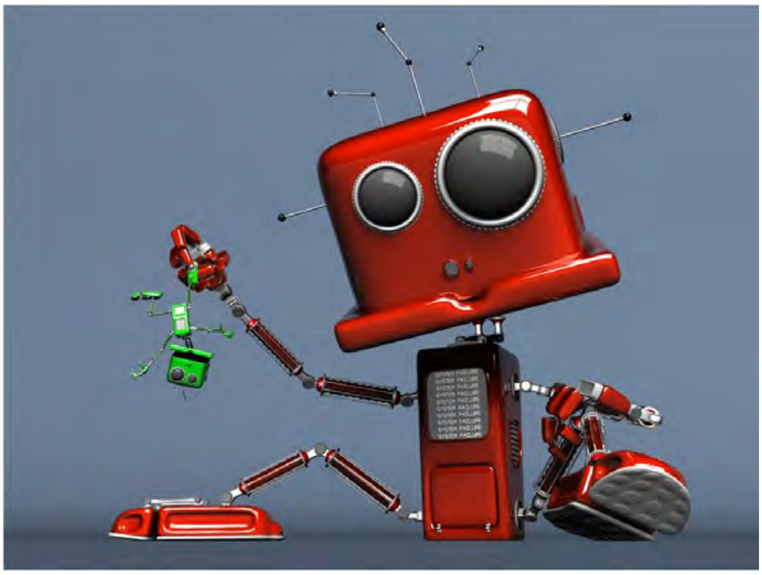
H.M. Kuipers
R.J.P. Bosch
M.E.C. Mutsaers
S. Diepen

Creative Electronics



Version 1d

Contents

1

Introduction

Assuming that all aspects of electrical engineering and electronics can be dealt with within an 5 ects-credits study assignment, could make you believe that it's all about a limited knowledge area. The opposite is true: there are numerous textbooks on electrical engineering and they often are very voluminous. However, we know that often a backpack loaded with a limited amount of professional skills is sufficient for roughly understanding how simple electronic circuits work. And it will even enable you to design, build and verify simple circuits yourself. The purpose of this assignment is to hand over this 'backpack' by briefly introducing the most important topics of electrical engineering and electronics. Besides that, exercises on test and measurement methods (and equipment) are added to let you gain hands-on experience in verifying the working of your circuit designs.

Because of the nature of this assignment, the various topics and its underlying theory are only briefly introduced. Often formulas are given without their derivations and they will only be superficially explained. Discussions are often interrupted by short stimulating exercises. Some of these exercises will ask you to verify your findings by doing real-life experiments and measurements. The supplement pages contain a lot of information on practical issues like measurement equipment, sensors, actuators, colour codes of resistors etc. You will be challenged to study the subjects which you find difficult to understand yourself by exploring web sites and textbooks. The assignment will be concluded by asking you to apply your freshly-gained knowledge on electronics in a small design idea. You will have to submit a report which describes the results on the exercises and your design idea as well.

This compilation has been set up as a reader / exercise book but can also serve as a compact reference book on electronics. Besides information, you'll find the following icons in this reader:

! an important note

? a question which you have to answer

E an example which clarifies the discussed theory

an optional exercise which will help you in understanding formulas and gaining insights

a practical assignment which you have to do

Summing up, this assignment serves the next goals:

- it allows you to gain basic skills for designing, implementing and testing simple analog and digital electronic systems by providing you with information and exercises on the various topics;
- it's a starting point for further exploration of the area of electrical engineering and electronics;
- it provides you with the requisite level of skills and knowledge demanded for starting one of the other electro-related ID assignments.

2

Voltage, current and power

2.1 Voltage, current and power

In and around the house, there are a lot of applications which need an electrical power source. You can think of big energy consumers, like washing machines and refrigerators, which get power from the electricity grid, but also of smaller ones, like watches and calculators, which use sources like batteries. In contrast to the electricity grid, the smaller power sources, which are most of the time used in small (portable) applications, need to be replaced or recharged, because they can run out of the stored energy.

The power P that devices request is supplied by a power source (e.g. the electricity grid or batteries) and can be expressed in terms of a voltage V and a current I:

$$\boxed{P = V \cdot I \qquad (2.1)}$$

The unities for these quantities are listed in Table 2.1, together with their symbols.

Note that we have to distinguish between power that is generated (e.g. by a power source) and power that's consumed or dissipated by a device (e.g. motor, lamp etc.). Also note that in Table 2.1 a new quantity is introduced ('potential difference'). Actually, when people talk about a voltage, they usually talk about a potential difference. In an electric circuit all nodes (i.e. a 'point' to which we connect our measurement equipment when a voltage is to be measured) have a certain potential (referred to a common or reference node). The voltage that can then be measured between two

Quantity	**Unity**	**Symbol**
Voltage, potential diff.	Volt (V)	V
Current	Ampere (A)	I
Power	Watt (W)	P

Table 2.1: *Electrical quantities with their respective unities and symbols.*

nodes (which is what we do as we connect the equipment to two nodes in the circuit) is called a potential difference.

The term (electric) power is frequently misused as an alternate name for electrical energy. Yet energy and power are two different things: electrical energy is the power consumed during a period of time. The unity of electrical energy is Joule (J). The unity used by electrical utility companies is the Watt-hour (W·h) or the kiloWatt-hour (kW·h). Instead of talking about "flow of power" and "consume a quantity of electric power", one should use "flow of energy" and "consume a quantity of electrical energy", respectively.

2.2 DC situations

In the previous section two kinds of electrical power sources were mentioned: the electricity grid and batteries. Besides the fact that both are usually used for different types of applications (e.g. portable applications in the case of batteries), there is another difference between these sources. Batteries are so called Direct Current (DC) sources, whereas the electricity grid is an Alternating Current (AC) source. In this section DC will be discussed and AC will be discussed in the next section.

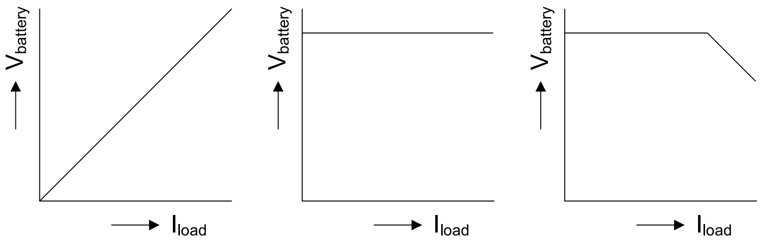
In DC situations, current always flows in the same direction. Furthermore, when talking about DC voltage sources, we can think of constant voltages that are supplied. Those sources are often presumed to be ideal, which means that the voltage is independent of the magnitude and duration of the current.

Batteries are not the only DC-sources. Often, batteries are not sufficient for an application. In these cases you can use DC-sources which are connected to the electricity grid through so called 'net-adapter'-circuitry. In Appendix A you can find a description of how this conversion can be done. Furthermore, conversion between different DC voltages is discussed there. In contrary to batteries, the DC-sources connected to the electricity grid behave like ideal DC-sources.

? Question 2.1

In the graphs below, three different relations between $V_{battery}$ and I_{load} are drawn.

1. Which graph depicts an ideal voltage source?
2. Which graph depicts a realistic voltage source with limited power?

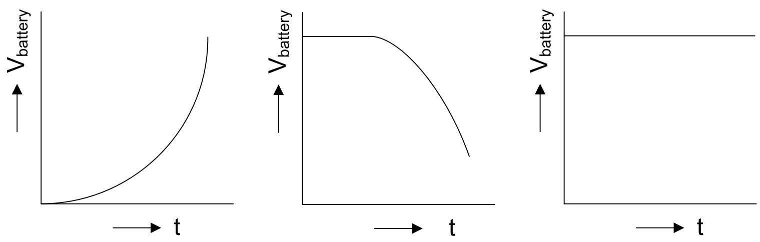


(a) (b) (c)

? Question 2.2

In the graphs below, three different relations between $V_{battery}$ and t (time) are drawn. The current I_{load} is taken constant.

1. Which graph depicts an ideal voltage source?
2. Which graph depicts a realistic voltage source with limited energy?



(a) (b) (c)

! Note

When doing experiments which require a constant voltage, you can make use of a DC-voltage source. These sources have at least two connections: the mass (black) and the positive potential (red). The mass can be seen as the ground and we take its potential as 0 V. The potential difference between the black and red connection is the voltage supplied by the source. In Appendix D you can find more information about the most common sources you will be using at the university.

? **Question 2.3**

For a DC-power supply the following properties are given:
$V_{out} = 50$ V
$I_{max} = 2$ A

An experimental audio-amplifier is designed to operate at a voltage of 50 V. For this amplifier, it is also given that the maximal output power is 50W, with an efficiency of 35%.

The definition of efficiency is $\frac{P_{out}}{P_{source}}$ x 100%.

Is the supply sufficient for this amplifier? Explain!

When a source supplies a constant current (independent of the load and duration) we talk about a current source. You won't come across such current sources as a laboratory power supply that often although a lot of the modern power supplies act as a current source when the current limiting circuit becomes active.

2.3 AC situations

As said before, we can also use the electricity grid as a power source. When we look to the potential difference between the two plugs of the contact, we see that it alternates (actually, if we put a resistance between the plugs, we could see that the current alternates). In fact, the potential difference between the plugs can be written as:

$$\boxed{V(t) = V_{top} \cdot \sin(\omega \cdot t) \qquad (2.2)}$$

where f is the frequency of the signal (which should be around 50 Hz), V_{top} the peak value or amplitude (which is about 325 V) and t the time. The period T of the sine wave equals one over the frequency. In Figure 2.1 an example of a sine wave is shown. The parameters V_{top} and T are indicated.

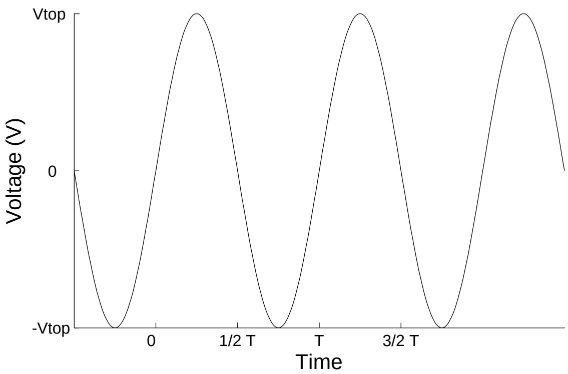


Figure 2.1: *Example of an AC sine waveform.*

The electricity grid in the Netherlands has a frequency f of 50 Hz, and an amplitude V_{top} of 325V. Note that a lamp connected to the electricity grid goes on and off twice during one cycle.

> **?** **Question 2.4**
>
> Our eyes are not fast enough to notice, but in fact a light bulb that is connected to the electricity grid flickers. At what frequency does this bulb flicker?

Sometimes we can have a combination or superposition of an AC voltage (named V_{AC}) and a DC (named V_{DC}) voltage. The latter is then called an offset voltage. This will be illustrated later in this reader, when you start working with a function generator.

2.3.1 RMS values

The value of an AC voltage changes continuously from zero up to the positive peak, through zero to the negative peak and back to zero again. For most of the time the (absolute) voltage is less than the peak voltage, so the latter is not a good measure of its real effect. Instead, we use the root mean square voltage (V_{RMS}).

RMS value - *The effective value of a varying voltage or current. It is the equivalent steady DC (constant) value which has the same heating potential. This means that the AC waveform will generate the same heat in a resistive load.*

What the definition says is that when you connect a device to an AC source (for example a lamp to the electricity grid), during one period the consumed (mean) power can be written in terms of DC currents and voltages. This means that:

$$\frac{V_{RMS}^2}{R} = \left(\frac{V(t)^2}{R}\right)_{\text{mean of period}} \tag{2.3}$$

where V_{RMS} is the RMS value (DC equivalent) of $V(t)$. Since R is constant, we get:

$$V_{RMS}^2 = (V(t)^2)_{\text{mean of period}} \tag{2.4}$$

Since V_{RMS} should be positive, this results in:

$$V_{RMS} = \sqrt{(V(t)^2)_{\text{mean of period}}}\,. \tag{2.5}$$

The value of $(V(t)^2)_{\text{mean of period}}$ can be calculated by summing up all the instantaneous values of $V^2(t)$ during one period, divided by the number of values $(\frac{1}{N}(V^2(t_1) + V^2(t_2) + ... + V^2(t_N)))$. This can be expressed as follows:

$$(V(t)^2)_{\text{mean of period}} = \frac{1}{T}\int_0^T V(t)^2 dt. \tag{2.6}$$

Sometimes, the RMS value is called the effective value.

For a true sine wave, the relations between V_{RMS} and V_{peak} are:

$$V_{RMS} = \frac{1}{sqrt(2)} V_{top} \approx 0.707 V_{top} \tag{2.7}$$

$$V_{top} = sqrt(2) V_{RMS} \approx 1.414 V_{RMS} \tag{2.8}$$

These equations also apply to current.

Note

The values 0.707 and 1.414 which were specified in Equation 2.7 and Equation 2.8, respectively, only hold for true sine waves (the most common type of AC)!

Exercise 2.1

Verify that the V_{RMS} of our electricity grid is approximately 230 V using the values specified earlier and the following relation:

$$V_{RMS} = \sqrt{\frac{1}{T} \int_0^T V^{(}t)^2 dt}.$$

Exercise 2.2

Besides sine waves there are more types of AC signals. For example, square waves. Such waves have two values: during half of the period the signal has a positive value A and during the other half, the signal has a negative value $-A$. Calculate the V_{RMS} for a square wave that has a peak voltage of 325 V and a frequency of 50 Hz using the following relation:

$$V_{RMS} = \sqrt{\frac{1}{T} \int_0^T V(t)^2 dt}.$$

When you connect a lamp to an AC-source with 6 V RMS at its contacts or to a 6 V DC-source, you should not see any difference in brightness of the lamp. However, when you connect the lamp to an AC-source with a peak voltage of 6 V, the brightness of the lamp is dimmer than in the DC case (in case of a sine wave, the DC-source should be set to approximately 4.2 V for equal brightness).

You may find it helpful to think of the RMS value as some kind of average. Please remember that it is NOT really the average! In fact, the long-time average voltage (or current) of an AC signal is zero because the positive and negative parts exactly cancel out!

2.3.2 Sine waves

Until now we focussed on sine waves when talking about AC signals, while all signals whose magnitude and direction vary cyclically are AC signals. However, there are several reasons to focus on sine waves. The main reason is that they are all around us: when connecting to the electricity grid, we deal with sine waves.

A combination of mechanical rotation and electromagnetic properties result in a sinusoidal signal. We all have (or should have!) lighting on our bicycles. Although battery powered LED-lights are gaining popularity, you should be familiar with a dynamo. When you bike while your dynamo is making contact with your wheel, the rotating wheel makes the small wheel on your dynamo rotate as well. A permanent magnet is connected to this small wheel and its rotating field is fed through a coil. The changing magnetic field induces a voltage across the coil. When you measure the potential difference between both ends of the coil, you can observe a sinusoidal shape. For the electricity grid, large generators are used in which turbines rotate (e.g. due to flow of water or steam or due to wind). In fact, a dynamo on your bike is a small generator!

The rotating field in the generator can be seen as a vector. The sine wave that we see is a projection of this vector onto a certain axis, like drawn in Figure 2.2.

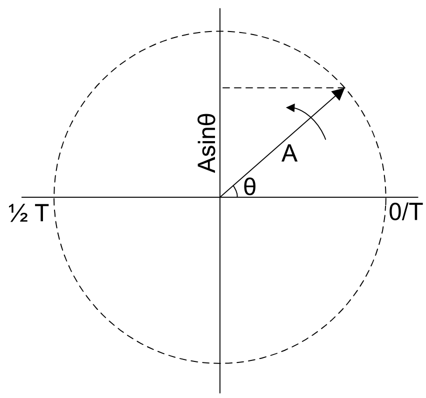
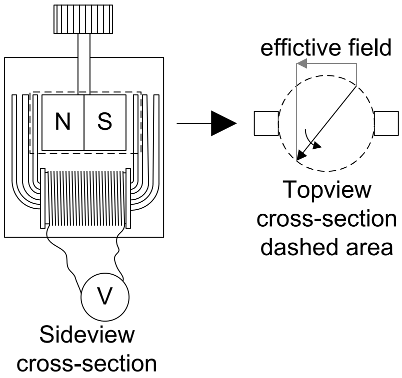


Figure 2.2: *The projection of a rotating vector on the y-axis results in a sine wave.*

At a certain point in time, the angle of the vector is θ. The potential difference that you can measure then is equal to $A \sin \theta$. The change in θ over time is ω, which is related to the period time T by $\omega = \frac{2\pi}{T}$.

Exercise 2.3

In the figure below you can see a cross-section of the side view of a schematic drawing of a dynamo. On the right side you can see the cross-section of the top view of the dashed area from the left. In this top view the 'effective' field that contributes to the induced voltage is highlighted.



Verify that, for a constantly rotating magnetic field, the induced voltage (which is linearly related to the effective field) has a sinusoidal shape.

2.4 Energy vs information

Until now we have focussed on voltages and currents which were needed for the electrical energy consumption of circuits and equipment. Besides this, voltages and currents are also used to contain and/or transport information. For example, the wave shape of a current that flows between an amplifier and a speaker represents information (sound) that will be transduced by the speaker.

Another example of using voltages and currents for transferring information is the transfer of data between computers using a network router. Note that the voltages on the in- and outputs of the switch are used for information (bits), whereas the router needs to be plugged into the electricity grid since it needs electrical energy to be able to work.

2.5 Symbols of sources and meters in electrical schematics

Electrical AC and DC sources have different symbols in schematics. These symbols are shown in Figure 2.3. The symbol for voltage and current meters are shown in this figure as well.

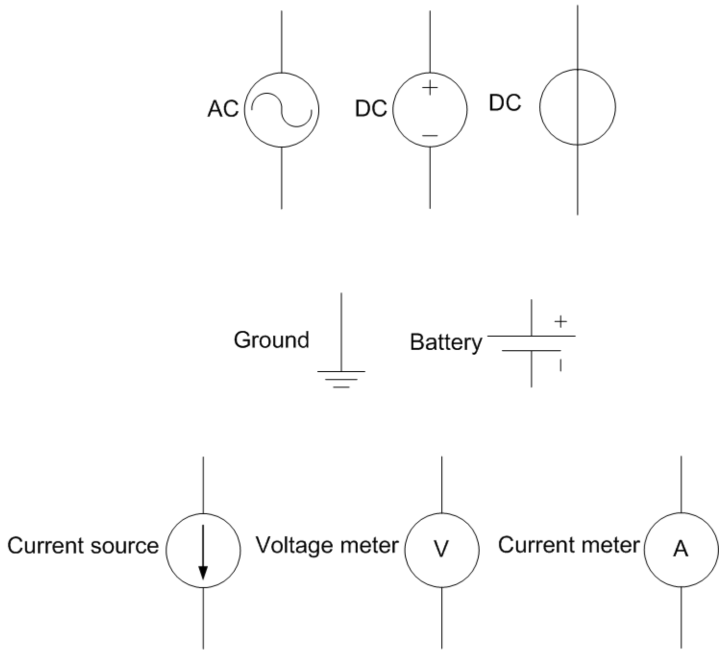


Figure 2.3: *Schematic symbols for AC source, DC sources, current source, ground, voltage and current meters.*

3

Resistance and Resistors

3.1 What is resistance and what are resistors?

Formally, in electrical engineering resistance (R) is defined as a measure of the degree of opposition that current is faced with.

Resistance only has a conceptual meaning; when building circuitry, you cannot get resistances from the shelf. Instead, you need to get a physical component that 'implements' the definition of resistance. This can be a resistor. Note that in Dutch both the concept resistance and its implementation are called 'weerstand'! In Figure 3.1 several resistors are shown.

In Figure 3.2 two commonly used symbols for drawing schematic electrical circuits are shown. The right symbol is often seen in schematics of American origin.

Resistors are made for many kinds of application. They are mostly made of carbon, metal film or just wire. The reciprocal of resistance is conductance.

3.2 Relation between voltage and current

It was said earlier that resistance is defined as a measure of the degree of opposition that current is faced with. This measure can be expressed in terms of voltage and current:

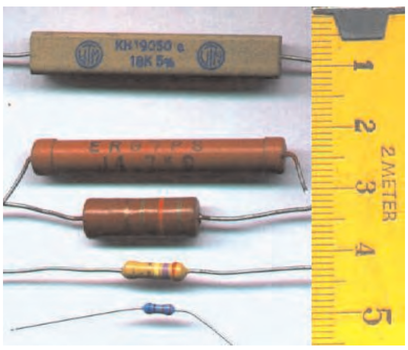


Figure 3.1: *Resistors that can each handle a different amount of power.*

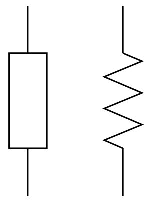


Figure 3.2: *Two symbols for resistances.*

$$R = \frac{V}{I} \tag{3.1}$$

where R is the resistance value specified in Ohm (Ω). This relation is often referred to as Ohm's Law.

> ? **Question 3.1**
>
> Draw the graph of $V = f(I)$ with $R =$ constant.

According to Ohm's Law, resistance resembles a constant relation between applied voltage and the current that flows through through the device. Ideal resistors don't exist but can be regarded be ing components that are constructed to have this relation regardless of:

- the applied voltage;
- the resulting current;
- the frequency;
- the temperature (of the resistor);
- the air humidity;
- the pressure.

3.3 Power behavior

In the previous chapter the relation between power, voltage and current was given. This relation can be applied to a resistor as well. The product of the voltage applied by a *source* to a *resistor* and the current that flows through it results in a power $P_{resistor}$ and a power P_{source}:

$$P_{resistor} = V_{resistor} \cdot I_{resistor} \quad \textbf{and} \quad P_{source} = -V_{resistor} \cdot I_{resistor} \tag{3.2}$$

where $V_{resistor}$ is the applied voltage and $I_{resistor}$ the current through the resistor.

We say that the power $P_{resistor}$ is dissipated in the resistor. The dissipated power is transformed into heat.

? Question 3.2

With a voltage source a voltage of 10V is applied over a resistor of $1k\Omega$.
Determine:

1. The current flowing **through** the resistor
2. The power **dissipated** in the resistor

? Question 3.3

Draw the graph of $P = f(I)$ with R = constant. Consider the fact that I can flow either ways, so can have a positive or negative sign.

Due to the fact that heat is generated, not every type of resistor can handle the same amount of power. The resistors you are going to work with the upcoming TV labs can handle a power of $\frac{1}{4}$ W or $\frac{1}{3}$ W. When you design a circuit you will have to take this into consideration and make sure that you do not overload a component.

Unfortunately, you cannot go to the store and ask for a resistor with an arbitrary resistance value. Resistors are produced in so called E-series. This means that only limited values are available and that you may have to combine several resistors (see the next section) in order to obtain the desired value.

Besides the fact that not all resistance values are (directly) available off the shelf, the values that are available are only (good) approximations of these values. Usually the resistance value of a resistor is guaranteed between upper and lower limits around the specified value. For example, the most commonly used low-power models (i.e. $\frac{1}{4}$ W and $\frac{1}{3}$ W) are the carbon and metal film resistors. The carbon resistors ($\frac{1}{4}$W) usually have a tolerance of ± 5%, whereas most metal film resistors ($\frac{1}{3}$ W) have a tolerance of ± 1%.

The previously mentioned resistors are provided with color codes that indicate the resistance value and tolerance. In Appendix B you can find more information on this topic.

? **Question 3.4**

Given a metal film resistor R with $P_{max} = \frac{1}{4}$ W (colour code: brown-green-black-brown-brown) and a voltage source: 19 V, $I_{max} = 10$ A. The resistor R is part of a complex circuit that is powered by the voltage source.

Show by calculation that it is not possible to let R dissipate more than the maximum amount of power that R can handle, no matter how R is connected in the circuit and considering the tolerance of its resistance.

3.4 AC behavior

As stated before, the resistance resembles a constant relation between voltage and current which is independent of the frequency. So, for different frequencies, equal powers are dissipated.

3.5 Connection: series and parallel

In the previous section it was mentioned that not all desired resistance values are available. To be able to construct a desired value, multiple resistance values have to be combined. There are three types of connections: series, parallel and combinations of these.

Series connection

In a series connection, multiple resistors are connected in such a way that the same current I_{re} flows through all resistors. If we were to replace all these resistors by one, we should need a value R_{re}. Because we know that the flow through all resistors is the same, R_{re} can be calculated easily: across each resistor there is a voltage drop (which is equal to the product of the resistance value times the current). There is thus a total voltage drop V_{re} which is equal to the sum of all single voltage drops. The resistance value R_{re} can now be calculated using Equation 3.2 and the values of I_{re} and V_{re}: $R_{re} = \frac{V_{re}}{I_{re}}$.

In Figure 5.5 an example of a series connection of three resistors is given. The three resistors in series can be replaced by a single resistor with a resistance value $R_{re} = R_1 + R_2 + R_3$. In general:

$$R_{re} = \sum_{i=1}^{N} R_i. \tag{3.3}$$

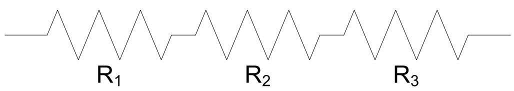


Figure 3.3: *Example of a series connection of three resistors.*

There are three observations that can be made:

- R_{re} is always higher than the largest (single) resistance value;
- If one of the single resistance values is much higher than the other two, R_{re} is almost equal to that value;
- The power dissipated in a single resistor is always lower than the power dissipated in R_{re}.

Parallel connection

In a parallel connection, multiple resistors are connected between the same nodes. The voltage across each of these resistors is equal. However, the current that flows between the two nodes divides itself over the resistors. Like with the series connection, a replacement resistance value R_{re} can be used. This value can be calculated by:

$$R_{re} = \frac{1}{\sum\limits_{i=1}^{N} \frac{1}{R_i}}. \tag{3.4}$$

Exercise 3.1

Verify the calculation of R_{re} for a parallel connection using the given relation.

In Figure 3.4 an example of a parallel connection of three resistances is given. The three resistances in parallel can be replaced by a resistance $R_{re} = \frac{1}{\frac{1}{R_1}+\frac{1}{R_2}+\frac{1}{R_3}}$.

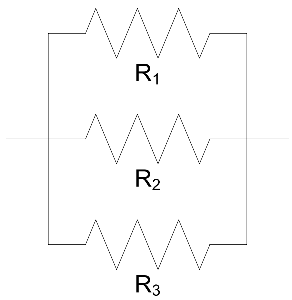


Figure 3.4: *Example of a parallel connection of three resistances.*

There are three observations that can be made:

- R_{re} is always lower than the lowest R;
- If one of the single resistance values is much lower than the other two, R_{re} is almost equal to that value;
- The power dissipated in a single resistor is always lower than the power dissipated in R_{re}.

? **Question 3.5**

You need a resistor of 500 Ω, $\pm 2\%$, $\frac{1}{2}$ W to be applied in a circuit.

Find two resistors from the E12 (see appendix B) series for resistors (the resistors having a tolerance of $\pm$ 1% and a maximum power dissipation of $\frac{1}{3}$ W) and connect them in such a way that R_{re} satisfies the demands for the resistance value, tolerance and the amount of power to handle.

3.6 Application

You will often see resistors in filters, switch circuitry, integrators and more. Another typical application for resistors is the voltage divider. As the name implies, a voltage divider can be used to divide the (input) voltage to obtain lower (output) values. For example, if the input voltage of a voltage divider is 9 V and we want to have a potential difference of 4 V, we can use the circuit drawn in Figure 3.5.

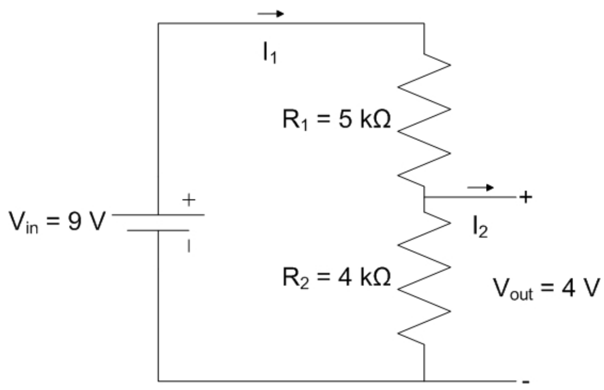


Figure 3.5: *Example of a voltage divider.*

Assume that $I_2 = 0$ A. Then:

$$
\begin{aligned}
I_1 &= \frac{V_{in}}{R_1 + R_2} \\
V_{out} &= I_1 * R_2 \\
&= \frac{V_{in}}{R_1 + R_2} * R_2 \\
&= \frac{R_2}{R_1 + R_2} * V_{in} \\
&= \frac{4000}{5000 + 4000} * 9 \\
&= \frac{4}{9} * 9 \\
so : V_{out} = &\ 4 \text{ V.}
\end{aligned}
$$

Note that the output voltage will equal 4 V **if and only if** $I_2 = I_{out} = 0$ **A**. In that particular case V_{out} only depends on the input voltage and the relation between R_1 and R_2:

$$\boxed{V_{out} = \frac{R_2}{R_{re}} \cdot V_{in} = \frac{R_2}{R_1 + R_2} \cdot V_{in} \qquad (3.5)}$$

In chapter 6 you learn how to approach networks like the voltage divider in a more formal way, by applying so-called nodal or loop analysis following Kirchoff's laws.

> **?** **Question 3.6**
>
> Create a voltage divider like depicted in Figure 3.5 only by applying resistors from the E12 series (see appendix B) having the following specifications: $V_{in} = 9$ V, $V_{out} = 4$ V, $I_1 = 0,5$ mA and $I_2 = 0$ mA.

> **Exercise 3.2**
>
> Using the voltage divider shown in Figure 3.5, show that for $I_2 > 0$ A the output voltage V_{out} is always lower than 4 V.

3.7 Potentiometers

There are situations where you need to adjust the resistance value of a resistor. Instead of interchanging resistors all the time, you can use adjustable resistors called potentiometers (or 'potmeters'/'pot' in short). A potmeter is made of two fixed connections and a slip contact, which is called the wiper. In Figure 3.6 a schematic drawing of a potmeter with its contacts is drawn.

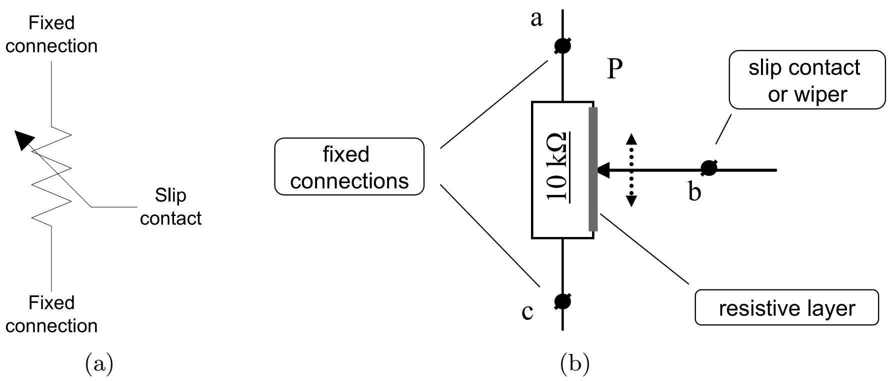


Figure 3.6: *Schematic symbol of a potmeter.*

Besides the schematic symbol of the potmeter drawn in Figure 3.6, you can also find the 'block'-symbol for a resistance (see Figure 3.2) with an arrow through it or an arrow to its side.

There are different types of potmeters: turn potmeters and slide potmeters. Turn potmeters have a rotatable axis which you can rotate using a screwdriver or a knob. Potmeters may be linear (i.e. the resistance value is proportional to the angle of rotation of the axis in the case of turn potmeters), but also logarithmic. Behind the turning knob of an audio amplifier that is used to regulate the volume you will often find a logarithmic potmeter.

When used in circuitry, potmeters usually have two applications: creating an adjustable resistance value needed for instance for filtering (see the next chapter) or creating an adjustable voltage divider (e.g. in an audio amplifier). In the first application, the wiper and one of the fixed connections are connected. An example of this is shown in Figure 3.7(a). In the second application the wiper contact is used for the output. An example of this is shown in Figure 3.7(b).

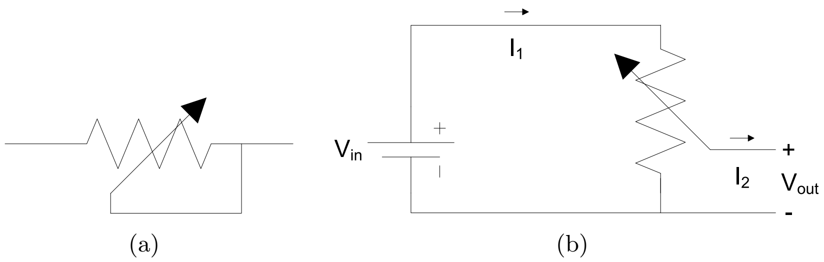


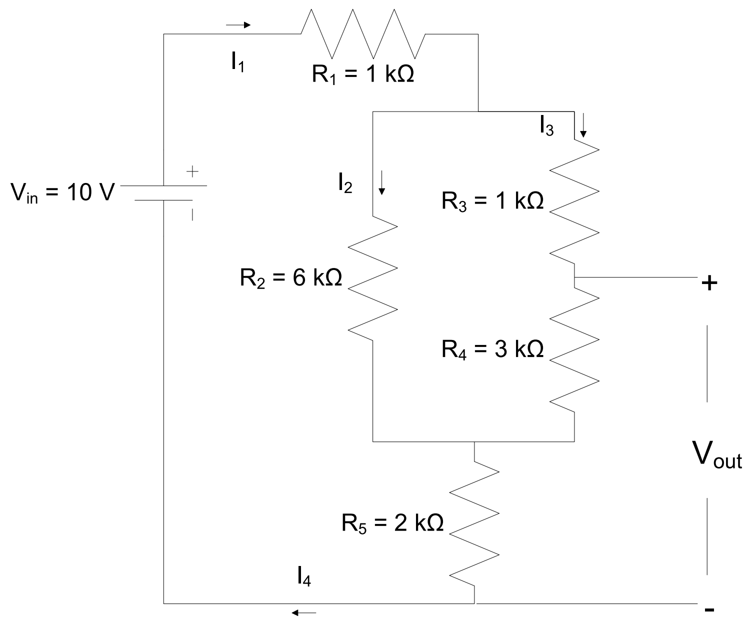
Figure 3.7: *Two application types for a potmeter: (a) 'plain' adjustable resistance value and (b) ajustable voltage divider.*

3.8 Example

In this chapter you have learned how to calculate the replacement resistance value for parallel and series connections of multiple resistors. In this section we give an example of how to calculate the replacement resistance value for a circuit with a combination of parallel and series connections.

Example 3.1

Calculate the output voltage V_{out} of the circuit shown below.



Solution:
To solve this problem, you will have to take several steps.

First note that $I_1 = I_4 = I_2 + I_3$.

From Ohm's law ($V = I \cdot R$) we find for V_{out}:
$V_{out} = V_{R5} + V_{R4} = I_4 \cdot R_5 + I_3 \cdot R_4$.

Now we have to find an expression for I_4 and I_3. You already know how to calculate a replacement resistance value for series and parallel connections. First we calculate the replacement resistance value R_{re1} for R_3 and R_4:
$R_{re1} = R_3 + R_4 = 4\ \text{k}\Omega$.

Now we can calculate the replacement resistance value R_{re2} for the parallel connection of R_2 and R_{re1}:
$R_{re2} = \frac{1}{\frac{1}{R_2} + \frac{1}{R_{r1}}} = 2.4\ \text{k}\Omega$.

Next, we can calculate the total replacement resistance value $R_{retotal}$ from the series connection of R_1, R_{re2} and R_5:
$R_{retotal} = R_1 + R_{r2} + R_5 = 5.4\ \text{k}\Omega$.

Using $R_{retotal}$, we can simply calculate I_1 with Ohm's law:
$I_1 = \frac{V_{in}}{R_{retotal}} = 1.85\ \text{mA}$.

The current I_1 also flows through R_{re2}, so we can calculate the voltage drop V_2 across R_{re2} (= voltage drop across R_2 and R_{re1}):
$V_2 = 1.85$ mA $\cdot 2.4$ k$\Omega = 4.44$ V.

With this voltage we can calculate the current I_3 through R_{re1}:
$I_3 = \frac{V_2}{R_{r1}} = 1.1$ mA.

Now we have everything we need to calculate the output voltage:
$V_{out} = I_1 \cdot R_5 + I_3 \cdot R_4 = 1.85$ mA $\cdot 2$ k$\Omega + 1.1$ mA $\cdot 3$ k$\Omega = 7.04$ V.

4

Capacitance and capacitors

4.1 What is capacitance and what are capacitors?

Capacitance (C) is defined as a measure of the amount of electrical charge (Q) stored for a given potential difference.

Just like resistance, capacitance only has a conceptual meaning; when building circuitry, you cannot get resistances nor capacitances from the shelf. Instead, you need to get a physical component that 'implements' the definition of capacitance: the capacitor. In Dutch the words 'capaciteit' and 'condensator' are used for capacitance and capacitor, respectively. In Figure 4.1 a capacitor is shown.

Figure 4.1: *A capacitor.*

The capacitor consists of two conductive plates which are separated by an insulator or dielectric. The **capacitance value** depends on the dielectric properties of the insulating material and is proportional to the surface area of the conductive plates and inversely proportional to the distance between the plates. The electrical charge is stored on the capacitor's plates (each plate stores equal charges with opposite signs). The capacitance value C is given in Farad (F) and electrical charge Q in Coulomb (C).

In Figure 4.2 schematic symbols for capacitances are shown.

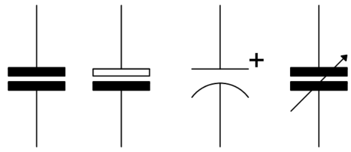


Figure 4.2: *Schematic symbols for capacitances.*

The left symbol in Figure 4.2 depicts a normal capacitance. The two symbols in the middle depict so called polarized capacitances. The right symbol depicts an adjustable capacitance. As can be seen, the polarized capacitances have different plates: the positive and negative plate. When used, the potential difference between the positive and negative plate should be positive. Polarized capacitors are often used for removing noise on DC sources.

When you build circuits and you need a capacitor, it is not always easy to check what its capacitance value is. For example, when a text on the capacitor is "104", this means that the capacitance value equals $10 \cdot 10^4$ pF $= 100$ nF $= 0.1$ μF. When the text is "12k", the value equals $12 \cdot 10^3$ pF $= 12$ nF. The tolerance of capacitors can be found in data sheets. Typical values for capacitance value that you will find are in the order of pF to μF. Capacitors are produced only having capacitances taken from the E-12 series.

4.2 Relation between voltage and current

Mathematically, the definition of capacitance can be written as follows:

$$\boxed{C = \frac{Q}{V}. \qquad (4.1)}$$

To obtain the relation between voltage and current, we first rewrite Equation 4.1:

$$\boxed{Q = C \cdot V. \qquad (4.2)}$$

Now realize that the current I (by definition) is the change in charge C per unit of time. We thus can say:

$$\boxed{I = \frac{dQ}{dt} = C\frac{dV}{dt}. \qquad (4.3)}$$

We call Equation 4.3 a differential equation. It tells us that for a capacitance the current depends on the change in voltage. It can be read as follows: when the voltage across a capacitance C of 1 F changes with 1 V per second, a current I of 1 A will flow.

The stored charge results in an electrical energy E, which depends on the potential difference between the plates and the capacitance value of the capacitor:

$$E = \frac{1}{2}CV^2. \tag{4.4}$$

The unit of energy is Joule.

? **Question 4.1**

A capacitor of 100 pF is connected to a DC-source of 12 V.
1. What is the total amount of charge stored on it?
2. What is the total amount of electrical energy is stored in it?

Now we de-charge this capacitor, then connect it to a constant current source and charge it with a constant current of 50 mA.

3. How long does it take to charge this capacitor to a voltage of 90 V?
(Note: Ampere is Coulomb per second!)

4.3 AC behavior

When connecting an AC voltage source to a capacitor and observing the potential difference between the plates and the current flowing through it, we can see a phase difference. This can be shown quite easily using Equation 4.3 and the assumption that the potential difference between the two plates has a sinusoidal shape: the derivative of a sine is a cosine, which is 90^o degrees ahead of the sine. For a sinusoidal current the phase difference between the capacitors current and voltage is shown in Figure 4.3.

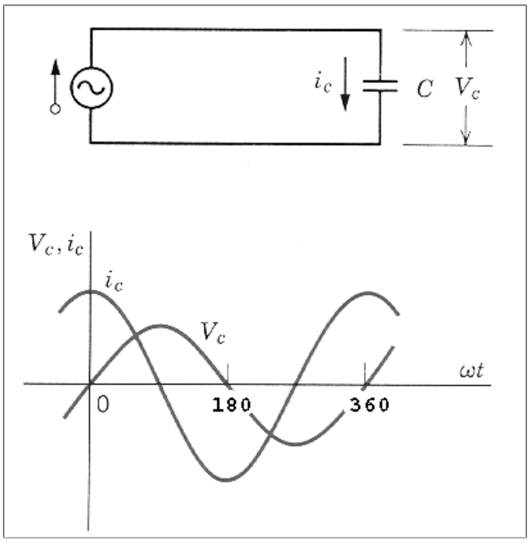


Figure 4.3: *For a capacitor, the current is 90^o ahead of the voltage.*

In Figure 4.3 it can be seen that, when the voltage increases/decreases the fastest (i.e. when it passes zero), the current is at its maximum/minimum. On the other hand, when the voltage is at its maximum or minimum, there is no current. Try to become familiar with this behavior.

> **!** **Note**
>
> In literature you can learn more about the 'complex' behavior of a capacitor. Because we do not want to introduce the required complex mathematics, this reader will discuss this component in a more informal way, if necessary giving equations without deriving them.

We introduce the term Reactance (X) to indicate the measure of the degree of opposition to AC currents an ideal capacitor is faced with. This reactance, with unit [Ω], is determined by the frequency of the current (f) and its capacitance (C).

$$\boxed{X_c(\omega) = \frac{1}{\omega C} = \frac{1}{2\pi f C}. \qquad (4.5)}$$

We can assume that Ohm's law will also hold for AC signals.

$$So,\ for\ reactance:\ X = \frac{Magnitude(V_{ac})}{Magnitude(I_{ac})} \tag{4.6}$$

$$Now\ let:\quad V_{ac} = sin(\omega)t,\ then\ \frac{dV}{dt} = \omega.cos(\omega)t \tag{4.7}$$

$$The\ current:\ i = C\frac{dV}{dt} = C.\omega.cos(\omega)t \tag{4.8}$$

So if the voltage across the capacitor is a sine wave with amplitude of 1 volt, then the current through the capacitor will be a cosine wave with an amplitude that is determined by ωC.

The fact the current is a cosine wave when the voltage is a sine wave tells us that the current in a capacitor lead the voltage across it by 90 degrees.

> **?** **Question 4.2**
>
> What will happen with the impedance X_c if:
>
> 1. The frequency f increases?
> 2. The frequency f decreases?

4.4 Power dissipation

For resistors we saw that the power was dissipated (electrical energy was transferred into heat). For capacitors this is not the case. Power is needed for building up the electrical field that exists between the positive and negative plates of the capacitor. For an ideal capacitor connected to an AC voltage source one can plot a diagram of the power dissipated as a function of time. Figure 4.4 shows that power is taken but given back the next quarter of a cycle. On average the power stays zero.

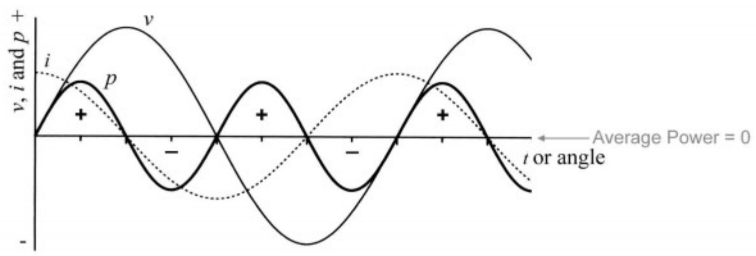


Figure 4.4: *Power in an AC capacitive circuit. Note that v, i and p are not plotted on the same scale.*

4.5 Connection: series and parallel

Just like resistance values, to get a desired value, it is also possible to connect capacitance values in series and parallel. In this section we will show how to calculate the replacement capacitance values for these connections.

Series connection

In Figure 4.5 a series connection of three capacitors is given. The current through all capacitors is the same. Also, the voltage drop across the three capacitors is equal to the sum of voltage drops across each capacitor.

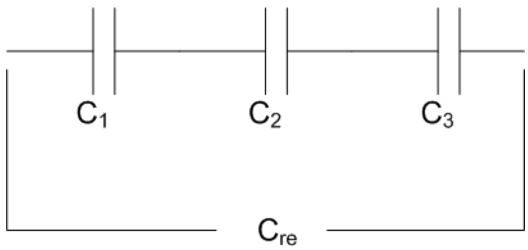


Figure 4.5: *Example of a series connection of three capacitors.*

We consider the situation where the capacitors are unloaded before we connect them to each other. To find the value of C_{re}, we need to realize that the charges on the plates are equal, but have an opposite direction. Take the charge on the left plate of C_1 Q, then the charge on the right plate has to be $-Q$. However, to account for the negative charge on this plate, the left plate of C_2 has to have a charge Q. This means that the charge on C_2's right plate equals $-Q$. Just like the left plate of C_2, the left plate of C_3 has a charge Q. The right plate of C_3 thus has a charge of $-Q$.

The replacement capacitor is thus charged with Q as well.

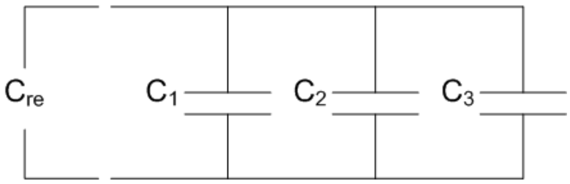


Figure 4.6: *Example of a parallel connection of three capacitors.*

We thus have: $Q = C_1 \cdot V_1 = C_2 \cdot V_2 = C_3 \cdot V_3 = C_{re}(V_1 + V_2 + V_3)$.

This means that $(\frac{Q}{C_1} + \frac{Q}{C_2} + \frac{Q}{C_3})C_{re} = Q$

And thus: $\frac{1}{C_{re}} = \frac{1}{C_1} + \frac{1}{C_2} + \frac{1}{C_3}$.

In general:

$$C_{re} = \frac{1}{\sum\limits_{i=1}^{N} \frac{1}{C_i}}. \tag{4.9}$$

Parallel connection

In Figure 4.6 a parallel connection of three capacitors is given. The replacement capacitance C_{re} is equal to the sum of all single capacitances: $C_{re} = C_1 + C_2 + C_3$. In general:

$$C_{re} = \sum_{i=1}^{N} C_i. \tag{4.10}$$

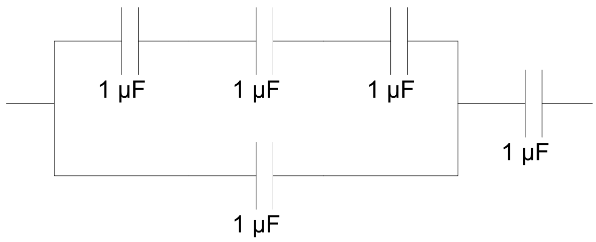
? **Question 4.3**

Derive that holds: $C_{re} = \sum\limits_{i=1}^{N} C_i$

Use the fact that the potential difference across each capacitor is the same, use Equation 4.1 and take the same steps as in the derivation of Equation 4.9 .

? **Question 4.4**

What is the equivalent capacitance of the circuit shown below?



4.6 Application

It was already mentioned that **polarized** capacitors can be used to remove noise on DC sources. This is not the only application of capacitors. Other applications even include capacitors constructed as sensors e.g. condenser microphones and humidity sensors. Furthermore, you will often see capacitors in filters and delay-units. These last two applications will be discussed in this section.

4.6.1 Capacitors and filters

In many applications it is necessary to filter out what we call 'frequency content'. Although we focussed on AC signals being sine waves with one frequency, in practice signals containing information are usually constructed of multiple frequencies and amplitudes. You may think of a bass

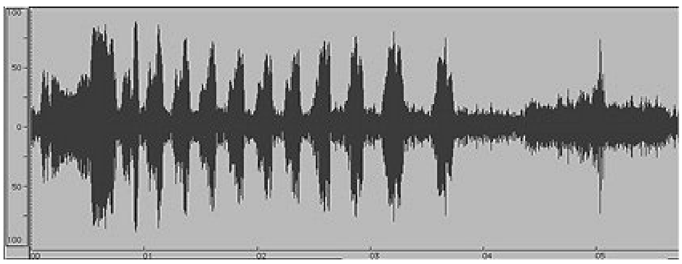


Figure 4.7: *Amplitude-time plot of a birdsong recording.*

loudspeaker: it needs to be controlled only by the low-frequency parts of the 'sound'-signal. So the high-frequency components need to be filtered out. In Figure 4.8 two identical simple (low-pass) filters which consist of a resistor and a capacitor are shown. They are only drawn in a different way. Try to regard them as frequency dependent voltage dividers. If necessary restudy section 3.6.

We first look to the total series impedance Z_{re} observed between the two input leads. Because the true impedance of a capacitance requires complex mathematics, we start with a different approach. It was already seen that the current through a capacitance is 90^o ahead of its potential difference. When nothing is connected to the output, the current through the resistance is the same as the current through the capacitance. This means that the potential differences across the resistance

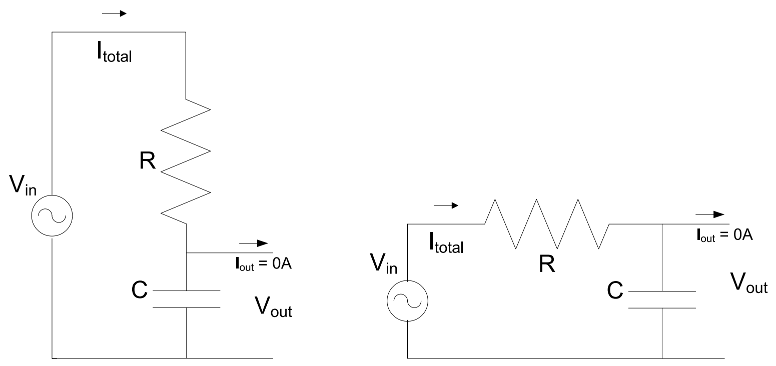


Figure 4.8: *Simple RC low-pass filters.*

and across the capacitance differ 90^o in phase. This is shown in Figure 4.9.

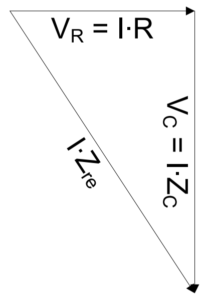


Figure 4.9: *For similar currents it holds that the vector of the potential difference across the resistance is perpendicular to the vector of the potential difference across the capacitance.*

According to the vectors drawn in Figure 4.9, we thus have a series impedance the modulus of which $|Z_{re}| = \sqrt{R^2 + X_c^2} = \sqrt{R^2 + \frac{1}{\omega^2 C^2}}$ which will increase for decreasing ω and decrease for increasing ω. When writing the output voltage V_{out} as a fraction of the input voltage V_{in}, we get:

$$\boxed{V_{out}(\omega) = \frac{Z_c}{Z_{re}} \cdot V_{in}(\omega) = \frac{\frac{1}{\omega C}}{\sqrt{R^2 + \frac{1}{\omega^2 C^2}}} \cdot V_{in}(\omega). \qquad (4.11)}$$

As can be seen, V_{out} does not only depend on V_{in}, but also on the frequency (compare this with the voltage divider derivation).

There are three typical frequencies:

1. $\omega = 0$;
2. $\omega = \omega_{cut-off}$;
3. $\omega \to \infty$.

When we substitute $\omega = 0$ in Equation 4.11, we can see that the transfer equals one. This means that DC-values are transferred fully by the filter. When ω increases to infinity, we can see that the signal is blocked, as V_{out} goes to zero. The last typical frequency ($\omega_{cut-off}$) is the frequency at which the input power is exactly divided by two and $\frac{V_{out}}{V_{in}} = \frac{1}{\sqrt{2}}$ and R $= X_C$. In Figure 4.10 the so called (amplitude) transfer-function is shown (for all frequencies). Note that both axes are logarithmic. Relations like V_{in}/V$_{out}$ (gain) on the vertical axis often are expressed in so-called Decibels.

A decibel is defined in two common ways.

When referring to measurements of power or intensity it is:

$$X_{dB} = 10 \log_{10} \frac{P_{out}}{P_{in}}. \tag{4.12}$$

But when referring to measurements of amplitude of voltages it is:

$$X_{dB} = 20 \log_{10} \frac{V_{out}}{V_{in}}. \tag{4.13}$$

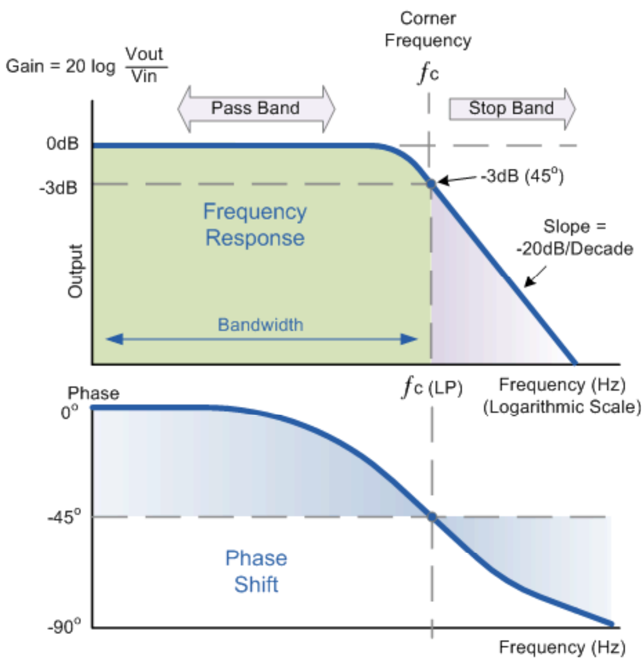


Figure 4.10: *Amplitude and phase transfer-function of a low-pass RC-filter.*

In Figure 4.10 the typical frequency f_c is indicated. Often, this frequency is called the -3dB frequency or the **cut-off frequency**. It is defined as the frequency where the input power is divided by two. Its value depends on R and C and at this frequency holds: R $= Z_c$ or:

$$\omega_{(cut-off)} = \frac{1}{RC} \; or: \; f_{-3dB} = \frac{1}{2\pi RC}. \tag{4.14}$$

Usually, filters, especially RC-filters, are specified by this frequency.

> **!** **Note**
>
> The used relations are just a simplification as we did not introduce complex mathematics. For example, phase shifts between the output voltage and input voltage are indicated but not calculated.

When the resistance and the capacitance are interchanged with each other, we obtain a different type of filter. Now the low frequencies appear attenuated on the output, while the high frequencies can pass the filter. Hence, we have a **high-pass** filter. In Figure 4.11 the amplitude transfer of such a filter for a high and a low frequency is given.

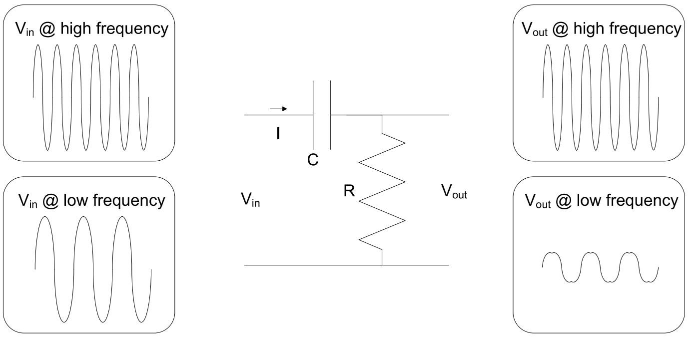


Figure 4.11: *Functionality of a simple high-pass RC-filter.*

The transfer function of such a high-pass filter is given below:

$$V_{out}(\omega) = \frac{R}{Z_{re}} \cdot V_{in}(\omega) = \frac{R}{\sqrt{R^2 + \frac{1}{\omega^2 C^2}}} \cdot V_{in}(\omega). \qquad (4.15)$$

> **?** **Question 4.5**
>
> Show by calculation that for both types of RC-filters at the cut-off frequency holds: $\frac{V_{out}}{V_{in}} = 0.707$

4.6.2 Capacitors and delay-units

Charging a capacitor takes time. This means that when you want to have a delay in your circuit (which you sometimes need in switch-logic), you can use capacitors.

The low-pass RC-filter that was derived in the previous subsection can be used to achieve a delay. The capacitor charges via the resistor. As the charge increases, the potential difference across the capacitor increases.(ref. equation 4.3). This potential difference will asymptotically increase to the level of the input voltage, but does not have this value instantaneously.

In Figure 4.12 an RC-filter is drawn which is used as a delay-unit. Also a sub circuit is drawn. From the sub circuit we know that the input current (I_x) is zero and that its output voltage will be 10 V when the input voltage (the potential difference across the capacitance) is higher than 5 V and 0 V when the input voltage is equal to or less then 5 V.

For calculations with this type of application we can use the fact that 50% of the potential difference across a capacitor when maximally charged will be reached in about $0.7RC$ seconds. The value is derived from the following relation between the potential difference across the capacitor $V_c(t)$ and the input voltage $V_{in}(t)$ and is given without derivation:

$$\boxed{V_c(t) = V_{in}(1 - e^{\frac{-t}{RC}}). \qquad (4.16)}$$

Suppose that we switch the input voltage of the schematic drawn in Figure 4.12 from 0 V to 10 V and after a while (when the potential difference across the capacitor is almost the input voltage) switch it back to 0 V. Now the output voltage V_{out} will be a delayed version of the input voltage. This is shown in Figure 4.13.

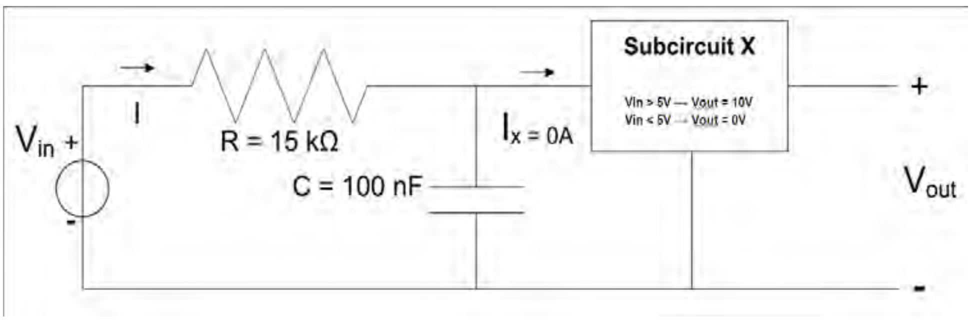


Figure 4.12: *Schematic of a digital delay-unit.*

? **Question 4.6**

Given the values for R and C from Figure 4.12 and the values for the voltages as indicated in the text, calculate the delay between V_{in} and V_{out}.

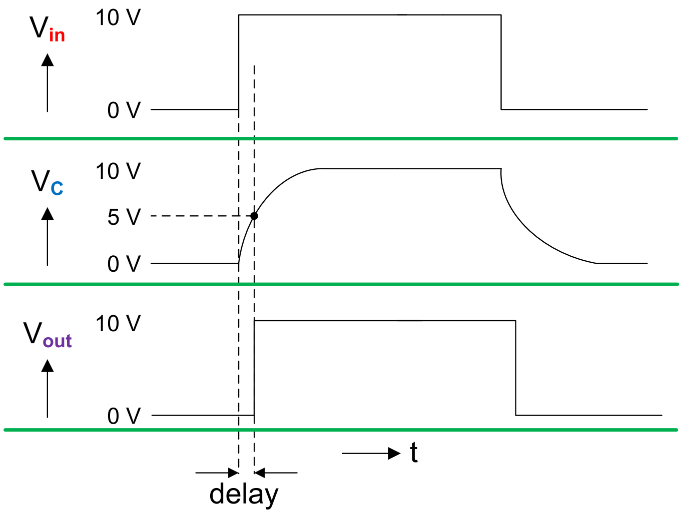


Figure 4.13: *Delay between input and output voltage due to delay-unit.*

Exercise 4.1

Again observe Figure 4.12:
Only based on Ohm's law, Kirchoff's Voltage Law, what you've learned about voltage dividers and the fact that holds:

$$I_{charge} = C\frac{dV_c}{dt} \tag{4.17}$$

reason why the potential difference across a capacitor charged by a voltage source through a resistor, like depicted in Figure 4.12, is **not** a linear function of time.

Hint: start reasoning the other way around: what must hold in (4.18) for $V_c = f(t)$ to be a linear function? Then start reasoning how I_{charge} develops after V_{in} is applied and t progresses.

5

Inductance and inductors

5.1 What is inductance and what are inductors?

Inductance (L) is defined as a measure of the amount of magnetic flux (Φ) produced for a given electric current.

This relation also works vice versa: a magnetic flux results induces an voltage (electro motive force) which on its turn can create an electric current in a closed circuit. Just like resistance and capacitance, inductance only has a conceptual meaning; when building circuitry, you cannot get them from the shelf. Instead, you need to get a physical component that 'implements' the definition of inductance: the inductor. In Dutch the words 'inductantie' and 'spoel' are used for inductance and inductor, respectively. In Figure 5.1 an inductor is shown.

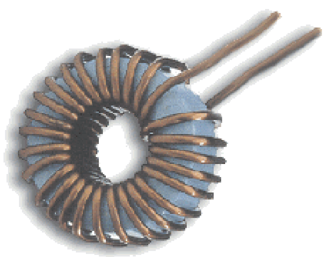


Figure 5.1: *An inductor.*

Basically, the inductor is just a piece of conducting wire which is winded. Often these windings are wrapped around so-called core material (iron, ferrite). This is done to enlarge the self-inductance of the inductor. The property of self-inductance is a particular form of electromagnetic induction. Self inductance is defined as the induction of a voltage in a current-carrying wire when the current in the wire itself is changing. In the case of self-inductance, the magnetic field created by a changing current in the circuit itself induces a voltage in the same circuit. Therefore, the voltage is self-induced. In Chapter 2 we already saw this when the dynamo was discussed. The inductance value of an inductor is given in Henry (H) and magnetic flux in Weber (W).

Typical values for inductance value that you will find are in the order of μH to mH and come in the E12 value series. The **inductance value** of inductors (indicated by the letter L) depends on the number of windings, the diameter of the windings, the length and the amount and type of core material.

Inductors exist in different sizes and shapes. Often you will see that the DC resistance of the inductor is mentioned. This resistance is due to the non-zero resistance of the wire used in the windings. Depending on the application (high vs. low frequencies, high vs. low power) the core material mentioned earlier is needed. The core often has the shape of a bar or a ring.

In Figure 5.2 schematic symbols for inductors are shown.

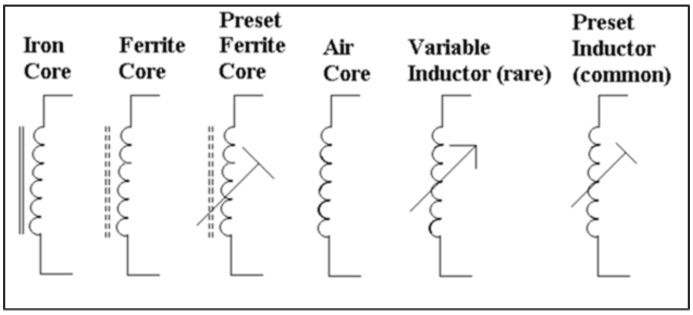


Figure 5.2: *Schematic symbols for inductance.*

5.2 Relation between voltage and current

Mathematically, the definition of inductance can be written as follows:

$$L = \frac{\Phi}{I}. \tag{5.1}$$

To obtain the relation between voltage and current, we first rewrite Equation 5.1:

$$\Phi = L \cdot I. \tag{5.2}$$

Now realize that the voltage V is the change in flux Φ per unit of time. We can thus say:

$$V = \frac{d\Phi}{dt} = L\frac{dI}{dt}. \tag{5.3}$$

Equation 5.3 can be read as follows: when the current through an inductance L of 1 H changes with 1 A per second, there will be a potential difference across the inductor of 1 V.

Note that capacitances behave electrically opposite to inductances: for capacitances the current depends on changes in the voltage, whereas for inductances the voltage depends on changes in the current. Furthermore, instead of electrical energy, magnetic energy is stored:

$$E_{magnetic} = \frac{1}{2} \cdot LI^2 \tag{5.4}$$

Question 5.1

When a current flows through an inductor and the current is suddenly cut off, this can damage your circuit. Explain why this can happen.
Hint: use Equation 5.3

5.3 AC behavior

When connecting an AC voltage source to an inductor and observing the potential difference across and the current flowing through it, we can see a phase difference. This can be shown quite easily using Equation 5.3 and the assumption that the potential difference across it has a sinusoidal shape: the derivative of a sine is a cosine, which is 90^o degrees ahead of the sine. For a sinusoidal voltage the phase difference between the inductors current and voltage is shown in Figure 5.3.

So contrary to the capacitor, the potential difference across the inductor is 90^o ahead of the current that flows through it. This is depicted in Figure 5.3.

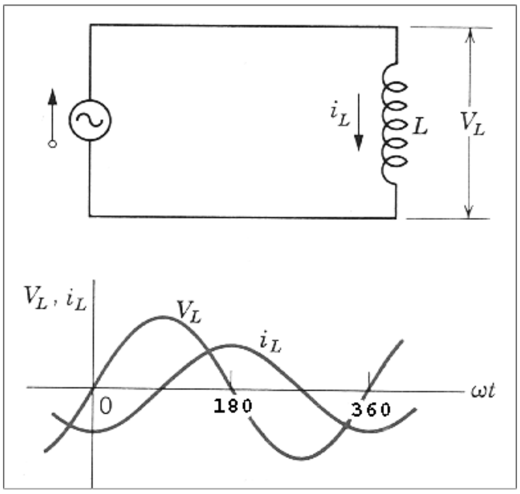


Figure 5.3: *For an inductor, the voltage is 90° ahead of the current.*

In Figure 5.3 it can be seen that, when the current increases/decreases the fastest (i.e. when it passes zero), the voltage is at its maximum/minimum. On the other hand, when the current is at its maximum or minimum, there is no voltage. Try to become familiar with this behavior.

> **!** **Note**
>
> In literature you can learn more about the 'complex' behavior of an inductor. Because we do not want to introduce the required complex mathematics, this reader will discuss this component in a more informal way, if necessary giving equations without deriving them.

We already introduced the term Reactance (X), in this case to to indicate the measure of the degree of opposition to AC currents an ideal inductor is faced with. This reactance, with unit $[\Omega]$, is determined by the frequency of the current (f) and its capacitance (C).

$$\boxed{X_L(\omega) = \omega L \ = 2\pi f L \qquad (5.5)}$$

We can assume that Ohm's law will also hold for AC signals.

$$So,\ for\ reactance:\ X = \frac{Magnitude(V_{ac})}{Magnitude(I_{ac})} \tag{5.6}$$

$$Now\ let:\ \ I_{ac} = sin(\omega)t,\ then\ \frac{dI}{dt} = \omega.cos(\omega)t \tag{5.7}$$

$$The\ voltage:\ v = L\frac{dI}{dt} = C.\omega.cos(\omega)t \tag{5.8}$$

So if the voltage across the inductor is a cosine wave with amplitude of 1 volt, then the current through the inductor will be a sine wave with an amplitude that is determined by ωL.

The fact the current is a sine wave when the voltage is a cosine wave tells us that the voltage across an inductor leads it's current by 90 degrees.

> **?** **Question 5.2**
>
> What will happen with the impedance X_L if:
>
> 1. The frequency f increases?
> 2. The frequency f decreases?

5.4 Power dissipation

For resistors we saw that the power was dissipated (electrical energy was transferred into heat). Also for ideal inductors this is not the case. For an ideal inductor connected to an AC voltage source one can plot a diagram of the power dissipated as a function of time. Figure 5.4 shows that power is taken but given back the next quarter of a cycle. On average the power stays zero.

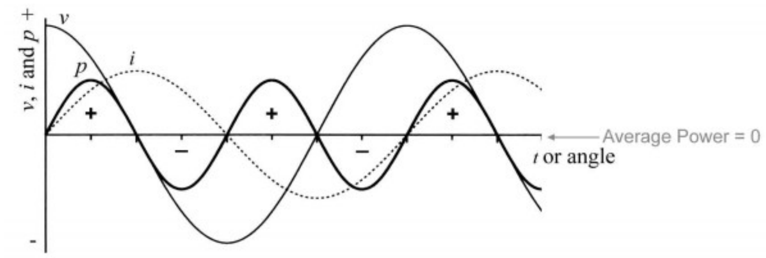


Figure 5.4: *Power in an AC inductive circuit. Note that v, i and p are not plotted on the same scale.*

5.5 Series and parallel connections

For series and parallel connections of inductor we can use the same formulas as for resistors. In fact, like we saw with the impedances for capacitances and resistances, we in fact apply formulas for series and parallel connections of impedances on resistances, capacitances and inductances.

Series connection

In Figure 5.5 a series connection of three inductors is given. The replacement inductor has an inductance value $L_{re} = L_1 + L_2 + L_3$. In general:

$$\boxed{L_{re} = \sum_{i=1}^{N} L_i. \qquad (5.9)}$$

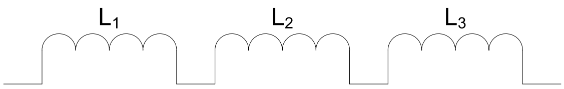


Figure 5.5: *Example of a series connection of three inductors.*

Parallel connection

In Figure 5.6 a parallel connection of three inductors is given. The replacement inductor has an inductance value $\frac{1}{L_{re}} = \frac{1}{L_1} + \frac{1}{L_2} + \frac{1}{L_3}$. In general:

$$\boxed{L_{re} = \frac{1}{\sum_{i=1}^{N} \frac{1}{L_i}}. \qquad (5.10)}$$

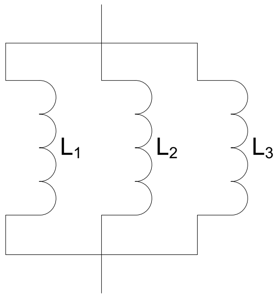


Figure 5.6: *Example of a parallel connection of three inductors.*

? Question 5.3

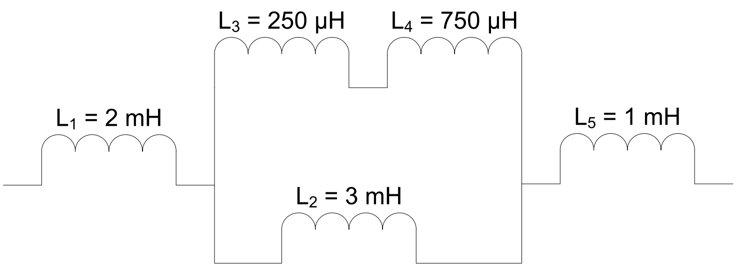
If you consider the parallel connection of three inductors with $L_1 = 0.33$ mH, $L_2 = 68$ μH and $L_3 = 12$ mH.

What can you tell, without calculating it, about the replacement inductance L_{re}? So, which of the three answers is correct and why is it correct ?

1. $L_{re} > 12$ mH
2. $L_{re} < 68$ μH
3. 68 μH $< L_{re} < 0.33$ mH

? Question 5.4

What is the equivalent inductance L_{re} of the circuit shown below?



5.6 Application

Inductors are often used for application in filters, relays, transformers and speakers. In this section these applications will be discussed.

5.6.1 Inductors and filters

In the previous chapter we saw that capacitors and resistors together can be used for filtering signals. Since the impedance of the inductor depends on the frequency as well, a combination of resistors and inductors can also be used to filter signals by creating a frequency dependent voltage divider. In Figure a circuit of a so called RL-filter is drawn.

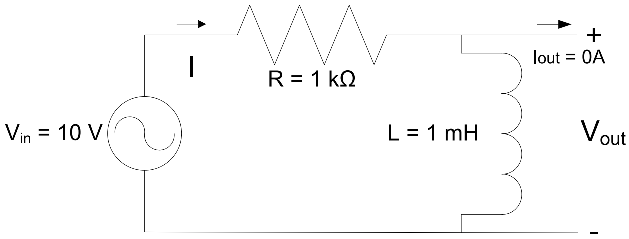


Figure 5.7: *Schematic of a simple RL-filter.*

Like with the RC-filter, we can start with calculating the modulus of the series impedance $|Z_{re}| = \sqrt{R^2 + X_L^2} == \sqrt{R^2 + \omega^2 L^2}$. The voltage division then results in:

$$\boxed{V_{out}(\omega) = \frac{X_L}{Z_{re}} \cdot V_{in}(\omega) = \frac{\omega L}{\sqrt{R^2 + \omega^2 L^2}} V_{in}(\omega). \qquad (5.11)}$$

? **Question 5.5**

1. Is the filter drawn in Figure 5.7 a high-pass or a low-pass filter?
2. What do you get by interchanging the position of L and R in Figure 5.7: a high-pass or a low-pass filter?
3. Mathematically derive the equation which describes the input-output relation (like Equation 5.11) of this newfound filter.

? **Question 5.6**

Calculate the current I in the circuit drawn in Figure 5.7 at 3 different V_{in} frequencies:

1. 100 kHz
2. 0 Hz
3. approaching infinity

In the previous chapter we introduced the cut-off frequency of an RC-filter. Such a frequency also exists for an RL-filter. Now its value depends on R and L and at this frequency holds: R = X_L.

$$\boxed{\omega_{(cut-off)} = \frac{R}{L} \; or: \; f_{-3dB} = \frac{R}{2\pi L}. \qquad (5.12)}$$

Besides RC/RL-filters, also LC- and RLC-filters exist. For example, in a basic loudspeaker box you will probably find two LC-filters, like drawn in Figure 5.8.

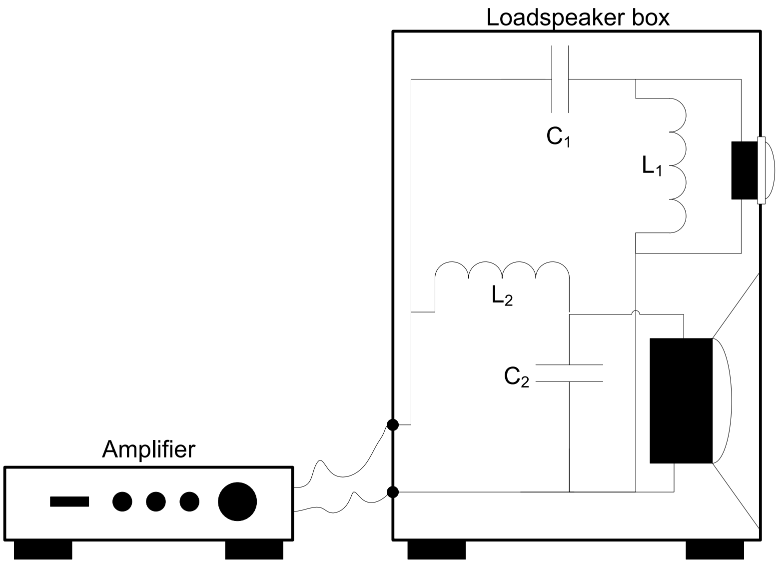


Figure 5.8: *A loudspeaker box with two LC-filters. One filter is used to pass high frequencies and the other is used to pass low frequencies.*

One filter is used to filter out the high frequencies and the other filter filters out the low frequencies. The inductance and capacitance values are chosen in such a way that there is a smooth transition from the low tone (frequency) area to the high tone area.

LC/RCL-filters contain a factor ω^2 in the denominator of their transfer functions and their transfer will therefore decrease/increase much faster (for decreasing/increasing frequency) than that of RC/RL-filters. In Figure 5.9. two low-pass transfer functions are drawn. The steepest graph belongs to a LC-filter the other one to a RC or LR-filter.

5.6.2 Inductors and mechanics

Besides filters, inductors are often used to transform mechanical energy into electrical energy (generators) and vice versa (motors). Another example with which you might also be familiar is the relay. When a magnetic field results from a current through the inductor, this field is used to switch a lever. Relays are mostly used as a distant switch or for switching large currents or voltages (you will often hear a ‘click’ when a relay switches, for example, when the doors of the elevator start opening) which need to be electrically separated from the circuit that operates the circuit (e.g. a

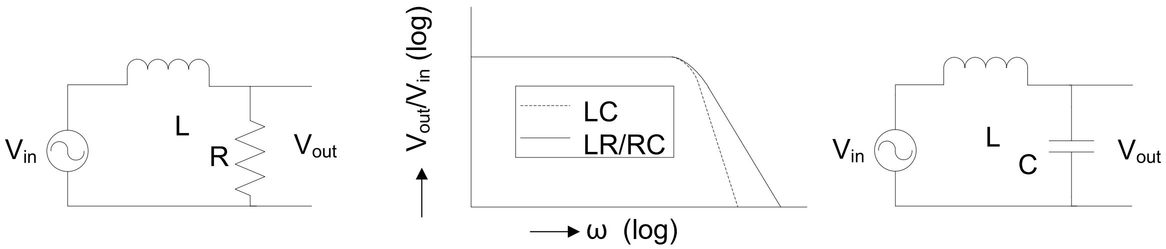


Figure 5.9: *The (amplitude) transfer of a LC/CL filter changes faster (is steeper) than a LR/RL/RC/CR filter.*

computer is not electrically connected to the elevator doors). In Figure 5.10 you can see how a relay is used to switch Circuit 2 using Circuit 1.

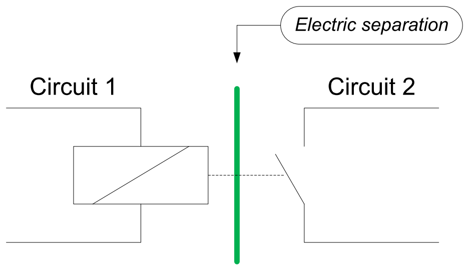


Figure 5.10: *A relay is used to switch Circuit 2 using Circuit 1.*

5.6.3 Inductors and transformers

A last type of application which require inductors are transformers. In their simplest form, transformers are components that consist of two electrically separated coils (primary and secondary) that are placed close to each other. In this way the magnetic fields are coupled and therefore energy can be transported from the primary to the secondary coil. An alternating voltage that is applied over the primary coil appears across the secondary coil as a secondary voltage that is equal to the input voltage multiplied with a winding factor (the ratio between the primary and secondary number of windings) and a current that is equal to the input current multiplied with the inverse of the winding factor. The coils are often wrapped around a common core to achieve a better energy transfer.

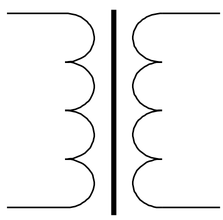


Figure 5.11: *Schematic of a transformer.*

In electric machines you will often find transformers which serve two important purposes:

- transformation of 230 V (RMS!) to a useful (mostly much lower) voltage that the machine can use;
- separation of the machine from the electricity grid (for reasons of safety) because the primary and secondary coil are electrically separated.

6

Kirchoff's laws

When you need to analyze complex electrical circuits (consisting of several components and having several nodes) just applying Ohm's law will not help you in calculating all the currents and potential differences inside the circuit. To be able to analyze those circuits you need Kirchoff's current and voltage laws which will be discussed in this chapter. Together with what you have seen in the preceding chapters, they will enable you in analyzing complex circuits consisting of resistors, capacitors and inductors.

6.1 Kirchoff's current law

Kirchoff defined a law for currents to be analyzed in a circuit. It is usually referred to as 'KCL'.

KCL - *The sum of currents entering a node is equal to the sum of currents leaving a node.*

The physical meaning of this law is that current cannot accumulate in a node: what goes in must come out.
This can be exemplified using tubes and marbles. When marbles enter a crossing of tubes, they do not stay at the crossing, but are pushed to a tube where they can leave the crossing. At each time instance the amount of marbles entering is equal to the amount of marbles leaving, as they cannot stay at the crossing.

For the KCL we can write:

$$\boxed{\sum I_{incoming} - \sum I_{outgoing} = 0. \qquad (6.1)}$$

In Figure 6.1 Equation 6.1 is illustrated. The incoming currents $I_{in1..n}$ split into the outgoing currents $I_{out1..n}$. According to KCL we have $I_{in1} + I_{in2} + ... + I_{inn} = I_{out1} + I_{out2} + ... + I_{outm}$.

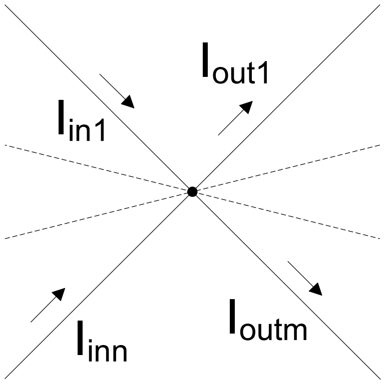
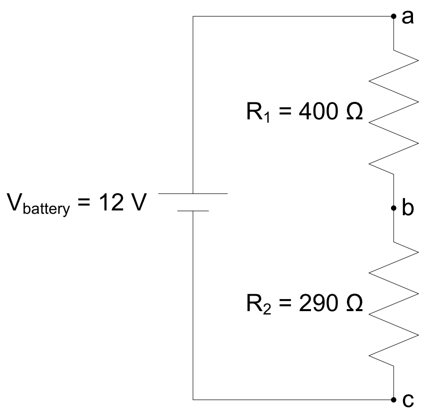


Figure 6.1: *Kirchoff's Current law.*

? Question 6.1

Find the potential for node b in the schematic below using KCL (this means that you should use the incoming and outgoing currents at node b). You may take node c as a reference.



6.2 Kirchoff's voltage law

Besides a law for current, Kirchoff defined a law for voltages as well. It is usually referred to KVL. Before we introduce it, we need to define closed loops in circuits. A closed loop is just a closed path which begins and ends in the same node of a circuit. In Figure 6.2 an example of a circuit is shown. Two closed loops are indicated.

KVL - *The sum of the branch voltage drops around any closed loop is zero.*

The idea behind KVL can be best explained using heights. If you are on the top of a mountain and you descent in several steps to ground level, you will have several drops of height. However if you then ascend to the top in several steps, you will have several (negative) drops of height. In total,

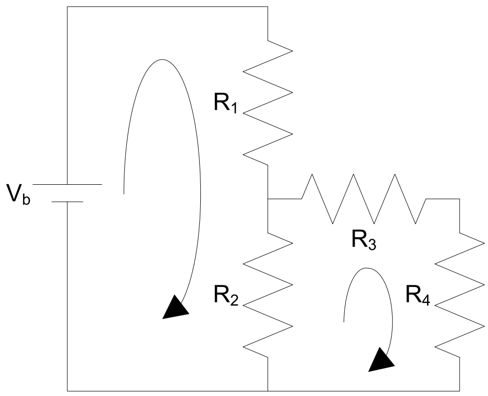


Figure 6.2: *A circuit with two closed loops indicated.*

you descended x meter. Since the top of the mountain did not change in height, you ascended x meter as well. In total, the difference between your starting and ending heights is zero meter!

Mathematically, KVL can be defined as follows:

$$\sum V_{drops\ in\ a\ closed\ loop} = 0 \tag{6.2}$$

? **Question 6.2**

Can you think of another closed loop in the circuit shown in Figure 6.2?

Example 6.1

In Question 6.1 a simple circuit containing one closed loop is given. The current I that will flow through the loop can be calculated using Ohm's law: $I = \frac{V_{battery}}{R_{re}} = \frac{V_{battery}}{R_1+R_2} = \frac{12}{690} = 0.0174$ A. Using the currents, we can calculate the voltage drops across the resistors: $V_{ab} = I \cdot 400 = 6.96$ V, $V_{bc} = I \cdot 290 = 5.04$ V. We also have a voltage drop across the battery, which is equal to V_{ca} if we start our loop in node a and we loop clockwise: $V_{ca} = -V_{battery} = -12$ V. If we now sum up all voltage drops we get: $V_{ab} + V_{bc} + V_{ca} = 6.96 + 5.04 + -12 = 0$ V!

? **Question 6.3**

Write out the KVL for all closed loops of the circuit shown in Figure 6.2.

Until now we chose the polarities intuitively. However, sometimes it can be that a certain potential difference is defined as the negative of a positive voltage drop. In these cases you should account for this fact by multiplying the potential difference by -1 ('all drops should face the same direction'). This is indicated in Figure 6.3.

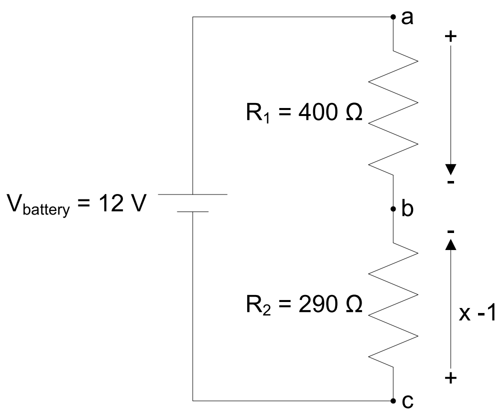


Figure 6.3: *All potential differences in a KVL should 'face the same direction'. This can be achieved by multiplying the potential differences that face the wrong direction with '-1'.*

6.3 Applying KCL and KVL to a circuit

In Figure 6.4 a circuit is shown that cannot just be analyzed by using the series and parallel replacement resistors. If we want to know all the currents and voltage drops, we need to apply KCL or KVL. Both methods must give the same result. Here we will show how to apply KCL and KVL in a structured manner.

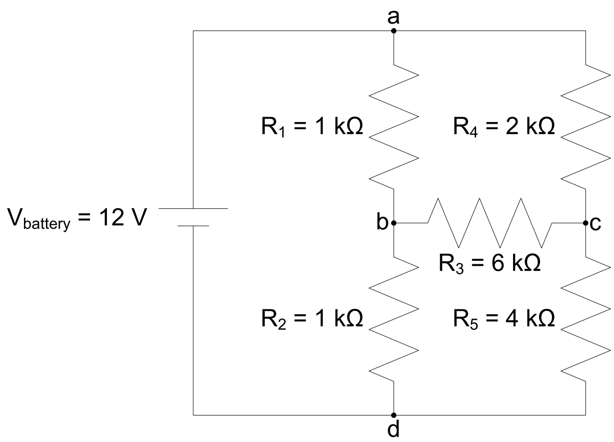


Figure 6.4: *A more complex circuit.*

6.3.1 KCL

First we need to check how many nodes we have. In this case there are four nodes: a, b, c, and d. Notice that nodes b and c are special nodes. We choose node d to be the reference node.

Now we need to define the direction of all currents. The current will flow from node a to node c and node b. From node b and c it flows to node d. Now we have one current left: from node b to node c or vice versa. In fact, choosing the wrong direction for this current is not a big issue: if we choose the wrong direction, the current as found by the solution will only be negative (indicating that we chose the wrong direction).

The next step involves applying KCL to the nodes. For node b we have one incoming current and two outgoing currents:

$I_{R_1} = \frac{V_a - V_b}{R_1} = \frac{12 - V_b}{1000}$

$I_{R_2} = \frac{V_b - V_0}{R_2} = \frac{Vb - 0}{1000}$

$I_{R_3} = \frac{V_b - V_c}{R_3} = \frac{V_b - V_c}{6000}$

By applying KCL, we now find an equation: $I_{R_1} - I_{R_2} - I_{R_3} = 0 \rightarrow \frac{3}{250} - \frac{13}{6000} V_b + \frac{1}{6000} V_c = 0$.

For node c we have two incoming currents and one outgoing current:

$I_{R_3} = \frac{V_b - V_c}{R_3} = \frac{Vb - V_c}{6000}$

$I_{R_4} = \frac{12 - V_c}{R_4} = \frac{12 - V_c}{2000}$

$I_{R_5} = \frac{V_c - V_d}{R_5} = \frac{V_c - 0}{4000}$

Applying KCL results in a second equation:: $I_{R_3} + I_{R_4} - I_{R_5} = 0 \rightarrow \frac{3}{500} - \frac{1}{6000} V_b + \frac{11}{12000} V_c = 0$.

We now have to equations with two unknown variables. When we solve this system, we find the values for V_b and V_c: $V_b = 6.1277$ V and $V_c = 7.6596$ V. With these two potentials, we can calculate all currents and voltages. Note that we chose the wrong direction for the current between nodes b and c as $V_b < V_c$. and thus $I_{bc} < 0$.

6.3.2 KVL

First we need to find the loops in the circuit. There are six loops in this circuit:

1. a → b → d → a;
2. a → d → c → b → a;
3. a → d → b → c → a;
4. a → d → c → a;
5. a → c → b → a;
6. b → d → c → b.

We do not need to apply KVL to all loops, because they give redundant information. Therefore, we choose the first loop and the last two loops. We will call the current in the first loop I_1, the current in the fifth loop I_2, and the current in the sixth loop I_3. The currents are indicated in Figure 6.5. With the currents we can calculate the voltage drops across the resistors for each loop.
$R_1 = 1k\Omega$, $R_2 = 1k\Omega$, $R_3 = 6k\Omega$, $R_4 = 2k\Omega$, $R_5 = 4k\Omega$.

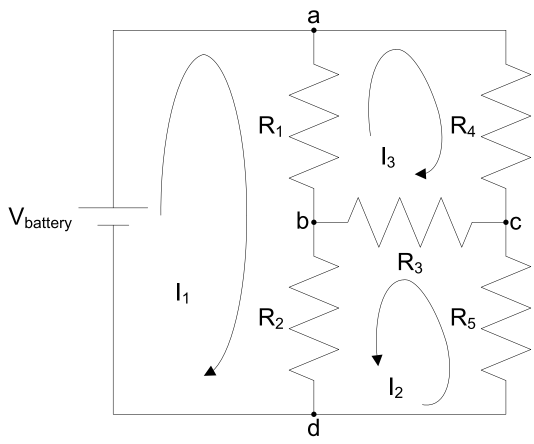


Figure 6.5: *A more complex circuit with the currents used in the calculation described below.*

Loop $\mathbf{I}_1 : a\rightarrow \mathbf{b} \rightarrow \mathbf{d} \rightarrow \mathbf{a}$

$$1000(I_1 - I_3) + 1000(I_1 + I_2) - 12 = 0 \rightarrow 12 - 2000I_1 - 1000I_2 + 1000I_3 = 0$$

Loop $\mathbf{I}_3 : a\rightarrow \mathbf{c} \rightarrow \mathbf{b} \rightarrow \mathbf{a}$

$$2000I_3 + 6000(I_2 + I_3) + 1000(I_3 - I_1) = 0 \rightarrow -1000I_1 + 6000I_2 + 9000I_3 = 0$$

Loop $\mathbf{I}_2 : b\rightarrow \mathbf{d} \rightarrow \mathbf{c} \rightarrow \mathbf{b}$

$$4000I_2 + 6000(I_2 + I_3) + 1000(I_1 + I_2) = 0 \rightarrow 1000I_1 + 11000I_2 + 6000I_3 = 0$$

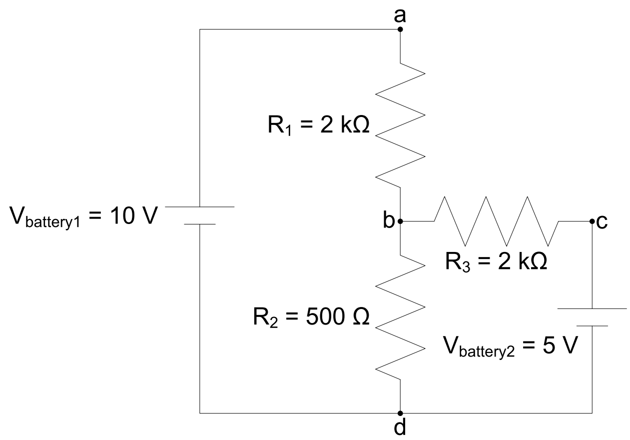
With the three equations, we can find values for the three unknown currents I_1, I_2, and I_3: $I_1 = 0.008$ A, $I_2 = 0.0019$ A and $I_3 = -0.0022$ A (so we chose the wrong direction for I_3). Using these currents and Ohm's law, we can calculate the values for V_b and V_c: $V_b = 6.1$ V and $V_c = 7.6$ V.

The difference in the results of both methods is due to rounding errors.

?

Question 6.4

1. Apply KCL to determine the voltages at the nodes of the circuit below.
2. Apply KVL to determine the voltages at the nodes of the circuit below.
3. Verify that both yield the same solution!



Tip: draw arrows that indicate the current in the various loops and take node ***d*** *as reference..*

7

Norton and Thevenin

Like we already saw while replacing series and parallel connections, it is more difficult to analyze complicated circuits containing many components than it is to analyze simpler circuits containing less components. We always try to simplify a circuit using simple concepts. This results in fewer equations which are more easy to solve (and usually they are faster to solve).

When you look to two of the nodes of a circuit, it is possible to represent all sources and components of that circuit by a single (independent) source and an impedance. In Figure 7.1 the concept of the (complicated) circuit of which you take two nodes (A and B) is shown. Calculation of the values of the source and impedance is straightforward: you only need to know the open circuit voltage V_{oc} that you can measure between the two nodes and the short circuit current I_{oc} that you can measure when you connect the two nodes to each other. In order to measure V_{oc}, you take resistance R infinite and measure the voltage drop across the resistance. Likewise, in order to measure I_{sc}, you take resistance R zero Ohm and measure the current through the resistor.

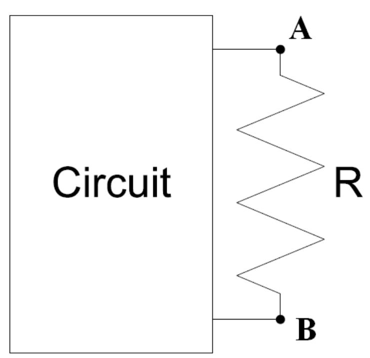


Figure 7.1: *You can take two arbitrary nodes out of the circuit.*

The terminal characteristics of the circuit and the two arbitrary nodes shown in Figure 7.1 can be described by either its Thevenin equivalent or its Norton equivalent. The Thevenin equivalent consists of a voltage source with value V_{oc} in series with a resistance R_T. The Norton equivalent consists of a current source with value I_{sc} in parallel with a resistance R_N. In Figure 7.2 the Thevenin and Norton equivalents are shown.

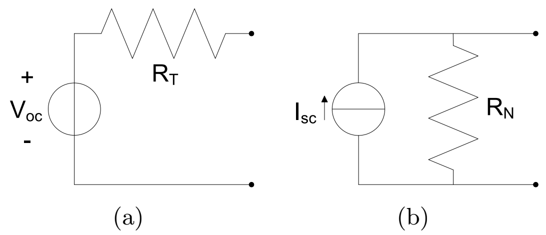


Figure 7.2: *Equivalent circuits: (a) Thevenin and (b) Norton.*

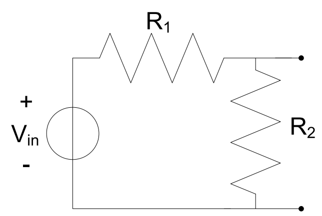
The resistances R_T and R_N can be calculated as follows:

$$\boxed{R_T = R_N = \frac{V_{oc}}{I_{sc}}. \qquad (7.1)}$$

Due to Equation 7.1, knowledge of two parameters suffices to determine all three parameter values. The open circuit voltage and closed circuit current can be measured easily. Their ratio determines the equivalent resistance. Alternatively, the resistance can be found directly by determining the resistance across the two terminals when all independent sources in the circuit have been replaced by their dead (zero-valued) equivalents. This means that voltage sources are short circuited and that current sources are open circuits.

? **Question 7.1**

Given the following circuit.



1. Calculate the open circuit voltage V_{oc}.
2. Calculate the short circuit current I_{sc}.
3. Calculate the equivalent resistance R_T/R_N by applying Equation 7.1.
4. Draw the equivalent Thevenin circuit and indicate the values for V_{oc} and R_T.

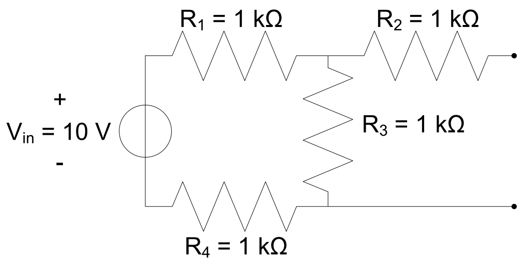
By 'calculating' we mean: express in V_{in}, R_1 and R_2.

Due to Equation 7.1, both equivalents can be interchanged. To do this, you need the following rules:

- Thevenin and Norton resistances are equal;
- The Thevinin voltage is equal to the Norton current times the Norton resistance;
- The Norton current is equal to the Thevenin voltage divided by the Thevenin resistance.

? **Question 7.2**

Given the following circuit.



Determine the Thevenin equivalent circuit paramaters of this circuit.
(*so calculate* V_{oc} *and* R_T *and draw the Thevenin equivalent schematic.*)

8

Diodes

8.1 What is a diode?

A diode is a one way conductor. This means that it ensures uni-directional current: it blocks current in one direction and lets the current pass in the other direction. Contrary to the components we have discussed until now, the diode is not linear. With linear we mean that when the voltage voltage is doubled, the current is doubled as well. In Figure 8.1 several diodes are drawn. All of these diodes are Light Emitting Diodes (LEDs): when current passes light is emitted. You are probably familiar with this type of diodes, since they are all around us (e.g. power indicators on your computer and lighting for your bicycle).

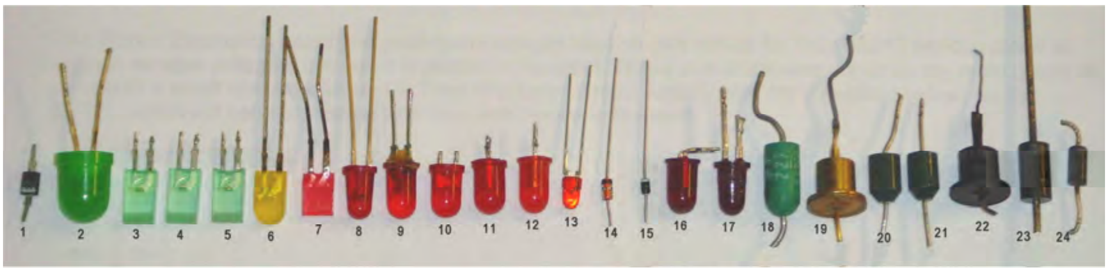


Figure 8.1: *Several diodes and LEDs (Light Emitting Diodes).*

The schematic symbol for a diode is drawn in Figure 8.2. The big arrow in the symbol indicates the 'pass direction'.

Figure 8.2: *Schematic symbol for a LED and a diode.*

It was already said that the relationship between the voltage across the diode (V_d) and the pass current (I_d) is not linear. Ideally, the diode would block all current whenever the voltage across it

(V_d) is negative. When conducting, the current through the diode (I_d) could be any value > 0 A and there should not be no voltage across the diode. In practice, however, the relationship between V_d and I_d is slightly different. In Figure 8.3 a typical example of this relationship is drawn.

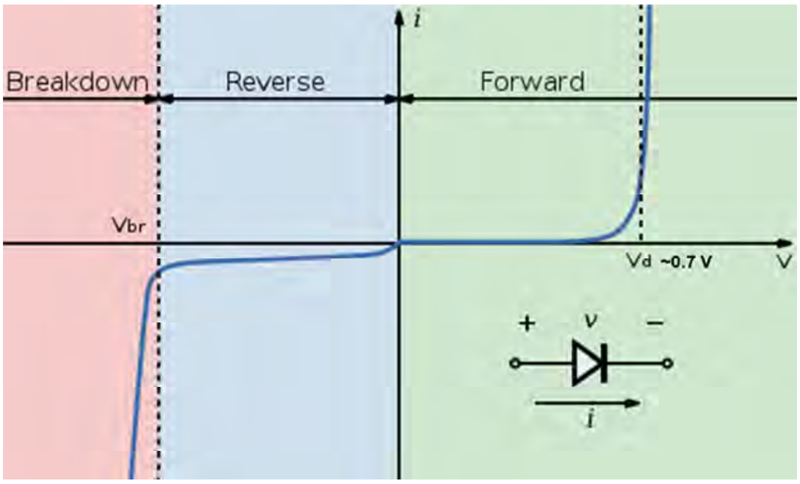


Figure 8.3: *Typical relationship between the voltage across a diode (V_d) and the pass current (I_d). Note that the forward and the reverse part have a different scale.*

In the 'forward' part of the relationship drawn in Figure 8.3 the so called 'Knee' voltage of the diode is indicated. This is the point at which the diode really starts conducting. For silicon diodes, this voltage usually equals ˜0.7 V. The pass current then is around 10 mA. Instead of the ideal situation where there is no voltage across the diode once it is conducting, we thus have a slight positive value. Furthermore, when we increase this voltage, we see that the current increases much faster. In fact, this increase is approximately exponential. Therefore, you should not use too high voltages across the diode or limit the current. In the 'reverse' direction hardly any current flows. The figure indicates what is called leakage current. At a certain reverse voltage (Vbr) the diode breaks down and the current increases rapidly. In most of these situations the diode will be broken.

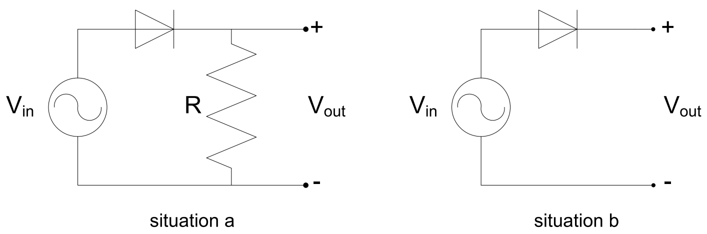
> **!** **Note**
>
> When using a diode in a circuit, make sure that the voltage across the diode does not become much higher than the 'knee' voltage: the energy dissipated in the diode (which is equal to $V_d \cdot I_d$ can become too high easily, thereby destroying the diode).

8.1.1 Background: the P-N junction

The diode consists of a series connection of P doped material and N doped material. Doped material is plain material to which extra charge carriers are added, which can move easily (either holes (P material) or electrons (N material). The behavior of the diode can be ascribed to the so-called depletion layer or depletion zone which exists at the junction of so-called P material and N material. When a P-N junction is first created, conduction band (mobile) electrons from the N-doped region diffuse into the P-doped region where there is a large population of holes with which the electrons "recombine". When a mobile electron recombines with a hole, the hole vanishes and the electron is no longer mobile thus, two charge carriers have vanished. The region around the P-N junction becomes depleted of charge carriers and thus behaves as an insulator. However, the depletion width cannot grow without limit. For each electron-hole pair that recombines, a positively-charged dopant ion is left behind in the N-doped region, and a negatively charged dopant ion is left behind in the P-doped region. As recombination proceeds and more ions are created, an increasing electric field develops through the depletion zone which acts to slow down and then finally stop recombination. At this point, there is a 'built-in' potential across the depletion zone. If an external voltage is placed across the diode with the same polarity as the built-in potential, the depletion zone continues to act as an insulator preventing a significant electric current. However, if the polarity of the external voltage opposes the built-in potential, recombination can once again proceed resulting in substantial electric current through the P-N junction. For silicon diodes, the built-in potential is approximately 0.6 V (the 'Knee' voltage!). Thus, if an external voltage of about 0.7 V is applied to the diode such that the P-doped region is positive with respect to the N-doped region, the diode is 'turned on' allowing an electric current.

? **Question 8.1**

Given the following circuit.



The signal of the AC-source is a sine wave.
- Sketch the course of the output signal $V_{out} = f(t)$ for both situations. Assume V_{in} to be $2V_{top-top}$. Also assume no breakdown problems. Now indicate the top values of V_{out} in both situations.

8.2 Application

Diodes are used for lots of applications. In this section we will discuss three of the most common application types.

8.2.1 Diodes and AC voltage rectifiers

Probably one of the most important applications of the the diode is using it to rectify AC voltages. This rectification is the first step in converting AC to DC. Different types of rectifier exist. The rectifier shown in the previous question is the most simple one: the single-sided rectifier. Only positive currents can pass, so the rectification consists of passing through positive values and blocking negative values. There also exist more complex rectifiers. They let pass the positive values and rectify the negative values into their positive equivalents, which can pass as well. So if the input signal of the rectifier is a sine wave, the output signal will be the absolute value of that sine wave (depending on the construction we, of course, have to take into account one or more forward diode voltage drops). These rectifiers are called bridge rectifiers. For further explanations you may search the Internet or Chapter 16.

8.2.2 Diodes and 'OR'-circuits

Diodes can also be used to let the highest of two voltages pass without influencing the other voltage (level). A situation in which this may be important, is drawn in Figure 8.4. Here a timing device is shown that is not allowed to stop when the supply voltage fails.

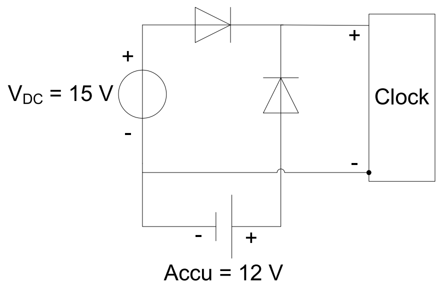


Figure 8.4: *Typical 'OR'-circuit: whenever the source V_{DC} fails, the accu will take over.*

As long as the 15 V is maintained by the source, there cannot flow any current through the battery. However, when the DC-source fails (i.e. the supply voltage drops below 12 V minus the diode knee voltage), the accu takes over.

The circuit drawn in Figure 8.4 is called an 'OR'-circuit: *either* the DC-source *or* the accu is used.

8.2.3 Diodes and voltage limiters

Inputs of electronic circuits often need protection. If input voltages become too high, the vulnerable input circuits can be damaged quite easily (e.g. by static discharges). To overcome this problem, diodes can be used to limit the input voltage. An example of a diode used in a voltage limiter circuit is shown in Figure 8.5.

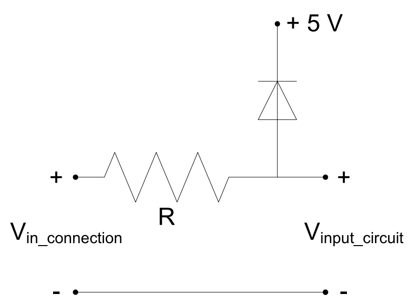


Figure 8.5: *A diode used in a voltage limiter circuit.*

For a voltage $V_{in-connection}$ above 5.7V (5V plus the diode's knee voltage) the diode will let current pass. If $V_{in-connection}$ is lower than 5.7 V, the diode blocks and the current will flow to the output.

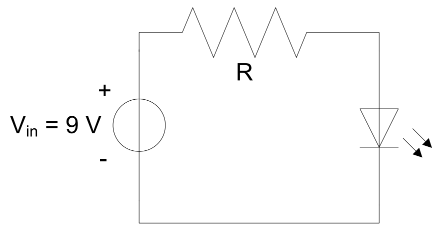
In this way $V_{input_circuit}$ is always lower than 5.7 V and possible peaks in the input signal will be carried away through the diode.

8.2.4 Diodes and light

Light Emitting Diodes (LEDs) were already mentioned. They are mostly used for applications such as (power) indicators and bicycle lighting. LEDs consume less power than light bulbs for the same amount of light and have a much longer life time. Since several years LEDs are available which can produce high intensities of light and they are expected to completely replace our traditional incandescent light bulbs within a few years. A regular LED needs a current of 10..20 mA. The voltage across the LED is usually 1.5 to 2 V. The latter depends on the color (white, yellow, orange, green, red or blue). LEDs are indicated in schematics by diodes with arrows that depart from it.

? Question 8.2

Given the following circuit.



Assume that the voltage across the LED is 2 V and that we want to have a current of 15 mA. What will be a good E-12 based value for R?

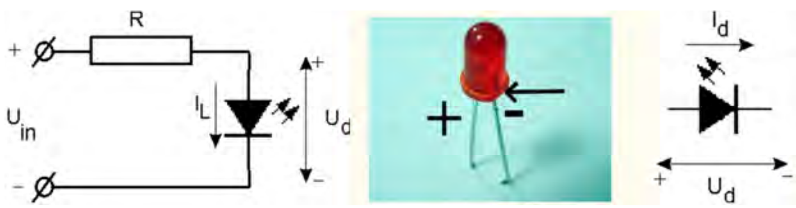


Figure 8.6: *A LED circuit, a LED and it's symbol.*

9

Transistors

9.1 What is a transistor?

Besides so-called passive components, there are active components and electromechanical ones.
Active components rely on a source of energy (usually from the DC circuit, which we have chosen to ignore) and usually can inject power into a circuit, though this is not part of the definition. Active components include amplifying components such as transistors, triode vacuum tubes (valves), and tunnel diodes.
Passive components can't introduce net energy into the circuit. They also can't rely on a source of power, except for what is available from the (AC) circuit they are connected to. As a consequence they can't amplify (increase the power of a signal), although they may increase a voltage or current (such as is done by a transformer or resonant circuit). Passive components include two-terminal components such as resistors, capacitors, inductors, and transformers.
Electromechanical components can carry out electrical operations by using moving parts or by using electrical connections.
The key active component, which will be discussed in this chapter, is the ***transistor***. It is considered by many to be one of the greatest inventions in modern history, ranking in importance with inventions such as the printing press, the automobile, and the telephone. Without transistors (easy) amplification, switching, voltage stabilization, signal modulation, and many other functions that we consider essential nowadays would not be possible. Furthermore, if we would still use electronic tubes for amplifiers, switching, and such, our notebook computer would not fit into the room.

The transistor is a solid state semiconducting device. Several types of transistors exist. In this reader we will shortly discuss the bipolar junction transistor and the field effect transistor. In Figure 9.1 some samples are shown.

9.2 Bipolar junction transistor

Like a diode, a bipolar junction transistor (or BJT in short) consists of doped N type and P type sections. However, instead of two sections, the BJT is a layered sandwich of three sections. Because the types of the sections alternate, we can distinguish between so called NPN-type BJTs and PNP-type BJTs. Without going into detail about the physical aspects, we state that the NPN-

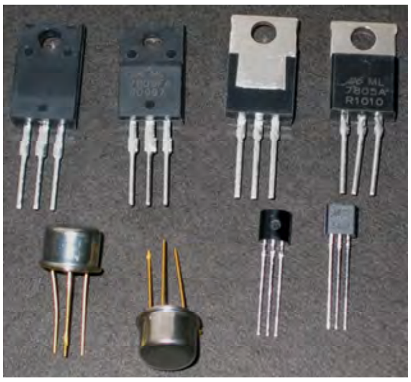


Figure 9.1: *Samples of transistor packages.*

and PNP-types have the same properties, except that their currents flow in opposite directions. In Figure 9.2 the schematic symbols for NPN- and PNP-type BJTs are drawn.

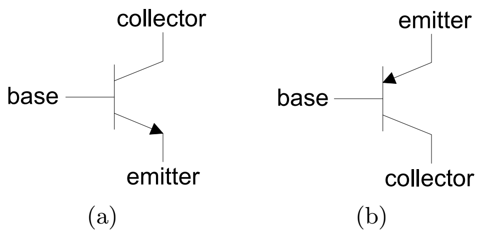


Figure 9.2: *Schematic symbols for (a) NPN and (b) PNP bipolar junction transistors.*

Both types of BJT have three connections. The left connection for both types is called the base (b). The connection with the arrow is called the emitter (e) and the other connection is called collector (c). In the remainder of this reader, we will indicate these connections with b, e, and c.

Since there are two PN-transition in each transistor, we can also draw them as diodes. In Figure 9.3 the 'equivalent' diode symbols for the NPN and PNP BJTs are given. These diode symbols can be misleading: don't think that you can construct a transistor by simply connecting two diodes to each other.

When the potential difference between the base and emitter of the NPN is positive, free electrons from the emitter (N) can move to the thin base-section (P); the base-emitter diode opens. Since the emitter-section contains many free electrons, the base-section will overflow and the electrons reach the top NP-transition. Although the diode is closed, the electrons will be attracted by the positive collector once they have reached the transition. So a (small) base-emitter current leads to a (large) collector-emitter current!

For a PNP BJT a similar derivation can be made. However, now a current from the emitter to the base leads to a negative collector-emitter current.

If the potential difference between the collector and emitter of the NPN BJT is positive, the relation between the base-emitter and collector-emitter currents, I_b and I_c, respectively, can be written as

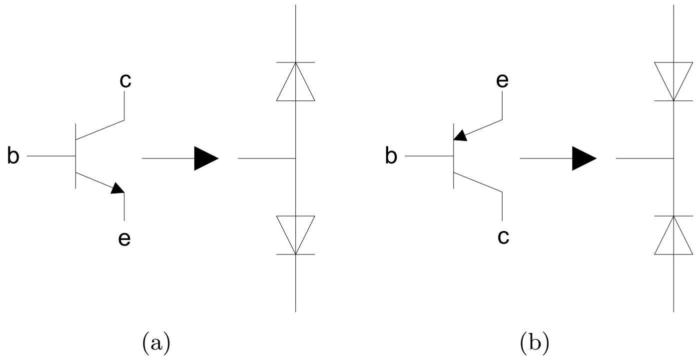


Figure 9.3: *Schematic symbols for (a) NPN and (b) PNP bipolar junction transistors with 'equivalent' diode schematics.*

follows:

$$\boxed{I_c = \beta \cdot I_b \qquad (9.1)}$$

where β is the current gain (usually in the range of 50 until 500, depending on the type of transistor). The transistor thus has a gaining property: a small current through the base yields a big current through the collector. The same formula holds for the PNP-transistor, however, for this type of transistor the potential difference between the emitter and collector must be positive.

> **!** **Note**
>
> It was stated that the potential difference between the base and emitter of the **NPN** must be positive to allow current to flow from its collector to its emitter. Actually, this value must exceed 0.6 V. Recall the discussion about the knee-voltage of a diode. For a **PNP** transistor, the potential difference between emitter and base must be **negative** but also exceed 0.6 V.

9.3 Applications of BJT

9.3.1 BJTs and switching

On of the most important applications of a BJT is switching: we can apply two levels of potential difference between the base and emitter in such a way that we have a collector current or not. Especially in digital circuitry (for logic gates as we will see later) switching is essential. Furthermore, we often need to control one circuit by another circuit. In Figure 9.4 an example of the latter is given. Here subcircuit X is needed to switch a lamp on or off. However we need an extra switching circuit, because subcircuit X cannot supply enough energy for the lamp (e.g. the current of the lamp needs to be 50 mA, the voltage drop across it needs to be 5 V and subcircuit X can only supply 2V and a maximum current of 1mA). Note that the minus connection points of the power supplies need to be connected.

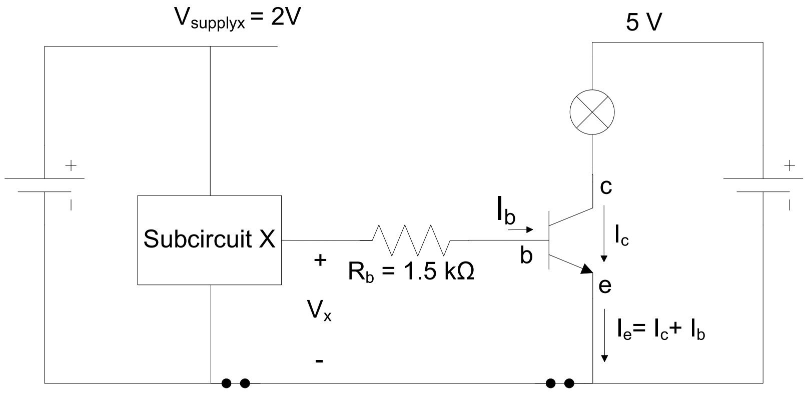


Figure 9.4: *Switching of a lamp by subcircuit X using an NPN-transistor.*

Before we explain the circuit, we have to introduce an aspect called **saturation**. In Figure 9.5 the collector currents as function of the potential differences between collector and emitter is given for different base currents.

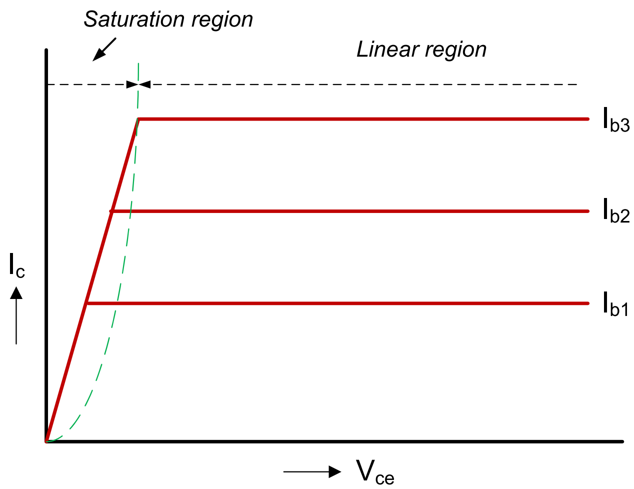


Figure 9.5: *Collector current as a function of collector-emitter potential difference for three different base currents.*

As can be seen in the figure, for each base current the relationship between collector current and collector-emitter potential consists of two parts: a steep part and a flat part. We call the flat part the linear part and the steep part the saturation part (the collector is saturated, as the collector-emitter potential does not increase anymore for increasing collector currents). When using BJTs for switching, we need to make sure that the transistor is in saturation when a collector current flows. The fact that the transistor needs to be in saturation can be explained by the functionality of the transistor: we want to use it as an ideal switch, like drawn in Figure 9.6.
For an ideal switch as drawn in Figure 9.6 it holds that the potential difference V_{switch} is equal to 0 V (for a closed switch). Although this potential difference for the BJT (i.e. V_{ce}) will never reach zero, it is very low when the transistor is in saturation (around 0.2 V).

To make sure that the transistor is in ***saturation***, you can use a rule of thumb. Adjust I_b (and so

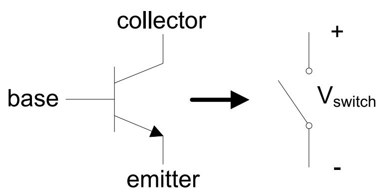


Figure 9.6: *Ideally, an NPN used for switching can be replaced by a switch which is controlled by the transistor's base.*

R_b) in such a way that holds:

$$I_b \approx 10. \frac{I_{c,limited}}{\beta}. \tag{9.2}$$

So you have to calculate I_b and then multiply it by a factor 10 to put the transistor into ***saturation***. Here $I_{c,limited}$ is the current that will flow through the load (and collector) when the transistor is switched on and is determined by the Ohmic characteristics of the load.

When switched in saturation, the amount of power dissipated in the transistor is usually very small, since the potential difference between collector and emitter (V_{ce}) will be very small (around 0.2 V). Remember that like in a resistor the power $P_{transistor}$ is equal to $I_c \cdot V_{ce} + I_b \cdot V_{be}$. The $I_b \cdot V_{be}$ is often ignored.

Example 9.1

Important !!!

The resistance value of the base resistor in the circuit of Figure 9.4 was found using rule of thumb 9.2 and that the current of the lamp will be 50 mA, the voltage drop across it then needs to be 5 V and subcircuit X can only supply 2V at a maximum current of 1mA. So $I_{c,limited}$ will be 50 mA, a value of 500 was assigned to β and $V_{be_{sat}}$ is assumed to be 0.6V. A three step method is applied:

First: calculate I_b

$I_b = \frac{I_{c,limited}}{\beta} = \frac{50.10^{-3}}{500} = 0.1mA$

Second: multiply I_b by a factor 10 for forcing saturation.

$I_b, saturation = 10.I_b = 1\text{mA}$

Third: calculate R_b. This is simple since V*over* R_b and I*through* R_b (I_b) are known.

$V_{be} = 0.6\text{ V} \rightarrow \frac{V_X - 0.6}{R_b} = I_b \rightarrow \frac{2-0.6}{R_b} = 0.001A \rightarrow R_b = \frac{1.4}{0.001} = 1400\ \Omega.$

We take a resistance value from the E12 series, so for R_b a resistor of 1.5 kΩ was chosen.

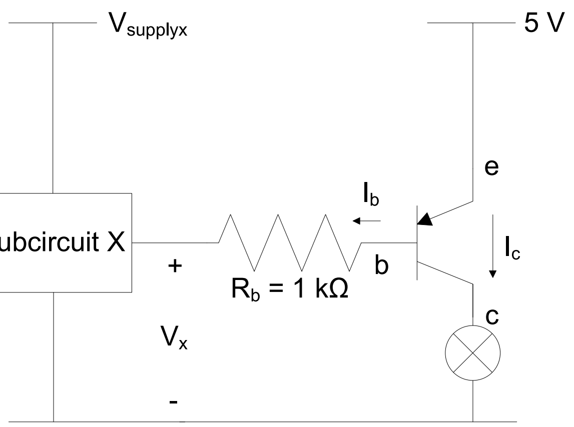
! Note

Whenever you use a BJT for switching, think of the following important issues:

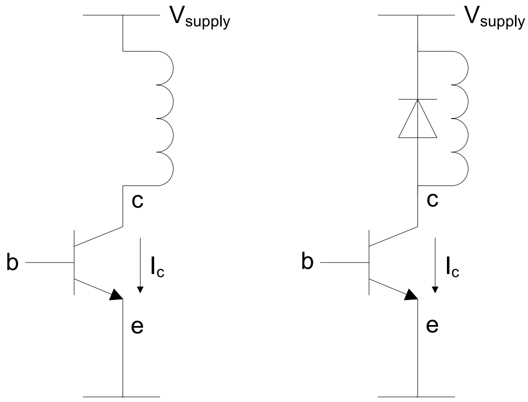
1. **Always connect a resistor R_b to the base of the transistor;** never connect the (controlling) subcircuit to the transistor directly. The base-emitter transition is simply a diode and the current flowing through it has to be controlled in order not to destroy the transistor.
2. When using an **NPN** transistor, you always need to connect the load between the positive supply contact and the **collector** of the transistor.
3. When using a **PNP** transistor, you always need to connect the load between the **collector** and the negative supply contact.
4. Since the current through a transistor can only flow in one direction (from collector to emitter in case of a NPN type and vice versa for a PNP type) a single transistor can not switch AC currents.
5. The (controlling) subcircuit can be everything! (output of a computer port, a digital circuit, a circuit designed by yourself, a function generator, etc. . .)

? Question 9.1

We replace the NPN-transistor in Figure 9.4 with a PNP-transistor (the lamp remains connected to the collector, so the positions of the transistor and the lamp are changed). This is shown in the following circuit:



The output voltage of subcicruit X can only have two levels: 0V or 5V.
What should the output voltage level of subcircuit X be if we want to switch the lamp on? Explain why!
Remember that now a PNP transistor is applied !



(a) (b)

Figure 9.7: *An NPN-transistor with (a) an inductor as load and (b) an inductor and diode in parallel as load.*

9.3.2 BJTs and switching inductors as load

When we replace the lamp in the circuit drawn in Figure 9.4 with an inductor (e.g. a relais), we may be in trouble, since there is a significant chance that the transistor will get damaged when the current is suddenly interrupted. An NPN-transistor with an inductor as load is drawn in Figure 9.7(a). The transistor is conducting, so a current I_c flows through the inductor. When the transistor is suddenly switched off, it will stop conducting, so there cannot flow (collector) current anymore. This means that at the instance of switching, $|\frac{dI_c}{dt}|$ grows to infinity. On the other hand, the inductor tries to keep the current stable (which is a property of an inductor, think of Newton's mass law), resulting in a voltage drop across the inductor, where the potential of the inductor-transistor connection becomes higher than the positive supply potential (a voltage drop of 1000 V can be reached easily!). The potential difference between collector and emitter therefore increases as well. This difference will exceed the breakdown-voltage of the transistor.

Fortunately, this problem can be solved easily by adding a diode to the circuit. This is drawn in Figure 9.7(b). When the collector current is interrupted, the diode starts conducting. The inductor will still try to keep the current stable, but now the current will flow away through the diode instead of through the transistor, thereby saving the transistor. This is called bleeding and the diode a ***bleeding diode***.

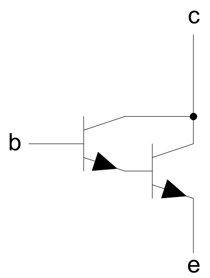


Figure 9.8: *NPN Darlington pair transistor.*

? **Question 9.2**

1. Argue, by showing a calculation and with the use of the relation $V = L\frac{dI}{dt}$, that suddenly interrupting even a small current will lead to a huge potential difference over the inductor.

Assume an inductance value of 1 mH, a collector current of 100 mA, and a switching time of 1 μs (switching times of less than 1 μs are no exception!).

2. Suppose no **bleeding** diode is available. Which of the three transistors can switch the above mentioned load condition without being damaged: a) BC550, b) 2N3439 or c) BC618

Study the data sheets of these transistors. They can be found on the internet.

9.3.3 BJTs and Darlington pairs

Sometimes the controlling subcircuit cannot supply enough current (I_b) to saturate the transistor. In those cases you can make use of a darlington transistor. Such a transistor is drawn in Figure 9.8 (NPN). As can be seen in the figure, a darlington transistor is in fact a combination of two regular transistors.

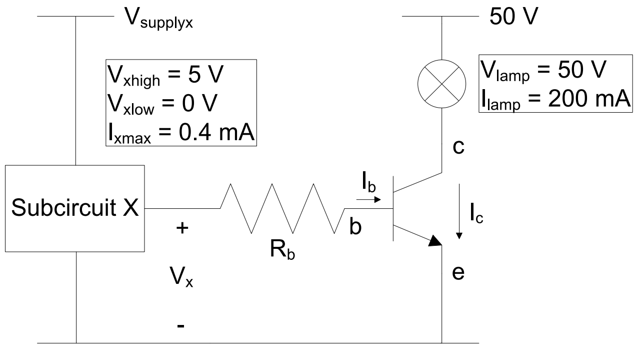
Like regular transistors, darlington (pairs) are available in various types.

? **Question 9.3**

Give an approximation of the darlington's base-emitter potential difference (over the 'whole' pair) that is needed to make it conduct.

? **Question 9.4**

Given the following system,



and the specifications for 4 different NPN-transistors:

Type:	2N3773	BC550	BD139	BC618
β_{max}	60	520	250	50000
$V_{ce,max}$	140 V	50 V	80 V	80 V
$I_{ce,max}$	16 A	0.1 A	1.5 A	1 A
$V_{be,saturation}$	0.8 V	0.9V	1 V	1.6 V

- Which of the four transistors will fulfill the requirements when switched into saturation?
- Then calculate the resistance value of R_b needed for the saturation condition ?

Hint: Use the step-wise calculation which was discussed earlier.

9.4 Field Effect Transistor

In the Bipolar Junction Transistor sections, we saw that the output Collector current of the transistor is proportional to input current flowing into the Base terminal of the device, thereby making the bipolar transistor a "CURRENT" operated device (Beta model). The Field Effect Transistor, or simply FET however, uses the voltage that is applied to their input terminal, called the Gate to control the current flowing through them resulting in the output current being proportional to the input voltage. As their operation relies on an electric field (hence the name field effect) generated by the input Gate voltage, this then makes the Field Effect Transistor a "VOLTAGE" operated device.

The FET is a three terminal unipolar semiconductor device that has very similar characteristics to those of their Bipolar Transistor counterparts ie, high efficiency, instant operation, robust and cheap and can be used in most electronic circuit applications to replace their equivalent bipolar junction transistors (BJT) cousins.

Field effect transistors can be made much smaller than an equivalent BJT transistor and along with their low power consumption and power dissipation makes them ideal for use in integrated circuits

such as the CMOS range of digital logic chips.
We remember from the previous sections that there are two basic types of Bipolar Transistor construction, NPN and PNP, which basically describes the physical arrangement of the P-type and N-type semiconductor materials from which they are made. This is also true of FET's as there are also two basic classifications of Field Effect Transistor, called the N-channel FET and the P-channel FET.
The FET is a three terminal device that is constructed with no PN-junctions within the main current carrying path between the **Drain** (D) and the **Source** (S) terminals, which correspond in function to the Collector and the Emitter respectively of the bipolar transistor. The current path between these two terminals is called the "channel" which may be made of either a P-type or an N-type semiconductor material.
The control of current flowing in this channel is achieved by varying the voltage applied to the **Gate** (G). As their name implies, Bipolar Transistors are "Bipolar" devices because they operate with both types of charge carriers, Holes and Electrons. The Field Effect Transistor on the other hand is a "Unipolar" device that depends only on the conduction of electrons (N-channel) or holes (P-channel).
The Field Effect Transistor has one major advantage over its standard bipolar transistor cousins, in that their input impedance, (Rin) is very high, (thousands of Ohms), while the BJT is comparatively low. This very high input impedance makes them very sensitive to input voltage signals, but the price of this high sensitivity also means that they can be easily damaged by static electricity.
There are two main types of field effect transistor, the Junction Field Effect Transistor or JFET and the Insulated-gate Field Effect Transistor or IGFET), which is more commonly known as the standard Metal Oxide Semiconductor Field Effect Transistor or MOSFET for short.

In Figure 9.9 the symbols for both P-type and N-type FETs are given. The different MOSFET types are not discussed in this document.

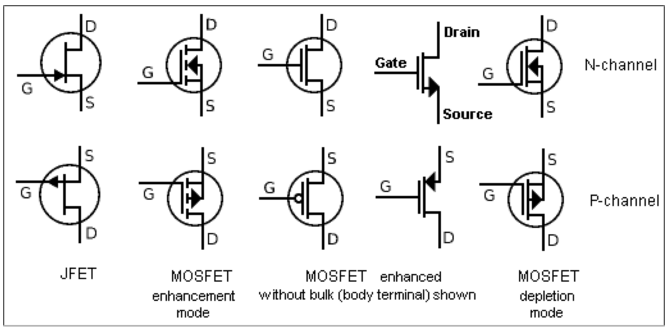


Figure 9.9: *Schematic symbol various types of FETs.*

9.5 An example of using the MOSFET as a switch

In Figure 9.10 an Enhancement-mode N-channel MOSFET is being used to switch a simple lamp "ON" and "OFF" (could also be an LED). The gate input voltage VGS is taken to an appropriate positive voltage level to turn the device and therefore the lamp either fully "ON", (VGS = $+V_{DD}$) or at a zero voltage level that turns the device fully "OFF", (VGS = 0). If the resistive load of the lamp was to be replaced by an inductive load such as a coil, solenoid or relay a "flywheel

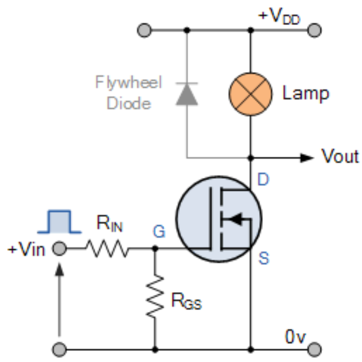


Figure 9.10: *a FET used for switching a lamp.*

diode" would be required in parallel with the load to protect the MOSFET from any self generated back-emf.
For the power MOSFET to operate as an analogue switching device, it needs to be switched between its "Cut-off Region" where VGS = 0 and its "Saturation Region" were VGS(on) = $+V_{DD}$. When switched ON you expect VDS to be as low as possible and so the "ON-resistance" of the channel given as RDS(on) also as low as possible. The power dissipated in the MOSFET depends upon the current flowing through the channel DS at saturation and RDS(on).

Often you see that Rin and Rgs are omitted. They however have to be implemented in some applications like high-power/high-frequency switching. MOSFET's require very little current to turn on, so any floating input is going to cause bad effects, where just waving your hand at an unconnected gate pin it can make it oscillate on and off. **Rgs** is there to hold the gate off until you apply the voltage to turn it on.
While very little current is needed to turn MOSFET's on, they do have a capacitance at the gate pin that you have to charge to turn on and discharge to turn off. Directly connecting this capacitance to whatever previous stage you are using to turn it on and off with is not the best idea. An ideal capacitor switched to an ideal voltage supply will have a pulse of infinite current, of course your driving stage won't be able to supply that and resistance in the line will limit it as well, the effect that it creates is that you have these large transients that could have unwanted effects of increased noise and EMF in the system, **Rin** stops the big inrush transients to the gate capacitance of the MOSEFT as it makes the gate capacitance charge up slower. While small MOSFET's don't have very large gate capacitance, bigger ones do.

Check ($http://www.electronics-tutorials.ws/transistor/tran7.html$) for details about how to apply a FET as an electronic switch.

10

Operational Amplifiers

10.1 What are operational amplifiers?

Operational amplifiers are active components that can be used to amplify signals in electrical circuitry (e.g. amplification of an audio signal inside your stereo amplifier). An operational amplifier (or opamp in short) usually has two inputs and one output. Furthermore, connections for the positive and negative supply voltages (V_+ and V_-, respectively), are present. The output voltage of the opamp V_{out} can be written as a function of the potential difference between the non-inverting (+) and inverting (-) inputs (V_{in+} and V_{in-}, respectively):

$$\boxed{V_{out} = G \cdot (V_{in+} - V_{in-}) \qquad (10.1)}$$

where G is the amplification. When no other components are connected, G is in the order of 100.000 to 1.000.000! However, although the gain is very high, the output voltage is limited by the positive and negative supply voltages. When the output voltage reaches one of these limits while Equation 10.1 states that it should be higher (or lower), we say that the opamp is clipping.

A schematic symbol for an opamp is given in Figure 10.1. You will see that the positive and negative supply voltage pins are often omitted.

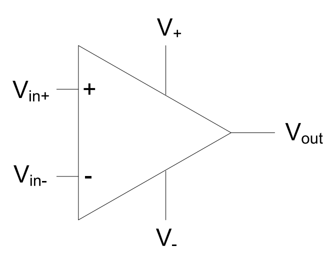


Figure 10.1: *Schematic symbol for an operational amplifier.*

In most applications, opamps are used in configurations with a negative feedback connection. This means that the output of the opamp is connected to the negative input, either directly or by means of extra components. You may ask yourself why one would connect an opamp in this way. To understand this, we recall the high gain of the (plain) opamp. Due to the high gain, small potential differences between the input pins (e.g. millivolts) already result in a clipping opamp. However, by applying negative feedback we are able to control the overall gain of the amplifier to be constructed. In Figure 10.2 an opamp without negative feedback (open loop). In Figure 10.3 the opamp has a negative feedback (closed loop) configuration.

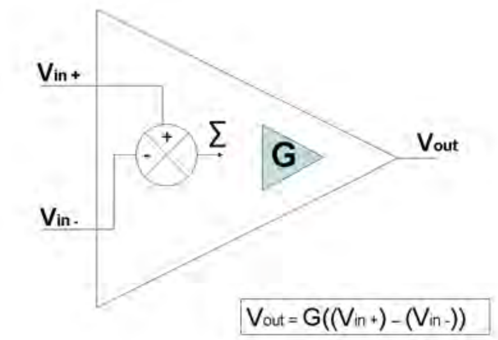


Figure 10.2: *Operational amplifier without negative feedback applied.*

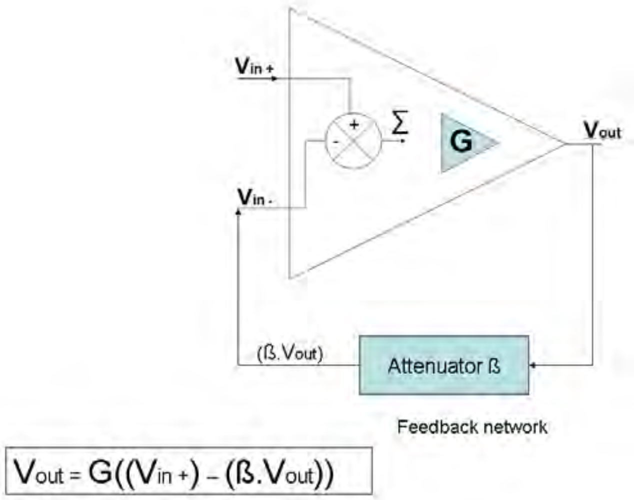


Figure 10.3: *Operational amplifier with a negative feedback network.*

From the equation given in Figure 10.3 can be derived that holds:

The closed loop amplification: $\frac{V_{out}}{V_{in}} = G_v = \frac{G}{1+\beta G}$

Since the open loop gain G has a very high value it appears that the closed loop gain G_v mainly depends on the feedback factor β . For ideal opamps with a negative feedback loop, we have two important premises:

1. No current will flow into the inputs;
2. The negative feedback reduces the potential difference on the input to 0 V.

Exercise 10.1

Actually derive that holds:

The closed loop amplification: $\frac{V_{out}}{V_{in}} = G_v = \frac{G}{1+\beta G}$.

10.2 Application

10.2.1 Opamps and inverting amplifiers

An opamp can be used for an inverting amplifier. We call the amplifier inverting because the output signal is an amplified version of the input signal with an altered sign. In Figure 10.4 a circuit of an inverting amplifier is given.

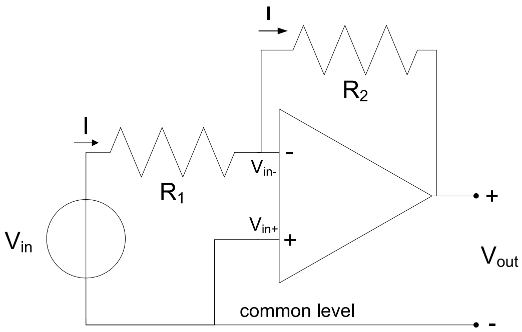


Figure 10.4: *Operational amplifier used in an inverting amplifier circuit.*

The transfer of this circuit (i.e. $\frac{V_{out}}{V_{in}}$; the gain of the amplifier), can be determined in three steps:

1. I = $V_{in}.R_1$; remember V$_{in-}$ = V_{in+} and so V_{in-} is at 0 V level.
2. Because the current cannot flow into the input, it flows through resistor R_2. With $V_{in-} = V_{in+}$, $V_{out} = 0 - I\cdot R_2$;
3. Rewrite the value for V_{out} in terms of V_{in} and find $V_{out} = -\frac{R_2}{R_1}\cdot V_{in}$.

For the inverting amplifier circuit drawn in Figure 10.4 we thus have the following transfer:

$$V_{out} = -\frac{R_2}{R_1} V_{in} \tag{10.2}$$

By choosing the right resistance values, we can thus realize every gain we want. For example, if a gain of -10 is needed, we can use $R_1 = 1$ kΩ and $R_2 = 10$ kΩ. Note that when $R_2 < R_1$ we have an inverting attenuator instead of an inverting amplifier.

10.2.2 Opamps and non-inverting amplifiers

Often it is not wanted to invert a signal when it is amplified. If this is the case, you can use the opamp in a non-inverting amplifier circuit. An example of such a circuit is drawn in Figure 10.5.

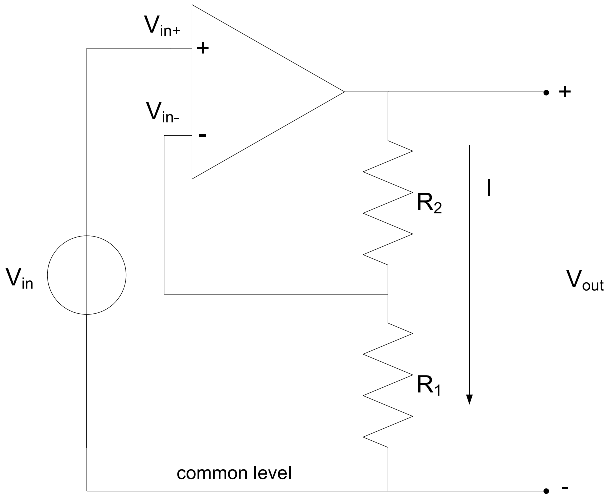


Figure 10.5: *Operational amplifier used in a n on-inverting amplifier circuit.*

For the non-inverting amplifier circuit drawn in Figure 10.5, we have the following transfer:

$$V_{out} = (1 + \frac{R_2}{R_1})V_{in} \tag{10.3}$$

? **Question 10.1**

Determine (the same way as we did for the inverting amplifier) that for the transfer of the non-inverting amplifier, drawn in Figure 10.5, holds:

$$\frac{V_{out}}{V_{in}} = 1 + \frac{R_2}{R_1}$$

Hint: use the fact that in fact V_{in-} is the output voltage of the voltage divider that divides the output voltage V_{out}.

! **Note**

The discussed operational amplifiers are supposed to be ideal. However, in practice, there are several limitations:

- the output cannot be regarded being an ideal voltage source. the output impedance Z_{out} is in general low (a few Ohms) and depends on the amount of negative feedback applied.
- the output current I_{out} is limited and thus the power that the opamp can deliver is limited.
- the output can saturate to the positive and negative supply voltages so it cannot be driven to these voltages completely;
- when choosing the wrong component values, a significant part of the output power may be lost in the feedback network. So the feedback resistors R_1 and R_2 have to be chosen in **k**Ω's or **M**Ω's. A suitable range would be: $> 1\mathbf{k}\Omega < 1\mathbf{M}\Omega$.
- for amplifying AC signals you will often need to apply a so-called symmetrical power supply for 'feeding' the opamp. See figure: 10.6

The significance of these limitations depends on the specific type of opamp and on the configuration in which it is used. The data sheet will provide you with the parameters needed to make a good choice of which opamp to use.

10.2.3 Opamps and comparators

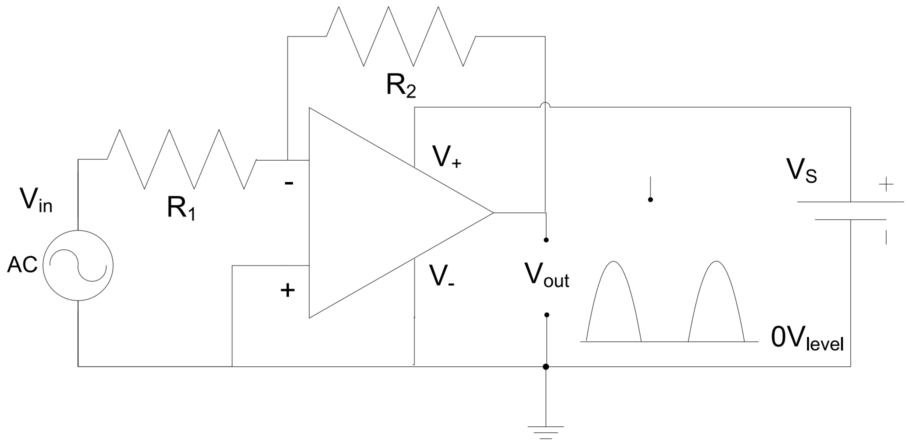
Clipping amplifiers are not always a problem. Moreover, at times this behavior is wanted. For example, if you compare two values and the only thing you need to know is whether value 1 is higher than value 2, such behavior is ideal! Circuits that perform the comparison are called comparators. In Figure 10.7 a typical comparator circuit is shown. Instead of the voltage division using two resistors, you can also use a potmeter!

The output voltage of the circuit shown in Figure 10.7 is equal to the positive supply voltage whenever $V_{in} < \frac{R_2}{R_1+R_2} \cdot V_x$ and equal to the negative supply voltage in the other cases.

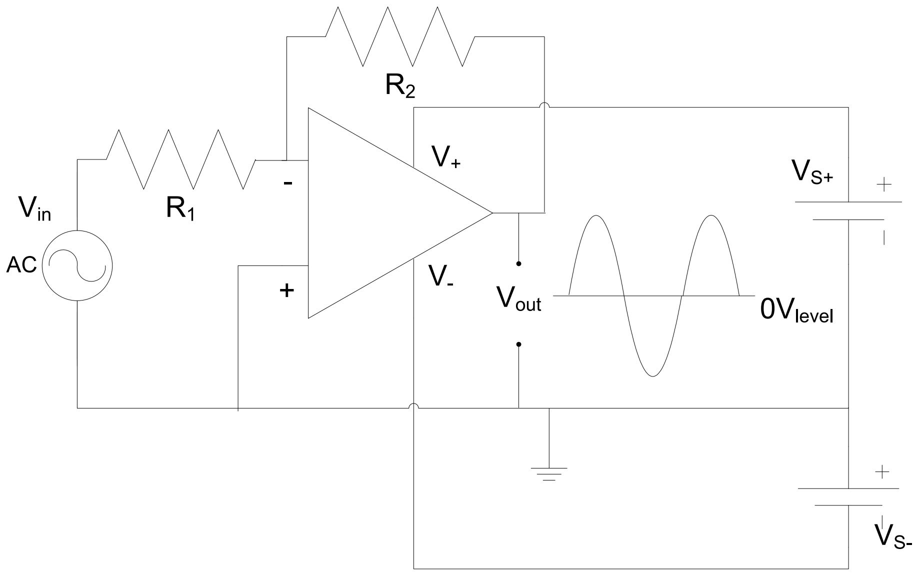
Although the discussed circuit works as expected, in most cases it is not usable. Take for example an input signal that is very noisy. Around the 'decision level', this will result in a toggling output value, as depicted in Figure 10.8.

To overcome the problem of the toggling output, a solution called hysteresis can be used. With this solution, the decision levels for increasing input values and decreasing input values are not the same. This is depicted in Figure 10.9.

When the input increases above the level at which the output switches from high to low, small decreases in the input do not result in an output switch from low to high. Similarly, when the input decreases below the level at which the output switches from low to high, small increases in the input



Inverting AC amplifier – single power supply applied



Inverting AC amplifier – symmetrical (Vs+ = Vs-) power supply applied

Figure 10.6: *Operational amplifier used as AC amplifier.*

do not result in an output switch from high to low.

Hysteresis can be implemented using an opamp with a positive feedback loop. Instead of connecting the output with the negative input, the output is now connected with a resistor to the positive input. Such a comparator circuit is drawn in Figure 10.10.

To simplify calculations, we take the resistance value of resistors R_1 and R_2 10 kΩ, the resistance value of resistor R_3 100 kΩ, V_x equal to $V_{out,high} = V_+ = 10V$, and $V_{out,low} = V_- = 0$ V. We can now distinguish between two cases.

Case 1: $V_{out} = 10$ V.
The resulting circuit is given in Figure 10.11(a). Since resistors R_1 and R_3 have the same voltage drop across them and are connected to the same node, they are connected in parallel. Then remains

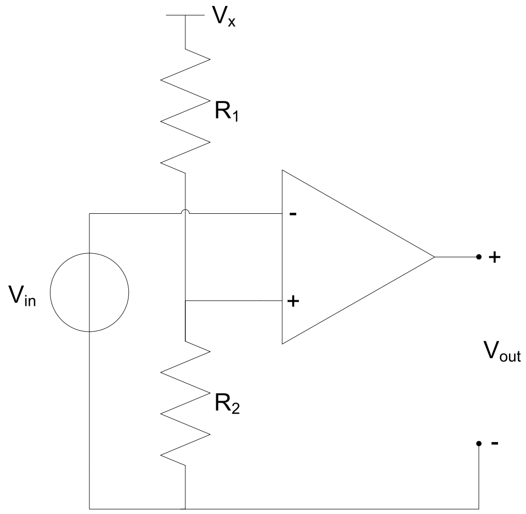


Figure 10.7: *Operational amplifier used in a comparator circuit with static comparison.*

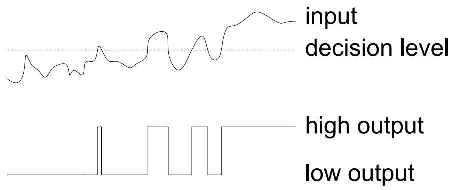


Figure 10.8: *The output of a comparator toggles when the input is noisy.*

that V_{in+} is a partial voltage of V_{out}: $V_{in+} = \frac{R_2}{R_2 + \frac{R_1 \cdot R_3}{R_1 + R_3}} \cdot V_x$. After substituting the given values, it follows that $V_{in+} = 5.24$ V.

Case 2: $V_{out} = 0$ V.
The resulting circuit is given in Figure 10.11(b). Since resistors R_2 and R_3 have the same voltage drop across them and are connected to the same node, they are connected in parallel. Then remains that V_{in+} is a partial voltage of V_{out}: $V_{in+} = \frac{\frac{R_2 \cdot R_3}{R_2 + R_3}}{R_1 + \frac{R_2 \cdot R_3}{R_2 + R_3}} \cdot V_x$. After substituting the given values, it follows that $V_{in+} = 4.76$ V.

Like indicated in Figure 10.9, we thus have two different transition levels and the transition level from low to high is lower than the transition level from high to low. For a noisy input signal, the output of the circuit drawn in Figure 10.10 is drawn in Figure 10.12.

Note that the resistance values of R_1, R_2, R_3 and the actual output levels (we considered the opamp to be ideal) determine the transition levels. When you design a circuit, you should decide whether the circuit needs to be very accurate in decision making or whether there is very much noise present in the input signal.

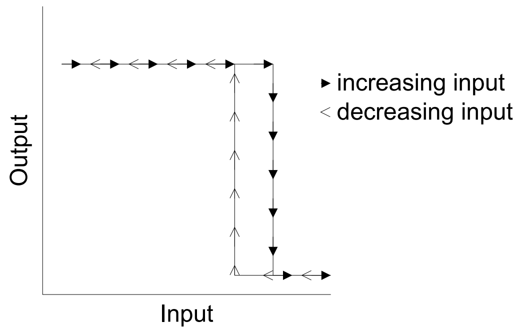


Figure 10.9: *Hysteresis.*

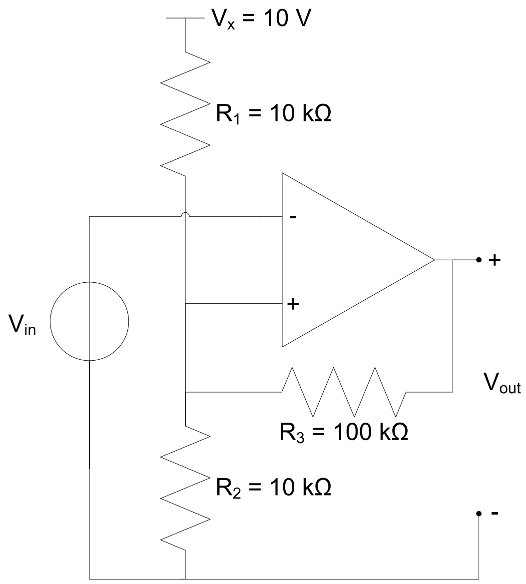


Figure 10.10: *An static opamp comparator circuit with hysterese.*

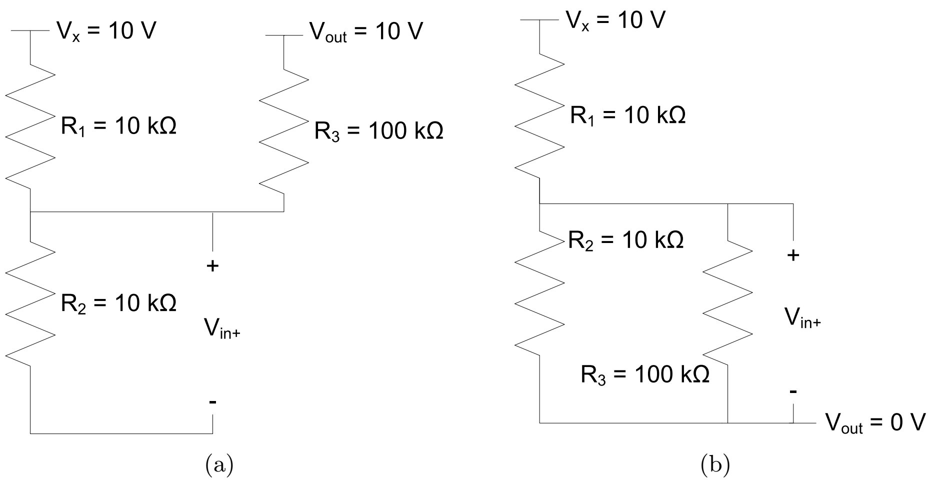


Figure 10.11: *Resulting circuits for (a) $V_{out} = 10\ V$ and (b) $V_{out} = 0\ V$.*

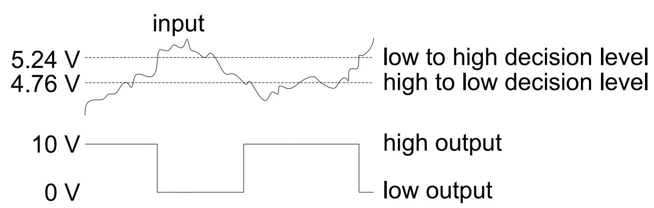


Figure 10.12: *The output of a comparator with hysterese does not make the output toggle for noisy input signals.*

11

Sensors and Actuators

11.1 What are sensors and actuators?

Basically, sensors and actuators are components that enable interaction between the physical world and electrical circuits: a sensor converts a physical phenomenon into an electrical signal for processing and an actuator converts a processed electrical signal to a physical phenomenon.

Sensors

Sensors are used to explore (changes) in the environment and convert them into (changing) electrical signals. Depending on the application, the (changes in) environment can be (changes in) lightness, sound pressure level, pressure, temperature, magnetic field, acceleration, etc. For example, a lightness-meter on your photo camera is used to determine whether a flash is necessary or not, the thermometer in the thermostat that senses the temperature, the receiver in your television that is needed for your remote control. A simple switch can also be seen as a sensor. The output signal of this sensor can then be seen as 'high' or 'low' instead of continuous values.

The signal which is produced by the sensor often needs to be processed before you know what the measured physical value was. You can think of amplification, linearization and comparing. In order to perform the right processing you need to know the characteristics of the sensor:

Transfer function
: The relationship between the physical and electrical signals (looking at the complete input-range).

Sensitivity
: The amount of change in the electrical signal induced by a change in the physical signal (e.g. for a temperature sensor this value can be specified in x V/Kelvin).

Operating or dynamic range
: The range of the physical signal in which the sensor produces a useful and reliable signal. Outside this range the sensor becomes inaccurate or it may not work at all.

Hysteresis

The fact that when an input signal increases the output signal goes to a certain value, but when the input signal decreases to its original value again, the output signal does not. This was discussed in the chapter about operational amplifiers.

Resolution

The smallest change in the physical signal that induces a change in the electrical signal.

(Non-)linearity

Most of the times the transfer function of a function is a curve, but not necessarily a line. The largest error between the relationship and an approximated line is a measurement for the linearity. Note that non-linear behavior is not always an issue.

Response time

The time it takes for the sensor to react on a change in the physical signal. For example, if there is a step-like change in the input signal it will take some time before the output goes to a steady value. Most of the times the response time is specified as the time that it takes for the output to come to 60%-65% of the new steady value when a step-like change occurred in the input signal.

Actuators

Actuators are in fact the reciprocal of sensors. Therefore, they can be specified by the same specifications that were given for sensors. Although sensors might be more appealing to you, actuators are all around us as well: (electric) motors, speakers, (electric) heating elements, electric powered light sources, monitors, disk controllers, etc.

Like sensors, actuators can have continuous values and discrete values as well. For example, you can have discrete lighting (on/off), but you can have continuous lighting using a dimmer as well. Another common aspect between actuators and sensors is that the signals need to be processed before they can be used. For sensors this is usually postprocessing (i.e. after the conversion), whereas for actuators this is usually preprocessing (i.e. before the conversion). You can think of an amplifier which amplifies an electrical signal that is passed to the speaker (the actuator which converts electrical signals into an electro-magnetic field which is needed to create pressure-changes in the air).

In Figure 11.1 an example of a closed-loop control system (a central heating system) is given.

The temperature is measured by the sensor. This temperature is compared to a reference temperature, which can be set by the user. Depending on the comparison result, a current is fed (or not fed) through a heating element (which heats up when current is fed through it). Because the result (the temperature) is constantly measured and used for control, we speak of a closed-loop control system.

In Figure 11.1 the actuator is given as the heating element (resistor). Usually, actuators are depicted as the symbol given in Figure 11.2(a). Often, the universal symbol is not used, but the electrical equivalent, like drawn in Figure 11.2(b).

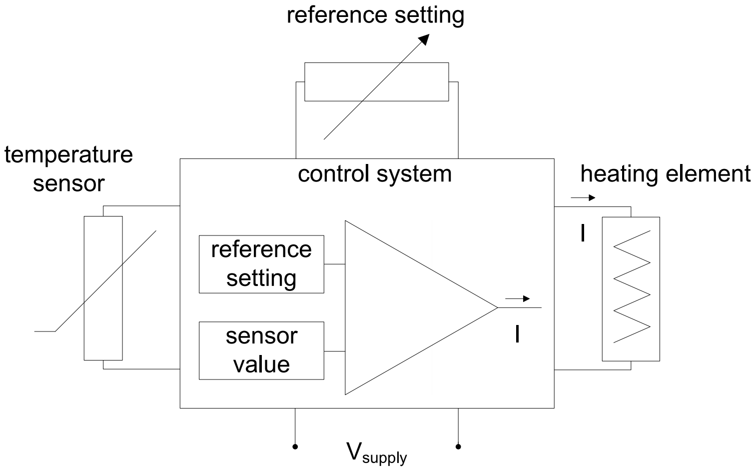


Figure 11.1: *Closed-loop temperature control system.*

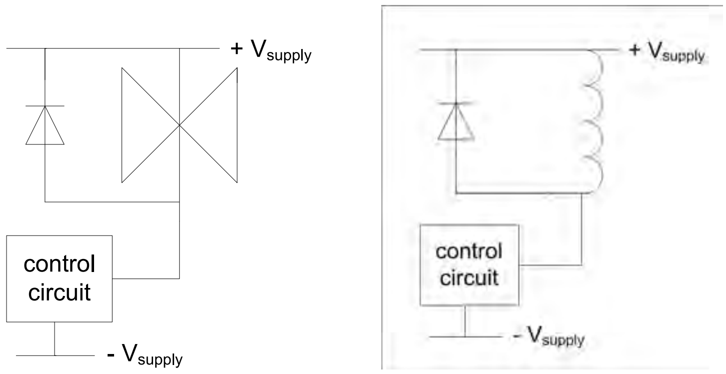


Figure 11.2: *Controlling an electro-magnetic (gas) valve: (a) functional symbol and (b) electrical symbol.*

11.1.1 Examples of sensors

We cannot give a complete overview of all kinds of sensor and actuator for all physical quantities. Therefore, we just give a small list of the most commonly used sensors for light, magnetic field and temperature.

Light-sensitive: LDR

A Light Dependent Resistor (LDR) is, as the name implies, a resistor which resistance depends on the lightness of the light falling on it. When the lightness increases, the value of the resistance decreases. Typical resistance values are 5.5 kΩ in dark and 55 Ω in a light environment. The schematic symbol for an LDR is drawn in Figure 11.3. Note that, in contrary to the symbol of a LED where arrows are used to indicate some 'outbound value', arrows in the symbol for an LDR indicate some 'inbound value'.

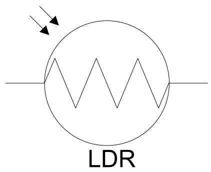


Figure 11.3: *Schematic symbol for an LDR*

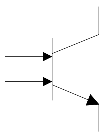


Figure 11.4: *Schematic symbol for a photo-transistor*

Light-sensitive: Photo-transistor

A photo-transistor is a transistor which starts conducting when light falls on it. More precise: a photo-transistor starts creating a base-current which depends on the lightness. You will often see photo-transistors in remote controlled applications (e.g. your television, or your stereo set). The schematic symbol for a photo-transistor is drawn in Figure 11.4.

Magnetic field-sensitive: Reed relay

A reed-relay or reed-contact is an electronic switch in a casing, which can be controlled by a magnetic field. This field can be generated by a permanent magnet, but also by an electro-magnet. The casing of the relay is often made of glass and contains some gas to keep the switch-contacts in good shape. In this way switching sparkles are prevented. Reed-relays can be used to switch small currents only. The working of the Reed relay is illustrated in Figure 11.5. Depending on the magnetic field, the contact switches on or off.

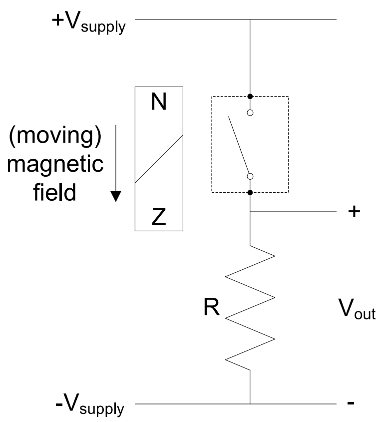


Figure 11.5: *Working of the Reed relay.*

In Figure 11.6 some types of reed relay are given.

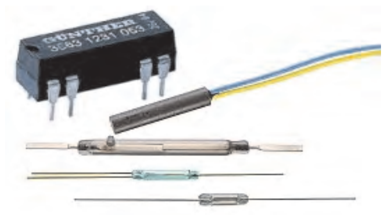


Figure 11.6: *Some types of Reed relay.*

Temperature-sensitive: NTC and PTC

There are resistors whose resistance value depends on the temperature. There are two types of them: ones that have a negative temperature coefficient (i.e. the resistance value decreases for increasing temperature) and ones that have a positive temperature coefficient. The former are called NTCs, whereas the latter are called PTCs. In Figure 11.7 the relationship between temperature and resistance value for three different NTCs is drawn.

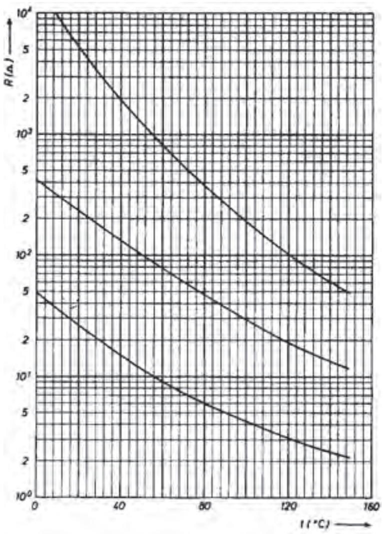


Figure 11.7: *Relationship between temperature and resistance value for three different NTCs.*

Note that the horizontal axis (temperature) is linear, whereas the vertical axis (resistance value) is logarithmic. When using such a sensor in (control) systems, one should be aware that the relation between resistance value and temperature is not linear!

In Figure 11.8 the schematic symbol for NTCs and PTCs is given.

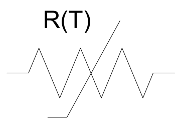


Figure 11.8: *Schematic symbol for NTCs and PTCs.*

12

Digital circuits

Until now we have discussed analog circuits. In these circuits voltages and currents can have all kinds of values (within certain limits). Besides analog circuits, we have digital circuits. Digital circuits are all around us: your (notebook) computer, your MP3-player, your mobile phone, etc. Contrary to analog circuits, in digital circuits we usually work with only two voltages: a 'high' voltage and a 'low' voltage. The values of these voltage depends on the standard used (e.g. 5 V). In the remainder of this chapter we will use '1' to indicate a high value and '0' to indicate a low value.

The power of digital circuits is their ability to perform (complex) calculations. They are made of components that can be either in the high state or the low state. That is why we call these components binary. Binary components are often referred to as gates or logic gates. Logic gates have one or more inputs and one output. The output value depends on the function of the gate as well as on the value(s) of the input(s). Circuits the output of which depends on the state of the inputs and it's previous output state as well, are called sequential circuits. A digital counter is an example of such circuit.

Providing you here with even a brief course on digital electronics is beyond the scope of this introduction. In the next section we will only discuss the most commonly used and basic gates. The function of these gates will be discussed using a truth table. This is a table in which you specify the output value of the gate for all combinations that you can make with the inputs. We will use a maximum of two inputs. Beside that some practical information on logic families is given.

12.1 Basic gates

12.1.1 The 'AND'-gate

The output value of an 'AND'-gate is 1 if and only if all of its input values are 1. In all other cases the output is 0. In words: input 1 should be 1 AND input 2 should be 1 AND ... AND input n should be 1. The truth table of a 2-input 'AND'-gate is given in Table 12.1.

In Figure 12.1 the schematic symbol for an 'AND'-gate is drawn.

input 1	input 2	output
0	0	0
0	1	0
1	0	0
1	1	1

Table 12.1: *Truth table of a 2-input 'AND'-gate.*

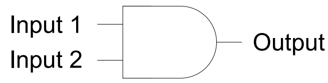


Figure 12.1: *Schematic symbol for an 'AND'-gate.*

12.1.2 The 'OR'-gate

The output value of an 'OR'-gate is 1 if at least one of its input values is 1. If none of the input values are 1, the output is 0. In words: input 1 should be 1 OR input 2 should be 1 OR ... OR input n should be 1 (actually, we have an AND/OR relation, because it is not exclusive like: either ... or ...). The truth table of a 2-input 'OR'-gate is given in Table 12.2.

input 1	input 2	output
0	0	0
0	1	1
1	0	1
1	1	1

Table 12.2: *Truth table of a 2-input 'OR'-gate.*

In Figure 12.2 the schematic symbol for an 'OR'-gate is drawn.

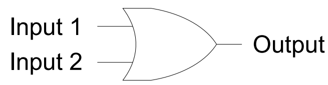


Figure 12.2: *Schematic symbol for an 'OR'-gate.*

A variant of the 'OR'-gate is the 'XOR'-gate, the so called exclusive 'OR'. The output value of the 'XOR'-gate is only 1 if exactly one of its inputs has value 1. The truth table of a 2-input 'XOR'-gate is given in Table 12.3.

input 1	input 2	output
0	0	0
0	1	1
1	0	1
1	1	0

Table 12.3: *Truth table of a 2-input 'XOR'-gate.*

In Figure 12.3 the schematic symbol for a 'XOR'-gate is drawn.

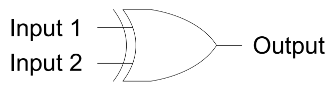


Figure 12.3: *Schematic symbol for a 'XOR'-gate.*

12.1.3 The inverter

An inverter has only one input. The output value is the inverted value of the input value. This means that whenever the input value is 1, the output value is 0 and whenever the input value is 0, the output value is 1. The truth table of an inverter is given in Table 12.4.

input 1	output
0	1
1	0

Table 12.4: *Truth table of an inverter.*

In Figure 12.4 the schematic symbol for an inverter is drawn.

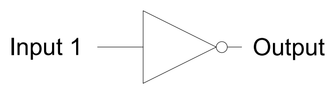


Figure 12.4: *Schematic symbol for an inverter.*

12.2 Combined gates

12.2.1 The 'NAND'-gate

In order to create (complex) digital circuits, multiple gates can be connected together. For example, if we connect the output of an 'AND'-gate to the input of an inverter, then the output of the inverter will be 0 if both the 'AND'-gate's inputs are 1 and the output will be one 1 in all other cases. We call such a combined gate a 'NAND'-gate. The truth table of a 'NAND'-gate is given in Table 12.5.

input 1	input 2	output
0	0	1
0	1	1
1	0	1
1	1	0

Table 12.5: *Truth table of a 2-input 'NAND'-gate.*

Instead of drawing the schematic symbol of an 'AND'-gate succeeded by that of an inverter, we use a single symbol. This symbol is drawn in Figure 12.5.

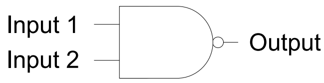


Figure 12.5: *Schematic symbol for a 'NAND'-gate.*

12.2.2 The 'NOR'-gate

An 'OR'-gate followed by an inverter is called a 'NOR'. The truth table of a 'NAND'-gate is given in Table 12.6.

input 1	input 2	output
0	0	1
0	1	0
1	0	0
1	1	0

Table 12.6: *Truth table of a 2-input 'NOR'-gate.*

The schematic symbol for a 'NOR'-gate is drawn in Figure 12.6.

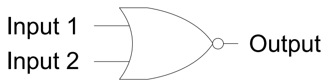


Figure 12.6: *Schematic symbol for a 'NOR'-gate.*

A combined 'XOR'-gate and inverter also exist: the 'XNOR'-gate. Its schematic symbol is drawn in Figure 12.7.

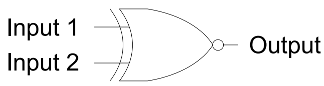


Figure 12.7: *Schematic symbol for an 'XNOR'-gate.*

? **Question 12.1**

Give the truth table for an 'XNOR'-gate.

12.2.3 More complex circuits

Digital circuits are usually more complex (think of your computer, which mainly consists of the discussed ports!). In Figure 12.8 an example of a more complex circuit is shown. This circuit can be used to detect whether cars are driving the wrong direction on a one way road.

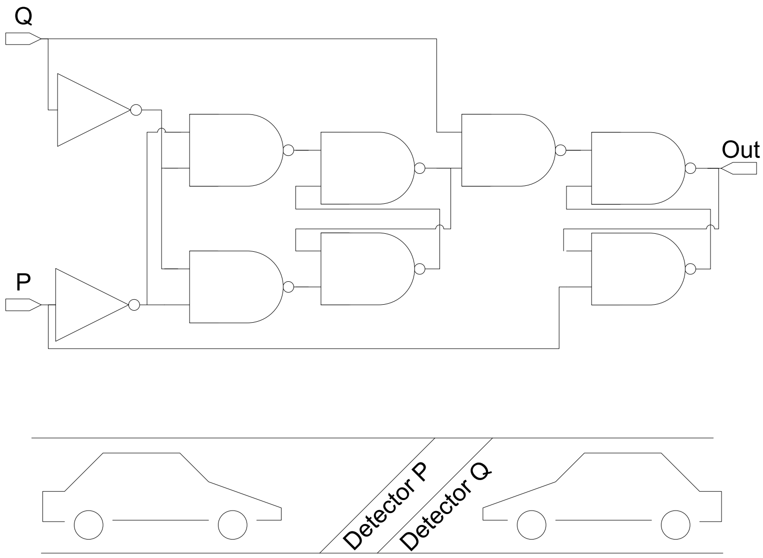


Figure 12.8: *Digital circuit for direction of driving detection.*

12.3 Logic integrated circuits

When you need logic gates for your implementation, you usually do not use single gates, but logic integrated circuits (IC's) that contain several gates (most of the time with the same function). There are two main types: 4000 CMOS series and the 7400/74LS/74HC/74HCT series. They differ in their physical (transistor) technology.

12.3.1 4000 CMOS series

IC's of this family are numbered from 4000 and 4500. Sometimes you find a trailing 'B', which indicates that it is an improved type of the original. Most of the IC's in this family have 14 or 16 pin housings. The CMOS (Complementary Metal Oxide Semiconductor) transistor technology allows for little current to be drawn from the power supply. The power supply voltage can be up from 3 V to 15 V. These IC's are very suitable for battery powered circuits. Unfortunately they are sensitive to static electricity: touching an input pin when you are statically charged can, in an occasional case, destroy the whole IC.

12.3.2 7400/74LS/74HC/74HCT series

The base family is the 7400 series. The other series are derived from this base serie. The letters indicate the transistor technology used. The original and the 74LS series are based on TTL (Transistor to Transistor Logic) technology, whereas the HC and HCT series are based on the CMOS series. Just like the 4000 CMOS series, the HC and HCT series usually have 14 to 16 pins housings. Most of the 74xx series require a stable 5 V source voltage. Although they consume more current than the 4000 CMOS series, their switching frequency is higher.

When you use IC's, there are two important specifications: the fan-out and sinking/sourcing outputs. With fan-out we mean the number of inputs that can be driven by one single output (i.e. the number of inputs of succeeding gates that you can connect to an output). Sinking and sourcing refer to the direction of the current on the output. If current flows into the output, the output is a sink. On the other hand, when the output supplies a current, we call it a source. In Figure 12.9 this is indicated.

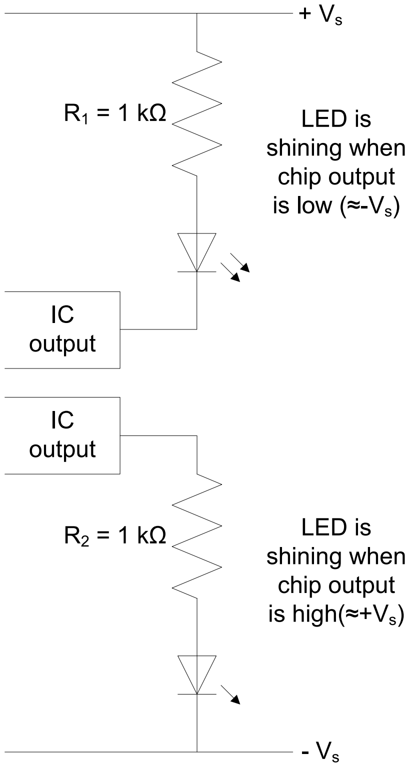


Figure 12.9: *Concept of sinking and sourcing currents.*

In Table 12.7 comparisons between the main properties of the 4000 and 7400 series are given.

Property	4000 series	7400/74LS/74HC/74HCT series
Source voltage	3 V to 15 V, tolerant for small fluctuations	5 V +/- 0.25 V, intolerant to small fluctuations (often a small capacitor with a capacitance value of 0.1 μF is placed over each of the source pins. The 74 HCT series works between 2 V and 6 V.
Inputs	Very high input resistance. This makes it very sensitive to disturbances. Always connect the non-used inputs to positive or negative source voltage.	Non-used (open) inputs become ‘high’ automatically. Shortcut to the negative source potential to make them ‘low’.
Outputs	About 1 mA sink and source	16 mA sink and 2 mA source.
Fan-out	Maximum 50 inputs.	Maximum 10 inputs.
Switching time	The reaction time on a change on the input is about 30 ns and depends on the source voltage.	The reaction time on a change on the input is about 10 ns.
Max. Switching frequency	About 1 MHz	About 35 MHz
Used power	Several μW, depends strongly on the switching frequency.	Several mW, depends on the switching frequency.

Table 12.7: *Comparisons between the properties of the 4000 and the 7400 series.*

13

Connecting circuits with each other

A loss-less signal transfer from one circuit to another demands so-called "matching". Each electrical (sub-) circuit has different properties. For example, different input- and output currents and impedances. You need to think about those properties when you connect circuits to each other. By connecting the output of some circuit to an input of another circuit, the signal can get lost or can get distorted. In Figure 13.1 two circuits are connected. You can think of the left circuit as the output of your CD-player and of the right circuit as the input of your amplifier. The output impedance of the CD-player is then equal to R_o and the input impedance of your amplifier is equal to R_i.

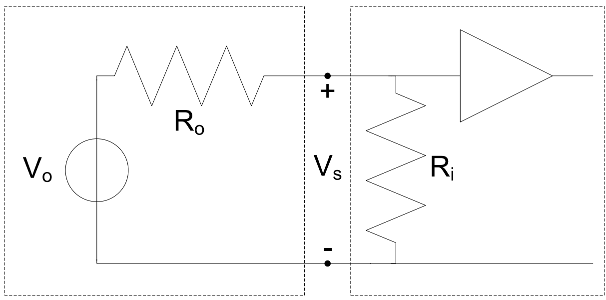


Figure 13.1: *Two connected circuits.*

When we calculate the value for V_s, we get a voltage divider. Depending on the values for R_o and R_i, we can a significant loss of signal.

> **?** **Question 13.1**
>
> Assume that the output impedance of your CD-player equals 2.2 kΩ and the input impedance of your amplifier equals 50 kΩ.
>
> What is the decrease in percent in the output signal of the CD-player?

The conclusion should be that when you connect two circuit to each other, you have to be aware of the discussed effects.

14

Practical Assignments

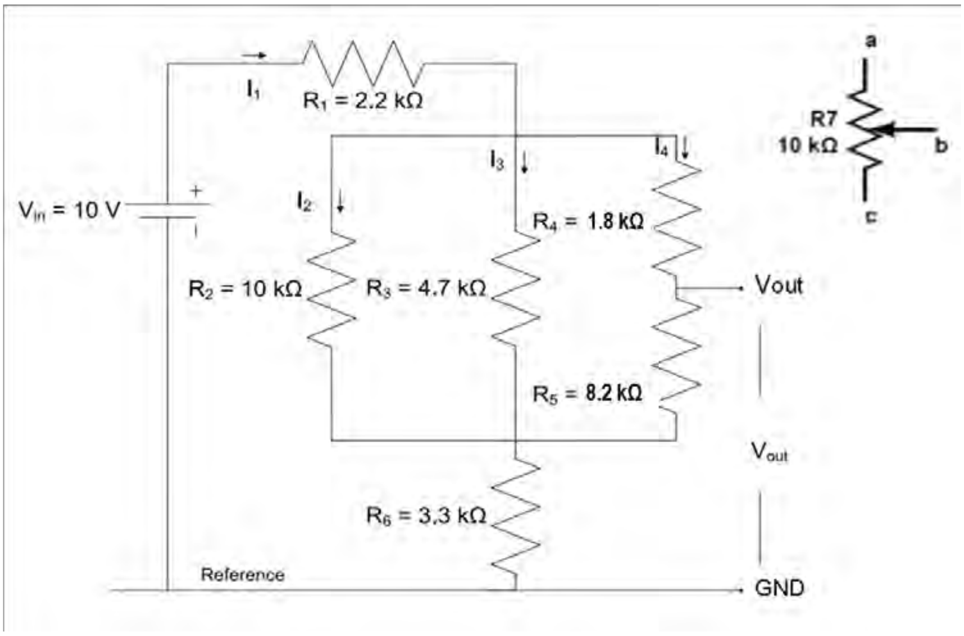
Introduction
These practical assignments are meant to let you become familiar with a 3 step sequence in which you model, build and verify your design. Within the *modeling* part you will design your circuit and do the necessary calculations in order to predict the outcome of your design. Then you *build* a prototype of your design on your breadboard. Finally you *verify* your design by doing the appropriate measurements. If necessary, you correct your model.

Regarding all of these assignments your report has to be structured as indicated in the '*Your final report DG291.pdf*' document which can be found in the DG291 Wiki pages.

Before you start a practical assignment, you should have studied all the chapters up to the one specified, Appendix B: Resistor series and values, Appendix D: Sources, Appendix E: Measurement equipment and errors, and Appendix G: building circuits. We are convinced that by first studying these chapters carefully you will gain time at the end. Try to make yourself familiar with the equipment by first doing some simple measurements like measuring the voltage over and the current through a resistor which you have connected to a power supply. Take in mind the power limits of the power supply, the resistors, transistors, and opamps.

Assignment (ref. Chapter 3)

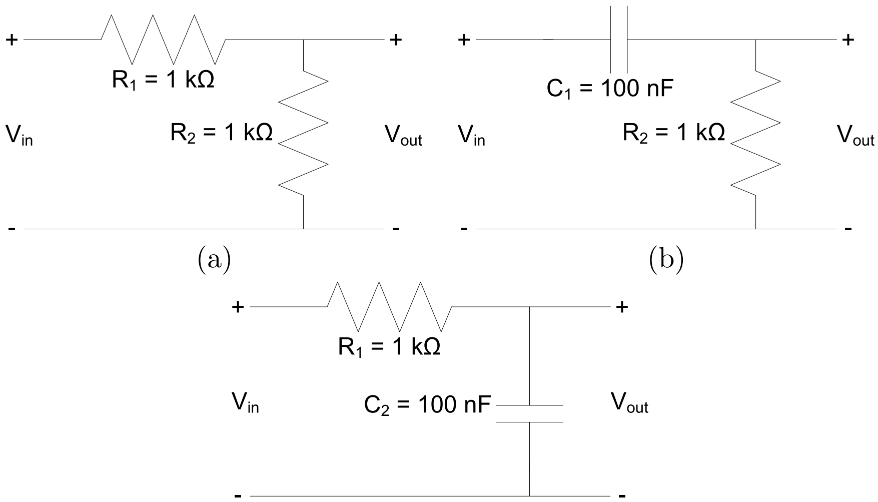
Assignment 1:
Given the following circuit:



1. Build the circuit on your breadboard (use resistors with a tolerance of 1% and 1/3 W).
2. Calculate $R_{retotal}$, I_1, I_2, I_3, I_4, and V_{out}.
3. Confirm your calculation by measuring $R_{retotal}$, I_1, I_2, I_3, I_4, and V_{out}.
   Make sure that you remove the source from the circuit before you measure $R_{retotal}$.
4. Take a potmeter with a resistance value of 10 kΩ. Put it somewhere on your breadboard and measure the resistance value between pins a and b, and between pins b and c while you turn the knob (see figure above for pins a, b and c).
5. Replace R4 and R5 by the potmeter R7 (connect pins a and c of the potmeter). Adjust the knob on the potmeter until you get an output voltage Vout = 5 V on potmeter pin b (between pin b and GND).
6. Now take the potmeter away from the circuit and, without readjusting the knob, measure the resistance value between pins a and b (Rab) and between pins b and c (Rbc).
7. Verify your measurement results by calculation: Imagine you replace R4 and R5 in the schematic above by Rab and Rbc respectively as measured in the previous question. Now calculate the voltage on the joint of these 2 resistors.

Assignment (ref. Chapter 4)

Assignment 2:
You have learned to do calculations on RC-filters. Now you are going to do measurements on them to verify the theory that was discussed in Chapter 4.
Given the following three circuits.



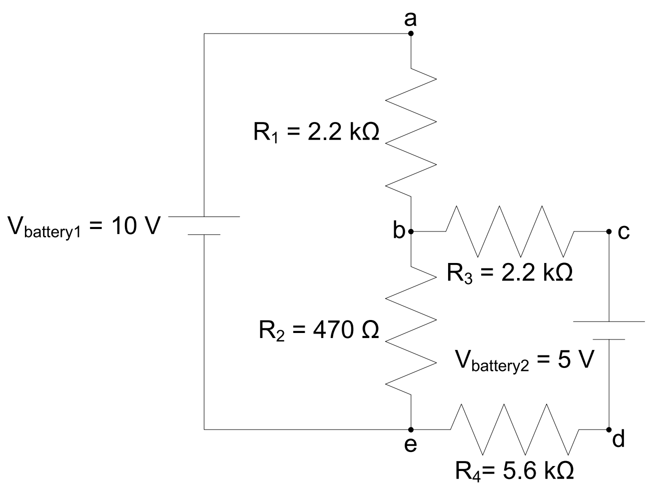
(a) (b) (c)

1. Build circuit (a). Connect a function generator to the input and choose a sine wave with an amplitude of 2V top-top (connect the input to channel one of your oscilloscope), and leave it unchanged. Connect channel two of your oscilloscope to the output.
2. Measure the amplitude of V_{out} at 100 Hz, 1 kHz and 100 Khz with an oscilloscope.
3. Build circuit (b) by replacing R_1 with a capacitor of 100 nF.
   First: calculate the amplitude of V_{out} for 100 Hz, 1 kHz and 100 kHz.
   Then: verify your result by measuring V_{out} at the three frequencies with the oscilloscope.
4. Build circuit (c) by interchanging the resistor and the capacitor.
   First: calculate the amplitude for V_{out} for 100 Hz, 1 kHz and 100 kHz.
   Then: verify your result by measuring V_{out} at the three frequencies with the oscilloscope.
5. Calculate the cut-off frequency f_c of the last filter. Verify your result by measuring V_{out} at this frequency with the oscilloscope.
6. What type of filter is circuit (c) (high- or low-pass)?

- *Show your calculations and present the results of both your calculations and measurements side by side in a table !*
- *Draw your conclusions !*

Assignment (ref. Chapter 6)

Assignment 3:
Given the following circuit:



1. Calculate the voltage drops across the resistors and the currents through them. Use KVL and indicate the loops you've chosen. Take node 'e' as reference.
2. Build the circuit and verify the calculated values.
3. Show your calculations and present the results of both the calculations and measurements side by side in a table.
4. Draw your conclusions.

Assignment (ref. Chapter 8)

Assignment 4:
Take a red LED and determine the relationship between I_d and V_d. Start at $V_d = 0$ V and increment V_d with steps of 0.1 V until just above the 'knee' voltage.

Before you start, you must limit the current of the power supply to 500 mA.

- Show the results of your measurements in a table.
- Present a graph which shows the relation between I_d and V_d .
- How did you detect the 'knee' voltage?

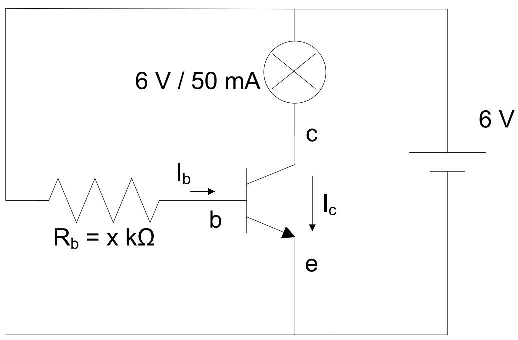
Assignment (ref. Chapter 9)

Assignment 5:
Study the saturation effect of a BC550 transistor by taking different resistance values for R_b (use resistors from the E12 series).

- Take $V_{powersupply} = 6$ V
- Take a lamp of 6 V / 50 mA
- Take as R_b resistance values 100 $K\Omega$, $10K\Omega$, $4K7$, $2K2$, $1K\Omega$ and 470Ω.
- For every R_b value measure I_c, V_{ce} and V_{be}
- Present your results in a table.

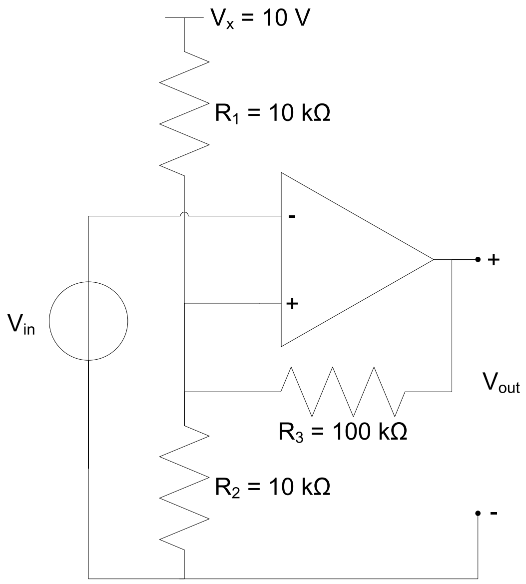
When you reach saturation, what can you tell about V_{ce} and V_{be}?
Did you expect this?

Hint: build a circuit like given below.



Assignment (ref. Chapter 10)

Assignment 6:
Given the following comparator circuit:



1. Build the circuit. You can use the LM324 type opamp, of which the datasheet can be found on the Internet.
2. Calculate the two transition levels and verify them.
3. Draw the hysteresis graph and indicate the measured values (see Figure 10.7).

15

Final assignment

Introduction
This final assignment provides you with a kind of ultimate challenge. You'll be asked to show that you are able to put together several aspects of what you've learned. A complete feedback controller based system has to be analysed, designed, build and tested.

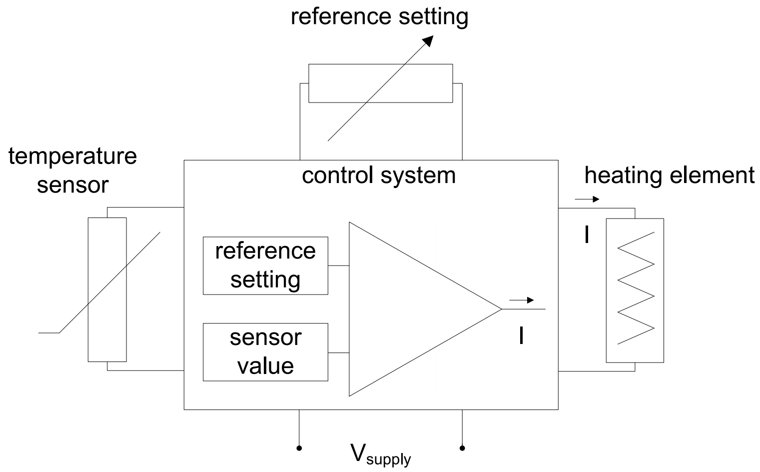
Regarding this final assignment your report has to contain:

1. **A short introduction to this assignment.**
2. **Your analysis of the sensor and actuator applied including data provided by the manufacturer.**
3. **A decent drawing of your circuit including indications of types and values.**
4. **An explanation of the working of your circuit including calculations.**
5. **Proof that you really build it by showing some pictures. Add an explanation, don't let us guess what it supposes to mean.**
6. **Draw conclusions, reflect on your findings. Are you able to quickly change the temperature inside the box? Why, or why not?**

Assignment (ref. Chapter 11)

Central heating

Given the closed-loop control system of a temperature controller:



Design and build the control system using a NTC resistor, a comparator, a potmeter and an electronic power switch. For the heating element you can use a power resistor (e.g. 10 Ohms/10 Watt)

Use a tiny closed foam or cardboard box to build the 'room' to be heated. The NTC sensor should be positioned close to the heating element. Furthermore, to verify the correct functioning of your control system, you need to use a calibrated electronic temperature meter. Add an LED to the output of your controller to check whether the output is switched on or of.

- Analyse your sensor: find the R_{value} vs T graph of your NTC.
- Analyse your actuator: what power source do you need to heat it up to it's maximum?
- Present your circuit: draw the schematic and explain how you think it works.
- Indicate the various parts: e.g. type of opamp, transistor, power resistor, value of parts etc.
- Show that you've build it and that it actually works !!
- Are you able to quickly change the temperature inside the box? Why, or why not?

Optional: Can you give a calibration graph of your system? (i.e. the relationship between the reference setting (resistance value) and the measured temperature).

16

Appendix A - The classical power source

In this reader you've been working with a power supply when you were doing measurements. In this appendix we'll shortly explain how this power-supplies work, and what kind of components they contain.
In figure 16.1 the electrical circuit of a voltage source is depicted.

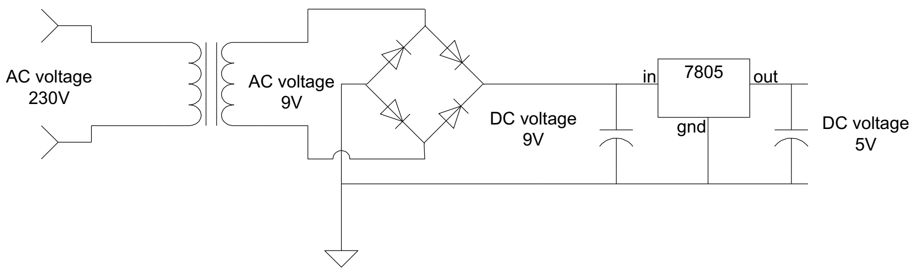


Figure 16.1: *Classical Voltage source.*

At the complete left you see the input, which is in our case a AC voltage of 230V. This voltage is supplied by the electrical grid in the Netherlands.
Because this is a very high voltage to work with, we lower this AC voltage to an AC voltage of 9V. This is done with a transformer, which was discussed in the chapter about inductors.
We want a DC voltage as output, so we have to rectify this AC voltage. This can be done with a so called diode-bridge, which contains four diodes.

16.1 Diode bridge

The diode bridge has four diodes. The input of this bridge is a AC signal, which has two connections.

When the input connected on the top of the diamond-structure is **positive** with respect to the one connected at the lowest corner, current flows from the top corner to the positive output, and returns to the other input connection via the negative output through the diode bridge.

When the input connected on the top of the structure is **negative** with respect to the one connected at the lowest corner, current flows from the lowest corner to the positive output and returns to the upper corner via the negative output of the bridge.

The outputs of this rectifier bridge can be depicted in figure 16.2 In this figure the upper graph is the input signal, which is AC. The second graph is an the half-wave, which can be seen as inputs for both situations that are discussed, and the third graph depicts the rectified output signal.

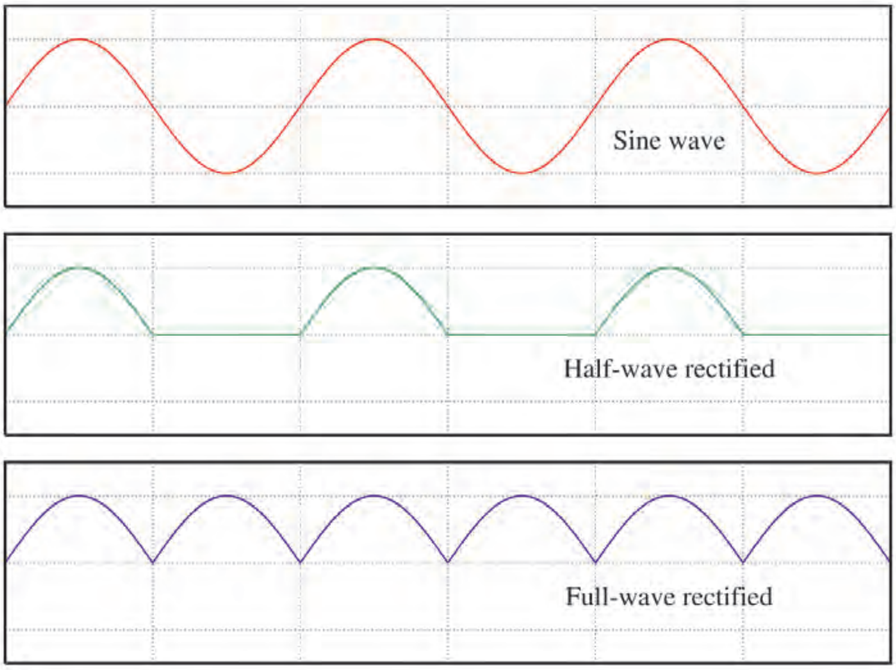


Figure 16.2: *Rectifier signals for input and output.*

You see that this output signal is not completely DC yet. That's why we connect a capacitor between the negative and positive output of the rectifier. This capacitor more or less flattens the fluctuations of the full wave rectified voltage.

16.2 Voltage regulator

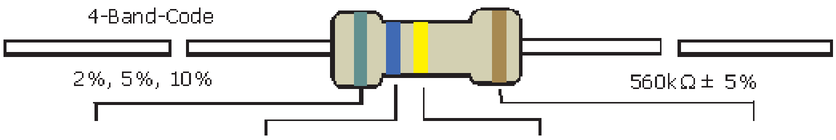
Now we've got a DC voltage of 9V, we want to downgrade it to a voltage of about 5V. This can easily be done with use of a voltage regulator. An example of a voltage regulator is the 7805. How this IC must be connected can be read in the data sheet of the component. Also the precise properties are discussed there.

At the output of this voltage regulator we place a capacitor, to get a really clean DC output voltage of 5V!

17

Appendix B - Resistor series and values

On most low-power resistors you see some color-code printed on it. This color code depicts the value of the resistor. The exact value of the resistor can be found using Figure 17.1.



COLOR	1st BAND	2nd BAND	3rd BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	±0.5% (D)
Blue	6	6	6	1MΩ	±0.25% (C)
Violet	7	7	7	10MΩ	±0.10% (B)
Grey	8	8	8		±0.05%
White	9	9	9		
Gold				0.1	± 5% (J)
Silver				0.01	± 10% (K)

Figure 17.1: *Resistor color code overview to calculate resistance value of a resistor.*

If a resistor has a total of four bands, it will contain two digits, a multiplier, and a tolerance band. If a resistor has five bands and is a newer one, it most likely has three digits, a multiplier, and a tolerance band.

Lets say you have a resistor with a yellow, violet, red, and gold band. The first band represents the first digit, and a yellow band means 4, so the first digit in the value of the resistor is 4. The next band is violet, meaning 7 is our next digit. The next band is our multiplier, and will tell us to what power of 10 we must multiply the first two digits by. A red band in the multiplier means 10^2, so to get the value of the resistor we must multiply 47 by 10^2. This gives us 4700 Ω, or 4.7 kΩ. The last (keep this on the right) band is the tolerance, a gold band means that the actual resistance value must be within ± 5% of the specified value. So the actual value of the resistor may be anywhere from 4,465 Ω to 4,935 Ω.

Sometimes figuring out what end is what can be difficult. Some resistors will have the bands close to one end, indicating the starting point. On others, the last band will be larger than any of the others. But in many resistors it is common for all stripes to be evenly distributed and equal in width. If you have a gold or silver stripe, the end that stripe is furthest from is your starting point, because we know gold and silver cannot be used for any of the digit values. But sometimes you might have a resistor such as brown, green, black, red, brown. It could be either be read as a 15 kΩ ± 1% or 12 MΩ ± 1% resistor. If you are stuck in a situation where you cannot figure out what end is what, the next best thing is to just get a DMM and measure it.

Resistors (and also capacitors) are manufactured in **standard** values, known as the E series. The most common series is the E12, where there are 12 resistors in a decade (**10-12-15-18-22-27-33-39-47-56-68-82**). Other series include the E6, E24, E48, and E96 series.

The Electronic Industries Association (EIA), and other authorities, specify standard values for resistors, sometimes referred to as the "preferred value" system. The preferred value system has its origins in the early years of the last century at a time when most resistors were carbon-graphite with relatively poor manufacturing tolerances. The rationale is simple - select values for components based on the tolerances with which they are able to be manufactured. Using 10% tolerance devices as an example, suppose that the first preferred value is 100 ohms. It makes little sense to produce a 105 ohm resistor since 105 ohms falls within the 10% tolerance range of the 100 ohm resistor. The next reasonable value is 120 ohms because the 100 ohm resistor with a 10% tolerance is expected to have a value somewhere between 90 and 110 ohms. The 120 ohm resistor has a value ranging between 110 and 130 ohms. Following this logic, the preferred values for 10% tolerance resistors between 100 and 1,000 ohms would be 100, 120, 150, 180, 220, 270, 330 and so on (rounded appropriately); this is the E12 series shown in the table below. The EIA "E" series specify the preferred values for various tolerances. The number following the "E" specifies the number of logarithmic steps per decade.

At our university you will only find resistor and capacitor values from the E12 series. The exact value of such a resistor can only be one of the values of the series multiplied with a power of ten. So in the E-lab you can ask for a resistor of 180 Ω ($18 \cdot 10^1$) or 3.3 kΩ ($33 \cdot 10^2$) but not for an 5.2 MΩ ($52 \cdot 10^5$). If you need such a value you will have to compound it by series and/or parallel connecting resistors which exist in the E12 range.

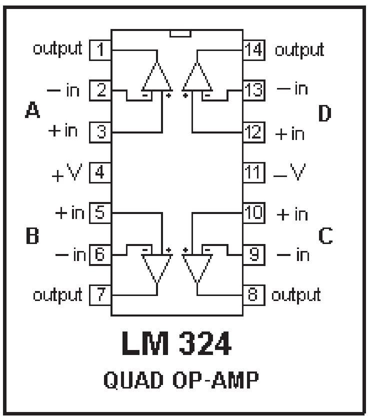
18

Appendix C - ICs and data sheets

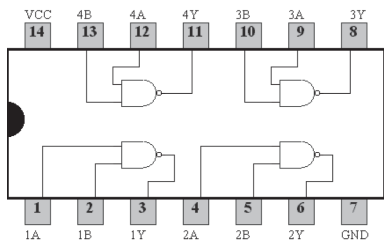
18.1 ICs

IC is an abbreviation for Integrated Circuit. An IC contains diodes, transistors, resistors and capacitors on one chip (made of silicon and in most situations some mm^2 large). IC's are categorized in different families: logical (digital) ICs, analog ICs, linear ICs and so on. IC's are also divided in the technology where they are produced. So there are sub-categories like CMOS, TTL and so on in the category digital ICs.

The figures 18.1a gives a picture of the pin-layout of the top-view of an IC, which contains 4 identical opamps in its case. Figure 18.2 is another kind of IC which contains 4 two-input NAND ports. Both ICs are accommodated in the same kind of case: a 14-pins DIL (Dual In Line) case. Figure 18.1a shows the top-view of the case of the LM324 IC.



(a) Pin layout.



(b) Photo.

Figure 18.1: *LM324.*

The LM324, which is depicted in figure 18.1 has common supply connections: The positive supply can be connected to pin 4 (+V), and the negative supply can be connected to pin 11 (-V) or (GND

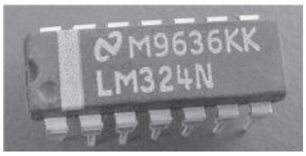


Figure 18.2: *4 NAND ports.*

or 0V). The digital IC depicted in figure 18.2 has also a common source supply.
For connection details and specification of most electrical components you have to consult a data sheet. When you know the type-number of such a component, those data sheets can be easily found on the internet or in the library.

The carving of an IC depicts where to start counting for the pin numbering. Just like with most electrical components, you've to count counter clock-wise.

Note: A special sub-category of ICs are the so called CMOS-switches (for example the 4066). In this reader we've already discussed the relay, which can switch circuits "on distance". If such a switch needs to switch a small current and voltage, and the on-resistance for the switch-contact doesn't need to be very small, you can use a CMOS-switch. One benefit of this type of switch is that there are more switches in one IC, which leads to compact circuits.

18.2 Data sheets

If you are designing a circuit and need some components, you often have to check some variants of those components. Each of those components have different electrical properties, frequency-dependencies or temperature-dependencies. For each of those components there will be a data sheet, which contains all this information.

A data sheet contains globally:

- A logical and/or functional description of the component
- An overview of the connection-pins and their names (or pin numbers)
- The most important properties
- Common applications which use the component
- The conditions under which the component must or can operate. You can think about temperature conditions or voltages
- Electrical specifications

Each manufacturer of ICs produces special handbooks which contain all data sheet of their electrical components. Nowadays most data sheets are also available on the Internet.
An example of a data sheet can be found in figure 18.3.

VISHAY

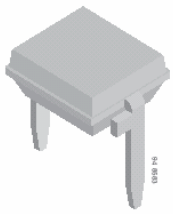
BPW34

Vishay Semiconductors

Silicon PIN Photodiode

Description

The BPW34 is a high speed and high sensitive PIN photodiode in a miniature flat plastic package. Its top view construction makes it ideal as a low cost replacement of TO-5 devices in many applications.
Due to its waterclear epoxy the device is sensitive to visible and infrared radiation. The large active area combined with a flat case gives a high sensitivity at a wide viewing angle.



Features

- Large radiant sensitive area (A = 7.5 mm^2)
- Wide angle of half sensitivity $\varphi = \pm 65°$
- High photo sensitivity
- Fast response times
- Small junction capacitance
- Suitable for visible and near infrared radiation
- Lead-free device

Applications

High speed photo detector

Absolute Maximum Ratings

T_{amb} = 25 °C, unless otherwise specified

Parameter	Test condition	Symbol	Value	Unit
Reverse Voltage		V_R	60	V
Power Dissipation	$T_{amb} \leq 25$ °C	P_V	215	mW
Junction Temperature		T_j	100	°C
Storage Temperature Range		T_{stg}	- 55 to + 100	°C
Soldering Temperature	$t \leq 3$ s	T_{sd}	260	°C
Thermal Resistance Junction/Ambient		R_{thJA}	350	K/W

Electrical Characteristics

T_{amb} = 25 °C, unless otherwise specified

Parameter	Test condition	Symbol	Min	Typ.	Max	Unit
Breakdown Voltage	I_R = 100 µA, E = 0	$V_{(BR)}$	60			V
Reverse Dark Current	V_R = 10 V, E = 0	I_{ro}		2	30	nA
Diode capacitance	V_R = 0 V, f = 1 MHz, E = 0	C_D		70		pF
	V_R = 3 V, f = 1 MHz, E = 0	C_D		25	40	pF

Figure 18.3: *Example of a data sheet (Photodiode).*

19

Appendix D - Sources

19.1 DC Voltage Sources

19.1.1 The battery

The battery is the most well known DC voltage source. It is broadly used as a portable power source when the required source needs to be compact. In a battery a chemical reaction will generate a constant voltage between its positive and its negative side, which in turn can cause a current to flow when the battery is attached to a circuit. The battery lasts only as long as the chemical reaction that generates the voltage lasts, so it can only provide a limited amount of power during a limited period of time. Because the voltage of the battery is constant, this means a certain current for a certain period of time. Therefore the amount of energy that a battery can provide is given in (milli) Amperes times the number of hours: (m)Ah. A battery that can provide 3000 mAh can for example provide a current of 3000 milli Amperes for one hour, or a current of 1500 Amperes for 2 hours before it fails.

When you use a battery you should take into account that it is not an ideal source. The voltage will not stay constant during the life time of the battery, but will slowly fall when more energy is used up. Further a battery can not provide an infinite current. Temperature may also influence the performance of the battery. In a cold environment the chemical reaction that generates the voltage will work slower, causing the battery voltage to drop.

19.1.2 The laboratory power supply

The laboratory power supply is a more sophisticated voltage source that is often used in experimental set ups. Their voltage can be adjusted to the desired value. This makes them ideal for experimental circuits when you will need to adjust the input voltage of your circuit. They are not practical for use in commercial products because they are large and need to be attached to the 230V net.

Mostly these sources have an analog display, 2 switches, 2 turning knobs and 3 sockets. One of the switches is used to turn the source on and off. The other switches the quantity you read on the display between Amperes and Volts. The turning knob marked V lets you adjust the voltage to the desired value and the turning knob marked I lets you adjust the maximum amount of current

Figure 19.1: *A laboratory power supply.*

the source can deliver. When this maximum current is reached, the voltage of the source will only be as high as is needed to cause this current. The sockets marked '+' and '-' are the positive and negative output of the source and are to be connected to the positive and negative supply of your circuit. The socket marked with a ground symbol should not be used as the negative input for your circuit. This is the ground of the chassis of the source and can be used to (as shown in Figure 19.1) to set the negative supply potential to the 'real' ground level.

A laboratory voltage supply behaves much more like an ideal source. The voltage will not drop when time passes. Further the voltage will not drop when the current rises, as long as the current will stay below the maximum value you have selected. And of course you can accurately select the desired value of the voltage.

19.2 The Function Generator

The function generator is a source that is used to generate signals. This can be very useful to test your circuit with. Function generators can generate a variety of waveforms like sine, square, triangular and ramp waveforms.
Depending on the type of function generator, there are different ways in which to adjust the prop-

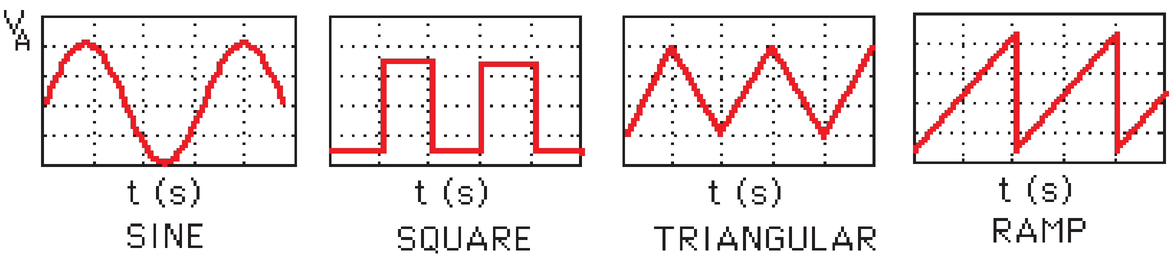


Figure 19.2: *Waveforms generated by a function generator.*

erties of the signal. With older function generators this is done with turning knobs, with modern function generators this is done with buttons and a display.

The properties of the signal you can adjust include:

- Amplitude: The amplitude of the signal in volts
- Frequency: The frequency of the signal (the number of repetitions per second of the signal) in Hertz
- DC offset value: The DC voltage around which the signal moves. For example a sine wave signal with an amplitude of 1V and a DC offset of +3V will have a maximum voltage of 4V (DC offset + amplitude) and a minimum of 2 V (DC offset - amplitude).

As an example, look at a sine function described by the following equation,
$v(t) = V_a \cdot sin(2\pi f) + V_{off}$
in which f is the frequency, V_a the amplitude, and V_{off} the offset voltage as shown in Figure 19.3. Instead of amplitude one often uses the RMS (Root Mean Square) value to express the signal voltage level. For a sine wave the RMS value is the amplitude divided by the square root of 2 or $V_{RMS} = V_a/1.41$.
More advanced function generators can let you select different wave forms, AM and FM modulated

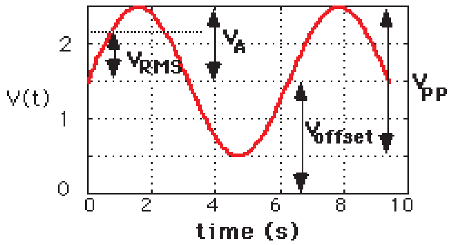


Figure 19.3: *Sine wave with amplitude V_a, frequency f, and offset V_{off}.*

signals and several other properties.

A function generator is connected to a circuit in the same way as a DC voltage source.

A function generator is not an ideal source. It has an output impedance (figure 19.4a) that can influence the actual voltage applied to the circuit. Further a function generator should not be used as a power supply. The example below explains the effect of an output impedance of 50Ω.

Important is that this output resistance of the function generator has a value of 50 Ohm. This implies that the actual output voltage one measures over the load will vary with the load resistance because of the voltage divider, as shown in Figure 19.4b. The output amplitude is calibrated for a 50 Ohm load resistance, which means that the voltage shown on the function generators display panel corresponds to the actual voltage V_{load} over the load only when the load is equal to 50 Ohm. In other words, the value of Vgen is double of the value displayed (or selected) by the function generator. If the function generator's output is measured with no load connected (=open circuit or infinite resistance), the output voltage will be twice the displayed amplitude. Thus, be careful when applying the output voltage of the function generator to a circuit whose input resistance is different from 50 Ohm. In general, it is a good practice to check the amplitude of the waveform using a modern oscilloscope or a high quality DMM instead of relying on the function generator display reading.

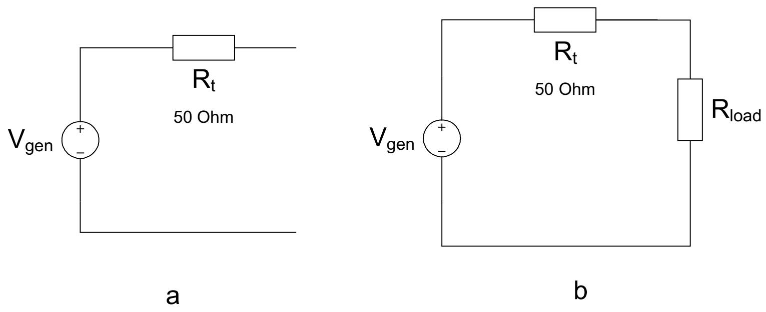
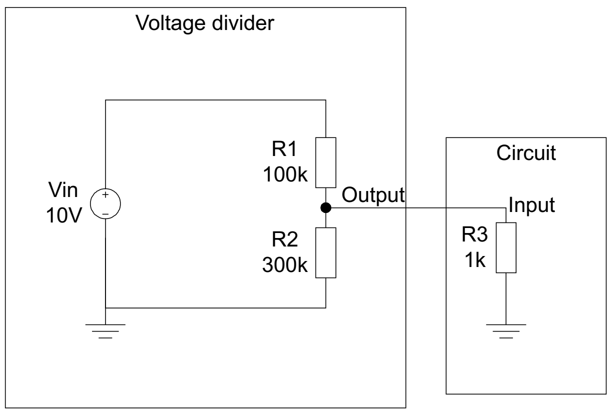


Figure 19.4: *A function generator has an output impedance (a) which makes a voltage divider when the generator is attached to a circuit (load resistance) (b).*

19.3 Other circuits as sources

Often you will use the output of another circuit as the input (signal) for your circuit. Then you should really pay attention that the circuits do not influence each other for your circuits will not have ideal input and output impedances. For example the voltage divider shown in figure 19.5, consisting of a $100k\Omega$ resistor and a $300k\Omega$ resistor to create an output voltage of 7,5 V from a source voltage of 10 V. For instance when you connect the output to a circuit having an input impedance of $1k\Omega$, the input voltage will drop from 7,5 V to almost 0V. Regard input and output circuit as one network and apply what you've learned in the 'Circuits' course to analyse it's behaviour.

When you work with signals and your loading (sub)system is a complex network (network consist of a mixture of capacitors, inductors and/or resistors), this can deform your input signal. For more information on connecting circuits see chapter 13.



(a) Network as source.

Figure 19.5: *The voltage divider will give an output of 7,5V, but when it is connected to a circuit with an input resistance of $1k\Omega$ the output voltage will drop to about 0.1V.*

20

Appendix E - Measurement equipment and errors

20.1 The Multimeter

The multimeter is a multi functional piece of equipment that can measure the basic circuit variables voltage, resistance and current. Multimeters can be found in many different forms, varying from simple hand held multimeters to sophisticated digital multimeters.

A handheld multimeter mostly has 3 inputs: The positive input, the negative or common input and a positive input for high currents. Figure 20.1 shows how you should measure resistance, voltage and current. Commonly red wires are used to connect the positive input to the objects positive node and black wires are used to connect the negative (common) input to the objects negative node. Figure 20.2 shows a handheld digital multimeter.

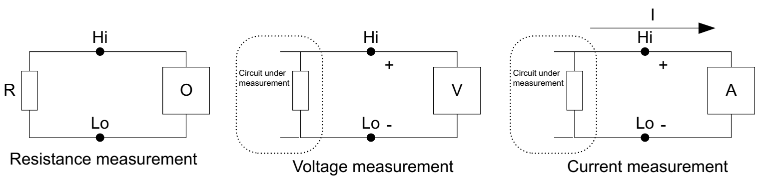


Figure 20.1: *Measurement of resistance, voltage and current with a multimeter*

For resistance measurements, you connect the input and output of the multimeter over the resistor. It does not matter at which side you put the positive and negative input. Make sure that all voltage sources are shut off when you perform a resistance measurement or else your multimeter might get damaged. When measuring inside a circuit, make sure that you are only measuring the part you want and that no other parts of the circuit are connected to your input and output to influence your results. For voltage measurements you connect the multimeter in *parallel* with the circuit element so that you measure the voltage *across* the element. Pay attention that you connect the positive input of the multimeter to the positive voltage and the negative input to the negative voltage. Make sure that you select the AC range when you are measuring AC voltages and the DC range for DC voltages. In case of a current measurement, one must put the multimeter in series with the element in order to measure the current through the element. That involves opening the circuit in order to

Figure 20.2: *A handheld multimeter. On the right the three inputs. The black is the negative or common input, the red one above the positive input for measuring resistance, voltage and low currents. The red input on top is only for measuring high currents. The knob in the middle is used to set the range of the measurement. This can change the values of your input resistance. This multimeter gives its results in $3\frac{1}{2}$ significant digits. You need to know the accuracy of this multimeter and its input resistance before you can interpret the results.*

insert the multimeter in the circuit loop. The current you measure will actually flow through your multimeter. A fuse is built in to prevent damage when the current is too high. Your multimeter will often show the maximum allowed current printed near the positive (red) input terminals. Most multimeters have a separate input to measure higher currents.

Before you start your measurement you should select the right range to measure, for example 200Ω, 20kΩ, 2V, 20V, 200V AC, 20mA or, after switching the input to the high current input, 10A. With handheld multimeters you can often select the range with a turning knob, with digital multimeters with buttons. When measuring small resistance values the resistance of the wires used to connect the resistor to the multimeter may be of a significant value. When measuring high resistance values avoid touching the circuit with your hands, since your skin and body resistance can be connected in parallel to your measurement circuit and thus influencing it's behaviour.

Figure 20.3: *The Agilent 34401A desktop multimeter. The ones used in the EE-labs.*

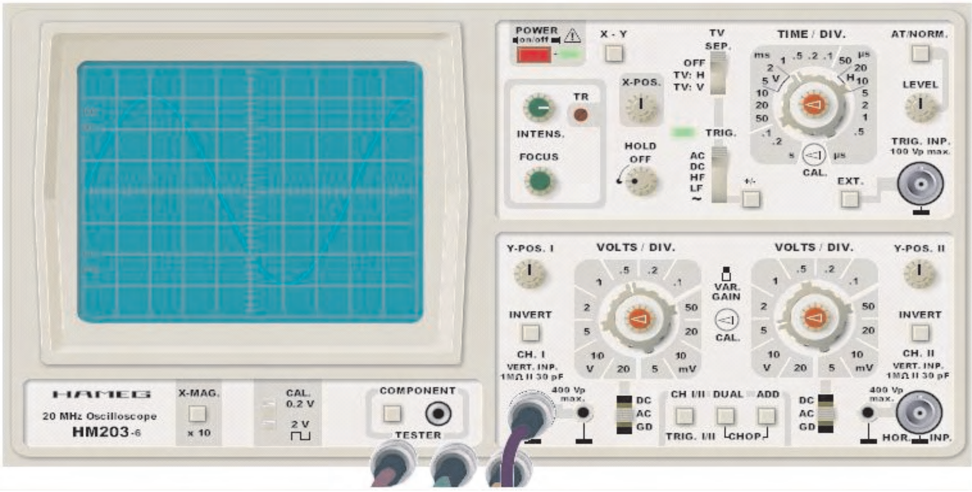


Figure 20.4: *A traditional oscilloscope with all its functions.*

20.2 The Oscilloscope

The oscilloscope is an often used piece of measurement equipment for measuring varying voltages. The oscilloscope, or in short scope, is ideal to study signals that vary in time, even if their frequency is very high. With most scopes you are able to view a maximum of 2 signals as a function of time on its display. For this purpose there are 2 connections (CH1 and CH2) on the front of the scope. Figure 20.4 shows an oscilloscope with all its functions.

An oscilloscope is in fact a voltmeter with a positive and a negative connection. To measure voltage across an element you will have to connect the scope in parallel with that object. But unlike the multimeter, where 2 cables are used, one for the positive input and one for the negative input, scopes use coax cables as an input. A coax cable is in fact nothing else then 2 combined wires. The wire in the middle of the cable is the positive wire, the wire that is wrapped around it is the negative wire. When you want to connect a scope to your circuit you will mostly have to use an adapter that splits the coax cable to 2 separate wires. These wires will be red and black. The red wire is the positive wire and the black is the negative wire and connected to the 'ground' of the oscilloscope.

Signals are displayed on a small screen as a beam that follows the signals voltage in the up and down direction and that moves to the right as a function of time and starts left again when the beam has reached the end of the display. The brightness of the beam can be adjusted with the **INTENS** knob and the beam can be focused by the **FOCUS** knob.

Axis
The display of the oscilloscope is divided in squares of about 1 cm. Each square is one division. The signal that is measured is displayed on the scope's display. The X-axis represents time and the Y-axis represents amplitude (voltage). The value of each division on the Y-axis can be adjusted with the knob **VOLTS/DIV**. When you put this knob pointing to 1V this means that each division

(1 cm in the Y direction) corresponds with 1V. The value of each division on the X-axis can be adjusted with the knob **TIME/DIV**. When you put this knob pointing to 1msec this means that each division (1 cm in the X direction) corresponds with 1 msec.

Some scopes have the ability to shift the displayed signal left or right with the knob **X POSITION** and up or down with the knob **Y POSITION**. To adjust the time division some more you can use the button **X MAGN**.

Signal
With the buttons in the MODE field you can select the signal that is to be shown:

- **CH1 (2)**: Only the selected signal is shown
- **ADD**: The sum of both signals is shown
- **CHOP**: Both signals are shown. Use this for low frequencies. With CHOP first a small part of CH1 is seen, and then a part of CH2 and so until the display is filled.
- **ALT**: Both signals are shown. Use this for high frequencies. With ALT a display (full sweep) is first filled with the signal of CH1 and then a sweep with signal of CH2 and so on.

When you want to view both signals you will have to choose between these two options. Take the one that generates the most comfortable view.

Some scopes have the ability to invert the signal:

- **INVERT**: The signal is shown inverted.

Input coupling
It is also possible to choose how each signal on its own is shown:

- **AC**: Only the AC voltage component is shown. The DC voltage level is removed by connecting a capacitor in series with the input.
- **GND**: The 0 level of the input is shown.
- **DC**: The complete signal is shown, inclusive its DC component or offset level.

Triggering
The experience learns that many people that work with a scope for the first time have problems with *triggering*. Therefore read this part carefully. With *triggering* the scope is meant fixing of the beginning (time) of the shown signal on the display, by starting the horizontal time-axis on a certain moment/phase (=trigger).

A good triggering, meaning a good tuning of the trigger function, is necessary to get a clear and still image on the display of the scope, see figure 20.5. The scope always shows the signal a few times sequentially. If the beginning of each period of showing the image is arbitrarily to the signal, than it can become hard to view the signal.

With a bad triggering there is no relation between the beginning moment and the signal. This has as a result that the different periods of the signal, that are shown sequentially, will appear

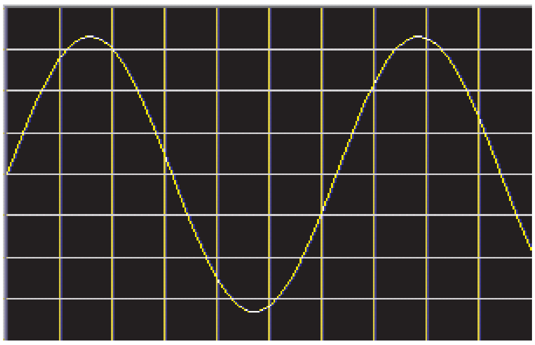


Figure 20.5: *A well triggered scope gives a clear signal on the display.*

totally different on the display. In the ideal situation all the periods are shown on top of each other and only 1 image is visible.

The scope determines the beginning moment using a signal that is to be chosen by the user. For that purpose each scope has several switches and buttons to make the right choices.

With **SOURCE** you decide from which signal the beginning moment has to be derived:

- **CH1 (2)**: The scope starts if the signal on the selected input satisfies the trigger conditions.
- **VERT MODE**: The scope starts if one of both signals satisfies the trigger conditions. This can cause confusing results if you have picked the ALT option to show the signal. Therefore you'd better not use this, and certainly not in combination with the **ALT** option. For the **ALT** option you actually need 2 beginning moments: one for channel 1 and one for channel 2. With the button **VERT MODE** you give the scope the freedom to choose a signal and so you will not know which signal the scope will choose on a certain moment.
- **EXT**: The scope starts if the signal on the input EXT satisfies the trigger conditions. This signal is not shown on the scope.
- **LINE**: The scope derives the beginning moment from the net voltage. This is very convenient if your signal is in phase with the net voltage.

Sometimes it will occur that starting conditions are selected that will never occur in the signal. With **MODE** you can determine what has to happen in that case.

- **AUTO**: The scope automatically starts after a while (about 100 ms). This waiting period is independent of the chosen time scale and therefore it is better to use **NORM** for low frequencies (till 10Hz).
- **NORM**: The scope does not start until all trigger conditions are satisfied.
- **XY**: With this function you plot the two signals CH1 (vertically) and CH2 (horizontally) to each other and in fact no triggering is used.

Apart from selecting a source for the trigger signal there should also be determined on which edge of the signal should be triggered. This can be selected with the knob/button **SLOPE**. The **LEVEL** knob allows for adjusting the level of the trigger signal on which triggering should start. Mostly an internal triggering is enough, the trigger moment is derived from the signal that is to be measured, and sometimes it is convenient to use for example the net voltage (LINE) as a trigger source.

NOTE 1: When you want to practice the use of an oscilloscope you should take a look at http://www.virtual-oscilloscope.com. This site has a very realistic simulator of a traditional oscilloscope and you can find more information about all its functions. You should practice a little on this site before you start to work with a scope.

NOTE 2: Nowadays more and more analog, valve based, oscilloscopes are being replaced by digitally controlled flat screen oscilloscopes. Beside the fact that they are more compact the main advance is that they can store waveforms into their digital memory. This feature allows for observing single occurrences of a wave. Often these types of scopes are loaded with extra features (and so extra knobs on the front) like performing various types of calculations on waveforms. Most of the times basic controls like vertical sensitivity and time-base can easily be identified. Providing you with in-depth information on these types of oscilloscopes goes beyond the scope of this reader.

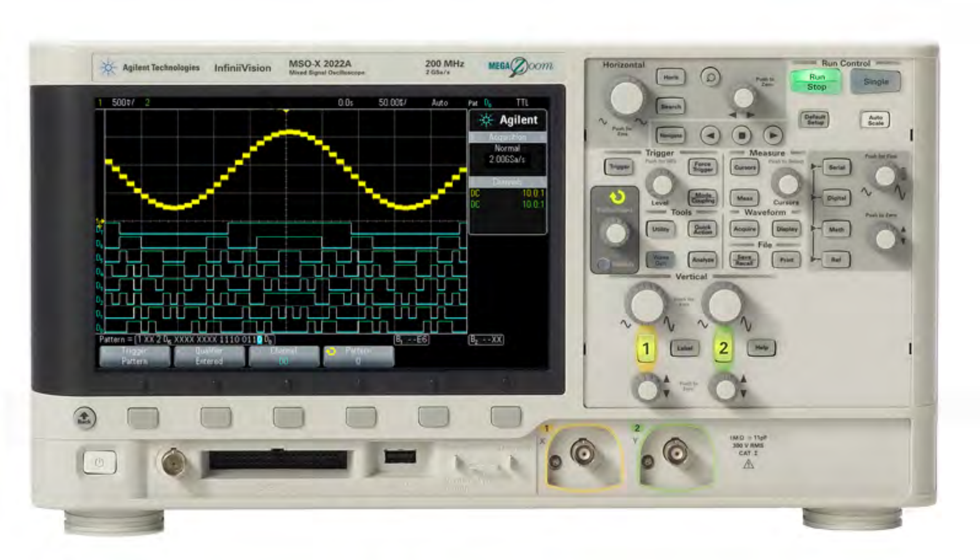


Figure 20.6: *The Agilent DSO-X2002a digital oscilloscope; the ones used in the EE-labs.*

20.3 Errors

When you are measuring values you should know how to interpret your results. There are several aspects you have to take into account.

Significant figures

The term significant figures (significant digits) refers to particular digits in a number, to be more precise: it refers to the digits that contribute to the precision of the number. Only the digits of a number that are the result of an actual measurement are significant.

When writing and interpreting measured numbers you can keep the following rules in mind when you want to know more about the significant digits, and therefore the precision, of that number.

- All non-zero digits are significant. *42 has two significant digits and 832.56 has five.*
- All zeros that appear between non-zero digits are significant. *1.051.001 has seven significant digits*
- Zeros in front of the first non-zero digit (leading zeros) are not significant, they don't contribute to the precision of the number *0.0012 has two significant digits.*
- All digits in a number that contain a decimal comma are significant, even in case of a zero at the end of the number (trailing zero). *3.56 Has three significant digits 2.500 has four*
- If there are trailing zeros but no decimal point the significance of the zeros can be argued about. In some cases the zeros are non-significant but in most cases, especially when the scientific notation is used, those trailing zeros are significant. *1200 Can have two or four significant digits depending on the contextual meaning of the number. As 10 dozen equals 1200, it has four, when you say "I paid 1200 Euro for that couch" then it most likely will have only two, because the chance that the real price was something like 1198.95 is very high.*

To make sure that there is no misunderstanding about the amount of significant digits in a number you can resort to the scientific or engineering notation. In the scientific notation all numbers are written like this: $a \cdot 10^b$ with $(1 \leq |a| < 10)$ and b an integer. This way, if there are any trailing zeros, they will be after the decimal point and therefore are significant. *1200 as in 10 dozen will then become* $1.200 \cdot 10^3$ *and 1200 as in the couch example will become* $1.2 \cdot 10^3$. The engineering notation is similar to the scientific one; only the powers of ten have to be multiples of three.
There are a few special cases when it comes to significant figures. Here are a few examples:

Defined numbers: Every number that has a mathematical definition is exact, and therefore every digit you display for this number is significant. *The square root of 2* $(= 1.4142135\ldots)$, *the base of the natural logarithms* e $(= 2.781828\cdots)$ *and* $\pi (= 3.14159\ldots)$ *have a mathematical definition and therefore are exact.* Defined unit conversion factors are also exact; *there are exactly 2.54 cm to the inch.*

Integers: Things you count are exact (assuming you counted right) so all the digits in an exact integer are significant.

Rational fractions: Any fraction made of exact integers is itself exact. So if you convert the fraction to a rational number every digit is significant. Be aware that the quantity described must be exact. Therefore the ratio 15cm to 10cm (15cm/10cm) is not exact, because we cannot measure lengths with unlimited precision.

Calculating with significant figures

Rounding off

When computing with measured values, either with a calculator or a computer, you will often end up with with many digits displayed. Because computing cannot increase the accuracy of the measurement we must decide how many of those digits are significant and then find a way to round the result back to the appropriate number of digits. Rounding uses a fixed set of rules. If you are going to round the value to n significant numbers then:

- If the first non-significant digit is greater than 5, round up the last significant digit.
- If the first non-significant digit is equal to 5 and followed by non-zero digits, round up the last significant digit.
- If the first non-significant digit is equal to 5 and not followed by any other digits or followed by zeros, round up or down so that the last significant digit that is left is even. (If you would always round up in this case the result would get distorted for very large data sets. This one is often applied in surveying).
- If the first non-significant digit is less than 5, discard all of the non-significant digits .

If you do not round after a computation you imply greater accuracy than you measured

Rounding off after mathematical operations
The number of significant digits you should keep depends on the mathematical operation you preform

- *Multiplication or division.* The case of multiplication or division is the simplest. The number of significant digits you should keep in the result is equal to the amount of significant digits in the least accurate value used in the computation. Least accurate means the smallest number of significant digits.

  $273.92 \cdot 3.25 = 890.24$ *is rounded to* 890 *(The least accurate value is* 3.25 *it has three significant digits so the outcome is rounded to three significant digits).*

  $\frac{1}{3} \cdot 5.20 = 1.73333\ldots$ *is rounded to* 1.73 *(The least accurate value is* 5.20 *it has three significant digits so the outcome is rounded to three significant digits. (* $\frac{1}{3}$ *is exact because 3 is an integer).*

  $1.97 \cdot 2 = 3.94$ *is rounded to* 4 *(The least accurate value is* 2 *it has one significant digit so the outcome is rounded to one significant digit).*

  $2.0 \cdot \pi = 6.28318\ldots$ *is rounded to* $6,3$ *(The least accurate value is* $2,0$ *it has two significant digits so the outcome is rounded to two significant digits).*

- *Addition or subtraction.* For addition and subtraction the procedure is a bit different. The amount of significant numbers is not as important any more, but the significance of the given numbers is. The results are rounded to the position of the least significant digit in the most uncertain number in the calculation.

  $45.67 + 65.765 = 111.435$ *is rounded to* 111.43 *(The least accurate value is* 45.67*. The position of the least significant digit is in the hundredths place; anything after that will be discarded).*

  $4 + 0.1232 = 4.1232$ *is rounded to* 4 *(The least accurate value is* 4*. This is significant to the ones place so the answer cannot have any significant digits past the ones place).*

Sequential calculations and intermediate results
You will often use the result of one calculation as one of the values for another calculation. And it is possible that that answer is used again. If you round the outcomes in-between, all the steps you take after that can introduce additional errors. A solution to this problem is to combine all the steps algebraically and compute the final result in one step. When the intermediate answers are of interest to your readers you should present these answers with the appropriate number of significant digits, but keep a copy of the calculated answer to use in further calculations.

Accuracy vs. Precision, and Errors

Accuracy
A digital multimeter will give you the result of your measurement in several digits on a display. This does not mean that the value you read from the display is the exact value you are measuring. All multimeters have a limited accuracy which you can find in the manual of your multimeter. For example when you are measuring the resistance of a resistor with a multimeter that gives its results in 4 digits, a result you might get is 5000Ω. This number on itself does not say anything. When your multimeter is inaccurate for 5%, the real value of the resistance can be anything between 4750Ω and 5250Ω. Always keep in mind that the number of digits your multimeter tells nothing about its accuracy!

Precision
Multimeters measure with a limited precision. For example when you are measuring a resistor with a resistance of 1467Ω with a multimeter that reads its results in 2 digits, you will measure $1.5k\Omega$ as a result. The real value of the resistor can be anything between $1.45k\Omega$ and $1.55k\Omega$. But this applies only when the accuracy of the multimeter is high enough. Also note that there is a difference between $1.5k\Omega$ and 1500Ω. The difference is the number of significant figures that is used. $1.5k\Omega$ uses 2 significant figures which means, assuming that the accuracy of the measurement was high enough, the real value is somewhere between $1.45k\Omega$ and $1.55k\Omega$. 1500Ω uses 4 significant figures which means, assuming that the accuracy of the measurement was high enough, the real value is somewhere between 1499.5Ω and 1500.5Ω. When working with values, make sure that you interpret them right and you are using the right number of significant figures to describe them. 1.5k means something else than 1500.

Multimeters show the RMS value when measuring AC voltages or currents. DC meters also show the RMS value when connected to varying DC providing the DC is varying quickly, if the frequency is less than about 10Hz you will see the meter reading fluctuating instead. Be aware that a lot of DMM's become unreliable when measuring RMS values of high frequency signals: always check the specifications.

What does '6.05V AC' really mean, is it the RMS or peak voltage? If the peak value is meant it should be clearly stated, otherwise assume it is the RMS value. In everyday use AC voltages (and currents) are always given as RMS values because this allows a sensible comparison to be made with steady DC voltages (and currents), such as from a battery. For example a '6V AC supply' means 6V RMS, the peak voltage is 8.6V. Our mains supply is 230V AC, this means 230V RMS so the peak voltage of the mains is about 320V!

So what does root mean square (RMS) really mean? First square all the values, then find the average (mean) of these square values over a complete cycle, and find the square root of this average. That is the RMS value.

Errors
While measuring you can make several errors that influence your results. When you are measuring a very small resistance you should take into account that the wires you use and any other contacts between the element you measure and the multimeter have a resistance as well. This resistance can be of significant influence if you are measuring a resistance of only a few Ohms. When you are measuring very high resistances you should take into account that objects you consider to be insulators are in fact resistors with very high, but not infinite, resistance. For example when you are measuring a resistor of a few $M\Omega$ and you touch it with your hand, you create a parallel connection between yourself and the resistor which can influence the value you read. There are many more external factors that can cause errors while measuring, so always keep in mind that the value you

read is not always correct.

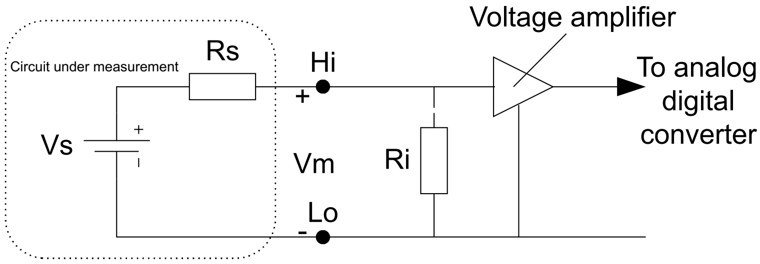


Figure 20.7: *While measuring voltage your result may be influenced by the input resistance of your multimeter. Here R_i can be very large ($10M\Omega - 10G\Omega$). The measured voltage V_m will be $V_s \cdot \frac{R_i}{R_i+R_s}$. The error will be (in percents) $\frac{R_s}{R_i+R_s} \cdot 100$.*

Errors caused by non ideal properties of your equipment
When you are measuring a voltage you connect your multimeter parallel to the component you are measuring. The multimeter will influence the actual voltage unless it is ideal and has an input resistance that is infinite, so no current will flow through it. For a real multimeter this input resistance is not infinite and the multimeter will act as a parallel connected resistor that influences your results. See figure 20.7. The input resistance of your multimeter is given in its manual. Note that the input resistance can change when you change the range you are measuring with.

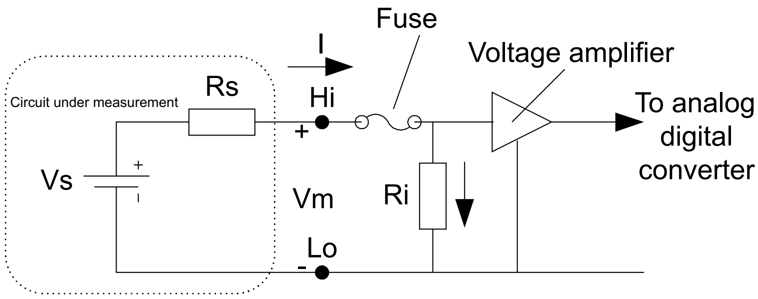


Figure 20.8: *While measuring current your result may be influenced by the input resistance of your multimeter. Now in this case R_i will be very small ($0.1\Omega - 5\Omega$)*

A current meter is taken in series with the current you want to measure. An ideal current meter should have a resistance of 0 if it is not to influence your results. In real life a current meter will have a very small but non 0 input resistance, putting a small resistance in series, so it will influence your results a bit. See figure 20.8. The value of this input resistance is given in the manual. This value may change when the range you are measuring with is changed. Notice that measuring current in fact is measuring voltage across a resistor with a known (low) R-value.

Precision of computed results

When you are calculating with uncertain numbers, the accuracy is changed after the calculations. This can be illustrated with a simple example:

Example 20.1

Suppose we have two resistors, one of $470k\Omega \pm 1\%$ and one of $180k\Omega \pm 2\%$, that are connected in series. To find the total resistance value, we have to add those two values. But how accurate is your answer then?
To find out your uncertainty you can use the following simplified rules. They will yield a slightly larger uncertainty than the probable uncertainty, but are sufficient for now.

- If two quantities are *added or subtracted*, the individual absolute uncertainty is added in the result.

$$470k\Omega \pm 1\% = 470k\Omega \pm 4.7k\Omega \; and \; 180k\Omega \pm 2\% = 180k\Omega \pm 3.6k\Omega$$
$$\text{The sum then becomes:}$$
$$(470k\Omega \pm 4.7k\Omega) + (180k\Omega \pm 3.6k\Omega) = 650k\Omega \pm 8.3k\Omega = 650k\Omega \pm 1.28\%$$

- If two quantities are *multiplied or divided*, the percentages of uncertainty are added to get the percentage of uncertainty in the result.

$$\text{The product then becomes:}$$
$$(470k\Omega \pm 1\%) \cdot (180k\Omega \pm 2\%) = 84.6M\Omega \pm 3\% = 84.6M\Omega \pm 2.5M\Omega$$
$$\text{And the quotient becomes:}$$
$$(470k\Omega \pm 1\%)/(180k\Omega \pm 2\%) = 2.6k\Omega \pm 3\% = 2.6k\Omega \pm 78.3\Omega$$

- When finding the square root of a quantity, we divide the percentage of uncertainty by two. For squaring, the percentage uncertainty is multiplied by two.

$$\sqrt{45.0 \pm 0,1} = \sqrt{45.0 \pm 0.2\%} = 6.708 \pm 0.1\% = 6.708 \pm 0.007$$
$$(45.0 \pm 0,1)^2 = (45.0 \pm 0.2\%)^2 = 2025 \pm 0.4\% = 2025 \pm 8$$

21

Appendix F - Drawing circuits

When designing a circuit, it is a must to draw a clear scheme. A good scheme clarifies how a circuit works and is a great help for locating and solving errors, a bad scheme only causes confusion. With a few rules and tips it becomes easy to draw a good scheme. In fact, it will take you the same amount of time to draw a good scheme as to draw a bad one. Therefore you should always be accurate and make a good scheme, this will save you time in the future.

General Principles

1. A scheme should be clear. Pin numbers, values of parts, voltages, etc. should be identified clearly.
2. A good scheme gives a clear insight in all the functions of the network. Therefore you should make a distinction between the different parts with different functions. Do not be afraid to use too much paper for your scheme.

Rules

1. A connection between wires is indicated with a thick black dot. When two wires cross no dot is drawn. Sometimes an arc to indicate crossing wires is used in handwritten diagrams.
2. Always use the same symbol for the same type of part.

Tips

1. Place the identification of the symbol always directly next to the symbol. For each type of symbol you should always give the same information like name, type or value in the same way.
2. Use logic names for your parts. Try to give all the parts logic names, like R1, R2, ., for resistors and C1, C2,.., for capacitors.
3. Do not put all information next to each other. You put the value of a resistor one line below its name.

4. Commonly signals travel from left to right in your scheme. Sometimes this can differ to for reasons of clarity.
5. Put the positive source voltages at the top of the page and the negative at the bottom. NPN transistors are drawn with the emitter at the bottom side, while PNP transistors are drawn with the emitter at the top side.
6. Do not try to connect all the wires to a source line or a common source node. In stead it often is better to use multiple ground symbols or multiple symbols for the source voltage.
7. Leave sufficient space around the components in your scheme.
8. It should be clear to follow the path of a signal. It is wrong if the reader should follow wires over the complete page to find out how a signal is connected.
9. The source connections for opamps and logic gates should not be drawn. Possible loose connections and not used inputs should be drawn.
10. It is convenient to include a small table with all ic's, their types and their connections with the source voltage.
11. Leave a blank space at the bottom of the page for general information and comment about the circuit.

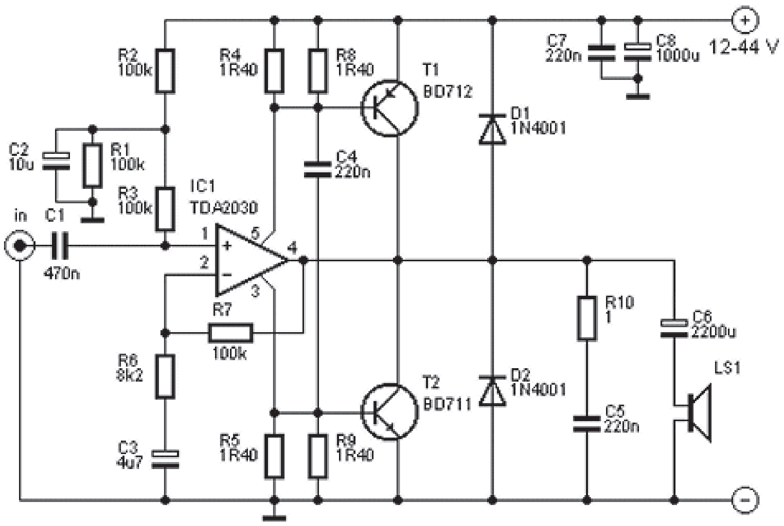


Figure 21.1: *An example of a well drawn circuit.*

22

Appendix G - Building Circuits

When you have designed a circuit and calculated all the values for the components you need, it is time to actually build the circuit. This appendix describes how you can realize your design into an actual circuit.

Step 1: The design

Before you start to realize any of your ideas, you should make sure that your design is finished. You should have decided which components you are going to need and you should have calculated the values for them and made sure that the currents and voltages can not get too high anywhere in your circuit. The next step is to draw a proper scheme of your circuit.
Appendix F describes how you should properly draw a circuit. You should always draw a scheme before you start realizing your circuit. This to prevent any confusion later on.

NOTE: Nowadays a wealth of CAD (Computer Aided Design) tools are available for the electronics and system designers. One of them, the so-called circuit simulator computer programs allow for drawing and in-depth testing and analyzing of a circuit to be designed. At a very early stage in the 'Circuits' course you will learn to apply these tools. The simulations you can perform will tell a lot about the behaviour of the circuit to be constructed and so help you performing the right analysis.

Step 2: Realization

The second step is to realize your circuit. You can either build your circuit on a breadboard (or socket board), or solder it on a circuit board.

Breadboard

Often prototyping your circuit means: plug your components into a breadboard. Realizing a circuit on a breadboard is easy and the circuit can quickly be modified which is ideal for testing. A disadvantage of a breadboard is that connections can go loose easily. Therefore a circuit on a breadboard is good for testing in a fixed position, but not ideal as a circuit that is meant to last or is to be transported a lot.

A breadboard has many small holes (sockets) in it in which you can attach wires or the legs of components by simply pushing them into the sockets.

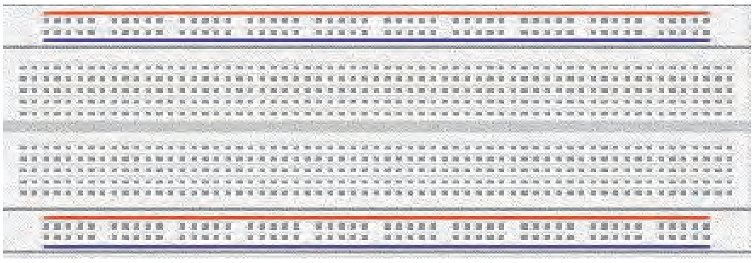


Figure 22.1: *A breadboard.*

The breadboard has many strips of metal (copper usually) which run underneath the board. The metal strips are laid out as shown in figure 22.2.

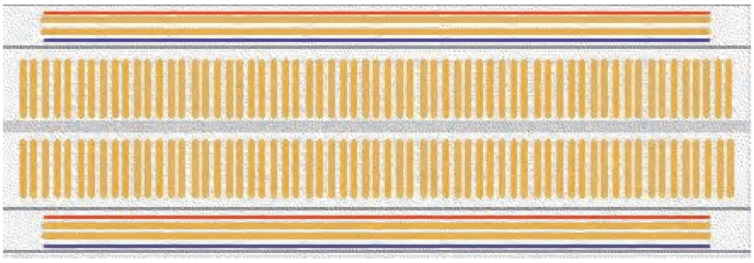


Figure 22.2: *The layout of the wires underneath the sockets in a breadboard.*

These strips connect the holes on the top of the board. This makes it easy to connect components together to build circuits. To use the breadboard, the legs of components are placed in the holes (the sockets). The holes are made so that they will hold the component in place. Each hole is connected to one of the metal strips running underneath the board.

Each wire forms a node. A node is a point in a circuit where two components are connected. Putting their legs in a common node forms connections between different components. On the breadboard, a node is the row of holes that are connected by the strip of metal underneath.

The long top and bottom row of holes are usually used for power supply connections. Be careful: in some models these rows are disconnected halfway. You'll have to put some jumper wires to connect them.

The circuit is built by placing components in the sockets and connecting them together with jumper wires. When a path is formed by wires and components from the positive supply node to the negative supply node and your circuit is finished, we can turn on the power and current will flow through the path and the circuit comes alive.

For chips with many legs (ICs), place them in the middle of the board so that half of the legs are on one side of the middle line and half are on the other side.

A completed circuit might look like figure 22.3. This circuit uses two small breadboards.

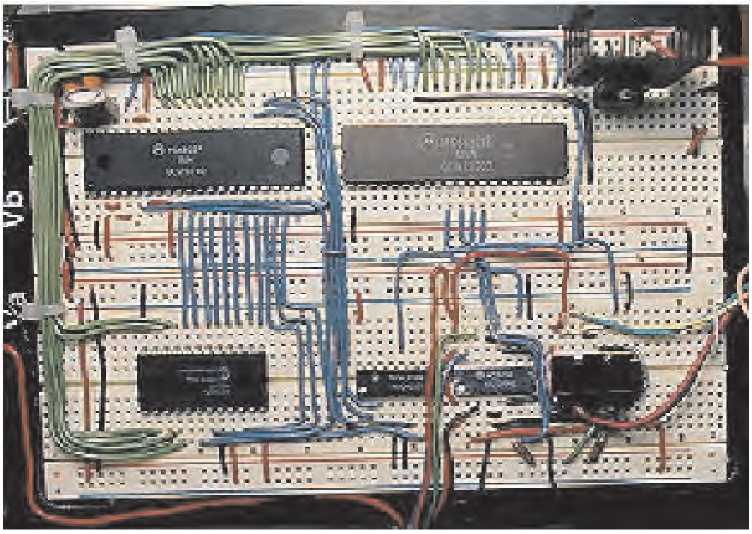


Figure 22.3: *A completed circuit on 2 breadboards.*

Soldering

When you have completed your design and finished testing it, you might want to construct your circuit in a more solid way. The disadvantage of a breadboard is that connections could go loose. The solution to this is to solder your circuit on a circuit board. By soldering your circuit all the components and wires are in a fixed position and will stay firmly attached. Therefore do not solder your circuit until you have completed testing it on a breadboard for making adjustments in a soldered circuit is trickier than when your circuit is put on a breadboard.

A circuit board is a non conducting plate with many tiny holes in it. On the back side of the circuit board there are small copper rings around all the holes.

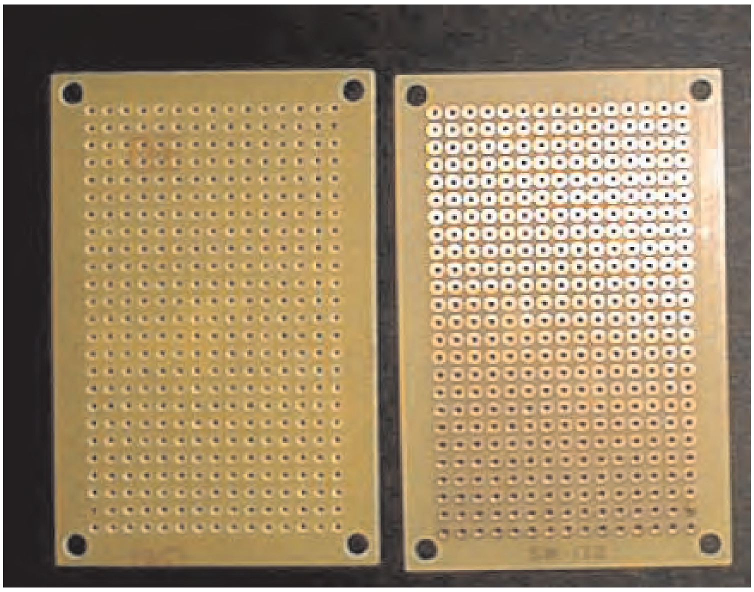


Figure 22.4: *Left the front side of a circuit board, right the back side.*

The components are put on the front side of the circuit board and their legs are put through the holes. On the back side the connections can be made by soldering wires to the legs of components.

When soldering, make sure that you do not heat components too long. It should only take 2 or 3 seconds to make a connection. When you heat them too long some components can get damaged. Furthermore you should make sure that you do not move the parts you are soldering while the tin is fluid or else you might get a bad connection. Also make sure that you do not connect parts together that you do not want to connect. Therefore you should use only a little tin.

Testing

When your circuit is completed you should always test your circuit. Connect the power source and use a multi meter or a scope to see if your circuit works as expected. Measure if the voltages on important nodes are as you expect them to be. More often than not, the circuit will not behave in the expected way for the first time.

If your circuit does not work as expected you start looking for the mistake. You can follow these steps.

1. First check the source voltage with a voltmeter.

2. Make sure that all your connections are made correctly. When using a breadboard, connections can go loose easily. Also make sure that no metal parts touch each other that are not supposed to be connected. Work neatly!

3. Check if your circuit matches your design. You could have put a wire in the wrong socket.

If you still can't find the error you should divide your circuit in several small sub circuits and look closer at them one by one to find the error. You should work systematically, for example from left to right if you are following your design, or from output to input or reverse. Look first at those sub circuits of which you have doubts if they are correct.

1. Measure if the input and output values of all your sub circuits are as expected and make your way through the complete circuit. In this way you can find in which sub circuit the error is located.

2. Test if the sub circuits work when they are separated from the complete circuit. With a source put the desired input on the sub circuit and test it separately to see if it has the desired behavior.

If you still can't find the mistake, ask yourself whether you made a mistake during the design stage. You could have made a miscalculation or the circuit might be behaving differently in real life. After all, a design is only a model of the circuit on paper. Use the results of your testing to modify your design until you have a circuit with the desired behaviour.

Tips on building circuits

Finally some tips on building circuits:

- Build your circuit as orderly as possible: What is left on your scheme is left on your bread board.
- Always make a scheme first, never start building without one or you surely will miss something.
- Keep the connections as short as possible.
- Do not put to thick wires (diameter of max. 0,8 mm) in the breadboard or else the contacts in the sockets will open in time, which leads to bad connections.
- Do not use long un-insolated wires or long legs of components. This increases the chance of undesired connections.
- If possible use (matching) plug-in sockets for i.c's.

23

Appendix H - Cables

Make it a good habit to build your networks systematically. Current loops are to be created with wires that are thick enough to transport the current without heating up. Below you will find the most important cables with their connectors.

Connection cables with banana plug
Mostly wires with two banana plugs are used to connect machines and components with each other. Choose the color and length of the wire in a consistent way to avoid a complete mess. For example:
- Red → (+)
- Black → ground
- White → (-)

Use the same colors for wires with the same voltages.

BNC-connectors
Coax-cables with BNC-connectors (Bayonet Navy Connector) are often used to avoid interference of anything outside the circuit. A coax-cable consists of two conductors: one central connector with coaxial around it a second conductor. These are isolated from each other. These cables are provided with a male BNC-connector (see figure 19.2 on the left). Each device that can measure or generate high frequencies (several kHz) is provided with a female BNC connector.

Oscilloscope probes
Because of the high frequencies often involved, oscilloscopes do not normally use simple wires to connect to the DUT (Device Under Test). Instead, a specific scope probe is used (see figure 19.3). Scope probes use a coaxial cable to transmit the signal from the tip of the probe to the oscilloscope, preserving high frequencies for more accurate oscilloscope operation.

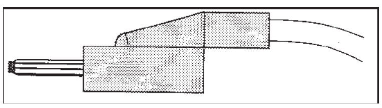


Figure 23.1: *A banana plug.*

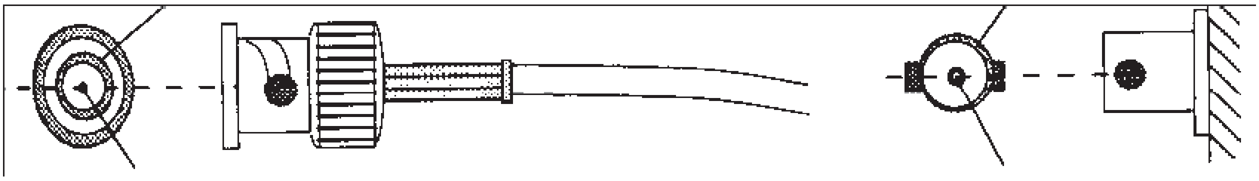


Figure 23.2: *BNC Connector.*

Figure 23.3: *Oscilloscope probe set.*

24

Appendix I - Safety and protection

The effect of getting electrocuted depends largely on:
- The value of the current through your body.
- The time that the current flows through your body.
- The path that the current follows trough your body.
- The frequency of the current.

Notice that the value of the voltage is not important! That is why you have no problem with static electricity (no current) of several kV.

Figure 24.1: *Ouch......*

Value of the current
Tests showed that:
1mA: perception limit, a current you can barely feel, although it can be lethal when injected directly into the heart.
10mA: breaking current limit, the current that causes to contract the mussels enough to make it impossible to break contact with the current.
An alternating current above 25mA can already cause heart failure (still reversible) and a current of 60mA can cause heart fibrillations. With currents of 3A and above the internal burns play a

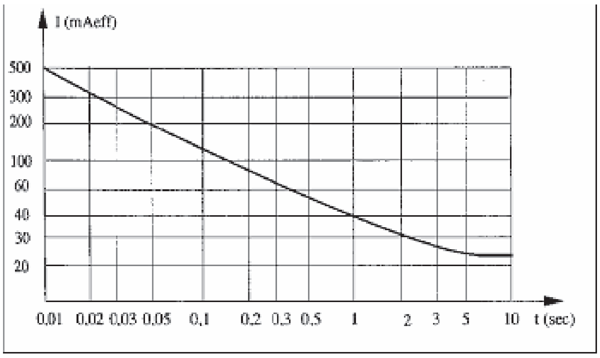


Figure 24.2: *Current versus time.*

major role.
The intensity of the current depends of the resistance of the human body, which is lower with tenser grips and in humid climates.

Duration of the current flow
Figure 24.2 shows a safety curve that shows how long the body can cope with a certain current in normal conditions.

Because in real life you will mostly encounter voltage sources, safety guides mostly work with time-voltage curves. To be safe in all possible conditions the maximum voltage is 25Vac for wet skin and 50Vac for dry skin.

Path of the current
The most dangerous paths are from hand to hand and from hand to the opposite foot. This is logic because on this path the current crosses heart and lungs.

The frequency of the current
The most dangerous frequencies are in the area between 10Hz and 200Hz. Above 5 kHz the current does not infiltrate the body but largely flows past the skin and is therefore less dangerous for the heart. However the chance on burns increases. With DC the perception limit is about 3 times higher than with 50Hz. However the risk of electrolytic decomposition of the blood is for low currents much higher with DC.

25

Appendix J - Internet websites

In this appendix you'll find a list with web sites on the internet, which contain more information about specific topics:

- **Hyper Physics**
  On this site you can find all kind of information about various physical subjects. Electrical parts are also discussed here.
  http://hyperphysics.phy-astr.gsu.edu/hbase/hframe.html

- **Electronics Tutorial**
  A nice collection of well written tutorials.
  http://www.electronics-tutorials.ws/

- **Circuits Online**
  Find many circuits for different applications on this site (in Dutch)!
  http://www.circuitsonline.net/

- **How Stuff Works**
  A site about some nice electronic applications.
  www.howstuffworks.com

- **Java Applets**
  A site with nice java applets for physical problems and laws (like Ohm's law)
  http://www.walter-fendt.de/ph14e/ohmslaw.htm

- **Electronics club**
  A site with some nice tips about soldering and circuits.
  http://www.kpsec.freeuk.com/index.htm

- **Elektuur**
  A nice online magazine about electronics.
  http://www.elektor.com/

- **Hobby Electronics**
  Information about electronic as a hobby.
  http://www.Hobby-Electronics.info/

- **Conrad**
  This is a seller of electrical components in the Netherlands.
  http://www.conrad.com/

- **Virtual oscilloscope**
  A site, which contains a very nice demo about the scopes we use at the TU/e
  http://www.virtual-oscilloscope.com/