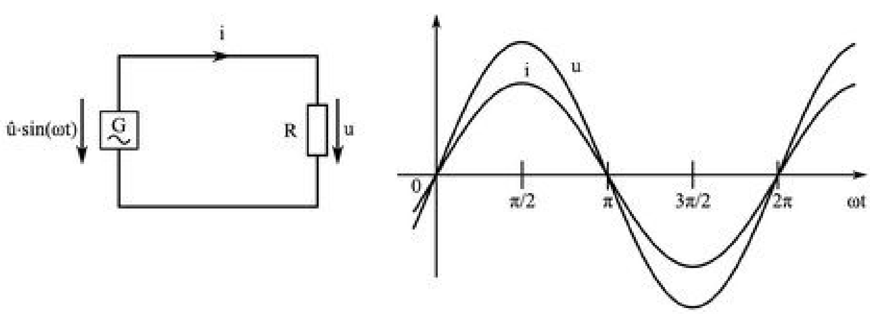


Fig. 7.28: A resistor connected to a sinusoidal voltage source: circuit (left) and line diagram (right) of current and voltage.

Inductor

Consider an inductor connected to a sinusoidal voltage source:

$$u(t) = \hat{u} \cdot \sin(\omega t)$$

The current through the inductor is given by:

$$i(t) = \frac{1}{L} \cdot \int_0^t u(\tau) d\tau = \frac{1}{L} \cdot \int_0^t \hat{u} \cdot \sin(\omega\tau) d\tau = -\frac{\hat{u}}{\omega L} \cdot \cos(\omega t) + i(0)$$

$$= \frac{\hat{u}}{\omega L} \cdot \sin\left(\omega t - \frac{\pi}{2}\right) + i(0)$$

Current and voltage are not in phase this time as depicted in Fig. 7.29, the voltage leads the current by Π/2 or 90 °.

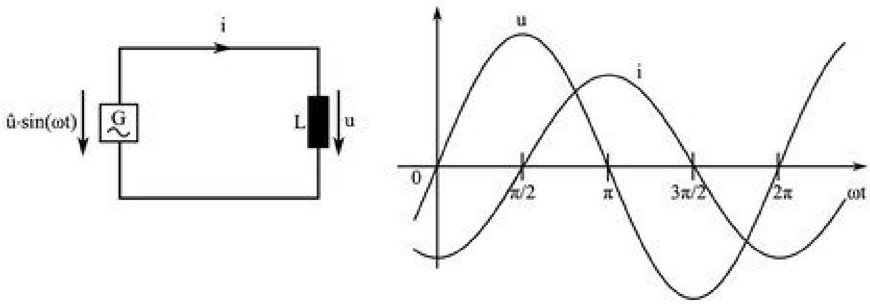


Fig. 7.29: An inductor connected to a sinusoidal voltage source: circuit (left) and line diagram (right) of current and voltage.

Capacitor

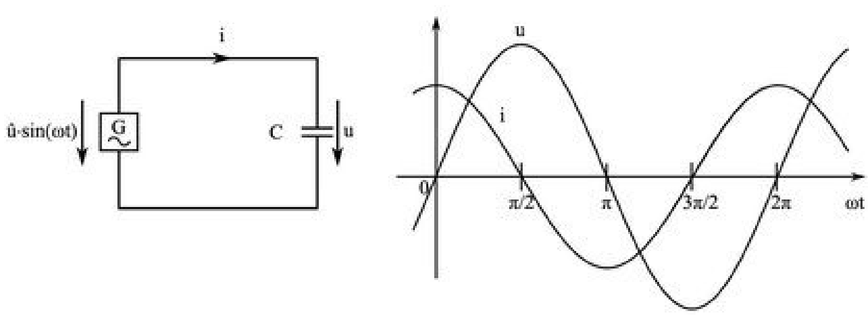
Consider a capacitor connected to a sinusoidal voltage source:

$$u(t) = \hat{u} \cdot \sin(\omega t)$$

The current in the circuit is given by:

$$i(t) = C \cdot \frac{du(t)}{dt} = C \cdot \hat{u} \cdot \frac{d}{dt}(\sin(\omega t)) = C \cdot \hat{u} \cdot \omega \cdot \cos(\omega t) = C \cdot \hat{u} \cdot \omega \cdot \sin\left(\omega t + \frac{\pi}{2}\right)$$

Current and voltage are not in phase this time, as the current leads the voltage by Π/2 or 90 °. This behavior is depicted in Fig. 7.30.



[Fig. 7.30](#): A capacitor connected to a sinusoidal voltage source: circuit (left) and line diagram (right) of current and voltage.

7.4.1 Vector diagram

Sinusoidal currents and voltages can be shown in line diagrams as depicted in the figures above and by equations using the sine and cosine functions. But for AC analysis these notations are complex to use. To simplify the calculation of AC circuits, pointers and vectors and the complex representation of voltages and currents are used. Just imagine a simple addition of two currents with same angular frequency but different phase angle as depicted in Fig. 7.31:

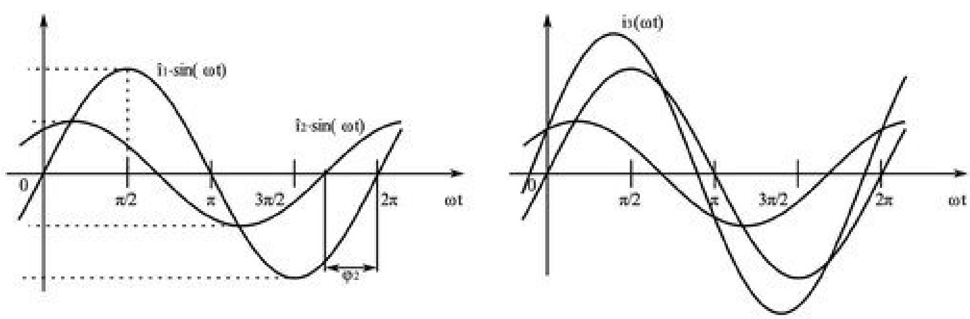


Fig. 7.31: Two sinusoidal currents that should be added (left); result of addition (right).

The resulting current will have a new peak value $\hat{i}_3$ and a new phase angle ϕ_3. Both values have somehow to be determined.
The addition can be done graphically as shown on the right side of Fig. 7.31. For any change in frequency of phase this graphical solution has to be repeated.
Another way is to use the representation with sine and cosine functions, e.g. using addition theorem:

$$\sin(\alpha+\beta)=\sin(\alpha)\cdot\cos(\beta)+\cos(\alpha)\cdot\sin(\beta)$$

The addition theorem used for addition of the currents yields:

$$\hat{\imath}_3 \cdot \sin(\omega t + \varphi_3) = \hat{\imath}_1 \cdot \sin(\omega t + \varphi_1) + \hat{\imath}_2 \cdot \sin(\omega t + \varphi_2)$$

$$\Rightarrow \hat{\imath}_3 \cdot (\sin(\omega t) \cdot \cos(\varphi_3) + \cos(\omega t) \cdot \sin(\varphi_3)) =$$
$$(\hat{\imath}_1 \cdot \cos(\varphi_1) + \hat{\imath}_2 \cdot \cos(\varphi_2)) \cdot \sin(\omega t) + (\hat{\imath}_1 \cdot \sin(\varphi_1) + \hat{\imath}_2 \cdot \sin(\varphi_2)) \cdot \cos(\omega t)$$

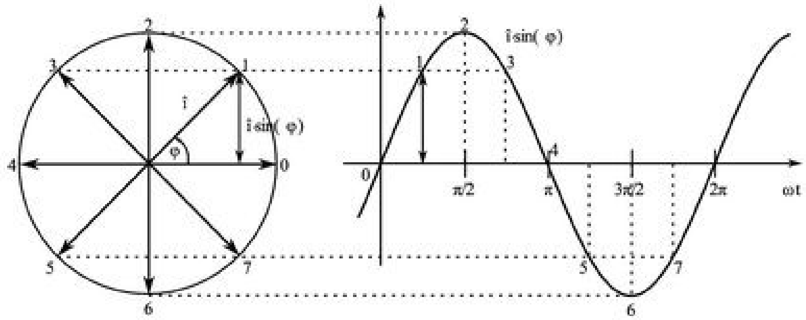
As the cosine and the sine function are independent of each other the equation has to be true for both functions and hence the corresponding coefficients have to be equal. This results in two equations for the values $\hat{\imath}_3$ and ϕ_3:

$$\hat{\imath}_3 = \sqrt{\hat{\imath}_1^2 + \hat{\imath}_2^2 + 2\hat{\imath}_1 \cdot \hat{\imath}_2 \cdot \cos(\varphi_1 + \varphi_2)}$$

$$\tan(\varphi_3) = \frac{\hat{\imath}_1 \cdot \sin(\varphi_1) + \hat{\imath}_2 \cdot \sin(\varphi_2)}{\hat{\imath}_1 \cdot \cos(\varphi_1) + \hat{\imath}_2 \cdot \cos(\varphi_2)}$$

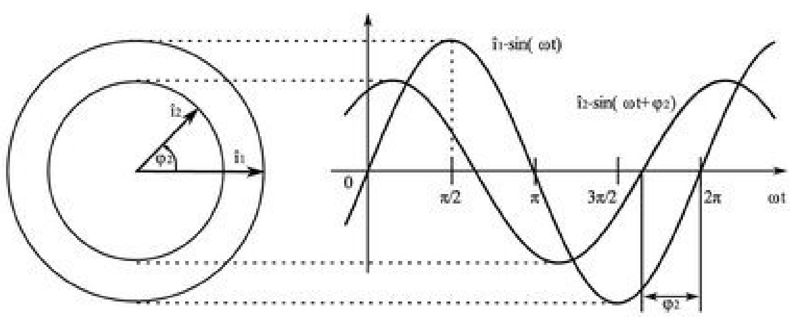
Even the simple addition of two currents is a rather complicated matter.

Pointer diagrams can be used to simplify the calculation with sinusoidal currents and voltages. Here currents and voltages are depicted as a vector, or pointer that rotates with an angular frequency of ω. Fig. 7.32 shows the graphical representation of a sinusoidal function $\hat{\imath} \cdot \sin(\omega t)$ on the right side. At every instance of time the value of the current is determined by the peak value $\hat{\imath}$ and the angular frequency ω. On the left side of this figure the corresponding vector diagram is shown. A pointer of length $\hat{\imath}$ rotates around the middle of the x-y-diagram with an angular frequency of ω. At any instance of time the angle ϕ corresponds to the angle of the sinusoidal, ωt. The sinusoidal on the right side is nothing other than the projection of the pointer in the vector diagram to the y-axis. The left side is the pointer representation of the sinusoidal.



[Fig. 7.32](#): Line (right) and vector representation (left) of a sinusoidal function.

This representation with rotating vectors makes it much easier to add two sinusoidal functions by just using vector addition. Consider again two currents that should be added as shown in [Fig. 7.31](#) on the right side. Current i_2 leads current i_1 by the phase ϕ_2. Transfer of this information to the vector diagram is shown on the left side of [Fig. 7.33](#). The instance of time (arbitrarily chosen) is t = 0 s. Current i_1 is represented by a vector in the direction of the x-axis of length $\hat{i}_1$. The projection to the y-axis is zero in correspondence to the value i_1 of at t = 0 s. At t = 0 s current i_2 is non-zero and rotated forward by phase difference ϕ_2 .The length of the vector is the peak value, $\hat{i}_2$.



[Fig. 7.33](#): Two currents with the same frequency but phase difference ϕ: line (right) and vector (left) representation..

The two vector representations are added by vector addition for any arbitrary time instance, here t = 0 s, the resulting vector is the total current that rotates around with angular frequency ω. The addition is done by graphical vector addition as depicted in [Fig. 7.34](#):

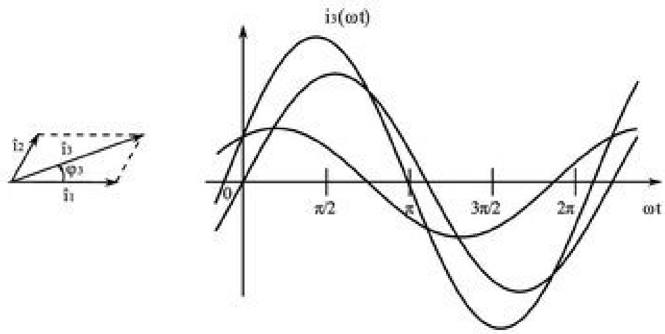


Fig. 7.34: Vector addition of two currents (left) and resulting line diagram (right).

The peak value of the total current is $\hat{i}_3$ and the resulting phase angle is ϕ_3 as depicted in Fig. 7.34. This resulting vector rotates with angular frequency ω. The transfer back to the line diagram is shown on the right side. So the addition of to time dependent currents with the same frequency is converted to a simple vector addition. This is still a graphical approach, not a mathematical calculation that would be preferred (in particular for simulations). However it is a good starting point for the representation by complex numbers, which is the next step in the description of AC values.
Of course the vector diagram method can also be used to determine the phase difference between current and voltage for a circuit element. Application of vector diagrams to the basic elements R, L and C yields:

Resistor
As we have seen previously current and voltage are in phase (no phase difference between current and voltage) for the resistor. Therefore the vectors for the current through and the voltage across the resistor are parallel.

Inductor
As calculated above, current and voltage are not in phase at the inductor, the voltage leads the current by Π/2 or 90 °. In the vector diagram (Fig. 7.35) the current is rotated clockwise by 90 ° (π/2) with respect to the voltage vector.

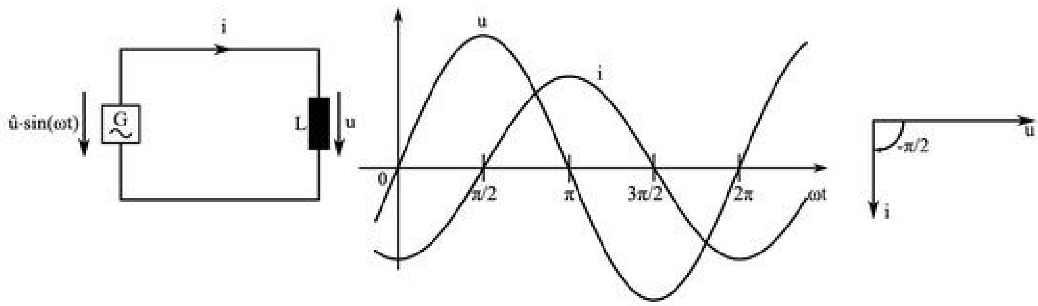


Fig. 7.35: Inductor connected to a sinusoidal voltage source: circuit (left), line diagram (mid) and vector diagram of current and voltage.

Capacitor

For the capacitor, the current and voltage are again not in phase, and this time the current leads the voltage by Π/2 or 90 °. In the vector diagram the current is rotated counterclockwise by 90° (π/2) with respect to the voltage vector (Fig. 7.36).

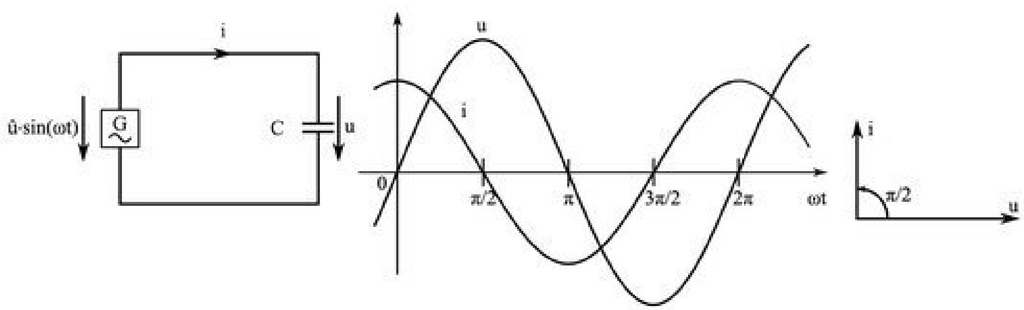


Fig. 7.36: Capacitor connected to a sinusoidal voltage source: circuit (left), line diagram (mid) and vector diagram (right) of current and voltage.

Example for the application of vector diagrams

The circuit depicted inFig. 7.37 is build up of two resistors and one capacitor and is excited by a sinusoidal voltage source û·sin(ωt). Resistor R_2 and the capacitor C are connected in parallel and together they are connected in series with resistor R_1. What about the voltages and currents in the elements and the total behavior of the circuit?

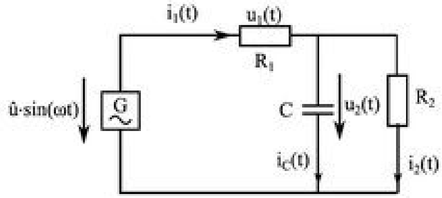


Fig. 7.37: A circuit with resistors, a capacitor and a sinusoidal voltage source.

The voltage drop $u_2(t)$ is the same for the parallel connection of R_2 and C. From Ohm's law we know that the current $i_2(t)$ through the resistor R_2 is in phase with the voltage drop $u_2(t)$. Using a vector diagram the magnitude $\hat{u}_2$ and $\hat{i}_2$ of the resistor point in the same direction (here arbitrarily to the right, see Fig. 7.38). For the capacitor the current $i_C(t)$ leads the voltage $u_2(t)$ by 90 ° and therefore points up as indicated in Fig. 7.38.
According to KCL the total current through R_2 and C has to be the same like the current $i_1(t)$. Hence the current $i_1(t)$ is the vectorial sum of the current vectors as given in Fig. 7.38. Again using Ohm's law the voltage across resistor R_1 is in phase with current $i_1(t)$ through R_1 and both voltage and current of R_1 point in the same direction. Current $i_1(t)$ is of course the current that is provided by the voltage source to the circuit.

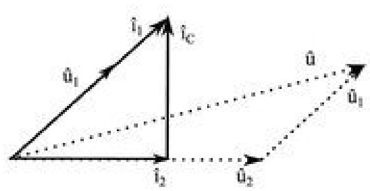


Fig. 7.38: Vector diagram of the RC circuit.

Using KVL for the left mesh (sinusoidal voltage source, R_1, C) yields the voltage by vectorial sum of $u_1(t)$ and $u_2(t)$ or $\hat{u}_1$ and $\hat{u}_2$.
As can be seen in Fig. 7.38 the current ($i_1(t)$) and voltage u(t)of the source are not in phase but the current leads the voltage. Therefore this circuit has a capacitve behavior.

7.4.2 Complex numbers

The vector diagram introduced above resembles the representation of complex numbers. Recall the imaginary number to be:

$$j = \sqrt{-1}$$

The sum of a real and imaginary number is called a complex number and a complex number $\underline{Z}$ can be represented in a Gaussian coordinate system by the rectangular form with the real part R = Re($\underline{Z}$) on the x-axis and the imaginary part X = Im($\underline{Z}$) on the y-axis:

$$\underline{Z} = R + jX$$

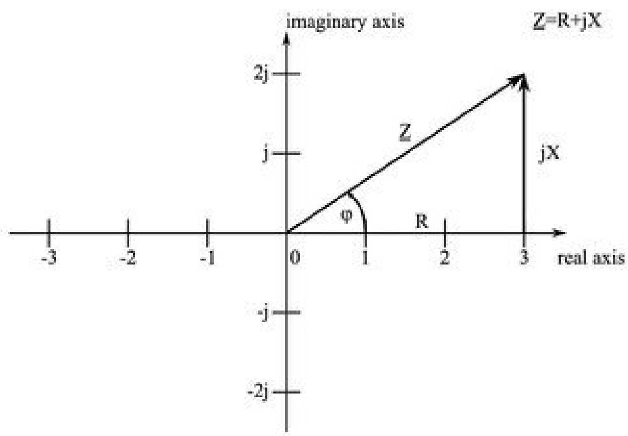


Fig. 7.39: Complex number $\underline{Z}$ in a Gaussian coordinate system.

The magnitude of the complex number (length of the vector from origin to the point in the Gaussian coordinate system) and the corresponding angle to the x-axis (real axis) are (see Fig. 7.39):

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

$$\varphi = \arctan\left(\frac{X}{R}\right)$$

Expressing the real and the imaginary part of the complex number by the magnitude and the angle yields:

$$R = Z \cdot \cos(\varphi)$$

$$X = Z \cdot \sin(\varphi)$$

Using Euler’s formula (see above) the complex number $\underline{Z}$ can be written in exponential or polar form:

$$\underline{Z} = Ze^{j\varphi}$$

Both representations of a complex number, rectangular and polar form, are used in AC analysis depending on the purpose. Sometimes the rectangular form is easier to handle, nevertheless most of the time the polar form is used as AC analysis deals a lot with differentiation, multiplication and division.
Some basic calculations with complex numbers:

Summation and subtraction
The easiest way for summation and subtraction of two complex numbers $\underline{Z}_1$ and $\underline{Z}_2$ is in rectangular form, as just the real and imaginary parts are added (subtracted) separately:

$$\underline{Z} = \underline{Z}_1 + \underline{Z}_2 = R_1 + R_2 + j(X_1 + X_2)$$
$$\underline{Z} = \underline{Z}_1 - \underline{Z}_2 = R_1 - R_2 + j(X_1 - X_2)$$

Multiplication and division
The easiest way for the multiplication and subtraction of two complex numbers $\underline{Z}_1$ and $\underline{Z}_2$ is in polar form, as the magnitude and the phase are treated separately. For multiplication the magnitudes are multiplied and the phases are added:

$$\underline{Z} = \underline{Z}_1 \cdot \underline{Z}_2 = Z_1 \cdot Z_2 \cdot e^{j(\varphi_1 + \varphi_2)}$$

For division the magnitudes are divided and the phases are subtracted:

$$\underline{Z} = \frac{\underline{Z}_1}{\underline{Z}_2} = \frac{Z_1}{Z_2} \cdot e^{j(\varphi_1 - \varphi_2)}$$

A special case of division (and of importance for AC analysis) is the reciprocal value of a complex number $\underline{Z}$:

$$\frac{1}{\underline{Z}} = \frac{1}{Z} \cdot e^{-j\varphi}$$

Differentiation of a complex harmonic time function can be easily done in polar form. Consider ϕ being time dependent, e.g. ϕ=ωt:

$$\underline{Z} = Ze^{j\omega t}$$

Differentiation yields:

$$\frac{d\underline{Z}}{dt} = j\omega \cdot Ze^{j\omega t} = j\omega\underline{Z}$$

So the differentiation in polar form is just a multiplication with jω. In terms of the vector diagram this multiplication with jω corresponds to counterclockwise rotation of the vector. For every complex number $\underline{Z}$ there is a complex conjugate number $\underline{Z}^*$ that differs just by the sign of the imaginary part:

$$\underline{Z} = R + jX = Ze^{j\varphi}$$

$$\underline{Z}^* = R - jX = Ze^{-j\varphi}$$

Multiplication of a complex number with its complex conjugate gives the square of the magnitude:

$$\underline{Z} \cdot \underline{Z}^* = R^2 + X^2 = Z^2$$

The sum of a complex number and the difference of a complex number with its complex conjugate number yields:

$$\underline{Z} + \underline{Z}^* = 2R$$

$$\underline{Z} - \underline{Z}^* = 2jX$$

7.4.3 Application of complex numbers to AC circuits

Consider a sinusoidal voltage with peak value û, angular frequency ω and phase angle ϕ_u:

$$u(t) = \hat{u} \cdot \cos(\omega t + \varphi_u)$$

This voltage can be depicted as the real axis projection of a rotating vector of length û in a vector diagram. Considering this diagram to be a Gaussian coordinate system we can write the voltage in complex form as:

$$\underline{u}(t) = \hat{u} \cdot (\cos(\omega t + \varphi_u) + j\sin(\omega t + \varphi_u)) = \hat{u}e^{j(\omega t + \varphi_u)}$$

The momentary value of the complex voltage is given by the real part of the complex voltage:

$$u(t) = \mathrm{Re}(\underline{u}(t)) = \hat{u} \cdot \cos(\omega t + \varphi_u)$$

If we are not interested in the actual value of the current or voltage, but just in the relation between these values (phase difference) the time independent part of the complex quantity can be considered only:

$$\underline{u} = \hat{u}e^{j(\varphi_u)}$$

This is the phase vector or phasor representation of the voltage for any given time (e.g. t = 0 s). It is useful in particular in all kinds of calculations with a common angular frequency of all components as it separates the time-dependent term from the time-independent terms.

7.4.4 AC circuits

When using complex numbers to describe AC circuits of course our basic rules hold true:

- –All events happen at the same time independent of the location within the circuit;
- –Kirchhoff's and Ohm's laws are valid for all instances of

time;
–Superposition is still valid for linear elements like resistors, inductors, capacitors (these elements have linear dependencies (direct linear or derivative) between electrical propoerties like voltage and current);

Very basic circuits, circuits with just a sinusoidal source connected to one element, will be analyzed first. The analysis of these circuits demonstrates the application and benefits of notation with complex numbers. Afterwards more complex circuits will be studied.

AC circuit with a resistor

Given is a sinusoidal voltage (arbitrarily using $\phi_u = 0$)

$$\underline{u}(t) = \hat{u}e^{j(\omega t)}$$

Using Ohm's law yields for the complex current:

$$\underline{i}(t) = \frac{\underline{u}(t)}{R} = \frac{\hat{u}}{R}e^{j(\omega t)} = \hat{i}e^{j(\omega t)}$$

As already seen with the trigonometric and the vector approach the current and the voltage at the resistor are in phase and the peak value of the current is given by the peak value of the voltage divided by the resistance R.

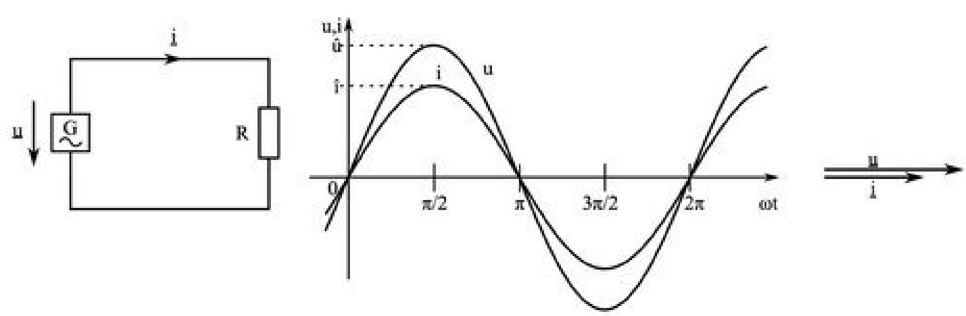


Fig. 7.40: A simple AC circuit with just a resistor (left), line diagram of current and voltage (center) and vector diagram (right).

AC circuit with an inductor

Given is a sinusoidal current (arbitrarily using $\phi_i = 0$)

$$\underline{i}(t) = \hat{i}e^{j(\omega t)}$$

Using the induction law yields for the complex voltage:

$$\underline{u}(t) = L \cdot \frac{d\underline{i}(t)}{dt} = j\omega L \cdot \hat{i}e^{j(\omega t)} = j\omega L \cdot \underline{i}(t)$$

As the time dependent term is the same for the voltage and the current we can rewrite this equation using the phasors of the voltage and the current:

$$\underline{u} = j\omega L \cdot \underline{i}$$

Thus the differentiation yields a multiplication with jω. Multiplication with jω corresponds to a counterclockwise rotation of 90 ° of the phasor. For the inductor the phasor of the voltage leads the current by 90 °. The vectors of voltage and current are not in phase but out of phase by 90 °. Formally this expression equals Ohm's law and the term $\underline{Z}_L = j\omega L$ is called the impedance of the inductor:

$$\frac{\underline{u}}{\underline{i}} = j\omega L = \underline{Z}_L$$

In general the ratio $\underline{u}/\underline{i}$ is called the impedance of an element. The unit for the impedance is Ohms (Ω) just as in case of the real resistance.
The reciprocal of the impedance is called the admittance $\underline{Y}$ and is given by the ratio of current phasor to the voltage phasor. The unit for the admittance is Siemens (S):

$$\underline{Y}_L = \frac{1}{\underline{Z}_L} = \frac{1}{j\omega L}$$

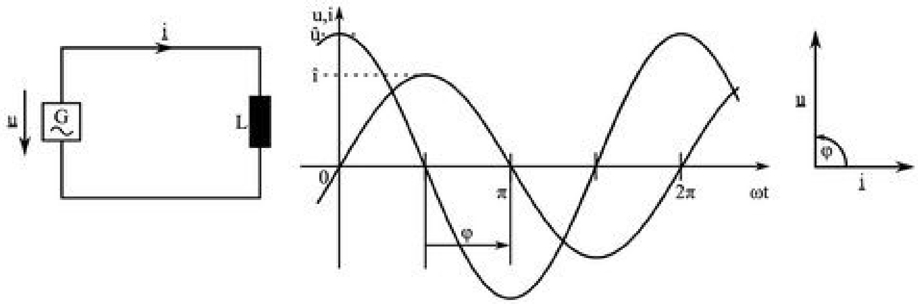


Fig. 7.41: A simple AC circuit with just an inductor (left), line diagram of current and voltage (center) and vector diagram (right).

AC circuit with a capacitor

Given is a sinusoidal voltage (arbitrarily using $\phi_u = 0$)

$$\underline{u}(t) = \hat{u}e^{j(\omega t)}$$

Using the capacitor's relation for current and voltage yields for the complex current:

$$\underline{i}(t) = C \cdot \frac{d\underline{u}(t)}{dt} = j\omega C \cdot \hat{u}e^{j(\omega t)}$$

In phasor notation it yields:

$$\underline{i} = j\omega C\underline{u}$$

The differentiation is again a multiplication with jω. Multiplication with jω corresponds to a counterclockwise rotation of 90 ° of the phasor. For the capacitor the phasor of the current leads the voltage by 90 °. The vectors of voltage and current are not in phase but out of phase by 90 °.
Formally this expression again equals Ohm's law and the factor $\underline{Z}_C$ = 1/jωC=-j/ωC is called the impedance of the capacitor. The admittance of a capacitor is:

$$\underline{Y}_C = \frac{1}{\underline{Z}_C} = j\omega C$$

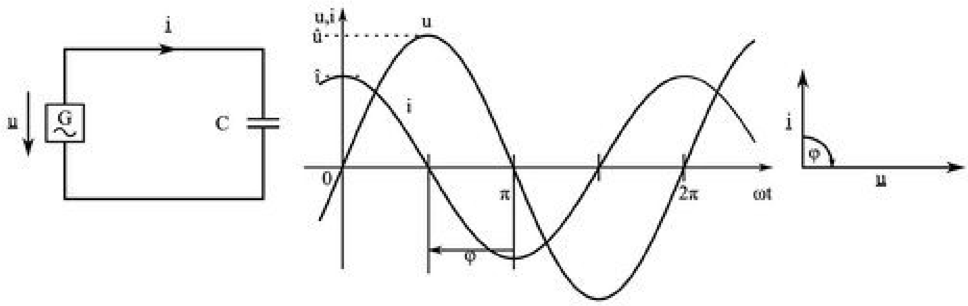


Fig. 7.42: A simple AC circuit with just a capacitor (left), a line diagram of current and voltage (center) and a vector diagram (right).

Summarizing the results of the simple R, L and C circuits we can write the voltage-current relations for these linear elements in phasor form as follows:

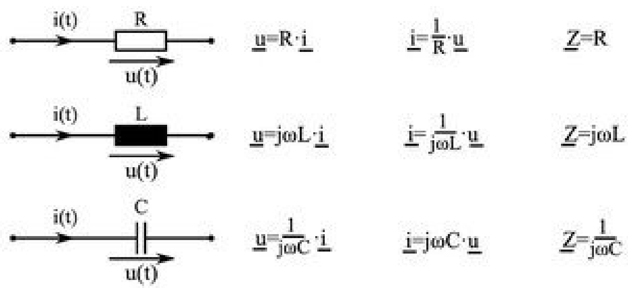


Fig. 7.43: Impedances for the basic elements resistor, inductor and capacitor in complex form.

Even though the use of complex number phasors seems complicated, this method is used to simplify the analysis of AC circuits. Fig. 7.44 shows the steps it takes to analyze an AC circuit, e.g. starting with a sinusoidal voltage. Without the use of complex numbers and phasors, differential equations have to be solved to calculate the corresponding currents. Depending on the complexity of the circuit this will be a very difficult task.

Using complex numbers and phasors transforms the differential equations into algebraic equations which are in general much easier to solve. After the current phasor is calculated it is transferred back to the time dependent form.

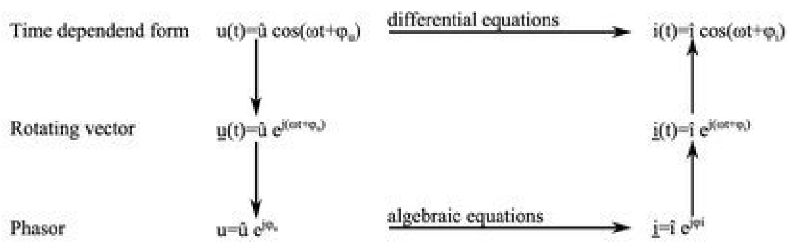


Fig. 7.44: Steps for analysis of an AC circuit using phasors.

7.4.5 Kirchhoff's laws for AC circuits

In general the behavior of a circuit is calculated using complex numbers and the real currents and voltages are determined by the resulting real part of the rotating vector associated with the phasor. The resulting impedance of a circuit can consist of a real and an imaginary part:

$$\underline{Z} = R + jX$$

The real part R is called resistance and the imaginary part X is called reactance. The impedance of a resistor has just a resistance and both an ideal capacitor and an ideal inductor just have a reactance.

As in the DC case, both Kirchhoff's current law and Kirchhoff's voltage law are still valid for AC circuits. In complex notation these two laws read like:

$$\sum_{mesh} \underline{u}_i = 0$$

$$\sum_{node} \underline{i}_i = 0$$

Consequently (as will be shown in the following section) also the rules for calculation of elements connected in parallel and in series are still valid.

Series connection of n elements:

$$\underline{Z} = \sum_{i=1}^{n} \underline{Z}_i$$

Parallel connection of n elements:

$$\underline{Y} = \sum_{i=1}^{n} \underline{Y}_i$$

Series connection of a resistor and an inductor

The analysis of circuits with single elements revealed that there is no phase shift between current and voltage in case of the resistor circuit and a phase shift of 90 ° with the voltage leading the current in the case of the inductor. The circuit with a series connection of a resistor and an inductor combines these two elements with different behavior. What about the current and the voltages?

Consider now Fig. 7.45 with a series connection of a resistor and an inductor. The voltage drop across the resistor and the inductor given in complex form are:

$$\underline{u}_R = R\underline{i}$$

$$\underline{u}_L = j\omega L\underline{i}$$

Applying KVL yields for this circuit:

$$\underline{u} = \underline{u}_R + \underline{u}_L = R\underline{i} + j\omega L\underline{i} = (R + j\omega L)\underline{i} = \underline{Z}\underline{i}$$

$$\underline{Z} = (R + j\omega L) = \underline{Z}_R + \underline{Z}_L$$

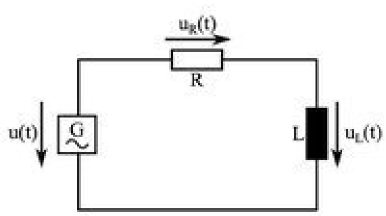


Fig. 7.45: An RL circuit with a sinusoidal voltage source.

The voltage of the voltage source is related to the current via the impedance of the circuit. As in case of series connection of resistors in DC circuits the total impedance of the circuit is the sum of the impedances of its elements. In polar form the impedance is:

$$\underline{Z} = \underline{Z}_R + \underline{Z}_L = Ze^{j\varphi}$$

The magnitude and the phase angle are given by:

$$Z = \sqrt{R^2 + (\omega L)^2}$$

$$\varphi = \arctan\left(\frac{\omega L}{R}\right)$$

Consider a source voltage:

$$u(t) = \hat{u} \cdot \cos(\omega t + \varphi_u)$$

In complex form this voltage is:

$$\underline{u} = \hat{u} e^{j(\omega t + \varphi_u)}$$

The resulting current yields:

$$\underline{i} = \frac{\underline{u}}{\underline{Z}} = \frac{\hat{u}}{Z} e^{j(\omega t + \varphi_u - \varphi)}$$

$$\Rightarrow i(t) = \mathrm{Re}(\underline{i}) = \frac{\hat{u}}{Z} \cdot \cos(\omega t + \varphi_u - \varphi)$$

The current lags behind the voltage by a phase shift of ϕ = arctan(ωL/R).
The voltage drop across the resistor is in phase with the current and yields:

$$\underline{u}_R = R\underline{i} = R\frac{\hat{u}}{Z} e^{j(\omega t + \varphi_u - \varphi)}$$

$$u_R(t) = \mathrm{Re}(\underline{u}_R) = R\frac{\hat{u}}{Z} \cdot \cos(\omega t + \varphi_u - \varphi)$$

The voltage drop across the inductor is given by:

$$\underline{u}_L = j\omega L\underline{i} = j\omega L\frac{\hat{u}}{Z}e^{j(\omega t+\varphi_u-\varphi)} = \omega L\frac{\hat{u}}{Z}e^{j\left(\omega t+\frac{\pi}{2}+\varphi_u-\varphi\right)}$$

$$u_L(t) = \mathrm{Re}(\underline{u}_L) = \omega L\frac{\hat{u}}{Z}\cdot\cos\left(\omega t+\frac{\pi}{2}+\varphi_u-\varphi\right)$$

As already known, the voltage at the inductor leads the current through the inductor by π/2 = 90 °.

Graphically this solution is depicted in Fig. 7.46. As there is just one current we use this current as a starting point for drawing the vector diagram. Current i is drawn horizontal. The voltage across the resistor u_R is in phase with the current and hence also horizontal, whereas the voltage across the inductor u_L leads the current by π/2 and points upwards. The total voltage u is the graphical sum of the two voltages represented by $\underline{Z}\cdot\underline{i}$. The angle ϕ is the phase shift of the voltage leading the current. As long as the voltage vector leads the current vector the behavior of a circuit is called inductive (if the current vector leads it is called capacitive).

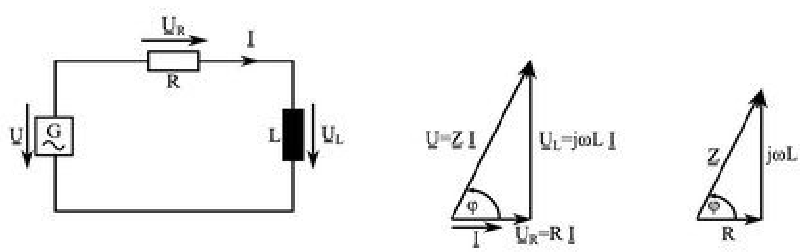


Fig. 7.46: Vector diagrams of the series connection of resistor and inductor.

Starting from the vector diagram of the series connection and dividing all terms by the current we get a similar triangle formed by the inductances of the elements and the total inductance. As we have seen before the phase angle is given by ϕ = arctan(ωL/R).

Series connection of a resistor and capacitor

The calculation for a series connection of a resistor and a capacitor is done in the same way as for the resistor-inductor series connection. The results are:

Impedance:

$$\underline{Z} = R - j\frac{1}{\omega C}$$

Magnitude of impedance:

$$Z = \sqrt{R^2 + \left(\frac{1}{\omega C}\right)^2}$$

Current:

$$\underline{i} = \frac{\underline{u}}{R - j\frac{1}{\omega C}}$$

Magnitude of the current:

$$I = \frac{U}{\sqrt{R^2 + \left(\frac{1}{\omega C}\right)^2}}$$

Magnitude of the total voltage:

$$U = \sqrt{U_R^2 + U_C^2}$$

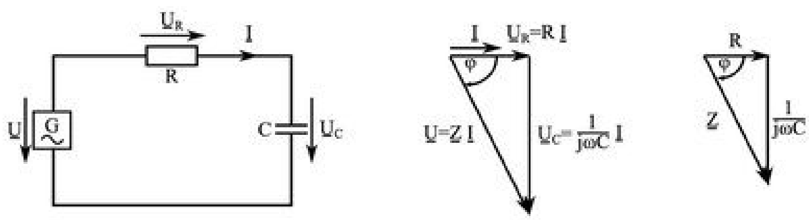


Fig. 7.47: Series connection of resistor and capacitor (left), vector diagram for voltages (center) and impedances (right).

Graphically this solution is depicted in Fig. 7.47. Again the current $\underline{i}$ is drawn horizontal. The voltage across the resistor u_R is in phase with the current and hence also horizontal, whereas the voltage across the inductor u_L lags the current by $\pi/2$ and points downwards. The total voltage u is the graphical sum of the two voltages represented by $\underline{Z}\cdot\underline{i}$. The angle ϕ is the phase shift of the voltage leading the current. Here the total current vector leads the total voltage and the circuit has a capacitive behavior.

Example of RC series circuit

A bulb should be operated at U_{bulb} = 230 V and I = 0.5 A using a voltage source of U = 300 V and f = 50 Hz. What is the value of the capacitance of a capacitor in series with the bulb to achieve the required operating conditions?

The bulb is a resistive element and the circuit with the series connection of R and C is shown in Fig. 7.48. The current in this circuit is the same for all elements. According to KVL the voltage of the source is the vectorial sum of the voltage across the resistor and the capacitor:

$$\underline{u} = \underline{u}_R + \underline{u}_C$$

The magnitudes of the voltage source and the bulb are given and the magnitude of the voltage of the capacitor is just:

$$U_C = \sqrt{U^2 - U_R^2} = 193V$$

Using this voltage of the capacitor the capacitance is given by the magnitude of the capacitor's impedance

$$Z_C = \frac{U_C}{I} = \frac{1}{\omega C}$$

$$\Rightarrow C = \frac{I}{\omega U_C} = 8.25\mu F$$

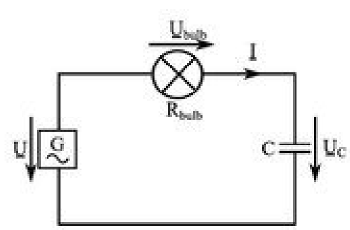


Fig. 7.48: A bulb operated in series with a capacitor.

Parallel connection of a resistor and a capacitor

After the study of series connection of two elements in AC circuits, the parallel connection of two elements is analyzed, here a resistor and a capacitor. The circuit is depicted in Fig. 7.49. The total current of the voltage source is split according to KCL:

$$\underline{i} = \underline{i}_R + \underline{i}_C$$

With

$$\underline{i}_R = \frac{\underline{u}}{R}$$

and

$$\underline{i}_C = j\omega C\underline{u}$$

KCL yields:

$$\underline{i} = \underline{i}_R + \underline{i}_C = \frac{\underline{u}}{R} + j\omega C\underline{u} = \underline{u} \cdot \left(\frac{1}{R} + j\omega C\right)$$

The factor (1/R+jωC) is the admittance of the total circuit and is, as expected, the sum of the admittances of the single elements:

$$\underline{Y} = \frac{1}{R} + j\omega C = \underline{Y}_R + \underline{Y}_C$$

The magnitude of the admittance is:

$$Y = \sqrt{\left(\frac{1}{R}\right)^2 + (\omega C)^2}$$

And the magnitude of the total current is:

$$I = U \cdot Y = U \cdot \sqrt{\left(\frac{1}{R}\right)^2 + (\omega C)^2}$$

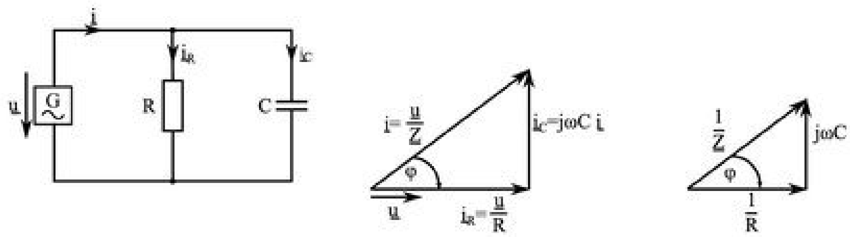


Fig. 7.49: Parallel connection of resistor and capacitor (left), vector diagram for currents (middle) and admittances (right).

Fig. 7.49 shows the vector diagram for the total voltage and the currents. Starting from the total voltage (in horizontal direction) the current through the resistor is in phase with the total voltage. The current through the capacitor leads the voltage by 90 ° and points upwards. The total current of the source is hence the geometrical sum of the two currents. Dividing the currents by the common total voltage transforms the triangle of the currents to the triangle of admittances.

Example of RLC circuit

A circuit with series and parallel connections is depicted in Fig. 7.50: A resistor R = 100 Ω is connected in series with an inductor of L = 10 mH and these two elements are connected in parallel to a sinusoidal voltage source (U = 50 V, f = 1000 Hz) and a capacitor with C = 10 µF.

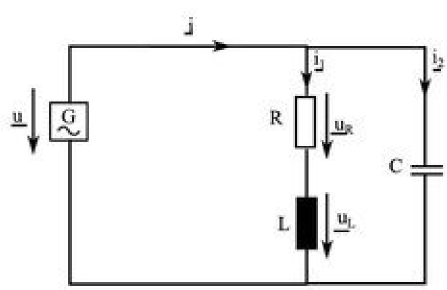


Fig. 7.50: RLC circuit with R and L in series.

To determine the total current $\underline{i}$ of the circuit we start with the RL series connection. The impedance of this connection is:

$$\underline{Z}_{RL} = \underline{Z}_R + \underline{Z}_L = (R + j\omega L) = 50\Omega + j62.8\Omega = 80.3\Omega \cdot e^{j51^\circ}$$

As the voltage drop across the series connection of R and L is equal to the voltage of the source ($\underline{u}$) the current i_1 through R and L yields:

$$\underline{i}_1 = \frac{\underline{u}}{\underline{Z}_{RL}} = \frac{50V}{80.3\Omega \cdot e^{j51°}} = 0.62A \cdot e^{-j51°} = 0.39A - j0.48A$$

The current through the capacitor is given by:

$$\underline{i}_2 = \frac{\underline{u}}{\underline{Z}_C} = \underline{u} \cdot j\omega C = 50V \cdot j \cdot 2 \cdot \pi \cdot 1000Hz \cdot 10\mu F = j3.14A = 3.14A \cdot e^{90°}$$

According to KCL the total current is:

$$\underline{i} = \underline{i}_1 + \underline{i}_2 = 0.39A - j0.48A + j3.14A = 0.39A + j2.66A = 2.68Ae^{82°}$$

The total circuit has a capacitive behavior as the current leads the voltage by 82 °. The total admittance of the circuit is:

$$\underline{Y}_{RLC} = \underline{Y}_{RL} + \underline{Y}_C = \frac{1}{\underline{Z}_{RL}} + j\omega C = 0.012S \cdot e^{-j51°} + j0.0628S$$
$$= 0.0078S - j0.0096S + j0.0628S = 0.0078S + j0.0532S = 0.0538S \cdot e^{j82°}$$

Automotive application

Parallel and series connections of resistors and capacitors are frequently used in automotive applications, e.g. for high-pass or low-pass filters. Another example is the use of a differential capacitor in a Wheatstone bridge. This capacitive bridge is used for example in micromechanical acceleration or angular rate sensors. The acceleration sensor makes use of the fact that the acceleration $\vec{a}$ is correlated to a force $\vec{F} = m \cdot \vec{a}$.

A differential capacitor is a series connection of two capacitors with one common electrode, such as depicted schematically in Fig. 7.51. It acts like a frequency dependent voltage divider. In sensors this common electrode with mass m is free to move if an external force due to the acceleration is exerted to it. By the movement of the common electrode the capacitances of the two capacitors change due to the changing distances Δd of the plates. One capacitance increases and the other decreases:

$$C_1 = \frac{\varepsilon \cdot A}{d + \Delta d}$$

$$C_2 = \frac{\varepsilon \cdot A}{d - \Delta d}$$

To measure these changes the differential capacitor is one leg of a Wheatstone bridge. The other leg is built out of two resistors. A sinusoidal voltage source excites the Wheatstone bridge with frequency ω. In this configuration the voltage difference $\underline{u}_a$ is a direct measure for the distortion of the differential capacitor and hence for the acceleration:

$$\underline{u}_a = \underline{u}_1 - \underline{u}_2 = \underline{u}_b \cdot \frac{R}{2R} - \underline{u}_b \cdot \frac{\frac{1}{j\omega C_2}}{\frac{1}{j\omega C_2} + \frac{1}{j\omega C_1}} = -\underline{u}_b \cdot \frac{\Delta d}{2d}$$

The output voltage is constant for constant acceleration and can be measured with an ADC.

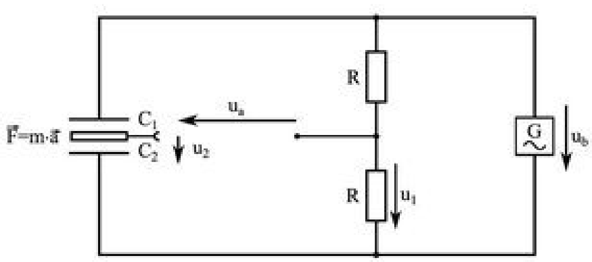


Fig. 7.51: The capacitive Wheatstone bridge of an acceleration sensor with a differential capacitor.

8 Building blocks

Just as any electric circuit with only two external terminals is called a two-terminal circuit, two port networks (or four-terminal networks) are circuits with two pairs of terminals such as shown in Fig. 8.1. In addition two port networks have to fulfill the port condition: current entering one terminal must be equal to the current flowing out of the other one of the same port. As for the two-terminal networks two-port networks can be active (containing sources) or passive (no sources inside). Furthermore, the two port network can be linear (containing just linear elements like resistors, capacitors, inductors) or non-linear (e.g. with diodes). The theory of two-port networks is not discussed here. Instead we make use of a special case of two-port networks: building blocks or system blocks with I_1 = 0 A and U_2 independent of I_2. This special case can be obtained by adding a unity gain buffer to the input and output ports respectively. Using this simplification with regard to two-port networks we are able to analyze building blocks of complex circuits and to determine the transfer characteristics of these blocks.

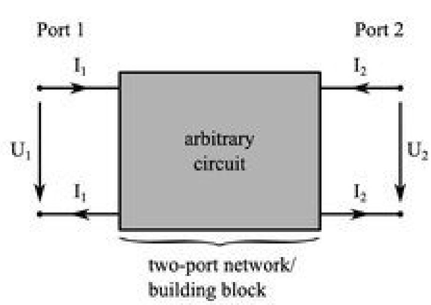


Fig. 8.1: An arbitrary two port network fulfilling the port conditions.

The ports connect to other circuits like in the case of the two-terminal circuits. The construct of building blocks is used to isolate parts of a larger circuit to simplify the analysis of the complete circuit. In this case often one port is the input port and the other one the output port, and the main property of the circuit is the transfer function: how does the output voltage depend on the input voltage and its frequency? If the transfer function is known, the two port network can be treated as a black box with the internal structure and components being of no further interest. The building blocks are often used for analysis of filters or transmission lines.

8.1 High-pass filter

A simple building block of just a resistor R and an inductor L is depicted in Fig. 8.2. It resembles the series connection of R and L treated above. The left port is the input port with an input voltage $\underline{u}_1(t)$ whereas the port on the right side is the output port with output voltage $\underline{u}_2(t)$.

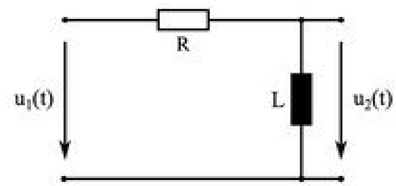


Fig. 8.2: -Two port network consisting of R and L; input voltage is $\underline{u}_1(t)$, voltage across the inductor is the output voltage $\underline{u}_2(t)$.

What about the transfer function for this building block? The circuit is a frequency dependent voltage divider and the ratio of the voltages is:

$$\frac{\underline{u}_2}{\underline{u}_1} = \frac{\underline{X}_L}{\underline{Z}_{total}} = \frac{j\omega L}{R + j\omega L} = \frac{\omega L}{\sqrt{R^2 + (\omega L)^2}} e^{j\left(\frac{\pi}{2} - \arctan\left(\frac{\omega L}{R}\right)\right)}$$

The analysis of this ratio can be split into two parts, the ratio of the magnitude and the phase difference between input and output voltage. The transfer function of the magnitude is also referred to as voltage gain.

Magnitude

$$\left|\frac{\underline{u}_2}{\underline{u}_1}\right| = \frac{\omega L}{\sqrt{R^2 + (\omega L)^2}} = \frac{\frac{\omega L}{R}}{\sqrt{1 + \left(\frac{\omega L}{R}\right)^2}}$$

The magnitude of the output voltage depends strongly on the angular frequency ω of the voltages. For $\omega \rightarrow 0\ s^{-1}$ the magnitude of $\underline{u}_2(t)$ tends to zero. In the DC case (in the limit of $\omega = 0\ s^{-1}$) the output voltage will be totally damped down to 0 V. This is exactly the behavior of an inductor that is expected in case of a DC circuit: the inductor acts as a short circuit and there is no voltage drop across the inductor. In this case the full voltage drop of $|\underline{u}_1|$ will be at the resistor R.

For $\omega \rightarrow \infty\ s^{-1}$ the term ω·L dominates the denominator and the ratio of the voltages tends to

1. In this case the output voltage will be (nearly) undamped and will have the same magnitude as the input voltage. The voltage at the resistor will tend to zero.

For frequencies between these two limits the magnitude of the voltage ratio is a steadily increasing function as depicted in Fig. 8.3 on the left side. How fast the function increases depends on the ratio of $\Omega = \omega \cdot L / R$. Therefore it is very common to rescale the graph logarithmically using this ratio Ω instead of the angular frequency (see Fig. 8.3).

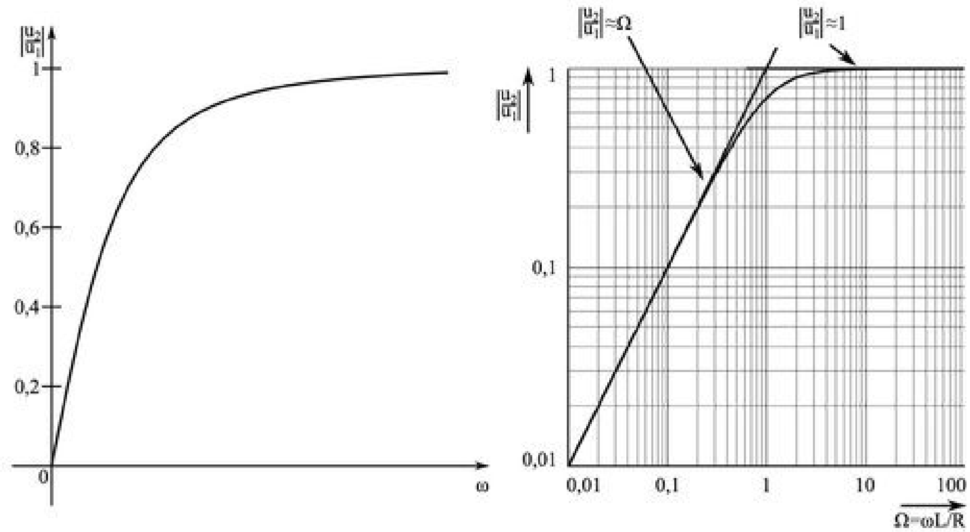


Fig. 8.3: The transfer function of the voltage; left: scaling using angular frequency; right: logarithmical scaling using Ω=ωL/R; straight lines show linear approximations for high and low frequencies.

Using the new scaling the functionality of the analyzed circuit becomes obvious: it is a high-pass filter. High frequencies well above $\Omega = 1$ can pass the circuit with very low damping. For this frequency range the voltage gain can be approximated by a straight line $|\underline{u}_1/\underline{u}_2| \approx 1$. Low frequencies well below $\Omega = 1$ cannot pass the circuit, they are strongly damped. Here the transfer function can be approximated by another straight line, $|\underline{u}_1/\underline{u}_2| \approx \Omega$. These two lines (approximations of the voltage gain for high and low frequencies) intersect at $\Omega = 1$ and the voltage gain at this point is:

$$\left|\frac{\underline{u}_2}{\underline{u}_1}\right|_{\Omega=1} = \frac{\Omega}{\sqrt{1+\Omega^2}} = \frac{1}{\sqrt{2}}$$

The corresponding angular frequency is called the cut-off frequency ω_0 and is given for the RL high-pass by:

$$\omega_0 = \frac{R}{L}$$

Frequencies above the cut-off frequency pass the circuit nearly undamped, and frequencies below the cut-off frequency are strongly damped and can hardly pass the circuit. In this blocking region the reduction of the frequency by a factor of 10 also reduces the voltage gain by a factor of 10.

Phase difference

Besides the voltage gain the phase difference also shows a characteristic behavior given by the exponential part of the transfer function:

$$\frac{\underline{u}_2}{\underline{u}_1} = \frac{\underline{X}_L}{\underline{Z}_{total}} = \frac{j\omega L}{R + j\omega L} = \frac{\omega L}{\sqrt{R^2 + (\omega L)^2}} e^{j\left(\frac{\pi}{2} - \arctan\left(\frac{\omega L}{R}\right)\right)}$$

In the limit of $\omega \to 0\ s^{-1}$ the arctan term tends to zero and the phase difference between output and input voltage is 90 °. For $\omega \to \infty\ s^{-1}$ the arctan term tends to Π/2 and the output voltage is in phase with the input voltage. In between the phase difference steadily decreases as shown on the left side of Fig. 8.4.

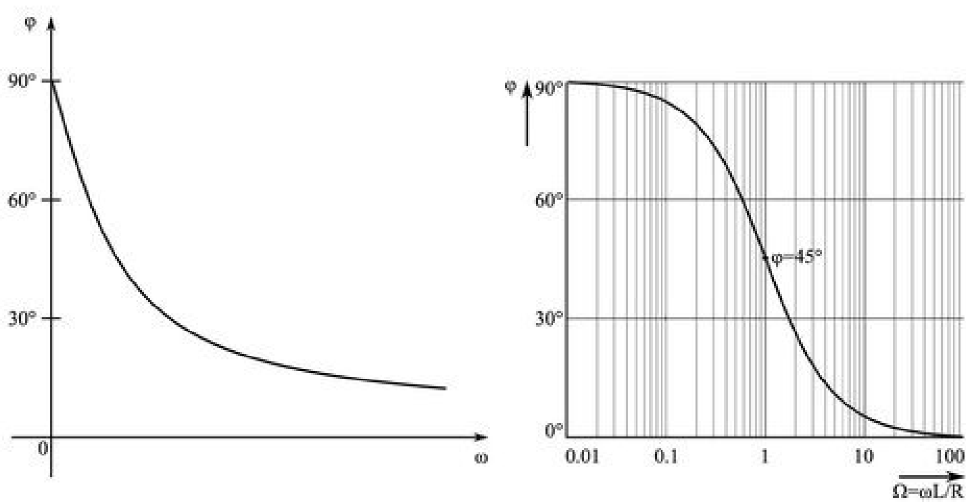


Fig. 8.4: The phase difference between output and input voltage; left: scaling using angular frequency; right: logarithmical scaling using $\Omega = \omega L/R$.

On the right side of Fig. 8.4 the scaling $\Omega = \omega L/R$ is used again and the characteristic behavior of the phase difference is clearly visible. At the cut-off frequency $\omega_0 = R/L$ the phase difference is 45 °.

8.2 Bode plot

Combination of the two logarithmic diagrams for the magnitude and the phase difference results in the so called Bode plot. In addition the gain of the magnitude's diagram is expressed in decibels:

$$gain = 20\log_{10}\left|\frac{\underline{u}_2}{\underline{u}_1}\right| dB$$

The Bode plot of a high-pass filter is shown in Fig. 8.5. For high frequencies the gain is approximately constant and equal to 0 dB. At the cut-off frequency ω_0 the gain drops down to –3 dB with regard to the high frequency limit. For low frequencies ($\omega < \omega_0$) it can be approximated by:

$$gain = 20\log_{10}\frac{\omega L}{R} dB = 20\log_{10}\frac{\omega}{\omega_0} dB$$

The gain is negative and strongly frequency dependent. As can be seen in the formula above and Fig. 8.5 the slope of the straight line at low frequencies is –20 dB/decade, where a decade denotes a change in frequency by a factor of 10. So using Bode plots the properties of the transfer functions for magnitude and phase difference can be easily seen. In addition Bode plots of complex networks can be constructed by the addition of simpler Bode plots.

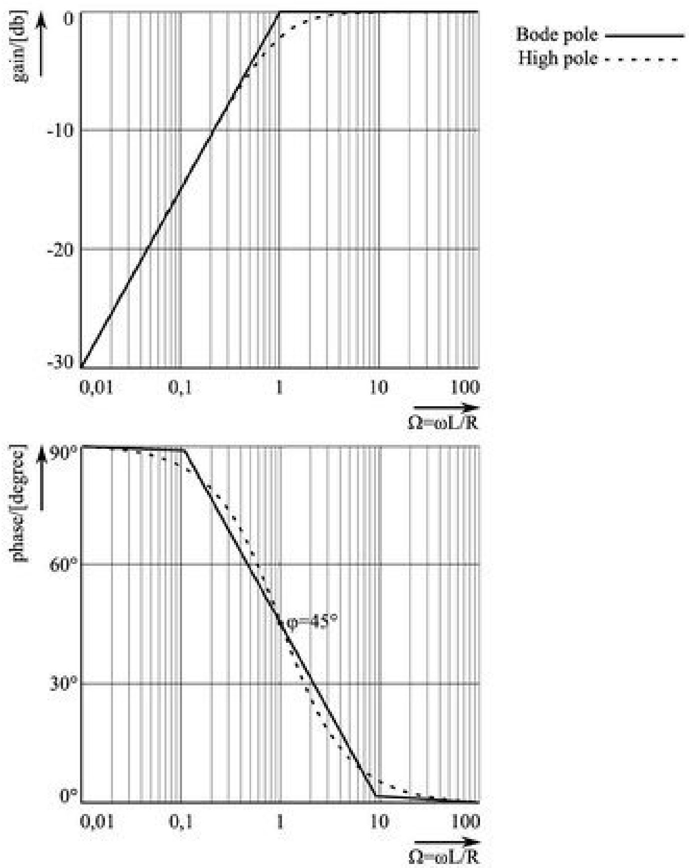


Fig. 8.5: A Bode plot of a high-pass filter.

8.3 Low-pass filter

The complementary building block to a high-pass filter is of course the low pass filter. Here low frequencies can pass the circuit and high frequencies are damped and filtered out. A simple low-pass filter can be designed by just changing the inductor of the previous high-pass filter to a capacitor as shown in Fig. 8.6.

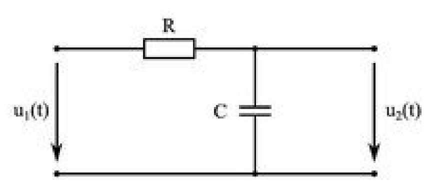


Fig. 8.6: An RC low-pass filter.

The transfer function of this RC low-pass filter is:

$$\frac{\underline{u}_2}{\underline{u}_1} = \frac{\underline{X}_C}{\underline{Z}_{total}} = \frac{\frac{1}{j\omega C}}{R + \frac{1}{j\omega C}} = \frac{1}{j\omega RC + 1} = \frac{1}{\sqrt{1 + (\omega RC)^2}} e^{j \arctan(\omega RC)}$$

The corresponding Bode diagram is depicted in Fig. 8.7. It clearly shows the filter functionality of this building block: low frequencies can pass the circuit, the gain is 0 dB. The cut-off frequency ω_0 is given for the RC low-pass filter by:

$$\omega_0 = \frac{1}{RC}$$

For the cut-off frequency the attenuation is again –3 dB and the phase shift between output and input voltage is –45 °. For higher frequencies the gain drops by –20 dB per decade:

$$gain = 20 \log_{10} \frac{1}{\sqrt{1 + (\omega RC)^2}} dB = -10 \log_{10} \left(1 + (\omega RC)^2\right) dB \approx -20 \log_{10} (\omega RC)$$

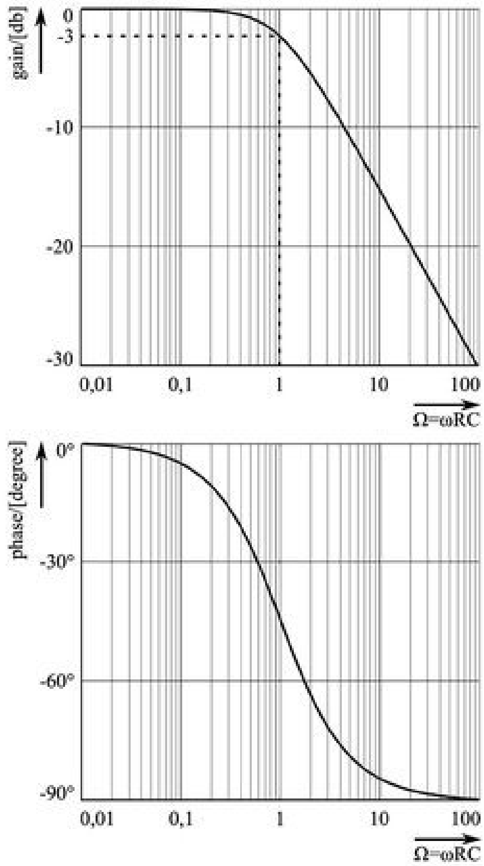


Fig. 8.7: A Bode diagram of a RC low-pass.

Automotive application

All kind of filters are commonly used in automotive applications. One example is the usage of a low-pass filter as an anti-aliasing filter. Consider an electronic sensor system as depicted in Fig. 8.8. A simple sensor like a temperature sensor is connected to an analog input pin of a microcontroller. The output of the sensor is an analog signal in the range of 0–5 V. An analog-to-digital converter (ADC) inside the microcontroller converts the analog sensor signal to a digital representation that can be used by the digital logic of the microcontroller. This conversion takes some time and therefore the sampling of the analog signal is at discrete time steps (e.g. every 10 µs, so sample rate is 100 kHz). To be able to recover the signal after the conversion correctly without any aliasing the Shannon-Nyquist criterion has to be fulfilled: the sample rate has to be at least twice the value of the highest frequency of the signal to be sampled. Therefore the high frequencies of the analog sensor signal have to be filtered out by using a low-pass filter like the RC low-pass filter. The cut-off frequency of this filter has to be adjusted to fit to the sampling rate of the ADC.

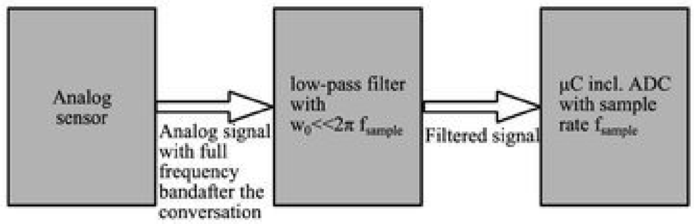


Fig. 8.8: A sensor system with microcontroller and low-pass (anti-aliasing) filter for the analog sensor signal.

8.4 Higher order filters

So far filters with just one energy storing element, an inductor or capacitor, have been used to introduce the basic concept of two-port network analysis and filters. These filters are called first order filters. First order filters have an attenuation far above (or below) the cut-off frequency of –20 dB per decade. To increase this attenuation, higher order filters can be used. The order of the filter corresponds to the number of energy storing elements, hence a filter of n^{th} order contains n energy storing elements. Furthermore the damping is increased by the order according to n time –20 dB per decade. A low-pass filter of 4^{th} order filters out frequencies well above the cut-off frequency with a damping factor of –80 dB per decade. One possible way to construct a higher order filter is to concatenate lower order filters (which is nothing else than connecting building blocks with known transfer function).
However, connecting any element to a building block, in particular to the output, may have a feedback to its behavior! So care has to be taken to avoid any feedback from one to the other network if building blocks are concatenated. This kind of feedback has already been discussed in terms of the unity gain buffer. The example of the two building blocks connected to

each other showed that the load circuit (the second building block) has an influence on the behavior of the source circuit (the first building block).

Fig. 8.9 shows an example of concatenated building blocks on the left side. The total network consists of two pure resistive networks each with resistor R. Separate analysis of these two network yields a transfer function of 1 for both. In case of the first one there is no current flow at any terminal, for the second a current enters and leaves via the input port and flows through the resistor. Any input voltage at the input of each network passes without change to the output. But after concatenation the behavior changes: the total transfer function is not again equal to 1 but equal to ½. The concatenated building block is nothing other than a voltage divider and the second network has an influence on the first one. As soon as the second network is connected to the first one a current also flows through the first resistor.

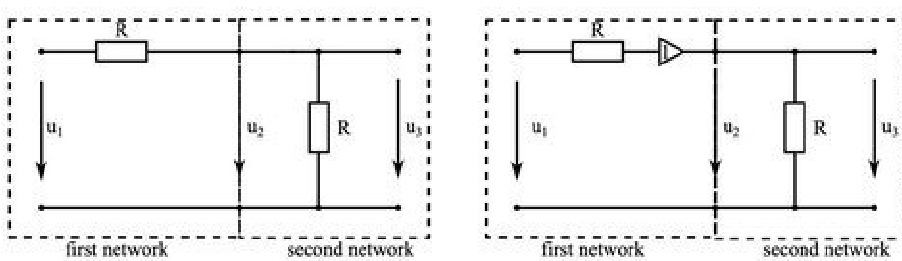


Fig. 8.9: Concatenation of two building blocks: without output termination (left), with an unity gain buffer at the output (right).

To avoid this kind of feedback the input impedance of the second building block has to be much higher than the output impedance of the first one. This can be done by terminating the output of each building block with a unity gain buffer as depicted on the right side of Fig. 8.9. The unity gain buffer makes the output impedance of the first circuit very small and its output voltage corresponds to the value given by the transfer function of the first network. The input impedance of the second network is much higher than the output impedance of the first network and there is no feedback. The transfer function of the total network is equal to 1 (product of the transfer functions of

the single networks).
If the output ports are terminated in a proper way, a higher order filter can be obtained by concatenation. Fig. 8.10 shows how a 2nd order low-pass filter is obtained by concatenation of two 1st order low-pass filters with the same values of R and C. The output of the first filter is the input of the second filter. The transfer function of the total 2nd order filter is the product of the transfer functions of the single filters:

$$\frac{\underline{u}_3}{\underline{u}_1} = \frac{\underline{u}_3}{\underline{u}_2} \cdot \frac{\underline{u}_2}{\underline{u}_1} = \frac{1}{1+(\omega RC)^2} e^{-j2\arctan(\omega RC)}$$

The cut-off frequency of the 2nd order filter, given by the frequency with a damping of –3 dB is:

$$\omega_0 = \frac{\sqrt{\sqrt{2}-1}}{RC}$$

Well above the cut-off frequency the damping is –40 dB per decade:

$$gain = 20\log_{10}\frac{1}{1+(\omega RC)^2} dB = -20\log_{10}\left(1+(\omega RC)^2\right) dB \approx -40\log_{10}(\omega RC)$$

Compared to the used 1st order filter the cut-off frequency is shifted to a lower frequency and the damping is doubled.
As a general rule Bode diagrams of higher order filters (and in more general of all complex two port networks), if constructed of lower order filters, can be obtained by the simple addition of the Bode plots of the lower order filters.

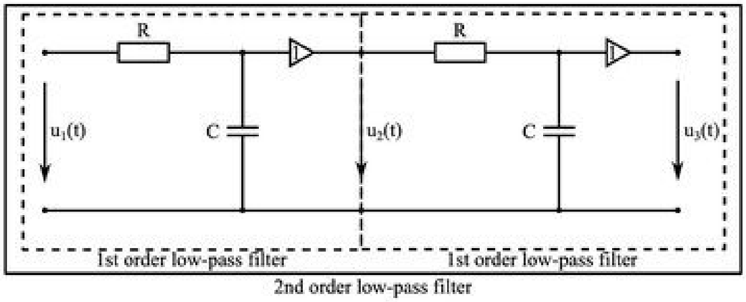


Fig. 8.10: A 2nd order low-pass filter, constructed by concatenation of two 1st order low-pass filters.

Also more complex functionalities can be realized by concatenation of simpler elements. Consider the circuit given in Fig. 8.11. It consists of a high-pass filter followed by a low-pass filter, both of 1st order. High frequencies are filtered out by the first element with cut-off frequency ω_1, low frequencies by the second element with a different cut-off frequency ω_2. In total frequencies lower than ω_2 and higher than ω_1 are filtered out. Just a frequency band between the two cut-off frequencies can pass the circuit. This kind of circuit is consequently called a band-pass circuit.
Again, the total transfer function is just the product of the single transfer functions (if output termination is done properly, e.g. by unity gain buffers). The transfer function is already well known for the low-pass filter, for the RC high-pass filter it is:

$$\frac{\underline{u}_2}{\underline{u}_1} = \frac{R_1}{\underline{Z}_{total}} = \frac{R_1}{R_1 + \frac{1}{j\omega C_1}} = \frac{1}{1 - \frac{j}{\omega R_1 C_1}} = \frac{1}{\sqrt{1 + \left(\frac{1}{\omega R_1 C_1}\right)^2}} e^{j \arctan\left(\frac{1}{\omega R_1 C_1}\right)}$$

Hence the total transfer function is given by:

$$\frac{\underline{u}_3}{\underline{u}_1} = \frac{1}{\sqrt{1 + \left(\frac{1}{\omega R_1 C_1}\right)^2}} \cdot \frac{1}{\sqrt{1 + (\omega R_2 C_2)^2}} e^{j\left(\arctan\left(\frac{1}{\omega R_1 C_1}\right) - \arctan(\omega R_2 C_2)\right)}$$

The Bode plot of this band-pass filter is shown on the bottom of

Fig. 8.11. At low frequencies the first term of the voltage gain tends to zero, at high frequencies the second term. At a frequency ω_r between ω_1 and ω_2 it has a maximum. This frequency is called resonant frequency and is given by the mean value of the cut-off frequencies.
Depending on the values of the resistors and capacitors the cut-off frequencies and the resonant frequency can be calculated:

$$\omega_1 = \frac{1}{R_1 C_1}$$

$$\omega_2 = \frac{1}{R_2 C_2}$$

$$\omega_r = \sqrt{\omega_1 \cdot \omega_2} = \sqrt{\frac{1}{R_1 C_1 \cdot R_2 C_2}}$$

The phase difference is 90 ° for $\omega \to 0\ s^{-1}$ and –90 ° for $\omega \to \infty\ s^{-1}$. At the resonant frequency it is 0 ° and output and input signal are in phase:

$$e^{j\left(\arctan\left(\frac{1}{\omega R_1 C_1}\right)-\arctan(\omega R_2 C_2)\right)} = e^0 = 1$$

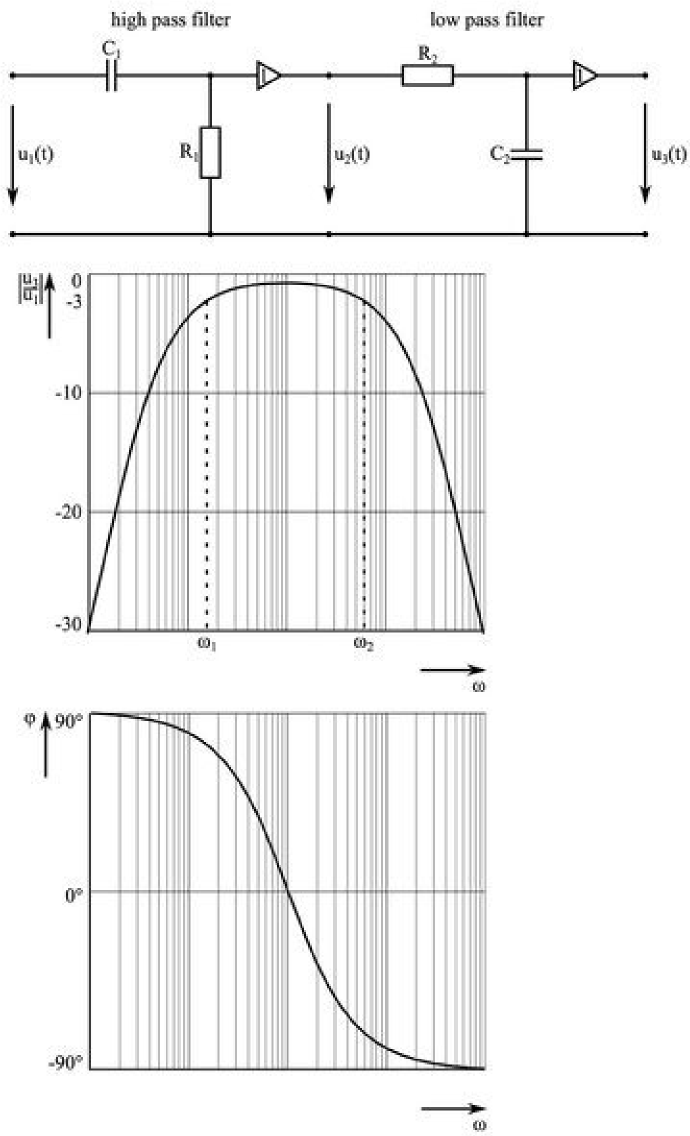


Fig. 8.11: A 2nd order band-pass filter, constructed by concatenation of a 1st order high-pass and low-pass filter (top); Bode plot of the band-pass filter (bottom).

8.5 Active filter

So far the filters have consisted of passive elements like capacitors and resistors. To avoid any feedback from a load connected to a passive filter an active element, the unity gain buffer, was added to the output terminal. An active filter now uses active elements like OpAmps to realize the required functionality and the output termination. Fig. 8.12 shows an active first order low-pass filter using an OpAmp in inverting amplifier configuration. The transfer function for this active filter yields (see transfer function of the inverting amplifier, feedback resistor replaced by parallel connection of R_2 and C_1):

$$\frac{\underline{u}_2}{\underline{u}_1} = -\frac{R_2}{R_1} \cdot \frac{1}{1 + jR_2C_1}$$

The first factor of the transfer function ($-R_2/R_1$) corresponds to the inverted amplification of the inverting amplifier, the second part is similar to the transfer function of the passive low-pass filter discussed above. In total this active filter combines the low-pass filter functionality with an amplification of the output voltage.

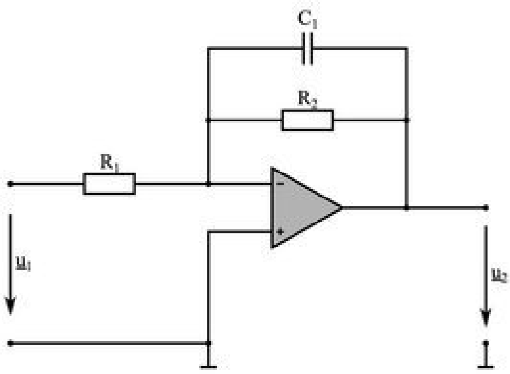


Fig. 8.12: An active high-pass filter.

9 AC power

Consider an arbitrary linear two terminal network, consisting of resistors, capacitor and inductors as depicted in Fig. 9.1.

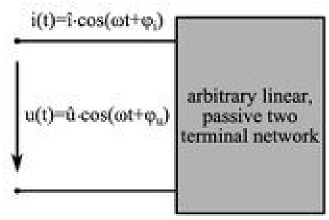


Fig. 9.1: Current and voltage of an arbitrary linear two terminal network.

The network has an internal impedance Z and the voltage u(t) and the current i(t) at the terminals. The instantaneous power inside the network is:

$$p(t) = u(t) \cdot i(t) = \hat{u} \cdot \hat{\imath} \cdot \cos(\omega t + \varphi_u) \cdot \cos(\omega t + \varphi_i)$$

Depending on the internal composition of the two terminal network the voltage $u(t) = \hat{u} \cdot \cos(\omega t+\phi_u)$ and the current $i(t) = \hat{\imath} \cdot \cos(\omega t + \phi_i)$ are in general not in phase Hence the instantaneous power can be positive or negative. In case of $p(t) > 0$ W power is consumed by the network, if $p(t) < 0$ W power is generated by the network.
Before the analysis of the AC power of an arbitrary linear passive network, two limit cases will be studied: a pure resistive and a pure inductive network.

9.1 AC power of a pure resistive two terminal network

For a pure resistive network voltage and current are always in phase, i.e. $\phi_i = \phi_u$.Using the trigonometric relation yields:

$$\cos(x)^2 = \frac{1}{2}\cdot(1+\cos(2x))$$

Finally the resulting instantaneous power is:

$$p(t) = \hat{u}\cdot\hat{\imath}\cdot(\cos(\omega t+\varphi_u))^2 = \frac{\hat{u}\cdot\hat{\imath}}{2}\cdot(1+\cos(2\omega t+2\varphi_u)) = U\cdot I\cdot(1+\cos(2\omega t+2\varphi_u))$$

The instantaneous power oscillates with the double frequency of the voltage and current (refer to Fig. 9.2) around a finite value with a peak power of $p(t)_{peak} = \hat{u}\cdot\hat{\imath}$. As current and voltage are in phase the power is always positive and the network consumes power at any instance of time.

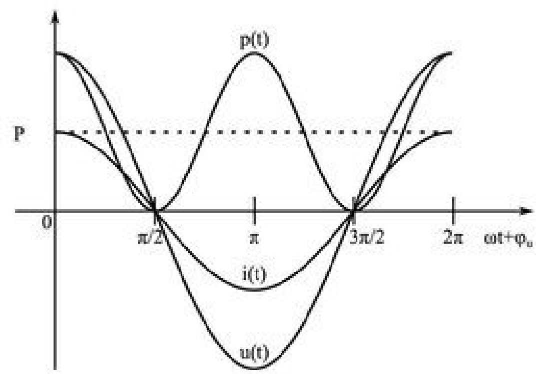


Fig. 9.2: AC power of a resistive network with voltage $\hat{u}\cdot\cos(\omega t+\phi_U)$ and current $i(t)=\hat{\imath}\cdot\cos(\omega t+\phi_i)$.

What about the average power? Due to the sinusoidal shape, both average voltage and average current are zero. But for the power the average value is:

$$\overline{p} = P = \frac{1}{T}\int_0^T p(t)dt = U\cdot I$$

The average power is just the product of the effective values of current and voltage:

If an arbitrary time-dependent current (voltage) dissipates the same power within a resistor as a DC current (voltage), then the RMS of the time-dependent current (voltage) is the same as the DC current (voltage).

This power is called active power (or effective or real power) as it describes the power that is transferred in one direction, here into the network, and that can be used inside the network.
The unit for the real power is, as usual for power, the Watt (W).

9.2 AC power of a pure inductive two terminal network

If the internal of the network is pure inductive it is the voltage that leads the current by 90 °:

$$\varphi_i = \varphi_u - 90°$$

Therefore the instantaneous power inside the network is:

$$p(t) = \hat{u} \cdot \hat{i} \cdot \cos(\omega t + \varphi_u) \cdot \cos(\omega t + \varphi_u - 90°) =$$
$$\hat{u} \cdot \hat{i} \cdot \cos(\omega t + \varphi_u) \cdot \sin(\omega t + \varphi_u) = U \cdot I \cdot \sin(2\omega t + 2\varphi_u)$$

Again the instantaneous power oscillates at double the frequency of the voltage and current, but this time around zero as can be seen in Fig. 9.3. For the first and third quarter of the period of the voltage, the power is positive and hence power is consumed by the network. The corresponding energy is stored within the inductor. In the second and fourth quarter, the power is negative. Power is generated by the network and the energy stored in the inductor declines to zero again. 1

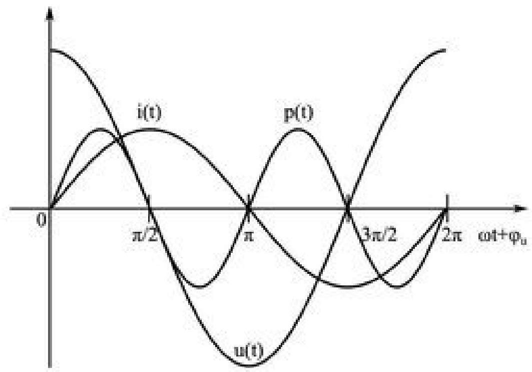


Fig. 9.3: AC power of a pure inductive network.

As the oscillation is around zero this time the average value of the power is zero:

$$\overline{p} = \frac{1}{T}\int_0^T p(t)dt = 0$$

For a pure inductive load no power is transferred to the network on average and the supply circuit (the circuit connected to the inductive two port network) does not have to provide any power to the network – on average. The amplitude of this oscillating power that is related to the temporary storage of energy within the inductor (and that has an average value of zero) is called reactive power as it is not associated with a permanent power transfer to the network.
Instead the power oscillates to and fro: for half of the time power is transferred to the network and the energy is stored within the inductor. For the other half it is transferred back from the network and the inductor is discharged again. So the source circuit has to provide power for half of the time and gets back power the other half. The peak power it has to provide and readopt is the product of the effective voltage and current, the reactive power Q:

$$Q = p(t)_{peak} = \frac{\hat{u} \cdot \hat{i}}{2} = U \cdot I$$

For a pure capacitive network the situation is similar to the inductive network: current and voltage are out of phase, this time by –90 °. Energy is temporarily stored in the capacitor, the power oscillates with double frequency around zero and the reactive power Q is the product of the effective voltage and current.

So in both cases, pure inductive and pure capacitive, no work at all can be done by the two port network as no energy is transferred to it on average. But the supply network has to provide power and hence current for half of the time (and to readopt the same amount of energy the other time). If the supply network has resistive elements, power will be dissipated and therewith wasted which is highly unwanted. Fig. 9.4 shows a simple example: a sinusoidal voltage source with an internal resistance is connected to a pure capacitive network. The average power is zero, but at the internal resistance power is dissipated (converted into heat) due to the reactive power and the associated the oscillating current flow.

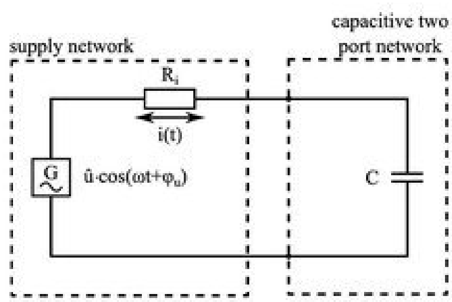


Fig. 9.4: A voltage source with internal resistance connected to a pure capacitive network.

The unit for the reactive power is the var (volt ampere reactive), unlike the Watt for the active power.

9.3 AC power of a mixed two terminal networkwith L, R and C

Based on the idealized configurations above, mixed networks containing capacitive, inductive and resistive elements can be analyzed. As already seen in the analysis of AC circuits, mixed networks will have a phase difference between voltage and current that can be any value between -90 ° and 90 °:

$$-90° \le \varphi_u - \varphi_i \le 90°$$

Using trigonometric functions the instantaneous power can be transformed:

$$\cos(a) \cdot \cos(b) = \frac{1}{2}\left(\cos(a-b) + \cos(a+b)\right)$$

$$\cos(a+b) = \cos(a) \cdot \cos(b) - \sin(a) \cdot \sin(b)$$

$$p(t) = \hat{u} \cdot \hat{i} \cdot \cos(\omega t + \varphi_u) \cdot \cos(\omega t + \varphi_i) = \frac{\hat{u} \cdot \hat{i}}{2} \cdot \left(\cos(\varphi_u - \varphi_i) + \cos(2\omega t + \varphi_i + \varphi_u)\right)$$
$$= U \cdot I \cdot \left(\cos(\varphi_u - \varphi_i) + \cos(2\omega t + 2\varphi_i + \varphi_u - \varphi_i)\right)$$
$$= U \cdot I \cdot \left(\cos(\varphi_u - \varphi_i) + \cos(\varphi_u - \varphi_i) \cdot \cos(2\omega t + 2\varphi_i) - \sin(\varphi_u - \varphi_i) \cdot \sin(2\omega t + 2\varphi_i)\right)$$

The final result for the instantaneous power is:

$$p(t) = U \cdot I \cdot \cos(\varphi_u - \varphi_i) \cdot \left(1 + \cos(2\omega t + 2\varphi_i)\right) - U \cdot I \cdot \sin(\varphi_u - \varphi_i) \cdot \sin(2\omega t + 2\varphi_i)$$

An example for the instantaneous power of a mixed network is depicted in Fig. 9.5. The two terms of the instantaneous power of a mixed two terminal network resemble the results of the pure resistive and inductive network.
The first term with a non-zero average corresponds to the active power of the pure resistive network multiplied by the so called power factor $\cos(\phi_u - \phi_i)$. The power factor has a value between 0 and 1. If the power factor is 1, the voltage and current are in phase and the total power U · I is transferred from the source to the network. If the power factor is smaller than one, less power is transferred. The active power $P = U \cdot I \cdot \cos(\phi_u - \phi_i)$ is always positive (as cos(x) = cos(–x)), no matter whether current or voltage is leading (capacitive or inductive behavior).
The second term with a zero average corresponds to the reactive power of the pure inductive (or capacitive) network multiplied by $\sin(\phi_u - \phi_i)$. The reactive power $Q = U \cdot I \cdot \sin(\phi_u -$

ϕ_i) can be positive (inductive network) or negative (capacitive network).

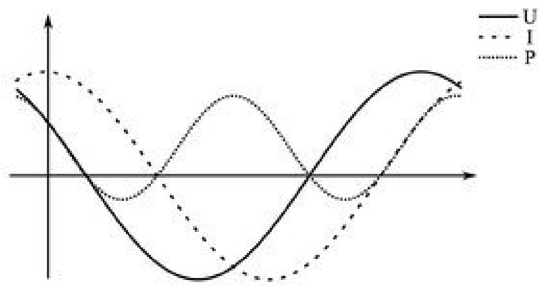


Fig. 9.5: AC power of a mixed resistive-capacitive network.

Reactive and active power have a phase shift of 90 ° and the vector sum of both power components results in the so called apparent power S. This yields for the magnitude of the apparent power:

$$S = \sqrt{P^2 + Q^2} = U \cdot I$$

Even though just it's just the active power that can be used to do any work within the network all elements of the network and the supply circuit has to be able to cope with the apparent power, e.g. the wires, generators, etc.
The common unit for apparent power is the VA (volt ampere).
A simple visualization of the AC power uses sinusoidal notation. Of course AC power can also be described in complex notation using complex voltage and current, e.g.:

$$\underline{u} = \hat{u}e^{j\varphi_u}$$

Using complex notation yields an apparent power of:

$$\underline{S} = \frac{1}{2} \cdot \underline{u} \cdot \underline{i}^* = P + jQ$$

Consequently active and reactive power are just the real and imaginary part of the complex apparent power respectively:

$$P = \mathrm{Re}(\underline{S})$$

$$Q = \mathrm{Im}(\underline{S})$$

As any two port network can be described by its impedance $\underline{Z}$ and admittance $\underline{Y}$ respectively the apparent power can also be written as:

$$\underline{S} = \frac{1}{2} \cdot \underline{u} \cdot \underline{i}^* = \frac{1}{2} \underline{i} \cdot \underline{Z} \cdot \underline{i}^* = \frac{1}{2} \hat{i}^2 \cdot \underline{Z}$$

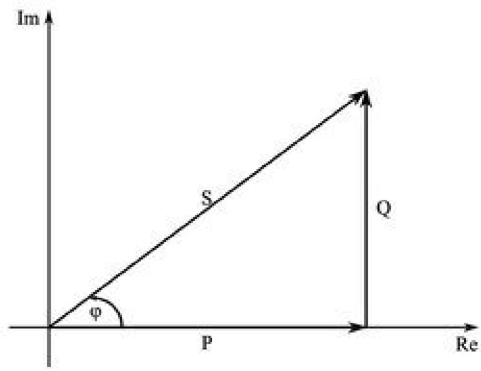


Fig. 9.6: An AC power diagram with active (P), reactive (Q) and apparent power (S).

Using apparent power the power factor can be defined as the ratio of active power by reactive power:

$$\cos(\varphi_u - \varphi_i) = \frac{P}{S}$$

Using this definition it becomes clear that a high power factor is desirable, as it indicates a high portion of active power compared to the total apparent power and hence is a measure for the efficiency of the power transfer. In other words: the higher the power factor, the smaller the reactive power and therefore the lower the unwanted power losses due to the reactive power. If a power of 1 kW has to be transferred to the two port network, it takes an apparent power of 1 VA in case of a power factor of 1 and 2 VA in case of a power factor of 0.5. This additional 1 VA has to be provided by the source and the corresponding currents generate power losses in resistive

elements.
As a high value of the power factor is desired, a lot of effort is spent to increase the power factor. For linear networks consisting of linear elements only (resistors, capacitors, inductors) this can be done rather simply by adding the complementary reactive element: In case of a network with inductive behavior, a capacitor is added and vice versa. This method of power factor correction is used for example for electric motors such as asynchronous motors: capacitors are placed accordingly close to the inductive motor windings. Non-linear loads require more complex measures for power factor correction.

Example: bulb in series with capacitor
Consider the bulb operated at a sinusoidal voltage source of U = 300 V and f = 50 Hz as discussed in chapter 7.4.5. The capacitor is 8.25 µF to achieve a voltage drop of U_R = 230 V across the bulb for a current of I = 0.5 A. What about the apparent power, the active power and the power factor?
The total impedance of the circuit is:

$$\underline{S} = \frac{1}{2}\hat{i}^2 \cdot \underline{Z} = I^2 \cdot \underline{Z} = P + jQ$$

The active power and reactive power are:

$$P = I^2 \cdot R = U_R \cdot I = 115W$$

$$Q = -\frac{I^2}{\omega C} = -96.5 \text{ var}$$

These results yield for the apparent power:

$$\underline{S} = P + jQ = (115 - j96.5)VA$$

Using the magnitude of the apparent power the power factor can be calculated:

$$S = \sqrt{P^2 + Q^2} = 150VA = U \cdot I$$

$$\cos(\varphi_u - \varphi_i) = \frac{115W}{150VA} = 0.77$$

The power factor is rather low and a rather high reactive power is oscillating to and fro and has to be provided by the source.

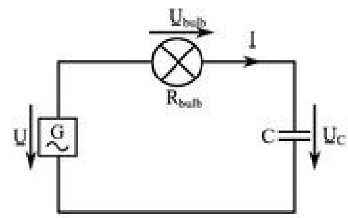


Fig. 9.7: A bulb in series with a capacitor to be operated by a sinusoidal voltage source.

10 Oscillating circuits

10.1 Series configuration

2nd order RLC circuits have been discussed previously and three cases were identified and analyzed: the overdamped, the critically damped and the underdamped case, depending on the values of R, L and C. The underdamped case can be obtained with small values of the resistor and in this case the voltages and the current oscillate with the damped frequency ω_d. In the limit of R = 0 Ω the circuit consists of an inductor and capacitor only and the voltage and the current oscillates with the natural angular frequency of:

$$\omega_n = \frac{1}{\sqrt{LC}}$$

Energy is transferred from the capacitor to the inductor back and forth.
If a sinusoidal voltage source (that acts as a driving force for the circuit) is added to the series RLC circuit (see Fig. 10.1) the voltages across the elements and the current will oscillate with the frequency of the source and the behavior of the circuit can be analyzed in terms of the complex impedance of the circuit:

$$\underline{Z} = R + j\left(\omega L - \frac{1}{\omega C}\right) = Z \cdot e^{j\varphi}$$

Magnitude and phase angle are:

$$Z = \sqrt{R^2 + \left(\omega L - \frac{1}{\omega C}\right)^2}$$

$$\varphi = \arctan\left(\frac{\omega L - \frac{1}{\omega C}}{R}\right)$$

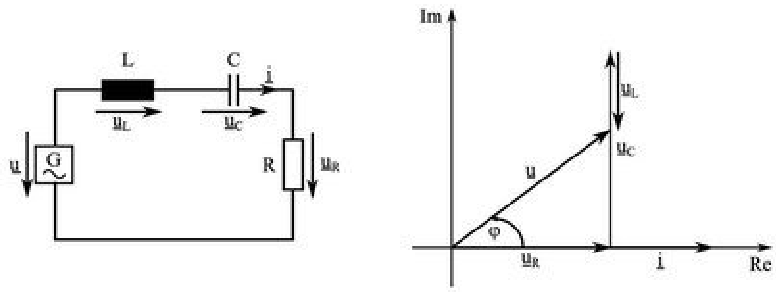


Fig. 10.1: An RLC oscillating circuit in series configuration (left); vector diagram of voltages and current (right).

The general behavior of the voltages and the current is depicted in the vector diagram on the right side of Fig. 10.1. As the voltage at the inductor leads the current by 90 ° and the voltage at the capacitor lags the current by 90 ° these two voltages have opposite directions in the vector diagram. Accordingly the voltage and the current of the source are in phase if the magnitudes of $\underline{u}_L$ and $\underline{u}_C$ are equal and therefore if the values of the reactance of the inductor and capacitor are equal:

$$\omega L = \frac{1}{\omega C}$$

If this condition is true the circuit is in resonance and the corresponding angular frequency is called the resonance angular frequency. It has the same value such as the natural angular frequency ω_n of the LC circuit:

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

The resonance case of the series RLC circuit has some interesting properties:

- $\underline{Z}$ = R is purely real
- Source voltage and current are in phase ($\phi = 0$)
- Smallest value of Z for given R, L, C
- ω_0 independent of R
- Maximum value of $|\underline{u}_R|$
- High voltages are possible at the inductor and the capacitor

The last two items can be seen by analysis of the magnitude of the voltages across R, L and C:

$$|\underline{u}_R| = |\underline{u}| \frac{R}{\sqrt{R^2 + \left(\omega_0 L - \frac{1}{\omega_0 C}\right)^2}} = |\underline{u}| \frac{R}{\sqrt{R^2 + \left(\sqrt{\frac{L}{C}} - \sqrt{\frac{L}{C}}\right)^2}} = |\underline{u}|$$

$$|\underline{u}_L| = |\underline{u}| \frac{\omega_0 L}{\sqrt{R^2 + \left(\omega_0 L - \frac{1}{\omega_0 C}\right)^2}} = |\underline{u}| \cdot \frac{1}{R} \cdot \sqrt{\frac{L}{C}}$$

$$|\underline{u}_C| = |\underline{u}| \frac{\frac{1}{\omega_0 C}}{\sqrt{R^2 + \left(\omega_0 L - \frac{1}{\omega_0 C}\right)^2}} = |\underline{u}| \cdot \frac{1}{R} \cdot \sqrt{\frac{L}{C}}$$

At the resonance angular frequency the magnitude of the voltage across the resistor is just the magnitude of the source voltage. As the circuit acts in resonance, the magnitudes of the inductor and capacitor voltage are the same and equal to the source voltage multiplied by a factor called the quality factor

$$Q = \frac{1}{R} \cdot \sqrt{\frac{L}{C}}$$

Depending on the values of R, L and C this factor can be significantly greater than one and hence the voltages at the capacitor and the inductor will be significantly greater than the source voltage. For example a RLC circuit with L = 1 mH, C = 1 μF and R = 4 Ω has a resonance angular frequency of about $\omega_0 = 32000\ s^{-1}$ and a quality factor of about Q = 8.

The magnitude of the voltages at the inductor and capacitor are 8 times higher than the source voltage and the magnitude of the resistor voltage. This voltage increase has to be taken into account when designing RLC circuits.

The reciprocal of the quality factor is the damping factor d given by:

$$d = \frac{1}{Q} = R \cdot \sqrt{\frac{C}{L}}$$

As in the mechanical case of a harmonic oscillating system with sinusoidal external force (e.g. a spring with damping and external excitation), energy for the RLC circuit is provided by the voltage source to the system. A part of the energy is dissipated by the resistor and the other part is accumulated in the circuit and resonates between inductor and capacitor. The energy stored in the circuit corresponds to the maximum energy stored in the inductor when the current is at its maximum value î (at this moment the voltage across the capacitor and hence the energy stored in the capacitor is zero due to the –90 ° phase shift):

$$E_{circuit} = E_{L\max} = \frac{1}{2} L \cdot \hat{i}^2$$

The energy that is dissipated at the resistor in each cycle is given by the effective value of the current:

$$E_{Rloss} = \frac{1}{2} R \cdot I^2 \cdot \frac{2\pi}{\omega_0} = R \cdot \hat{i}^2 \cdot \pi \cdot \sqrt{L \cdot C}$$

The ratio of the maximum energy stored in the circuit (e.g. in the inductor) to the dissipated energy in one cycle is related again to the quality factor Q:

$$Q = \frac{2\pi \cdot E_{max}}{E_{Rloss}} = \frac{2\pi \cdot \left(\frac{1}{2} L \cdot \hat{\imath}^2\right)}{R \cdot \hat{\imath}^2 \cdot \pi \cdot \sqrt{L \cdot C}} = \frac{1}{R} \cdot \sqrt{\frac{L}{C}}$$

The quality factor is therefore a measure for the amount of energy stored in the RLC circuit. Besides the resonance case, the frequency response of the circuit is also important. According to the formulas for the magnitude of the impedance and the phase angle, the series RLC circuit shows a very characteristic behavior: capacitive behavior at low frequencies, purely resistive behavior at the resonance frequency and inductive behavior at high frequencies. Fig. 10.2 shows this characteristic for the magnitude of the impedance and the phase angle.

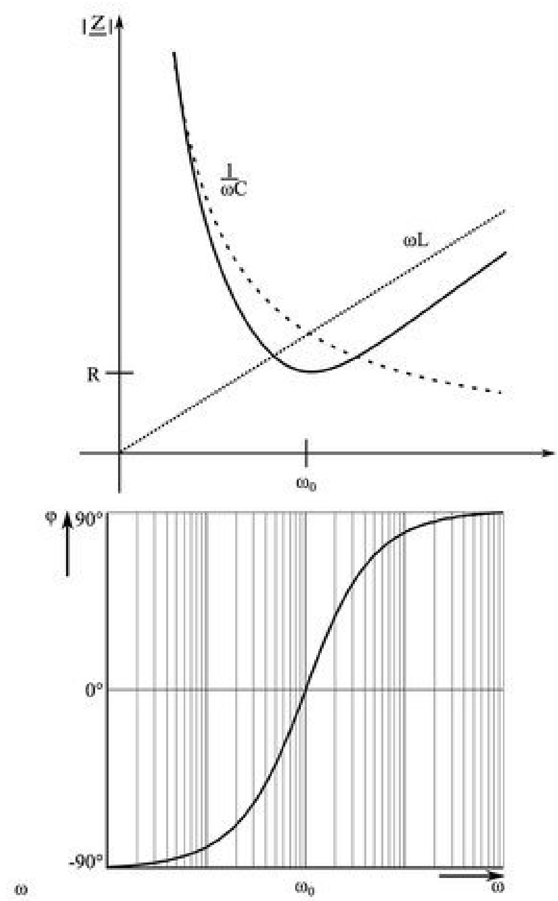


Fig. 10.2: Magnitude of the impedance of the series RLC circuit with the minimum value of R at resonance frequency (top); the corresponding phase angle (bottom).

The capacitor dominates both the magnitude of the impedance and the phase angle at low frequencies: a nearly 1 / (ω · C) behavior for the former one and a phase angle of about –90 °. In contrast, the almost linear behavior of the magnitude and the phase angle of about 90 ° clearly show the dominating behavior of the inductor at high frequencies. At the resonance frequency the inductive and capacitive parts cancel each other out and just the resistive part remains: $|\underline{Z}| = R$ and the phase angle is 0 °.

The voltages at the resistor, the inductor and the capacitor have their maximum value at the resonance angular frequency as the impedance is minimal. Close to the resonance angular frequency the voltages drop more or less sharply as depicted in Fig. 10.3. The cut-off angular frequencies ω_1 and ω_2 are the angular frequencies on both sides of the maximum where the voltage dropped down to 1/√2 or to –3 dB respectively.

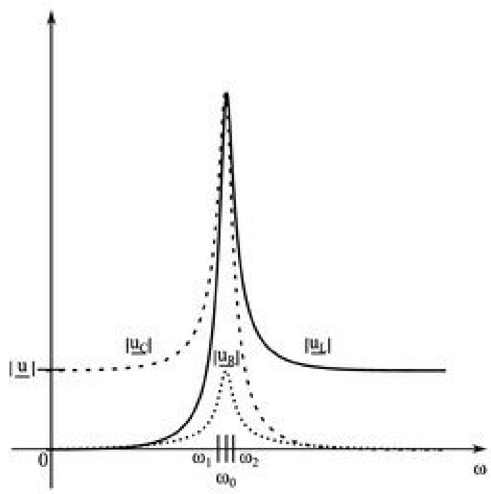


Fig. 10.3: The frequency dependence of the voltages across the inductor, capacitor and resistor, ω_1 and ω_2 are the cut-off frequencies.

In terms of a two port network the series RLC oscillating circuit acts as a band-pass filter. Considering the voltage source to be the input voltage of a two port network and the resistor voltage to be the output voltage just frequencies close to the resonance frequency can pass the network. The narrower the peak is, the better the filtering of a small frequency band is around the resonance frequency. A characteristic of the band pass filter is the bandwidth given by the difference of the cut-off frequencies (cut-off angular frequencies divided by 2π)

$$b = f_2 - f_1 = \frac{\omega_2 - \omega_2}{2\pi}$$

The bandwidth can be calculated using the frequency dependence of $|\underline{u}_R|$. At the cut-off frequencies the resistor voltage dropped down to 1/√2 and hence the denominator of $|\underline{u}_R|$ has to be:

$$\sqrt{R^2 + \left(\omega_1 L - \frac{1}{\omega_1 C}\right)^2} = \sqrt{R^2 + \left(\omega_2 L - \frac{1}{\omega_2 C}\right)^2} = \sqrt{2}R$$

Correspondingly the cut-off angular frequencies yield

$$\omega_1 = -\frac{R}{2L} + \sqrt{\frac{1}{LC} + \left(\frac{R}{2L}\right)^2}$$

$$\omega_2 = \frac{R}{2L} + \sqrt{\frac{1}{LC} + \left(\frac{R}{2L}\right)^2}$$

Finally the bandwidth is related to the quality factor of the circuit:

$$b = \frac{\omega_2 - \omega_2}{2\pi} = \frac{1}{2\pi} \cdot \frac{R}{L} = \frac{1}{2\pi} \cdot \frac{\omega_0}{Q} = \frac{f_0}{Q}$$

The bandwidth of an oscillating circuit is the resonance frequency divided by the quality factor Q. By changing the quality factor the bandwidth can be tuned even if the resonance frequency stays equal. Fig. 10.4 shows the frequency dependence of the resistor voltage for different values of L and C.

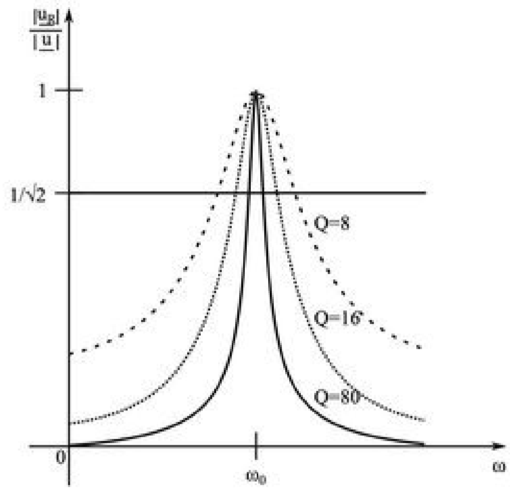


Fig. 10.4: The frequency response of a series RLC circuit with different quality factors but same resonance frequency.

The resonance angular frequency stays the same for the three parameters sets of R, L and C, but the quality factor is changed by a factor of 10. The bandwidth his reduced by a factor of 10 accordingly and the filtering functionality of this oscillating circuit is highly enhanced. The higher the quality factor, the more pronounced the frequency response is, the smaller the bandwidth is and the higher the voltage amplification is at the energy storing elements (inductor and capacitor).

10.2 Parallel configuration

There are many different topologies for oscillating circuits beside the series RLC circuit given here. They are all described by the parameters derived for the series RLC circuit such as resonance frequency, quality factor and bandwidth even though the formulas for the calculation of these parameters differ. Just as an example the parallel RLC circuit such as given in Fig. 10.5 has the same resonance angular frequency and the same bandwidth as the series circuit:

$$\omega_0 = \frac{1}{\sqrt{LC}}$$

$$b = \frac{1}{2\pi} \cdot \frac{\omega_0}{Q}$$

In contrast to this identity of the formulas the quality factor is the inverse of the quality factor of the series circuit:

$$Q = R \cdot \sqrt{\frac{C}{L}}$$

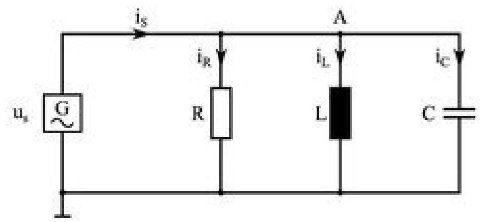


Fig. 10.5: A parallel RLC circuit.

Automotive application

Oscillating circuits are used in numerous automotive applications. Analog radios use these circuits as tuning circuits. Radio stations transmit at different frequencies and the radio antenna receives a superposition of all these radio signals. The oscillating circuit filters a small frequency band out of the antenna signal to receive just one radio station. Tuning to different stations can be done by using a variable capacitor to change the resonance frequency.

Keyless entry systems for vehicles or electronic immobilizer systems are other applications for oscillating circuits. These systems are closely related to RFID (radio frequency identification). Using keyless entry systems vehicles can be unlocked without the use of a (mechanical) key. As soon as the vehicle detects an approximation (e.g. by capacitive or optical proximity sensors) antennas of the keyless entry system start to transmit signals, e.g. with frequencies of some hundred kHz. The key has an oscillating circuit (very often just an LC oscillating circuit) with a fitting resonance frequency to receive the signal of the antennas. Afterwards the key sends a response back to the vehicle and in case of a correct response the vehicle is unlocked.

11 Semiconductor devices

Basic elements like resistors, capacitors or inductors are part of almost every electronic circuit. But besides these elements in particular semiconductor devices are extremely important to realize any complex electronic circuit. Semiconductors are used for devices like diodes or transistors as well as for rather complex integrated circuits (IC) like microprocessors and microcontrollers. These devices in general make use of the properties of doped semiconductors and combine n-and p-doped semiconductors and metals to realize different functionalities.

The basis for most semiconductor devices is pure silicon and compound semiconductors like SiC or GaN to some extent. Highly sophisticated processes are used to produce these semiconductor devices in the form of small rectangular dies. Depending on the functionality of the device (MOSFET, microprocessor, etc.) different process technologies are used, but to some extent these techniques are rather similar. The semiconductor industry is highly innovative in order to continuously improve their technologies. In particular semiconductor structures have shrunk very rapidly. According to Moore’s law the number of transistor elements per area doubles every 12–24 months.

The starting point for silicon dies, or chips is an extremely pure (>99,99999999 % purity) and crystallographically very well defined (very few crystal defects) cylindrical tube of silicon called an ingot. The diameter of the ingots ranges from 100–400 mm. From the ingot thin plates of silicon of some hundred micrometers thickness are cut. The surface of each wafer is separated into small rectangular areas known as dies. Dedicated process steps like photolithography, ion implantation for n-and p-doping, chemical etching, oxidation or vapor deposition are used repetitively to produce the required

structures and elements onto the wafer. Small structures of the elements like the gate length of the transistors are is just about 20 nm in 2014! Up to several billion transistors on one die of some cm^2 size can be realized by these techniques.
The connection of the billions of transistors on a die is achieved using many metal lines deposited on top of the wafer. At the end of the wafer process the wafer is separated into the single dies of some mm^2 or cm^2 size. Finally the silicon dies are mounted into dedicated packages or modules.

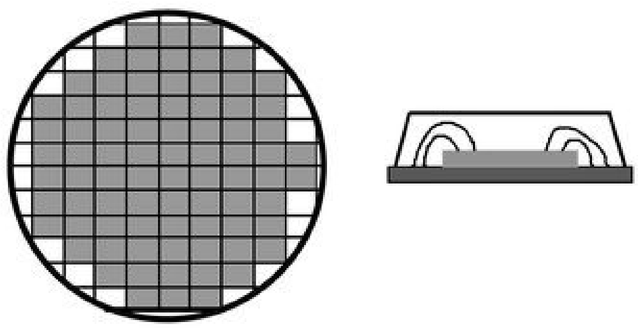


Fig. 11.1: A wafer with dies, complete dies are marked grey (left); packaged die (light grey) on a leadframe (dark grey) with bond wires (lines) in a package (right).

11.1 Diode

One of the simplest semiconductor devices is the combination of an n-and p-doped semiconductor to form a pn-junction as depicted in Fig. 11.2. The n-doped semiconductor has free electrons and stationary holes localized at the dopand. For the p-doped semiconductor it is vice versa. Both the n-and the p-doped semiconductors are electrically neutral. At the pn-boundary there is a strong concentration gradient of the free charge carriers: free electrons in the n-doped region and free holes in the p-doped region. Due to the concentration gradient, free charge carriers will diffuse into the other semiconductor and recombine: electrons will diffuse into the p-doped region and recombine with the holes of the p-doped region and vice versa. This diffusion and recombination results in a space-charge region around the junction as a small region of the p-doped semiconductor is now negatively charged and the n-doped semiconductor positively.

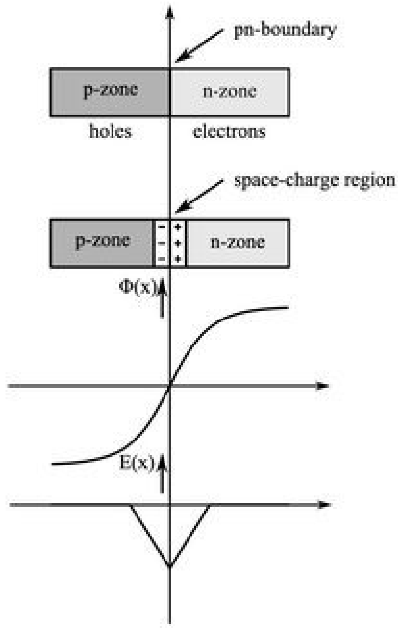


Fig. 11.2: From top to bottom: A theoretical pn-junction without electron transfer; a pn-junction with charge carrier diffusion and space-charge region; an electric field in x-direction and electric potential of the pn-junction.

These charged regions generate an electric field (see Fig. 11.2). Diffusion takes place as long as the electric field is not too strong and the potential difference is not too big. For silicon the diffusion stops at a diffusion voltage between the two regions of about 0.6–0.7 V. The size of the space-charge region depends on the number of charge carriers that recombine and within the space-charge region there are no more free charge carriers.
Applying an external voltage to the pn-junction will change the electric potential and the size of the space-charge region of the pn-junction as the internal and the external potential superpose. Depending on the polarity of the external voltage the pn-junction will show a different behavior.
If the higher potential of the external voltage is applied to the n-type semiconductor, the internal and the external electric field have the same direction and the electric potentials add as depicted on the left side of Fig. 11.3. The potential difference at the terminals of the pn-junction increases and also the space-

charge region enlarges. The pn-junction blocks any current flow.

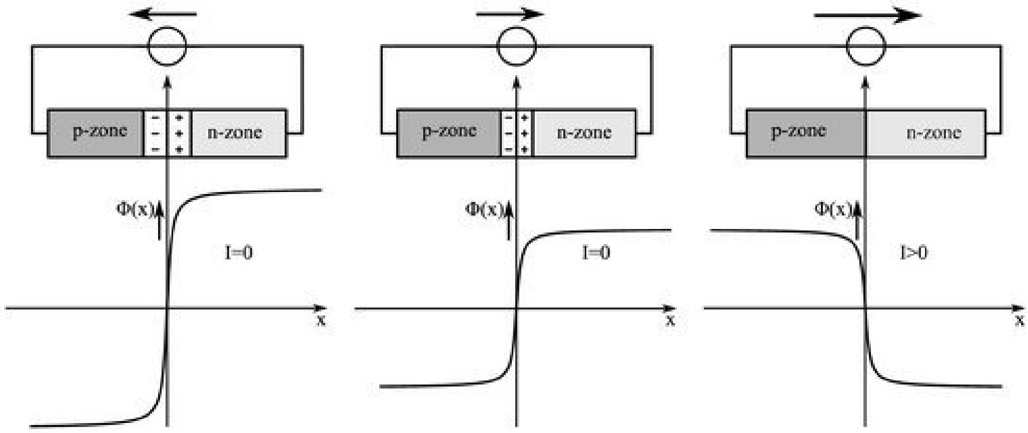


Fig. 11.3: An electric potential of a pn-junction with external voltage source.

If the higher potential of the external voltage is applied to the p-type semiconductor the internal and the external electric fields have the opposite direction. The internal electric potential is reduced by the external electric potential (right side of Fig. 11.3). The voltage at the terminals of the pn-junction decreases and also the space-charge region gets smaller. As soon as the external voltage is greater than the internal voltage, conduction is possible and a current can start to flow.
The semiconductor device built out of a pn-junction is called a diode. The two terminals of a diode are called the anode (p-type semiconductor) and cathode (n-type semiconductor). Fig. 11.4 shows the symbol of a diode with the anode and the cathode. The behavior of a real semiconductor diode differs slightly from the ideal pn-junction.

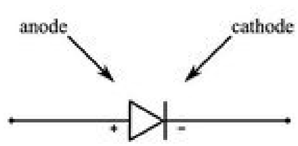


Fig. 11.4: Symbol of a diode with anode and cathode.

The characteristic of a diode is depicted in Fig. 11.5. In reverse direction the anode is connected to the lower potential and the diode blocks the current flow almost completely. Due to small amounts of minority charge carriers that diffuse into the space-charge region a very small reverse saturation current I_S of about some pA or nA can flow in real semiconductor diodes. This reverse saturation current depends strongly on temperature and on the semiconductor technology. At a high reverse voltage (50–1000 V) the reverse current increases sharply. This voltage is called the breakdown voltage and depends on the doping concentration, the semiconductor material and the technology for example. Most diodes should not be operated in breakdown mode as this operation may destroy the diode. An exception is the Zener diode (see below).

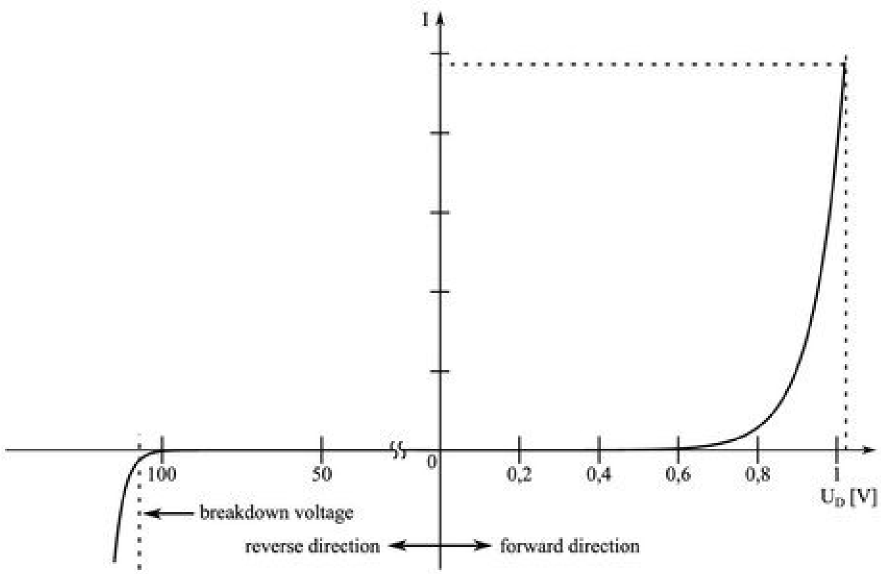


Fig. 11.5: A characteristics of a diode.

In the forward direction the anode is connected to the higher potential. For small voltages (< 0.7 V) only a very small current will flow. For voltages greater than about 0.7 V a significant current will start to flow and the current I depends on the voltage across the diode U_D in an exponential manner (ideal Shockley equation):

$$I = I_S \cdot \left(e^{\frac{U_D}{U_T}} - 1 \right)$$

U_T is the thermal voltage given by (e is the elementary charge):

$$U_T = \frac{k_B T}{e}$$

At room temperature the thermal voltage is about 26 mV.
The functionality of the diode corresponds to a valve. In reverse direction any current flow is (almost completely) blocked. But in the forward direction a current can flow if the applied voltage is high enough. Based on this functionality diodes are commonly used for any kind of rectification, or switching. Other applications include light emitting diodes (LED), photo diodes or voltage protection.
In Fig. 11.6 a schematic cross-section of a vertical diode is shown. A p-doped region is built up by ion implantation into the n-doped silicon wafer. The boundary of the two regions forms the pn-junction in the vertical direction. The metallization on top of the p-doped region is the electric contact for the anode. The other parts of the top surface are coated with SiO_2 for insulation. The bottom surface of the die is also covered with a metallization layer to form the cathode’s contact.

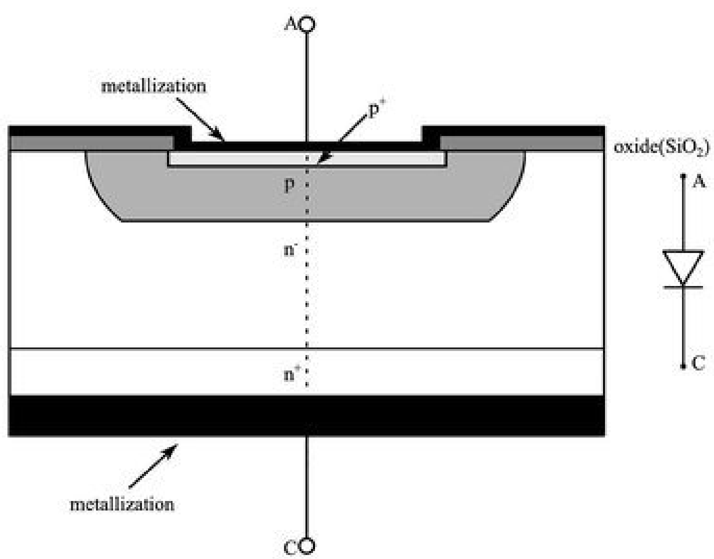


Fig. 11.6: Cross-section of a diode.

Several different packages are available for the packaging of the silicon dies of a diode, and most of these packages are standardized. These packages include cylindrical shape

packages with long wires as well as packages in surface mount and through hole technology (SMD and THD). Three typical package types for diodes are shown in Fig. 11.7. The small packages SOD-323 and SC-74 are SMD packages with short pins. The cathode of the SOD-323 package is marked with a stripe, for the SC-74 package the first pin (out of six pins) is marked with a dot. The dimensions of these two packages are rather small, just 1.25 mm by 2.5 mm and a height of 0.9 mm for the SOD-323 and 2.9 mm by 2.5 mm and a height of 1.1 mm for the SC-74.
The TO-220 is a through-hole device package (THD) for larger die sizes. Packages size is 10.5 mm by 16 mm and a height of 7.7 mm. The pins of this package are 13.6 mm.
Which package is used in an application depends on the power requirements of the application, the available space and the assembly technology for example.

Fig. 11.7: Diode packages: SOD-323 SMD package (left), SC-74 SMD package (mid), TO-220 THD package (right). Package drawings by Infineon Technologies AG.

Application

As the diode blocks the current in one direction it can be used to rectify an AC current as shown in Fig. 11.8. A sinusoidal input voltage is applied to the circuit of a diode and a resistor. During the negative half of the sinusoidal input voltage the diode blocks the current and the voltage drop across the resistor is zero. During the positive half the diode conducts if the input voltage is greater than about 0.7 V and according to KVL the voltage drop across the resistor corresponds to:

$$u_R(t) = u_i(t) - U_D = u_I(t) - 0.7V$$

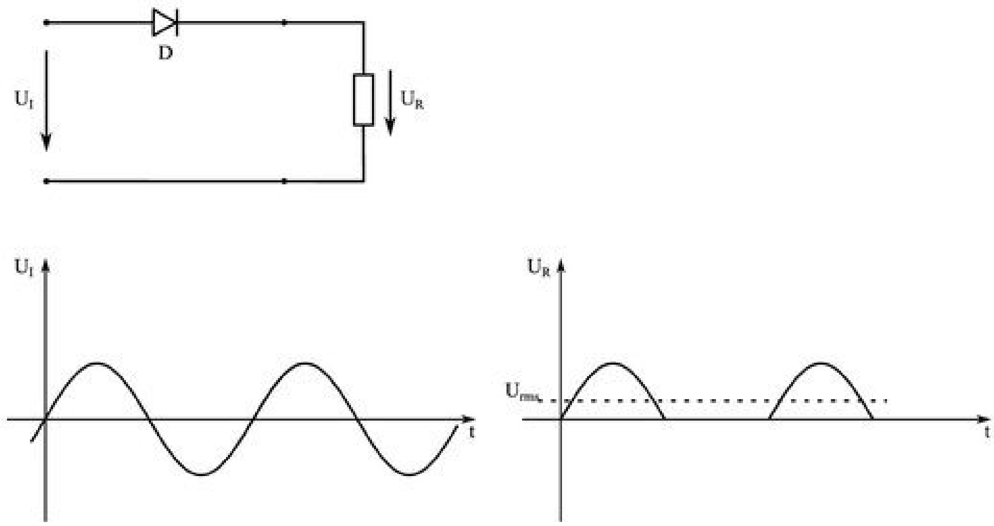


Fig. 11.8: A rectifier circuit with diode and resistor(top); an AC input voltage (bottom left) and a schematic drawing of rectified voltage at resistor (bottom right)

As depicted in Fig. 11.8 the resistor's voltage is a periodical function: half of the time it's a sinusoidal, the other half zero. The RMS value of the resistor voltage is rather low and hence the power that is transferred to the resistor. The disadvantage of this kind of rectification is that half of the period of the input frequency is blocked by the diode. To make use of the total period a full bridge circuit of four diodes can be used as depicted in Fig. 11.9.

For the positive half of the input voltage's period, diodes D_1 and D_4 conduct (if the input voltage is greater than $2 \cdot 0.7$ V) and diodes D_2 and D_3 block. The current flows via D_1 through the resistor and then via D_4. The resistor's voltage has a sinusoidal shape. For the negative half D_1 and D_4 block and D_2 and D_3 conduct. The current flow is D_3, resistor, D_2 and it flows again in the same direction through the resistor as in the positive half. In total the resistor's voltage is a periodic function again, but the RMS value is higher than in the simple one way rectifier with just one diode. By adding a capacitor parallel to the resistor the resistor's voltage can be smoothened after the rectification to get a more DC-like voltage.

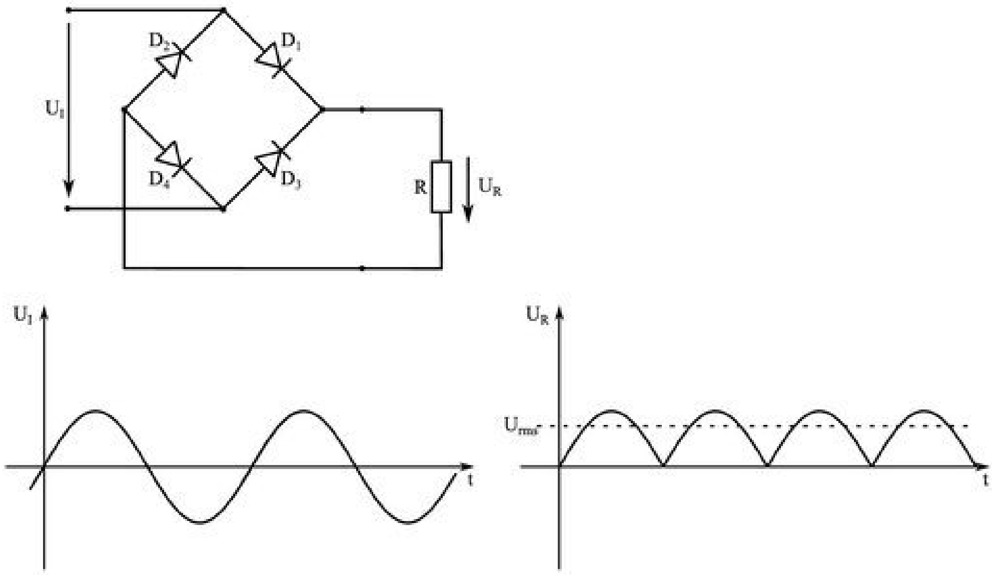


Fig. 11.9: A rectifier circuit with full bridge and resistor(top); an AC input voltage (bottom left) and a schematic drawing of rectified voltage at resistor.

Another application for diodes is to realize voltage stabilization or overvoltage protection using a Zener diode. The Zener diode is a special type of diode that is particularly designed for operation in breakdown mode (special doping and very thin junction). For reverse voltages above the breakdown, or Zener voltage this type of diode is able to conduct high currents and the voltage drop across the diode stays nearly constant and equal to the Zener voltage. To achieve a well defined Zener voltage it can be tuned and controlled during the fabrication process (doping level, size of the very thin pn-junction). Zener voltages may range from of about 3–100 V. In forward direction the Zener diode behaves like a normal diode.

Fig. 11.10 shows the symbol of a Zener diode and a circuit for overvoltage protection. The Zener diode is used in parallel to the load resistor. Using this reverse biased Zener diode limits the voltage across the load resistor to the Zener voltage and hence protects the load from overvoltage.

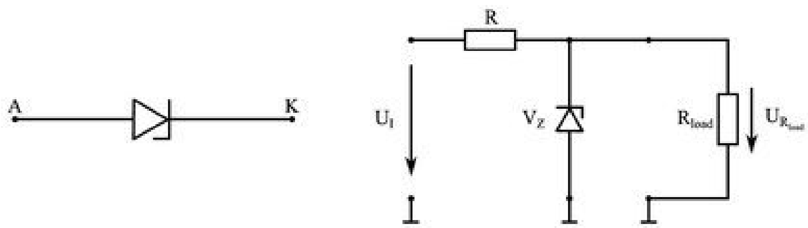


Fig. 11.10: The symbol of a Zener diode (left) and circuit for overvoltage protection (right).

Besides purely electrical applications, diodes are also used for optical applications in the form of LEDs and photo diodes. For LEDs compound semiconductors like AlGaAs or InGaN are used. The LEDs are forward biased. Electrons from the n-doped region cross the pn-junction and recombine with the holes in the p-doped region. The energy that is set free during the recombination is emitted in form of photons of a dedicated wavelength and hence color. The luminous flux strongly depends on the current through the LED. Therefore LEDs are driven by a constant current source. LEDs emit different colors like red, blue or yellow depending on the semiconductor material. An emission of white light from LEDs is not directly possible without further optical components. One way to generate white light is to use a blue LED and to cover it with a photoluminescence material. This material converts the single color blue into white light.

Automotive Application

Diodes are used very often in all kinds of electronic control unit (ECU) in cars, e.g. for rectification, overvoltage and electrostatic discharge (ESD) protection. The use of LEDs ranges from small signal lights in the interior to high brightness LEDs for headlights. Photo diodes are used as light sensors. One particular requirement for many automotive ECUs is reverse polarity protection. Reverse polarity means that the battery is connected in the reverse direction. This can happen e.g. during maintenance work on the electronic system even though the connectors are marked with colors or are mechanically different. During reverse polarity short circuits can occur via elements like internal diodes or transistors. In Fig. 11.11 a simple ECU is shown with a Zener diode for overvoltage protection. If the voltage is applied in the correct direction (and is smaller than the Zener voltage) the Zener diode is reverse biased and the current is limited by the load resistor. If the voltage rises the Zener diode protects the load by limiting the voltage to the Zener voltage.

In case of reverse applied voltage the Zener diode is forward biased and a short circuit current via the Zener diode occurs. This excessive current may damage the ECU.

Reverse polarity protection is required to prevent any damage to an ECU. A simple and cheap way to realize reverse polarity protection is to insert a diode into the power line of an ECU as depicted in Fig. 11.11. If the battery is now connected in reverse direction the diode D prevents any current flow and there is no short circuit via the Zener diode. Hence the additional diode protects the ECU. If the battery is connected correctly the diode D is forward biased.

A disadvantage of this solution for reverse polarity protection is the reduction of the voltage at the ECU by the forward voltage of the additional diode (0.7 V). In addition, the power loss at this diode reduces the efficiency of the system. The power loss of the diode is:

$$P_{diode} = 0.7V \cdot I_{diode}$$

In case of high currents (e.g. the current in applications like electric power steering, EPS, might be rather high, at more than 100 A) the power losses can be high and a non-negligible amount of power is dissipated into heat by the diode. A proper selection of the diode is needed to cope with this heat and to avoid excessive heating of the device, e.g. a package that provides a good thermal path to conduct the heat from the silicon die to the environment.

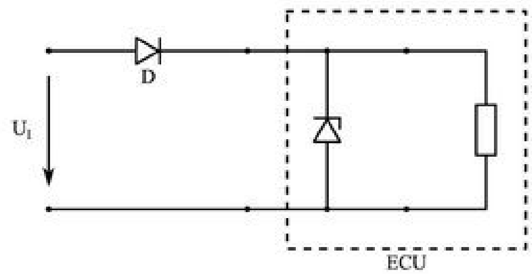


Fig. 11.11: A diode for reverse polarity protection of an ECU.

11.2 Bipolar transistor

The bipolar junction transistor (BJT) is a semiconductor device with two pn-junctions. Two different types of bipolar junction transistors exist, npn-and pnp-type. This nomenclature describes the structure of the BJT, e.g. n-doped layer, p-doped layer, n-doped layer like for the npn-type (see Fig. 11.12). One of the n-doped layers is heavily doped and called the emitter. The other n-doped layer is called the collector and the p-doped layer in between is the base. For the required functionality (see below) the base has to be very thin. Each of the three layers is connected to external terminals.

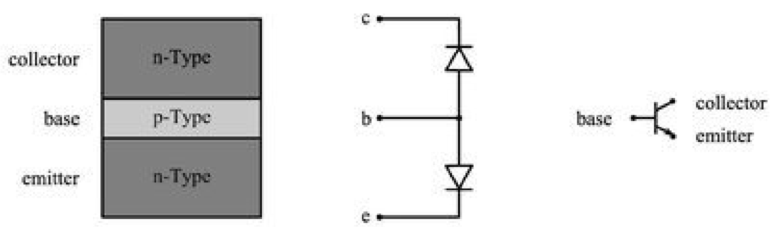


Fig. 11.12: An npn-BJT: layer structure (left); antiparallel diodes (center); circuit symbol (right).

Due to this npn structure there are two antiparallel diodes within the path from collector to emitter. If a voltage U_{CE} is applied between collector and emitter the base-collector diode blocks any current flow. If a voltage U_{BE} is additionally applied between base and emitter the situation changes. As soon as U_{BE} is greater than 0.7 V (and the collector diode is still reverse biased, $U_{CE} > U_{BE}$) the pn-diode between base and emitter becomes conductive. A small current I_B starts to flow: holes flow from base to emitter and electrons are emitted from the emitter towards the base. As the base is very thin, most of the electrons are able to cross the space-charge region of the base-collector pn-junction (which is still reverse biased). These electrons form a current I_C from the emitter to the collector. Some of the electrons emitted by the emitter do not cross the base-collector diode, but recombine within the base with the holes. This recombination would stop any further current flow. To prevent this stopping of the current, the base current I_B removes the electrons. As a consequence the base current I_B can control the collector current I_C.

As most of the electrons cross the base into the collector, the collector current is significantly greater than the base current:

$$I_C >> I_B$$

The ratio of the two currents is the current gain:

$$\frac{I_C}{I_B} = B$$

The current gain for a real BJT can be in the range from 4 to 1000. It depends on many technological and geometrical parameters such as density of donators in the emitter and base, the size of the base and diffusion parameters.

This is an important functionality of a BJT: an input current (base current I_B) controls an output current (collector current I_C) and the output current is the input current amplified by the current gain. The input current itself is controlled by the base-emitter voltage U_{BE}.

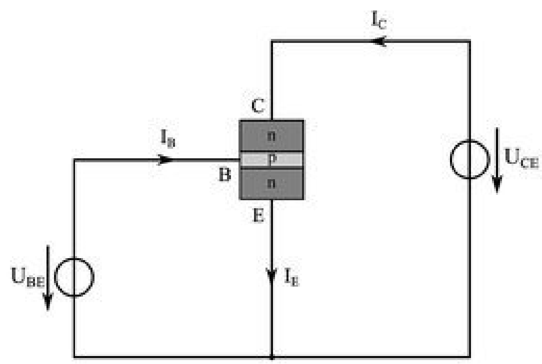


Fig. 11.13: An npn-BJT with external circuit: the base current drives the collector current.

The behavior of a pnp-type BJT is very similar to the npn-type, but the polarities of the external voltages have to be reversed. The structure and symbol of a pnp BJT are given in Fig. 11.14.

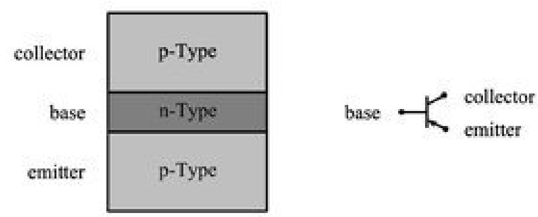


Fig. 11.14: A pnp-BJT: layer structure (left) and circuit symbol (right).

The characteristics of the BJT are mainly controlled by the voltages U_{BE} and U_{CE}. In Fig. 11.16 the diode characteristics of the base current is clearly visible. For base-emitter voltages greater than 0.7 V a small base current I_B flows, e.g. in µA range. As the output current I_C depends on the base current its shape is very similar to I_B (Fig. 11.16, right). Starting at about U_{BE} = 0.7 V (and $U_{CE} > U_{BE}$) a significant output current starts to flow. Slightly increasing the base-emitter voltage rises the output current I_C sharply. Depending on the current gain of the BJT the output current is much greater than the control current. In the example given in Fig. 11.16 the BJT operates in forward mode. The current gain is about 1000 and a control current in µA range controls the current in the mA range.

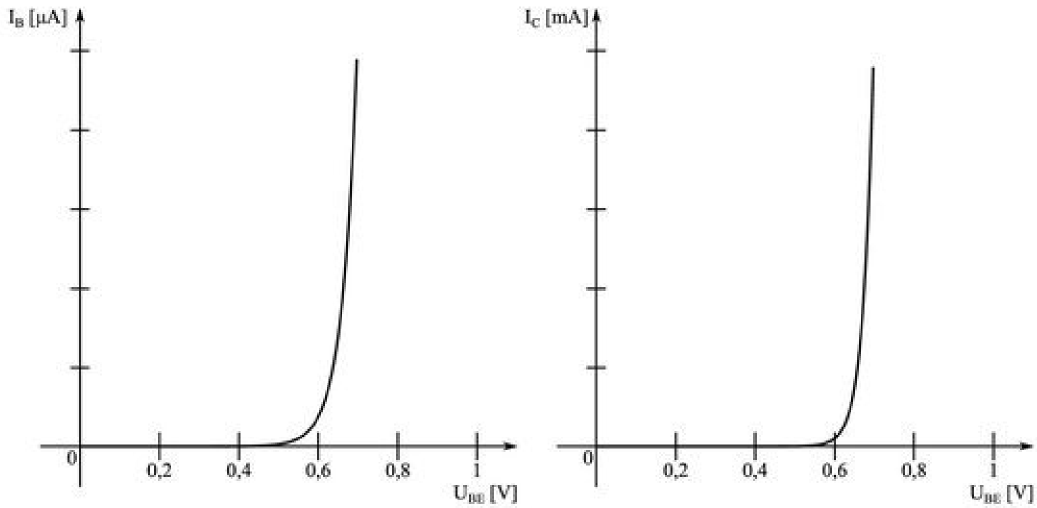


Fig. 11.15: Input characteristics of a npn-BJT: base current (control current, left); collector current (output current, right).

Besides the dependence of U_{BE} the collector current I_C also depends on the collector-emitter voltage U_{CE}. This output characteristics is depicted in Fig. 11.16. For small collector-emitter voltages up to the saturation voltage U_{CE}, sat the collector current rises sharply. Above the saturation voltage I_C just slightly increases linearly with U_{CE}. Important areas of operating are the cut-off, forward and saturation regions.

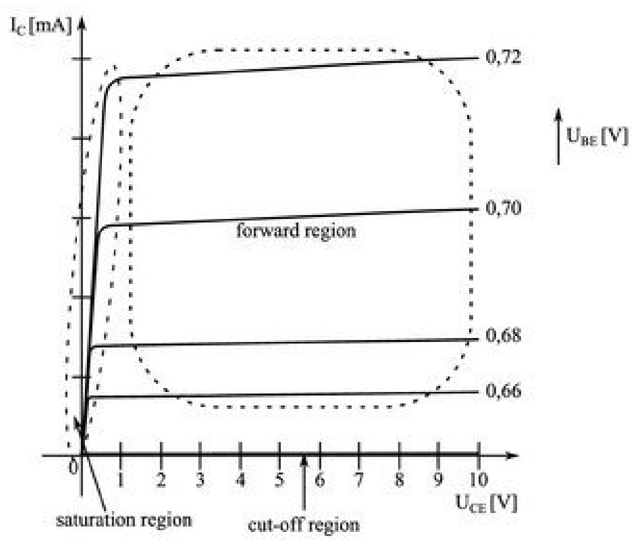


Fig. 11.16: Output characteristics of an npn-BJT: the parameter for the collector current is the base-emitter voltage.

In the cut-off region both pn-junctions serve to block and no collector current flows. In this case U_{BE} is too small (< 0.6 V) to drive a base current. In the output characteristics this operating mode is a straight horizontal line with I_C = 0 A in an ideal case. In reality there will be small leakage currents. Considering the BJT to be a switch, it is off in this operating mode.

The forward region has already been described in detail above. The emitter diode is forward biased and the collector diode is reversed biased, $U_{CE} > U_{BE}$. The collector-emitter voltage is higher than the saturation voltage $U_{CE,sat}$. In this operating mode the collector current is given by the current gain and the base current, $I_C = B \cdot I_B$ and the BJT acts as an amplifier for a small current. Small changes in the base current result in large changes in the collector current. Fig. 11.17 shows the example already discussed in terms of depending sources. The base current and the base-emitter voltage are set by resistors R_1 and R_2 to operate the BJT in forward mode. By the current amplification of B = 100 the collector current of 1.75 A is driven by the BJT to light the 21 W-bulb. The collector-emitter voltage is 2 V.

In saturation mode both diodes, emitter and collector diode, are forward biased. In terms of the circuit in Fig. 11.17 this operating mode can be reached by increasing the base current (e.g. by changing the resistors R_1 and R_2): the higher the base current, the higher the collector current. A higher collector current corresponds to a higher voltage drop across the bulb (resistance of the bulb is about 6.9 Ω). If the base current is increased to 19 mA the voltage drop across the bulb is 13 V and the collector-emitter voltage of the BJT drops down to 1 V. For a dedicated base current the saturation voltage $U_{CE,sat}$ of about 0.2 V is reached and both diodes are forward biased. In this case the collector current does not depend on the base current anymore and the collector-emitter resistance (= U_{CE} / I_C) has its smallest value. Considering the BJT to be a switch, it is on in this operating mode with smallest resistance value.

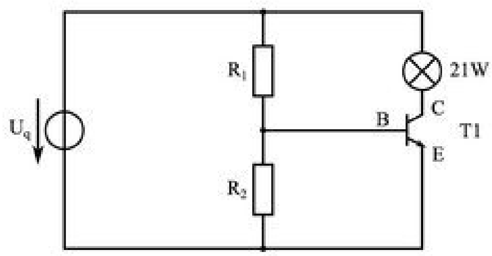


Fig. 11.17: A circuit with a bipolar transistor, the bulb acts as a resistive element with a resistance of 6.9 Ω.

Based on the output characteristics, two major applications for the BJT are amplification and switching. For amplification the BJT is operated in forward mode as in the example of Fig. 11.17. With a small control current a much higher current is controlled. In the other application the BJT is used as a switch. It is operated either in the off, or on mode to switch on and off a load.

To realize the required functionality the operating point of the BJT has to be set, i.e. the operating voltages U_{BE} and U_{CE} and currents I_B and I_C. Due to the interdependence of these values two of these values determine the operating point. In the example above the power and voltage of the bulb determine the BJT's operating parameters U_{CE} and I_C. With these values given the other two values U_{BE} and I_B were calculated using the BJT's properties such as current gain.

In all applications power is dissipated within the BJT due to the two currents, I_C and I_B. The total power loss is a sum of the base and the collector losses:

$$P_{total} = U_{CE} \cdot I_C + U_{BE} \cdot I_B \approx U_{CE} \cdot I_C$$

The base loss is much smaller than the collector loss as the base current is much smaller than the collector current. This electrical power is converted into heat and has to be conducted from the die to the environment by proper packaging and mounting of the device.

As for the diode the layer structure of a diode is obtained by regions of different doping within a bulk semiconductor. For an npn-BJT a typical layer structure is depicted in Fig. 11.18. The smaller p-and n-doped regions are implemented within the n-doped bulk semiconductor by ion implantation. The emitter and base contacts are on the top surface of the die whereas the collector contact is at the bottom side. Hence the collector current flows in a vertical direction through the die.

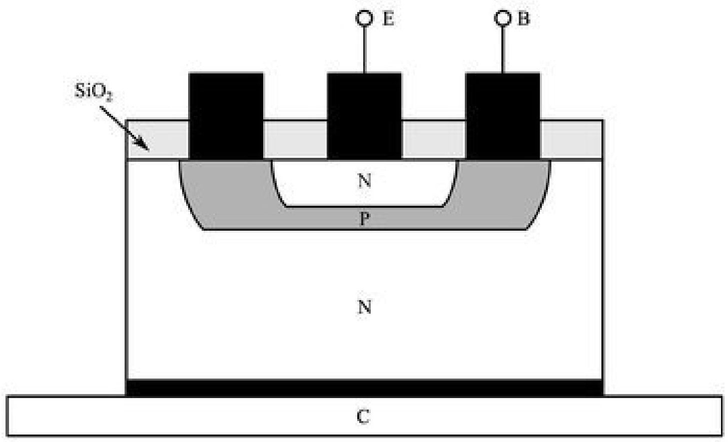


Fig. 11.18: The layer structure of an npn-BJT.

Packages for BJT are manifold and many of these are standardized. Both through-hole devices (THD) and surface mount devices (SMD) are available in different forms. The fitting device has to be selected depending on application requirements such as build space, mounting technology and electrical and thermal properties. For example the SOT-23 package (2.9 mm by 2.4 mm) with short pins is significantly smaller than the DPAK package (6.5 mm by 6.2 mm with a pin length of 3.7 mm). But the maximum collector current for the smaller package is much smaller than for the bigger package. TO-92 is a THD package with 5.2 mm by 4.2 mm and a height of 5.2 mm with a pin length of 14.5 mm

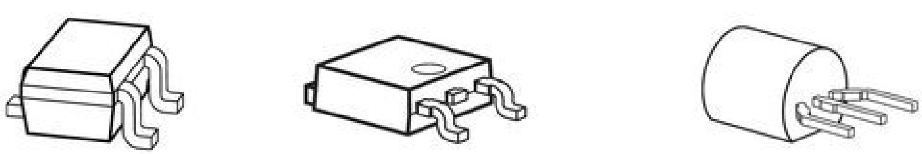


Fig. 11.19: Typical packages for BJT: two surface mount devices (SMD), small SOT-23 (left) and DPAK (TO-252, mid); TO-92 through hole device (THD, right). Package drawings by Infineon Technologies AG.

Automotive application

The use of BJT as current amplifier has already been demonstrated in the example of the bulb lighting above. The BJT acts as a constant current source to drive the bulb. If the base of the BJT is driven by a microcontroller the bulb can be switched on and off by the small base current. Instead of a bulb other loads that require a constant current source, like LEDs can be connected to this simple constant current source.

If the BJT transistor is used as a switch it is either off (cut-off region), or on (saturation region). In the on-state the power dissipated within the BJT is rather low as the voltage drop is just $U_{CE,sat}$. Seat heating is an application that can be realized with BJT use as a switch. In this typical convenience application a heating wire is embedded in the seat. As soon as a current flows through this wire, power is dissipated in the wire. The corresponding heating of the wire is the required functionality to make the driver feel more comfortable. To switch the heating wire BJT can be used as shown in Fig. 11.20. A microcontroller controls the switching of the seat heating. It drives the npn-BJT to operate in the forward region. Thus the small output current of the microcontroller is amplified to a much larger current to drive the pnp-BJT. This BJT operates in saturation mode to drive a rather large current of 5-10 A required by the heating wire with a low voltage drop $U_{CE,sat}$ and hence minimal power dissipation. The two resistors are used to set the operating points of the BJT.

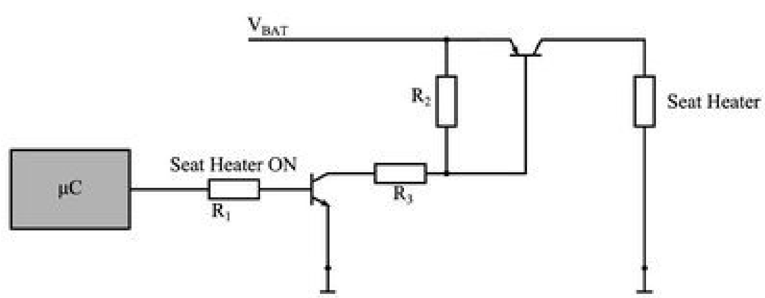


Fig. 11.20: BJT in a switching application, e.g. for seat heating.

11.3 MOSFET

Like a BJT a MOSFET (metal-oxide-semiconductor field effect transistor) is a semiconductor device with two pn-junctions. But the structure and the operating principle of a MOSFET differs significantly from that of a BJT. Like the BJT a MOSFET has three external terminals called the gate, source and drain as depicted in Fig. 11.21. A fourth connection, the bulk, is internally connected to the source terminal. Both source and drain are directly contacted to the semiconductor. But between the gate contact and the semiconductor there is an insulating layer, in most cases it is silicon oxide. This structure is reflected in the naming of the device, as it has a metal (gate contact)-oxide (insulator)-semiconductor (MOS) structure to build a field effect transistor (FET). In modern MOSFETs the metal of the gate contact is replaced by poly silicon, nevertheless the naming of the device remains. The gate is the switching part of the device as it controls the current flow from drain to source (or vice versa). Several types of MOSFET exist but here just the normally-off or enhancement MOSFET will be introduced. As with BJT (npn-and pnp-type) two different types of enhancement MOSFET exist: n-type and p-type.

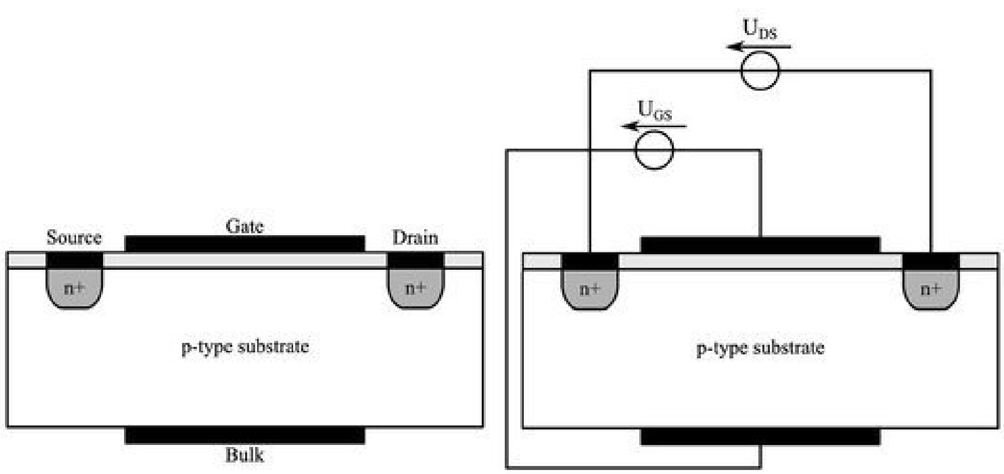


Fig. 11.21: Structure of a lateral n-type MOSFET with the four connections source, drain, gate and bulk (left); external connections for operation of the MOSFET.

In Fig. 11.21 the basic structure of an n-type MOSFET is depicted. For normal operation a drain-source voltage $U_{DS} > 0$ V is applied to the two terminals. As long as this voltage does not exceed the breakdown voltage of the device (the breakdown voltage depends on the technology, and is given in the data sheet of the device and should not be exceeded) and the gate-source voltage U_{GS} is zero, there is no current flowing as the drain-substrate diode is reverse biased. The gate and the bulk connection form a capacitor that is charged by applying a charge to it. As the bulk is internally short to the source, the capacitor's voltage corresponds to the gate-source voltage U_{GS}. If the gate-source voltage rises the electrical field between gate and bulk (electrical short to source) will attract electrons (minority charge carriers in the p-doped substrate) towards the gate. Due to the insulating oxide these electrons will accumulate beneath the gate. The higher U_{GS} gets the more electrons will be accumulated. If U_{GS} is sufficiently high, the electrons form an n-type channel beneath the gate from drain to source. For gate-source voltages above this threshold voltage U_{th} this n-type channel enables a current flow from drain to source. The threshold voltage is in the range of 2–3 V for MOSFET. The size and shape of the n-type channel depends strongly on U_{GS}. The behavior of p-type MOSFETs is similar to the n-type, but the gate source voltage has to be negative to switch the p-type MOSFET on.

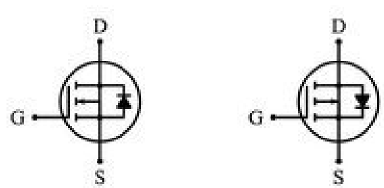


Fig. 11.22: Circuit symbols of an n-type MOSFET (left) and a p-type MOSFET (right).

As can be seen in the structure of a MOSFET there are two antiparallel diodes between the drain and source contact. As the source is in general short to the bulk (and hence to the substrate), the source-substrate diode has no functionality anymore. In contrast the drain-substrate diode is functional and forms the intrinsic body diode of a MOSFET. In the symbols of the MOSFET this body diode is also depicted (see Fig. 11.22). If the device is reverse biased ($U_{DS} < 0$ V) it behaves like a diode.

In contrast to the BJT, which is a current controlled device, the MOSFET is a voltage controlled device. The voltages U_{DS} and U_{GS} control the behavior of the MOSFET as depicted in Fig. 11.23. Four regions of operation can be distinguished in the output characteristics of MOSFETs.

In the cut-off region the gate-source voltage is smaller than the threshold voltage, $U_{GS} < U_{th}$ and the drain-source voltage is forward biased ($U_{DS} > 0$ V). There is no (or just very small) drain-source current I_D. The MOSFET blocks the current and considering the MOSFET to be a switch it is off in this operating mode.

In the ohmic region the voltage drop from drain to source is rather small ($U_{DS} < U_{GS} - U_{th}$). The gate-source voltage is above the threshold voltage, $U_{GS} > U_{th}$ and a conductive n-type channel is formed. If the gate-source voltage is well above the threshold voltage the drain current I_D is rather independent of U_{GS} but depends in a nearly linear manner from the drain-source voltage U_{DS}. This behavior corresponds to the behavior of an ohmic resistance. In this operation mode the MOSFET is switched on and behaves like a resistor with a drain-source resistance R_{DS}(on). Besides in the cut-off region (no current flow corresponds to no power loss inside the MOSFET) the power loss of the MOSFET in the ohmic region is lowest.

In saturation mode the drain-source voltage drop is high ($U_{DS} > U_{GS} – U_{th}$) and the $I_D – U_{DS}$ characteristics are almost parallel to the U_{DS} axis. Increasing the drain-source voltage has nearly no effect on the drain current. Instead the drain current can be controlled by the gate-source voltage. The higher the gate source voltage the higher the current, the MOSFET behaves like a voltage controlled current source. In this operation mode high power is dissipated in the MOSFET due to the high drain-source

voltage U_{DS} and high drain current I_D.
In the reverse region (U_{DS} < 0 V) the MOSFET behaves like a diode due to the intrinsic body diode. So in reverse operation the MOSFET does not block a drain current but it starts conducting is the forward voltage of the body diode is exceeded.

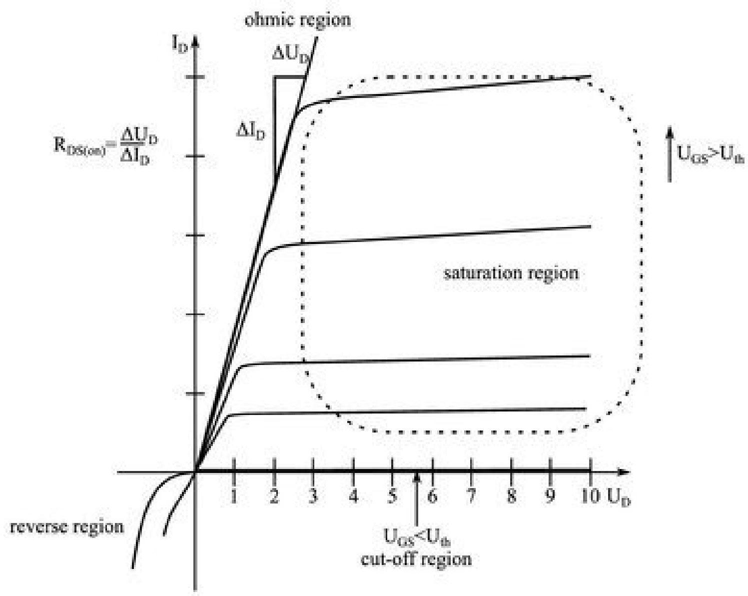


Fig. 11.23: Output characteristics of a n-type MOSFET.

Due to the output characteristics, MOSFETs are mainly used in switching applications to act as a switch. If drain-source voltage is forward biased the MOSFET operates in cut-off and ohmic mode. In the first mode the resistance of the MOSFET is infinite and the switch is off. In the ohmic mode it provides a (very low) on-state resistance R_{DS}(on) and the switch is on. In this mode the power loss is minimal for a conduction state. The on-state resistance for Power MOSFETs (MOSFETs designed in particular for high power applications) can be less than 1 mΩ and hence very low. To achieve this low on-state resistance the structure of Power MOSFETs differ from the structure introduced here. Instead Power MOSFETs have a vertical trench structure and the drain contact is on the bottom side of the Power MOSFET.
An operation in saturation mode is not desired most of the time. But it cannot be avoided at least for short times during

switching of the device (either on-off or off-on): during switching the gate capacitance has to be charged (switching on) or uncharged (switching off). During these switching events the device operates in saturation mode for a short time with significant power losses due to the simultaneously occurring drain current and drain source voltage.
The MOSFET is a voltage controlled device and the output is determined by the gate-source voltage (and the drain-source voltage). If the MOSFET is on or off no current has to be supplied to the gate, just a voltage. To operate a Power MOSFET in on-state a gate-source voltage of 5 V (so called logic level MOSFET) or 10 V (standard level MOSFET) has to be applied. But for switching, the gate capacitor has to be charged or discharged. To keep the switching time short a suitable gate current has to be provided.
With MOSFETs the power loss is determined by the drain current and the drain-source voltage. If used as a switch the total power loss is the sum of the losses during on-and off-state and during switching:

$$P_{total} = P_{off} + P_{on} + P_{switch}$$

The power loss in off-state is (nearly) zero and can be neglected. In on-state the MOSFET acts like a resistor with a resistance R_{DS}(on) and the on-loss is:

$$P_{on} = I_{DS} \cdot R_{DS}(on)^2$$

The switching losses depend on many device specific parameters and the external operating conditions and can hardly be estimated in general. A rough estimation shows the dependence of the power losses of switching time t_{sw} and switching frequency f:

$$P_{switch} = \frac{1}{2} \cdot I_{DS} \cdot U_{DS} \cdot t_{sw} \cdot f$$

Packages for MOSFETs are manifold and many of these are standardized. Both THD and SMD packages are available in different forms. Depending on application requirements like build space, mounting technology and electrical and thermal properties the fitting device has to be selected. Standard packages for Power MOSFET are DPAK and D2PAK in SMD technology and TO-220 and TO-262 in THD technology. For small signal MOSFETs also small packages like SOP-8 or SOT-23 are available.

Automotive application

MOSFETs and in particular Power MOSFET are frequently used in automotive applications, for example for reverse polarity protection (replacing the diode) or in any kind of switching application. DC/DC converter is an application where the MOSFET is used as a switch.

The standard automotive supply system on board has a voltage level of 12 V. But many devices, such as microcontroller need another voltage level, e.g. 5 V or 3.3 V. To convert DC voltages DC/DC converters can be used. A buck converter is a DC/DC converter that generates a lower output voltage. E.g. it can provide a 5 V output from a 12 V input voltage.

A schematic of a buck converter is depicted in Fig. 11.24. The DC input voltage U_E is converted to a lower output voltage U_A. It consists of a MOSFET, a diode, an inductor and a capacitor. The MOSFET switches on and off with a high frequency of some hundred kHz (e.g. 400 kHz). Using pulse width modulation (PWM) the duty cycle d of the switching can be adjusted:

$$d = \frac{t_{on}}{T}$$

Here T is the period of the switching and t_{on} is the time the MOSFET is switched on.

For the description of the behavior some simplifications can be made: the voltage drop across the MOSFET in the on-state is neglected (good approximation if a device with low R_{DS}(on) is used). In addition, the voltage drop across the diode is neglected (this changes the calculation slightly if the forward voltage of 0.7 V of the diode is taken into account). Also the current through the inductor is never zero (continuous mode) and a steady state situation is analyzed.

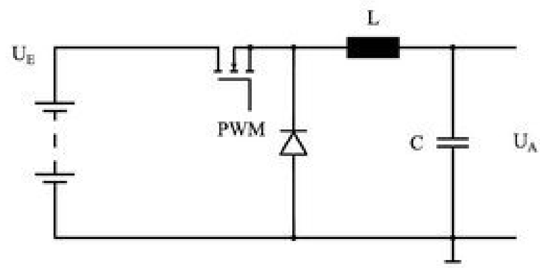


Fig. 11.24: Schematic of a buck converter.

If the MOSFET is switched on, the diode blocks any current flow and the voltage across the inductor is according to KVL:

$$U_L = U_E - U_A$$

During the time the MOSFET is switched on (t_{on}) the current I_L through the inductor rises linearly:

$$U_L = L \cdot \frac{dI_L}{dt}$$

During the off time (t_{off}) of the MOSFET the current keeps on flowing (as it cannot change in a step function) through the diode and the voltage drop across the inductor is:

$$U_L = -U_A$$

Accordingly the current decreases linearly. In steady state operation the rise of inductor current during on time equals the decrease during off time:

$$\Delta I_L = \frac{(U_E - U_A) \cdot t_{on}}{L} = \frac{U_A \cdot t_{off}}{L}$$

Using this steady state condition the output voltage can be calculated:

$$U_A = U_E \cdot \frac{t_{on}}{t_{on} + t_{off}} = d \cdot U_E$$

The output voltage just depends on the duty cycle (and the input voltage of course). By modulation of the duty cycle (that’s why PWM is used) the output voltage can be changed over a wide range.

Both the output voltage and the inductor current are not constant, but do change with the PWM frequency as depicted in Fig. 11.25. The capacitor is used to filter the output voltage to get a more DC-like behavior.

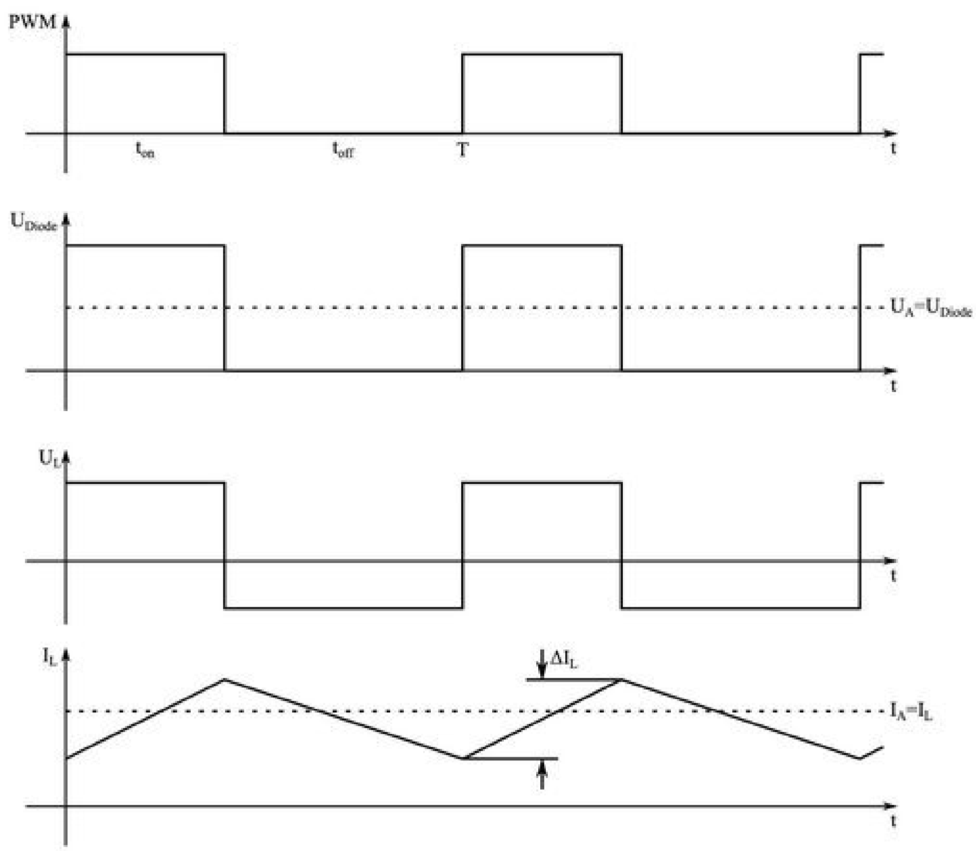


Fig. 11.25: Signals of a buck converter in continuous mode.

12 Circuit simulation

Circuit analysis can be achieved using the techniques introduced so far. Depending on the circuit under investigation, equation systems can be derived. Whether these equation systems can be solved at all depends on the complexity of the circuit, the size of the circuit, the elements used (e.g. linear, non linear) and the problem. Besides analytical calculations by hand, another way of finding the solution to a given problem of any circuit is circuit simulation.

Simulation in general transforms a complex system into an adequate model representation and analyzes the model. The result of this analysis is then transferred back to the original system. Key topics for simulation are the development of a proper model and the usage of the correct analysis and calculation methods.

In circuit simulation a real system is modeled by a circuit of lumped elements. These models can be as simple as a linear resistor with just a resistance, or very complex like semiconductors with parasitic inductances, capacitances, etc. Even for the simple elements the level of idealization has to be considered, depending on the purpose of the simulation: is a capacitor just an ideal capacitor, or do parasitic elements like an ESR or a parallel resistor have to be taken into account? So setting up a suitable representation of the circuit under investigation is a major task. Once the model is developed the calculations can be made by computer programs like PSPICE which is introduced here.

Circuit simulation can be used for different purposes. One purpose is visualization: to observe a general behavior of a circuit, e.g. the frequency response of a two port network. It can also be very useful for teaching and learning. Another purpose is for supporting circuit design for determining the behavior of a new circuit, checking for alternative solutions, determining working points and fitting parameters for the elements used. Or it can be used for design validation, to prove that a given design behaves as required and specified.

No matter what the purpose of simulation is, two points are always valid: A simulation is not reality and cannot replace reality, but it can help to improve reality. And a simulation without knowledge is worthless, or even dangerous.

Most circuit simulation programs are based on the SPICE (Simulation Program with Integrated Circuit Emphasis) software developed by the Electrical Engineering and Computer Sciences department at the University of California in Berkley in the early 1970s. This software can be used for all kinds of DC or AC circuit analysis, time or frequency domain analysis or power analysis. In SPICE the circuit is described in a netlist, and an ASCII text file that describes the circuit elements and their interconnection. The circuit elements are described by models, either simple ones such as for a resistor with just a resistance value or more complex ones such as for a MOSFET. The topology of the circuit and its elements determine the differential equations. Finally the algorithms of the SPICE software are used to solve these differential equations.

As the SPICE software is an open source software, several companies offer simulation software with additional features based on SPICE. Additional features are, for example, a GUI for the schematic entry of circuits, model editors to create own models or a graphical output of the simulation data. Examples of simulation software are PSPICE by Cadence Design Systems, LTspice by Linear Technology or Multisim by National Instruments. The examples in this chapter use PSPICE as a simulation program. For students a free student version of PSPICE is available for download and simulation.

PSPICE

The workflow for a simulation with PSPICE is shown in Fig. 12.1. It is split into several parts: Circuit editors like Capture or

Schematics are used to design the circuit in a graphical way. The schematic of a circuit is designed by drag and drop of models of circuit elements and the wiring of these elements. The models of circuit elements are stored in libraries including a graphical representation and its electrical behavior. Examples of models are any kind of sources, resistors, capacitors, or semiconductor devices. In addition a model editor can be used to describe own models if necessary. There is only one mandatory element that has to be used in all schematics: the ground or reference potential.

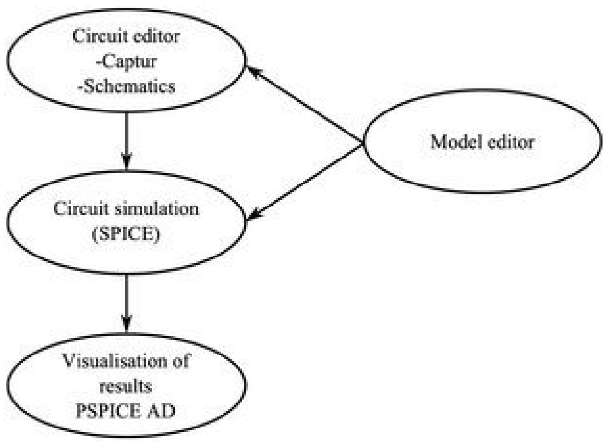


Fig. 12.1: Workflow of PSPICE simulation.

Afterwards the graphical schematic can be automatically translated into a netlist that can be used by the SPICE algorithms for circuit simulation. Different types of analysis are available:

- Bias point: determination of DC operating point;
- DC sweep: variation of a DC parameter in a given range (e.g. voltage source from 0–10 V in steps of 0.1 V);
- AC sweep: variation of operating frequency (e.g. for transfer functions);
- Time domain: simulation of time dependent effects (e.g. transient effects).

The graphical schematic is translated into a netlist that can be used by the SPICE algorithms for circuit simulation. The results of the simulation are graphically visualized in another PSPICE

module called PSPICE AD. Besides graphs of electrical parameters like currents and voltages, derived parameters like power or any mathematical value can be calculated and displayed.
A simple AC circuit with a sinusoidal voltage source, a resistor and a diode is used as an example (see Fig. 12.2). The mandatory ground element is denoted with 0 V. Simple elements like the ideal AC voltage source, or a resistor are described by a circuit element name (here V1, R1) and the corresponding parameter like the peak voltage of the source (10 V) and the frequency (50 Hz) or 2 Ω for the resistor. These parameters can easily be changed after the model is placed to the schematic. More complex element like the diode D1 in the circuit use more complex models with given parameters. These models are in general provided by the manufacturer of the device. Here the diode D1N914 is to be used in the design and the corresponding model is placed into the schematic. After the models are place the wiring can be done by just connecting the elements with lines.
After the circuit is completed the simulation setup has to be done. This includes the selection of simulation type and simulation parameters like simulation time.

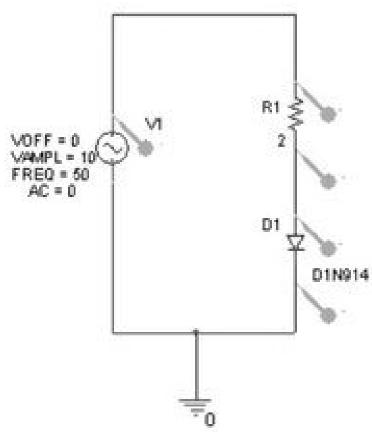


Fig. 12.2: A schematic of a simple AC circuit with probe marks in PSPICS capture.

The next step is the generation of the netlist for the simulation. The netlist of the AC circuit is depicted in Fig. 12.3. The three non-ground elements are listed: in the first column the type and name of the element, in the next columns (here 2^{nd} and 3^{rd}) the nets, or wires that are connected to the element are listed. The wiring information is followed by the information about the parameters of the element, e.g. 2 for the 2 Ω. The parameter information may extend to the next line like for the voltage source.

```
* source  TEST  START
V_V1  N00603  0
+SIN  0  10  50  0  0  0
D_D1  N00759  0  D1N914
R_R1  N00603  N00759  2
```

Fig. 12.3: The netlist of the circuit depicted in Fig. 12.2.

Before finally starting the simulation probe marks can be set in the circuit to probe voltages or currents. The values of these marks are in the end graphically displayed in the simulation output. The time domain simulation of the example circuit yields the expected behavior (see Fig. 12.4): The diode blocks the current during the negative half period of the source voltage. If the source voltage is above about 1 V the diode starts to conduct and the current causes a voltage drop across the resistor.

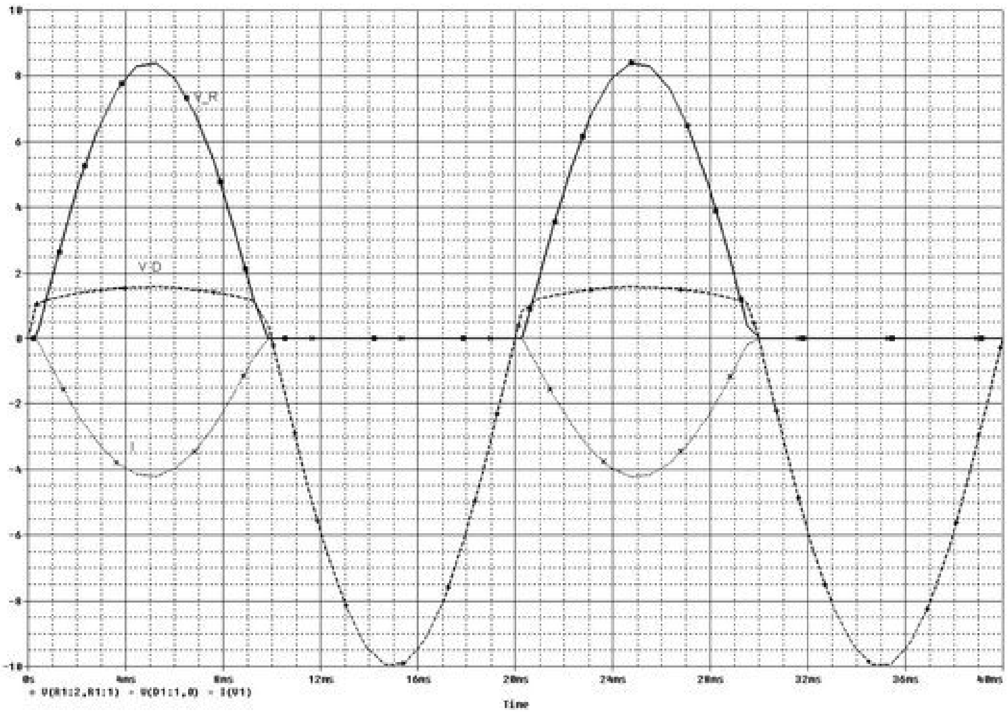


Fig. 12.4: Time domain simulation with PSPICE.

The models that are used can be rather simple (e.g. resistor, capacitor) or rather complex (e.g. diode, transistor). Complex models of dedicated elements that should be used are most of the time available from the manufacturer of this device. Examples are all kind of semiconductor devices like bipolar transistors or MOSFETs. Fig. 12.5 shows the electrical PSPICE model of the NP50N04YUK Power MOSFET by Renesas Electronics. The model is generated to reflect the real behavior of the device as well as possible. Besides the basic property of the on-resistance R_{DS}(on) it takes capacitances like the gate-source capacitance (C_{GS}), and parasitic resistances like the gate resistance R_G or the body diode into account.

```
.SUBCKT NP50N04YUK 1 2 3
**************************************
*      Model Generated by Renesas    *
*         All Rights Reserved        *
*Commercial Use or Resale Restricted *
**************************************
* Model generated on December 1, 2012
* MODEL FORMAT: SPICE2G.6
* POWER MOSFET Model (Version 3.1)
* External Node Designations
* Node 1 -> Drain
* Node 2 -> Gate
* Node 3 -> Source
***************************************
M1  4 5 3 3 NMOS W=5198515.2u L=0.4u
DDS   3  1    DDS
CGS   5  3    5.880E-10
RG    2  5    3.57
RD    1  4    RTEMP 0.805264E-3
FGD   1  5    VFGD 1
EVGD  7   0 1 5 1
DDG1  8  7    DD1
DDG2  8  0    DD1
EGD1  9   0 7 8 1
EGD2 10   0 8 0 1
COX  10 11    9.07886E-10
DCRR 11  9    DDG
VFGD 11  0    0
**************************************************************************
.MODEL  NMOS       NMOS  (LEVEL  = 3           TOX    = 500E-10
+ XJ     = 0.14E-06      LD     = 0            WD     = 0
+ TPG    = 1             RS     = 0.9E-3       RD     = 0.8235604E-3
+ RG     = 0             NSUB   = 2.811E17     IS     = 0
+ UO     = 600           KAPPA  = 0.006
+ NFS    = 0.146E12      THETA  = 0.241
+ KP     = 2.4061E-5     PHI    = 0.87296      VMAX   = 1.51E5
+ CGSO   = 0             CGDO   = 0            CGBO   = 0
+ XQC    = 1.0           AF     = 1            CBD    = 0
+ CBS    = 0             CJ     = 0            CJSW   = 0
+ FC     = 0.5           JS     = 0            KF     = 0
+ MJ     = 0.5           MJSW   = 0.33         PB     = 0.8
+ RSH    = 0)
**************************************************************************
.MODEL DDS D (CJO=3.06687E-9 VJ=1.542717618 M=1.027746599
+RS=0.001593006 IS=2.543E-12 TT=0.9876E-8 N=1.012594482 BV=40)
**************************************************************************
```

```
.MODEL DDG D (CJO=7.93101E-10 VJ=0.483405441 M=0.45294397 IS=1E-32 N=50
FC=1E-08)
**************************************************************************
.MODEL DD1 D (CJO=0 N=1)
**************************************************************************
.MODEL RTEMP RES (TC1=15.825349E-03  TC2=4.84641E-05)
**************************************************************************
.ENDS NP50N04YUK
```

Fig. 12.5: PSPICE model of a Power MOSFET NP50N04YUK by Renesas Electronics.

Automotive Application

Like for any design of electronic systems circuit simulation is carried out to a large extent for automotive applications. As with all circuit simulations it serves as a design support tool, a virtual circuit prototyping and circuit validation tool. This speeds up the design phase and increases the quality of the design as many elements can be tested in advance. Besides the simulation of electrical properties circuit simulation programs can perform thermal simulations to some extent. In particular for power electronics this additional feature can be very helpful in finding suitable designs and solutions.

References

Albach, Manfred, Elektrotechnik, Pearson Studium, 2011

Bobrow, Leonard S., Fundamentals of Electrical Engineering, 2nd edition, Oxford University Press, 1996

Dorf, Richard C., Circuits, Signals, and Speech and Image Processing, 3rd edition, CRC Press, Taylor & Francis Group, 2006

Dorf, Richard C., Electronics, Power Electronics, Optoelectronics, Microwaves, Electromagnetics, and Radar, 3rd edition, CRC Press, Taylor & Francis Group, 2006

Hagmann, Gerd, Grundlagen der Elektrotechnik, 13th edition, AULA-Verlag, 2008

Morris, Alan S. and Langari, Reza, Measurement and instrumentation: theory and application, 1st edition, Elsevier, 2012

Rizzoni, Giorgio, Fundamentals of Electrical Engineering, 1st edition, McGraw-Hill, 2009

Index