ENGR 260: CIRCUITS AND DEVICES



Ramki Kalyanaraman
Cañada College



ENGR 260: Circuits and Devices

This text is disseminated via the Open Education Resource (OER) LibreTexts Project (https://LibreTexts.org) and like the thousands of other texts available within this powerful platform, it is freely available for reading, printing, and "consuming."

The LibreTexts mission is to bring together students, faculty, and scholars in a collaborative effort to provide an accessible, and comprehensive platform that empowers our community to develop, curate, adapt, and adopt openly licensed resources and technologies; through these efforts we can reduce the financial burden born from traditional educational resource costs, ensuring education is more accessible for students and communities worldwide.

Most, but not all, pages in the library have licenses that may allow individuals to make changes, save, and print this book. Carefully consult the applicable license(s) before pursuing such effects. Instructors can adopt existing LibreTexts texts or Remix them to quickly build course-specific resources to meet the needs of their students. Unlike traditional textbooks, LibreTexts' web based origins allow powerful integration of advanced features and new technologies to support learning.



LibreTexts is the adaptable, user-friendly non-profit open education resource platform that educators trust for creating, customizing, and sharing accessible, interactive textbooks, adaptive homework, and ancillary materials. We collaborate with individuals and organizations to champion open education initiatives, support institutional publishing programs, drive curriculum development projects, and more.

The LibreTexts libraries are Powered by NICE CXone Expert and was supported by the Department of Education Open Textbook Pilot Project, the California Education Learning Lab, the UC Davis Office of the Provost, the UC Davis Library, the California State University Affordable Learning Solutions Program, and Merlot. This material is based upon work supported by the National Science Foundation under Grant No. 1246120, 1525057, and 1413739.

Any opinions, findings, and conclusions or recommendations expressed in this material are those of the author(s) and do not necessarily reflect the views of the National Science Foundation nor the US Department of Education.

Have questions or comments? For information about adoptions or adaptions contact info@LibreTexts.org or visit our main website at https://LibreTexts.org.

This text was compiled on 05/15/2026

TABLE OF CONTENTS

Licensing

1: Basic Concepts and Quantitites

- 1.1: Introduction
- 1.2: An Atomic Model
- 1.3: Charge and Current
- 1.4: Energy and Voltage
- 1.5: Power and Efficiency
- 1.6: Energy Cost and Battery Life
- 1.7: Resistance and Conductance
- 1.8: Instrumentation and Laboratory
- 1.9: Summary
- 1.10:Exercises

2: Fundamental Laws

- 2.1: Ohms Law and Kirchhof's Voltage Law
  - 2.1.1: Introduction
  - 2.1.2: Conventional Current Flow and Electron Flow
  - 2.1.3: The Series Connection
  - 2.1.4: Combining Series Components
  - 2.1.5: Ohm's Law
  - 2.1.6: Kirchhoff's Voltage Law
  - 2.1.7: Series Analysis
  - 2.1.8: Potentiometers and Rheostats
  - 2.1.9: Summary
  - 2.1.10: Exercises
- 2.2: Network Topology- Nodes, Branches and Loops
- 2.3: Kirchhof's Current Law
  - 2.3.1: Introduction
  - 2.3.2: The Parallel Connection
  - 2.3.3: Combining Parallel Components
  - 2.3.4: Kirchhoff's Current Law
  - 2.3.5: Parallel Analysis
  - 2.3.6: Current Limiting - Fuses and Circuit Breakers
  - 2.3.7: Summary
  - 2.3.8: Exercises
- 2.4: Series-Parallel Resistive Circuits
  - 2.4.1: Introduction
  - 2.4.2: Series-Parallel Connections
  - 2.4.3: Simplifying Series-Parallel Components
  - 2.4.4: Series-Parallel Analysis
  - 2.4.5: Summary
  - 2.4.6: Exercises

3: Circuit Simulations Using Computers

4: Analysis Theorems and Techniques

- 4.1: Introduction
- 4.2: Source Conversions
- 4.3: Superposition Theorem
- 4.4: Thévenin's Theorem
- 4.5: Norton's Theorem
- 4.6: Maximum Power Transfer Theorem
- 4.7: Advanced Topic- Delta-Y Conversions
- 4.8: Summary
- 4.9: Exercises

5: Nodal and Mesh Analysis, Dependent Sources

- 5.1: Introduction
- 5.2: Nodal Analysis
- 5.3: Mesh Analysis
- 5.4: Dependent Sources
- 5.5: Summary
- 5.6: Exercises

6: Capacitors and Inductors

- 6.1: Capacitors
  - 6.1.1: Introduction
  - 6.1.2: Capacitance and Capacitors
  - 6.1.3: Summary
  - 6.1.4: Exercises
- 6.2: Inductors
  - 6.2.1: Introduction
  - 6.2.2: Inductance and Inductors
  - 6.2.3: Summary
  - 6.2.4: Exercises

7: Advanced Topic- Operational Amplifiers

- 7.1: Introduction
- 7.2: The Decibel
- 7.3: Bode Plots
- 7.4: Combining the Elements - Multi-Stage Effects
- 7.5: The Differential Amplifier
- 7.6: Summary
- 7.7: Problems
- 7.8: Operational Amplifier Internals
  - 7.8.1: Introduction to Operational Amplifiers
  - 7.8.2: What Is An Op Amp?
  - 7.8.3: Simple Op Amp Comparator
  - 7.8.4: Op Amp Manufacture
  - 7.8.5: Summary

- 7.8.6: Problems
- 7.9: Negative Feedback
  - 7.9.1: Introduction to Negative Feedback
  - 7.9.2: What Negative Feedback Is and Why We Use It
  - 7.9.3: Basic Concepts
  - 7.9.4: The Four Variants of Negative Feedback
  - 7.9.5: Limitations On The Use of Negative Feedback
  - 7.9.6: Summary
  - 7.9.7: Problems
- 7.10: Basic Op Amp Circuits
  - 7.10.1: Introduction
  - 7.10.2: Inverting and Noninverting Amplifiers
  - 7.10.3: Single Supply Biasing
  - 7.10.4: Current Boosting
  - 7.10.5: Summary
  - 7.10.6: Problems
- 7.11: Integrators and Differentiators
  - 7.11.1: Introduction
  - 7.11.2: Integrators
  - 7.11.3: Differentiators
  - 7.11.4: Analog Computer
  - 7.11.5: Alternatives to Integrators and Differentiators
  - 7.11.6: Extended Topic- Other Integrator and Differentiator Circuits
  - 7.11.7: Summary
  - 7.11.8: Problems

8: Transient Behavior- First and Second Order Circuits

- 8.1: Introduction
- 8.2: Initial and Steady-State Analysis of RC Circuits
- 8.3: Transient Response of RC Circuits
- 8.4: Initial and Steady-State Analysis of RL Circuits
- 8.5: Transient Response of RL Circuits
- 8.6: Initial and Steady-State Analysis of RLC Circuits
- 8.7: Summary
- 8.8: Exercises

9: AC Signal Fundamentals

- 9.1: Introduction
- 9.2: Sinusoidal Waveforms
- 9.3: Basic Fourier Analysis
- 9.4: Complex Numbers
- 9.5: Reactance and Impedance
- 9.6: Phasor Diagrams
- 9.7: Summary
- 9.8: Exercises

10: Series RLC Circuits

- 10.1: Introduction
- 10.2: The Series Connection
- 10.3: Series Impedance
- 10.4: Series Circuit Analysis
- 10.5: Concerning Practical Inductors
- 10.6: Summary
- 10.7: Exercises

11: Parallel RLC Circuits

- 11.1: Introduction
- 11.2: The Parallel Connection
- 11.3: Parallel Impedance
- 11.4: Parallel Circuit Analysis
- 11.5: Summary
- 11.6: Exercises

12: AC Circuit Analysis Theorems and Techniques

- 12.1: Introduction
- 12.2: Source Conversions
- 12.3: Superposition Theorem
- 12.4: Thévenin's and Norton's Theorems
- 12.5: Maximum Power Transfer Theorem
- 12.6: Advanced Topic- Delta-Y (Pi-T) Conversions
- 12.7: Summary
- 12.8: Exercises

13: AC Power

- 13.1: Introduction
- 13.2: Power Waveforms
- 13.3: Power Triangle
- 13.4: Power Systems
- 13.5: Summary
- 13.6: Exercises

14: Advanced Topic- Magnetic Circuits and Transformers

- 14.1: Introduction
- 14.2: Electromagnetic Induction
- 14.3: Magnetic Circuits
- 14.4: Transformers
- 14.5: Summary
- 14.6: Exercises

Index

Glossary

Detailed Licensing

Licensing

*A detailed breakdown of this resource's licensing can be found in **Back Matter/Detailed Licensing**.*

CHAPTER OVERVIEW

1: Basic Concepts and Quantitites

Learning Objectives

After completing this chapter, you should be able to:

- Describe a basic, functional atomic model.
- Describe fundamental quantities and relations including charge, current, energy, voltage, power, resistance and conductance; and perform basic computations using them.
- Compute efficiency, energy cost, battery life and DMM accuracy.
- Utilize the resistor color code.

1.1: Introduction
1.2: An Atomic Model
1.3: Charge and Current
1.4: Energy and Voltage
1.5: Power and Efficiency
1.6: Energy Cost and Battery Life
1.7: Resistance and Conductance
1.8: Instrumentation and Laboratory
1.9: Summary
1.10:Exercises

This page titled 1: Basic Concepts and Quantitites is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

1.1: Introduction

In this chapter we shall examine the major quantities of interest in electrical systems and describe their relationships. These include voltage, current, power, energy and resistance. We begin with basic definitions and proceed to example computations. We shall go beyond the theoretical and investigate practical aspects as found in electrical laboratories. This includes measurement devices and power sources, as well as various resistive devices. We shall examine batteries from a functional perspective, and perform computations regarding efficiency and cost of energy. We will also introduce some of the device symbols used to create electrical circuit schematics. This material presents the foundation upon which we will build circuit analysis techniques in subsequent chapters. Treat it accordingly.

This page titled 1.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.2: An Atomic Model

In order to understand electrical circuit behavior we must first define what it is that we are chasing. Ultimately, electrical circuits are all about the movement of electric charge. What do we mean by charge? To answer that question, we need to come up with a usable model of an atom, in other words, a functional description of its internal structure. To be sure, atoms are far smaller than most people imagine. There are far more atoms in a gallon of water than there are gallons of water in all of the oceans, lakes and rivers of planet Earth combined. It would be pointless to ask what an atom might “look like” because its components are all smaller than the shortest wavelengths of light that humans can see. That would be a little like asking a human what a dog whistle sounds like, when in fact it's at a pitch that's higher than humans can hear. Ultimately though, all we really need is a model to explain its observed behavior.

Perhaps the most prolific model in the popular imagination is the planetary model shown in Figure 2.2.1 , so named because it is reminiscent of simple models of our solar system. In this model, the core, or nucleus, is drawn at the center and contains positively charged protons and non-charged neutrons. Revolving around this core are negatively charged electrons, each following a nice, regular, planar path much like a planet around the sun. Unfortunately, this model is not accurate to say the least, although its popularity can sometimes lead to wildly erroneous and darkly humorous conclusions. In spite of its inaccuracy, the components (nucleus, protons, neutrons and electrons) are perfectly valid. Before we come up with a more accurate and useful model, let's take a closer look at these items.

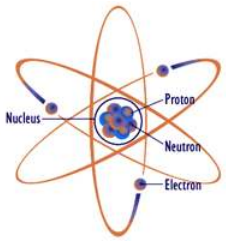


Figure 2.2.1 : Planetary atomic model: Popular but sadly incorrect. Image source (modified)

Unlike people, all particles of a type are indistinguishable: every proton is identical to every other proton and every electron is identical to every other electron. The number of protons determine what the element is. The simplest element, hydrogen, has a single proton in its nucleus. In contrast, helium has two protons in its nucleus and copper has 29. The nucleus may also contain a number of neutrons as well (not necessarily equal to the number of protons). In any stable atom, the number of electrons will equal the number of protons. If they aren't the same, the atom is called an ion (technically, a cation has fewer electrons than protons and an anion is its opposite). With an external energy source, it is possible to separate an electron from an atom. This leaves us with an electron and an atom that now has a net positive charge. It is this ability to separate these particles, and thus separating charge, that gives rise to our concept of electricity including such items as current and voltage, as we shall see.

Most of the mass of any given atom is from the protons and neutrons. Protons and neutrons have similar masses, about 1.67E−24 grams each. The mass of an electron is roughly 2000 times smaller. The radius of a proton is approximately 0.87E−15 meters and the mean distance to the nearest electron is about 5.3E−11 meters. Therefore an electron is about 60,000 times farther away from the proton than the size of said proton. To put this into perspective, that's roughly the same as the ratio between a golf ball and a sphere with a radius of 3/4ths of a mile or 1200 meters. This would be the case for a hydrogen atom as it consists of a single proton and electron. The magnitude of this ratio is not much different for other substances, including things like diamond and quartz that are very hard and solid. If you think about that for a moment, you realize that the idea of “solidity” is in some ways an illusion because the vast majority of what we call “something” is really just empty space (like a golf ball rattling around inside a sphere that's a mile and a half wide). In reality, the feeling of solidity when you grab something with your hand is just the result of the interaction of atomic forces between your hand and the object. Indeed, if we could somehow get rid of that “empty space”, we could fit a mountain inside of a car (although it would probably require a severely upgraded suspension).

For our immediate work, perhaps the most important issue is that of charge, an essential part of the theory of electromagnetism. It was mentioned that protons are positively charged and electrons are negatively charged. What then is charge? Charge is not an obvious physical attribute like someone's height or eye color. It's closer to a behavioral characteristic. Saying a particle is charged is

like saying someone has an attractive personality. The magnitude of the charge on an electron is the same as the magnitude of the charge on a proton. The only difference is the polarity: protons are positively charged while electrons are negatively charged. An important thing to remember is that particles with the same polarity repel each other while opposites attract[1].

Charge is measured in coulombs, named after Charles-Augustin de Coulomb a French physicist of the eighteenth century. The charge on a single electron is tiny, a mere 1.602E−19 coulombs. Alternately, it takes 6.242E18 electrons to create a charge of one coulomb. More on this in a moment.

One of the major issues with the planetary model is the idea that electrons whirl around the nucleus in stable, planet-like orbits. That's simply not true. First, the electron inhabits a region of 3D space and does not simply move through a plane. Second, due to the Heisenberg Uncertainty Principle, we can't precisely plot the position and trajectory of a given electron. The best we can do is make a plot of where the electron is likely to be. This is called a probability contour. To understand this concept, imagine that you could record the position of a specific electron relative to the nucleus at a specific time. A moment later you record its new position, a moment after that you record the next position, and so on for thousands of measurements. If you attempted to plot them all, you would wind up with a cloud of dots around the nucleus. This cloud is referred to as an orbital. You wouldn't know how the electron got from one position to the next but you would get a general idea of where it was likely to be. Ultimately, an orbital looks nothing like a planetary orbit.

A good analogy to an electron probability contour is the mapping of the flight of migratory birds. For example, a certain group of Canada geese might migrate from their summer range in the far north of Canada to their summer abode in the southern USA via s section of the Atlantic Flyway, perhaps through central New York State, as shown in purple, in Figure 2.2.2 . The flyway tells you where these birds are likely to be found while migrating. If they are taking the route through central New York, at some point it is likely that any particular goose in that flock will be found near the cities of Utica, Syracuse and so forth. It is much less likely that any particular goose from that flock will be found in Ohio or New Hampshire. Texas, of course, is right out. The flyway gives a good bit of data regarding where these birds are likely to be but in no way can it predict with any accuracy precisely where any individual bird will be on a specific date, nor precisely where it will end its migration.

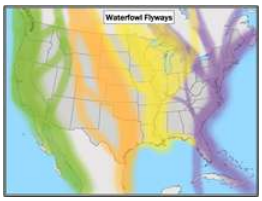


Figure 2.2.2 : Electron behavior versus migratory birds.

Returning from our avian diversion, we note that there are several potential orbitals. Due to quantum physics, only certain orbitals are allowed. The permissible electron energy levels are first grouped into shells, then subshells and finally orbitals. It is important to remember that orbitals indicate the electron energy level. That is, a higher orbital implies a higher energy level. Further, orbitals fill in first from lowest energy level to highest energy level.

Shells are denoted by their principal quantum number, n; 1, 2, 3, etc. The higher the number, the more subshells it can contain. Subshells are designated by letters, the first four being s, p, d, and f. Shell 1 contains only subshell s while shell 2 contains subshell types s and p. Shell 3 contains subshell types s, p and d, and so on. These subshells may also have variations within them. There is one variation on s, three variations on p, five variations on d, etc. These variations are the orbitals and each orbital can hold a maximum of two electrons.

Putting this all together, we find that the first shell can contain a maximum of two electrons: two in the single s subshell orbital ($1s$). The second shell can contain a maximum of eight electrons: two in the s subshell ($2s$) plus two in each of the three p subshell orbitals ($2p$). In like manner the third shell can contain a maximum of 18 electrons: two in $3s$, six in $3p$ and two in each of the the five d subshell orbitals ($3d$). You can condense this into a simple formula:

$$\text{Number of electrons per shell } = 2n^2, \text{ where } n \text{ is the shell number}$$

Figure 2.2.3 shows the electron probability contour of the innermost orbital, namely $1s$ (i.e., principle quantum number 1, subshell s). As you can see, it is spherical in shape. The nucleus is located at the center, obscured here. All s orbitals are similarly spherically

shaped although the internals change. $1s$ is the lowest energy orbital.



Figure 2.2.3 : Electron probability contour for innermost orbital, 1s.

Orbitals are not limited to simple spherical shapes. Higher order orbitals can take on a variety of forms. Figure 2.2.4 shows a much more complex electron probability contour. The nucleus is situated in the center of the concentric rings. Obviously, this is nothing like the well-behaved elliptical orbits of planets around the sun (nor the paths of migratory birds, unless they are very confused).

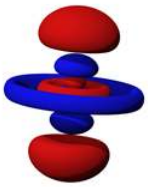


Figure 2.2.4 : A higher order electron probability contour.

As visually interesting as these graphics are, they are cumbersome to work with. Consequently, a more functional graphic is called for. Such a device is the Bohr model, named after Danish physicist Niels Bohr. An example is shown in Figure 2.2.5 .

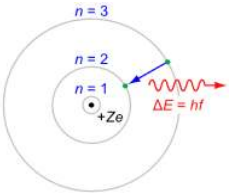


Figure 2.2.5 : Generic Bohr model.

It is important to understand that the Bohr model is an energy description of the atom, not an attempt to mimic its physical appearance or structure. The nucleus is placed at the center. It is surrounded by concentric rings that represent the electron shells. The higher the number, the larger the ring and the greater the energy level. If an electron were to move from a higher level to a lower level, the energy difference is radiated out. This could be in the form of heat or light. This is a point worth remembering. For example, this transition is what makes light emitting diodes (LEDs) function. The inverse is also possible, namely that by absorbing energy, an electron can move into a higher orbital. This is an equally powerful concept.

Using the Bohr model we can create diagrams to represent individual elements. For example, copper has an atomic number of 29 meaning that it has 29 protons and 29 electrons. The electron shell configuration is 2-8-18-1. That is, the first three shells are completely filled and there is a single electron in the fourth shell. The Bohr model for copper would simply show four rings, the first three being filled and with a single electron in the fourth ring. This is shown in Figure 2.2.6 . In this version, the individual electrons are drawn in each shell and the atomic number is indicated at the nucleus. Again, please do not imagine this representing individual electrons orbiting the nucleus in lanes. This is an energy level depiction.

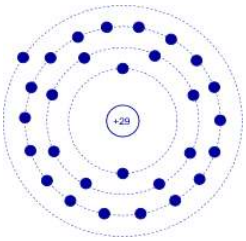


Figure 2.2.6 : Bohr model of copper.

It is worth noting that a lone outer electron is only loosely bound to the nucleus thus making the movement of charge through said material relatively easy, given the application of some external energy source. Consequently, copper is a very good conductor. This is a characteristic shared by common metals: they are good conductors because they only have one or two loosely held electrons in their outer shell. Examples include silver, gold and aluminum (one outer electron for each, all of them being very good conductors); and iron, tin and nickel (two outer electrons for each, and this group being not quite as conductive as those in the first group).

References

[1]The obvious question might be “Why doesn't the nucleus break apart?” The answer is that the sub-atomic strong nuclear force has a greater effect at atomic distances than electromagnetism, and thus binds the protons and neutrons together in the nucleus.

This page titled 1.2: An Atomic Model is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.2: An Atomic Model** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.3: Charge and Current

As already noted, charge is an attractive force. It is denoted by the letter Q and has units of coulombs. Electrons are negatively charged and protons are positively charged. All electrons and protons exhibit the same magnitude of charge, roughly 1.602E−19 coulombs. Thus, one coulomb is equivalent to the charge exhibited by approximately 1/1.602E−19, or 6.242E18 electrons. Further, opposite charges attract while like charges repel, similar to the poles of a magnet.

It is possible to move charge from one point to another. The rate of charge movement over time is called current. It is denoted by the letter I and has units of amps (or amperes)[1]. One amp of current is defined as one coulomb per second.

$$1 \text{ amp } \equiv 1 \text{ coulomb } /1 \text{ second} \tag{1.3.1}$$

That is, one amp can be visualized as approximately 6.242E18 electrons passing through a wire in a period of one second. Consider Figure 2.3.1 .

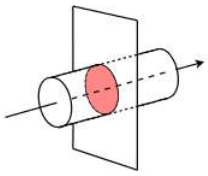


Figure 2.3.1 : Defining current as charge flow through a wire.

Here we have a wire with electrons flowing through it in the direction of the arrow. We cut this wire with an imaginary plane, leaving us with the highlighted disk. Now imagine that we could count the number of electrons passing through this disk over the course of one second. Because we know the charge possessed by one electron, we simply multiply the number of electrons by the charge for each to yield the total charge, and thus we arrive at the current. As a formula:

$$I = Q/t \tag{1.3.2}$$

where

- I is the current in amps,
- Q is the the charge in coulombs,
- t is the time in seconds.

A common analogy for electric current is the flow of water through a pipe or river. Just as we can imagine water flow as “gallons or liters per minute”, we imagine electric current as “coulombs per second”.

Example 2.3.1

In the course of one-half second, a certain battery delivers a charge of three coulombs. Determine the resulting current.

Solution

$$I = \frac{Q}{t}$$

$$I = \frac{3C}{0.5s}$$

$$I = 6A$$

Example 2.3.2

A device delivers a current of 25 mA. Determine the charge transferred in two seconds along with the equivalent total number of electrons.

Solution

$$I = \frac{Q}{t}$$

$$Q = I \times t$$

$Q = 25mA \times 2s$ from Definition 1.3.1, amp-seconds is coulombs

$$Q = 50mC$$

As one coulomb is equivalent to 6.242E18 electrons, just multiply to find the total number of electrons transferred.

$$\text{Total electrons } = Q \times \text{ number of electrons per coulomb}$$

$$\text{Total electrons } = 50mC \times 6.242E18$$

$$\text{Total electrons } = 3.121E17 \text{ electrons}$$

In sum, the larger the charge transferred within a given time period, the greater the current. Modern electrical and electronic systems might deal with currents of under a picoamp or, at the other extreme, thousands of amps. That is an astonishing range. It is roughly equivalent to a single drop of water dripping from a leaky faucet each second compared to 1000 times the flow of water over Niagara Falls.

References

[1] *I* stands for *I*ntensity (of current), and was so named by André-Marie Ampère.

This page titled 1.3: Charge and Current is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.3: Charge and Current** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.4: Energy and Voltage

Energy is defined as the ability to do work. It is denoted by the letter W. The basic unit is the joule although other units are sometimes used (for example, the calorie or the kilowatt-hour, kWh).

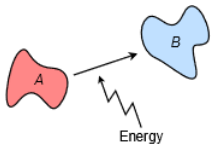


Figure 2.4.1 : Defining voltage as work to move charge.

If we were to move a charge from one point to another (for example, separating an electron from an atom), we would have to expend energy to do so. This is illustrated in Figure 2.4.1 . In this Figure, we would say that B has a higher electric potential than A. In other words, there is a potential difference between B and A. We refer to this change as voltage. It is denoted by the letter V (or sometimes E[1]) and has units of volts, in honor of Alessandro Volta. One volt is defined as one joule per coulomb.

$$1 \text{ volt } \equiv 1 \text{ joule } /1 \text{ coulomb} \tag{1.4.1}$$

As you might guess, the bigger the charge to be moved, the greater the energy required. Expressed as a formula,

$$V = W/Q \tag{1.4.2}$$

Where

V is the voltage in volts,

W is the energy in joules,

Q is the charge in coulombs.

Unlike current, voltage always implies two points for measurement because it involves a difference. Often, one of the points is a common reference, such as earth ground or a circuit ground (i.e., chassis ground). Sometimes people will refer to a point in a circuit as having a certain voltage, as in “point X is 12 volts”. Although common in use, this is somewhat sloppy and not strictly correct. It is important to always remember that this value is relative to some reference point. As a general rule, a voltage will be denoted using the two points as subscripts, for example, V_{AB}, that is, the voltage at point A relative to point B. If only a single subscript is used, as in V_A, then the second, or reference, point is assumed to be the system common or ground. In this case, we're referring to the voltage at point A relative to the system common point. Finally, by definition, $V_{AB} = V_A - V_B$, as they have the same reference.

Example 2.4.1

100 joules are expended to move a 20 coulomb charge from point A to point B. Determine the resulting voltage.

$$V_{BA} = \frac{W}{Q}$$

$$V_{BA} = \frac{100J}{20C}$$

$$V_{BA} = 5V$$

Note that it is possible for a voltage to be negative. This simply means that the potential at the point of interest is less than that of the reference point. In Example 2.4.1 we discovered that point B is five volts above point A. We could just as easily say that point A is five volts below point B, or $V_{AB} = -5$ V. Further, we can state the difference in terms of the individual ground-referenced voltages, or $V_{BA} = V_B - V_A$.

Static Electricity and ESD

While it is obviously true that higher voltages imply higher associated energies (charge being held constant), it is not true that a particularly high voltage is necessarily lethal. This is because a very high voltage can be achieved by moving a small charge with only modest energy input. A good example of this is static electricity, so called because it is not associated with a moving current.

Static electricity is commonly generated through the triboelectric effect which involves the transfer of electrons from one material to another via physical contact such as rubbing or scraping. If said materials are good electrical insulators, charges will remain on the materials and can build to very high levels, creating a large voltage. Many plastics, such as polystyrene and polyester, are good candidates. The effect can be noticed with certain fabrics, especially under low humidity. For example, removing a polyester fleece pullover or jacket can elicit a certain crackling sound. The sliding of the fleece builds up the charge and eventually the voltage will be become so large that it will arc through the air to surrounding objects which have a lower voltage. This happens quickly over many parts of the garment, each crackle being an individual arc. In fact, if tried in darkness, it is possible to see a cascade of small sparks. This is the same phenomenon that causes a spark when you touch a car after sliding off the seat on a cold and dry winter's day, or a small shock when you touch an object (or another person) after walking across a carpet in a dry library.



Figure 2.4.2 : Triboelectric effect and static electricity: Cat fur meets polystyrene.

Anyone who has opened a box filled with polystyrene packing peanuts can attest to the troublesome nature of the triboelectric effect, as the very light peanuts can easily adhere to other objects due to the electric charge generated through their displacement. No amount of manic brushing or throwing of the pieces will reduce the effect and may, in fact, make it worse. A simple solution in some instances is to spray a fine mist of water on the packing peanuts as the water will provide a conduction path, draining off the charges. Of course, this will not be appropriate in all situations, particularly in the one shown in Figure 2.4.2 . In the prior examples, the static voltage may be on the order of a few thousand volts but the associated energy may be just a few microjoules. In spite of the high voltage, this is not enough to kill someone. On the other hand, the same voltage achieved with a much higher charge and energy could be lethal.

Beyond its simple inconvenience and inadvertent feline entertainment capabilities, high static potential can damage sensitive electronic devices. Care must be taken to prevent the accidental buildup of damaging charges. In the electronics industry this is commonly referred to as ESD, or electrostatic discharge. Steps to reduce ESD include humidity control and the use of conductive devices such as resistive wrist straps for technicians to continually bleed off the charges, thus preventing the creation of a high static potential.

The Height Analogy

Just as the flow of water can be seen as an analogy for electrical current, a serviceable analogy for voltage involves pressure or height. Indeed, sometimes voltage is referred to as “electrical pressure”. The height analogy ties together the concepts of voltage and energy. In this analogy, height corresponds to voltage and mass corresponds to charge.

To begin, we note that there are two kinds of energy: kinetic energy, or energy of motion; and potential energy, or energy by virtue of position. Potential energy is the product of mass, gravity and height, or $w = mgh$. Keeping gravity equal, we see that the more mass something has or the higher up it is, the greater its potential energy. We might think of potential energy as the object's potential to inflict damage when released.



Figure 2.4.3 : Understanding voltage: David Hume does a header.

To illustrate, we shall call upon 18th century Scottish philosopher and noted soccer fan[2], David Hume. If a soccer ball is held stationary over Mr. Hume's head, as shown in Figure 2.4.3 , the ball has energy by virtue of its position. Releasing the ball from the position shown will scarcely bother Mr. Hume as there is little energy associated with this position relative to the top of his noggin. In fact, he would be hard pressed to head the ball to another player. If, on the other hand, the ball were held considerably higher, its potential energy would be much greater. Therefore, the impact on Mr. Hume's head would be increased dramatically and he would have little difficulty heading the ball down field (i.e., the transformation of potential energy into kinetic energy), although the chances of him gaining a concussion are greatly increased.

Thus, we note that the height of the ball gives us some insight into its potential energy, although these terms are not synonymous. That is to say, if, instead of a soccer ball, we had used a ping-pong ball, even when dropped from an extreme height, the chances of a concussion are non-existent[3]. On the other hand, if the ball had been replaced with one made of solid iron, a drop from even a modest height could render our most excellent philosopher seriously dead. So it is with voltage. If the charge associated with the voltage is small, even a relatively high voltage will not be lethal (as in the case of simple static electricity on clothing), however, if associated with a sufficiently large charge, a much lower voltage can be deadly

References

[1]E is used for voltage sources such as batteries. It is short for EMF, or electromotive force.

[2]Hume, author of An Enquiry Concerning Human Understanding, died in 1776. The modern game of soccer was established in 1863. Let's not let that deter our own inquiry.

[3]The construction of the referred sentence owes a certain debt to the writing style of Mr. Hume.

This page titled 1.4: Energy and Voltage is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.4: Energy and Voltage** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.5: Power and Efficiency

The terms power and energy are often used incorrectly as synonyms. Although related, they are not the same thing. As already mentioned, energy is the ability to do work. In contrast, power is the rate of energy usage. Power is denoted by the letter P and has units of watts, although other units are sometimes used such as the horsepower (1 horsepower $\approx$ 746 watts). One watt is defined as one joule of energy consumed per second.

$$1 \text{ watt } \equiv 1 \text{ joule } /1 \text{ second} \tag{1.5.1}$$

As a formula,

$$P = W/t \tag{1.5.2}$$

Where

P is the power in watts,

W is the energy in joules,

t is the time in seconds.

To better understand the concept, consider for a moment a delicious peanut butter and banana sandwich. This sandwich contains a certain number of food calories, let's say 300 in total. A food calorie refers to a certain amount of energy that humans can extract from an item of food. That energy enables us to do some manner of work such as walking, swimming or just breathing. The sandwich can be seen as an energy storage medium, a battery for biological units called humans. The question is, what do we do with the energy, and more to the point, how fast do we use it? For example, that sandwich might be sufficient to allow someone to run a 5k (3.1 mile) road race in 17 minutes. In contrast, it also might be sufficient to allow that same person to watch television for three hours. It's the same amount of energy being used, it's just being used at a much faster rate in the former case. That rate is power. The 5k runner has a much higher power output than the TV watcher.

Example 2.5.1

100 joules are consumed by a device in 0.1 seconds. Determine the power in watts and in horsepower.

$$P = \frac{W}{t}$$

$$P = \frac{100J}{0.1s}$$

$$P = 1000W$$

As one horsepower is approximately 746 watts, this is equivalent to

$$P_{hp} = \frac{P_W}{746W/hp}$$

$$P_{hp} = \frac{1000W}{746W/hp}$$

$$P_{hp} = 1.34hp$$

Power can also be found by multiplying a current by the associated voltage. To begin, we note the definitions of current and voltage, Equations 2.3.2 and 2.4.2 respectively, and then combine them.

$$I = \frac{Q}{t}$$

$$V = \frac{W}{Q}$$

$$I \times V = \frac{Q}{t} \times \frac{W}{Q} = \frac{W}{t}$$

From Equation 1.5.2, we know that $P = W/t$, thus $P = IV$. This is known as power law.

$$P = I \times V \tag{1.5.3}$$

Where

P is the power in watts,

I is the current in amps,

V is the voltage in volts.

Example 2.5.2

If a 9 volt battery delivers a current of 0.1 amps, determine the power delivered in watts.

$$P = I \times V$$

$$P = 0.1amps \times 9volts$$

$$P = 0.9W$$

Efficiency

Efficiency is the ratio of useful output power to applied power expressed as a percentage. It is denoted by the Greek letter η (eta) and is always less than 100%. Expressed as a formula,

$$\eta = \frac{P_{out}}{P_{in}} \times 100\% \tag{1.5.4}$$

Where

η is the efficiency in percent,

P_{out} is the output power,

P_{in} is the input power.

Generally speaking, the higher the efficiency, the better. This implies less waste. In other words, if a system is 30% efficient, then 70% of the input power is wasted, whereas if a system is 99% efficient, then only 1% of the input power is wasted. The concept is illustrated graphically in Figure 2.5.1 . In most systems, waste power turns into heat which is not a desired commodity, and in fact often reduces the lifespan of electrical components.

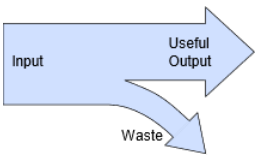


Figure 2.5.1 : Basic concept of efficiency.

Example 2.5.3

If a device draws 200 watts of power and has a useful output of 120 watts, determine the efficiency.

$$\eta = \frac{P_{out}}{P_{in}} \times 100\%$$

$$\eta = \frac{120W}{200W} \times 100\%$$

$$\eta = 60\%$$

In this case, the device is wasting 40% of the input power, or 80 watts.

Example 2.5.4

An audio amplifier has a maximum rated output of 100 watts to a loudspeaker. If it exhibits an efficiency of 70%, determine the input power required and the amount of power wasted.

$$\eta = \frac{P_{out}}{P_{in}} \times 100\%$$

$$P_{in} = \frac{P_{out}}{\eta} \times 100\%$$

$$P_{in} = \frac{100W}{70\%} \times 100\%$$

$$P_{in} = 142.9watts$$

As 142.9 watts are drawn by the amplifier and only 100 watts are delivered to the loudspeaker, then the difference, or 42.9 watts, is wasted power (most likely just making the amplifier hot).

This page titled 1.5: Power and Efficiency is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.5: Power and Efficiency** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.6: Energy Cost and Battery Life

As we have seen, knowing the voltage and current demands of a given device allows us to determine its power rating and energy consumption. The next steps are to determine the cost of operating a device and, if it's battery powered, how long the device will last before needing new batteries.

Computing Energy Cost

Once we know the power drawn by the device and for how long it will be used, the cost to operate a device can be determined given the per unit cost of energy. In spite of the fact that many people refer to their local electricity supplier as “the power company”, we do not buy “power”, per se. Rather, we are billed for energy. Although it would be possible to determine the cost per joule (or more practically, per megajoule), suppliers normally bill based on kilowatt-hours, or kWh, the product of the power and time[1]. This unit is used because most residential and commercial devices and appliances are rated in terms of power consumption in watts. Multiplying the power consumption in watts by the length of time the device is used in hours yields a watt-hour value. This is then scaled by a factor of one thousand to arrive at kWh. For example, a 1500 watt toaster oven used for 30 minutes (i.e., 0.5 hours) yields 750 watt-hours, or 0.75 kWh. Finally, knowing the per kWh cost, a simple multiplication yields the cost of electricity. Thus, if the utility charges 10 cents per kWh, then the cost to run that toaster oven is 7.5 cents. If it is used for a full hour then it costs 15 cents, and so on.

To put usage in perspective, a typical home in the USA or Canada uses about 900 kWh per month while households in many countries in Europe might use one half to one quarter of that amount. Global electricity generation is around 25 million gigawatt hours per year. Typical electricity rates in the USA are between ten and twenty cents per kWh, depending on geographic region and sector (e.g., residential or commercial).

Example 2.6.1

A 100 watt incandescent light bulb is left on for 24 hours. If the cost of electricity is 15 cents per kWh, determine the cost to run the light.

$$Cost = P \times t \times rate$$

$$Cost = 100W \times 24hours \times 0.15\$/kWh$$

$$Cost = \$0.36$$

At this point, it should be obvious that the more efficient a device is, the less expensive it will be to run. In fact, it is quite possible that in the long run, a device that is more expensive than a similar, though less efficient, device may be considerably less expensive to use over the course of its lifespan. A good example is a comparison between an ordinary incandescent light bub and an LED light. LED lighting can be an order of magnitude more efficient than incandescent lighting. Indeed, incandescent bulbs may convert less than five percent of their input into usable light output, the remaining 95 percent just turning into heat. While they are less expensive to purchase initially, their operating cost is much higher. A proper way to compare lights is to examine their light output in lumens, not their power consumption. For example, a 60 watt incandescent light produces about 800 lumens of illumination. That same level of illumination can be obtained with an LED drawing just 9 watts. Further, typically LED lights last ten to twenty times longer than incandescent bulbs.

This will be illustrated in the next example.

Example 2.6.2

A certain 14 watt LED light produces the same illumination as a 75 watt incandescent light bulb. Assume the LED costs $11 and has an expected life of 15,000 hours. The incandescent costs 50 cents each and has an expected life of 1000 hours. If the cost of electricity is 12 cents per kWh, determine the cost to run each version for 15,000 hours.

First, it should be noted that 15 incandescent bulbs will be needed. At 50 cents each, that's $7.50 for the bulbs. The cost to run them is,

$$Cost = P \times t \times rate$$

$$Cost = 75W \times 15000hours \times 0.12\$/kWh$$

$$Cost = \$135.00$$

The total is $ 142.50. Now for the single LED required:

$$Cost = P \times t \times rate$$

$$Cost = 14W \times 15000hours \times 0.12\$/kWh$$

$$Cost = \$25.20$$

The total is $36.20, a considerable savings, not to mention other positive factors including only having to change the light once instead of fifteen times; a considerable reduction in wasted energy, thus lowering demand and environmental impact; and finally, a reduction in burned out light bulbs for a further lowering of environmental impact (waste stream reduction).

Batteries

A battery is a device used to store electrical energy, generally in the form of a chemical cell. Ideally, it presents a constant voltage, its current varying according to what it drives. In reality, as the battery is used, its voltage will begin to decrease. Eventually, the energy stored in the battery will be exhausted and its voltage will drop to zero. The storage capacity of a battery is measured in amp-hours, Ah (or milliamp-hours, mAh, for smaller batteries). All other factors being equal, the battery with the higher amp-hour rating will last longer before being depleted. This is true for the same size of battery using different compositions (e.g., zinc-carbon versus alkaline), as well as batteries of different physical sizes but having the same voltage. For example, all AAA, AA, C and D cell batteries have a nominal voltage of 1.5 volts. If they are all of the same type, such as alkaline, then the immediate practical difference is that the larger the physical size, the greater its energy storage, and thus the longer it will last.

Over a modest range of currents, the expected lifespan of a battery can be computed based on its amp-hour rating and the current drawn from it.

$$\text{Battery life} \approx \text{amp-hour rating} / \text{current draw} \tag{1.6.1}$$

This equation is best used as a rough guide. If the current draw is considerably higher than the current at which the battery was tested, the predicted lifespan will be overly optimistic. On the other hand, if the current draw is considerably lower, it is likely that the battery will last longer than predicted.

The maximum current output of a battery is likewise limited. If such were not the case, we might expect even very small batteries to produce phenomenally large currents for very shorts periods of time. This is not the case.

A graph of discharge curves at various load currents is shown in Figure 2.6.1 . Notice how the battery voltage begins at the rated 1.5 volts and then begins to fall. After a certain point, the rate of decrease accelerates and “drops off of a cliff”. The actual service life will also be application dependent in that some devices can tolerate a lower voltage than others. For example, an old-fashioned flashlight will still work with largely depleted batteries, it simply won't be that bright. If we were to consider 75% of rated voltage as the useful lower limit (a little over 1.1 volts), we see that at a 50 mA draw, the battery will last around 18 hours, achieving about 900 mAh. At 100 mA, it will last around 7 hours, yielding 700 mAh. If we used 1.0 volts as our lower usable limit, we would arrive at 1.05 Ah and 910 mAh, respectively.

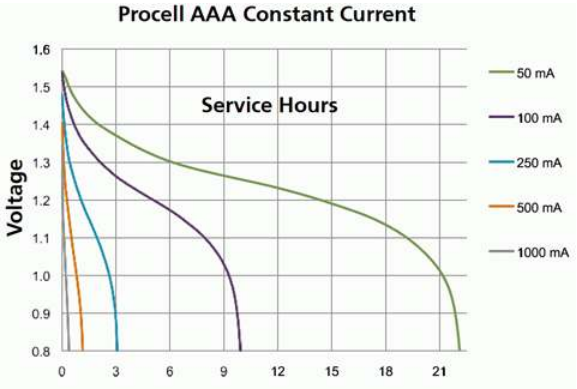


Figure 2.6.1 : Battery discharge curves for various load currents at room temperature. Courtesy of Duracell

Example 2.6.3

A certain battery is rated at 10 Ah. Approximately how long will it last with a 0.5 amp draw?

$$Lifespan \approx \frac{Ah}{I}$$

$$Lifespan \approx \frac{10Ah}{500mA}$$

$$Lifespan \approx 20hours$$

Remember, 20 hours is only an approximation. For a more accurate rendering, consider the following example.

Example 2.6.4

Using the graph of Figure 2.6.1 , determine the expected lifespan with a 100 milliamp draw for a lower voltage limit of 1.2 volts. Also determine the effective amp-hour rating at this point.

The 50 mA curve (purple) passes through 1.2 volts at approximately 5 hours. This is the expected lifespan. The corresponding amp-hour rating is

$$Ah \approx I \times t$$

$$Ah \approx 50mA \times 5hours$$

$$Ah \approx 250mAh$$

Batteries also tend to show decreased performance with reductions in temperature. This is shown in Figure 2.6.2 . For this particular battery, the capacity at freezing (0°C) is roughly half of what it is at room temperature (21°C). Peak current capacity may also be reduced with temperature. This is true of many kinds of batteries, including large 12 volt automotive batteries. This is a major reason why cars are often much more difficult to start on very cold days; their current capacity is reduced, thereby reducing the output of the starter motor.

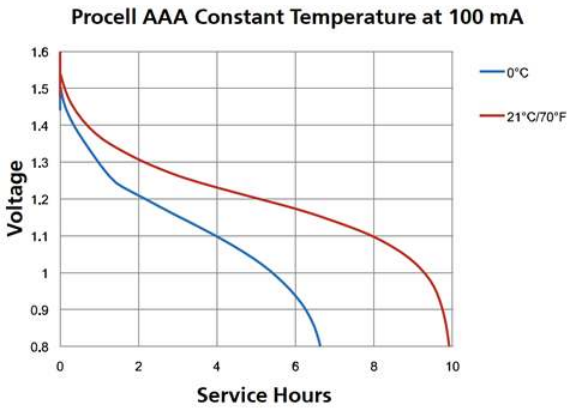


Figure 2.6.2 : Battery discharge curves for different temperatures. Courtesy of Duracell

A table of typical amp hour ratings for common battery sizes is shown in Table 2.6.1 . These values are appropriate for good quality alkaline batteries. Rechargeable NiMH (nickel-metal hydride) would be around the same. Remember, these are just approximations.

Table 2.6.1 : Typical battery capacities.

Battery Size	Capacity (mAh)
AAA	1000
AA	2500
C	5000
D	10000
9 Volt	500

References

[1]Remember, power is the rate of energy usage per unit time, and thus multiplying power by time yields energy. One kWh is approximately equal to 3.6 megajoules.

This page titled 1.6: Energy Cost and Battery Life is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.6: Energy Cost and Battery Life** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.7: Resistance and Conductance

We have seen that both current and voltage deal with the movement of charge. Consequently, in any electrical system, voltage and current are interrelated. Let us consider the most simple case. This would involve a single voltage source, such as a battery, driving a single, homogeneous item such as a length of wire or a block of a given material. The physical characteristics of this item will dictate how much current will flow. In general terms,

$$Effect = \frac{Cause}{Oppostion}$$

In this case the cause is the voltage source and the effect is the resulting current. The opposition is the characteristic of the item in question, in other words, its ability to restrict current flow. We call this characteristic resistance. In other words, resistance is a measure of how difficult it is to establish a flow of current (i.e., "resistance to current flow") under a given set of circumstances. It is denoted by the letter R and has units of ohms, in honor of Georg Ohm, a researcher from the early 1800s. The unit is denoted by the capital Greek letter omega, Ω.

Sometimes it is more convenient to use the inverse view of this phenomenon, and instead of referring to how difficult it is to establish a current, we would be interested in how easy it is to establish a current. This is called conductance and it is the reciprocal of resistance. Conductance is denoted by the letter G and has units of siemens, named after Werner von Siemens. The unit is abbreviated as S.

$$R = \frac{1}{G} \tag{1.7.1}$$
$$G = \frac{1}{R}$$

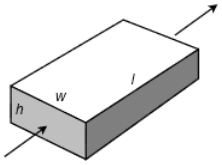


Figure 2.7.1 : Defining resistance.

In this simple scenario, resistance is a function of the material the current is passing through along with its shape. This is illustrated in Figure 2.7.1 where the arrow shows the direction of current flow. An obvious question is "What is this block made of?" It should come as no surprise that the material chosen has a great impact on the current. We have already seen that metals such as copper are good conductors of electricity. Other materials, such as certain plastics and ceramics, are not. These materials are referred to as insulators.

The measure of a material's inherent and general ability to restrict current flow is called resistivity. Resistivity is denoted by the Greek letter ρ (rho). All other factors being equal, the higher the resistivity, the greater will be the resistance. Further, the greater the length of the material that the current must pass through, the greater the resistance. Finally, as the surface area of the cross section (i.e, the face) grows, the resistance decreases. Expressed as a formula

$$R = \frac{\rho l}{A} \tag{1.7.2}$$

Where

R is the resistance in ohms,

ρ is the resistivity,

l is the length of the material,

A is area of the face (h times w).

Resistivity is often specified in ohm-centimeters with the length and area similarly specified in centimeters and square centimeters, respectively. A table of resistivity values for a variety of materials is shown in Table 2.7.1 . Note that resistivity is not necessarily

constant across temperature. Indeed, this change can be exploited as a means of measuring temperature. In other applications, we might need it to be as stable as possible across temperature. This need led to the creation of the alloys Constantan and Manganin in the late 1800s which exhibit very high stability across temperature.

From this table we can see that silver has lower resistivity than copper which in turn is lower than gold. This means that if we made identically sized wires of these three materials, the silver version would have the least resistance and gold the highest. Why then, do audio and video cables often feature gold plating? Certainly, it isn't due to lower resistivity and enhanced conductivity. The reason is that gold is a noble metal, meaning that it does not tarnish. In contrast, the surface of both silver and copper will oxidize, creating a patina (the dark “stain” noticeable on old silver and copper implements). The oxide will create a high resistance layer and reduce the integrity of the connection.

It is important to note that Formula 1.7.2 does not include a term for the mass of the material. It is only concerned with the shape of the item. This is an important distinction. If we were to alter the mass but keep the ratio of the length versus the area the same, the resulting resistance would be unchanged. In general, the increase in mass by itself does not necessarily alter the resistance but it may have an impact on the power handling capability of the device.

Table 2.7.1 : Resistivity values for various materials.

Material	Resistivity $\rho(\Omega\cdot\text{cm})$ at 20 °C	Temperature coefficient (K^{-1})
Silver	1.59×10^{-6}	0.0038
Copper	1.68×10^{-6}	0.00404
Gold	2.44×10^{-6}	0.0034
Aluminium	2.65×10^{-6}	0.0039
Zinc	5.90×10^{-6}	0.0037
Nickel	6.99×10^{-6}	0.006
Iron	9.7×10^{-6}	0.005
Platinum	1.06×10^{-5}	0.00392
Tin	1.09×10^{-5}	0.0045
Titanium	4.20×10^{-5}	0.0038
Manganin	4.82×10^{-5}	0.000002
Constantan	4.90×10^{-5}	0.000008
Stainless steel	6.90×10^{-5}	0.00094
Nichrome	1.10×10^{-4}	0.0004
Carbon (amorphous)	5×10^{-4} to 8×10^{-3}	−0.0005
Silicon	6.4×10^{4}	−0.075
Glass	1×10^{13} to 1×10^{17}	
Carbon (diamond)	1×10^{14}	
Hard rubber	1×10^{15}	
Air	10^{11} to 10^{17}	
PET	1×10^{23}	
Teflon	1×10^{25} to 1×10^{27}	

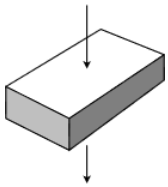


Figure 2.7.2 : Resistance will change due to to altering area and length with mass unchanged.

In contrast, if we take that that original mass and reshape it, or just apply the current to a difference face, such that the effective surface area and length change, then the resulting resistance will also change. This is illustrated in Figure 2.7.2 . Here we have taken the item shown in Figure 2.7.1 and directed the current flow from top to bottom rather than through the side. In this orientation, the surface area is much increased and the length through which the current must travel is greatly reduced. Consequently, the effective resistance in this orientation will be considerably less than that seen in the original.

Example 2.7.1

A certain material has a resistivity of 0.2 ohm-centimeters. Determine the resistance of a piece that is 0.3 cm wide, 0.5 cm high and 4 cm long.

$$A = h \times w$$

$$A = 0.5cm \times 0.3cm$$

$$A = 0.15cm^2$$

$$R = \frac{\rho l}{A}$$

$$R = \frac{0.2\Omega cm \times 4cm}{0.15cm^2}$$

$$R = 5.333\Omega$$

For our next example, let's consider a spool of wire. In many cases we treat wire ideally, as if it has no resistance. While this can be a good approximation in many instances, especially with relatively short runs of wire, such is not always the case.

Example 2.7.2

A certain gauge of copper wire has a diameter of 0.6 mm. Determine the resistance if the spool is 200 meters long.

Table 2.7.1 indicates that the resistivity of copper is 1.68E−6 ohm-centimeters. The trick here is that we must keep the units consistent. As there are 100 centimeters to the meter, the length is 200 meters times 100, or 20,000 centimeters (i.e., 20E3). Given that there are 10 millimeters to the centimeter, the diameter must be decreased by a factor of ten, yielding a diameter of 0.06 cm and thus a radius of 0.03 cm.

$$A = \pi r^2$$

$$A = \pi(0.03cm)^2$$

$$A = 2.83E-3cm^2$$

$$R = \frac{\rho l}{A}$$

$$R = \frac{1.68E-6\Omega cm \times 20E3cm}{2.83E-3cm^2}$$

$$R = 11.9\Omega$$

The amount of resistance seen in Example 2.7.2 would be considered excessive if the item to be connected is something as simple as a loudspeaker, which typically would be around 8 Ω. And while no one would likely need 200 meters of cable to connect a loudspeaker in their home, that sort of distance would be unremarkable in a large stadium or airport terminal. Don't forget, we'd need wire to and from the loudspeaker, achieving a total separation of 100 meters at most.

Various thicknesses of cables and wires are used for a wide variety of purposes. To make this easier, wire thicknesses have been standardized. The most common standard in North America is the American Wire Gauge, or AWG. This is a nonmetric specification with origins in the mid nineteenth century. The larger the gauge number, the smaller the wire diameter, and the less current it safely can carry. To put the gauge numbers in perspective, typical small home appliances use 16 or 18 gauge wire, basic home wiring uses 12 gauge (with a 20 amp circuit breaker), and hook-up wire used in an electrical circuits or electronics laboratory solderless breadboard is commonly 22 or 24 gauge.

For general purpose wiring, copper is by far the most common metal used because it is highly conductive and relatively inexpensive. Some other metals are used in special cases, for instance, aluminum is used for long distance power transmission lines.

Table 2.7.2 : AWG wire data for copper conductors.

AWG	Diameter (in)	Diameter (mm)	Resistance/length (Ω/km)	Resistance/length (Ω/1000ft)
0000 (4/0)	0.4600	11.684	0.1608	0.04901
00 (2/0)	0.3648	9.266	0.2557	0.07793
0 (1/0)	0.3249	8.251	0.3224	0.09827
2	0.2576	6.544	0.5127	0.1563
4	0.2043	5.189	0.8152	0.2485
6	0.1620	4.115	1.296	0.3951
8	0.1285	3.264	2.061	0.6282
10	0.1019	2.588	3.277	0.9989
12	0.0808	2.053	5.211	1.588
14	0.0641	1.628	8.286	2.525
16	0.0508	1.291	13.17	4.016
18	0.0403	1.024	20.95	6.385
20	0.0320	0.812	33.31	10.15
22	0.0253	0.644 5	52.96	16.14
24	0.0201	0.511	84.22	25.67
26	0.0159	0.405	133.9	40.81
32	0.00795	0.202	538.3	164.1
40	0.00314	0.0799	3441	1049

A table of gauge sizes and associated parameters is shown in Table 2.7.2 . This table assumes copper is being used for the wire. Note that as the diameter of the wire decreases, the amount of resistance for a particular length increases, as expected from Equation 1.7.2.

Using this table we can perform a quick crosscheck of Example 2.7.2 . The wire diameter used in that example was 0.6 mm which is just a little smaller than AWG 22 as listed in the table. Further, #22 wire is listed as having a resistance of approximately 53 Ω per km. For 200 meters, as used in the example problem, this works out to 10.6 Ω. As #22 wire is slightly larger in diameter, we expect it to show slightly less resistance than the calculated value of 11.9 Ω, which it does.

While it may not be immediately apparent, gauge numbers proceed in a logarithmic fashion based on diameter. Stepping up one gauge number (e.g., from #10 to #11) decreases the diameter by a factor of approximately 0.89. As resistance is inversely proportional to the square of the diameter (i.e., the area), the resistance increases by over 25%. Even numbered sizes are particularly common in use and a jump to the next higher even gauge number (e.g., #18 to #20) produces a resistance increase of nearly 60%.

Resistors and the Resistor Color Code

Resistors are devices used to control the currents and voltages in a circuit. They are available in many shapes and sizes, and are normally designed to maintain stable ohmic values in spite of environmental changes such as temperature and humidity. A sample of different resistor styles is shown in Figure 2.7.3 . As a general rule, the larger the resistor, the more power it can handle. At the back of the figure is a large ceramic power resistor using a rectangular aluminum housing. This device is rated for 200 watts of dissipation. Immediately in front and to the right of it are several smaller ceramic power resistors with ratings in the 5 to 20 watt range. Along the left side is a set of carbon composition and carbon film resistors ranging from 1 watt down to one-tenth watt in dissipation. Toward the center is a multi-lead chip resistor that contains several resistors in one package. With few exceptions, all of these items are classified as “through-hole” components, that is, their leads are designed to go through pre-drilled holes in a printed circuit board. These are also the most commonly used components in an electrical lab as the leads fit into solderless breadboards and the components are a convenient size.

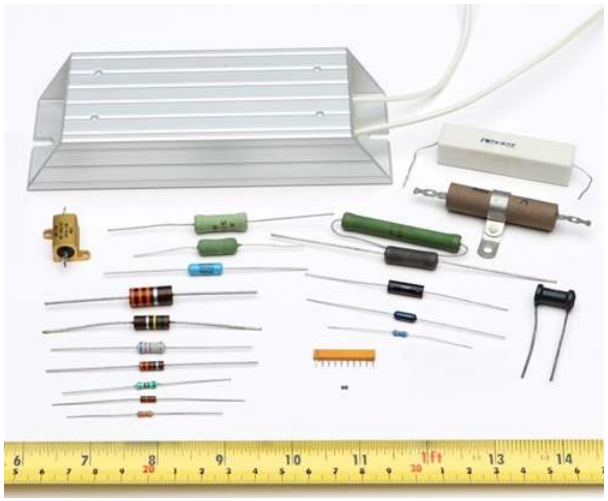


Figure 2.7.3 : A variety of different resistors.

With the increasing desire to shrink components, modern production designs use surface mounting techniques in place of through-hole construction. At the bottom-center is a small dot which is, in fact, a surface mount resistor capable of 1/2 watt of power dissipation. A close-up is shown in Figure 2.7.4 . Obviously, these devices are too small to be practical to use without special equipment.



Figure 2.7.4 : Close up of a surface mount resistor.

Resistors are linear bilateral devices. Being linear, their current-voltage relation can be drawn as a straight line. Bilateral means that the polarity of voltage or direction of current will not matter. In other words, unlike a battery, these devices cannot be inserted into

a circuit backwards because either orientation works the same way. If the horizontal axis is voltage and the vertical is current, then the slope of the line yields the conductance. Thus, the steeper the line, the lower its resistance. This is illustrated in Figure 2.7.5 .

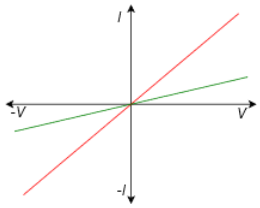


Figure 2.7.5 : Current-voltage plot of two resistors.

Not all electrical components are linear and bilateral. A good example is the semiconductor diode, a commonly used device in electronic systems. The current-voltage plot of a diode is shown in Figure 2.7.6 for comparison. Note that the plot is not a simple straight line, thus it is not linear. Further, the first and third quadrant responses are wildly different, indicating that polarity is of utmost importance. Clearly, it matters which way these kinds of devices are inserted into a circuit.

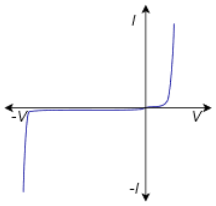


Figure 2.7.6 : Current-voltage plot of a diode: neither linear nor bilateral.

Resistors are available in standardized ohmic values and at standardized power ratings (see Appendix A). Along with their resistance value, resistors also have a specified tolerance. This specifies an allowable range of variation of the stated value. For example, a 220 ohm resistor may have a tolerance of 10%. This means that the actual value of any given specimen from a box of these resistors may be off of the nameplate or nominal value by 10% or 22 ohms. Thus, any particular resistor might be as high as 242 ohms or as low as 198 ohms.

General purpose resistors use a color code to denote their value and tolerance. Typically, this will involve four color stripes: two for the precision/mantissa, one for the power of ten, and the fourth to indicate the tolerance. For higher precision, a five stripe version may be used with the first three denoting the precision/ mantissa. Alternately, high precision resistors may have their nominal value printed directly on them. Refer to Figure 2.7.7 for an example of the basic variety.



Figure 2.7.7 : Example of basic four stripe resistor color code.

The first two bands, here yellow and violet, indicate the precision or leading digits. The third band, here orange, indicates the power of ten or “number of zeroes” to add. The fourth band, silver in this example, indicates the tolerance. Note that the fourth band is spaced away from the others to avoid accidentally reversing the order.

The color code is shown in Figure 2.7.8 . To assist in remembering the sequence, a number of mnemonic aids have been used, the first letter of each word starting with the same letter as the corresponding color. One example is the “Picnic Basket Mnemonic” which is: Black Bears Robbed Our Yummy Goodies Beating Various Gray Wolves. Another possibility is to note that the middle section follows a rainbow with black and white at the extreme ends.

Color		Numeral
	Black	0
	Brown	1
	Red	2
	Orange	3
	Yellow	4
	Green	5
	Blue	6
	Violet	7
	Gray	8
	White	9
For small resistor values If the multiplier band is gold then divide by10; if silver, divide by 100.		

Figure 2.7.8 : Resistor color code.

The tolerance band colors are as follows: For basic parts silver is $\pm 10\%$ and gold is $\pm 5\%$. If the fourth band is omitted, this indicates a tolerance of $\pm 20\%$, although it is seldom used in modern designs. For precision parts some colors are reused but follow the color code numerals: Brown is $\pm 1\%$ tolerance and red is $\pm 2\%$ tolerance. Tighter tolerances are also available. Sometimes an extra band is added that indicates a reliability rating or temperature coefficient. In such cases, it is best to refer to the manufacturer's data sheets for details. We will not pursue these further.

Example 2.7.3

Determine the nominal, maximum and minimum acceptable resistance values for the resistor pictured in Figure 2.7.2 .

The colors are yellow-violet-orange. This translates to 4, 7 and 3. The value is “47 with 3 zeroes”, or 47000 ohms. The silver fourth band indicates 10% tolerance. Thus, the resistor pictured is nominally 47 kΩ with $\pm 10\%$ variation around the nominal value being acceptable. The tolerance yields ± 4.7 kΩ, so the acceptable range is from 42.3 kΩ to 51.7 kΩ.

Example 2.7.4

A precision resistor has a color code of orange-blue-green-brown-brown. Determine its value and acceptable range.

The final band indicates that this is a $\pm 1\%$ tolerance component. The first four colors translate to 3, 6, 5 and 1. The value is “365 with 1 zero”, and thus 3650 ohms, or 3.65 kΩ. The allowable range is 3.65 kΩ $\pm$ 36.5 Ω.

Example 2.7.5

Determine the nominal value and tolerance of a resistor with the color code green-blue-gold-silver.

The first two colors translate to 5 and 6. Gold in the third band means “multiply by 0.1”. The final band indicates that this is a $\pm 10\%$ tolerance component. Therefore, nominal value is 5.6 Ω. The allowable range is 5.6 Ω $\pm$ 0.56 Ω.

An example of a resistor data sheet is shown in Figure 2.7.9 . This data sheet is for a series of surface mount chip resistors. The available tolerance grades range from 0.5% to 20%. Also, for each variant there are two temperature coefficients available with the

most stable being 100 ppm/◦C (ppm is short for “parts per million”, thus 100 ppm is equivalent to 0.01%). Note that these devices are too small to use the resistor color code. Instead, a numeric code is printed directly on them that follows the rules for color bands, meaning that the final digit is the multiplier. Thus “150” is 15 Ω (1-5 with no zeroes). The exception is very small values where the letter “R” is used in place of a decimal point. Consequently, “2R4” is 2.4 Ω.

RPC Series
Pulse Withstanding Thick Film Chip Resistor

Stackpole Electronics, Inc.
Resistive Product Solutions

Features:

- Excellent pulse withstanding performance
- Broad resistance range
- Higher anti-surge performance compared with RMCF Series
- Standard power RPC, 5% and wider tolerances, are untrimmed
- 1% and wider tolerances are qualified to AEC-Q200
- RoHS compliant and halogen-free
- Lower values may be available – contact factory



Electrical Specifications

Type/Code	Power Rating (Watts) @ 70ºC	Maximum Working Voltage	Maximum Overload Voltage	Resistance Temperature Coefficient	Ohmic Range (Ω) and Tolerance		
					0.5%	1%	5%, 10%, 20%
RPC0402	0.2W	50V	100V	±300 ppm/ºC	-	1 - 20	
				±100 ppm/ºC	1 - 1M	20.5 - 1M	22 - 1M
RPC0603	0.1W	50V	100V	±200 ppm/ºC	10 - 294	1 - 294	10 - 270
				±100 ppm/ºC	300 - 1M		
RPC0805	0.25W	150V	300V	±200 ppm/ºC	10 - 294	1 - 294	1 - 270
				±100 ppm/ºC	300 - 20M		
RPC1206	0.33W	200V	400V	±200 ppm/ºC	10 - 20	1 - 20	
				±100 ppm/ºC	20.5 - 20M		22 - 20M
RPC1210	0.5W	200V	400V	±200 ppm/ºC	10 - 20	1 - 20	
				±100 ppm/ºC	20.5 - 20M		22 - 20M
RPC2010	0.75W	400V	800V	±200 ppm/ºC	10 - 20	1 - 20	
				±100 ppm/ºC	20.5 - 20M		22 - 20M
RPC2512	1.5W	500V	1000V	±200 ppm/ºC	10 - 20	1 - 20	
				±100 ppm/ºC	20.5 - 20M		22 - 20M

Working Voltage = √(P*R) or Max. Working Voltage listed above, whichever is lower.
Overload Voltage = 2.5*√(P*R) or Max. Overload Voltage listed above, whichever is lower.

Figure 2.7.9 : Example resistor data sheet. Courtesy of Stackpole Electronics

A particularly important characteristic of most electronic devices is their power handling capacity. For this series there are several variations with power handling from 0.1 watts to 1.5 watts, and maximum working voltages up to 500 volts. Power handling is also a function of ambient temperature. In general, devices have a maximum internal temperature that they can reach before they are damaged. Control of heat tends to be a major issue in many system designs. If the ambient temperature increases, there is less “headroom” for the device's temperature rise, and thus the device will not be able to dissipate as much power. This is illustrated in the graph of Figure 2.7.10 .

Note how the power dissipation is constant at temperatures at and below 70 ◦C. This temperature is considered the maximum normal operating temperature for this device and the power dissipation at this temperature is the one quoted in the data sheet. If the device is operated in a warmer environment, the power dissipation is derated by the percentage given in the graph. For example, at 100 ◦C the dissipation is only about 65% of the nominal rating. Of particular importance, at 155 ◦C the device can no longer dissipate any power, and therefore this temperature serves as the absolute ceiling. After all, a resistor that cannot dissipate any power is a resistor that cannot have any current through it or voltage across it without burning up. It is essentially non-functional.

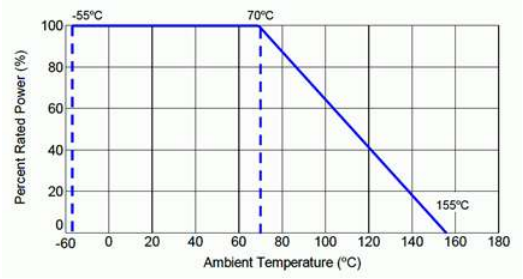


Figure 2.7.10 : Resistor power derating graph. Courtesy of Stackpole Electronics

Of course, just like a skillet on a stove, these devices do not heat up instantly, that is, they exhibit a thermal time constant. They take a certain amount of time to heat (and also to cool). For short periods, devices can handle considerably more power than their long term rating. The graph of Figure 2.7.11 , shows the power dissipation for single pulses. In some cases, these resistors can handle powers of an order of magnitude greater. For example, consider the type RPC0402 (bottom-most line). From the data sheet of Figure 2.7.9 this device is specified as having a long term rating of 0.2 watts. In contrast, the graph of Figure 2.7.11 shows it is

https://eng.libretexts.org/@go/page/52872

capable of withstanding a single pulse of greater than 20 watts for 100 microseconds, just under 10 watts for a millisecond pulse and roughly 2 watts for a one-tenth second pulse.

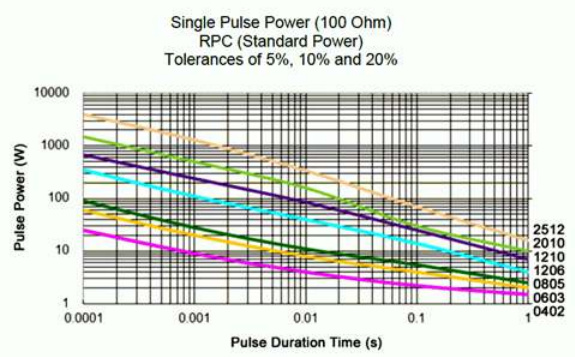


Figure 2.7.11 : Pulse power dissipation. Courtesy of Stackpole Electronics

Other Resistive Devices

Along with standard fixed resistors, there are several other kinds of resistive devices that have been designed to be sensitive to changes in their environment. Thus, they can serve as sensors because as their resistance changes, it impacts the flow of current and the resulting voltage. We will look at examples of this in upcoming work. Some of the environmental inputs include temperature, force and light levels.

Force Sensing Resistor (FSR)

The force sensing resistor consists of two layers of material with a nominal separation distance. It is presented as a flat membrane that is usually round or rectangular, and perhaps with an adhesive backing. An example of an FSR is shown in Figure 2.7.12 . With no force applied, the device shows an extremely high resistance, well into the megohms. As force is applied to the surface, the two layers come into better contact which decreases the net resistance. This is borne out in the graph of Figure 2.7.13 . The graph shows roughly a straight line response between resistance and force when plotted on a log-log scale. At the highest force levels, the resistance may drop to just a few hundred ohms.



Figure 2.7.12 : A force sensing resistor (FSR).

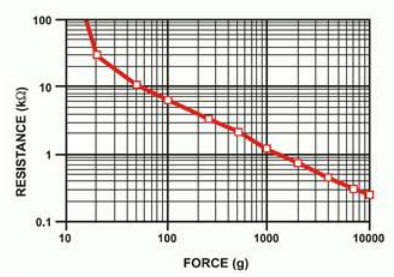


Figure 2.7.13 : FSR response curve. Courtesy of Interlink Electronics

Photoresistor

As their name implies, photoresistors are sensitive to changes in light level. They are also called LDRs, short for Light Dependent Resistor. Different materials may be used in their construction, but the most common is cadmium sulfide, CdS. As a consequence,

photoresistors are sometimes generically referred to as “CdS cells”.

A photoresistor is shown in Figure 2.7.14 along with its corresponding resistance graph in Figure 2.7.15 . In total darkness the device exhibits a very high resistance. As light levels increase, the resistance decreases. As with the FSR, we see an reverse relation between the resistance and the environmental factor: as the environmental input increases (force, light level), the resistance does the opposite and decreases. And once again, we see a straight line when plotted with log-log scales. Technically, we refer to this as a negative relation because the slope of the plot line is negative.



Figure 2.7.14 : A photoresitor, or LDR.

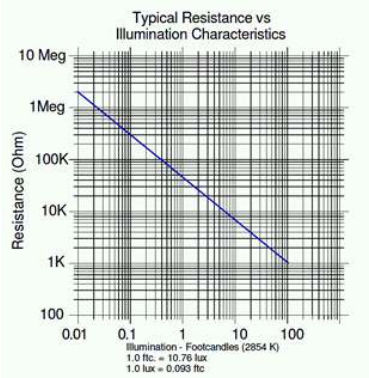


Figure 2.7.15 : Photoresitor response curve. Courtesy of Advanced Photonix

To put the brightness of the light into common terms, 0.01 foot-candles (roughly 0.1 lux) is equivalent to a clear moon-lit evening. At this level, the photoresistor is showing over 1 megohm of resistance. In contrast, 100 foot-candles (roughly 1000 lux) is equivalent to an overcast day (for reference, direct sunlight is perhaps 100 times stronger). At this level, the photoresistor's value has dropped to about 1000 ohms.

A point worth noting is that the light that the photoresistor “sees” is not necessarily the same as what a human sees. The sensitivity of the device at various wavelengths (i.e., colors) may differ starkly from human vision. In fact, differing materials exhibit different sensitivities at various wavelengths. Some of these may be wavelengths that the unaided human eye cannot see at all (infrared or ultraviolet, for example).

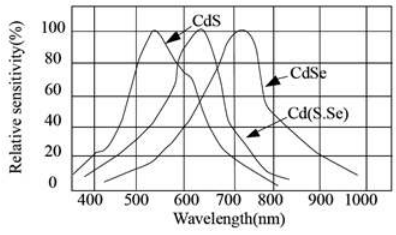


Figure 2.7.16 : Photoresitor sensitivity curves. Courtesy of Token Electronics

An example of sensitivity curves is shown in Figure 2.7.16 . Note the variations between the different materials, CdS being one of the three (left side). The peak sensitivities vary as do the precise shapes. In practice, this means that some of these units will be more or less sensitive to certain colors than other units. The CdS curve indicates peak sensitivity of about 540 nm, which corresponds to green. In comparison, the CdSe (cadmium selenide) cell exhibits a peak of just over 700 nm, which corresponds to red. At some wavelengths, the relative response of one material may be no more than 10% of the response of a different material.

It is important to note that the use of cadmium, such as in CdS cells and the like, is severely restricted by the RoHS directive (see Chapter One).

https://eng.libretexts.org/@go/page/52872

Thermistor

A thermistor is a device whose resistance is a function of temperature. These devices are available in two basic types. Either PTC, for Positive Temperature Coefficient; or NTC, for Negative Temperature Coefficient. PTC devices show an increase in resistance as temperature increases and NTC devices show a decrease in resistance as temperatures rise. Ideally, these are linear relationships with the plots showing straight lines. The reality is that linearity can only be assumed across fairly narrow ranges of temperature. For wider ranges, there will be noticeable deviation from a straight line as the curve is logarithmic in nature. A basic NTC thermistor is shown in Figure 2.7.17 .



Figure 2.7.17 : NTC thermistor.

Generic thermistor performance graphs are shown in Figure 2.7.17 with the idealized straight line response at top and the more realistic non-linear response below. Thermistors will be specified in terms of their room temperature resistance (usually taken to be 25 °C) along with their sensitivity which is denoted as beta (β). The larger the value of beta, the steeper the curve and the greater the sensitivity. The curves labeled Beta 2 show increased sensitivity, meaning that there will be a greater change in resistance for a certain change in temperature.

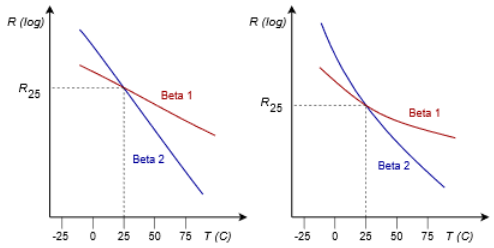


Figure 2.7.18 : NTC thermistor response curves: Ideal (top) and real.

The following equation can be used to determine the resistance of a thermistor at some other temperature of interest with greater accuracy than using a simple linear approximation. All that is needed is a reference temperature and corresponding resistance, the beta value and the new temperature of interest. Using a reference temperature of 25 °C:

$$R_T = R_{25} e^{\beta\left(\frac{1}{T} - \frac{1}{298.15}\right)} \tag{1.7.3}$$

Where

R_T is the resistance at the new kelvin temperature, T,

R_{25} is the resistance at 25 °C,

T is the temperature of interest in kelvin,

β is the device beta.

Note that the constant “298.15” in Equation 1.7.3 is equivalent to the reference temperature of 25 °C (0 K is −273.15 °C). Consequently, if a different reference point is used, simply insert the new reference temperature in its place and use the corresponding resistance in place of R_{25}.

Varistor

Varistors are used as limiting devices, primarily to suppress unwanted voltage spikes in electronic equipment. The varistor is a unique device in that it has a highly nonlinear current-voltage characteristic. This is shown in Figure 2.7.19 . Remember, when plotted with the voltage on the horizontal axis, the slope of the line represents the conductance. Consequently, the varistor shows a region of near zero conductance or extremely high resistance (the horizontal section), and two sections that are nearly vertical, which indicate extremely high conductance or near zero resistance. This characteristic allows the varistor to act as a limiting device.

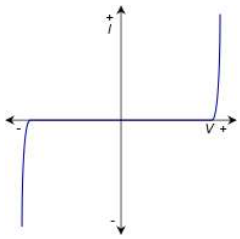


Figure 2.7.19 : Current-voltage characteristic of a varistor

Imagine that a lightning strike effects a local power line. This will create a sudden but short-lived spike in the voltage. A normal 120 volt wall outlet normally produces regular peaks of approximately 170 volts. A lightning strike might add several hundred volts to this. The resulting voltage could be so high that it would damage electronic equipment attached to the outlet. To alleviate this problem, a varistor can be placed across the incoming voltage lines. The vertical break point voltages would be set for a value just over 170 volts, the normal maximum. Under typical circumstances the varistor would see a voltage in its horizontal region and thus behave as a very high resistance. It would drain virtually no current from the outlet, and consequently it would have no impact on the rest of the circuitry. On the other hand, if a large spike hits the line, the varistor will swing into the vertical region, show a much reduced resistance, and act as a shunting path for the spike's current. It will effectively clamp the voltage to some maximum rated value. Of course, the varistor has to absorb the energy presented by that spike, and important parameters of a varistor include the amount of energy that it can absorb (in joules) and its maximum current capacity, along with the maximum clamping voltage.

Strain gauge

The strain gauge is device used to measure mechanical strain on some item. Strain occurs when an item is under compression (its length gets shortened or squished) or when it's under tension (it gets elongated or stretched). Both of these things can happen simultaneously, for example when a bar is experiencing bending or torque (one side is under compression while the other is under tension). Engineering strain is defined as the change in length over the initial length. If an item experiences too much strain, it can become permanently deformed or fail (for example, the landing gear in an aircraft or the suspension components in a car). Simply put, strain gauges are used to measure this effect.

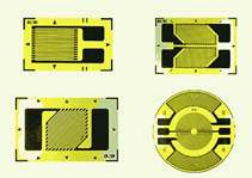


Figure 2.7.20 : Strain guages. Courtesy of Zemic

A strain gauge is made typically of very thin metal foil in a specific pattern. See Figure 2.7.20 for example shapes. Typically, the shape is that of a series of fine lines connected in a back-and-forth pattern, the two ends terminating into larger pads for soldering on connecting leads. In operation, the strain gauge is glued to the material being investigated, for example, a metal bar that is part of a suspension system. As they are glued together, the strain gauge experiences the same deformation as the metal bar. Any deformation will create changes in the length of the strain gauge's foil wires as well as their frontal surface area. For example, under tension, the length increases while the surface area decreases (the surface area must decrease because the foil wire has finite mass). Recalling the basic resistance relation, Equation 1.7.2, both of these effects will cause the resistance to rise. The greater the strain under tension, the greater the rise in resistance. The opposite will occur under compression and the resistance will decrease. These changes in resistance are not large but they are sufficient to alter the associated voltage or current which can then be calibrated to determine the applied strain.

This page titled 1.7: Resistance and Conductance is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.7: Resistance and Conductance** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.8: Instrumentation and Laboratory

At this point we need to shift gears and focus on a few practical aspects, the sort of things we might deal with in an electrical laboratory. It is one thing to discuss abstract concepts such as voltage and current, and quite another to deal with them in the real world.

Sources

For starters, let's consider the idea of power sources, that is, devices that can produce reliable, stable and constant voltages and/or currents. As mentioned previously, a battery approximates an ideal voltage source. The problem, of course, is that batteries only last a finite period of time and their voltage begins to sag after being used for a while. Also, their voltage is fixed and not adjustable. An adjustable voltage source would be much more flexible in a laboratory.

To address this problem, most electrical labs use a variable DC voltage supply in place of simple batteries. These sources typically are adjustable from 0 to 25 volts or higher and can produce one amp of current or more. An example is shown in Figure 2.8.1 . This particular unit has two main outputs plus a third auxiliary output. All outputs are independent, meaning that the voltage of each can be set independent of the others. Further, the two main outputs offer programmable current limiting. This feature can be thought of as a programmable fuse or circuit breaker: it limits the amount of current a circuit can draw to some preselected level. It is worth noting that even if a source is rated for, say, two amps, that only refers to the maximum value that can be drawn; that is not the current that it would produce by default.



Figure 2.8.1 : A multi-output adjustable DC voltage supply.

Multiple output power supplies are usually wired in a “floating” form, meaning that the negative terminal is not tied back to earth ground. A separate connector is presented (usually green) that is tied back to earth ground if it is needed. Having floating supplies gives the user much more flexibility. For example, two floating supplies can be wired in sequence to create a single higher voltage. They can also be wired to present a positive voltage and a negative voltage. The unit shown in Figure 2.8.1 also features twin LED displays for voltage and current for each of the main outputs. It is worthwhile to point out that the color coding used for electronic equipment is not the same as that used for residential wiring in North America. The electronics standard is that the common or ground terminal is black while the “hot” or positive terminal is red.

While there are a large number of makes and models of affordable laboratory DC voltage sources available, the same is not true of DC current sources. This is not usually a problem. As we shall see in upcoming work, it is possible to emulate a constant current source using a voltage source and some attached components.

Schematic Symbols

Clearly, it would not be practical to create circuit drawings (schematics) using pictures of the actual devices. Instead, we use simple schematic symbols to represent them. There are two widely used schemes: ANSI (American National Standards Institue) and IEC (International Electrotechnical Commission). For many electronic components the ANSI and IEC versions are the same, but there are notable differences. ANSI tends to dominate in North America (surprise!) while IEC tends to dominate elsewhere.

Figure 2.8.2 shows the symbols for DC current and voltage sources. The long bar at the top of the voltage source is the positive terminal while the shorter bar at the bottom denotes the negative terminal. For the current source, the arrow points in the direction of conventional current flow. If the source is adjustable, it will often be shown with a diagonal arrow drawn through it. In some schematics, a DC voltage source will simply be drawn as a “connection dot” or node with the voltage labeled. It is assumed that the other end of the source is tied back to the system common or ground.

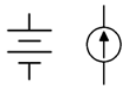


Figure 2.8.2 : Voltage (left) and current source schematic symbols.

Speaking of ground, there are three different symbols used for a circuit's common connection point. These are: earth ground, chassis ground and signal or digital ground. These are shown in Figure 2.8.3 . Earth ground is used when the circuit common is tied back to true earth (e.g., to the third pin on an AC receptacle). Chassis ground is a more generic term which would represent a common reference point that did not go back to true earth (e.g., a portable device). Finally, digital or signal ground is used to distinguish between common points where they might be separated for noise or interference concerns (e.g., a sensitive analog signal common separate from a digital logic common).

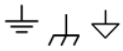


Figure 2.8.3 : Schematic symbols for ground. (L-R) earth, chassis, signal.

The symbols for resistors and other resistive devices vary considerably between ANSI and IEC versions. ANSI uses a zig-zag line while IEC uses a simple rectangle. These are illustrated in Figure 2.8.4 . This text will focus on using the ANSI standard symbols for resistors. One final note about component values: As decimal points are easy to lose on photocopies or small display screens, engineering notation multipliers are sometimes placed in the position of a decimal point. For example, a 4.7 k Ω resistor may be listed as simply 4k7.

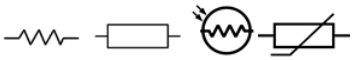


Figure 2.8.4 : Schematic symbols for resistive devices. (L-R) resistor (ANSI), resistor (IEC), photoresistor, thermistor.

Measurement – The Digital Multimeter

Perhaps the most handy measurement device in the electrical laboratory is the digital multimeter, or DMM for short. These handheld devices are used to measure voltage, current and resistance; and depending on model may have other measurement capabilities as well. A pair of typical units are shown in Figure 2.8.5 .



Figure 2.8.5 : Digital Multimeters.

Beyond the measurement functions of the DMM, perhaps the single most important characteristic is its accuracy. It is important to remember that no measurement device is perfect and there is always some uncertainly between what the device indicates and the true value of the parameter being measured. As a DMM uses a digital display, it has both finite range and resolution. As you might guess, the more digits the meter can display, the greater its potential accuracy. DMMs are commonly described as having 3 1/2 digits, 3 3/4 digits, 4 1/2 digits, and so on for their displays. A fractional digit is simply a leading digit that cannot go up to nine. By common use, the term “1/2 digit” means that the leading digit can be no more than one. A 3/4 digit specification typically means that the first digit can't be larger than three (it might also be four or even five as this terminology is not standardized)[1] . To clarify, a 3 1/2 digit display is also referred to as a “2000 count” display. This is because there are 2000 possible values, from 0000 to 1999. Similarly, a 4 3/4 digit display is called a 40,000 count display because it has 40,000 possible values, from 00,000 to 39,000.

Using a 2000 count display as an example, the next question to ask is where to place the decimal point. This display could be set up to read from 0 to 1999 volts. For convenience, we would call that a “2000 volt scale”, meaning that the maximum voltage that can be displayed is approximately 2000 volts. With an otherwise perfect meter, the error can be as large as 1 volt because we have no

way of indicating fractions of volts. To solve this limitation we could add other scales by shifting the decimal point. For instance, we could make a nominal “20 volt scale” which would range up to 19.99 volts, giving us resolution down to 10 millivolts. We could go further and make a 2 volt scale with a maximum of 1.999 volts and 1 millivolt resolution as well as a 200 millivolt scale with a maximum of 199.9 millivolts yielding a resolution of one-tenth millivolt. We might need to measure very large as well as very small voltages, so the scale setting is user adjustable.

It is important to use the lowest scale that can display the desired value otherwise a loss of resolution and accuracy will occur. For example, if we need to measure a voltage somewhere around five or six volts with this 2000 count meter, it should be set for the 20 volt scale. A lower setting, such as a 2 volt or 200 millivolt scale, will not be able to display the measurement and instead will indicate an overload condition (usually by flashing an abbreviated error message such as “Err” or “OL”). On the other hand, if the higher 2000 volt scale is used, the meter will only be able to resolve the measurement to the nearest volt. Indeed, this is so important that some meters have auto-ranging, meaning that they will automatically choose the scale setting to give the best result. The accuracy specification for a DMM is in two parts. The first part is the percent deviation around the measured value. The second part is the added number of counts.

The accuracy specification for a typical 3 1/2 digit meter might be $\pm$ 2% of reading plus 3 counts. To determine the range of possible values for a reading, first determine the percentage of the reading and then add in the number of counts (i.e., one count represents the resolution of the meter on that particular scale). The resulting value represents the uncertainty of the reading. It can also be visualized as an “error envelope” surrounding the displayed value. Somewhere in that envelope is the true value measured. This is shown in the following examples.

Example 2.8.1

A 3 1/2 digit (2000 count) DMM has an accuracy specification of $\pm$ 2% of reading plus 3 counts. On its 20 volt scale it measures 5.01 volts. Determine the uncertainty in this measurement.

On its 20 volt scale this meter can display up to 19.99 volts. Therefore its resolution, or single count value, is 0.01 volts. Thus, 3 counts is 0.03 volts. To this we add 2% of the reading of 5.01 volts, or 0.1002 volts. The total is 0.1302 volts or 130.2 millivolts. This represents the error envelope on either side of the reading. That is, the true value is within the range of 5.01 volts $\pm$ 130.2 millivolts, or somewhere between 4.8798 volts and 5.1402 volts. In other words, the ambiguity is about 130 millivolts out of about 5 volts.

Example 2.8.2

A 4 3/4 digit (40,000 count) DMM has an accuracy specification of $\pm$ 0.1% of reading plus 8 counts. On its 4 volt scale it measures 3.0035 volts. Determine the uncertainty in this measurement.

On its 4 volt scale this meter can display up to 3.9999 volts. Therefore its resolution is 0.0001 volts or 0.1 millivolts. 8 counts is 0.8 millivolts. To this we add 0.1% of the reading of 3.0035 volts, or 3.0035 millivolts. The total is 3.8035 millivolts. This represents the error envelope on either side of the reading and the true value is somewhere between 2.9996965 volts and 3.0073035 volts. The ambiguity here is just a few millivolts out of about 3 volts. Clearly, this meter's level of ambiguity is much reduced compared to the meter used in Example 2.8.1 .

References

[1]You might rightly ask how it is that having just one numeral out of nine counts as “half” and three out of nine counts as “three fourths”. Well, that's marketing for you.

This page titled 1.8: Instrumentation and Laboratory is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.8: Instrumentation and Laboratory** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.9: Summary

In this chapter we have examined the basic quantities that make up an electrical circuit. Charge, measured in coulombs, is a characteristic of subatomic particles; protons being positively charged and electrons being negatively charged. Like charges repel and opposite charges attract. The Bohr model of an atom shows that electrons are contained in energy shells and a single electron in the outermost shell is only loosely bound to an atom. With an applied external energy source these electrons (and thus their charge) can be moved from place to place. The rate of charge movement over time is called current and is measured in amps. Energy is defined as the ability to do work and is measured in joules. Voltage is defined as the energy required to move a charge divided by that charge, with units of volts. Power is the rate of energy usage over time and is measured in watts.

Some materials allow the easy flow of current and are called conductors. In contrast, materials that inhibit the flow of electrical current are called insulators. The measure of this inhibition is called resistance and has units of ohms. Its reciprocal is called conductance and is measured in siemens.

Resistors are devices designed to restrict the flow of current. They are available in a wide range of resistive values, power ratings, and physical sizes and configurations. Common resistors use a series of colored stripes, or color code, to signify their resistance value and tolerance.

Efficiency is the ratio of useful power output to applied power input, normally expressed as a percentage. The higher a system's efficiency, the less energy it will use, the less it will cost to run, and generally the less heat it will generate. If the system is battery powered, higher efficiency leads to longer battery life. Battery capacity is measured in amp-hours. All other factors being equal, the higher the amp-hour rating, the longer the battery will last.

The digital multimeter, or DMM, is a versatile tool for measuring voltage, current, resistance, and possibly other circuit or electrical device parameters. The accuracy of a DMM is dependent on its resolution which is in turn set by the number of digits used, or its total count. The count specification is added to a percent tolerance to arrive at a worst case error value for some specific reading.

Review Questions

1. Describe the Bohr atomic model.
2. How does charge relate to current and voltage?
3. What is the relationship between energy and power?
4. What is the relationship between resistance and conductance?
5. Define efficiency. How does it relate to operating cost?
6. Define count in terms of a digital multimeter accuracy specification.
7. Describe the resistor color code and its use.

This page titled 1.9: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.9: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

1.10:Exercises

Analysis

1. A 2000 count digital multimeter has an accuracy specification of $\pm 1\%$. If it reads a voltage of 34.2 volts, what is the possible range of the true voltage?

▼ Answer 1

33.858V to 34.542V

2. Determine the expected voltage scales for a standard “three and a half” digit meter between 1 and 1000 volts.

▼ Answer 2

2 Volt, 20Volt, and 200 Volt scales

4. Determine the expected current scales for a standard “three and a half” digit meter between 1 mA and 1 amp.

5. A 2000 count digital multimeter has an accuracy specification of $\pm 2\%$ and ± 5 counts. If it reads a voltage of 7.00 volts, what is the possible range of the true voltage?

▼ Answer 5

6.81V to 7.19V

6. A 50000 count digital multimeter has an accuracy specification of $\pm 0.1\%$ and ± 4 counts. If it reads a voltage of 7.00 volts, what is the possible range of the true voltage?

7. What is the charge in coulombs of a million million (10^{12}) electrons?

▼ Answer 7

$1.602 \times 10^{-7} Coulombs$

8. What is the charge in coulombs of 10^{15} electrons?

9. How many electrons would be needed for a charge of 20 coulombs?

▼ Answer 9

$1.248 \times 10^{20} electrons$

10. How many electrons would be needed for a charge of 1 microcoulomb?

11. If a charge of 2 coulombs passes through a wire in 5 seconds, what is the current?

▼ Answer 11

0.4 Amps

12. If a charge of 300 millicoulombs passes through a wire in 0.1 seconds, what is the current?

13. How much charge must be transferred in 0.1 seconds in order to achieve a current of 5 amps?

▼ Answer 13

0.5 Coulombs

14. How much charge must be transferred in 20 seconds in order to achieve a current of 10 microamps?

15. Determine the resulting voltage if it takes 2 joules to move 10 coulombs of charge.

▼ Answer 15

0.2 Volts

16. Determine the voltage if 15 joules is used to move 0.5 coulombs of charge.

17. How much energy is required to create a 10 volt potential difference with a 2 coulomb charge?

▼ Answer 17

20 Joules

18. How much energy is required to create a 50 millivolt potential difference with a 0.1 coulomb charge?

19. What is the wattage equivalent of two horsepower (2 hp)?

▼ Answer 19

1492 Watts

20. What is the horsepower equivalent of 1000 watts?

21. If a device draws 2 amps of current from a 12 volt battery, determine the power delivered.

▼ Answer 21

24 Watts

22. If a device draws 10 milliamps of current from a 1.5 volt battery, determine the power delivered.

23. A 2 hp motor draws 1800 watts from its source. Determine its efficiency.

▼ Answer 23

82.9%

24. A 5 hp motor draws 4.5 kw from its source. Determine its efficiency.

25. An audio power amplifier is rated for 500 watts of maximum audio output at an efficiency of 80%. Determine the amount of wasted power.

▼ Answer 25

125 Watts

26. A compressor draws 10 amps of current from a 120 volt source. Its rated output is 1 hp. Determine the efficiency.

27. An application requires a battery to deliver 15 mA for at least 200 hours. Determine the required amp-hour rating (assume the battery can supply 100% of its energy).

▼ Answer 27

3Ah

28. Determine the required rating for a battery to deliver 0.8 A for at least 30 hours.

29. A certain 12 volt battery has a rating of 6 Ah. Determine the expected battery life using a 5 mA draw. Assume 100% energy usage.

▼ Answer 29

1200 hours

30. A certain AA battery has a rating of 800 mAh. Determine the expected battery life using a 20 mA draw.

31. Assume that a certain piece of material has a resistance of 80 ohms. Determine the new resistance if the length of the piece is doubled and no other parameters are changed.

▼ Answer 31

160Ω

32. Assume that a certain piece of material has a resistance of 2 k ohms. Determine the new resistance if the width and height of the piece are doubled and no other parameters are changed.

33. Assume that a certain piece of material has a resistance of 4 ohms. Determine the new resistance if the resistivity is doubled and no other parameters are changed.

▼ Answer 33

8Ω

34. Assume that a certain piece of material has a resistance of 10 k ohms. Determine the new resistance if the length, width and height of the piece are all halved.

35. A certain material has a resistivity of 100 ohm-centimeters. Determine the resistance of a piece that is 1 cm wide, 0.5 cm high and 6 cm long.

▼ Answer 35

1200Ω

36. A certain material has a resistivity of 2000 ohm-centimeters. Determine the resistance of a piece that is 2 mm wide, 4 mm high and 10 mm long.

37. A 40 ohm resistor has dimension of 0.4 cm wide by 0.2 cm high by 1 cm long. Determine the resistivity in ohm-centimeters.

▼ Answer 37

$3.2\Omega \cdot cm$

38. A 5000 ohm resistor has dimension of 5 mm wide by 3 mm high by 6 mm long. Determine the resistivity in ohm-centimeters.

39. A resistor with the color code yellow-violet-red-silver has a measured value of 4806 ohms. Is this resistor within tolerance? As a percentage, how far is it from the nominal value?

▼ Answer 39

yes, +2.26%

40. A resistor with the color code orange-orange-yellow-gold has a measured value of 33.9 k ohms. Is this resistor within tolerance? As a percentage, how far is it from the nominal value?

41. A resistor with the color code brown-black-orange-silver has a measured value of 9980 ohms. Is this resistor within tolerance? As a percentage, how far is it from the nominal value?

▼ Answer 41

yes, -0.2%

42. A resistor with the color code green-blue-black-gold has a measured value of 50 ohms. Is this resistor within tolerance? As a percentage, how far is it from the nominal value?

43. Determine the value of the resistors pictured in Figure 1.10.1 (left-to-right: red-black-yellow, blue-gray-orange, red-violet-red-silver).

▼ Answer 43

$200k\Omega, 68k\Omega, 2.7k\Omega$

44. Determine the value of the resistors pictured in Figure 1.10.2 (left-to-right: red-red-orange-gold, brown-black-red-gold, yellow-violet-orange-silver).

45. Determine the value of the resistors pictured in Figure 1.10.3 (left-to-right: orange-orange-green-silver, white-brown-black-silver, orange-white-brown-gold).

▼ Answer 45

$3.3M\Omega, 91\Omega, 390\Omega$

46. Determine the value of the resistors pictured in Figure 1.10.4 (left-to-right: silver-orange-violet-yellow, green-blue-yellow-gold, gold-black-orange-yellow).

47. Determine the maximum and minimum allowed values of the resistors pictured in Figure 1.10.1 .

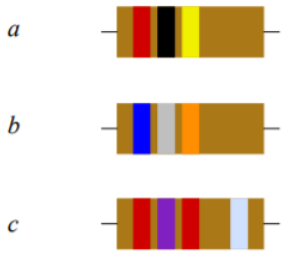


Figure 1.10.1

▼ Answer 47

- 160k to 240k
- 54.4k to 81.6k
- 2.43k to 2.97k

48. Determine the maximum and minimum allowed values of the resistors pictured in Figure 1.10.2 .

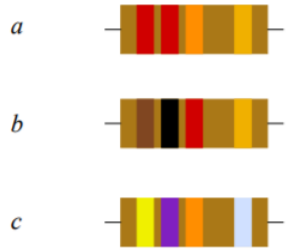


Figure 1.10.2

49. Determine the maximum and minimum allowed values of the resistors pictured in Figure 1.10.3 .

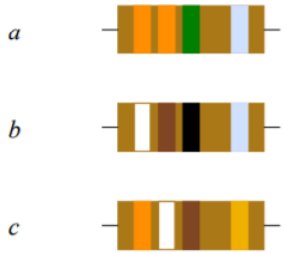


Figure 1.10.3

▼ Answer 49

- 2.97M to 3.63M
- 81.9 to 100.1
- 370.5 to 409.5

50. Determine the maximum and minimum allowed values of the resistors pictured in Figure 1.10.4 .

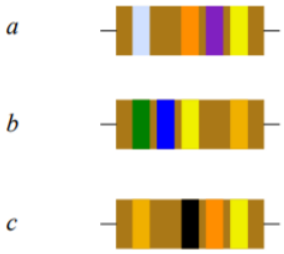


Figure 1.10.4

Design

51. Determine the resistor color code for the following ohmic values using 5% tolerance:

a) 47 Ω

b) 22 kΩ

c) 390 kΩ

d) 2.2 kΩ

e) 560 Ω

▼ Answer 51

a. yellow-violet-black-gold
b. Red-red-orange-gold
c. orange-white-yellow-gold
d. brown-red-oragne-gold
e. violet-green-brown-gold

52. Determine the resistor color code for the following ohmic values using 10% tolerance:

a) 56 Ω

b) 33 kΩ

c) 470 kΩ

d) 1.2 kΩe) 750 Ω

Challenge

53. A radio transmitter is rated for 100 watts of maximum output at an efficiency of 90%. If it is fed from a 120 volt source, determine the current draw.

▼ Answer 53

- Input power: $frac100W0.9 = 111.11W$
- Current: $I = fracPV = frac111.11120 = 0.93A$

54. A certain 75 watt incandescent bulb produces 71 watts worth of heat and the remainder in the form of light. Determine its efficiency as a lighting device and its efficiency as a heating device.

55. Assume it takes an 1800 watt toaster 3 minutes to toast a bagel “just right”. If you toast a bagel every morning for a year and electricity costs 15 cents/kWh, how much will you have spent in that year to toast bagels?

▼ Answer 55

- Minutes/year = $3 \times 365 = 1095\ minutes$
- Hours/year = $1095\ minutes \times \frac{1hour}{60minutes} = 18.25\ hours$
- Energy = $1.8kW \times 18.25\ hours = 32.85kWh$
- Cost = $32.85kWh \times 15\frac{cents}{kWh} + = \$4.93$

56. Assume that you can buy standard 60 watt incandescent light bulbs for 50 cents each and that each has an expected life span of 1000 hours. In comparison, you can buy an LED light bulb that produces the same amount of light but only consumes 7 watts. The LED bulbs cost $5.50 each and have an expected life of 20,000 hours. Assuming electricity costs 14 cents/kWh, determine the total cost of running incandescent lights versus LEDs for 40,000 hours.

57. Given a 1.5 volt battery with a 500 mAh rating, how much current can it produce continuously for 25 hours?.

58. Given a 9 volt battery with a 100 mAh rating, determine the total energy storage in joules.

59. A 2000 count DMM with $\pm 1\%$ accuracy is used to measure the resistance of a spool of AWG 22 wire. If the measurement is 4.50 Ω, determine the length of the wire and the possible inaccuracy (in feet or meters).

60. A length of AWG 26 wire is attached to one end of a length of AWG 32 wire. The total length of the two wires is 200 meters. Determine the length and resistance of each piece if the total resistance is 75 Ω.

This page titled 1.10:Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.10: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

2: Fundamental Laws

2.1: Ohms Law and Kirchhof's Voltage Law

2.1.1: Introduction

2.1.2: Conventional Current Flow and Electron Flow

2.1.3: The Series Connection

2.1.4: Combining Series Components

2.1.5: Ohm's Law

2.1.6: Kirchhoff's Voltage Law

2.1.7: Series Analysis

2.1.8: Potentiometers and Rheostats

2.1.9: Summary

2.1.10: Exercises

2.2: Network Topology- Nodes, Branches and Loops

2.3: Kirchhof's Current Law

2.3.1: Introduction

2.3.2: The Parallel Connection

2.3.3: Combining Parallel Components

2.3.4: Kirchhoff's Current Law

2.3.5: Parallel Analysis

2.3.6: Current Limiting - Fuses and Circuit Breakers

2.3.7: Summary

2.3.8: Exercises

2.4: Series-Parallel Resistive Circuits

2.4.1: Introduction

2.4.2: Series-Parallel Connections

2.4.3: Simplifying Series-Parallel Components

2.4.4: Series-Parallel Analysis

2.4.5: Summary

2.4.6: Exercises

This page titled 2: Fundamental Laws is shared under a not declared license and was authored, remixed, and/or curated by James M. Fiore.

SECTION OVERVIEW

2.1: Ohms Law and Kirchhof's Voltage Law

Learning Objectives

After completing this chapter, you should be able to:

- Describe the differences between conventional current flow and electron flow.
- Identify series resistive circuits that include one or more voltage sources or a single current source.
- Compute equivalent resistance, and component and node voltages for series resistive circuits.
- Compute circulating current and component powers for series resistive circuits.
- Utilize Ohm's law, Kirchhoff's voltage law (KVL) and the voltage divider rule (VDR) to aid in the analysis of series resistive circuits.
- Identify and describe the usage of potentiometers and rheostats.
- Utilize computer simulation tools to investigate and verify basic electric circuit quantities such as component voltages.

2.1.1: Introduction

2.1.2: Conventional Current Flow and Electron Flow

2.1.3: The Series Connection

2.1.4: Combining Series Components

2.1.5: Ohm's Law

2.1.6: Kirchhoff's Voltage Law

2.1.7: Series Analysis

2.1.8: Potentiometers and Rheostats

2.1.9: Summary

2.1.10: Exercises

This page titled 2.1: Ohms Law and Kirchhof's Voltage Law is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

2.1.1: Introduction

With the requisite background information of the first two chapters now behind us, in this chapter we shall begin our analysis of electrical circuits. Here we will introduce the most simple configuration, the series loop. It can contain any number of resistors and voltage sources, or in place of the voltage sources, a single current source. We shall examine how to determine the current flow through each component, the voltage across each component and the power either dissipated or generated by each component. Other practical issues will also be examined, such as the effect of component tolerance on the circuit parameters.

This page titled 2.1.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.2: Conventional Current Flow and Electron Flow

Before we dive into series circuits we need to consider an interesting question involving the direction of current flow. Does it flow from positive to negative or from negative to positive? For that matter, does it even make a difference as far as our analyses will be concerned?

Benjamin Franklin (pictured in Figure 3.2.1) began experimenting with the phenomenon of electricity in 1746. In 1752 he performed his famous kite experiment proving that lightning is a form of electricity by capturing charge from storm clouds in a leyden jar (an early form of an electrical charge storage device)[1]. At this time the modern concept of an atomic model with electrons and protons did not exist and electricity was conceived of as a sort of fluid. Franklin surmised that the “electrical flow” moved from positive to negative. This idea was accepted and became the conventional view. Today we call this idea conventional current flow. In this model, current flows from a more positive voltage to a less positive voltage. We know now that the electron is the charge carrier in metals and the electrons travel in the reverse direction. Essentially, Franklin guessed wrong. Electrons move from a lower potential to a higher potential. We call this model electron flow. For most work, engineers and technicians use conventional flow, although in some cases, such as the explanation of semiconductors, electron flow is easier to visualize for some people. In short, conventional flow exists for historical reasons, and it is the model used for most analyses, including this text.



Figure 3.2.1 : Benjamin Franklin: Technically incorrect but it doesn't really matter.

You might think the current direction would make a big difference in an analysis; after all, it certainly makes a big difference if you drive a car in the wrong direction. It turns out that both forms will achieve the desired results, we just have to be consistent with the usage.

To better understand this, consider that the movement of a net negative charge in one direction can be thought of as a movement of a net positive charge in the other direction. That is, the movement of an electron creates a “hole” where it used to be and that hole is net positive. This is illustrated in Figure 3.2.2 . Here we start at the top with a tube of identical marbles, all pushed to the right. In each step below it we move a marble to the left, mimicking the flow of electrons in a circuit. When we reach the bottom, each marble has been pushed left by one place. We can also arrive at the bottom drawing by simply taking the right-most marble in the top drawing and inserting it to the extreme left by jumping over the other three marbles. Here's the important bit: instead of imagining the marbles moving left, we can also think in terms of “negative space” and imagine the empty slot moving to the right. That's hole flow. The two views are functionally identical as they lead to the same outcome.

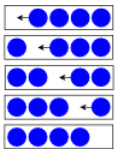


Figure 3.2.2 : Electron versus hole flow.

References

[1]It is worth noting that Franklin's kite was not struck by lightning. If it had been, he likely would have been killed. The hemp string that was used for the kite was sufficiently wet from rain that it was possible to transfer charge from the atmosphere to the leyden jar, and subsequently to a metal key which would emit a spark.

This page titled 2.1.2: Conventional Current Flow and Electron Flow is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.2: Conventional Current Flow and Electron Flow** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.3: The Series Connection

The word circuit comes from the Latin root circ, meaning “ring” or “around”. An electrical circuit consists of at least one ring or loop through which current flows. For example, if we have a battery attached to a lamp as in Figure 3.3.1 , the current exits the battery, flows through the lamp, and then returns to the other side of the battery creating a loop or completed circuit. Without a path back to the battery, current will not flow. Thus, if we cut one of the wires connecting the battery and lamp, there is no path for current and no current flows. This is referred to as an open circuit and is a common fault that occurs when electronic systems are dropped or struck forcefully. Obviously, this will tend to render the circuit unusable. In this example, the lamp will not illuminate. The opposite of the open circuit is the short circuit. In a short circuit, an unintended alternate path for current flow exists and this also can create a malfunction. In the case of our battery and lamp, a short circuit can occur if a piece of wire or metal accidentally fell across the terminals of the lamp. The current would then have a high conductance (i.e., low resistance) path around the lamp. The vast majority of the current would take this low resistance path instead of the higher resistance path presented by the lamp. The result would be that the lamp would not light. In either the open or short case, the light does not function but there is an important difference: for the short circuit, excessive current will flow out of the battery because there is little to resist the flow of current. Thus, the battery will be drained very quickly. For the open, no current flows and thus the battery is not drained.

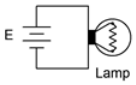


Figure 3.3.1 : Battery and lamp circuit.

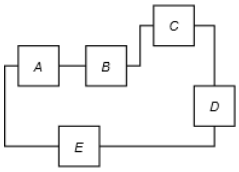
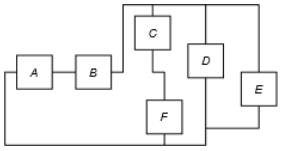


Figure 3.3.2 : A generic series configuration.

Series connections are not limited to just two components. In general, a series connection is any connection of components configured such that the current through each component must be the same. This is illustrated in Figure 3.3.2 . Inside each of the lettered boxes would be a component such as a resistor or a voltage source. Note that for each component, there is one entrance point and one exit point. No matter which box you pick, the current flowing through it must be the same as the current flowing into the next box or out of the preceding box, and it doesn't matter if you follow this path in a clockwise or counterclockwise fashion. Thus, this entire configuration is a series connection. The idea that current is consistent throughout should be selfevident. After all, the only way the current through, say, item B could be different from that flowing through item C or D is if some of it somehow “disappeared” along the way by magic. It is important to remember that consistent current is the hallmark feature defining a series connection:

$$\text{The current is the same everywhere in a series connection.} \tag{2.1.3.1}$$

It is possible that only a portion of a circuit exhibits a series connection. Consider the more complex diagram presented in Figure 3.3.3 . Some of these items are in series and some are not. For example, items A and B are in series with each other but not in series with the remaining items. Why? Because if we imagine the current flowing through A and then through B they must be the same, however, once beyond B, the current could split and flow down other branches: a portion entering C, a portion entering D and the remainder flowing into E. On the other hand, items C and F are in series with each other because whatever current is flowing through one of them must be flowing through the other. Thus, items A and B are in series with each other and items C and F are in series with each other, although all four are not in series as a group.



3.3.3 : A more complex configuration.

It is possible that no two items in a circuit are strictly in series. We will see examples of this in upcoming chapters. Further, just because the currents through two items happen to be the same, that does not necessarily mean they are in series. Identical currents could be just a by-product of the component values chosen. For example, even if items D and E in Figure 3.3.3 have the same numeric value for current, we would not say that they are in series, anymore than we would say that any two people with the same last name would have to be siblings.

This page titled 2.1.3: The Series Connection is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.3: The Series Connection** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.4: Combining Series Components

Typically, a series connection will include multiple resistors. Referring back to the resistance equation presented in Chapter 2, Equation 2.7.2, we can see that resistors in series add.

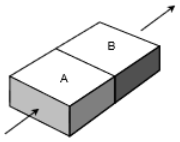


Figure 3.4.1 : Resistors in series.

$$R = \frac{\rho l}{A}$$

If we consider two identical resistors placed in series, one after the other as in Figure 3.4.1 , the effective length would double while keeping the resistivity and area unchanged. The combined result would be a doubling of the resistance of just one of them. If we then generalize this to two arbitrary resistors of identical resistivity and area, the lengths would dictate the resistance of each, and the combined lengths would then reflect the resistance of the pair. We can generalize this further for N resistors. Thus we find that the equivalent resistance of a group of series resistors is their sum:

$$R_{Total} = R_1 + R_2 + R_3 + \cdots + R_N \tag{2.1.4.1}$$

Consequently, as resistors in series add, total resistance may be found by summing the individual resistors.

Example 3.4.1

A string of resistors is placed in series as shown in Figure 3.4.2 . Their values are: 120 Ω, 390 Ω, 560 Ω and 470 Ω. Determine the equivalent series value.

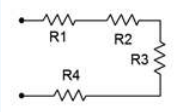


Figure 3.4.2 : Resistor string for Example 3.4.1 .

$$R_T = R_1 + R_2 + R_3 + R_4$$

$$R_T = 120\Omega + 390\Omega + 560\Omega + 470\Omega$$

$$R_T = 1540\Omega$$

Multiple voltage sources in series may also be added, however, polarities must be considered as opposing voltages partially cancel each other (i.e., adding a negative). This concept is presented in the next example.

Example 3.4.2

Determine the equivalent series value of the voltage sources presented in Figure 3.4.3 .

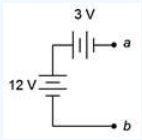


Figure 3.4.3 : Voltage sources in series.

If we use point b as our reference, by inspection the top of the 12 volt source is 12 volts above point b (reminder, the long bar denotes the positive terminal). Also, by inspection, the right side of the 3 volt source (point a) is negative with respect to its left side. As the left side of this source is connected to the positive terminal of the 12 volt source, then it too must be 12 volts above

point b. As its right side is 3 volts less than this side, point a must be 3 volts less than 12 volts, or 9 volts above point b. Thus V_{ab} = 9 volts.

Chasing this further, if the 3 volt source had been flipped so that it had the same polarity as the 12 volt source (positive toward the right) then V_{ab} = 15 volts. If the 12 volt source had been flipped (positive toward bottom) with the 3 volt source as drawn originally, then V_{ab} = −15 volts. If both sources had been flipped then V_{ab} = −9 volts. Finally, if point a had been taken as the reference then these four potentials would have the opposite polarity because, by definition, $V_{ba} = -V_{ab}$.

In contrast to voltage sources, differing current sources are not placed in series as they would each attempt to establish a different series current, a practical impossibility. Refer to Figure 3.4.4 as a reminder!

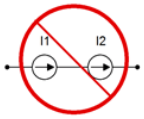


Figure 3.4.4 : Placing current sources in series generally is evil.

This page titled 2.1.4: Combining Series Components is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.4: Combining Series Components** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.5: Ohm's Law

In Chapter 2, the concept of resistance was introduced using the following generality:

$$Effect = \frac{Cause}{Oppostion}$$

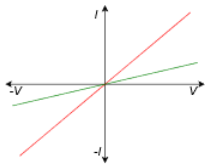


Figure 3.5.1 : Current-voltage plots for simple resistors.

To review, the cause is a voltage source, the opposition is the resistance, and the effect is the resulting current. If the item the current passes through is linear, such as the simple resistors depicted in Figure 3.5.1 , then this relationship can be rewritten as:

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

or more commonly, solved for the voltage and expressed as:

$$V = I \times R \tag{2.1.5.1}$$

This is called Ohm's law, and is named in honor of Georg Ohm, a researcher from the early nineteenth century. It is, along with Kirchhoff's laws that we shall see shortly, one of the most important and useful equations available for the analysis of DC electrical circuits. For the sake of completeness, Ohm's law can also be visualized in terms of resistance:

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

All three forms of this equation are useful. For example, we can use the first version if we have a known voltage applied across a resistor and we want to determine the resulting current. Similarly, Equation 2.1.5.1 can be used to find the voltage drop across a resistor if we know the current through it. Finally, we can use the last version if we need to set the current to a certain value and have to find the resistance required to do so.

Example 3.5.1

A 9 volt battery is used in series to power a 40 Ω resistor as shown in Figure 3.5.2 . Determine the circulating current. Also determine the minimum power rating for the resistor.

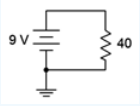


Figure 3.5.2 : Circuit for Example 3.5.1 .

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

$$I = \frac{9V}{40\Omega}$$

$$I = 0.225amps$$

From power law, Equation 2.5.3,

$$P = I \times V$$

$$P = 0.225A \times 9V$$

$$P = 2.025watts$$

Power could also be determined using V^2/R or I^2R. Try it. You should get the same result.

Computer Simulation

To verify the results of the preceding example, the circuit is entered into a simulator, as shown in Figure 3.5.3 . Multisim™ is used here, although any quality circuit simulator will do. Virtual DMMs are used to measure the current and voltage. Remember, current is the rate of charge flow and is measured at a single point; the middle of a connecting wire if you will. Consequently, the ammeter is inserted between the battery and resistor. In contrast, voltage is a potential difference which implies two points for the measurement, and thus, the voltmeter is placed across the resistor.

Note the voltage is exactly 9 volts as expected. The current is just shy of the expected 225 milliamps. This is due to the fact that all real world ammeters exhibit some finite internal resistance. This extra resistance, though very much smaller than the resistor, adds to the total resistance and thus slightly decreases the current. The simulator has been programmed to mimic this behavior and thus we see a very slight decrease in the current in the simulation, which is precisely the effect we would see in a proper physical lab.

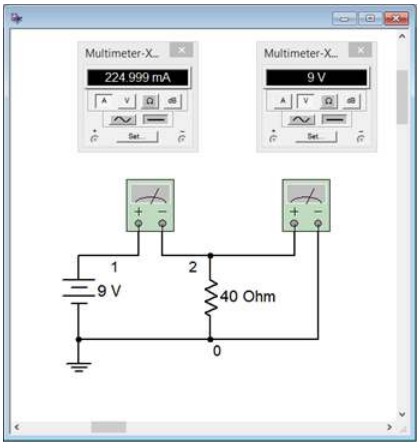


Figure 3.5.3 : The circuit of Example 3.5.1 in a simulator.

This simulation uses virtual instruments because they echo the layout of a physical lab and offer some familiarity. Unfortunately, they become cumbersome in larger circuits. We shall look at other, more direct, simulation techniques later in this chapter.

Example 3.5.2

Refer to the battery and lamp circuit shown back in Figure 3.3.1. Determine the resistance of the lamp if the current flowing through it is 300 mA and the battery voltage is 6 volts. Also determine the power dissipated by the lamp.

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

$$R = \frac{6V}{0.3A}$$

$$R = 20\Omega$$

From power law,

$$P = I^2 \times R$$

$$P = 0.3A^2 \times 20\Omega$$

$$P = 1.8watts$$

Let's continue with a couple of examples utilizing current sources.

Example 3.5.3

Determine the voltage developed across the resistor in the circuit of Figure 3.5.4 . Also determine the power dissipated.

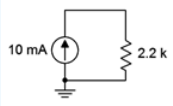


Figure 3.5.4 : Circuit for Example 3.5.3 .

$$V = I \times R$$

$$V = 10mA \times 2.2k\Omega$$

$$V = 22volts$$

From power law,

$$P = I^2 \times R$$

$$P = (10mA)^2 \times 2.2k\Omega$$

$$P = 0.22watts$$

Computational hint: don't forget the “milli” part of the current (milli squared is micro).

Example 3.5.4

Determine the resistor value required in the circuit of Figure 3.5.5 in order for the source to generate 24 watts of power.

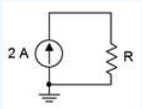


Figure 3.5.5 : Circuit for Example 3.5.4 .

First, note that power generated must equal power dissipated. Thus, the power generated by the source must equal the power dissipated by the resistor. From power law,

$$P = I^2 \times R$$

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

$$R = \frac{24W}{(2A)^2}$$

$$R = 6\Omega$$

As a crosscheck, note that Ohm's law indicates that a 6 Ω resistor would produce a 12 volt drop given a 2 amp current, and that 12 volts times 2 amps yields the expected 24 watts.

It is now time to move onto series circuits using multiple resistors and/or current sources.

This page titled 2.1.5: Ohm's Law is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.5: Ohm's Law** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.6: Kirchhoff's Voltage Law

Along with Ohm's law, the key law governing series circuits is Kirchhoff's voltage law, or KVL. Named after nineteenth century German physicist Gustav Kirchhoff, this law states that the sum of voltage rises and voltage drops around a series loop must equal zero (the rises and drops having opposite polarities). Alternately, it may be reworded as the sum of voltage rises around a series loop must equal the sum of voltage drops. As a pseudo formula:

$$\sum V \uparrow = \sum V \downarrow \tag{2.1.6.1}$$

The Voltage Divider Rule (VDR)

An outgrowth of KVL is the voltage divider rule (VDR). In a series connection, the current is the same through each component. Thus, the voltage drops in a series connection must be directly proportional to the size of the resistances: the larger the resistor, the larger its voltage, and the larger its share of the total voltage applied to the series connection. Thus, the voltage across any resistor must equal the net supplied voltage times the ratio of the resistor of interest to the total resistance:

$$V_{Rx} = E \cdot R_X / R_{TOTAL} \tag{2.1.6.2}$$

In fact, Equation 2.1.6.2 is just a combination of two Ohm's law calculations into a single formula. The circulating current is equal to E/R_{TOTAL}. This current is then multiplied by the resistor of interest (R_X) to arrive at the voltage across that resistor (V_{Rx}). It is worth pointing out that “the resistor of interest” can, in fact, be the sum of multiple resistors in series. While VDR is not required for any particular analysis, it serves two purposes: first, it saves some time because it skips over the computation of current, and second, it reinforces the ideal of a proportional division of voltage in a series connection. For example, if there are two resistors in series and one of them is twice the size of the other, then it must be the case that the larger resistor sees twice the voltage of the smaller resistor.

This page titled 2.1.6: Kirchhoff's Voltage Law is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.6: Kirchhoff's Voltage Law** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.7: Series Analysis

Ohm's law, Kirchhoff's voltage law, the voltage divider rule and series component combinations are the tools we will use to solve general series circuit problems. There are multiple techniques for analyzing these circuits:

- If all voltage source and resistor values are given, the circulating current can be found by dividing the equivalent voltage by the sum of the resistances. Once the current is found, Ohm's law can be used to find the voltage drops across individual resistors. Alternately, the individual voltage drops can be found using the voltage divider rule. At that point, power law can be used to find the power dissipation in each resistor or the power developed by the source(s).
- If the circuit uses a current source instead of a voltage source, then the circulating current is known and the voltage drop across any resistor may be determined directly using Ohm's law.
- If the problem concerns determining resistance values, the basic idea will be to use these rules in reverse. For example, if a resistor value is needed to set a specific current, the total required resistance can be determined from this current and the given voltage supply. The values of the other series resistors can then be subtracted from the total, yielding the required resistor value. Similarly, if the voltages across two resistors are known, as long as one of the resistance values is known, the other resistance can be determined using either the voltage divider rule or Ohm's law.

Often, there will be more than one way to solve a given problem. We shall refer to these as solution paths. No particular path is “more correct” than any other, although some paths may be shorter or easier in a given case, as will be illustrated next.

Example 3.7.1

Determine the voltage V_b (i.e., the voltage across the 100 Ω resistor) in the circuit of Figure 3.7.1 . Also determine the circulating current.

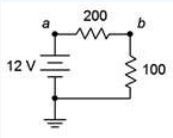


Figure 3.7.1 : Circuit for Example 3.7.1 .

First, note that there is a single voltage source that will create a clockwise current flowing through the two resistors. KVL says that the combined drops across those resistors must equal the rise of the source, or 12 volts. One solution path involves first finding the total resistance. Dividing this into the source voltage will yield the current. Ohm's law can then be used to determine the resistor voltage.

$$R_T = R_1 + R_2$$

$$R_T = 200\Omega + 100\Omega$$

$$R_T = 300\Omega$$

$$I = \frac{E}{R_T}$$

$$I = \frac{12V}{300\Omega}$$

$$I = 40mA$$

$$V_b = I \times R_2$$

$$V_b = 40mA \times 100\Omega$$

$$V_b = 4 \text{ volts}$$

A second solution path involves first finding the resistor voltage via the voltage divider rule. The current can then be found via Ohm's law.

$$V_b = E\frac{R_2}{R_T}$$

$$V_b = 12V\frac{100\Omega}{300\Omega}$$

$$V_b = 4 \text{ volts}$$

$$I = \frac{V_b}{R_2}$$

$$I = \frac{4V}{100\Omega}$$

$$I = 40mA$$

Computer Simulation

Let's verify the preceding example using a computer simulation. Instead of using virtual instruments, we shall use a more direct route, namely a listing of node voltages. A node voltage is the voltage at a given point with respect to some other point, normally ground, as in V_b in the prior example. Note that as we treat connecting wires ideally (having no resistance), a node is the entire connection wire, not a specific location on it. Normally, simulators will automatically number each node in a circuit with ground being node 0. This method skips the entire business of inserting virtual meters in the circuit and reduces schematic clutter.

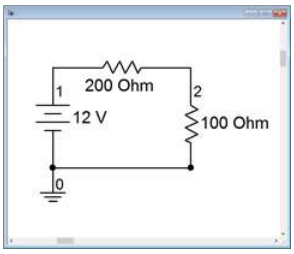


Figure 3.7.2 : The circuit of Example 3.7.1 in the simulator.

Note that ground is node 0, the power supply connection to the first resistor is node 1 and the connection between the two resistors is node 2. Thus, node 2 is V_b. A DC operating point analysis is performed. The results are shown in a separate output window, as depicted in Figure 3.7.3 . The values are as expected and with no loading effects as might be seen with virtual instruments.

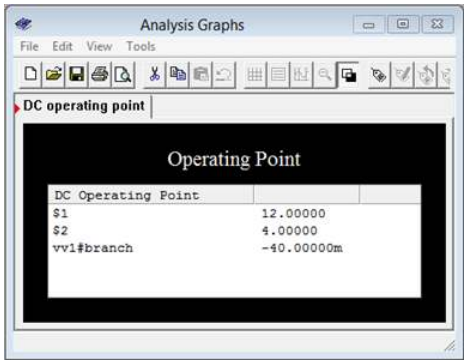


Figure 3.7.3 : Simulation results for the circuit of Figure 3.7.2 .

Example 3.7.2

Determine the voltage V_a (i.e., the voltage across the current source) in the circuit of Figure 3.7.4 .

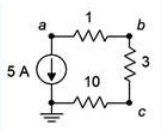


Figure 3.7.4 : Circuit for Example 3.7.2 .

Given the direction of the current source, the current will flow counterclockwise. This produces voltage drops (plus to minus) left to right across the 10 Ω, bottom to top across the 3 Ω, and right to left across the 1 Ω resistor. This means that node a is negative with respect to ground (i.e., the voltage keeps dropping across the resistors as we travel counterclockwise from ground, indicating that the resulting potential keeps getting more and more negative with respect to ground). To find V_a, we may simply sum the resistors and then use Ohm's law to determine their combined voltage drop, which of course, must equal the voltage rise from the current source.

$$R_T = R_1 + R_2 + R_3$$

$$R_T = 1\Omega + 3\Omega + 10\Omega$$

$$R_T = 14\Omega$$

$$V_a = -I \times R_T$$

$$V_a = -5A \times 14\Omega$$

$$V_a = -70 \text{ volts}$$

One practical point when designing circuits is to appreciate the variations in current and voltage produced by component tolerances. For example, in the preceding problem, we determined that there will be 70 volts across the current source, but this assumes that the resistor values are exactly as stated. In the previous chapter we discovered that all resistor values will vary somewhat, and we grade this variation as a percent tolerance. Thus, if the true resistor values are off a bit, we would expect this to impact the associated voltages. Often, we would like to determine worst case values such as a maximum voltage or a minimum current. Sometimes these conditions occur when all resistors are at their maximum or minimum value. For example, the maximum value of source voltage in the previous problem occurs when all of the resistor values are at their maximum. This is not always the case, though. Sometimes the minima and maxima are achieved with odd combinations, such as one resistor being at maximum and another being at minimum. For instance, in the circuit used for Example 3.7.1 , V_b will be at minimum when R_2 is at minimum and when R_1 is at maximum. This can be verified easily using VDR.

Computer Simulation

Along with worst case, we might also like to know the sort of typical spread we get on a particular parameter using normal component variations. This can be performed in a simulator and it is usually called Monte Carlo Analysis or something similar. Basically, the simulator computes a series of simulation runs, each with component values chosen randomly within the specified tolerances of the chosen components, mimicking the effect of pulling the components out of a parts bins. To illustrate the Monte Carlo technique, the circuit of Example 3.7.2 is entered in a simulator as shown in Figure 3.7.5 .

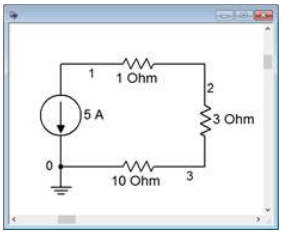


Figure 3.7.5 : The circuit of Example 3.7.2 in the simulator.

The resistors are given 10% tolerances. Five runs are specified for node voltage 1 which is equivalent to V_a in the original circuit. The results are shown in Figure 3.7.6 . Note that the nominal value run indicates −70 volts as expected. The other runs show variations on this voltage, some above and some below the nominal result.

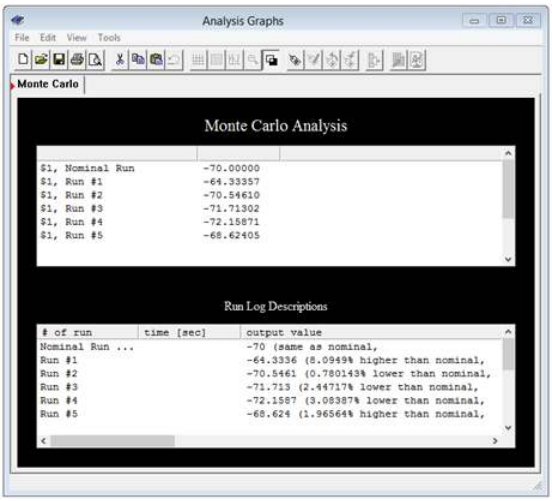


Figure 3.7.6 : Simulation results for the circuit of Figure 3.7.5 .

The Importance of Polarity

A key item of importance when analyzing series circuits, or indeed any electrical circuit, is noting the polarity of the voltages. This was apparent in Example 3.7.2 . Determining polarity for voltage sources is easy as their polarity is fixed (positive is always at the "long bar" and negative at the "short bar"). The polarity of any resistor will depend on the direction of current flow. To avoid confusion we will use the following standard:

- Where conventional current enters a resistor, we label this point with a plus sign, and where the current leaves, a minus sign.
- During analysis, moving from plus to minus is a voltage drop. That is, we are moving from a more positive or higher potential to a less positive or lower potential, thus "dropping" voltage.
- Similarly, traversing from minus to plus is a voltage rise.

Remember, Kirchhoff's voltage law states that the sum of voltage rises around a series loop must equal the sum of voltage drops.

Along with determining the voltage across single resistors, we are often interested in determining the voltage between arbitrary points in a circuit. These may span several components. It is imperative that we know the polarities of the individual voltages in order to determine the voltage between any two circuit points. The foregoing is illustrated using the series circuit schematic of Figure 3.7.7 .

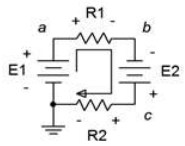


Figure 3.7.7 : A multi-source, multi-resistor series circuit.

Here, the two voltage sources, E_1 and E_2, aid each other because they are both trying to establish a clockwise current. This produces a net voltage of $E_1 + E_2$. Their polarities are fixed. The current direction will be as drawn because conventional current flows out of the positive terminal of a voltage source. The value of this current will be $(E_1 + E_2)/(R_1 + R_2)$ via Ohm's law. Knowing the direction of current, the polarities of the voltage drops across the two resistors may be found. The point where the current enters the resistor is positive, and negative where it exits. Once these are labeled, either Ohm's law or the voltage divider rule can be used to determine the voltages developed across the two resistors. Note that KVL states that the sum of these two resistor voltages must equal the net supplied voltage, or $E_1 + E_2$ in this case.

To determine the voltage from point a to point c, or V_{ac}, we start at point a and sum the voltage drops and rises until we get to point c. Here, the voltage across R_1 shows up as a drop (+ to − and taken as positive) and the voltage across E_2 shows up as a rise (− to +, and thus negative). The result is the voltage across R_1 minus the voltage across E_2. Depending on the specific values, the sum may wind up being either a positive or negative value. If it's positive, this signifies that point a is at a higher potential than is point c.

Conversely, if it's negative, this signifies that point a is at a lower potential than is point c. If we reverse the order, starting at point c and moving to point a, or V_{ca}, we will wind up with the same voltage magnitude but the sign will flip. In the laboratory, the first point letter (the a in V_{ac}) is where the red lead of the voltmeter is connected and the second letter is where the black lead is connected. As a reminder, if a single connection point is used, as in V_a, the second letter is assumed to be ground. Thus, V_a is the voltage from point a to ground.

To illustrate the importance of voltage source polarity, consider the series circuit shown in Figure 3.7.8 . This circuit is identical to the previous circuit with the exception that the polarity of source E_2 has been flipped. This can have a drastic change on the resulting current and voltage drops. For example, if E_2 is larger than E_1, the net supplied voltage will be $E_2 - E_1$ and the direction of conventional current will be as drawn. Effectively, E_2 will be charging E_1. Note that this is opposite to the prior circuit. Because both the direction and the magnitude of the current have changed, the voltage drops and polarities of R_1 and R_2 will change. Consequently, any point-to-point voltage, such as V_{ac}, will also change.

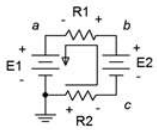


Figure 3.7.8 : Series circuit with altered polarity.

Example 3.7.3

Determine the voltages V_b and V_{ca} in the circuit of Figure 3.7.9 .

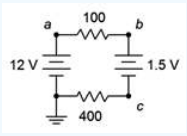


Figure 3.7.9 : Circuit for Example 3.7.3 .

The first step here would be to note that the two sources are in opposition. The result is an effective source of 10.5 volts with the same polarity as the 12 volt source, meaning it creates a clockwise current (rather the opposite of Figure 3.7.8). The total resistance will be the sum of the resistors, or 500 Ω. From this we can compute the current and then the voltage drops across each resistor. The polarity across the 100 Ω will be + to − left to right, and across the 400 Ω it will be + to − right to left. The polarities for both sources are top to bottom + to −. This is shown in Figure 3.7.10 .

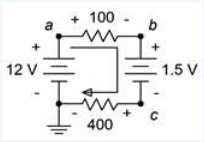


Figure 3.7.10 : Circuit for Example 3.7.3 with current direction and voltage polarities shown.

$$I = \frac{E_{Total}}{R_{Total}}$$

$$I = \frac{10.5V}{500\Omega}$$

$$I = 21mA$$

$$V_{100} = I \times R_{100}$$

$$V_{100} = 21mA \times 100\Omega$$

$$V_{100} = 2.1 \text{ volts}$$

$$V_{400} = I \times R_{400}$$

$$V_{400} = 21mA \times 400\Omega$$

$$V_{400} = 8.4 \text{ volts}$$

To find V_b we start at node b and work our way to ground, summing the potentials as we go. First we see the 1.5 volt source. Even though this is a source, it is still considered a voltage drop because the polarity is + to −. The polarity on the 400 Ω resistor is the same, so we add in its 8.4 volts for 9.9 volts total. That is, node b is 9.9 volts above ground.

For V_{ca} we proceed similarly. Starting at node c we move through the 1.5 volt source, but this time the polarity is − to +, or a rise of 1.5 volts (i.e., taken as negative). Now at node b, we continue through the 100 Ω resistor, also seeing a − to + polarity, or a rise of 2.1 volts. That adds up to 3.6 volts in magnitude, however, the starting node is lower than the ending node, thus V_{ca} = −3.6 volts. We could also say V_{ac} = 3.6 volts.

Back in Chapter 2 the height analogy for voltage was presented. This can be applied nicely in the prior example. You can think of ground as literally ground: the earth under your feet. Voltage rises would be like climbing up a ladder and voltage drops would be like climbing down a ladder. Think of each volt as being analogous to one foot or one meter. Thus, for V_b, you climb up the ladder 8.4 feet for the 400 Ω and another 1.5 feet for the 1.5 volt source, leaving you 9.9 feet (volts) above the ground. Therefore, V_b must be positive with respect to ground by 9.9 volts. This analogy works for voltages below ground as well: just think in terms of digging a hole. For voltages that are not referenced to ground, think of climbing up and down ladders, and then comparing the height of where you stopped to the height of where you started.

Precisely labeling every voltage polarity may seem like unnecessary work, but as circuit complexity increases so does the opportunity for polarity errors. This important step should not be omitted. Consider the following example which is only slightly more complex than the one preceding.

Example 3.7.4

Determine the voltage V_{be} in the circuit of Figure 3.7.11 .

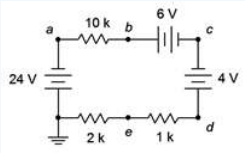


Figure 3.7.11 : Circuit for Example 3.7.4 .

The 24 volt and 6 volt sources are aiding each other while the 4 volt source is in opposition. The result is an effective supplied potential of 26 volts with a current circulating counterclockwise. The current direction and voltage polarities are shown in Figure 3.7.12 .

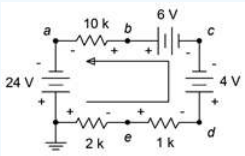


Figure 3.7.12 : Circuit for Example 3.7.4 with current direction and voltage polarities shown.

First, find the circulating current and then find the voltage drops across the resistors of interest. The total resistance is the sum of the three, or 13 kΩ.

$$I = \frac{E_{Total}}{R_{Total}}$$

$$I = \frac{26V}{13k\Omega}$$

$$I = 2mA$$

https://eng.libretexts.org/@go/page/52884

$$V_{10k} = I \times R_1$$

$$V_{10k} = 2mA \times 10k\Omega$$

$$V_{10k} = 20 \text{ volts}$$

Similarly, V_{2k} = 4 volts and V_{1k} = 2 volts with the polarities shown. To determine V_{be}, we can start at node b and move to node e by either going to the left or right side. We shall do both and compare the results.

Going to the left, we see + to − 20 volts across the 10 kΩ, − to + 24 across the left side source, and then + to − 4 volts across the 2 kΩ. This adds up to 0 volts, meaning that nodes b and e are at the same potential! Double checking by going down the right side we see + to − 6 volts for the top source, − to + 4 volts for the right side source, and finally − to + 2 volts for the 1 kΩ. This is also 0 volts, as expected.

It is important to understand that saying V_{be} = 0 volts does not imply that V_b or V_e are 0 volts. In fact, we can see that V_e is below ground by the voltage dropped across the 2 kΩ, or −4 volts. Similarly, V_b is above node a by 20 volts, and node a itself is 24 volts below ground given the polarity of the left side source. The result, once again, is −4 volts.

One more example to better illustrate the voltage divider rule in a larger circuit.

Example 3.7.5

Determine the voltages, V_b, V_d, and V_{bd} in the circuit of Figure 3.7.13 .

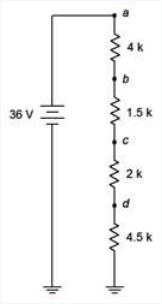


Figure 3.7.13 : Circuit for Example 3.7.5 .

While we could find the circulating current, then use Ohm's law to find the individual voltage drops, and finally add these pieces to find the potentials we're looking for, we shall instead focus on using VDR which may be faster.

The voltage divider rule states that the voltage across any resistor or group of resistors in a series loop is proportional to their resistance compared to the total resistance in that loop. The total resistance in this circuit is 12 kΩ. To find the voltage across any resistor or group of resistors, we simply make a ratio of the resistance of interest to the total resistance and multiply by the applied source voltage. Numbering the resistors R_1 at top to R_4 at bottom:

$$V_d = E\frac{R_4}{R_T}$$

$$V_d = 36V\frac{4.5k\Omega}{12k\Omega}$$

$$V_d = 13.5 \text{ volts}$$

$$V_b = E\frac{R_2 + R_3 + R_4}{R_T}$$

$$V_b = 36V\frac{8k\Omega}{12k\Omega}$$

$$V_b = 24 \text{ volts}$$

$$V_{bd} = E\frac{R_2 + R_3}{R_T}$$

$$V_{bd} = 36V\frac{3.5k\Omega}{12k\Omega}$$

$$V_{bd} = 10.5 \text{ volts}$$

There is another way to find V_{bd} here, and that's to notice that by definition $V_{bd} = V_b - V_d$. Therefore, V_{bd} = 24 volts − 13.5 volts, or 10.5 volts.

This page titled 2.1.7: Series Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.7: Series Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.8: Potentiometers and Rheostats

Some applications require the use of variable resistances. One example is a basic volume control, but other examples include equipment calibration and user adjustable parameters such as the bass, treble and balance controls on a home audio system. Adjustable resistances fall under two categories, potentiometers and rheostats. Generally, these would be the same physical device, with the application and usage determining its name.

A potentiometer is a resistor with three leads. Along with the typical end leads, there is a third lead referred to as the wiper or tap. This third lead is connected to a small metallic arm that makes contact with the resistive material and which can be moved along the length of the material. Potentiometers, or pots for short, often are formed in a circular shape (rotary travel) but they can also be in the form of a straight line (linear travel). Several different types are shown in Figure 3.8.1 . The larger circular units with shafts and the straight unit at the top are designed to be adjusted by the end user (appropriately stylish knobs being attached to said shafts, of course), while the smaller units are intended to be adjusted only by a technician (note the small adjustment screw slots intended to receive fine screwdrivers). The smaller units are designed to be mounted directly on a printed circuit board while the user adjustable units are mounted on a front panel, and have wiring lugs instead of straight leads.



Figure 3.8.1 : A variety of potentiometers.

A cutaway view of a rotary panel mount pot with its back casing removed is shown in Figure 3.8.2 .

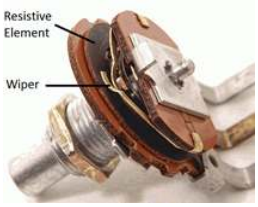


Figure 3.8.2 : Cutaway view of a rotary potentiometer.

Most circular potentiometers utilize a 270° rotation from end to end, also known as “3/4 turn”. Some precision pots used for calibration may use an internal worm gear to move the wiper and achieve 20 or more turns to shuttle the wiper from one end to the other. This results in the ability to make very fine adjustments. The rectangular units in the right-center of Figure 3.8.1 are examples of this type.

The schematic symbol for a basic potentiometer is shown in Figure 3.8.3 . For convenience, its terminals are labeled A and B for the two ends, and W for the wiper arm.



Figure 3.8.3 : Schematic symbol for a potentiometer, with labels.

A pot can be visualized as two resistors in sequence with a tap between them. The total resistance never changes. What does change is the ratio of the two pieces.

$$R_{AB} = R_{AW} + R_{WB} \tag{2.1.8.1}$$

At the extreme left of travel, R_{AW} will be 0 and R_{WB} will be equal to R_{AB}. At the extreme right, R_{AW} will equal R_{AB} and R_{WB} will be 0. If the wiper is precisely in the middle, then $R_{AW} = R_{WB} = R_{AB}/2$, and so on[1].

As a result, potentiometers are very useful as controllable voltage dividers (hence the name). Sometimes it is desirable to control two signals simultaneously from a single knob (e.g., one volume knob for a hi-fi stereo controls both the left and right channels). For this application, multiple resistive elements can be stacked and use a common shaft to control all of the wiper arms in tandem. These are known as ganged potentiometers. The tall, round unit in the left-center of Figure 3.8.1 is a dual gang pot. Note that it has a total of six solder lugs, three for the top element and three for the back element. Finally, unlike fixed resistors, pots are available in limited sizes. Typical values use multiples of 1 or 5 for the total resistance (for example, 1 kΩ, 5 kΩ, 10 kΩ, etc.) although a limited selection of other sizes are also available.

If only the wiper and only one of the two end terminals is used, then the device is referred to as a rheostat. Rheostats are often used to limit current. The value of the rheostat ranges from 0 up to the nominal value of the potentiometer. The schematic symbols for a rheostat are shown in Figure 3.8.4 . The ANSI symbol is shown at the left and the IEC version at the right.

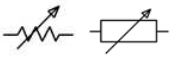


Figure 3.8.4 : Schematic symbols for a rheostat: ANSI (right) and IEC (left).

Time for a couple of examples.

Example 3.8.1

For the circuit depicted in Figure 3.8.5 , determine the maximum and minimum voltages for V_c, as well as its value when the pot is adjusted to midway (assume linear taper).

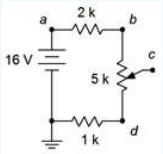


Figure 3.8.5 : Circuit for Example 3.8.1 .

The voltage divider rule states that the voltage across any resistor or group of resistors in a series loop is proportional to their resistance compared to the total resistance in that loop. The total resistance in this circuit is 8 kΩ. The voltage at node c will depend on the setting of the pot. The maximum voltage occurs when the wiper is all the way up, contacting node b.

$$V_c(max) = E\frac{R_i}{R_T}$$

$$V_c(max) = 16V\frac{6k\Omega}{8k\Omega}$$

$$V_c(max) = 12 \text{ volts}$$

For the minimum case, the wiper is set all the way down, contacting node d.

$$V_c(min) = E\frac{R_i}{R_T}$$

$$V_c(min) = 16V\frac{1k\Omega}{8k\Omega}$$

$$V_c(min) = 2 \text{ volts}$$

For the midway case, the pot can be treated as two 2.5 kΩ resistors.

$$V_c(mid) = E\frac{R_i}{R_T}$$

$$V_c(mid) = 16V\frac{3.5k\Omega}{8k\Omega}$$

$$V_c(mid) = 7 \text{ volts}$$

Note that the mid-position achieves a voltage midway between the extremes.

Thus the pot allows a range of voltage adjustment between a minimum of 2 volts and a maximum of 12 volts. The upper and lower fixed resistors serve to set these limits. The larger the values of these resistors compared to the value of the pot, the more narrow the range of adjustment. If these two resistors were excluded, the pot would allow the entire range of adjustment, from ground (0 volts) up to the power supply value of 16 volts.

Example 3.8.2

The potentiometer in the circuit of Figure 3.8.6 is connected as a rheostat. Determine the maximum and minimum current in the circuit.

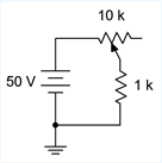


Figure 3.8.6 : Circuit for Example 3.8.2 .

The rheostat will present a resistance value between 0 Ω and 10 kΩ. The maximum current case occurs when the wiper is set to the extreme left, producing 0 Ω.

$$I(max) = \frac{E}{R_T}$$

$$I(max) = \frac{50V}{1k\Omega}$$

$$I(max) = 50mA$$

For the minimum case, the wiper is set all the way to the right, producing the full 10 kΩ.

$$I(min) = \frac{E}{R_T}$$

$$I(min) = \frac{50V}{11k\Omega}$$

$$I(min) = 4.545mA$$

References

[1]This 50/50 division is true if the pot has a linear taper, meaning that the resistance change is proportional to the wiper position. While this is typical, it is possible to have a taper that is not linear for special applications, such as audio taper. In these devices the resistance ratio at mid-position might be something like 10/90 instead of 50/50. This would be desired for a volume control because the human ear responds to changes in sound pressure in a logarithmic fashion. Using a linear taper pot would result in a volume control that is too sensitive at one end and yet produces hardly no change at the other end.

This page titled 2.1.8: Potentiometers and Rheostats is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.8: Potentiometers and Rheostats** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.9: Summary

Electrons are the charge carrier in metals. Their direction of travel, called electron flow, runs from the negative terminal of a voltage source to the positive terminal. The more commonly used conventional current flow runs from the positive terminal to the negative terminal. As conventional current enters a resistor, we mark this point as positive, and where it exits, negative. Thus, any voltage with a + to − polarity is deemed a voltage drop (which dissipates energy in a resistor). A polarity of − to + is deemed a voltage rise.

A series connection is any connection in which the current through one component must be identical to the current flowing through any other component in that connection. In other words, the current exiting any component must be the only current flowing into the next component in line; there can no intervening connections for current flow to or from other parts of the circuit. The equivalent resistance of a string of resistors placed in series is simply the sum of their resistance values. Consequently, the larger resistors dominate the equivalent value.

Ohm's law states that the voltage across a resistor must equal the product of its resistance and the current flowing through it.

Kirchhoff's voltage law (KVL) states that the sum of voltage rises around any series loop must equal the sum of voltage drops around that loop. Alternately, it may be stated as the sum of voltage rises and drops around a series loop must be zero, as the rises and drops have opposite polarity.

The voltage divider rule (VDR) is a convenient computational shortcut based on Ohm's law and KVL. It states that the voltage across any given resistor or group of resistors in a series connection is proportional to their resistance as a percentage of the total resistance. For example, if a resistor makes up one quarter of the total resistance in a series string of resistors, then that resistor will get one quarter of the total voltage applied to that string.

A potentiometer, or pot, is a three terminal variable resistance. The resistance of the entire device is fixed, however, a third lead is attached internally to an adjustable wiper, effectively creating two resistors. The sum of the two parts always equals the nominal value of the pot with the values of each piece being determined by the position of the wiper. Pots may be either rotary or linear (straight line) in motion and are used typically to adjust a voltage level. If the wiper and only one of the end terminals is used, the device is called a rheostat. A common application for a rheostat is the control of current.

Finally, computer simulation tools may be used to build virtual circuits and determine various parameters such as voltages and currents. They may use virtual instruments that echo real world instruments or they may use more direct means, offering results in the form of tables.

Review Questions

1. Describe the differences between electron flow and conventional current flow.

2. Define the term series connection.

3. How is the equivalent resistance for a string of series connected resistors computed? Is the process identical for series connected voltage sources?

4. Why are current sources generally not placed in series?

5. Define Ohm's law.

6. Define Kirchhoff's voltage law.

7. Define the voltage divider rule.

8. What is meant by a Monte Carlo analysis?

9. What is the difference between a potentiometer and a rheostat?

This page titled 2.1.9: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.9: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.1.10: Exercises

Analysis

1. For the circuit of Figure 3.10.1 , determine the circulating current.

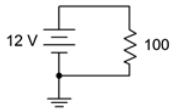


Figure 3.10.1

▼ Answer 1

- 120mA

2. For the circuit of Figure 3.10.2 , determine the circulating current.

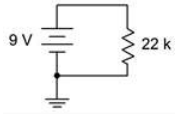


Figure 3.10.2

3. For the circuit of Figure 3.10.1 , determine the power dissipated in the resistor.

▼ Answer 3

- 1.44W

4. For the circuit of Figure 3.10.2 , determine the power dissipated in the resistor.

5. Determine the voltage at the open terminals of Figure 3.10.3 .

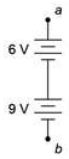


Figure 3.10.3

▼ Answer 5

- 15 V

6. Determine the voltage at the open terminals of Figure 3.10.4 .

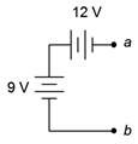


Figure 3.10.4

7. Determine the voltage at the open terminals of Figure 3.10.5 .

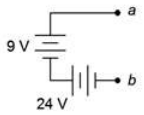


Figure 3.10.5

▼ Answer 7

- 33 V

8. Determine the equivalent resistance of circuit shown in Figure 3.10.6 .



Figure 3.10.6

9. Determine the equivalent resistance of circuit shown in Figure 3.10.7 .



Figure 3.10.7

▼ Answer 9

- $2.6k\Omega$

10. Determine the equivalent resistance of circuit shown in Figure 3.10.8 .



Figure 3.10.8

11. For the circuit of Figure 3.10.9 , determine the circulating current.

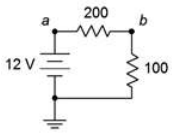


Figure 3.10.9

▼ Answer 11

- 40 mA

12. For the circuit of Figure 3.10.9 , determine the voltages across each resistor and find V_{ab}.

13. Given the circuit of Figure 3.10.9 , determine the power dissipated by each resistor and the power delivered by the source.

▼ Answer 13

- 320mW (200Ω)
- 160mW (100Ω)
- 480mW (source)

14. For the circuit of Figure 3.10.10 , determine the circulating current.

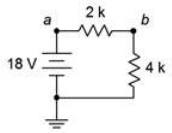


Figure 3.10.10

15. Given the circuit of Figure 3.10.10 , determine the voltages across each resistor and find V_{ba}.

▼ Answer 15

- 6V($2k\Omega$)
- 12V($4k\Omega$)
- -6V (V_{ba})

16. For the circuit of Figure 3.10.10 , determine the power dissipated by each resistor and the power delivered by the source.

17. For the circuit of Figure 3.10.11 , determine the circulating current.

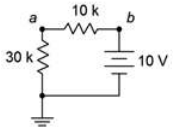


Figure 3.10.11

▼ Answer 17

- 0.25 A

18. For the circuit of Figure 3.10.11 , determine the voltages across each resistor and find V_a.

19. For the circuit of Figure 3.10.12 , determine the circulating current and indicate all voltage polarities.

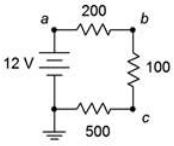


Figure 3.10.12

▼ Answer 19

- I = 15mA
- V_{ab}: + on left, - on right
- V_{bc}: + on top, - on bottom
- V_c: + on right, - on left

- V_{src}: + on top, - on bottom

20. Given the circuit of Figure 3.10.12 , determine the voltages across each resistor and find V_b, V_{bc}, and V_{ca}.

21. For the circuit of Figure 3.10.12 , determine the power delivered by the source.

▼ Answer 21

- P = 180mW

22. For the circuit of Figure 3.10.13 , determine the circulating current and indicate all voltage polarities.

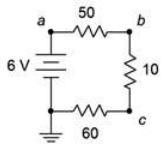


Figure 3.10.13

23. Given the circuit of Figure 3.10.13 , determine the voltages across each resistor and find V_c, V_{ac}, and V_a.

▼ Answer 23

- $V_{50} = 2.5V$
- $V_{10} = 0.5V$
- $V_{60} = 3V$
- $V_c = 3V$
- $V_{ac} = 3V$
- $V_a = 6V$

24. For the circuit of Figure 3.10.13 , determine the power dissipated by the 10 Ω resistor.

25. For the circuit of Figure 3.10.14 , determine the circulating current and indicate all voltage polarities.

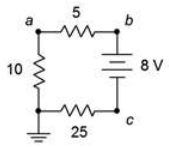


Figure 3.10.14

▼ Answer 25

- I = 200mA
- V_5: + on right, - on left
- V_{10}: + on top, - on bottom
- V_{25}: + on left, - on right
- V_{src}: + on top, - on bottom

26. For the circuit of Figure 3.10.14 , determine the voltages across each resistor and find V_b, V_c, and V_{ca}.

27. Referring to the circuit of 3.10.15 , determine the voltages across each resistor and find V_b, V_c, and V_{ac}.

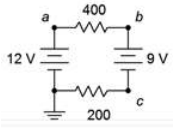


Figure 3.10.15

▼ Answer 27

- $V_{400} = 2V$
- $V_{200} = 1V$
- $V_b = 10V$
- $V_c = 1V$
- $V_{ac} = 11V$

28. Referring to the circuit of Figure 3.10.15 , determine the circulating current and indicate all voltage polarities.

29. Given the circuit of 3.10.16 , determine the voltages across each resistor and find V_b, V_c, and V_{ac}.

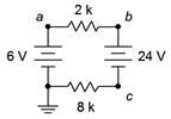


Figure 3.10.16

▼ Answer 29

- $V_{2k} = 3.6V$
- $V_{8k} = 14.4V$
- $V_b = 9.6V$
- $V_c = -14.4V$
- $V_{ac} = 20.4V$

30. Referring to the circuit of Figure 3.10.16 , determine the circulating current and indicate all voltage polarities.

31. Given the circuit of 3.10.17 , determine the circulating current and indicate all voltage polarities.

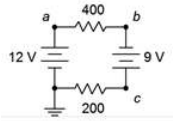


Figure 3.10.17

▼ Answer 31

- I = 35mA
- V_{400}: + on left, - on right
- V_{200}: + on right, - on left
- V_{12Vsrc}: + on top, - on bottom
- V_{9Vsrc}: + on bottom, - on top

32. Referring to the circuit of Figure 3.10.17 , determine the voltages across each resistor and find V_b, V_c, and V_{ac}.

33. Given the circuit of 3.10.18 , determine the circulating current and indicate all voltage polarities.

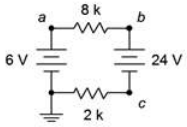


Figure 3.10.18

▼ Answer 33

- I = 1.89mA
- V_{2k}: + on left, - on right
- V_{8k}: + on right, - on left
- V_{6Vsrc}: + on top, - on bottom
- V_{24Vsrc}: + on top, - on bottom

34. Referring to the circuit of Figure 3.10.18 , determine the voltages across each resistor and find V_b, V_c, and V_{ac}.

35. Using the voltage divider rule, determine the voltages V_b, V_c and V_{ac} for the circuit shown in Figure 3.10.19 .

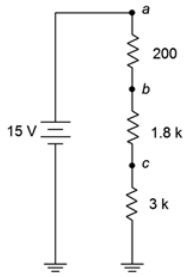


Figure 3.10.19

▼ Answer 35

- $V_b = 14.4V$
- $V_c = 9V$
- $V_{ac} = 6V$

36. Using the voltage divider rule, determine the voltages V_b, V_c and V_{bd} for the circuit shown in Figure 3.10.20 .

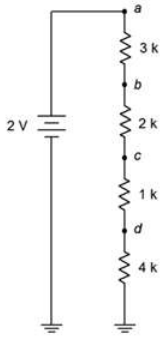


Figure 3.10.20

37. For the circuit of Figure 3.10.20 , determine V_b if the 4 kΩ resistor is accidentally shorted. How does this compare to the original circuit?

▼ Answer 37

- $V_b = 1V$. The 2V is now split across the three remaining resistors

38. For the circuit of Figure 3.10.20 , determine V_b if the 4 kΩ resistor is accidentally opened. How does this compare to the original circuit?

39. Given the circuit shown in Figure 3.10.21 , find the voltage drop across the resistor.

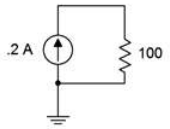


Figure 3.10.21

▼ Answer 39

- 20V

40. Given the circuit shown in Figure 3.10.22 , find the voltage drops across the resistor.

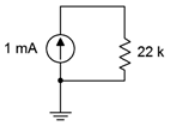


Figure 3.10.22

41. Find the voltage drops across the resistors in the circuit of Figure 3.10.23 .

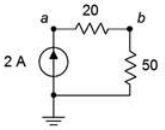


Figure 3.10.23

▼ Answer 41

- $V_{20} = 40V$
- $V_{20} = 100V$

42. Find the voltage drops across the resistors in the circuit of Figure 3.10.24 .

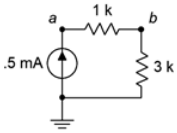


Figure 3.10.24

43. Find the voltage drops across the resistors in the circuit of Figure 3.10.25 .

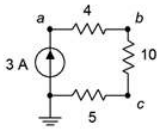


Figure 3.10.25

▼ Answer 43

- $V_4 = 12V$
- $V_{10} = 30V$
- $V_5 = 15V$

44. Find the voltage drops across the resistors in the circuit of Figure 3.10.26 .

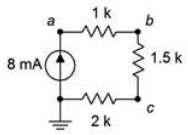


Figure 3.10.26

45. The circuit of Figure 3.10.27 uses a linear taper potentiometer. Determine V_b when the wiper arm is at position a, position b, and at the halfway point.

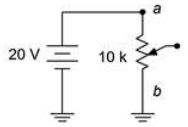


Figure 3.10.27

▼ Answer 45

- $V_b = 0V$. This is true no matter where the wiper arm of the potentiometer is

46. What is the maximum current flowing through the potentiometer of Figure 3.10.27 ? At what position(s) does this occur?

Design

47. Redesign the circuit of Figure 3.10.1 using a new resistor such that the current from the 12 volt battery is 0.1 A.

▼ Answer 47

- Change the 100 Ω resistor to 120 Ω

48. Redesign the circuit of Figure 3.10.2 using a new resistor such that the current from the 9 volt battery is 2 mA.

49. For the circuit of Figure 3.10.6 , find the value of a series voltage source that would generate 1 mA of current if it was connected across the terminals.

▼ Answer 49

- 5.5V

50. For the circuit of Figure 3.10.8 , find the value of a series voltage source that would generate 1 mA of current if it was connected across the terminals.

51. Determine values for the resistors in Figure 3.10.28 such that R_1 is four times the size of R_2 and R_2 is three times the size of R3, with the total resistance equaling 8 kΩ.



Figure 3.10.28

▼ Answer 51

- $R_1 = 6k\Omega$
- $R_2 = 1.5k\Omega$

- $R_3 = 500\Omega$

52. For the circuit shown in Figure 3.10.29 , determine values for R_1 and R_2 such that V_{ab} is 6 volts if E is a 9 volt battery and the total current draw is 20 mA.

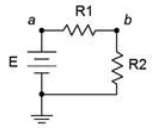


Figure 3.10.29

53. Consider the circuit shown in Figure 3.10.30 . If all resistors have the same value, determine that value if E, a 24 volt source, generates a total power of 10 watts.

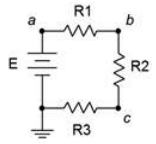


Figure 3.10.30

▼ Answer 53

- 19.2Ω

54. For the circuit shown in Figure 3.10.31 , determine values for R_1 and R_2 such that V_{ab} is 6 volts if I is a 2 mA source and the total voltage drop is 24 volts.

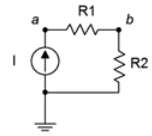


Figure 3.10.31

55. Consider the circuit of Figure 3.10.20 . Is it possible to add a fifth resistor such that the circulating current is 0.1 mA? If so, what is that resistor value?

▼ Answer 55

- Yes, 10 $k\Omega$

56. Consider the circuit of Figure 3.10.20 . Is it possible to add a fifth resistor such that the circulating current is 2 mA? If so, what is that resistor value?

Challenge

57. Assume that two AA cells, E_1 and E_2, rated at 900 mAh each are used to drive a 2 watt lamp as shown in Figure 3.10.32 . Determine the expected life of the batteries.

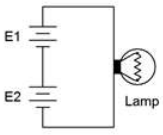


Figure 3.10.32

58. Given the circuit of Figure 3.10.33 , determine the required values of E, R_1, R_2 and R_3 if there is one volt across R_3, the total current draw is 10 mA, the voltage across R_1 is twice the size of voltage across R_2 and the power dissipation in R_2 is 100 mW.

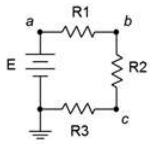


Figure 3.10.33

59. Given the circuit of Figure 3.10.33 , determine the required source voltage if R_1 is 1 kΩ, R_2 is 1 kΩ, the power dissipation in R_1 is 4 mW and the power dissipation in R_3 is 2 mW.

60. Given the circuit of Figure 3.10.34 , determine V_c, V_{db} and V_{ce}.

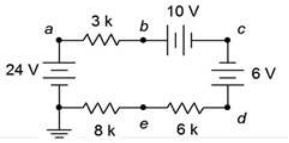


Figure 3.10.34

61. Given the circuit of Figure 3.10.34 , determine V_{ac}, V_{eb} and V_d.

62. Refer to the circuit of Figure 3.10.20 . Assuming each resistor has a 10% tolerance, determine the maximum and minimum values for V_c.

Simulation

63. Simulate the solution of design problem 43 and determine if the values produce the required results.

64. Perform a DC simulation on the circuit of Figure 3.10.13 and find all of the node voltages along with the circulating current.

65. Perform a DC simulation on the circuit of Figure 3.10.14 and find all of the node voltages along with the circulating current.

66. Perform a DC simulation on the circuit of Figure 3.10.26 and find all of the node voltages.

67. Perform a DC simulation on the circuit of Problem 34 and find all of the node voltages.

68. Perform a DC simulation on the circuit of Problem 36 and find all of the node voltages.

69. Simulate the solution of Challenge problem 58 and determine if the values produce the required results.

70. Simulate the circuit of Figure 3.10.34 (Challenge problems 60 and 61) and determine if the node voltages produced match the expected results.

71. Perform a Monte Carlo simulation on the circuit of Figure 3.10.19 . Set each resistor to 5% tolerance and run at least ten variations for V_b to determine a typical spread of values.



https://eng.libretexts.org/@go/page/52887

Source: www.xkcd.com

This page titled 2.1.10: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.10: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.2: Network Topology- Nodes, Branches and Loops

This page titled 2.2: Network Topology- Nodes, Branches and Loops is shared under a not declared license and was authored, remixed, and/or curated by James M. Fiore.

SECTION OVERVIEW

2.3: Kirchhof's Current Law

Learning Objectives

After completing this chapter, you should be able to:

- Identify parallel resistive circuits that include a single voltage source or one or more current sources.
- Compute equivalent resistance of parallel resistive networks.
- Determine the equivalent of multiple parallel current sources.
- Compute component and total current for parallel resistive networks.
- Compute system voltage and component powers for parallel resistive networks.
- Utilize Ohm's law, Kirchhoff's current law (KCL) and the current divider rule (CDR) to aid in the analysis of parallel resistive circuits.
- Describe the function of fuses and circuit breakers.

2.3.1: Introduction

2.3.2: The Parallel Connection

2.3.3: Combining Parallel Components

2.3.4: Kirchhoff's Current Law

2.3.5: Parallel Analysis

2.3.6: Current Limiting - Fuses and Circuit Breakers

2.3.7: Summary

2.3.8: Exercises

This page titled 2.3: Kirchhof's Current Law is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

2.3.1: Introduction

In the preceding chapter we examined series circuits, starting with series configurations of resistors and voltage sources. In this chapter we shall examine the “mirror twin” of series circuits, namely parallel circuits. Again, we shall begin by defining the parallel configuration, and explore how to combine sources and resistors in parallel. From there we shall introduce new laws and rules unique to parallel configurations. For everything said about series circuits, there are corresponding statements regarding parallel circuits. Just as there were specific laws and techniques useful in the series case, such as Kirchhoff's voltage law and the voltage divider rule, there are corresponding laws and techniques useful in the series case (e.g., Kirchhoff's current law and the current divider rule).

A parallel circuit may contain any number of resistors and current sources, or in place of the current sources, a single voltage source. We shall examine how to determine the current flow through each component, the voltage across each component and the power either dissipated or generated by each component. As usual, other practical issues will also be examined along with appropriate computer simulations.

Parallel circuits are in many ways the complement of series circuits. The hallmark of a series connection is that all components in that connection see the same current. Similarly, the most notable characteristic of a parallel connection is that all components see the same voltage. This implies that parallel connections have only two nodes, or two points of connection upon which everything is connected. Just as a series connection can be envisioned as a sort of chain with every element a link in that chain, a parallel connection can be envisioned as a sort of ladder with every element a rung on that ladder.

This page titled 2.3.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.2: The Parallel Connection

Consider the generic layout of Figure 4.2.1 , each component represented by a box.

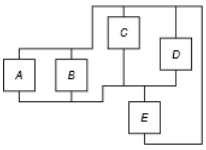


Figure 4.2.1 : A parallel configuration.

Although it may be drawn oddly, there are only two common connection nodes in this configuration: one along the top and the other along the bottom (if you're having difficulty seeing the bottom node, simply slide box E to the right and then rotate it counterclockwise 180 degrees so that its bottom winds up on top). Each of these blocks is like a rung on a ladder. If we were to take a voltmeter and place it across any of these blocks, the meter would read the same voltage across each because the probes are always contacting the same two nodes (remember, ideally the connecting wires have negligible resistance). Thus, we come to the first observation of parallel circuits:

$$\text{In a parallel connection the voltage is the same across each component.} \tag{2.3.2.1}$$

A more complex configuration is shown in Figure 4.2.2 . In this layout some elements are in series and some are in parallel. Only elements D and E are strictly in parallel here as they are the only two that connect to the same two nodes, and thus must see the same voltage. Items C and F might at first appear to be in parallel with them but upon closer inspection it should be noted that C and F are in series with each other. That is, C and F will split the voltage that D or E sees, in accordance with the voltage divider rule. In other words, it is not true that the voltage across C or across F must be the same as the voltage across D or E. The same can be said for items A and B.[1]

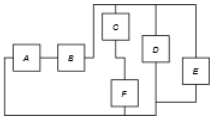


Figure 4.2.2 : A more complex configuration.

References

[1]What can be said is that the series combination of C and F together is in parallel with D which is, in turn, in parallel with E. This is the basic idea behind series-parallel networks, the theme of the next chapter.

This page titled 2.3.2: The Parallel Connection is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.2: The Parallel Connection** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.3: Combining Parallel Components

Our first step is to determine how to combine parallel components in order to create a single equivalent component. Unlike series connections, this can be a little more time consuming.

Sources in Parallel

First, voltage sources are not placed in parallel as a general rule, see Figure 4.3.1 . The reason is because a parallel connection requires the same voltage across each component. This would be impossible to achieve with each source trying to maintain a different voltage across the same two nodes. This may result in damage to the sources (e.g., exploding batteries). The main exception to this rule is if the sources have the same potential and the goal is to extend operational life (e.g., multiple D cell batteries in parallel).

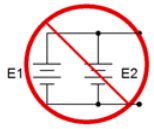


Figure 4.3.1 : Do not place voltage sources in parallel.

When it comes to current sources, they may simply be added together, however, like series voltage sources, polarity is important. Referring to the three parallel current sources in Figure 4.3.2 , the middle and right sources are pumping current into the top node while the left source is feeding the bottom node. You can also think of the left source as draining the top node. Thus, these three sources together behave like a single source of five amps feeding the top node (eight amps in, three amps out).

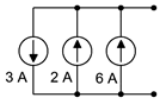


Figure 4.3.2 : Current sources in parallel.

Resistors in Parallel

When placed in parallel, resistor values do not add they way they do in series connections. The reason for this is obvious if we look at the basic resistance equation from Chapter 2, Equation 2.11:

$$R = \frac{\rho l}{A}$$

If we consider two identical resistors placed in parallel, side by side as in Figure 4.3.3 , the effective area would double while keeping the resistivity and length unchanged. The combined result would be a halving of the resistance of just one of them. Thus we see that placing resistors in parallel results in a decrease in net resistance, the opposite of the series case.

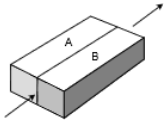


Figure 4.3.3 : Resistors in parallel.

This situation is simplified if we consider conductance instead of resistance. Recalling that conductance, G, is the reciprocal of resistance, we can rewrite the resistance equation:

$$G = \frac{A}{\rho l}$$

Now we can see that the increase in area creates a proportional increase in conductivity. If we generalize this for N resistors we find that the equivalent conductance of a group of parallel resistors is their sum:

$$G_{Total} = G_1 + G_2 + G_3 + \cdots + G_N \tag{2.3.3.1}$$

As we deal normally with component resistance values instead of conductance values, Equation 2.3.3.1 may be rewritten in terms of equivalent resistance:

$$\frac{1}{R_{Parallel}} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \cdots + \frac{1}{R_N} \tag{2.3.3.2}$$
$$R_{Parallel} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \cdots + \frac{1}{R_N}}$$

Among other things, Equation 2.3.3.2 tells us that the equivalent resistance of a group of parallel resistances will always be smaller than the smallest resistance in that group. It also indicates that if we have N identical resistors of value R, the equivalent will be R/N (e.g., 3 parallel 3.6 kΩ resistors are equivalent to a single 1.2 kΩ resistor). As a convenience, it is common to use two parallel lines, $||$, as a shortcut way of saying "in parallel with", as in $R_1||R_2$. Also, $||$ has higher operational precedence than +.

Product-Sum Rule

A handy shortcut for two parallel resistors may be created from Equation 2.3.3.2. We start by writing the version for just two parallel resistors,

$$R_{Parallel} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2}}$$

Multiply the fraction through by $R_1\ R_2$ and simplify:

$$R_{Parallel} = \frac{R_1 R_2}{R_1 + R_2} \tag{2.3.3.3}$$

For obvious reasons, Equation 2.3.3.3 is referred to as the product-sum rule. It is only applicable to two resistors, however, it can be used repeatedly on pairs of resistors within larger groups instead of using Equation 2.3.3.2. For example, a group of four resistors can be reduced to two pairs, and then those two equivalents can be reduced as a third pair. This can be a handy technique for obtaining quick estimates.

We can take the product-sum rule a step further by writing R_2 as a multiple of R_1:

$$R_2 = NR_1$$

Substituting this into Equation 2.3.3.3 yields:

$$R_{Parallel} = \frac{R_1 N R_1}{R_1 + N R_1}$$
$$R_{Parallel} = \frac{N R_1^2}{(N+1) R_1}$$
$$R_{Parallel} = \frac{N}{N+1} R_1 \tag{2.3.3.4}$$

Equation 2.3.3.4 is useful for quick estimates of parallel resistor pairs. For example, if we have a 24 kΩ resistor in parallel with an 8 kΩ resistor, that's a ratio (N) of 3:1. Thus the equivalent will be $N/(N+1)$, or 3/4ths, of the smaller resistor, yielding 6 kΩ. Similarly, a 180 Ω in parallel with a 90 Ω would yield 2/3rds of 90 Ω, or 60 Ω.

Example 4.3.1

A group of resistors is placed in parallel as shown in Figure 4.3.4 . Determine the equivalent parallel value.

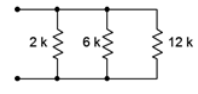


Figure 4.3.4 : Resistor arrangement for Example 4.3.1 .

There are several solution paths. We'll try them all and compare. First, let's try the conductance form:

$$R_{Parallel} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \cdots + \frac{1}{R_N}}$$

$$R_{Parallel} = \frac{1}{\frac{1}{2k\Omega} + \frac{1}{6k\Omega} + \frac{1}{12k\Omega}}$$

$$R_{Parallel} \approx 1.333k\Omega$$

We could also use the product-sum rule twice. We'll use the middle and right-most resistor first (although any pairing would do, but this pairing is convenient):

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

$$R_{23} = \frac{6k\Omega 12k\Omega}{6k\Omega + 12k\Omega}$$

$$R_{23} = 4k\Omega$$

$$R_{Parallel} = \frac{R_1 R_{23}}{R_1 + R_{23}}$$

$$R_{Parallel} = \frac{2k\Omega 4k\Omega}{2k\Omega + 4k\Omega}$$

$$R_{Parallel} \approx 1.333k\Omega$$

Finally, we could use the ratio technique. First, consider the 6 kΩ and 12 kΩ resistor pair. That's a 2:1 ratio, so the result will be 2/3rds of the smaller resistor, and 2/3rds of 6 kΩ is 4 kΩ. The ratio between this and the 2 kΩ resistor is also 2:1, and 2/3rds of 2 kΩ is approximately 1.333 kΩ.

The ratio technique happened to work out well here because of the convenient resistor values with perfect integer ratios. That will not always be the case.

This page titled 2.3.3: Combining Parallel Components is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.3: Combining Parallel Components** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.4: Kirchhoff's Current Law

Just as Kirchhoff's voltage law is a key element in understanding series circuits, Kirchhoff's current law (KCL) is the operative rule for parallel circuits. It states that the sum of all currents entering and exiting a node must sum to zero. Alternately, it can be stated as the sum of currents entering a node must equal the sum of currents exiting that node. As a pseudo formula:

$$\sum I \rightarrow = \sum I \leftarrow \tag{2.3.4.1}$$

Recalling that a node is a connection area wherein the voltage is the same (ignoring the resistance of connecting wires), we see that there are two nodes in a parallel configuration. For each of these nodes, whatever currents flow in must be balanced by the same total current flowing out. For example, referring to the circuit shown in Figure 4.4.1 , there is a current source feeding the top node. At the three-way connection with R_1 and R_2, KCL along with Ohm's law dictate that the incoming current from the source must split with one portion exiting by flowing down through R_1 and the remainder exiting by flowing down through R_2. At the bottom node (ground) these two resistor currents must combine and equal the current that is being drawn back into the current source, which must be the same as the current originally produced by the source. This is perfectly sensible because it would be impossible for the current source to establish two different currents; one at its input and the other at its output. To assert otherwise would be a little like saying you're two different heights depending on whether you measure from your toes up to your head versus from your head down to your toes. Thus, a simplistic way of stating KCL is “What goes in must come out”.[1]

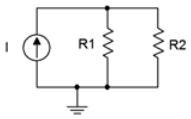


Figure 4.4.1 : A simple parallel circuit.

The Current Divider Rule (CDR)

Just as series circuits follow the voltage divider rule (voltage dividing in proportion to resistance), parallel circuits follow the current divider rule which states that current divides in reverse proportion to resistance (i.e., in direct proportion to conductance). This can be reduced to a simple formula when only two resistors are involved. Consider two parallel resistors, R_1 and R_2, fed by a current, I_{Total}, as in Figure 4.4.1 . First, we note that the voltage across the pair must equal the entering current times the effective resistance of the pair.

$$V = I_{Total} \frac{R_1 R_2}{R_1 + R_2}$$

The current through each of the resistors must be this voltage divided by the corresponding resistance. For the current through R_1:

$$I_{R1} = \frac{V}{R_1}$$

$$I_{R1} = I_{Total} \frac{R_1 R_2}{R_1 (R_1 + R_2)}$$

$$I_{R1} = I_{Total} \frac{R_2}{R_1 + R_2}$$

Similarly, for the current through R_2:

$$I_{R2} = \frac{V}{R_2}$$

$$I_{R2} = I_{Total} \frac{R_1 R_2}{R_2 (R_1 + R_2)}$$

$$I_{R2} = I_{Total} \frac{R_1}{R_1 + R_2}$$

Thus, the current through one of the resistors will equal the total current times the ratio of the opposite resistor over the sum of the two resistors. In general:

$$I_X = I_{Total}\frac{R_Y}{R_X + R_Y} \tag{2.3.4.2}$$

This rule is convenient in that you don't have to compute the parallel equivalent resistance, but remember, it is valid only when there are just two resistors involved. A more general version that can be used for any number of resistors is $I_i = I_{Total} \cdot R_P/R_i$, where R_P is the total equivalent parallel resistance and R_i is the resistor of interest. This is, in essence, merely a rewriting of the fact that all components must see the same voltage: $I_i \cdot R_i = I_{Total} \cdot R_P$. This is not as much of a shortcut as the two resistor version because the parallel resistance still must be calculated.

Example 4.4.1

A simple parallel network is shown in Figure 4.4.2 . Determine the current through each resistor.

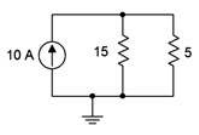


Figure 4.4.2 : Circuit for Example 4.4.1 .

The current divider rule can be used to find the currents through the two resistors. At the top node the total entering current is 10 amps.

$$I_{R1} = I_{Total}\frac{R_2}{R_1 + R_2}$$

$$I_{15} = 10A\frac{5\Omega}{15\Omega + 5\Omega}$$

$$I_{15} = 2.5A$$

$$I_{R2} = I_{Total}\frac{R_1}{R_1 + R_2}$$

$$I_5 = 10A\frac{15\Omega}{15\Omega + 5\Omega}$$

$$I_5 = 7.5A$$

At this point it is worth taking a moment to verify the voltage polarities and current directions. Given the direction of the source, the currents through the two resistors must be flowing from top to bottom. Consequently, the voltage polarities must be + to − from top to bottom. This is illustrated in Figure 4.4.3 . At the top node, entering current is in red with exiting current in blue. Note that KCL is satisfied as 10 amps enters while a total of 10 amps (2.5 amps plus 7.5 amps) exits.

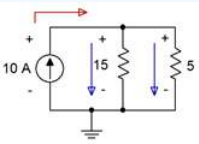


Figure 4.4.3 : Polarities and directions for Example 4.4.1 .

References

[1]For details on this corollary to Fudd's First Law of Opposition, see Firesign Theatre, I Think We're All Bozos On This Bus. Remember, the future can't wait.

This page titled 2.3.4: Kirchhoff's Current Law is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.4: Kirchhoff's Current Law** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.5: Parallel Analysis

Keeping in mind that the voltage across each element in a parallel configuration is constant, Ohm's law dictates that currents divide among parallel resistors in proportion to their conductance (i.e., in inverse proportion to their resistance). As a consequence, Ohm's law, Kirchhoff's current law, the current divider rule and parallel component combinations are the tools we will use to solve general parallel circuit problems. There are multiple techniques for analyzing these circuits:

- If the circuit uses a voltage source and its value along with the resistor values are given, the resistor currents can be found by dividing the source voltage by each resistance. Once these currents are found, KCL can be used to determine the source current. At that point, power law can be used to find the power dissipation in each resistor or the power developed by the source, as needed.
- If the circuit uses current sources, then the total circulating current can be found by combining their values (taking into account the current directions, of course). The individual resistor branch currents then can be found using the current divider rule (repeatedly, if needed). Alternately, the effective parallel resistance may be found first. Multiplying this by the total source current via Ohm's law will determine the system voltage, and from there the individual resistor branch currents may be found using Ohm's law.
- If the problem concerns determining resistance values, the basic idea will be to use these rules in reverse. For example, if a resistor value is needed to set a specific voltage, the equivalent parallel resistance can be determined from this voltage and the given current source. The conductance values of the other parallel resistors can then be subtracted from the total conductance (i.e., the reciprocal of the equivalent parallel resistance), yielding the remaining required conductance value, the reciprocal being the required resistance. Similarly, if the currents through two parallel resistors are known, as long as one of the resistance values is known, the other resistance can be determined using either the current divider rule in reverse (i.e., the currents divide in inverse proportion to the resistances) or via Ohm's law.

Example 4.5.1

A simple parallel network is shown in Figure 4.5.1 . Determine the current through each resistor as well as the total current exiting the voltage source.

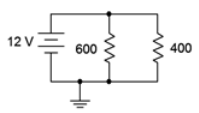


Figure 4.5.1 : Circuit for Example 4.5.1 .

The most direct solution is to use Ohm's law to determine each of the resistor branch currents. KCL can then be used to determine the current flowing from the source. Recalling the voltage is the same across parallel branches:

$$I_{600} = \frac{E}{R_1}$$

$$I_{600} = \frac{12V}{600\Omega}$$

$$I_{600} = 20mA$$

$$I_{400} = \frac{E}{R_2}$$

$$I_{400} = \frac{12V}{400\Omega}$$

$$I_{400} = 30mA$$

KCL dictates that the entering current must equal the sum of the exiting currents, or 50 mA.

An alternate technique would be to determine the parallel resistance and divide this into the source voltage to determine the exiting source current.

$$R_{Parallel} = \frac{R_1 R_2}{R_1 + R_2}$$

$$R_{Parallel} = \frac{400\Omega 600\Omega}{400\Omega + 600\Omega}$$

$$R_{Parallel} = 240\Omega$$

$$I_{Total} = \frac{E}{R_{Parallel}}$$

$$I_{Total} = \frac{12V}{240\Omega}$$

$$I_{Total} = 50mA$$

Now CDR can be used to find the currents through the two resistors.

$$I_{R1} = I_{Total}\frac{R_2}{R_1 + R_2}$$

$$I_{400} = 50mA\frac{600\Omega}{400\Omega + 600\Omega}$$

$$I_{400} = 30mA$$

$$I_{R2} = I_{Total}\frac{R_1}{R_1 + R_2}$$

$$I_{600} = 50mA\frac{400\Omega}{400\Omega + 600\Omega}$$

$$I_{600} = 20mA$$

The voltage polarities and current directions are illustrated in Figure 4.5.2 . The voltage polarity is + to − from top to bottom, as set by the voltage source. With these polarities, the currents through the two resistors must be flowing from top to bottom and the current from the source is flowing right, from the positive terminal. At the top node, entering current is in red with exiting current in blue.

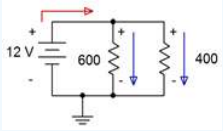


Figure 4.5.2 : Polarities and directions for Example 4.5.1 .

Computer Simulation

The circuit of Example 4.5.1 is entered into a simulator as shown in Figure 4.5.3 . In this example, virtual instruments are used. Three ammeters are inserted in line to measure the current exiting the voltage source as well as the currents flowing through the two resistors. The polarities of the ammeters are set to match those illustrated in Figure 4.5.2 . As a consequence, we expect to see all positive currents.

The result is pretty much as expected. The current through the 600 Ω resistor is precisely 20 mA and the source current is precisely the sum of the two resistor currents. The only wrinkle is that the current through the 400 Ω resistor is very slightly less than expected, 29.999 mA versus 30 mA calculated. This is due to meter loading effects. In Chapter 3 we saw that ammeters present a very low internal resistance, but this cannot always be ignored, especially when measuring in series with resistors having very small values. This slight increase in resistance forces a slight decrease in current due to Ohm's law.

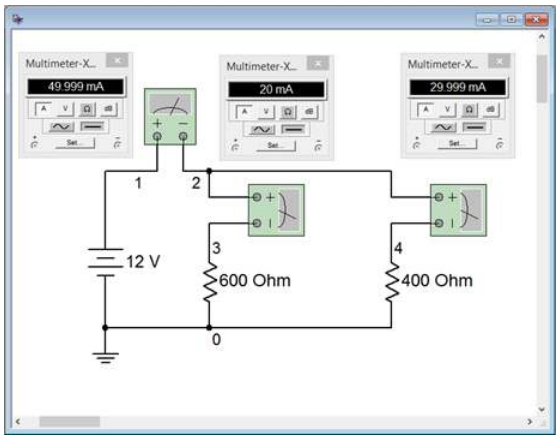


Figure 4.5.3 : The circuit of Example 4.5.1 in a simulator.

It turns out that a similar situation exists for voltmeters. Ideally, voltmeters present a very high internal resistance which, when placed across a resistor, has minimal impact. This effect cannot always be ignored, especially when measuring across resistors having large values, as the current divider rule will come into play. Fortunately, the internal resistance of many virtual instruments is adjustable and can be set to extreme values to minimize any impact on measurements. In the world of physical instruments this is not possible so the internal resistance of any real world meter is something that must always be kept in mind.

Example 4.5.2

A parallel network is shown in Figure 4.5.4 . Determine the current through each resistor.

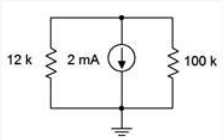


Figure 4.5.4 : Circuit for Example 4.5.2 .

The current divider rule can be used to find the currents through the two resistors. At the bottom node (ground) the total entering current is 2 mA.

$$I_{R1} = I_{Total}\frac{R_2}{R_1 + R_2}$$

$$I_{12k} = 2mA\frac{100k\Omega}{12k\Omega + 100k\Omega}$$

$$I_{12k} \approx 1.7857mA$$

The current through the 100 kΩ can be found via KCL as follows:

$$I_{R2} = I_{Total} - I_{R1}$$

$$I_{100k} = 2mA - 1.7857mA$$

$$I_{100k} \approx 0.2143mA$$

The polarities and directions are shown in Figure 4.5.5 . Note that the currents are flowing up through the resistors producing voltage drops of + to − from ground up to the top. This means that the voltage of the top node is negative with respect to ground.

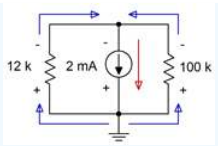


Figure 4.5.5 : Polarities and directions for Example 4.5.2 .

Computer Simulation

In order to get a better handle on the idea of meter loading, the circuit shown in Example 4.5.2 is entered into a simulator. This is shown in Figure 4.5.6 with the simulation using virtual instruments. Using Ohm's law, the voltage drop across the system should be 100 kΩ times its current, or approximately 21.4286 volts if we carry the digits out a little further.

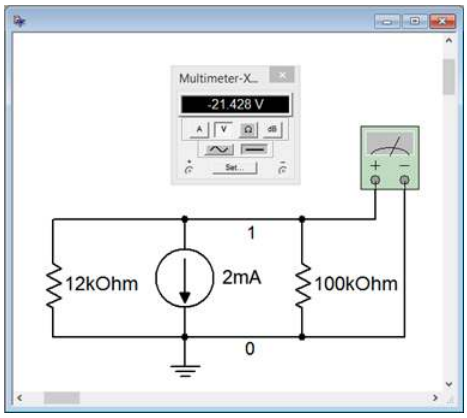


Figure 4.5.6 : The circuit of Example 4.5.2 in a simulator.

The virtual voltmeter has been set for an internal resistance of 1 GΩ. The results are nearly the same as those calculated previously. To test the impact of the voltmeter's internal resistance the simulation is run a second time with the internal resistance set to 1 MΩ, a value seen commonly in general purpose digital multimeters. The result is shown in Figure 4.5.7 . The voltage magnitude has dropped more than 200 mV, entirely due to meter loading.

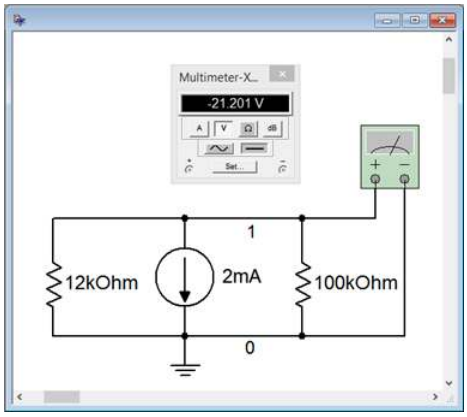


Figure 4.5.7 : Simulation repeated using a voltmeter internal resistance of 1 MΩ.

Depending on component values, the results could be much worse or hardly noticeable at all. As a general rule, make sure the voltmeter's internal resistance is at least 10 times (and preferably, 100 times) larger than any resistance it is placed across in order to avoid loading errors caused by unwanted current division. In the case of an ammeter, its internal resistance should be at least 10 times (and preferably, 100 times) smaller than any resistance with which it is placed in series.

✔ Example 4.5.3

A parallel network is shown in Figure 4.5.8 . Determine the current through the 200 Ω resistor.

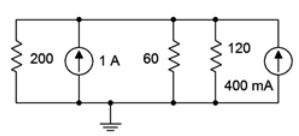


Figure 4.5.8 : Circuit for Example 4.5.3 .

The first step is to simplify the circuit. The two current sources aid each other as they both feed current into the top node. Thus they are equivalent to a single 1.4 amp current source with the same direction. Further, the 60 Ω and 120 Ω resistors are in parallel and can be treated as a single unit. Note this is a 2:1 ratio, and thus the result will be 2/3rds of 60 Ω, or 40 Ω. The basic product-sum rule or conductance formula could also be used.

We have simplified the circuit down to a single 1.4 amp source feeding a 200 Ω resistor in parallel with a 40 Ω resistor. The current divider rule can be used to find the current through the 200 Ω resistor as follows:

$$I_{R1} = I_{Total}\frac{R_2}{R_1 + R_2}$$

$$I_{200} = 1.4A\frac{40\Omega}{200\Omega + 40\Omega}$$

$$I_{200} \approx 233.33mA$$

As a crosscheck, Ohm's law indicates the system voltage must be 200 Ω times 233.33 mA, or approximately 46.667 volts. Via Ohm's law, the current through the 60 Ω must be 46.667 volts divided by 60 Ω, or approximately 777.78 mA. Similarly, the current through the 120 Ω can be determined to be approximately 388.89 mA. Via KCL, these three currents should add up to the total supplied current of 1.4 amps, which they do.

An alternate solution path would be to find the equivalent of the three parallel resistors, or $200||60||120$, which is 33.333 Ω. This is fed by the combined sources yielding 1.4 amps. Ohm's law is then used to find the system voltage of approximately 46.667 volts (1.4 amps times 33.333 Ω). From there Ohm's law is used again to find the current flowing through the 200 Ω (46.667 volts divided by 200 Ω, or approximately 233.33 mA).

Let's continue with the prior example, this time with a design twist.

Example 4.5.4

Using the circuit of Figure 4.5.8 , determine the value of an additional fourth parallel resistor such that the system voltage will drop to 42 volts.

The total source current in this circuit is fixed at 1.4 amps. If the resistance it feeds is reduced, then by Ohm's law, the system voltage must also be reduced. One way to solve this problem is to first determine the required parallel equivalent resistance. Then the conductances of the three known resistors can be subtracted out, leaving the conductance of the new resistor, from which we determine its resistance.

$$R_{Parallel} = \frac{V}{I}$$

$$R_{Parallel} = \frac{42V}{1.4A}$$

$$R_{Parallel} = 30\Omega$$

$$G_4 = G_{Parallel} - G_1 - G_2 - G_3$$

$$G_4 = \frac{1}{30\Omega} - \frac{1}{200\Omega} - \frac{1}{60\Omega} - \frac{1}{120\Omega}$$

$$G_4 \approx 3.333mS$$

And finally, $R_4 = 1/G_4$, thus $R_4 = 300\Omega$.

The smaller the value of this fourth resistor, the more current it will siphon off from the other three resistors, thus reducing the system voltage further.

Time for an example involving power.

Example 4.5.5

A parallel network is shown in Figure 4.5.9 . Determine the total power dissipated by the four resistors.

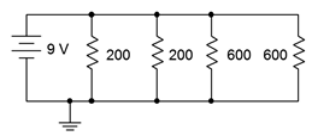


Figure 4.5.9 : Circuit for Example 4.5.5 .

While the powers for each resistor can be calculated individually and then added, it is worthwhile to note that the total power dissipated must be equal to the power generated. The power generated by the battery is dictated by power law, in other words, the battery voltage times its output current. To find this current, we can determine the effective resistance and then apply Ohm's law.

The total resistance can be found directly by using the conductance formula, however, the values can be divided into convenient pairs first. The pair of 200 Ω resistors is equivalent to a single 100 Ω. Similarly, the two 600 Ω resistors are equivalent to a single 300 Ω. Product-sum rule may be used next or $100||300$ can be determined using the ratio rule. This is a 3:1 ratio, so the result will 3/4ths of the smaller resistor, or 75 Ω.

$$I_{Total} = \frac{E}{R_{Total}}$$

$$I_{Total} = \frac{9V}{75\Omega}$$

$$I_{Total} = 120mA$$

$$P = E \times I$$

$$P = 9V \times 120mA$$

$$P = 1.08W$$

Slightly quicker, we could also use the following approach:

$$P = \frac{E^2}{R}$$

$$P = \frac{(9V)^2}{75\Omega}$$

$$P = 1.08W$$

This page titled 2.3.5: Parallel Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.5: Parallel Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.6: Current Limiting - Fuses and Circuit Breakers

Most electrical and electronic devices are designed to run off of a constant voltage source. Examples include battery powered devices and the numerous items that connect to standard residential and commercial electrical systems (e.g., the 120 VAC system in North America). As such, parallel connections are ideal when a variable number of devices are to be powered because, unlike a series connection, each device will see the same voltage rather than having that voltage split among them. There is another advantage to a parallel connection. Consider the two lighting schemes presented in Figure 4.6.1 ; one utilizing a series configuration and the other a parallel configuration.

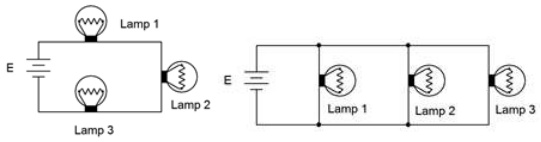


Figure 4.6.1 : Lamp connections: Series (left) and parallel (right).

Typically, when a lamp fails, it fails creating an open. If any of the lamps in the series connection fails, current in the loop is interrupted and all of the lamps go out. In contrast, if one of the lamps in the parallel connection fails, the remaining lights will stay on. Further, in the series connection, there is no immediate way to determine which lamp has failed as none of them will be lit. Each lamp has to be tested in turn to determine the faulty unit. In the parallel system, it is obvious which lamp has failed because that's the one that's out.

In a home, school or commercial setting, any number of devices may be plugged into wall sockets with the expectation that each unit will receive the same voltage and operate correctly due to the parallel nature of the system. As convenient as this is, there is a practical problem. Every device that is added to the system presents another path for current flow. Due to KCL, this means that the total current drawn from the source must increase. This creates a potential problem because if too much current is drawn, there can be an appreciable voltage drop across the wires feeding the outlets as they can no longer be assumed to be zero ohms. This lowered voltage may present a problem for the attached devices, but more importantly, the large current results in unexpected heating of the wires because of the power dissipation in them (I^2R) and creates a possible fire hazard. Consequently, we must include some method to limit the current to a safe maximum value.

While some electronic systems use active current limiting schemes[1], the more common approach is a fuse or circuit breaker. These devices are placed between the voltage source and the various loads, and see their combined current. If this current becomes too great, the device interrupts the current flow by opening the circuit. Nothing will operate but nothing catches fire, either. Fuses are used typically in electronic systems, motors, automotive subsystems and the like. An array of various types of fuses is shown in Figure 4.6.2 . Whether they're the cylindrical type found in electronic instrumentation or the blade type common in automobiles, fuses are one-shot devices. They consist of a fine wire or metal link that will heat up and melt if it experiences too high of a current. Upon melting, circuit continuity is lost and current flow stops. This is called a “blown” fuse. The fuse will then need to be replaced once the problem has been fixed (i.e., whatever it was that caused excessive current to flow in the first place).



Figure 4.6.2 : Various styles of fuses.

Fuses have a nominal current rating, for example, two amps. This does not mean that as soon as two amps is reached, the fuse blows. It will take some time for the fuse to heat up. The higher the current is above the nominal rating, the faster the fuse will blow. For example, it might take this nominal two amp fuse several seconds before it blows if the current is, say, 2.5 amps, but less than a second if the current is five amps. Fuses are also available in fast-blow and slow-blow forms. Fast-blow fuses will activate

many times faster than standard fuses while slow-blow types will require much long to activate (this prevents the fuse from activating accidentally with certain kinds of loads that exhibit a high initial startup current but that falls back to a more modest level once running smoothly, such as a motor). The schematic symbols for the fuse are shown in Figure 4.6.3 . The top two are IEC standard and the bottom version is the ANSI (North American) standard.

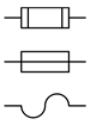


Figure 4.6.3 : Schematic symbols for fuses: IEC (top and middle), ANSI (bottom).

Fuses are generally inexpensive and effective. The obvious downside to a fuse is that they need to be replaced once they do their job. A circuit breaker solves this problem. A circuit breaker, or just breaker for short, can be thought of as a resettable fuse. Breakers do not have a melting link that interrupts current flow. Instead, a breaker operates like an intelligent switch: if the current is too large, the switch is thrown open. The operator can then fix the problem and reset the breaker (i.e., return the switch to the operating position). A selection of breakers is shown in Figure 4.6.4 . Circuit breakers are available in a variety of sizes. For residential wiring, 15 and 20 amp sizes are common for general use although higher sizes such as 30 and 50 amps are available for circuits feeding high power appliances such as electric stoves, water heaters and clothes dryers.



Figure 4.6.4 : Residential circuit breakers including cutaway view.

References

[1]For details, see Fiore, J., Semiconductor Devices as well as Operational Amplifiers and Linear Integrated Circuits, both OER titles.

This page titled 2.3.6: Current Limiting - Fuses and Circuit Breakers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.6: Current Limiting - Fuses and Circuit Breakers** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.7: Summary

In this chapter we have introduced and examined parallel circuits using either one or more current sources or a single voltage source, along with two or more resistors. The hallmark of a parallel configuration is that all components are connected to just two nodes. This means that all elements in a parallel configuration see the same voltage. If multiple current sources are present, they may be combined into a single equivalent current source by adding their values, taking care to watch for the directions of source currents; like directions add while opposing directions subtract, in much the same manner as series-connected voltage sources. Voltage sources generally are not placed in parallel as it would be a practical impossibility to establish different voltages across the same two nodes. The major exception to this rule is if the sources use the same voltage and the goal is to extend battery life. Dissimilar parallel voltage sources will most likely cause excessive heat and an unstable voltage.

The equivalent resistance of a group of parallel resistors will always be smaller than the smallest resistor in the group because each resistor adds another path for current flow, thus enhancing conductivity and reducing resistance. In general, the effective resistance is found by summing the individual conductances to find the total conductance of the group and then taking the reciprocal of this value. Two shortcut techniques when just two resistors are involved are the product-sum rule and the ratio rule (which is particularly handy when one resistor is the integer multiple of the other).

Kirchhoff's current law (KCL) states that the sum of currents entering a node must equal the sum of currents leaving that node. Another way of stating this is that the sum of all currents entering and exiting a node must be zero (obeying polarities). The current divider rule (CDR) is useful for determining the division of current between two parallel resistors. It states that the current through one resistor must equal the current entering the pair times the ratio of the other resistance value to the summed resistance of the pair.

To determine individual branch currents in a circuit driven by a voltage source, Ohm's law may be used by dividing the source voltage by the individual resistor values. These branch currents must sum to the total current delivered by the voltage source due to KCL. If the parallel network is driven by current sources, the individual branch currents can be found by determining the effective parallel resistance and then using Ohm's law to find the system voltage. Once the voltage is known, Ohm's law is used again on each resistor to find the associated branch current. Alternately, the resistors can be simplified into successive pairings and then CDR can be applied repeatedly.

Real world ammeters and voltmeters exhibit internal resistances that can load down circuit elements and cause errors in measurement. It is best if voltmeters have an internal resistance that is lat east an order of magnitude (preferably two) larger than the resistors they are placed across in order to limit undesired current divider effects. Similarly, ammeters should have an internal resistance at least an order of magnitude (preferably two) smaller than the resistors with which they are placed in-line to limit undesired Ohm's law effects.

Fuses and circuit breakers are used to limit potentially damaging current in a system. Both devices will activate by opening the circuit if the current passing through them exceeds their rated value for a specified period of time. This results in zero current flow. Fuses are one-shot devices that must be replaced after being activated. In contrast, circuit breakers can be activated numerous times and are simply reset via a switch or button after being tripped.

Review Questions

1. Define the term parallel connection.
2. How is the equivalent resistance for a group of parallel connected resistors computed?
3. How is the equivalent value for parallel connected current sources computed?
4. Why are voltage sources generally not placed in parallel?
5. Define the product-sum rule and its use.
6. Define Kirchhoff's current law.
7. Define the current divider rule.
8. Explain the basic operation and usage of fuses and circuit breakers.

This page titled 2.3.7: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.7: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.3.8: Exercises

Analysis

1. Determine the effective resistance of the network shown in Figure 4.8.1 .

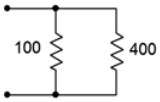


Figure 4.8.1

▼ Answer 1

- 80 Ω

2. Determine the effective resistance of the network shown in Figure 4.8.2 .

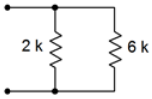


Figure 4.8.2

3. Determine the effective resistance of the network shown in Figure 4.8.3 .

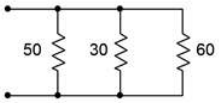


Figure 4.8.3

▼ Answer 3

- 14.3 Ω

4. Find the effective source current of the network shown in Figure 4.8.4 .

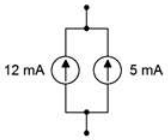


Figure 4.8.4

5. Determine the effective source current of the network shown in Figure 4.8.5 .

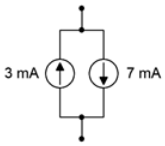


Figure 4.8.5

▼ Answer 5

- 4mA, down

6. Find the source and resistor currents for the circuit of Figure 4.8.6 .

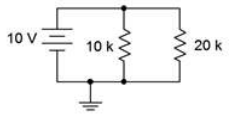


Figure 4.8.6

7. Determine the source and resistor currents for the circuit of Figure 4.8.7 .

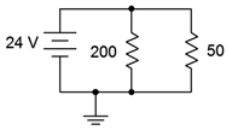


Figure 4.8.7

▼ Answer 7

- $I_{200} = 120mA$
- $I_{50} = 480mA$
- $I_{src} = 600mA$

8. Find the source and resistor currents for the circuit of Figure 4.8.8 .

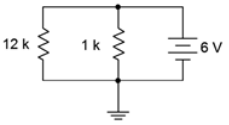


Figure 4.8.8

9. For the circuit of Figure 4.8.9 , determine the source and resistor currents.

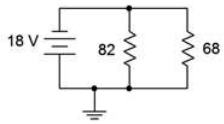


Figure 4.8.9

▼ Answer 9

- $I_{82} = 219.5mA$
- $I_{69} = 264.7mA$
- $I_{src} = 484.2mA$

10. Determine the current through each resistor in the circuit of Figure 4.8.10 . Also determine the total power generated by the source.

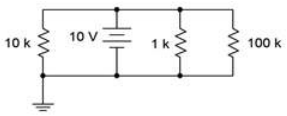


Figure 4.8.10

11. Consider the circuit shown in Figure 4.8.10 . Assume that the 100 kΩ is replaced with another resistor ten times as large. Will this have a major impact on the current exiting source? Why/why not?

▼ Answer 11

- No. It is already the smallest current by an order of magnitude, and this makes it smaller

12. Consider the circuit shown in Figure 4.8.10 . Assume that the 1 kΩ is replaced with another resistor ten times smaller. Will this have a major impact on the current exiting source? Why/why not?

13. Find the current through each resistor in the circuit of Figure 4.8.11 .

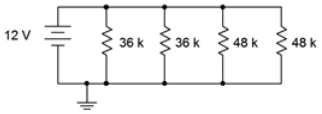


Figure 4.8.11

▼ Answer 13

- $I_{36k} = 333.3uA$
- $I_{48k} = 250uA$

14. Determine the current through each resistor in the circuit of Figure 4.8.12 . Also determine the total current exiting by the source.

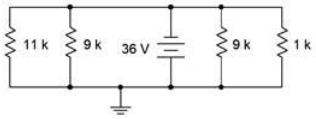


Figure 4.8.12

15. Determine the current through each resistor in the circuit of Figure 4.8.13 .

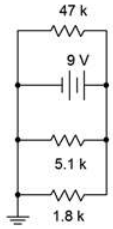


Figure 4.8.13

▼ Answer 15

- $I_{47k} = 0.1915mA$
- $I_{5.1k} = 1.765mA$
- $I_{1.8k} = 5mA$

16. For the circuit shown in Figure 4.8.14 , determine the current through each resistor and the source voltage.

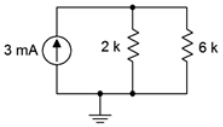


Figure 4.8.14

17. For the circuit shown in Figure 4.8.15 , determine the current through each resistor and the source voltage.

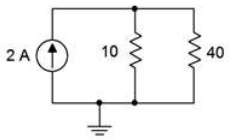


Figure 4.8.15

▼ Answer 17

- $I_{10} = 1.6A$
- $I_{40} = 0.4mA$
- $V_{src} = 16V$

18. For the circuit shown in Figure 4.8.16 , determine the current through each resistor and the source voltage.

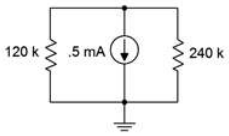


Figure 4.8.16

19. For the circuit shown in Figure 4.8.17 , determine the current through each resistor and the source voltage.

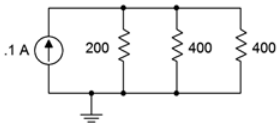


Figure 4.8.17

▼ Answer 19

- $I_{200} = 50mA$
- $I_{400} = 25mA$
- $V_{src} = 10V$

20. For the circuit shown in Figure 4.8.18 , determine the current through each resistor and the source voltage.

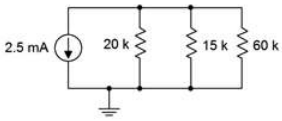


Figure 4.8.18

21. For the circuit shown in Figure 4.8.19 , determine the current through each resistor and the source voltage.

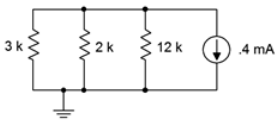


Figure 4.8.19

▼ Answer 21

- $I_{3k} = 145.5uA$

- $I_{2k} = 218.2uA$
- $I_{12k} = 28.57uA$
- $V_{src} = 0.4364V$

22. For the circuit shown in Figure 4.8.20 , determine the current through each resistor and the source voltage.

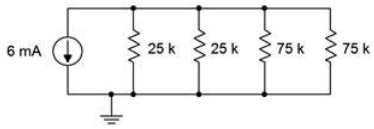


Figure 4.8.20

23. Referring to the circuit of Figure 4.8.20 , determine the resistor currents if the right-most 75 kΩ resistor is accidentally opened (i.e., unconnected). How do these results compare to those of problem 22?

▼ Answer 23

- $I_{25k} = 2.571mA$
- $I_{75k} = 0.8571mA$
- Currents will be higher in the 25k resistors because there is one fewer path for current to go

24. Referring to the circuit of Figure 4.8.20 , determine the resistor currents if the right-most 75 kΩ resistor is accidentally shorted. How do these results compare to those of problems 22 and 23?

25. For the circuit shown in Figure 4.8.21 , determine the current through each resistor and the source voltage.

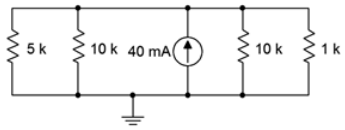


Figure 4.8.21

▼ Answer 25

- $I_{5k} = 15.714mA$
- $I_{10k} = 2.857mA$
- $I_{1k} = 28.57mA$
- $V_{src} = 28.57V$

26. Given the circuit of Figure 4.8.22 , find the currents through the two resistors.

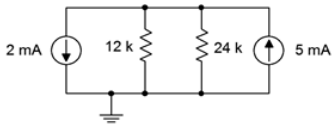


Figure 4.8.22

27. If the 5 mA current source shown in Figure 4.8.22 is accidentally wired in upside down, does the voltage across the 12 k Ω resistor become more positive or more negative with respect to ground?

▼ Answer 27

- The voltage is larger, but now negative, so it becomes more negative

28. Given the circuit of Figure 4.8.23 , find the voltages across the three resistors.

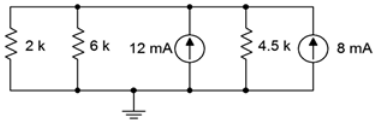


Figure 4.8.23

29. Find the currents through the three resistors in Figure 4.8.23 .

▼ Answer 29

- $I_{2k} = 11.25mA$
- $I_{6k} = 3.75mA$
- $I_{4.5k} = 5mA$

Design

30. For the network shown in Figure 4.8.24 , determine a for values for R_1 given that R_2 is 12 kΩ and the equivalent combination is 8 kΩ.

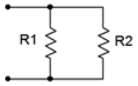


Figure 4.8.24

31. Add a third parallel resistor to the circuit of Figure 4.8.8 such that the source current is 10 mA.

▼ Answer 31

- 1.714kΩ

32. Add a fourth parallel resistor to the circuit of Figure 4.8.10 such that the source current is 20 mA.

33. Consider the circuit shown in Figure 4.8.14 . Determine a new value for the current source such that the source voltage equals 10 volts.

▼ Answer 33

- 6.667mA

34. Consider the circuit shown in Figure 4.8.16 . Determine a new value for the current source such that the source voltage equals 20 volts.

35. Given the circuit of Figure 4.8.25 , if the source is 6 volts and R_1 is 2 kΩ, what must be the value of R_2 if the total current exiting the source is 10 mA?

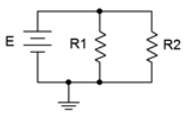


Figure 4.8.25

▼ Answer 35

- 857Ω

36. For the circuit shown in Figure 4.8.26 , determine values for resistors R_2 and R_3 such that the current through R_2 is twice the current through R_1 and the current through R_3 is half the current through R_1. The source is 6 volts and R_1 is 2 kΩ.

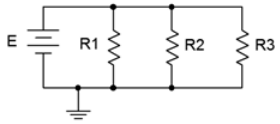


Figure 4.8.26

37. For the circuit shown in Figure 4.8.27 , determine values for resistors R_1 and R_2 such that the current through R_2 is twice the current through R_1. The source is 10 mA and R_1 is 6 kΩ.

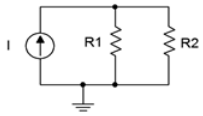


Figure 4.8.27

▼ Answer 37

- 3kΩ

Challenge

38. For the circuit shown in Figure 4.8.12 , determine a new value for the 11 kΩ resistor such that the supply current is 50 mA.

39. For the circuit shown in Figure 4.8.14 , determine a new value for the 2 kΩ resistor such that the voltage drop across the 6 kΩ is 15 volts.

40. Consider the circuit shown in Figure 4.8.21 . If the current source was replaced with a voltage source, what value is needed so that the same currents flow through the resistors as in the original circuit?

41. For the network shown in Figure 4.8.24 , determine values for R_1 and R_2 such that R_2 is twice the size of R_1 and the equivalent combination is 6 kΩ.

42. Given the network of Figure 4.8.3 , is it possible to replace the 60 Ω resistor with another value such that the equivalent combination of the three resistors is 25 Ω? If so, what is that value?

43. Given the network of Figure 4.8.11 , is it possible to add a fifth parallel resistor such that the source current is 1 mA? If so, what is that value?

44. For the circuit shown in Figure 4.8.26 , determine values for the three resistors such that the current through R_1 is twice the current through R_2 and four times the current through R_3. The source is 12 volts and should produce a total of 9 mA of current.

45. For the circuit shown in Figure 4.8.27 determine values for the two resistors such that the current through R_1 is half the current through R_2. The source is 24 mA and should produce a drop of 16 volts across R_1.

46. Given three current sources with values of 1 mA, 2 mA and 7 mA; how would they need to be connected in order to deliver 4 volts across a 1 kΩ load resistor?

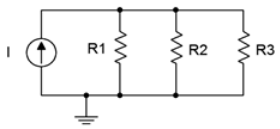


Figure 4.8.28

47. Consider the circuit of Figure 4.8.28 . Assume I is a 4 mA source. Using only 5% standard resistor values (see Appendix A), pick values for the three resistors such that the voltage across R_1 is within 10% of 10 volts as long as the resistors are within tolerance.

Simulation

48. Verify the currents found in problem 11 via a DC simulation.
49. Verify the currents found in problem 15 via a DC simulation.
50. Verify the currents and voltages found in problem 17 via a DC simulation.
51. Verify the results found in problem 25 via a DC simulation.
52. Verify the results found in problem 27 via a DC simulation.
53. Perform a DC simulation on the design of problem 44 to verify its performance.
54. Perform a DC simulation on the design of problem 45 to verify its performance.
55. Perform a DC simulation on the design of problem 46 to verify its performance.
56. Perform a Monte Carlo or worst-case simulation on the design of problem 47 to verify its performance.

This page titled 2.3.8: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.8: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

SECTION OVERVIEW

2.4: Series-Parallel Resistive Circuits

Learning Objectives

After completing this chapter, you should be able to:

- Identify series-parallel resistive circuits that include a single effective voltage or current source.
- Compute component and node voltages for single-source series-parallel resistive networks.
- Compute branch currents for single-source series-parallel resistive networks.
- Break down series-parallel networks into smaller series-only and parallel-only sub-units, and utilize KVL, KCL and other previously established techniques to aid in the analysis of these circuits.

2.4.1: Introduction

2.4.2: Series-Parallel Connections

2.4.3: Simplifying Series-Parallel Components

2.4.4: Series-Parallel Analysis

2.4.5: Summary

2.4.6: Exercises

This page titled 2.4: Series-Parallel Resistive Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

2.4.1: Introduction

In the preceding two chapters, series circuits and parallel circuits were examined. Each offered unique laws and rules for their configuration, such as Kirchhoff's voltage law and the current divider rule. Distinct techniques of analysis were offered to determine system current and voltage, and the currents and voltages associated with individual components. This almost mechanical approach was successful mainly because there are so few variations on the theme. In the case of a series circuit, more resistors might be added in the “daisy chain”, and likewise multiple voltage sources, but once you have experience with a few of these types of circuits, there is not much concern when simply adding more of the same because a pattern becomes apparent. The same holds true with parallel circuits: more resistors might be added as further “rungs on the ladder”, as well as multiple current sources similarly arranged, but again, once familiarity has been gained, obvious patterns emerge. This is not the case with series-parallel circuits, and thus, there are no simple recipes for our solution paths. Fortunately, laws and rules such as Ohm's law, KCL and KVL still hold true for these circuits, thus the focus shifts to their appropriate application.

To begin with, there are infinite variations of series-parallel circuits. This chapter deals with a subset, namely those that are driven by a single current or voltage source (after reducing trivial combinations), and which may be simplified using series and parallel resistor combinations. More complex configurations using multiple sources await in subsequent chapters. The key to analyzing basic series-parallel circuits is in recognizing portions of the circuit, or sub-circuits, that exhibit a series or parallel configuration by themselves, and then applying the series and parallel analysis rules to those sections. Ohm's law, KVL and KCL may be used in turn to “chip away” at the problem until all currents and voltages are found. As individual voltages and currents are determined, this makes it easier to apply these rules to determine other values. Given this observation, the number of potential solution paths tends to grow exponentially as the number of components increases. As a consequence, when faced with the same circuit, six people may solve it six different ways, no particular way being more or less correct than any other. The only thing we can say is that some solution paths might be more computationally efficient than others, meaning they take less work. Do not let this bother you. The fact that these circuits can be solved in a diverse number of ways is a strength, not a weakness. After all, you only need to recognize one of those ways, not all of them, in order to be successful. In this chapter's examples, various competing methods will be explored, but not every solution path will be spelled out for each one. Flexibility of thought and view will prove to be an asset.

This page titled 2.4.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.4.2: Series-Parallel Connections

A basic **series-parallel system** is shown in Figure 5.2.1 . It should be obvious that this system is neither a simple series connection nor a simple parallel connection, but rather some combination of the two. In order for it to be series, the current through each component would have to be the same. This does not have to be true here. Consider the connection node directly above block C. This also connects to items B, D and E. All we really know about that node is that there are four paths leading to it and that the four associated currents must sum to zero, thanks to KCL. There is nothing that requires the current through, say, block D to be the same as that through block E. Similarly, there is nothing that stipulates that the voltage across block C must be the same as that across block F. After all, that is the hallmark of a parallel configuration, and that would require those two blocks to be connected to the same two nodes, which they aren't.

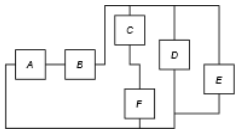


Figure 5.2.1 : A series-parallel configuration.

What we can say about the configuration of Figure 5.2.1 is that some of the components are in series among themselves, and some are in parallel among themselves. We might recognize these groupings as sub-circuits. For example, it is true that blocks A and B are in series with each other because the current through one of them must be the same as the current through the other. The same is true of blocks C and F; they are in series with each other. Similarly, we can say that blocks D and E are in parallel with each other because they must see the same voltage as they are each connected to the same two nodes.

This page titled 2.4.2: Series-Parallel Connections is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.2: Series-Parallel Connections** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.4.3: Simplifying Series-Parallel Components

We can go beyond the simple observations presented above. Precisely how we approach this will depend on the components represented by the blocks (most often resistors, but possibly a voltage source or a current source) and how they are wired together. For starters, if all of the items in a sub-circuit are of the same type, they can just be combined using the techniques discussed in the previous chapters. Resistors in series are added, as are voltage sources in series-being wary of polarity. Resistors in parallel are combined using conductance or the product-sum rule while parallel current sources are added, again being wary of polarity. Things get a little trickier when there is a mix, such as two resistors in series with a voltage source, or a resistor in parallel with a couple of current sources. Precisely how this sort of situation is handled will become apparent shortly.

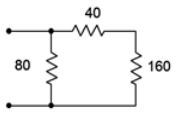


Figure 5.3.1 : A simple series-parallel resistor network.

To illustrate, let us consider the resistor configuration presented in Figure 5.3.1 . Imagine that an ohmmeter is connected to the two open terminals. What would it read? The ohmmeter would apply a small current to the circuit to determine the effective resistance of the group. The three resistors pictured here are not a simple series configuration because we can identify a current split at the 40 Ω and 80 Ω resistors. Similarly, it is not a parallel network because the 40 Ω and 160 Ω resistors are not both tied to the same two nodes. What we can say is that the 40 Ω and 160 Ω resistors are in series with each other as they must each see the same current. Thus, we can combine the two into a single equivalent resistance of 200 Ω. This equivalent is in parallel with the 80 Ω resistor and 80 || 200 is approximately 57.14 Ω. That's what the ohmmeter should read. Indeed, wherever we see this configuration with these precise values, we can replace it with a single resistor of 57.14 Ω.

The simplification technique can be outlined as follows:

- Identify subgroups of resistors that exhibit series or parallel configurations within themselves.
- Replace the subgroups with a single equivalent resistance.
- Repeat the above steps until the circuit is reduced to a single resistance, or alternately, a simple series-only or parallel-only configuration.

The looping nature of the technique outlined above may need to be invoked many times, depending on the complexity of the network. In simpler networks, a single pass may be sufficient. Initially, it may be wise to redraw the circuit for each iteration of the loop. Eventually, with practice, this will not be needed except with the most complex networks. Let's look at a somewhat more involved example.

Example 5.3.1

A series-parallel resistor network is shown in Figure 5.3.2 . Determine its equivalent resistance.

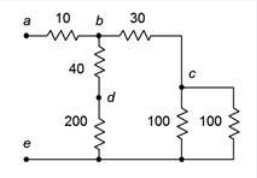


Figure 5.3.2 : Network for Example 5.3.1 .

The first step is to recognize those subgroups that are in series or in parallel with themselves. One obvious candidate is that the two 100 Ω resistors are in parallel with each other. Two resistors of equal value in parallel are equivalent to half the resistance, or 50 Ω in this case.

The other candidate is the 40 Ω, 200 Ω pair. These are in series. The equivalent resistance of the pair is the series combination, or 240 Ω.

There are no other subsets of resistors that can be reduced (yet). The network is redrawn in Figure 5.3.3 with the equivalent resistances. The process is now repeated.

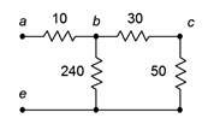


Figure 5.3.3 : Partially simplified network.

In the newly reduced network, the 50 Ω and 30 Ω resistances are in series, yielding 80 Ω.

Without redrawing, the process may be repeated again, this time the 240 Ω being in parallel with the new 80 Ω equivalent. 240 || 80 is equal to 60 Ω. This sequence has reduced the three right-most resistances down to one: the new 60 Ω equivalent.

Finally, the 60 Ω equivalent is in series with the 10 Ω resistor, yielding a final equivalent resistance of 70 Ω.

Ladders

A ladder is a unique series-parallel configuration. It is arranged in a cascade of series and parallel connections. Ladders are used in a variety of applications, one of which will be examined in the next section.

The name is obvious based on the appearance of the network. An example resistive ladder network is shown in Figure 5.3.4 . The main thing to note is that the ladder network consists of “sections loading sections” repeatedly. By loading, we mean that this element (the load) draws off current and power from the prior section. As a result, simple two-resistor voltage dividers cannot be used. For example, it is a common error to assume that R_1 and R_2 create a voltage divider which can be used to determine V_{bf} if V_{af} is known. Similarly, the uninitiated may believe incorrectly that R_3 and R_4, or R_5 and R_6 create voltage dividers.

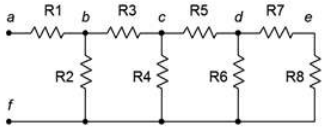


Figure 5.3.4 : A resistive ladder network.

In fact, the only proper voltage divider in this configuration is between R_7 and R_8. The reason for this becomes apparent if we imagine determining the equivalent resistance of the network by placing an ohmmeter at terminals a and f. The ohmmeter will apply a current to node a which passes through R_1. At node b this current splits, part going down R_2 and part continuing through R_3. Clearly then, R_1 and R_2 are not in series as they do not see the same current. Further, the current that flows through R_3 enters node c where it splits again, part down R_4 with the remainder flowing through R_5. Thus, R_3 and R_4 are not in series, either. The same sort of thing happens again at node d. This would continue for as many rungs as are on the ladder, the sole exception being the final two resistors which are in series (R_7 and R_8 here).

How then do we find the equivalent resistance of this network? We begin at the end farthest from the open terminals. We have already noted that R_7 and R_8 are in series. This pair, if treated as a single resistance, is in parallel with R_6, or $R_6||(R_7 + R_8)$. This group of three is in series with R_5, yielding $R_5 + R_6||(R_7 + R_8)$[1]. This new group of four is in parallel with R_4, yielding $R_4||(R_5 + R_6||(R_7 + R + 8))$. This group is in series with R_3, and that new group is in turn in parallel with R_2. Finally, that penultimate grouping is in series with R_1. Therefore, the equivalent resistance of the network is $R_1 + R_2||(R_3 + R_4||(R_5 + R_6||(R_7 + R_8)))$. Surely, it is a tad tedious to compute the value, but particularly difficult. A word to the wise: if the goal is determine the voltages at the various nodes or the myriad branch currents, it will be helpful to keep track of the equivalent resistances of the various groupings. For example, if V_{cf} is known and the goal is to find V_{df}, the voltage divider would be $R_6||(R_7 + R_8)/(R_5 + R_6||(R_7 + R_8))$.

Example 5.3.2

A resistive ladder network is shown in Figure 5.3.5 . Determine its equivalent resistance.

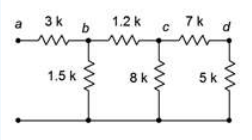


Figure 5.3.5 : Network for Example 5.3.2 .

Starting at the far end from the open terminals, we note that the 7 kΩ and 5 kΩ resistors are in series, yielding 12 kΩ. This is in parallel with the 8 kΩ. 12 k || 8 k is 4.8 kΩ. This is in series with the 1.2 kΩ yielding 6 kΩ. The 6 kΩ equivalent is in parallel with the 1.5 kΩ which yields 1.2 kΩ.

Finally, the 1.2 kΩ equivalent is in series with the 3 kΩ, yielding a final equivalent resistance of 4.2 kΩ.

Remember, always start at the far end and work toward the open terminals, i.e., the terminals into which you are trying to find the equivalent resistance.

Example 5.3.3

A resistive ladder network is shown in Figure 5.3.6 . Determine its equivalent resistance.

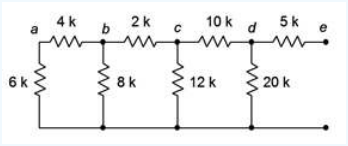


Figure 5.3.6 : Network for Example 5.3.3 .

The far end from the open terminals is the extreme left. First, the 6 kΩ and 4 kΩ resistors are in series, yielding 10 kΩ. This is in parallel with the 8 kΩ, achieving approximately 4.444 kΩ. In turn, this is in series with the 2 kΩ yielding 6.444 kΩ. That result is in parallel with the 12 kΩ which yields 4.193 kΩ. Placing this in series with the 10 kΩ resistor yields 14.193 kΩ. This is in parallel with the 20 kΩ yielding 8.302 kΩ.

Finally, the 8.302 kΩ equivalent is in series with the 5 kΩ, yielding a final equivalent resistance of approximately 13.3 kΩ.

References

[1]Recall that || has higher precedence than + or –, therefore || is performed first, rather like multiplication when mixed with addition and subtraction.

This page titled 2.4.3: Simplifying Series-Parallel Components is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.3: Simplifying Series-Parallel Components** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.4.4: Series-Parallel Analysis

Analysis of series-parallel networks involves recognizing those sub-circuits that are in series or that are in parallel among themselves, performing simplifications as needed, and winding up with a simple series-only or parallel-only equivalent. Then the various laws such as Ohm's law, KVL, KCL, VDR and CDR are applied to the various simplified networks to determine the parameters of interest. There is no single solution technique, each circuit being unique.

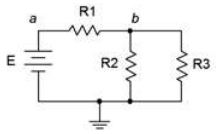


Figure 5.4.1 : A simple series-parallel circuit.

Let's begin by considering the circuit of Figure 5.4.1 . To review, this is neither just a series circuit nor just a parallel circuit. If it was a series circuit then the current through all components would have to be same, that is, there would no nodes where the current could divide. This is clearly not the case as the current flowing through R_1 can divide at node b, with one portion flowing down through R_2 and the remainder through R_3. On the other hand, if it was strictly parallel, then all of the components would have to exhibit the same voltage and therefore there would be only two connection points in the circuit. This is also not the case as there are three such points: a, b and ground.

What is true is that resistors R_2 and R_3 are in parallel. We know this because both components are attached to the same two nodes, b and ground, and must exhibit the same voltage, V_b. As such, we can find the equivalent resistance of this pair and treat the result as a single resistance, let's call it R_x. In this newly simplified circuit, Rx is in series with R_1 and the source, E. We have reduced the original circuit down to a simple series circuit and thus the series analysis rules may be applied.

There are many solution paths at this point. For example, we could find the total resistance, R_t, by adding R_1 to R_x. Dividing this by E yields the total current flowing out of the source, I_{total}. This current must flow through R_1 so Ohm's law can be used to find the voltage drop across R_1. This same current must be flowing through R_x, so Ohm's law can be used to find the associated voltage (V_b). The currents through R_2 and R_3 may then be found using Ohm's law for each resistor (e.g., the current through R_2 must be V_b/R_2). Alternately, these currents may be found by using the current divider rule between R_2 and R_3 (e.g., the current through R_2 must be $I_{total} \cdot R_1/(R_1 + R_2)$; remember, the current divider rule uses the ratio of the opposite resistor over the sum).

Another solution path would be to apply the voltage divider rule to R_1 and R_x in order to derive the two voltage drops (or the rule can be applied to find just one of the drops and the other voltage may be found by subtracting that from the source, an application of KVL). Once the voltages are determined, Ohm's law can be used to find the currents. If powers are needed, they are easily determined once the voltages and currents are found.

Example 5.4.1

A series-parallel circuit is shown in Figure 5.4.2 . Determine V_b.

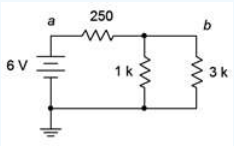


Figure 5.4.2 : Circuit for Example 5.4.1 .

As we are only interested in finding one voltage, the voltage divider rule is a good candidate. The 1 kΩ is in parallel with the 3 kΩ, yielding an equivalent resistance of 750 Ω. From here we apply VDR.

$$V_b = E\frac{R_{eq}}{R_1 + R_{eq}}$$

$$V_b = 6V\frac{750\Omega}{250\Omega + 750\Omega}$$

https://eng.libretexts.org/@go/page/52902

$$V_b = 4.5V$$

To clarify the current directions and voltage polarities, the circuit is redrawn and appropriately labeled in Figure 5.4.3 . We can crosscheck the answer in a variety of ways. For example, Figure 5.4.3 indicates, via KVL, that the voltage drop across the 250 Ω resistor plus V_b must equal the source voltage of 6 volts. Consequently, V_{250} must be 6 volts minus 4.5 volts, or 1.5 volts. Ohm's law then indicates that the current through the 250 Ω must be 1.5 V/250 Ω, or 6 milliamps. Based on the earlier resistive equivalence, the total resistance as seen by the source must be 250 Ω in series with 750 Ω, or 1 kΩ. This indicates the source current must be 6 V/1 kΩ, or 6 milliamps. As the source and the 250 Ω are in series, they must see the same current. This crosscheck shows that they do: happy happy joy joy.

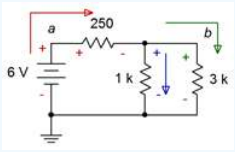


Figure 5.4.3 : Circuit for Example 5.4.1 with polarities and directions included.

We can go further and verify the currents via KCL. For the first parallel resistor we find:

$$I_{1k} = \frac{V_b}{1k\Omega}$$

$$I_{1k} = \frac{4.5V}{1k\Omega}$$

$$I_{1k} = 4.5mA$$

And for the second resistor we see:

$$I_{3k} = \frac{V_b}{1k\Omega}$$

$$I_{3k} = \frac{4.5V}{3k\Omega}$$

$$I_{3k} = 1.5mA$$

Obviously, these sum to the entering current of 6 mA.

Example 5.4.2

Determine V_b and the source current in the circuit of Figure 5.4.4 .

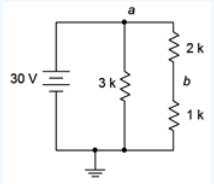


Figure 5.4.4 : Circuit for Example 5.4.2 .

In this circuit the 3 kΩ resistor is in parallel with the series combination of the 2 kΩ and 1 kΩ. This leads to an equivalent resistance of 3 kΩ in parallel with 3 kΩ, or 1.5 kΩ. From here we can find the source current.

$$I_s = \frac{E}{R_{Total}}$$

$$I_s = \frac{30V}{1.5k\Omega}$$

$$I_s = 20mA$$

This current should split evenly down the two vertical paths as they each present 3 kΩ of resistance (10 mA each achieves 30 volts for V_a, which is the source).

The voltage divider rule is a good choice for V_b as we know the applied voltage.

$$V_b = E\frac{R_{eq}}{R_1 + R_{eq}}$$

$$V_b = 30V\frac{1k\Omega}{1k\Omega + 2k\Omega}$$

$$V_b = 10V$$

For clarity, the circuit is redrawn and relabeled with current directions and voltage polarities in Figure 5.4.5 . As a further exercise, try to verify the answers above using alternate means, as illustrated in Exercise 5.4.

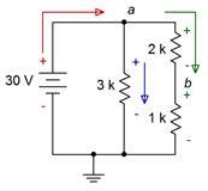


Figure 5.4.5 : Circuit for Example 5.4.2 with polarities and directions included.

As the circuit grows, more and more solution paths exist. Consider the circuit of Figure 5.4.6 . In this case, R_3 and R_4 are in parallel. This parallel combination is in series with R_2. Finally, this set of three resistors is in parallel with R_1 and E, reducing to a parallel circuit. Consequently, we know that the voltage across R_1 must be equal to E. Also, the currents through R_1 and R_2 must add up to the current exiting the source (KCL). Further, the currents through R_3 and R_4 must add up to the current flowing through R_2 (KCL). Also, the voltages across R_3 and R_4 must be the same (they are in parallel) and that this voltage plus the voltage across R_2 must equal E (KVL). One solution path would be to find the total resistance as seen by the source, $R_1||(R_2 + (R_3||R_4))$, and use this to find the total current flowing out of the source. The current divider rule can then be used between R_1 and the equivalent resistance $(R_2 + (R_3||R_4))$. Ohm's law can be used subsequently to find various voltage drops. Alternately, the voltage divider rule can be used between R_2 and the parallel equivalent $(R_3||R_4)$ as this combination is driven by E. Knowing the voltages, the currents may be determined. With so many possible solution paths for large circuits, it is often worthwhile to take a moment to map out a strategy rather than just “diving in” and hoping it will all work out.

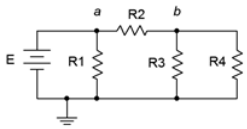


Figure 5.4.6 : A slightly more complex series-parallel circuit.

Example 5.4.3

The circuit of Figure 5.4.7 is the same as the resistive network presented in Example 5.3.1 but with the addition of a voltage source. Determine V_b, V_c, V_d, the source current I_s, and the current flowing through the 40 Ω resistor, I_{40}.

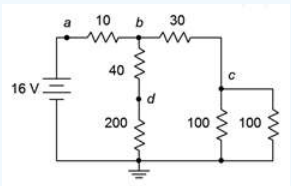


Figure 5.4.7 : Circuit for Example 5.4.3 .

Thanks to Example 5.3.1, the equivalent resistances of various portions have already been determined, saving some time. To reiterate, the pair of 100 Ω resistors are in parallel for 50 Ω. That's in series with the 30 Ω for 80 Ω. In the middle path, the 200 Ω and 40 Ω are in series, yielding 240 Ω. 240 || 80 = 60 Ω. This is in series with the 10 Ω, yielding a total resistance loading the source of 70 Ω.

$$I_s = \frac{E}{R_{Total}}$$

$$I_s = \frac{16V}{70\Omega}$$

$$I_s \approx 228.6mA$$

The three voltages can be quickly found via VDR as we know all of the associated sub-circuit resistances.

$$V_b = E\frac{R_x}{R_x + R_y}$$

$$V_b = 16V\frac{60\Omega}{60\Omega + 10\Omega}$$

$$V_b \approx 13.71V$$

$$V_c = V_b\frac{R_x}{R_x + R_y}$$

$$V_c = 13.71V\frac{50\Omega}{50\Omega + 30\Omega}$$

$$V_c \approx 8.57V$$

$$V_d = V_b\frac{R_x}{R_x + R_y}$$

$$V_d = 13.71V\frac{200\Omega}{200\Omega + 40\Omega}$$

$$V_d \approx 11.43V$$

Lastly, the current through the 40 Ω resistor can be found by dividing its voltage by its resistance. Its voltage is $V_b - V_d$. A quicker method is to note that because it is in series with the 240 Ω, their currents are the same.

$$I_{40} = \frac{V_b}{R_{Series}}$$

$$I_{40} = \frac{13.71V}{240\Omega}$$

$$I_{40} \approx 57.1mA$$

Once again the circuit is redrawn in Figure 5.4.8 to illustrate the current directions and voltage polarities. The usefulness of this will become even more apparent in the next example.

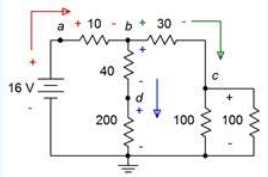


Figure 5.4.8 : Circuit for Example 5.4.3 with directions and polarities.

Example 5.4.4

Determine V_{ac} and the voltages across each resistor in the circuit of Figure 5.4.9 .

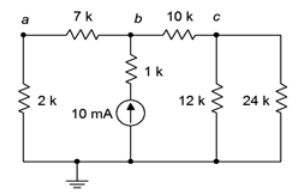


Figure 5.4.9 : Circuit for Example 5.4.3 .

This circuit is perhaps the most complex offered so far. It may not be immediately apparent what the voltage polarities are for the various resistors or the proper current directions. Without these data, it will be impossible to determine V_{ac}. To complicate matters, the voltage of interest is not ground referenced, but instead refers to two nodes located in arbitrary points of the circuit. It is worth recognizing that, by definition, $V_{ac} = V_a - V_c$, but how are those voltages determined?

To assist with this minor quandary, the circuit has been redrawn in Figure 5.4.10 . Note that the current source and associated 1 kΩ resistor have been flipped to the side, essentially trading places with the 2 kΩ and 7 kΩ series combination. This is entirely acceptable as both subgroups share node b and ground, meaning they are in parallel. Of course, the order of parallel elements in a situation like this does not matter. This orientation, having the source at one end of the circuit, tends to make visualization of current flows a little easier.

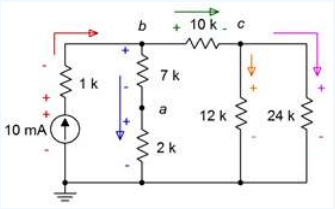


Figure 5.4.10 : Circuit for Example 5.4.3 redrawn and with polarities and current directions added.

In any case, the current exits the source and then splits at node b, some heading through the 10 kΩ with the rest flowing down the 7 kΩ with 2 kΩ series combo. At node c the current that entered the 10 kΩ splits again, flowing down into the parallel subgroup of the 12 kΩ and 24 kΩ. Knowing the current directions, the voltage polarities are determined directly.

Now that the circuit has been redrawn, several solution paths might come to mind. For starters, the voltage drop across the 1 k Ω is trivial as both its current and resistance are known. From there, a current divider can be used to find the current flowing down into the series 7 kΩ plus 2 kΩ combo. Ohm's law could then be used to find their voltages. Following that, KCL can be used to find the current through the 10 kΩ (and thus its voltage), and then Ohm's law can be applied to find V_c by treating the two right-most parallel resistors as a single unit.

Another solution path would be to determine the equivalent resistance that loads the current source. This allows the source voltage to be determined, after which a voltage divider may be applied between the 1 kΩ and the remaining five resistors to find V_b. From there, two more voltage dividers can be used to find V_a and V_c which will allow determination of the remaining resistor voltages via KVL. There are other possible paths as well but for lack of a better reason, we shall chase the first path that was outlined.

Before getting too deep, it would be wise to determine the equivalent resistances of a few subgroups. First, the 12 kΩ || 24 kΩ combo reduces down to 8 kΩ. This is in series with the 10 kΩ, leaving 18 kΩ for the three right-most resistors. In the middle branch, the 2 kΩ + 7 kΩ series combo is equivalent to 9 kΩ. The parallel combination of 9 kΩ || 18 kΩ yields 6 kΩ. Finally, this last bit is in series with the 1 kΩ producing 7 kΩ for the equivalent resistance of all resistors.

The source voltage and the voltage across the 1 kΩ resistor are determined via Ohm's law:

$$V_{source} = I_{source} \times R_{equivalent}$$

$$V_{source} = 10mA \times 7k\Omega$$

$$V_{source} = 70V$$

$$V_{1k} = I_{source} \times R$$

$$V_{1k} = 10mA \times 1k\Omega$$

$$V_{1k} = 10V$$

As the source produces 70 volts and the drop on the 1 kΩ is 10 volts, then by KVL, V_b must be 70 volts − 10 volts, or 60 volts (which will be useful for a crosscheck in a moment). Using a current divider, the 10 mA source splits between the middle and right branches. The equivalent resistances of those branches are 9 kΩ and 18 kΩ, as found earlier.

$$I_{middle} = I_{source}\frac{R_{right}}{R_{right} + R_{middle}}$$

$$I_{middle} = 10mA\frac{18k\Omega}{18k\Omega + 9k\Omega}$$

$$I_{middle} \approx 6.667mA$$

Via KCL, the right branch current must be 10 mA − 6.667 mA, or approximately 3.333 mA. As a crosscheck, if 6.667 mA passes through a total of 9 kΩ, that produces a voltage drop of 60 volts, exactly as expected from the earlier calculation for node b. At this point, the current through each resistor is known (treating the 12 kΩ || 24 kΩ combo as a unit) and therefore Ohm's law may be used to determine their voltage drops. They are summarized below.

$$V_{7k} \approx 46.67V$$

$$V_{2k} \approx 13.33V$$

$$V_{10k} \approx 33.33V$$

$$V_{12k} = V_{24k} \approx 26.67V$$

Finally, to determine V_{ac} we know that $V_{ac} = V_a - V_c$, and by observation V_a is the drop across the 2 kΩ while V_c is the drop across the 12 kΩ || 24 kΩ combo.

$$V_{ac} = V_a - V_c$$

$$V_{ac} \approx 13.33V - 26.67V$$

$$V_{ac} \approx -13.33V$$

Do not let the negative sign be bothersome. All it indicates is that node a is at a lower potential than node c, i.e., that node a is 13.33 volts below node c. It is also purely coincidental that V_{ac} and V_a have the same magnitude. In the lab, if a voltmeter is connected such that the red (positive) lead is attached to node a and the black (negative or reference) lead is attached to node c, the meter will show a negative value. If the leads are connected in reverse, thus measuring V_{ca}, the voltage will show up as positive (meaning that node c is higher in potential than node a, the reverse way of looking at the situation)[1].

Any Route Works

In the example just completed it was noted that, by definition, $V_{ac} = V_a - V_c$, and this dictated the method employed to find said voltage. It turns out that this is just one of many ways of determining a voltage from one point to another. In general:

> To find some voltage V_{XY}, start at point X and proceed along any convenient route to Y, subtracting voltage rises (polarities of − to +) and adding voltage drops (polarities of + to −) along the way.

In the prior example, an obvious route for V_{ac} would be to go down the 2 kΩ and then up the 12 kΩ. That gives a drop of approximately13.33 volts (positive). Going up the 12 kΩ produces a rise of approximately 26.67 volts (negative). The result is approximately −13.33 volts, as previously calculated. The technique states that any route may be used, no matter how inefficient or wacky it might be. To verify this, one odd route might be to go up the 7 kΩ, down through the 1 kΩ and source, and then up through the 24 kΩ. The summation would be − 46.67 − 10 + 70 − 26.67, or approximately −13.33 volts once again. An easy way to remember whether the potential is added or subtracted is to just look at the polarity sign where the component is entered along your chosen route.

R-2R Ladder

As promised earlier in the chapter, it's time to look at a ladder network. While ladders can use any resistance values desired, certain ratios turn out to be eminently practical. Of particular note is the R-2R ladder, an example of which is shown in Figure 5.4.11 , and which is being driven by a current source. In this configuration, only two different resistor values are used; some initial value and a second value of precisely twice that size. As drawn in Figure 5.4.11 , all horizontal resistors use value R (6 kΩ here) while all of the vertical resistors use 2R (12 kΩ here) with the exception of the very last, or terminating, resistor which uses R.

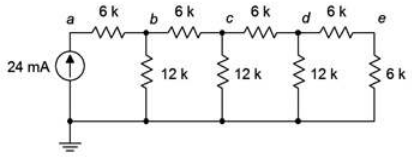


Figure 5.4.11 : An R-2R ladder network.

It turns out that this arrangement offers something very unique: a halving of successive node voltages and branch currents. This is extremely useful in digital to analog conversion circuits; the sort of hardware that would turn the digital bits of an MP3 or WAV file into listenable analog signals to feed loudspeakers or headphones.

Here is how it works. Starting at the far right end, it should be obvious that the last two 6 kΩ resistors comprise a 50% voltage divider, and thus V_e will half of V_d. The combination of the final three resistors works out to 12 kΩ || (6 kΩ + 6 kΩ), or 6 kΩ again. Thus, the divider from c to d is also 50%. This process repeats as we move to the left, each equivalent working out to R and resulting in a 50% voltage divider. It will not matter what value is chosen for R. As long as 2R is used for the other set of resistors, the result will always be successive 50% voltage dividers.

Example 5.4.5

Determine all lettered node voltages in the circuit of Figure 5.4.11 .

Continuing the prior analysis, the entire resistor network presents an equivalent resistance of 12 kΩ. The node a voltage is determined by Ohm's law:

$$V_a = I \times R$$

$$V_a = 24mA \times 12k\Omega$$

$$V_a = 288V$$

The prior analysis indicated that the subsequent node voltages should be, from left to right, 144 volts, 72 volts, 36 volts and 18 volts. A quick check follows. The source of 24 mA flows into the left-most 6 kΩ, producing a drop of 144 volts by Ohm's law. This can be subtracted from V_a (288 V) to arrive at V_b, or 144 volts, or half of the prior node voltage, as expected. The current through the left-most vertical resistor must be V_b/12 kΩ, or 12 mA. That's half of the entering current, as expected. KCL indicates that the remaining current, 24 mA − 12 mA, or 12 mA, must be flowing into the second 6 kΩ between nodes b and c. This will produce a voltage drop of 72 volts across that resistor. In other words, V_c must be 72 volts less than V_b (144 volts), or 72 volts. This is half of V_b, as expected. Likewise, the current flowing down through the second 12 kΩ must be V_c/12 kΩ, or 6 mA, once again half of the preceding current, as expected. The process continues as we move to the right, each subsequent node seeing half the voltage of the prior node and each current getting cut in half as well.

Computer Simulation

In order to verify the results of Example 5.4.4 , the R-2R ladder circuit is entered into a simulator, as shown in Figure 5.4.12 . The numbered nodes 1 through 5 in the simulation schematic correspond to the lettered nodes of the original circuit, a through e, respectively. This is one example where the use of virtual instruments would result in a very cluttered display with five multimeters. Consequently, the simulation uses a DC operating point analysis with a single output window, as shown in Figure 5.4.13 .

Although the node numbers are not in ascending order, the voltages are a perfect match to the results computed previously. Each successive node sees half the voltage of the prior node. Although not explicitly listed here, it should be obvious that the currents will also be cut in half when progressing from left to right, via a quick application of Ohm's law.

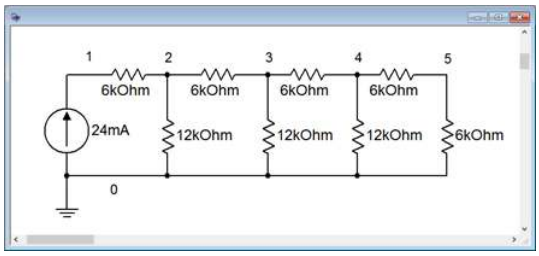


Figure 5.4.12 : The circuit of Example 5.4.3 in a simulator.

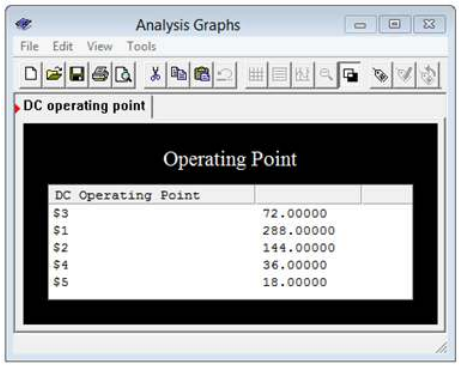


Figure 5.4.13 : Simulation results of the circuit of Example 5.4.4 .

Bridges

A bridge is a particular series-parallel configuration consisting of two pairs of series connected elements placed in parallel. An example of a resistive bridge being driven by a voltage source is shown in Figure 5.4.14 . In this circuit, R_1 and R_2 create one series connection while R_2 and R_3 create the other. The two pairs are then placed in parallel. While this circuit is shown with a voltage source, it could also be driven by a current source. Other elements might also be added.

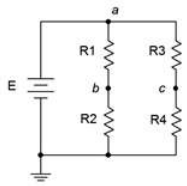


Figure 5.4.14 : A resistive bridge network.

A typical use for a bridge is the measurement of some environmental quantity. As stated in Chapter 2, there are certain resistive devices that are sensitive to environmental change. Examples include photoresistors, whose resistance is a function of light level; and thermistors, whose resistance is a function of temperature.

To understand the operation, the voltages V_b and V_c are established by the voltage dividers comprised of R_1 and R_2; and R_3 and R_4, respectively. With ordinary resistors, these voltages would be unchanging and therefore V_{bc} would also be a fixed value. We shall set it up such that V_b and V_c are the same voltage, and thus V_{bc} will be zero. Now assume that we replace R_1 with a photoresistor. As the light level increases, its resistance decreases. This will cause V_b to rise which will cause V_{bc} to go positive. If the light level decreases, the resistance of R_1 will increase, forcing V_b to drop which will cause V_{bc} to go negative. The greater the change in light, the larger V_{bc} will be, and its sign indicates whether the light level has increased or decreased compared to the original set point. By simply placing some manner of voltmeter between points b and c, we can create a display that indicates light levels.

If we want to flip the sign, we would place the photoresistor in the position of R_2 instead of R_1. Further, it is possible to increase the sensitivity by using sensors at opposite corners (e.g., R_1 and R_4), and comparative or differential measurements are possible by using both sides (e.g., R_1 with R_3).

Example 5.4.6

Determine V_{bc} in the circuit of Figure 5.4.15 .

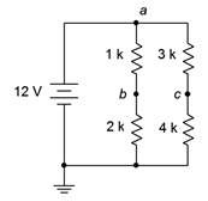


Figure 5.4.15 : Circuit for Example 5.4.5 .

Perhaps the most straightforward way to do this is to treat each half as a voltage divider and then subtract V_c from V_b.

$$V_b = E\frac{R_2}{R_2 + R_1}$$

$$V_b = 12V\frac{2k\Omega}{2k\Omega + 1k\Omega}$$

$$V_b = 8V$$

$$V_c = E\frac{R_4}{R_4 + R_3}$$

$$V_c = 12V\frac{4k\Omega}{4k\Omega + 3k\Omega}$$

$$V_c \approx 6.857V$$

Thus, V_{bc} is 8 volts minus 6.857 volts, or 1.143 volts.

Computer Simulation

In order to verify the results of Example 5.4.5 , the bridge network of Figure 5.4.15 is entered into a simulator, as depicted in Figure 5.4.16 .

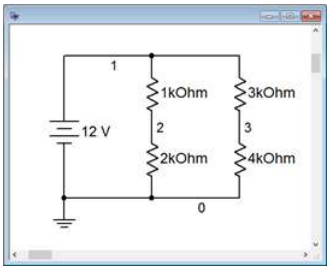


Figure 5.4.16 : The circuit of Example 5.4.5 in a simulator.

A DC operating point analysis is run and the results are shown in Figure 5.4.17 . The voltages are precisely as expected.

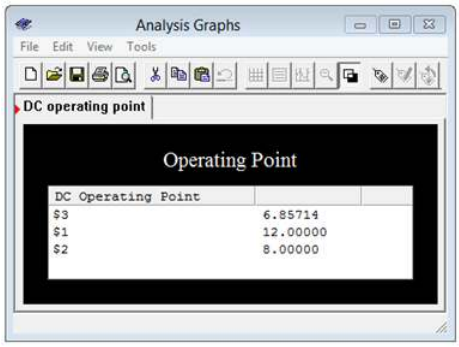


Figure 5.4.17 : Simulation results for the bridge circuit of Example 5.4.5 .

Final Comments

Series-parallel simplification techniques will not work for all circuits. Complex multi-source circuits and some resistive networks such as delta configurations or five-element bridge configurations require other techniques that will be addressed in later chapters.

References

[1]Or, if you prefer this situation posed as a question: When standing normally, is your head above your feet or are your feet below your head? Answer: Yes!

This page titled 2.4.4: Series-Parallel Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.4: Series-Parallel Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.4.5: Summary

In this chapter we have determined how to identify basic series-parallel networks driven by a single effective voltage or current source. A key element here is to identify sub-circuits or subgroups of resistors that are comprised of either series-only or parallel-only configurations within themselves. These groupings can then be reduced to equivalent resistances using series and parallel combination techniques examined in earlier chapters. This process may be repeated until the entire circuit is simplified down to either a single series loop or parallel arrangement of resistors driven by a voltage or current source.

Once a circuit has been simplified, series and parallel analysis techniques, and laws such as Ohm's law, KVL, KCL, VDR and CDR, may be employed to determine various voltages and currents in the simplified equivalent. Given these results, the circuit may be expanded back into its original form in stages, reapplying these rules and techniques to determine voltages and currents within the sub-circuits. The process may be iterated until every current and voltage in the original circuit is discovered, if desired. Once these values are determined, power calculations are trivial.

There is an infinite variety of series-parallel configurations and consequently no single solution technique will work for all of them. Indeed, the more complex the circuit, the more solution paths there are for that circuit. It is therefore wise to plan some strategy for a solution instead of randomly “diving in” in order to ease the ultimate effort.

Two series-parallel configurations of note are the ladder network and the bridge network. A resistive ladder may be used to derive several different currents or voltages from a single source; the R-2R variation being particularly useful in that it divides the source down by powers of two. A basic resistive bridge is comprised of four elements arranged as a pair of series resistors in parallel with another pair of series resistors. Bridges can be used as part of a measurement scheme. For example, one or more of the resistors could be environmentally sensitive, such as a photoresistor, force sensing resistor or thermistor. As the associated variable changes, the resistance of the sensor changes which unbalances the bridge, and the resulting voltage indicates both the magnitude and sign of said environmental change.

Review Questions

1. In general, describe the process of reducing a series-parallel resistive network down to a single equivalent resistance.
2. Do Ohm's law, KVL and KCL still apply in series-parallel networks? Why?
3. Is there a finite number of variations of series-parallel networks? Why/why not?
4. Describe the primary functional characteristic of an R-2R ladder network.
5. What is meant by the terms “load” and “loading”, as in “R_1 loads R_2”?
6. How might a bridge network be used to sense changes in temperature?
7. Describe a general procedure to find the voltage between two arbitrary points in a series-parallel circuit.

This page titled 2.4.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

2.4.6: Exercises

Analysis

1. In the circuit of Figure 5.6.1 , which individual resistors are strictly in series and which are in parallel?

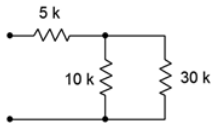


Figure 5.6.1

2. In the circuit of Figure 5.6.2 , which individual resistors are strictly in series and which are in parallel?

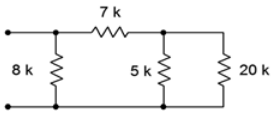


Figure 5.6.2

3. In the circuit of Figure 5.6.3 , which individual resistors are strictly in series and which are in parallel?

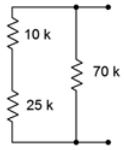


Figure 5.6.3

4. In the circuit of Figure 5.6.4 , which individual resistors are strictly in series and which are in parallel?

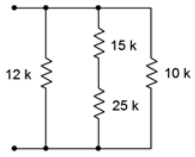


Figure 5.6.4

5. In the circuit of Figure 5.6.5 , which individual resistors are strictly in series and which are in parallel?

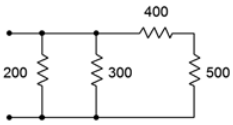


Figure 5.6.5

6. In the circuit of Figure 5.6.6 , which individual resistors are strictly in series and which are in parallel?

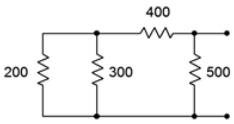


Figure 5.6.6

7. In the circuit of Figure 5.6.7 , which individual resistors are strictly in series and which are in parallel?

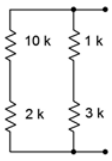


Figure 5.6.7

8. In the circuit of Figure 5.6.8 , which individual resistors are strictly in series and which are in parallel?

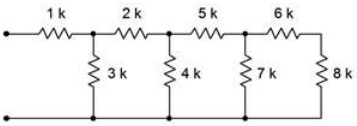


Figure 5.6.8

9. Determine the equivalent resistance of the network shown in Figure 5.6.1 (i.e., as if an ohmmeter is connected to the two terminals).

10. Determine the equivalent resistance of the network shown in Figure 5.6.2 .

11. Determine the equivalent resistance of the network shown in Figure 5.6.3 .

12. Determine the equivalent resistance of the network shown in Figure 5.6.4 .

13. Determine the equivalent resistance of the network shown in Figure 5.6.5 .

14. Determine the equivalent resistance of the network shown in Figure 5.6.6 .

15. Determine the equivalent resistance of the network shown in Figure 5.6.7 .

16. Determine the equivalent resistance of the network shown in Figure 5.6.8 .

17. For the circuit of Figure 5.6.9 , find voltages V_a, V_b and V_{ab}.

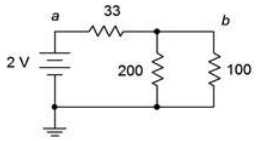


Figure 5.6.9

18. For the circuit of Figure 5.6.9 , find the current through each resistor.

19. For the circuit of Figure 5.6.10 , find the current through each resistor.

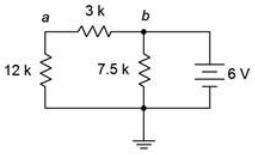


Figure 5.6.10

20. For the circuit of Figure 5.6.10 , find voltages V_a, V_b and V_{ab}.

21. For the circuit of Figure 5.6.11 , find voltages V_a, V_b and V_{ab}.

https://eng.libretexts.org/@go/page/52904

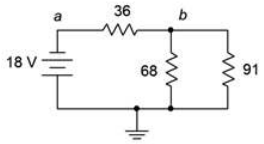


Figure 5.6.11

22. For the circuit of Figure 5.6.11 , find the current through each resistor.

23. For the circuit of Figure 5.6.12 , find the current through each resistor.

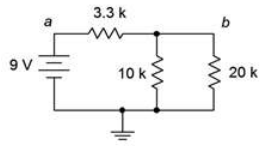


Figure 5.6.12

24. For the circuit of Figure 5.6.12 , find voltages V_a, V_b and V_{ab}.

25. In the circuit of Figure 5.6.13 , find voltages V_a, V_b and V_{ab}.

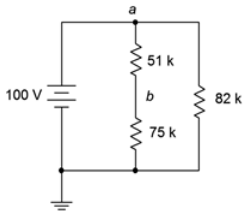


Figure 5.6.13

26. In the circuit of Figure 5.6.13 , find the current through each resistor.

27. In the circuit of Figure 5.6.14 , find the current through each resistor.

28. In the circuit of Figure 5.6.14 , find voltages V_a, V_b and V_{ab}.

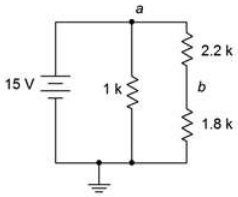


Figure 5.6.14

29. For the circuit of Figure 5.6.15 , find voltages V_b, V_c and V_{cb}.

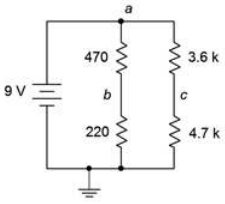


Figure 5.6.15

30. For the circuit of Figure 5.6.15 , find the current through the 470 Ω and 3.6 kΩ resistors along with the source current.

31. For the circuit of Figure 5.6.16 , find the current through the 2 kΩ and 5.1 kΩ resistors along with the source current.

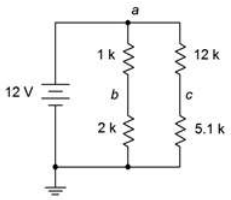


Figure 5.6.16

32. For the circuit of Figure 5.6.16 , find voltages V_c, V_b, V_{bc} and V_{ab}.

33. For the circuit of Figure 5.6.17 , find voltages V_a and V_b.

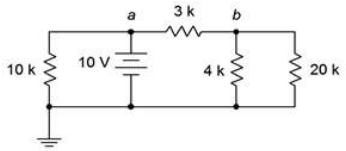


Figure 5.6.17

34. For the circuit of Figure 5.6.17 , find the current through the 10 kΩ and 4 kΩ resistors along with the source current.

35. For the circuit of Figure 5.6.18 , find the current through the 30 kΩ and the left-most 36 kΩ resistors along with the source current.

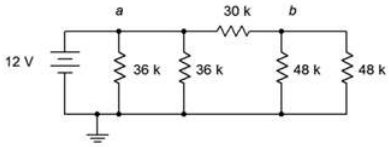


Figure 5.6.18

36. For the circuit of Figure 5.6.18 , find voltages V_a, V_b and V_{ab}.

37. In the circuit of Figure 5.6.18 , must the currents through the two 36 kΩ resistors be the same? Why/why not?

38. In the circuit of Figure 5.6.18 , must the currents through the two 48 kΩ resistors be the same? Further, must these currents be the same as the currents through the two 36 kΩ resistors Why/why not?

39. For the circuit of Figure 5.6.19 , find voltages V_a, V_b and V_{ab}.

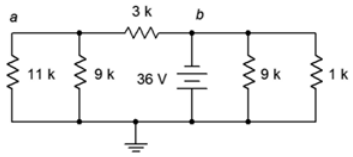


Figure 5.6.19

40. For the circuit of Figure 5.6.19 , find the current through the 3 kΩ, 11 kΩ, and both 9 kΩ resistors.

41. Given the circuit of Figure 5.6.20 , find the current through the 47 kΩ, 5.1 kΩ, and 3.9 kΩ resistors.

https://eng.libretexts.org/@go/page/52904

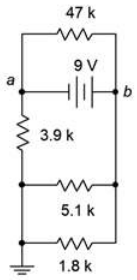


Figure 5.6.20

42. Given the circuit of Figure 5.6.20 , find voltages V_a, V_b and V_{ab}.

43. For the circuit of Figure 5.6.21 , find voltages V_b, V_c and V_d.

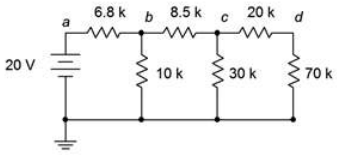


Figure 5.6.21

44. For the circuit of Figure 5.6.21 , find the current through the 6.8 kΩ, 8.5 kΩ, and 20 kΩ resistors.

45. The circuit of Figure 5.6.22 is called an R-2R ladder. Find the current through each of the 60 kΩ resistors.

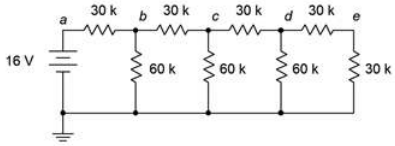


Figure 5.6.22

46. Given the circuit of Figure 5.6.22 , find voltages V_b, V_c, V_d and V_e. What is unique about this configuration?

47. For the circuit of Figure 5.6.23 , find voltages V_a, V_b and V_{ab}.

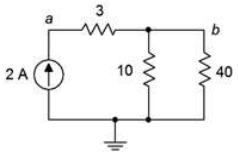


Figure 5.6.23

48. For the circuit of Figure 5.6.23 , find the current through each of the resistors.

49. For the circuit of Figure 5.6.24 , find the current through each of the resistors.

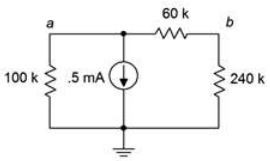


Figure 5.6.24

50. For the circuit of Figure 5.6.24 , find voltages V_a, V_b and V_{ab}.

51. Given the circuit of Figure 5.6.25 , find voltages V_a, V_b and V_{ab}.

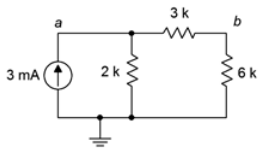


Figure 5.6.25

52. Given the circuit of Figure 5.6.25 , find the current through each of the resistors.

53. For the circuit of Figure 5.6.26 , find the current through the 200 Ω, 400 Ω, and 100 Ω resistors.

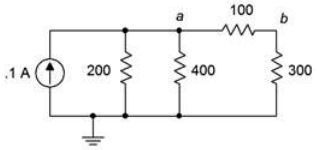


Figure 5.6.26

54. For the circuit of Figure 5.6.26 , find voltages V_a, V_b and V_{ab}.

55. Given the circuit of Figure 5.6.27 , find voltages V_a, V_b and V_{ab}.

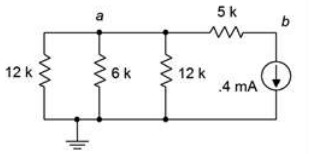


Figure 5.6.27

56. Given the circuit of Figure 5.6.27 , find the current through the 5 kΩ, and 6 kΩ resistors.

57. For the circuit of Figure 5.6.28 , find the current through the 9 kΩ and rightmost 82 kΩ resistors.

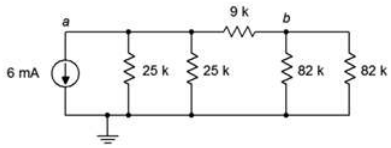


Figure 5.6.28

58. For the circuit of Figure 5.6.28 , find voltages V_a, V_b and V_{ab}.

59. Must the current through the 82 kΩ resistors be identical in the circuit of 4.8.6? Why/why not?

60. Must the current through the 25 kΩ resistors be identical in the circuit of 4.8.6? Must they be the same as the currents through the 82 kΩ resistors? Why/why not?

61. For the circuit of Figure 5.6.29 , find voltages V_a, V_b and V_{ab}.

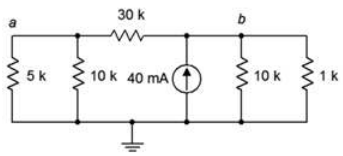


Figure 5.6.29

62. For the circuit of Figure 5.6.29 , find the current through the 30 kΩ, 5 kΩ, and 1 kΩ resistors.

63. For the circuit of Figure 5.6.30 , find the current through the 1 kΩ, 2.2 kΩ, and 18 kΩ resistors.

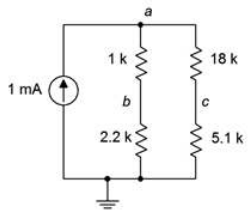


Figure 5.6.30

64. For the circuit of Figure 5.6.30 , find voltages V_a, V_b and V_c.

65. Given the circuit of Figure 5.6.31 , find voltages V_a, V_b and V_{ab}.

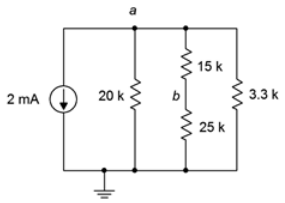


Figure 5.6.31

66. Given the circuit of Figure 5.6.31 , find the current through the 20 kΩ, 15 kΩ, and 3.3 kΩ resistors.

67. The circuit of Figure 5.6.32 is referred to as an R-2R ladder network. Find the current through each of the 4 kΩ resistors.

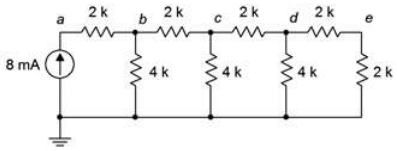


Figure 5.6.32

68. For the circuit of Figure 5.6.32 , find voltages V_a, V_b and V_{ab}. What is the unique characteristic of this configuration?

Design

69. Determine a new value for the 33 Ω resistor in Figure 5.6.9 such that the source current is 10 mA.

70. Determine a new value for the 20 kΩ resistor in Figure 5.6.12 such that V_b is 4 volts.

71. Determine a new value for the 470 Ω resistor in Figure 5.6.15 such that V_b is 6 volts.

72. Determine a new value for the 1 kΩ resistor in Figure 5.6.14 such that the source current is 20 mA.

73. Determine a new value for the 12 kΩ resistor in Figure 5.6.16 such that V_{bc} is 0 volts.

74. Given the circuit of Figure 5.6.24 , determine a new value for the 100 k Ω resistor such that V_a is 75 volts.

75. Given the circuit of Figure 5.6.23 , determine a new value for the current source such that V_b is 8 volts.

76. Determine a new value for the 2.2 kΩ resistor in Figure 5.6.30 such that V_{bc} is 0 volts.

77. Given the circuit of Figure 5.6.32 , determine a new value for the current source such that V_e is 1 volt.

Challenge

78. Utilizing only 1 kΩ resistors, create a series-parallel combination that achieves 1.25 kΩ of total resistance.

79. Utilizing only 12 kΩ resistors, create a series-parallel combination that achieves 9 kΩ of total resistance.

80. Consider the circuit of Figure 5.6.22 . Alter the values of the two right-most 30 kΩ resistors such that V_e is 1.2 volts. The remaining node voltages should be unchanged from the original circuit.

81. Alter the circuit shown in Figure 5.6.22 such that another “wrung” is added to the ladder creating a new right-most node f. If the former terminating 30 kΩ resistor (i.e., the vertical one) is reset to 60 kΩ, determine the values of the resistors on the new wrung such that V_f is 0.5 volts.

82. Given the circuit of Figure 5.6.25 , determine a new value for the 2 k Ω resistor such that V_b is 12 volts.

83. Given the circuit of Figure 5.6.32 , determine new values for the resistors such that all of the node voltages are twice the value of the original circuit's node voltages.

Simulation

84. Perform a DC simulation on the result of problem 24 to verify the node voltages.

85. Perform a DC simulation on the result of problem 25 to verify the node voltages.

86. Perform a DC simulation on the result of problem 43 to verify the node voltages.

87. Perform a DC simulation on the result of problem 46 to verify the node voltages.

88. Perform a DC simulation on the result of problem 51 to verify the node voltages.

89. Perform a DC simulation on the result of problem 68 to verify the node voltages.

90. Perform a DC simulation on the proposed solution to problem 69 to verify the new design.

91. Perform a DC simulation of the alteration presented in Challenge problem 80. Does the new design meet all of the requirements?

92. Perform a DC simulation of the alteration requested in Challenge problem 83. Does the new design meet all of the requirements?

This page titled 2.4.6: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

3: Circuit Simulations Using Computers

With the advent of low cost personal computers, there are many alternatives to hand sketching of plots. One method involves the use of commercial or public domain software packages designed for circuit analysis. One common package is the public domain program SPICE. SPICE is an acronym that stands for Simulation Program with Integrated Circuit Emphasis, and was originally written in the mid- 1970's by Dr. Laurence Nagel of the University of California. This program is available for many different computing platforms for minimal cost. SPICE also serves as the core for a number of commercial packages. The commercial versions generally add features such as schematic capture (the ability to "draw" circuits using the computer's mouse), graphical input and output of data, interactive analysis, analog-digital mixed signal analysis and large device libraries. Examples of popular simulation packages include the commercial offerings OrCAD PSpice and Electronics Workbench Multisim, while freeware packages include Linear Technology LTspice and Texas Instruments TINA-TI. The choice of a specific simulation tool depends on the needs of the user, available facilities, costs, and so forth. Of course, everyone has their own working style, so personal preference also plays a role. Generally, any quality SPICE-based simulator will be sufficient for the circuits presented in this book.

In order to use a simulator, a circuit is "described" with a special data file. For non-graphical simulators, the data file can be created using an ordinary text editor. The data file is then used by the simulation program to estimate the circuit response. Simulation results can include items such as Bode gain and phase plots.

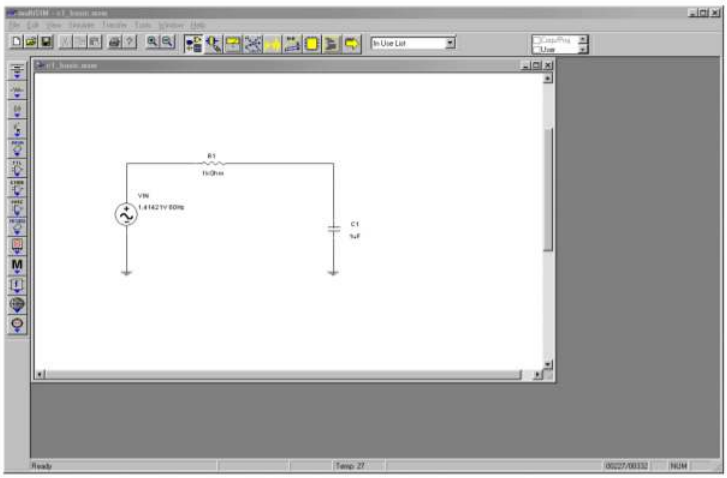


Figure 3.1a: Multisim schematic for a simple lag network.

With graphical input (AKA schematic capture), components are usually dragged onto the work area and interconnected through the use of a mouse or other pointing device. Consequently, text-based input files are not needed, although many programs can import them and create the circuit from there. Similarly, simulation results are generally shown in graphical form using a plotting window or virtual oscilloscope instead of using a text-based output file. Graphical input and output using Multisim is shown in Figure 3.1. Here a simple lag network is drawn on the worksheet. A Bode plot is then generated by selecting the AC Analysis option of Simulate.

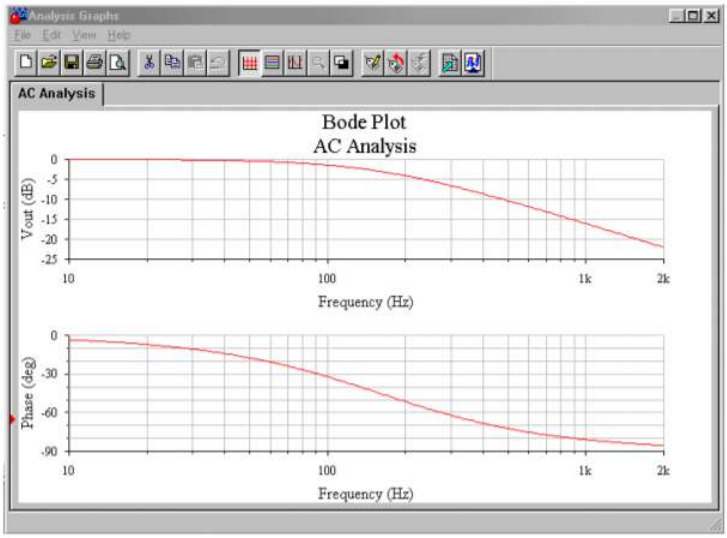


Figure 3.1*b*: Multisim gain and phase output graphs.

Note that accurate models for specific op amps or other devices that are not included in the simulator's libraries may be obtained from their manufacturer. This is often as simple as downloading them from the manufacturer's Web site. A listing of manufacturer's Web sites may be found in the Appendix.

Simulators are by no means small or trivial programs. They have many features and options, not to mention the variations produced by the many commercial versions. This text does not attempt to teach all of the intricacies of SPICE-based simulators. For that, you should consult your simulator user's manual, or one of the books available on the subject. The examples in this book assume that you already have some familiarity with computer circuit simulators.

We will be using simulations in the following chapters for various purposes. One thing that you should always bear in mind is that simulation tools should not be used in place of a normal "human" analysis. Doing so can cause no end of grief. Simulations are only as good as the models used with them. It is easy to see that if the description of the circuit or the components within the circuit is not accurate, the simulation will not be accurate. Simulation tools are best used as a form of doublechecking a design, not as a substitute for proper analysis.

This page titled 3: Circuit Simulations Using Computers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.5: Circuit Simulations Using Computers** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

CHAPTER OVERVIEW

4: Analysis Theorems and Techniques

Learning Objectives

After completing this chapter, you should be able to:

- Define whether or not a circuit is a linear bilateral network.
- Find the voltage source equivalent of a current source and vice versa.
- Compute voltages and currents in multi-source networks using superposition.
- Simplify networks using Thévenin's and Norton's theorems.
- Determine conditions for maximum power transfer and compute the maximum power.
- Utilize delta-Y and Y-delta conversions for circuit simplification and analysis.

4.1: Introduction
4.2: Source Conversions
4.3: Superposition Theorem
4.4: Thévenin's Theorem
4.5: Norton's Theorem
4.6: Maximum Power Transfer Theorem
4.7: Advanced Topic- Delta-Y Conversions
4.8: Summary
4.9: Exercises

This page titled 4: Analysis Theorems and Techniques is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

4.1: Introduction

As presented in the previous chapter, series-parallel analysis broadened our ability to analyze circuits beyond simple series and parallel networks, and leveraged the laws and techniques learned in earlier chapters. In this chapter we shall examine a number of theorems and techniques to help us analyze yet more complex circuits and address specialized applications. Specifically, we will address a method of analyzing circuits that contain multiple current and/or voltage sources that are connected in a non-trivial fashion (i.e., not just series voltage sources or parallel current sources). It is called the superposition theorem and can be applied to any circuit or parameter that meets certain requirements, including circuits that have both current sources and voltage sources together.

We shall also examine several ways of simplifying circuits or making functional equivalents for them. These are more tools to aid in the process of circuit simplification and analysis. The first item in this category is to make more accurate models of our idealized constant voltage and current sources. Once this is completed, it will be possible to convert from one type of source to another, such as creating a current source that is the functional equivalent of a voltage source. By this we mean that if we swap one for the other in any circuit, the remainder of the circuit will behave identically, producing the same component voltage drops and branch currents. This technique is useful in a number of ways, not the least of which is that it can help reduce more complex circuits to ease analysis.

The concept of equivalence can be extended beyond just a single source to an entire network. For this we will explore Thévenin's and Norton's theorems. Using these theorems, entire circuits utilizing dozens of components can be modeled as a single source with an associated resistance. When coupled with the maximum power transfer theorem, these theorems will allow us to determine component values that produce the maximum amount of load power.

Finally, we will examine how to find equivalent circuits for certain resistor arrangements that use three connecting points, in other words, resistor arrangements shaped like the letter Y or like a triangle. These are known as delta and Y configurations. These configurations are difficult to address with basic seriesparallel simplification techniques but the conversion equivalences will help solve that issue.

This page titled 4.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.2: Source Conversions

We begin by considering a more realistic model for constant voltage and constant current sources. The ideal voltage source produces a stated potential forever, and without regard to what it is connected to. The ideal current source behaves similarly; it will always produce the same current regardless of its load. These expectations are not realistic. For example, if we were to place a solid bar of copper across the terminals of a voltage source, the bar may exhibit a resistance of mere milliohms, implying an output current of thousands of amps. Similarly, if we were to disconnect a current source from any load, its effective load would be the resistance of the air between its terminals and Ohm's law would dictate an output voltage of perhaps thousands or even millions of volts. Real world sources do not behave like this.

Realistic Models for Sources

A highly precise model for any voltage source or current source could be fairly complex but for general purpose work, we can greatly improve our ideal sources by simply adding a resistor to them. This resistance is referred to as the internal resistance of the source. It is important to understand that this is not a resistor, as in an internal component that can be altered, but rather a mathematical addition to the source that better predicts how it will behave. Further, the value of this effective resistance can be deduced in a laboratory via proper measurements.

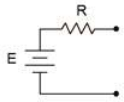


Figure 6.2.1 : Practical voltage source model.

The model for a voltage source adds a resistance in series, as shown in Figure 6.2.1 . This resistance sets an upper limit on the source's current output. Even if the output terminals are shorted, the maximum current will be dictated via Ohm's law to be the source voltage divided by the internal resistance, or E/R. Obviously, this internal resistance will create some voltage divider effect with the attached load. To minimize this effect, the internal resistance should be as small as practicably possible. Thus,

$$\text{The ideal internal resistance of a voltage source is zero ohms (a short).}$$

Given a zero ohm internal resistance, this improved model reverts back to the original ideal source. In the case of a lab power supply, the internal resistance typically will be a small fraction of an ohm.

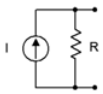


Figure 6.2.2 : Practical current source model.

For a current source, the improved model adds a resistance in parallel, as shown in Figure 6.2.2 . This resistance sets an upper limit on the source's voltage output. If the output terminals are opened, the maximum voltage will no longer produce a huge voltage. Instead, it is dictated by Ohm's law to be the source current times the internal resistance, or $I \cdot R$. This internal resistance will create some current divider effect with the attached load. To minimize this effect, the internal resistance should be as large as practicably possible. Thus,

$$\text{The ideal internal resistance of a current source is infinite ohms.}$$

Given an infinite ohm internal resistance (i.e., an open), this improved model reverts back to the original ideal source. From here on, whenever we deal with practical voltage and current sources, we understand that these sources have some associated internal resistance, even if they're not shown explicitly in a schematic diagram. Further, whenever we talk about ideal sources, we simply use a short for the internal resistance of a voltage source and an open for the internal resistance of a current source.

Source Equivalences

For any simple voltage source consisting of an ideal voltage source with a series internal resistance, an equivalent current source may be created. Similarly, for any simple current source consisting of an ideal current source with a parallel internal resistance, an equivalent voltage source may be created. By “equivalent”, we mean the load currents must be identical for both circuits given any value of load resistance. (Note that if the currents are the same then the voltages must also be the same due to Ohm’s Law.) Consider the simple voltage source of Figure 6.2.1 . Its equivalent current source would be that shown in Figure 6.2.2 . The opposite is also true.

For reasons that will become apparent under the section on Thévenin's theorem following, the internal resistances of these two circuits must be identical if they are to behave identically. Knowing that, it is a straightforward process to find the required value of the other source. As the current/voltage characteristic is linear for these circuits, a plot line can be defined by just two points. The two obvious points to use are the opened and shorted load cases. In other words, if it's equivalent for these two situations, it must work for any load. The shorted load case produces the maximum load current with zero load voltage, while in contrast, the opened load case produces the maximum load voltage with zero load current.

Given a voltage source, the maximum current is developed when the load is shorted producing a current of E/R. Under that same load condition, all of the current from the current source version must be flowing through the load. Therefore, the value of the equivalent current source must be the maximum current of E/R. It would be nonsensical to use a current source that was larger or smaller than this value.

To continue, if we look at the open load case, for the voltage source the load current would be zero and the load voltage would be the entire source voltage of E. For the current source, the load would also see no current and its voltage would be the voltage appearing across its internal resistance which is R times the current E/R, or just E. Thus, the two behave identically at the load limits.

Similarly, if we start with a current source, an open load produces the maximum load voltage of $I \cdot R$. Therefore, the equivalent voltage source must have a value of $I \cdot R$. For the current source, a shorted load would produce a load current equal to the source value, or I. The voltage source version would produce a current of E/R, where the value of E was just found to be equal to $I \cdot R$, and thus the load current would be $I \cdot R/R$, or just I. Once again, the two versions behave identically at the load limits.

To summarize the process of source conversion:

- The internal resistance will be the same for both versions.
- If converting from a voltage source to a current source, the value of the current source will be the maximum current available from the voltage source (shorted load case), and is equal to E/R.
- If converting from a current source to a voltage source, the value of the voltage source will be the maximum voltage available from the current source (opened load case), and is equal to $I \cdot R$.

If a multi-source is being converted (i.e., voltage sources in series or current sources in parallel), first combine the sources to arrive at the simplest source and then do the conversion. Do not convert the sources first and then combine them as you will wind up with series-parallel configurations rather than simple sources.

Judicious use of source conversions can sometimes simplify multi-source circuits by allowing converted sources to be combined, resulting in a single source.

Example 6.2.1

Determine the current source equivalent for the source of Figure 6.2.3 .

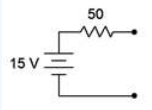


Figure 6.2.3 : Source for Example 6.2.1 .

First off, the resistance value is simply copied over, therefore the internal resistance of the current source is 50 Ω. The value of the current source is computed using Ohm's law based on the maximum current produced under the shorted load case. Thus, all 15 volts drops across the 50 Ω resistance.

$$I_s = \frac{E}{R_s}$$

$$I_s = \frac{15V}{50\Omega}$$

$$I_s = 0.3A$$

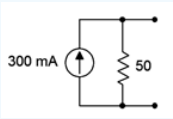


Figure 6.2.4 : Equivalent current source for the source of Figure 6.2.3 .

The equivalent current source is shown in Figure 6.2.4 . We know that this will work for the shorted an opened cases, but if any doubt is left as to its universal nature, simply substitute any other resistance value and compare the outcomes of the two circuits. For no particular reason, let's try using a load of 200 Ω and see if the load currents are identical.

For the original voltage source we can use Ohm's law:

$$I_L = \frac{E}{R_s + R_L}$$

$$I_L = \frac{15V}{50\Omega + 200\Omega}$$

$$I_L = 60mA$$

For the equivalent current source we can use CDR:

$$I_L = I_s \frac{R_s}{R_s + R_L}$$

$$I_L = 0.3A \frac{50\Omega}{50\Omega + 200\Omega}$$

$$I_L = 60mA$$

Try this with any other load resistor. The results should always be identical.

Now let's try one going the other way.

Example 6.2.2

Determine the voltage source equivalent for the source of Figure 6.2.5 .

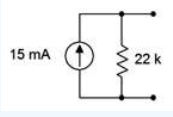


Figure 6.2.5 : Source for Example 6.2.2 .

Once again, the resistance value is simply copied over, therefore the internal resistance of the voltage source is 22 kΩ. The value of the voltage source is based on the maximum voltage produced under the opened load case and is computed using Ohm's law. For the open case, all 15 milliamps flows through the 22 kΩ resistance.

$$E_s = I \times Rs$$

$$E_s = 15mA \times 22k\Omega$$

$$E_s = 330V$$

The equivalent voltage source is shown in Figure 6.2.6 . Again, let's try using another value of load resistor to see if the load current and voltage results are identical between the two sources. This time, we'll match the load to the internal resistance of 22 kΩ.

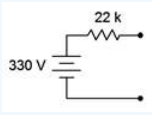


Figure 6.2.6 : Equivalent current source for the source of Figure 6.2.5 .

For a voltage source with matched resistances we wind up with a simple 50% voltage divider, thus the load voltage will be half the source voltage, or 165 volts. The original current source sees the current split in half due to the current divider rule. Thus, the load current should be 7.5 mA. Given this current, the load voltage will be,

$$V_L = I \times Rs$$

$$V_L = 7.5mA \times 22k\Omega$$

$$V_L = 165V$$

Once again the results turn out to be identical.

Now that we have it is possible to substitute one kind of source for another, it is time to investigate how we might put this to use beyond just giving us a different way of driving a circuit. If applied intelligently, source conversions can be used to simplify and reduce complex circuits, and thus ease computational difficulty. For example, consider the circuit of Figure 6.2.7 .

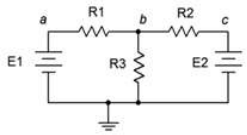


Figure 6.2.7 : A dual source circuit.

This circuit is unlike any circuit we have seen so far. Although we have analyzed circuits using multiple voltage sources, they have always been in a simple series loop. As such, their voltages may be added together to find a single equivalent voltage source. Such is not the case here. In this circuit the voltage sources are part of a series-parallel network and thus their potentials cannot be merely added. In fact, there are no further simplifications to be performed on this circuit using basic series-parallel techniques. We appear to be stuck.

But we're not. This circuit can be simplified into a straight all-parallel network through the use of source conversions. The E_1, R_1 combo can be converted into one current source while the E_2, R_2 combo can be converted into a second source. Once these are converted, the resulting circuit will consist of two current sources and three resistors, all in parallel. This is the sort of circuit we solved back in Chapter 4.

✔ Example 6.2.3

Determine V_b for the circuit of Figure 6.2.8 .

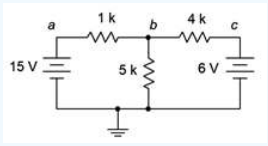


Figure 6.2.8 : Circuit for Example 6.2.3 .

The first step will be to convert the voltage sources into current sources. We will treat the resistors attached to their positive terminals as their internal resistances. In other words, we have a 15 volt source with 1 kΩ resistance and a 6 volt source with a

4 kΩ resistance.

For the first source, the current will be:

$$I_s = \frac{E}{R_s}$$

$$I_s = \frac{15V}{1k\Omega}$$

$$I_s = 15mA$$

And for the second source:

$$I_s = \frac{E}{R_s}$$

$$I_s = \frac{6V}{4k\Omega}$$

$$I_s = 1.5mA$$

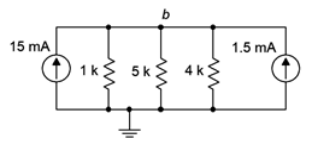


Figure 6.2.9 : Equivalent current source version of the circuit depicted in Figure 6.2.8 .

The equivalent converted circuit is shown in Figure 6.2.9 . Before continuing, it is worth noting that connection nodes a and c no longer exist in this circuit. More on this in a moment. This new circuit consists of a pair of current sources that sum to 16.5 mA and which drive three parallel resistors, 1 kΩ || 4 kΩ || 5 kΩ, or approximately 689.7 Ω. Ohm's law tells us V_b is:

$$V_b = I_{Total} \times R_{Equivalent}$$

$$V_b \approx 16.5mA \times 689.7\Omega$$

$$V_b = 11.38V$$

We can now take this voltage and apply it back into the original (unconverted) circuit. With this potential known, it is relatively easy to determine the other currents and voltages using KVL and Ohm's law. Looking at the 1 kΩ resistor, the voltage across it must be 15 V − 11.38 V, or 3.62 V. Therefore, the current through it must be 3.62 mA. Similarly, the voltage across the 4 kΩ resistor must be 11.38 V − 6 V, or 5.38 V, which yields a current of 1.345 mA. Both of these currents are flowing left to right. Then third current, flowing down through the 5 kΩ, is 11.38 V/5 kΩ, or 2.276 mA. KCL states that the current entering node b must equal the currents leaving. The entering current is 3.62 mA. The exiting currents are 1.345 mA and 2.276 mA, or 3.62 mA when rounded to three places (like the entering current).

As mentioned, nodes a and c disappeared in the converted circuit in Example 6.2.3 . This brings up an important point. Equivalent circuits are equivalent in terms that the items connected to the equivalent behave the same way as they do with the original circuit. That does not mean that the items within the equivalent see the same current or voltage. We do not expect the voltage across the 1 k Ω or 4 kΩ resistors in the converted version to be the same as those in the original version.

This page titled 4.2: Source Conversions is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.2: Source Conversions** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.3: Superposition Theorem

As useful as the source conversion technique proved to be in Example 6.2.3, it will not work for all circuits. Thus, more general approaches are needed. One of these methods is superposition.

Superposition allows the analysis of multi-source series-parallel circuits. Superposition can only be applied to networks that are linear and bilateral. Further, it cannot be used to find values for non-linear functions, such as power, directly. Fortunately, if the circuit contains nothing but resistors, and ordinary voltage sources and current sources, the circuit will be a linear bilateral network. Also, although power is a square law function (i.e., it is proportional to the square of voltage or current), it can be computed from the resulting voltage or current values so this presents no limits to analysis.

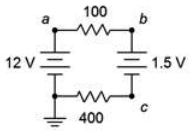


Figure 6.3.1 : A dual source circuit.

The basic idea is to determine the contribution of each source by itself, and then adding the results to get the final answer(s). Let's start with an almost trivial example to illustrate the idea. Consider the circuit of Figure 6.3.1 . Here we have two voltage sources driving a pair of resistors. It is a basic series circuit. The way this was approached in Chapter 3 was to combine the sources and the resistors into a single source and resistor. The circulating current could then be found using Ohm's law. As the two sources are in opposition, the net voltage is 10.5 volts while the total resistance is 500 Ω. This yields a circulating current of 21 mA.

Instead of this approach, let's consider the contribution of each source by itself. To do that, we will replace all other sources in the circuit with their ideal internal resistance. From earlier work we have discovered that the ideal internal resistance of a voltage source is a short, or zero ohms. Thus, we wind up with two circuits: one with a 12 volt source that drives 100 Ω, 400 Ω and a short; and a second circuit with a 1.5 volt source that drives pretty much the same thing, but with an opposing current direction. The first circuit produces a clockwise current of 12 V/ 500 Ω, or 24 milliamps. Meanwhile, the second circuit produces a counterclockwise current of 1.5 V / 500 Ω, or 3 mA. As these two currents oppose each other, the resulting current is 24 mA − 3 mA, or 21 mA, the same value originally computed. Granted, this second method is less efficient than the original method but it illustrates the process. More importantly, this process can be applied to a variety of multi-source series-parallel circuits. A key element is to remember the directions of the currents and polarities of the voltages created in the sub-circuits. Without these data, it will be impossible to determine whether the various pieces of the puzzle are added to or subtracted from the total.

To summarize the superposition technique:

- For every voltage or current source in the original circuit, create a new subcircuit. The sub-circuits will be identical to the original except that all sources other than the one under consideration will be replaced by their ideal internal resistance. This means that all remaining voltage sources will be shorted and all remaining current sources will be opened.
- Label the current directions and voltage polarities on each of the new subcircuits, as generated by the source under consideration.
- Solve each of the sub-circuits for the desired voltages and/or currents using standard series-parallel analysis techniques. Make sure to note the voltage polarities and current directions for these items.
- Add all of the contributions from each of the sub-circuits to arrive at the final values, being sure to account for current directions and voltage polarities in the process.

In order to get a better handle on the superposition technique, let's reexamine the the dual source circuit shown in Figure 6.2.8 (repeated in Figure 6.3.2 for ease of reference). We will solve this using superposition. As the circuit has two sources, it will require two sub-circuits.

Example 6.3.1

Determine V_b for the circuit of Figure 6.3.2 using superposition.

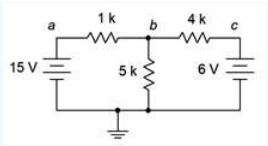


Figure 6.3.2 : Circuit for Example 6.3.1 .

As this circuit has two voltage sources, two sub-circuits will be needed. The first sub-circuit will use the 15 volt source. Consequently, the 6 volt source will be replaced with its ideal internal resistance, a short. This new circuit is shown in Figure 6.3.3 .

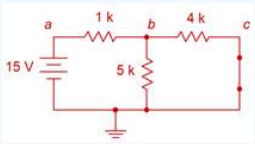


Figure 6.3.3 : First sub-circuit for the circuit of Figure 6.3.2 .

The current directions are as follows: current exits the source and travels through the 1 kΩ producing a voltage drop + to − from left to right. At node b the current splits. The 4 kΩ and 5 kΩ are in parallel so they both see the same voltage. Some of the current flows down through the 5 kΩ resistor producing a voltage drop + to − from top to bottom. The remainder of the current flows to the right, through the 4 kΩ, producing a voltage drop + to − from left to right. V_b can be determined via voltage divider between the 1 kΩ and the parallel combo of the 5 kΩ and 4 kΩ. 5 kΩ in parallel with 4 kΩ is approximately 2.222 kΩ.

$$V_b = E\frac{R_x}{R_x + R_y}$$

$$V_b = 15V\frac{2.222k\Omega}{2.222k\Omega + 1k\Omega}$$

$$V_b \approx 10.34V$$

The second sub-circuit will use the 6 volt source. Therefore, the 15 volt source will be replaced with its ideal internal resistance, a short. This new circuit is shown in Figure 6.3.4 .

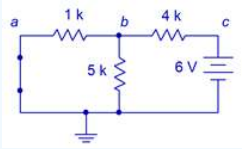


Figure 6.3.4 : Second sub-circuit for the circuit of Figure 6.3.2 .

The current directions are as follows: current exits the source and travels through the 4 kΩ producing a voltage drop + to − from right to left. At node b the current splits. Now the 1 kΩ and 5 kΩ are in parallel so they both see the same voltage. Some of the current flows down through the 5 kΩ resistor producing a voltage drop + to − from top to bottom, just as it did in the first sub-circuit. The remainder of the current flows to the left, through the 1 kΩ, producing a voltage drop + to − from right to left. Once again, V_b can be determined via voltage divider, this time between the 4 kΩ and the parallel combo of the 5 kΩ and 1 kΩ is approximately 833.3 Ω.

$$V_b = E\frac{R_x}{R_x + R_y}$$

$$V_b = 6V\frac{833.3\Omega}{833.3\Omega + 4k\Omega}$$

$$V_b \approx 1.034V$$

Both sub-circuits show polarities of + to − from top to bottom for V_b, therefore these two voltages simply add, for a total of approximately 11.37 volts. This agrees nicely with the result obtained using source conversions. The slight deviation in the

final digit is no doubt due to the rounding of intermediate values, such as resistor combinations.

It is instructive to pursue this example further and investigate a current. In the source conversion version it was discovered that the current through the 1 kΩ resistor is 3.62 mA and flowing left to right. Using superposition, the corresponding current in the first sub-circuit can be found using KVL and Ohm's law, (15 V − 10.34 V)/ 1 kΩ, or 4.66 mA flowing left to right. In the second sub-circuit, this current can be obtained using Ohm's law alone, 1.034 V / 1 kΩ, or 1.034 mA flowing right to left. As this current opposes the current from the first sub-circuit, the two values are subtracting leaving approximately 3.63 mA flowing left to right, agreeing with the original answer within rounding error.

Computer Simulation

For further verification, the circuit of Example 6.3.1 is entered into a simulator as shown in Figure 6.3.5 .

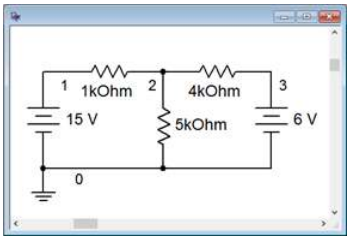


Figure 6.3.5 : The circuit of Example 6.3.1 in a simulator.

A DC operating point analysis is performed, as shown in Figure 6.3.6 .

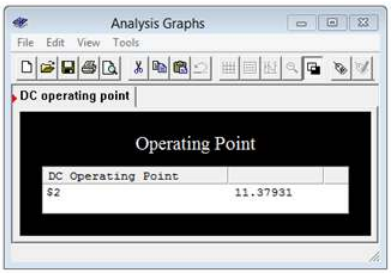


Figure 6.3.6 : Simulation results for the circuit of Example 6.3.1 .

The voltage at node 2, which is V_b in the original circuit, agrees nicely with the values computed previously.

This exploration shows that the superposition technique offers a fairly straightforward method of solving a variety of multi-source series-parallel circuits. It is not without its limitations, though. First, if a circuit has a large number of sources, superposition can become a bit tedious as it requires as many sub-circuits as there are sources, and every one of these sub-circuits requires separate solutions. Second, there are some series-parallel configurations that superposition along with seriesparallel analysis techniques will not solve. We will investigate methods of dealing with these issues later in this chapter and in the following chapter.

This page titled 4.3: Superposition Theorem is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.3: Superposition Theorem** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.4: Thévenin's Theorem

Thévenin's theorem, named after Léon Charles Thévenin, is a powerful analysis tool. For DC, it states:

> Any single port linear network can be reduced to a simple voltage source, E_{th}, in series with an internal resistance, R_{th}.

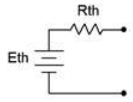


Figure 6.4.1 : Thévenin equivalent circuit.

An example is shown in Figure 6.4.1 . The phrase “single port network” means that the circuit is cut in such a way that only two connections exist to the remainder of the circuit (two connection points makes up one port). The remainder of the circuit may be a single component or a large multi-component sub-circuit. As there are many ways to cut a typical circuit, there are many possible Thévenin equivalents. The important thing is that there are only two connection points between the two portions of the circuit and neither point has to be ground.

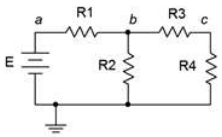


Figure 6.4.2 : Example circuit.

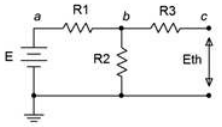


Figure 6.4.3 : Finding E_{th}..

Consider the circuit shown in Figure 6.4.2 . Suppose we cut the circuit immediately to the left of R_4. That is, we will find the Thévenin equivalent that drives R_4. The first step is to make the cut, removing the remainder of the circuit (in this case, just R_4). We then determine the open circuit output voltage. This is the maximum voltage that could appear between the cut points and is called the Thévenin voltage, E_{th}. This is shown in Figure 6.4.3 . In a circuit such as this, basic series-parallel analysis may be used to find E_{th}. This process turns out to be quite straightforward in this particular circuit. Due to the open, no current flows through R_3, thus no voltage is developed across R_3, and therefore E_{th} must equal the voltage developed across R_2 which may be obtained via a voltage divider with resistor R_1 and source E.

The second part is finding the Thévenin resistance, R_{th}. Beginning with the “cut” circuit, replace all sources with their ideal internal resistance (thus shorting voltage sources and opening current sources). From the perspective of the cut point, look back into the circuit and simplify by making appropriate series and parallel combinations to determine its equivalent resistance. This is shown in Figure 6.4.4 . Looking in from where the cut was made (right-to-left here), we find that R_1 and R_2 are in parallel, and this combination is then in series with R_3. Thus, $R_{th} = R_3 + (R_1||R_2)$. A common error is to determine the equivalent resistance that the source drives. Remember, everything is determined from the vantage point of the cut or port.

Thévenin equivalents are not limited to single source circuits. It is possible to find the equivalent of a network with several sources. Finding the open circuit output voltage will undoubtedly require some extra work, for example the use of superposition of source conversions. Finding the Thévenin resistance is unchanged, just remember to replace every source with its ideal internal resistance before simplifying the network.

As noted earlier, the original circuit could be cut in a number of different ways. We might, for example, want to determine the Thévenin equivalent that drives R_2 in the circuit of Figure 6.4.2 . This cut appears in Figure 6.4.5 . Clearly, this will result in different values for both E_{th} and R_{th}. For example, R_{th} is now $R_1||(R_3 + R_4)$.

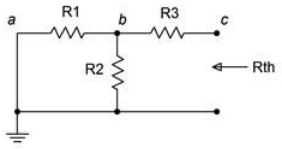


Figure 6.4.4 : Finding R_{th}.

Finding Thévenin Equivalents in Lab

The procedure for experimentally determining an equivalent in the laboratory mimics the analytical approach. The first step is to figuratively cut the circuit and isolate the section that is to be converted. At this point we have the open circuit version of the circuit and all we need do is connect a multimeter to the open port to determine E_{th}.

There are two methods for determining R_{th}. The first method works well for simple circuits that use only resistors and current and/or voltage sources. The second method is more generally applicable and will work for circuits with active devices such as transistors. To use the first method, the sources are physically removed from the circuit and replaced with their ideal internal resistance. Thus voltage sources are replaced with a shorting wire and current sources are left as opens. Do not simply place shorting wires across the terminals of voltage sources as doing so will cause an overload and potentially damage the sources. Once this is completed, a multimeter is placed at the open port and set to read resistance. The indicated value is R_{th}.

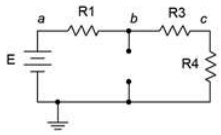


Figure 6.4.5 : An alternate cut or port location.

The second method uses a variable resistance, namely either a rheostat or a decade box, and exploits the voltage divider rule. In this version, the sources are left active (powered up) and are not replaced with opens or shorts. Once E_{th} is measured, the variable resistor is placed across the port connections. This resistance is adjusted until the port voltage drops to exactly half of E_{th}. In the equivalent circuit, there are only two resistances of concern, R_{th} and this variable load resistance, and they are in series. Consequently, if the load voltage is now half of the open circuit voltage (E_{th}), then the other half of the voltage must be dropping across the equivalent internal resistance (R_{th}). For this to be true in a series circuit, the two resistances must have the same value. Thus, we simply remove the rheostat from the circuit and use a multimeter to determine its precise value. If a decade box is used instead, the value is determined directly by reading the knob settings.

Whether determined analytically or empirically, the Thévenin equivalent circuit can replace the original single port network regardless of what the original was connected to. The same voltages and currents will be seen in this other portion, and it won't matter if the other portion is comprised of a single resistor, multiple resistors, multiple resistors and multiple sources, or even multiple resistors and sources wired to a selection of piquant cheeses. The Thévenin equivalent is a true functional equivalent and can be used on any linear bilateral network.

Example 6.4.1

Determine the Thévenin equivalent of the circuit driving the 1 kΩ in Figure 6.4.6 . Verify that the equivalent produces the same voltage across this resistor as the original circuit.

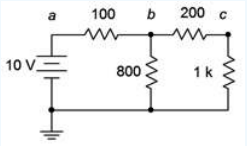


Figure 6.4.6 : Circuit for Example 6.4.1 .

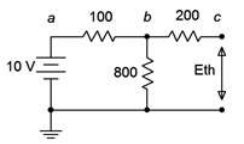


Figure 6.4.7 : Finding E_{th}.

First, we'll redraw the circuit showing the portion to be Théveninized, as shown in Figure 6.4.7 . The open circuit output voltage will be the voltage across the 800 Ω resistor as there will be no voltage drop across the 200 Ω resistor (as no current flows through it into the open). This is found via a simple voltage divider.

$$E_{th} = E\frac{R_x}{R_x + R_y}$$

$$E_{th} = 10V\frac{800\Omega}{800\Omega + 100\Omega}$$

$$E_{th} \approx 8.889V$$

The equivalent resistance is found by shorting the voltage source and then simplifying the circuit. The result is 200 Ω in series with the parallel combination of the 100 Ω and 800 Ω. $200 + 100||800 \approx 288.89\Omega$.

To determine Vc, we can use a voltage divider between the 1 kΩ and the 288.89 Ω along with the equivalent source voltage of 8.889 volts.

$$V_c = E_{th}\frac{R_L}{R_L + R_{th}}$$

$$V_c = 8.889V\frac{1k\Omega}{1k\Omega + 288.89\Omega}$$

$$V_c \approx 6.897V$$

Now let's determine V_c in the original circuit using series-parallel analysis techniques. Perhaps the quickest approach is a pair of voltage dividers. V_b is found via a divider between the parallel combination of the 1 kΩ + 200 Ω and the 800 Ω, against the 100 Ω. V_c is is then found using that voltage with a divider between the 1 kΩ and 200 Ω. Using this approach, V_b is approximately 8.276 volts and V_c is approximately 6.897 volts, providing an excellent match.

It might seem that the Thévenin method is the “long way home” in this example, and it is, but it has the advantage of being more efficient if several different loads are being considered. For example, suppose we decided to determine the output voltage not just for the 1 kΩ, but for a group of a half dozen different resistors. The double voltage divider would have to be determined for each load resistor using the straight series-parallel method but only a single divider needs to be computed for each load when using the Thévenin method. The Thévenin method is also of great use when determining maximum power transfer, as we shall see a later in this chapter.

Computer Simulation

To verify the results of Example 6.4.1 , the original circuit is recreated in a simulator along its Thévenin equivalent, as shown in Figure 6.4.8 .

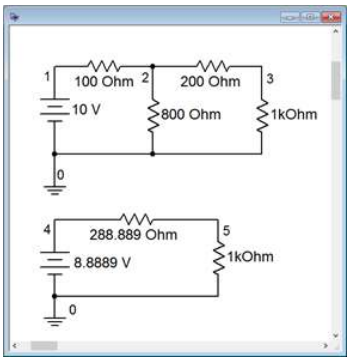


Figure 6.4.8 : The original circuit of Example 6.4.1 in a simulator along with the equivalent.

A DC operating point analysis is performed and the results are shown in Figure 6.4.9 . Note that the voltages across the identical 1 kΩ loads (nodes 3 and 5) are virtually the same, indicating functional equivalence between the two. The slight deviation is due to the rounding of the Thévenin voltage and resistance.

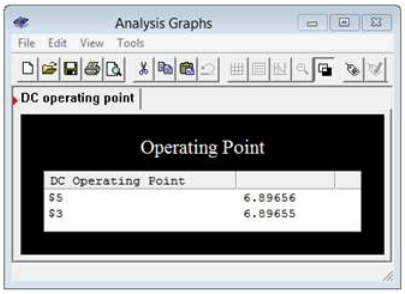


Figure 6.4.9 : Results of the simulation showing equivalence.

As a further check, the simulation is run again, but this time using an alternate load resistor value. The value chosen is the Thévenin resistance value. By matching the resistances, this should produce a 50/50 voltage divider and the load voltage should equal half of the Thévenin source voltage, or approximately 4.444 volts. That is precisely what we find, as seen in Figure 6.4.10 .

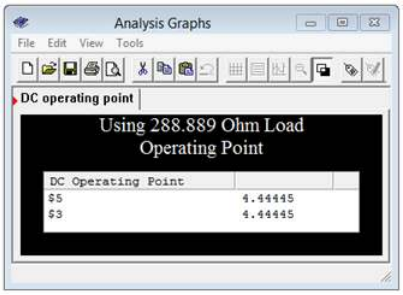


Figure 6.4.10 : Results of the simulation for matched resistance.

As mentioned previously, Thévenin's theorem can be applied to much more complex multi-source circuits, and the item being driven need not be just a single resistor. This is illustrated in the next example.

Example 6.4.2

Determine the Thévenin equivalent of the circuit driving the resistor/voltage source combo in Figure 6.4.11 . Verify that the equivalent produces the same voltage across this resistor as the original circuit.

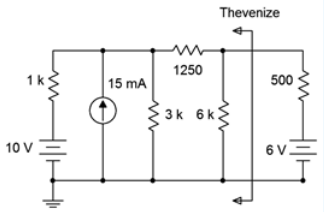


Figure 6.4.11 : Circuit for Example 6.4.2 .

Determining R_{th} is not particularly difficult here. After shorting the 10 volt source and open the current source, it can be seen that the 1 kΩ and 3 kΩ are in parallel, yielding 750 Ω. This is in series with the 1250 Ω for a total of 2 kΩ, which is in turn in parallel with the 6 kΩ. The final result is 1.5 kΩ.

Finding E_{th} is a little more involved. One possibility is to use a source conversion on the 10 volt source as that will leave us with two parallel current sources that may be combined directly. The converted source will be 10 V/ 1kΩ, or 10 mA, in parallel with 1 kΩ. This results in a total of 25 mA feeding upwards with 1 kΩ || 3 kΩ, or 750 Ω. We can use the current divider rule to determine the current flowing through the 1250 Ω plus 6 kΩ branch, and then use Ohm's law to find the open circuit voltage (i.e., the voltage across the 6 kΩ).

$$I_{6k} = I_S \frac{R_x}{R_x + R_y}$$

$$I_{6k} = 25mA \frac{750\Omega}{750\Omega + 7250\Omega}$$

$$I_{6k} = 2.34375mA$$

$$E_{th} = I_{6k} \times R_{6k}$$

$$E_{th} = 2.34375mA \times 6k\Omega$$

$$E_{th} = 14.0625V$$

Another approach to determine this value would be to use superposition. This is left as an additional exercise.

Turning our attention to the voltage produced across the 6 volt with 500 Ω sub-circuit, we have a simple series loop with E_{th} opposing this 6 volt source, leaving 8.0625 volts to produce a clockwise current through R_{th} in series with the 500 Ω (a total of 2 kΩ). That current will be 8.0625 V / 2 kΩ, or 4.03125 mA. This will produce a drop across the 500 Ω of 2.015625 volts with a polarity of + to − from top to bottom. This will add to the 6 volt source resulting in a final potential of 8.015625 volts. The results are verified via the simulation technique used previously. The node voltages for both the original and equivalent circuits are shown in Figure 6.4.12 .

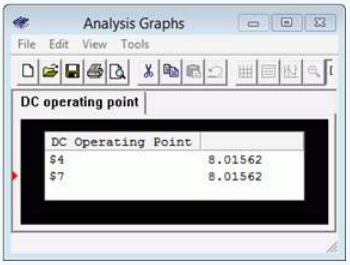


Figure 6.4.12 : Results of the simulation using the original and equivalent circuits.

This page titled 4.4: Thévenin's Theorem is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.4: Thévenin's Theorem** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.5: Norton's Theorem

Norton's theorem is credited to Edward Lawry Norton. In a nutshell, Norton's theorem is the current source version of Thévenin's theorem. That is, a single port DC network can be reduced to a single current source, I_N, with parallel internal resistance, R_N. Indeed, once you understand one of them, the other is but a minor extension, so we will not need to spend a great deal of time here once the basics are outlined sufficiently.

The process of determining the Norton equivalent current and resistance is very similar to that employed for the Thévenin equivalent. First, the Norton resistance is found the same way as is the Thévenin resistance: replace all sources with their ideal internal resistance and then perform appropriate series and parallel combinations to reduce this to a single resistance value. Consequently, the Norton and Thévenin resistance values are identical, $R_N = R_{th}$. Second, instead of finding the open circuit output voltage, we find the short circuit output current. This is the Norton current. Instead of thinking in terms of a voltmeter at the opened load, we think in terms of connecting a shorting ammeter across the load. Either way, we're looking for the maximum value that can be obtained.

Perhaps the most useful thing to remember here is that if we can create a Thévenin equivalent for a network then it must be possible to create a Norton equivalent. Indeed, once a Thévenin equivalent is found, a source conversion can be performed on it to yield the Norton equivalent! The opposite, of course, is also true.

This page titled 4.5: Norton's Theorem is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.5: Norton's Theorem** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.6: Maximum Power Transfer Theorem

Given a simple voltage source with internal resistance, a useful question to ask is “What value of load resistance will yield the maximum amount of power in the load?” While it is not true that maximizing load power is a goal of all circuit designs, it is a goal of a portion of them and thus worth a closer look. Consider the basic circuit depicted in Figure 6.6.1 with source E, source internal resistance R_i and load resistance R.

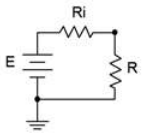


Figure 6.6.1 : Defining maximum power transfer.

We would like to describe the load power in terms of the load resistance. To make the job easier, we may normalize the voltage source E to 1 volt and the source resistance R_i to 1 Ohm. By doing this, R also becomes a normalized value, that is, it no longer represents a simple resistance value but rather represents a ratio in comparison to R_i. In this way the analysis will work for any set of source values. Note that the value of E will equally scale the power in both R_i and R, so a precise value is not needed, and thus, we may as well chose 1 volt for convenience.

The power in the load can be determined by using I^2R where $I = E/(R_i + R)$. Using our normalized values $I = 1/(1 + R)$ and thus the load power is:

$$P = \left(\frac{1}{1+R}\right)^2 R \text{ or after expanding, } P = \frac{R}{R^2+2R+1}$$

We now have an equation that describes the load power in terms of the load resistance. Before we go any further, take a look at what this equation tells you, in general. It is obvious that maximum power will not occur at the extremes. If $R = 0$ or $R = \infty$ (i.e., shorted or opened load) the load power is zero. While shorting the load will yield the maximum load current, it will also yield zero load voltage and thus no power. Similarly, opening the load will yield maximum load voltage but it will also yield zero load current, and again, no load power. To find the precise value that produces the maximum load power, the proof can be divided into two major portions. The first involves graphing the function and the second requires differential calculus to solve for a precise value. We shall proceed with the graphing portion which will lead us to the answer. The more rigorous proof of the second method is detailed in Appendix C.

Graphing the Power Function $P = R/(R^2 + 2R + 1)$

This may be done in parts, looking at the contribution of each term, and then combining to form the final result. First, consider the denominator $R^2 + 2R + 1$. It consists of three segments. The most simple is the horizontal line at +1. The $2R$ term creates a straight line with slope 2 starting at the origin. The R^2 term creates a simple curve with increasing slope that starts at the origin, crosses the horizontal +1 line at $R = 1$ and also crosses the $2R$ line at $R = 2$. These items are drawn individually and then added together as illustrated in Figure 6.6.2 .

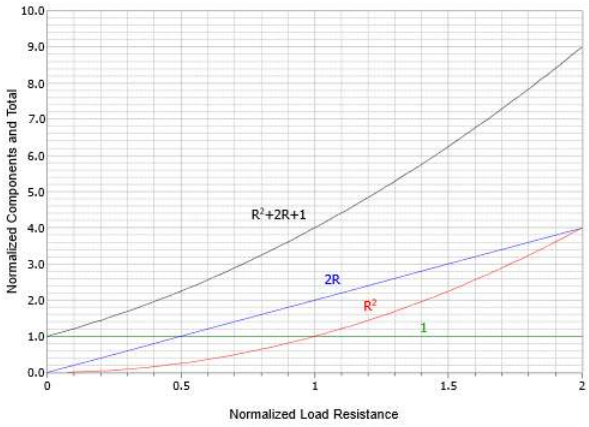


Figure 6.6.2 : The three terms of the power equation plotted individually and then summed.

Of course, we must remember that we really want $1/(R^2 + 2R + 1)$ and so we plot the reciprocal of the combo as shown in Figure 6.6.3 (red curve). We also include the numerator term, R. This is shown as a straight line (blue) with a slope of 1. Finally, these two curves are multiplied together to produce the load power equation (black).

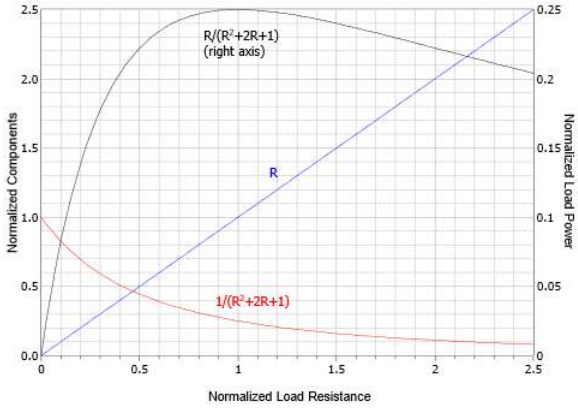


Figure 6.6.3 : The power equation and components plotted.

A close examination of the power curve will show that the peak occurs at $R = 1$. This is easier to see if we plot the completed power curve using a logarithmic horizontal axis and also scale the vertical axis to 100%, as shown in Figure 6.6.4 . The peak is more apparent and the curve is symmetrical in shape rather than lopsided. This reinforces the idea that the ratio of the resistances is what matters.

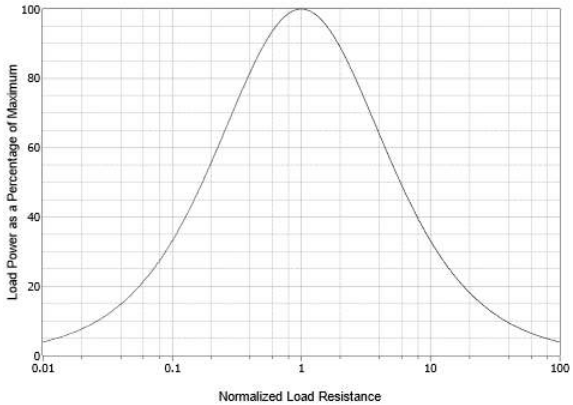


Figure 6.6.4 : The load power curve with logarithmic axis showing symmetry.

Finally, we can state:

> Maximum load power will be achieved when the load resistance is equal to the internal resistance of the driving source.

No other value of load resistance will produce a higher load power. For the circuit of Figure 6.6.1 , this means that R must equal R_i. As an exercise, we can try substituting a few values around the peak to verify this. For example, given E = 1 V and R_i = 1 Ω, we calculate P for R = 0.5 Ω, 1 Ω, and 2 Ω:

$$P_{0.5} = 0.5/(0.5^2 + 2 \cdot 0.5 + 1) = 2/9$$

$$P_1 = 1/(1^2 + 2 \cdot 1 + 1) = 2/8 = 1/4$$

$$P_2 = 2/(2^2 + 2 \cdot 2 + 1) = 2/9$$

You can also try this with extremely small variations such as R = 0.9999 Ω along with R = 1.0001 Ω and you will not achieve a value equal to or greater than 1/4 Watt.

While matching the resistance produces the maximum load power, it does not produce maximum load current or maximum load voltage. In fact, this condition produces a load voltage and a load current that are half of their maximums. Their product, however, is at the maximum. Further, efficiency at maximum load power is only 50% (i.e., only half of all generated power goes to the load with the other half being wasted internally). Values of R greater than R_i will achieve higher efficiency but at reduced load power. Sometimes we favor efficiency over maximal load power.

As any linear two point network can be reduced to something like Figure 6.6.1 by using Thévenin's theorem, combining the two theorems allows us to determine maximum power conditions for any resistor in a complex circuit.

This page titled 4.6: Maximum Power Transfer Theorem is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.6: Maximum Power Transfer Theorem** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.7: Advanced Topic- Delta-Y Conversions

Certain component configurations, such as bridged networks, cannot be reduced to a single resistance using basic series-parallel conversion techniques. One method for simplification involves converting sections into more convenient forms. The configurations in question are three-point networks containing three resistors. Due to the manner in which they drawn, they are referred to as delta networks and Y networks[1]. These configurations are shown in Figures 6.7.1 . In this particular case the delta version is drawn upside down so that its terminal designations match those of the Y configuration.

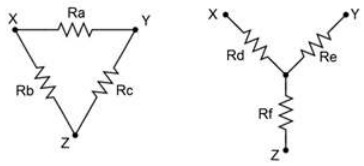


Figure 6.7.1 : Delta and Y (Δ-Y) networks.

Alternately, if they are slightly redrawn they are known as pi (also called "π") networks and T (also called "tee") networks. These configurations are shown in Figure 6.7.2 .

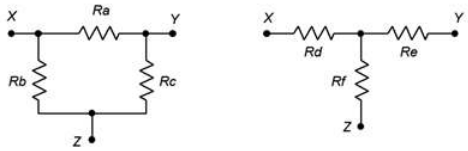


Figure 6.7.2 : Alternate form: Pi and T(π-T) networks.

It is possible to convert back and forth between delta and Y networks. That is, for every delta network, there exists a Y network such that the resistances seen between the X, Y and Z terminals are identical, and vice versa. Consequently, one configuration can replace another in order to simplify a larger circuit.

Δ-Y Conversion

A true equivalent circuit would present the same resistance between any two terminals as the original circuit. Consider the unloaded case for the circuits of Figure 6.7.1 (i.e., just these networks with nothing else connected to them). The equivalent resistances seen between each pair of terminals for the delta and Y respectively are:

$$R_{XY} = R_a||(R_b + R_c) = R_d + R_e \tag{4.7.1}$$

$$R_{XZ} = R_b||(R_a + R_c) = R_d + R_f \tag{4.7.2}$$

$$R_{ZY} = R_c||(R_b + R_a) = R_e + R_f \tag{4.7.3}$$

Assuming we have the delta and are looking for the Y equivalent, note that we have three equations with three unknowns (R_d, R_e and R_f). Thus, they can be solved using a term elimination process. If we subtract Equation 4.7.3 from Equation 4.7.1 we eliminate the second resistance (R_e) and arrive at a difference between the first and third unknown resistances ($R_d - R_f$). This quantity can then be added to Equation 4.7.2 to eliminate the third resistance (R_f), leaving just the first unknown resistance (R_d).

$$(R_d + R_e) - (R_e + R_f) = (R_d - R_f) = R_a||(R_b + R_c) - R_b||(R_a + R_c)$$

$$(R_d + R_f) + (R_d - R_f) = 2R_d = 2(R_b||(R_a + R_c) + R_a||(R_b + R_c) - R_c||(R_a + R_b))$$

Therefore,

$$R_d = R_b||(R_a + R_c) + R_a||(R_b + R_c) - R_c||(R_a + R_b)$$

which, after simplifying, is:

$$R_d = \frac{R_a R_b}{R_a + R_b + R_c} \tag{4.7.4}$$

Similarly, we can show that

$$R_e = \frac{R_a R_c}{R_a + R_b + R_c} \tag{4.7.5}$$

$$R_f = \frac{R_b R_c}{R_a + R_b + R_c} \tag{4.7.6}$$

Note that if three identical resistors are used, the values of the Y equivalent will all be one-third of that value.

Helpful Hint for Remembering Δ-Y Conversion Equations

Examine the equations for R_d, R_e, and R_f. You will note that they all have the same denominator. What kind of mnemonic (memory helper) can you come up with to help you remember the relationship between the resistor whose value you are trying to calculate and the resistors in the numerator on the right-hand side of the equation.

▼ Suggestion

- The resistances in the numerator are the two resistors in the delta network that are connected to the same node as the resistor in the Wye network whose value you are trying to determine.
  - For example, if you are calculating R_d, you'll notice it is connected to node X. In the delta configuration, R_a and R_b are connected to node X, so the product of R_a and R_b is used in the calculation of R_d

Y-Δ Conversion

For the reverse process of converting Y to delta, start by noting the similarities of the expressions for R_d, R_e and R_f. If two of these expressions are divided, a single equation for R_a, R_b or R_c will result. For example, using Equations 4.7.4 and 4.7.5:

$$\frac{R_d}{R_e} = \frac{\frac{R_a R_b}{R_a + R_b + R_c}}{\frac{R_a R_c}{R_a + R_b + R_c}}$$

$$\frac{R_d}{R_e} = \frac{R_a R_b}{R_a R_c}$$

$$\frac{R_d}{R_e} = \frac{R_b}{R_c}$$

Therefore,

$$\frac{R_b}{R_c} = \frac{R_d}{R_e}$$

$$R_b = \frac{R_c R_d}{R_e}$$

This process can be repeated for Equations 4.7.4 and 4.7.6 to obtain an expression for R_a. The two expressions for R_a and R_b can then be substituted into Equation 4.7.4 to obtain an expression for R_c that utilizes only R_d, R_e and R_f. A similar process is followed for R_a and R_b resulting in:

$$R_a = \frac{R_d R_e + R_e R_f + R_d R_f}{R_f} \tag{4.7.7}$$

$$R_b = \frac{R_d R_e + R_e R_f + R_d R_f}{R_e} \tag{4.7.8}$$

$$R_c = \frac{R_d R_e + R_e R_f + R_d R_f}{R_d} \tag{4.7.9}$$

If three identical resistors are used, the values of the delta equivalent will all be three times that value, the inverse of the situation when converting from delta to Y.

Helpful Hint for Remembering Y-Δ Conversion Equations

Examine the equations for R_a, R_b, and R_c. You will note that they all have the same numerator.

What kind of mnemonic can you come up with to help you remember the relationship between the resistor whose value you are trying to calculate and the resistor in the denominator on the right-hand side of the equation.

▼ Suggestion

- The resistance in the denominator is the value of the resistor in the Wye network that is connected to the node opposite the resistor in the Delta network whose value you are trying to determine
  - For example, if you are calculating R_a, you'll notice that the node opposite is is Z and in the delta network, R_f is connected to Z, so R_f is used in the denominator for the calculation of R_a

Thus, equations 4.7.4, 4.7.5 and 4.7.6 can be used to convert a delta network into a Y network, and equations 4.7.7, 4.7.8 and 4.7.9 can be used to convert a Y network into a delta network. An example of how this will tame an otherwise incorrigible series-parallel network is next.

Example 6.7.1

Determine the equivalent of the bridge circuit shown in Figure 6.7.3 .

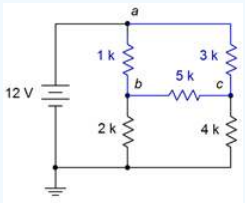


Figure 6.7.3 : Circuit for Example 6.7.1 .

This circuit uses a five resistor bridge which cannot be further simplified using basic series-parallel combinations. As a result, the techniques presented in Chapter 5 will not be sufficient to obtain a solution. A delta-Y conversion is proposed to simplify the circuit instead. We begin by defining a delta configuration using the top three resistors (shown in blue). When replaced with a Y configuration, we will have resistors in series with the 2 kΩ and 4 kΩ, and a third resistor in series with the source. This new network can solved using basic series-parallel techniques.

Note that a Y-delta is not as good of a choice. For example, using the 1 kΩ, 2 kΩ and the 5kΩ as a sideways Y, the delta version leaves resistors in parallel with the 3 kΩ and the 4 kΩ, and another in parallel with the source. This is straightforward to solve, but unfortunately, node b disappears in the process.

We shall use Equations 4.7.4, 4.7.5 and 4.7.6 to perform the delta-Y conversion. The orientation in Figure 6.7.3 is skewed compared to the reference of Figure 6.7.1 . To make them match, imagine rotating the networks of Figure 6.7.1 slightly clockwise so that point X is at the very top (matching node a), leaving Y and Z at the bottom (matching c and b, respectively). Given this new orientation, R_a is the 3 kΩ, R_b is the 1 kΩ and R_c is the 5 kΩ.

$$R_d = \frac{R_a R_b}{R_a + R_b + R_c}$$

$$R_d = \frac{3k\Omega 1k\Omega}{3k\Omega + 1k\Omega + 5k\Omega}$$

$$R_d \approx 333.333\Omega$$

$$R_e = \frac{R_a R_c}{R_a + R_b + R_c}$$

$$R_e = \frac{3k\Omega 5k\Omega}{3k\Omega + 1k\Omega + 5k\Omega}$$

$$R_e \approx 1.666667k\Omega$$

$$R_f = \frac{R_b R_c}{R_a + R_b + R_c}$$

$$R_f = \frac{1k\Omega 5k\Omega}{3k\Omega + 1k\Omega + 5k\Omega}$$

$$R_f \approx 555.556\Omega$$

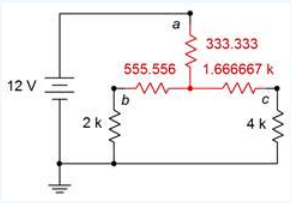


Figure 6.7.4 : Equivalent for the circuit of Figure 6.7.3 .

The equivalent network is placed back into the original circuit as shown in Figure 6.7.4 (red replaces blue). We now have a 333.333 Ω in series with 2.555556 kΩ || 5.666667 kΩ. To determine V_b and V_c the total equivalent resistance can be used to find the source current. From there, a current divider may be employed along with Ohm's law to find the node voltages.

$$R_{Total} = 333.333\Omega + 2.555556k\Omega || 5.666667k\Omega$$

$$R_{Total} \approx 2.0946k\Omega$$

$$I_s = \frac{E}{R_{Total}}$$

$$I_s \approx \frac{12V}{2.0946k\Omega}$$

$$I_s \approx 5.729mA$$

$$I_b = I_s \frac{R_{right}}{R_{left} + R_{right}}$$

$$I_b \approx 5.729mA \frac{5.666667k\Omega}{2.555556k\Omega + 5.666667k\Omega}$$

$$I_b \approx 3.9484mA$$

$$V_b = I_b \times R$$

$$V_b \approx 3.948mA \times 2k\Omega$$

$$V_b \approx 7.8968V$$

By similar process, V_c is approximately 7.1226 volts.

To verify the results of Example 6.7.1 , we can take the two node voltages, apply them back to the original circuit, and determine whether or not they satisfy KVL or KCL.

For example, examining node b, the current flowing through the 1 kΩ resistor has to equal the sum of the currents flowing through the 2 kΩ and 5 kΩ resistors, via KCL.

$$I_{1k} = \frac{V_a - V_b}{R}$$

$$I_{1k} \approx \frac{12V - 7.8968V}{1k\Omega}$$

$$I_{1k} \approx 4.1032mA$$

$$I_{5k} = \frac{V_b - V_c}{R}$$

$$I_{5k} \approx \frac{7.8968V - 7.1226V}{5k\Omega}$$

$$I_{5k} \approx 0.1548mA$$

$$I_{2k} = I_{1k} - I_{5k}$$

$$I_{2k} \approx 4.1032mA - 0.1548mA$$

$$I_{2k} \approx 3.9484mA$$

This result matches the current I_b calculated previously. For completeness sake, the process can be replicated for node c.

Computer Simulation

For further verification, both the original circuit of Figure 6.7.3 and the converted circuit of Figure 6.7.4 are entered into a simulator, as shown in Figure 6.7.5 .

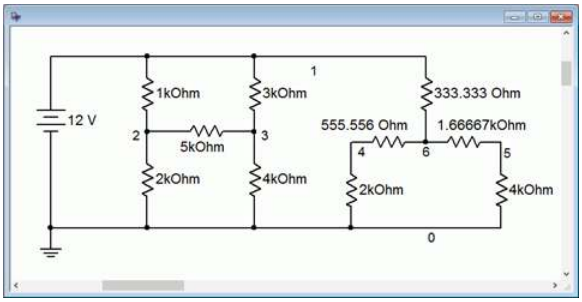


Figure 6.7.5 : Original and converted bridge circuits in simulator.

Both resistive networks are connected to a common power supply. Nodes 2 and 4 correspond to V_b, while nodes 3 and 5 correspond to V_c. The results of a DC operating point simulation are shown in Figure 6.7.6 .

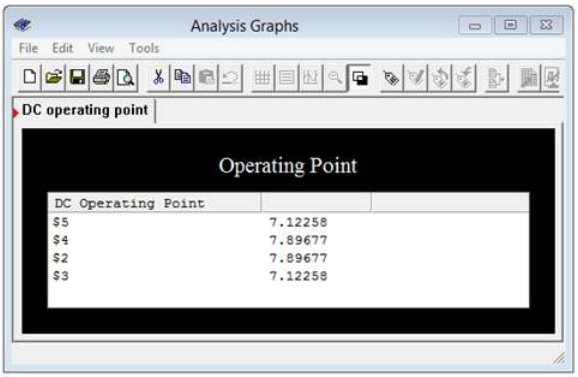


Figure 6.7.6 : Simulation results for original and converted bridges.

The voltages match each other perfectly, along with the manual calculation. In closing, we see that it is possible to swap delta networks with Y networks, and achieve identical results.

References

[1]In some sources the capital Greek letter delta (Δ) is used instead of spelling out “delta” and the letter Y is spelled out as “wye”. Thus, you may come across discussion of “Δ-Y ”, “Δ-wye” or “delta-wye” networks. It's all the same stuff.

This page titled 4.7: Advanced Topic- Delta-Y Conversions is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.7: Delta-Y Conversions** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.8: Summary

In this chapter we have examined several new techniques and theorems to assist with the analysis of DC electrical circuits. Beginning with more practical models for voltage and current sources, we added an internal resistance which sets limits on the source's maximum output. For a voltage source, this resistance is in series, its ideal value being a short. For current sources, the resistance is in parallel, its ideal value being an open.

Source conversions allow us to create an equivalent current source for any practical voltage source and vice versa. An equivalent source is one that will create the same voltage across (and current into) whatever the new source is connected to as did the original source. In some cases, this swap allows differing sources to be combined into a single source, simplifying analysis.

The superposition theorem states that, for any multi-source linear bilateral network, the contributions of each source may be determined independent of all other sources, the final result being the summation of the contributions, cognizant of current directions and voltage polarities. Thus, the original circuit of N sources generates N new circuits, one for each source under consideration and with all other sources replaced by their ideal internal resistance.

Thévenin's and Norton's theorems allow the simplification of complex linear single port (i.e., two connecting points) networks. The Thévenin equivalent consists of a voltage source with series a resistance while the Norton equivalent consists of a current source with a parallel resistance. These equivalents, when replacing the original sub-circuit, will create the same voltage across the remainder of the circuit with the same current draw. That is, the remainder of the circuit will see no difference between being driven by the original sub-circuit or by either the Thévenin or Norton equivalents.

The maximum power transfer theorem states that for a simple voltage source with an internal resistance driving a single resistor load, the maximum load power will be achieved when the load resistance equals the internal resistance. At this point, efficiency will be 50%. If the load resistance is higher than the internal resistance, the load power will not be as great, however, the system efficiency will increase.

Delta-Y conversions allow the generation of equivalent “three connection point” resistor networks. Resistor networks with three elements in the shape of a triangle or delta (with one connection point at each corner) can be converted into a three element network in the shape of a Y or T, or vice versa. The two versions will behave identically to the remainder of the circuit. This allows the simplification of some circuits and eases analysis.

Review Questions

1. Define the term linear bilateral network.
2. What are the ideal internal resistances of voltage sources and current sources?
3. Outline the process of converting a voltage source into a current source, and vice versa.
4. In general, describe the process of using superposition to analyze a multisource circuit.
5. What do Thévenin's and Norton's theorems state? How are they related?
6. What does the maximum power transfer theorem state? How might it be used with Thévenin's or Norton's theorems?
7. What are delta and Y configurations? How are they related?

This page titled 4.8: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.8: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

4.9: Exercises

Analysis

1. For the circuit shown in Figure 6.9.1 , determine the equivalent current source.

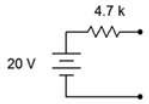


Figure 6.9.1

▼ Answer 1

- 4.26mA source in parallel with 4.7k resistor

---

2. Given the circuit shown in Figure 6.9.2 , determine the equivalent current source.

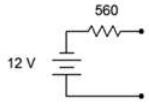


Figure 6.9.2

---

3. Determine the equivalent current source for the circuit shown in Figure 6.9.3 .

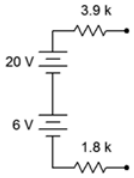


Figure 6.9.3

▼ Answer 3

- 4.56mA source in parallel with 5.7k resistor

---

4. For the circuit shown in Figure 6.9.4 , determine the equivalent current source.

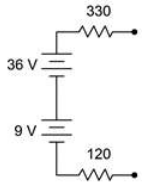


Figure 6.9.4

---

5. For the circuit shown in Figure 6.9.5 , determine the equivalent voltage source.

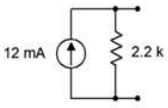


Figure 6.9.5

▼ Answer 5

- 26.4V in series with 2.2k resistor

---

6. Given the circuit shown in Figure 6.9.6 , determine the equivalent voltage source.

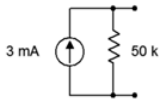


Figure 6.9.6

---

7. Determine the equivalent voltage source for the circuit shown in Figure 6.9.7 . 800\ohm

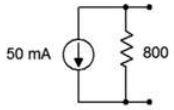


Figure 6.9.7

▼ Answer 7

- -40V in series with 800 Ω resistor

---

8. For the circuit shown in Figure 6.9.8 , determine the equivalent voltage source.

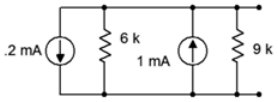


Figure 6.9.8

---

9. Given the circuit shown in Figure 6.9.9 , determine the equivalent voltage source.

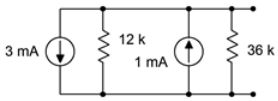


Figure 6.9.9

▼ Answer 9

- -18V in series with 9kΩ resistor

---

10. Using source conversion, find V_b for the circuit shown in Figure 6.9.10 .

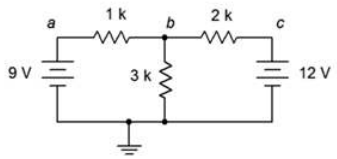


Figure 6.9.10

---

11. Using source conversion, find the current through the 3 kΩ resistor in the circuit of Figure 6.9.11 .

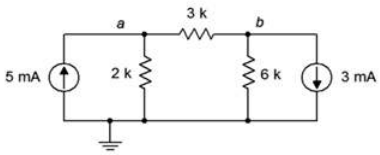


Figure 6.9.11

▼ Answer 11

- 2.55mA left to right

---

12. Using source conversion, find V_b for the circuit shown in Figure 6.9.12 .

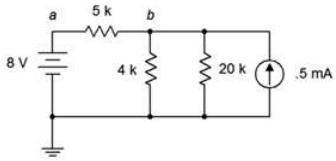


Figure 6.9.12

---

13. Using source conversion, find V_a for the circuit shown in Figure 6.9.13 .

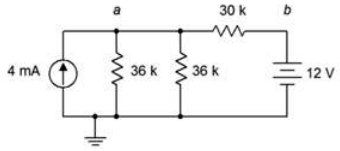


Figure 6.9.13

▼ Answer 13

- 49.5V

---

14. Using source conversion, find V_b for the circuit shown in Figure 6.9.14 .

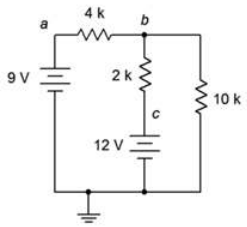


Figure 6.9.14

15. Using superposition, determine V_b for the circuit shown in Figure 6.9.10 .

▼ Answer 15

- 8.18V

16. Using superposition, find the current through the 3 kΩ resistor for the circuit of Figure 6.9.10 .

17. Using superposition, find the current through the 3 kΩ resistor for the circuit of Figure 6.9.11 .

▼ Answer 17

- 2.55mA

18. Using superposition, determine V_{ab} for the circuit shown in Figure 6.9.11 .

19. Using superposition, determine V_b for the circuit shown in Figure 6.9.12 .

▼ Answer 19

- 4.2V

20. Using superposition, find the current through the 4 kΩ resistor for the circuit of Figure 6.9.12 .

21. Using superposition, find the current through the 30 kΩ resistor for the circuit of Figure 6.9.13 .

▼ Answer 21

- 1.25mA

22. Using superposition, determine V_a for the circuit shown in Figure 6.9.13 .

23. Using superposition, determine V_{ba} for the circuit shown in Figure 6.9.14 .

▼ Answer 23

- 0.7V

---

24. Using superposition, find the current through the 10 kΩ resistor for the circuit of Figure 6.9.14 .

---

25. Using superposition, find the current through the 1.5 kΩ resistor for the circuit of Figure 6.9.15 .

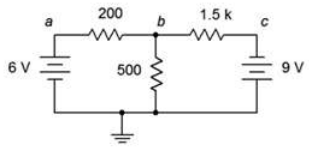


Figure 6.9.15

▼ Answer 25

- 8.09mA

---

26. Using superposition, determine V_{ab} for the circuit shown in Figure 6.9.15 .

---

27. Using superposition, determine V_b for the circuit shown in Figure 6.9.16 .

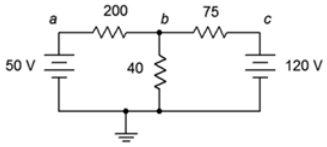


Figure 6.9.16

▼ Answer 27

- 42.7V

---

28. Using superposition, find the current through the 200 Ω resistor for the circuit of Figure 6.9.16 .

---

29. Using superposition, find the current through the 4 kΩ resistor for the circuit of Figure 6.9.17 .

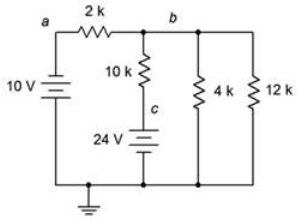


Figure 6.9.17

▼ Answer 29

- 0.696mA

---

30. Using superposition, determine V_b for the circuit shown in Figure 6.9.17 .

---

31. Is it possible to determine V_b in Figure 6.9.17 by using source conversions instead of superposition? Why/why not?

▼ Answer 31

- Yes, it's only current sources and resistors in parallel

---

32. Using superposition, determine V_{bc} for the circuit shown in Figure 6.9.18 .

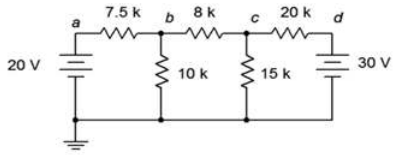


Figure 6.9.18

---

33. Using superposition, find the current through the 10 kΩ resistor for the circuit of Figure 6.9.18 .

▼ Answer 33

- 1.17mA

---

34. Using superposition, find the currents through the 100 Ω and 700 Ω resistors for the circuit shown in Figure 6.9.19 .

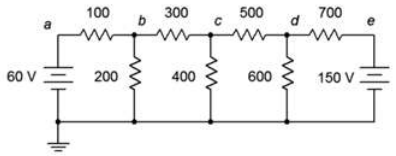


Figure 6.9.19

---

35. Using superposition, determine V_{bd} for the circuit shown in Figure 6.9.19 .

▼ Answer 35

- -15.65V

---

36. Is it possible to determine V_{bd} in Figure 6.9.19 by using source conversions instead of superposition? Why/why not?

37. Using superposition, determine V_{ad} for the circuit shown in Figure 6.9.20 .

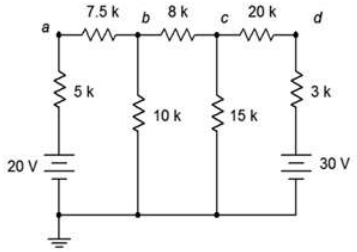


Figure 6.9.20

▼ Answer 37

- -11.6V

38. Using superposition, find the current through the 20 kΩ resistor for the circuit shown in Figure 6.9.20 .

39. Using superposition, find the current through the 12 kΩ resistor for the circuit shown in Figure 6.9.21 .

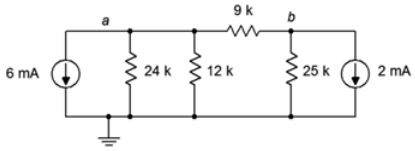


Figure 6.9.21

▼ Answer 39

- 4.03mA (up)

40. Using superposition, determine V_b for the circuit shown in Figure 6.9.21 .

41. Using superposition, determine V_b for the circuit shown in Figure 6.9.22 .

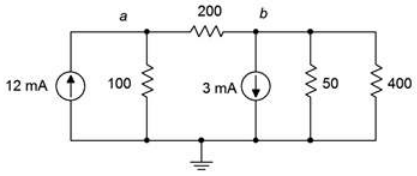


Figure 6.9.22

▼ Answer 41

- 38.7mV

42. Using superposition, find the current through the 100 Ω resistor for the circuit of Figure 6.9.22 .

---

43. Using superposition, find the current through the 5 kΩ resistor for the circuit of Figure 6.9.23 .

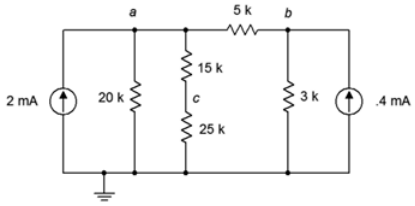


Figure 6.9.23

▼ Answer 43

- 1.19mA left to right

---

44. Using superposition, determine V_c for the circuit shown in Figure 6.9.23 .

---

45. Given the circuit shown in Figure 6.9.24 , determine the Thévenin equivalent circuit that is driving the 4 kΩ resistor.

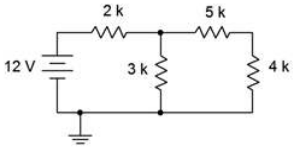


Figure 6.9.24

▼ Answer 45

- 7.2V with 6.2kΩ

---

46. Given the circuit shown in Figure 6.9.24 , determine the Norton equivalent circuit driving the 4 kΩ resistor.

---

47. Given the circuit shown in Figure 6.9.24 , determine the Norton equivalent circuit driving the 12 kΩ resistor.

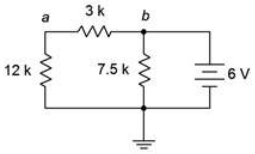


Figure 6.9.25

▼ Answer 47

- 2mA with 3kΩ resistor

---

48. Determine the Thévenin equivalent circuit driving the 12 kΩ resistor for the circuit shown in Figure 6.9.25 .

---

49. Given the circuit shown in Figure 6.9.26 , determine the Thévenin equivalent circuit that is driving the 20 kΩ resistor.

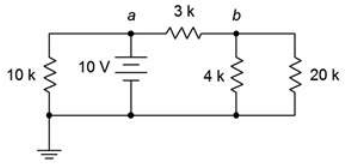


Figure 6.9.26

▼ Answer 49

- 5.71V with 1.71$k\Omega$ resistor

---

50. For the circuit shown in Figure 6.9.26 , determine the Norton equivalent circuit driving the 4 kΩ resistor.

---

51. Given the circuit shown in Figure 6.9.27 , determine the Norton equivalent circuit driving the 40 Ω resistor.

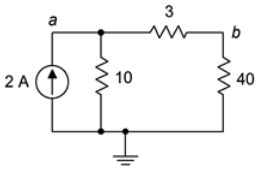


Figure 6.9.27

▼ Answer 51

- 1.538A with 13 Ω resistor

---

52. Determine the Thévenin equivalent circuit driving the 10 Ω resistor for the circuit shown in Figure 6.9.27 .

---

53. Given the circuit shown in Figure 6.9.28 , determine the Norton equivalent circuit driving the 12 kΩ resistor.

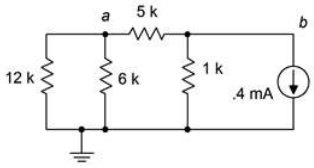


Figure 6.9.28

▼ Answer 53

- 66.7 μA with 3 $k\Omega$ resistor

---

54. Given the circuit of Figure 6.9.24 , determine the power in the 4 kΩ resistor. If this resistor can be replaced with any other value, is it possible to achieve a higher power? Why/why not?

---

55. Given the circuit of Figure 6.9.25 , determine the power in the 12 kΩ resistor. If this resistor can be replaced with any other value, is it possible to achieve a higher power? Why/why not?

▼ Answer 55

- 1.92mW. Yes, match R_{Th}

---

56. Given the circuit of Figure 6.9.27 , determine the power in the 40 Ω resistor. If this resistor can be replaced with any other value, is it possible to achieve a higher power? Why/why not?

---

57. Given the circuit of Figure 6.9.28 , determine the power in the 6 kΩ resistor. If this resistor can be replaced with any other value, is it possible to achieve a higher power? Why/why not?

▼ Answer 57

- $4.27\mu W$. Yes, match R_{Th}

Design

58. Consider the 4 kΩ resistor to be the load in Figure 6.9.24 . Determine a new value for the load in order to achieve maximum load power. Also determine the maximum load power.

---

59. Consider the 12 kΩ resistor to be the load in Figure 6.9.25 . Determine a new value for the load in order to achieve maximum load power. Also determine the maximum load power.

▼ Answer 59

- 1.92mW. Yes, match R_{Th}

---

60. Consider the 40 Ω resistor to be the load in Figure 6.9.27 . Determine a new value for the load in order to achieve maximum load power. Also determine the maximum load power.

---

61. Consider the 6 kΩ resistor to be the load in Figure 6.9.28 . Determine a new value for the load in order to achieve maximum load power. Also determine the maximum load power.

▼ Answer 61

- 4kΩ, $4.44\mu W$

---

62. Redesign the circuit of Figure 6.9.15 so that it uses only current sources and produces the same component currents and voltages as the original circuit.

---

63. Redesign the circuit of Figure 6.9.17 so that it uses only current sources and produces the same component currents and voltages as the original circuit.

▼ Answer 63

- Replace 10V+2kΩ with 5mA||2kΩ and 24V+10kΩ with 2.4mA||10kΩ

---

64. Redesign the circuit of Figure 6.9.21 so that it uses only voltage sources and produces the same component currents and voltages as the original circuit.

---

65. Redesign the circuit of Figure 6.9.23 so that it uses only voltage sources and produces the same component currents and voltages as the original circuit.

▼ Answer 65

- Replace 2mA||20kΩ with 40V + 20kΩ, and 0.4mA||3kΩ with 1.2V + 3kΩ

---

66. Convert the delta network of Figure 6.9.29 into a Y network.

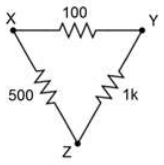


Figure 6.9.29

---

67. Convert the pi network of Figure 6.9.30 into a T network.

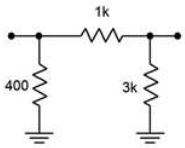


Figure 6.9.30

▼ Answer 67



68. Convert the Y network of Figure 6.9.31 into a delta network.

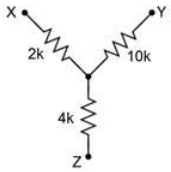


Figure 6.9.31

69. Convert the T network of Figure 6.9.32 into a pi network.

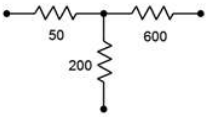
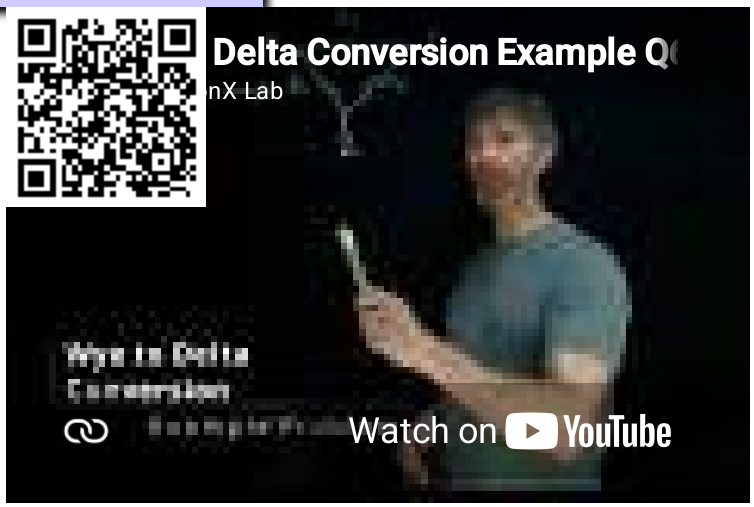


Figure 6.9.32

▼ Answer 69



Challenge

70. Redesign the circuit of Figure 6.9.33 so that it uses only current sources and produces the same node voltages as the original circuit.

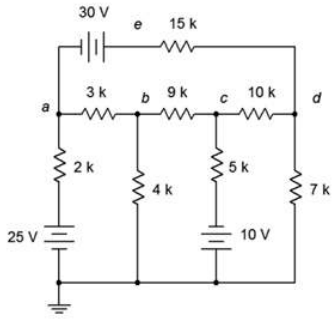


Figure 6.9.33

---

71. Using any combination of techniques, find the current through the 9 kΩ resistor for the circuit of Figure 6.9.33 .

---

72. Using any combination of techniques, determine V_{bc} for the circuit shown in Figure 6.9.33 .

---

73. Is it possible to determine V_{bc} in Figure 6.9.33 by using just source conversions? Why/why not?

---

74. Using superposition, find the current through the 25 Ω resistor for the circuit of Figure 6.9.34 .

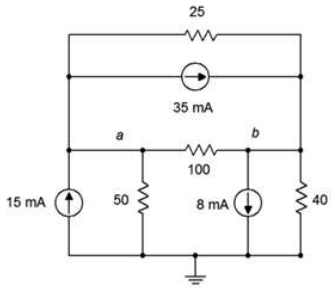


Figure 6.9.34

---

75. Using superposition, find V_{ab} in the circuit of Figure 6.9.34 .

---

76. Redesign the circuit of Figure 6.9.34 using only voltage sources so that it achieves the same node voltages as the original.

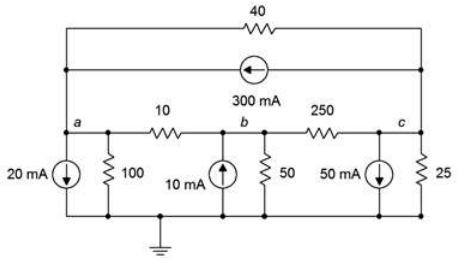


Figure 6.9.35

77. Is it possible to determine V_{bc} in Figure 6.9.35 by using just source conversions or just superposition? Why/why not?

78. Using any combination of techniques, find the current through the 100 Ω resistor for the circuit shown in Figure 6.9.35 .

79. Redesign the circuit of Figure 6.9.35 so that it uses only voltage sources and produces the same node voltages as the original circuit.

80. Determine the Thévenin and Norton equivalents driving the 40 Ω resistor for the circuit shown in Figure 6.9.16 .

81. Determine the Thévenin and Norton equivalents driving the 12 kΩ resistor for the circuit shown in Figure 6.9.17 .

▼ Answer 81

Thevenin's Equivalent Answer:



82. Given the circuit of Figure 6.9.26 , determine if the 4 kΩ resistor is the optimal value to achieve maximum power dissipation in that resistor. If it is not, determine the value that will produce maximum power in the resistor along with the resulting power.

83. For the circuit of Figure 6.9.36 , determine an equivalent circuit using just a single voltage source.

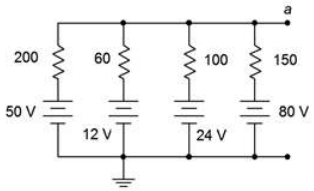


Figure 6.9.36

Simulation

84. Using DC bias simulations, compare the original circuit of problem 1 to its converted equivalent. Do this by connecting a resistor to the output terminals, trying several different resistance values and checking to see if the two circuits always produce the same voltage across this resistor.

85. Using DC bias simulations, compare the original circuit of problem 5 to its converted equivalent. Do this by connecting a resistor to the output terminals, trying several different resistance values and checking to see if the two circuits always produce the same voltage across this resistor.

86. Perform a DC bias simulation on the circuit of problem 11 to verify the node voltages.

87. Perform a DC bias simulation on the circuit of problem 13 to verify the node voltages.

88. Perform a DC bias simulation on the circuit of problem 19 to verify the node voltages.

89. Perform a DC bias simulation on the circuit of problem 21 to verify the resistor current.

90. Create DC bias simulations of the original and equivalent circuits generated in problem 45 to determine if the equivalent circuit is truly equivalent. Do this by substituting several different values for the 4 kΩ resistor in both circuits to see if the same load voltage is obtained for both circuits.

91. Create DC bias simulations of the original and equivalent circuits generated in problem 49 to determine if the equivalent circuit is truly equivalent. Do this by substituting several different values for the 20 kΩ resistor in both circuits to see if the same load voltage is obtained for both circuits.

92. Create DC bias simulations of the original and equivalent circuits generated in problem 53 to determine if the equivalent circuit is truly equivalent. Do this by substituting several different values for the 12 kΩ resistor in both circuits to see if the same load voltage is obtained for both circuits.

93. Perform a DC bias simulation on the circuit of problem 63 to verify that the node voltages of the new design match those of the original.

---

94. Perform a DC bias simulation on the circuit of problem 65 to verify that the node voltages of the new design match those of the original.

---

95. Using either Monte Carlo simulation or multiple DC bias simulations, verify that the resistance calculated in problem 55 achieves maximum power. Do this by trying several resistor values near the calculated value, and determining the power for each based on the squared load voltages.

---

96. Using either Monte Carlo simulation or multiple DC bias simulations, verify that the resistance calculated in problem 56 achieves maximum power. Do this by trying several resistor values near the calculated value, and determining the power for each based on the squared load voltages.

This XKCD comic is too large to fit on this page. Just go to this link instead:

https://xkcd.com/1732/

This page titled 4.9: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **6.9: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

5: Nodal and Mesh Analysis, Dependent Sources

Learning Objectives

After completing this chapter, you should be able to:

- Utilize nodal analysis techniques to solve for voltages in multi-source series-parallel networks.
- Utilize mesh analysis techniques to solve for currents in multi-source series-parallel networks.
- Analyze networks using dependent voltage and current sources.

5.1: Introduction

5.2: Nodal Analysis

5.3: Mesh Analysis

5.4: Dependent Sources

5.5: Summary

5.6: Exercises

This page titled 5: Nodal and Mesh Analysis, Dependent Sources is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

5.1: Introduction

This chapter presents the culmination of the analysis of DC circuits that employ resistors along with any number of voltage and/or current sources. Here we shall focus on nodal analysis and mesh analysis. Both nodal analysis and mesh analysis generate a system of simultaneous linear equations that are used to solve the circuit for various voltages or currents. There are several methods that can be used to solve the simultaneous equations including substitution, Gauss-Jordan elimination and expansion by minors. These methods are reviewed in Appendix B and are not covered in this chapter. Instead, to keep clutter to a minimum and focus on the circuit analysis aspects, the explanations and examples will simply detail the process of examining the circuit and applying basic circuit laws to create the system of equations. From there, the specific technique used to solve these simultaneous equations is up to you based on your personal preferences.

It is particularly efficient to obtain a more advanced scientific calculator that can solve these equations directly rather than plowing through the solution manually. By doing so, you can spend your precious time mastering the process of circuit analysis and creating the equations. Manual solution techniques, though not necessarily difficult, can be tedious, time consuming and error prone. If you plan on continuing your study into AC electrical circuits, you should consider obtaining a calculator that can solve simultaneous equations with complex coefficients. Such calculators can be expensive when purchased new, such as the Texas Instruments TI-89 and Nspire models. On the used market, perfectly satisfactory older models such as the TI-85 and TI-86 can be found at considerable discount. Another model to consider is the Casio FX-9750GII, although it is not quite as powerful as some of the other units mentioned.

Along with nodal and mesh, we shall also introduce the concept of dependent sources. These are current and voltage sources whose value is not fixed to some particular value, but rather is dependent on some other current or voltage in the circuit. What makes this interesting is that this controlling current or voltage may itself be affected by the value produced by the dependent source. Dependent sources are often used to model the behavior of active electronic devices such as bipolar and field effect transistors.

This page titled 5.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

5.2: Nodal Analysis

Nodal analysis is a technique that can be applied to virtually any circuit. In general use, it might be considered a universal solution technique as there are no practical circuit configurations that it cannot handle. Nodal analysis relies on the application of Kirchhoff's current law to create a series of node equations that can be solved for node voltages. These equations are based on Ohm's law and will be of the form $I = V/R$, or more generally, $I = (1/RX) \cdot VA + (1/RY) \cdot VB ...$ Once the node voltages are obtained, finding any branch currents or component powers becomes an almost trivial exercise. We will examine two variations; a general version that can be used with both voltage and current sources, and a second somewhat quicker version that can be used with circuits only driven by current sources.

General Method

Consider the circuit shown in Figure 7.2.1 . We begin by labeling connection nodes and assigning current directions. We are particularly interested in current junctions, that is, places where currents can combine or split. These are also known as summing nodes and are circled in green on the figure. The current directions are chosen arbitrarily and may be the opposite of reality. This is not a problem. If we assign directions that are wrong, the resulting directions will ultimately show up reversed but the computed node voltages will be just fine.

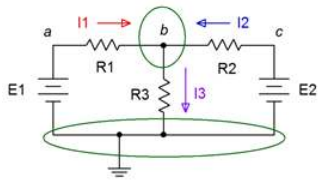


Figure 7.2.1 : Basic dual voltage source circuit with currents and nodes defined.

One node is chosen as the reference. This is the point to which all other node voltages will be measured against. Typically, the reference node is ground, although it does not have to be.

We now write a current summation equation for each summing node, except for the reference node. In this circuit there is only one node where currents combine (other than ground) and that's node b. Points a and c are places where components connect, but they are not summing nodes, so we can ignore them for now. Using KCL on node b we can say:

$$I_1 + I_2 = I_3$$

Next, we describe these currents in terms of the node voltages and associated components via Ohm's law. For example, I_3 is the node b voltage divided by R_3 while I_1 is the voltage across R_1 divided by R_1. This voltage is $V_a - V_b$. Therefore,

$$\frac{V_a - V_b}{R_1} + \frac{V_c - V_b}{R_2} = \frac{V_b}{R_3}$$

Noting that Missing argument for \frac and Missing argument for \frac, with a little algebra this can be reduced to a series of products of conductances and voltages:

$$\left(\frac{1}{R_1}\right) E_1 + \left(\frac{1}{R_2}\right) E_2 = \left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}\right) V_b$$

All quantities are known except for V_b and thus it is easily found with a little more algebra. As we shall see, this conductance-voltage product format turns out to be a convenient way of writing these equations. Also, note that the first two terms on the left reduce to fixed current values. If there had been more nodes, there would have been more equations, one for each node.

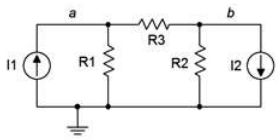


Figure 7.2.2 : Basic dual current source circuit.

For current sources, a more direct approach is possible. Consider the circuit of Figure 7.2.2 . We start as before, identifying nodes and labeling currents. This is shown in Figure 7.2.3 . We then write current summation equations at each node (except for ground). We consider currents entering a node as positive, and exiting as negative.

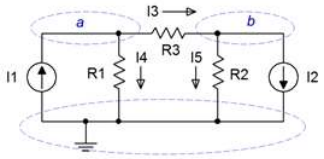


Figure 7.2.3 : Basic dual current source circuit with currents and nodes defined.

$$\text{Node } a : I_1 = I_3 + I_4$$

$$\text{Node } b : I_3 = I_2 + I_5, \text{ and rearranging in terms of the fixed source,}$$

$$\text{Node } b : -I_2 = -I_3 + I_5$$

The currents are then described by their Ohm's law equivalents:

$$\text{Node } a : I_1 = \frac{V_a - V_b}{R_3} + \frac{V_a}{R_1}$$

$$\text{Node } b : -I_2 = -\frac{V_a - V_b}{R_3} + \frac{V_b}{R_2}$$

Expanding and collecting terms yields:

$$\text{Node } a : I_1 = \left(\frac{1}{R_1} + \frac{1}{R_3}\right) V_a - \left(\frac{1}{R_3}\right) V_b$$

$$\text{Node } b : -I_2 = -\left(\frac{1}{R_3}\right) V_a + \left(\frac{1}{R_3} + \frac{1}{R_2}\right) V_b$$

As the resistor values and currents are known, simultaneous equation solution techniques may be used to solve for the node voltages. Once again, there will be as many equations as node voltages. It is very important that the terms “line up” when the final system of equations is written out. That is, there should be a column for the V_a terms, a column for the V_b terms, and so on. They should not be written out in random order. This format will make it much it easier to enter the coefficients into a calculator or solve manually.

Example 7.2.1

Determine V_b in the circuit of Figure 7.2.4 .

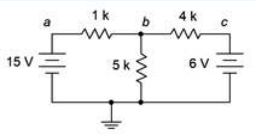


Figure 7.2.4 : Circuit for Example 7.2.1 .

This circuit has two current summing nodes, ground and node b. Let us assume that the currents from the two sources flow into node b and the exiting current flows down through the 5 kΩ resistor. It will be convenient if we find the conductance equivalents of the resistors before continuing. Taking the reciprocals we find: 1 kΩ = 1 mS, 4 kΩ = 0.250 mS, and 5 kΩ = 0.2 mS. Through KCL we can say:

$$I_{1k} + I_{4k} = I_{5k}$$

Replacing these currents with their Ohm's law equivalents yields,

$$\frac{15V - V_b}{1k\Omega} + \frac{6V - V_b}{4k\Omega} = \frac{V_b}{5k\Omega}$$

collecting terms,

$$15mA + 1.5mA = \left(\frac{1}{1k\Omega} + \frac{1}{4k\Omega} + \frac{1}{5k\Omega}\right) V_b$$

and solving for V_b,

$$V_b = \frac{16.5mA}{1mS + 0.25mS + 0.2mS}$$

$$V_b = 11.379 \text{ volts}$$

As V_b is higher than the 6 volt source, our assumed current direction for the 4 kΩ resistor was incorrect; we assumed right to left but it is in fact left to right, flowing from 11.379 volts to 6 volts. As you can see, this did not present a problem.

For comparison sake, this circuit was solved in Chapter 6 using source conversions and also using superposition.

Inspection Method

The system of equations can be obtained directly through inspection if the circuit contains no voltage sources. For the node under inspection, sum all of the current sources connected to it to obtain the current constant. The conductance term for that node will be the sum of all of the conductances connected to that node. For the other node conductances, determine the conductances between the node under inspection and these other nodes. These terms will all be negative. As a crosscheck, the set of equations produced must exhibit diagonal symmetry, that is, if a diagonal is drawn from the upper left to the lower right through the conductance-voltage pairs, then the coefficients found above the diagonal will have to match those found below the diagonal as we move perpendicular from the diagonal. This is illustrated in Figure 7.2.5 . Diagonal symmetry can be seen in Example 7.2.1 by noting the matching "$-1/R_3$" coefficients in the final pair of equations. As an example of the inspection method, focusing on node a in Figure 7.2.2 , we find the fixed current source I_1 feeding it (entering, therefore positive). The conductances directly connected to node a are $1/R_1$ and $1/R_3$, yielding the coefficient for V_a. The only conductance common between nodes a and b is $1/R_3$, yielding the V_b coefficient.

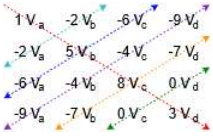


Figure 7.2.5 : Diagonal symmetry.

The inspection method is summarized as follows:

1. Verify that the circuit uses only current sources with resistors and no voltage sources. If voltage sources exist, they must be converted to current sources before proceeding.

2. Find all of the current summing nodes and number them. Also decide on the reference node (usually ground).

3. To generate an equation, locate the first node. This is the node of interest and the next few steps will be associated with it.

4. Sum the current sources feeding the node of interest. Entering is deemed positive while exiting is deemed negative. The sum is placed on one side of the equals sign.

5. Next, find all of the resistors connected to the node of interest and write them as a sum of conductances on the other side of the equals sign, the group being multiplied by this node's voltage (e.g., V_1).

6. Now find all of the resistors that are connected to the node of interest and to other nodes (except the ground reference). For each of these other nodes, multiply the sum of the conductances between the node of interest and this other node by this other node's voltage, and then subtract that product from the equation built so far. Once all other nodes are considered, this equation is finished.

7. Find the next node and treat this as the new node of interest.

8. Repeat steps 4 through 7 until all nodes have been treated as the node of interest. Each iteration creates a new equation. There will be as many equations as there are nodes, less the reference node. Check for diagonal symmetry and solve.

The inspection method is best observed in action; as in the following example.

Example 7.2.2

Determine V_a and V_b in the circuit of Figure 7.2.6 . Also determine the current flowing through the 100 Ω resistor.

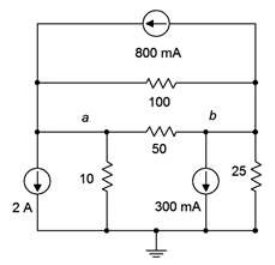


Figure 7.2.6 : Circuit for Example 7.2.2 .

This circuit has three current summing nodes; ground, node a and node b. Also, it uses only current sources and no voltage sources. Therefore, we are able to use the inspection method to obtain the system of equations (in this case, two equations).

We begin by focusing on node a, our first node of interest. We will build the first expression piece by piece. First we find the current sources feeding this node:

$$800mA - 2A = ...$$

Next, we find all of the resistors connected to node a and write them as conductances, the group being multiplied by node voltage a:

$$800mA - 2A = \left(\frac{1}{10\Omega} + \frac{1}{50\Omega} + \frac{1}{100\Omega}\right) V_a ...$$

Now find all of the resistors that are connected to this node and to the other nodes. Multiply those resistors (expressed as conductances) by the other associated node voltages and subtract the products from the expression built so far. Repeat for all remaining nodes except the ground reference. In this example there is only one other node, node b, and thus only one iteration.

$$800mA - 2A = \left(\frac{1}{10\Omega} + \frac{1}{50\Omega} + \frac{1}{100\Omega}\right) V_a - \left(\frac{1}{50\Omega} + \frac{1}{100\Omega}\right) V_b$$

Finally, simplify the constants and coefficients, and the first expression is complete:

$$-1.2A = 130mSV_a - 30mSV_b$$

Now we repeat the entire process for the next equation. Node b is our new node of interest. The fixed current sources are:

$$-800mA - 300mA = ...$$

Next, we find all of the resistors connected to node b and write them as conductances, the group being multiplied by node voltage b:

$$-800mA - 300mA = \left(\frac{1}{25\Omega} + \frac{1}{50\Omega} + \frac{1}{100\Omega}\right) V_b ...$$

Now we find all of the resistors that are connected to this node and to the other nodes. Multiply those resistors (expressed as conductances) by these other node voltages and subtract those products from the expression built so far. Make sure the terms align vertically based on the node voltages.

$$-800mA - 300mA = -\left(\frac{1}{50\Omega} + \frac{1}{100\Omega}\right) V_a + \left(\frac{1}{25\Omega} + \frac{1}{50\Omega} + \frac{1}{100\Omega}\right) V_b$$

Finally, simplify the constants and coefficients, and this expression is complete:

$$-1.1A = -30mSV_a + 70mSV_b$$

We now have two equations with two unknowns. Let's check for diagonal symmetry:

$$-1.2A = 130mSV_a - 30mSV_b$$

$$-1.1A = -30mSV_a + 70mSV_b$$

The perpendicular coefficient is −30 mS on each side. We can now solve the system. The results are V_a = −14.27 volts and V_b = −21.83 volts. The current flowing through the 100 resistor is (−14.27 V − (−21.83 V))/100 Ω, or approximately 75.6 mA flowing left to right.

Let's verify that these values are correct. We can perform a KCL summation at node a and see if it balances. We already know that 2 amps and 75.6 milliamps exit while 800 milliamps enters. We only need to find the currents through the 10 Ω and 50 Ω resistors. First, notice that the 50 Ω sees the same voltage as the 100 Ω resistor. As it is half the resistance, it must produce twice the current, or 151.2 milliamps exiting. The current through the 10 Ω is found via Ohm's law, or −14.27 V/10 Ω, which is 1.427 amps entering.

$$\text{Entering: } 0.8A + 1.427A = 2.227A$$

$$\text{Exiting: } 2A + 0.0756A + 0.1512A \approx 2.227A$$

KCL is satisfied. To complete the verification we perform the same summation on node b. This is left as an exercise.

Example 7.2.3

Write the node equations for the circuit of Figure 7.2.7 .

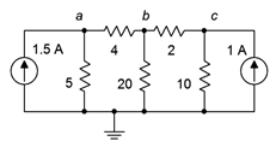


Figure 7.2.7 : Circuit for Example 7.2.3 .

This circuit has four current summing nodes; ground and nodes a, b, and c. Also, it uses only current sources and no voltage sources. Therefore, we are able to use the inspection method to obtain the system of equations (three equations).

As we have already seen, we will wind up with conductance values in the final equations. We can save some work later by finding the conductance value of each resistor right now. The result is shown in Figure 7.2.8 .

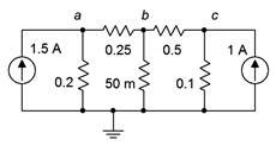


Figure 7.2.8 : Circuit of Figure 7.2.7 shown with conductances instead of resistances.

We begin by focusing on node a, our first node of interest. We will build the first expression piece by piece. First we find the current sources feeding this node. That's just the 1.5 A source, entering is positive.

$$1.5A = ...$$

Next, we find all of the conductances connected to node a and multiply by node voltage a:

$$1.5A = (0.2S + 0.25S)V_a ...$$

Find all of the conductances that are connected to this node and to the other nodes. Multiply those conductances by the other associated node voltages and subtract those products from the expression built so far. Repeat for all remaining nodes except the ground reference.

$$1.5A = (0.2S + 0.25S)V_a - (0.25S)V_b - (0)V_c$$

Finally, simplify the constants and coefficients, and the first expression is complete:

$$1.5A = 0.45SV_a - 0.25SV_b - 0V_c$$

We leave the zero term in just for alignment. Now we repeat the entire process for the next equation. The node of interest is now node b. Here is the result:

$$0 = -0.25SV_a + 0.8SV_b - 0.5V_c$$

And finally for node c we have:

$$1A = 0V_a - 0.5SV_b + 0.6V_c$$

Our three equations are (with some padding to see the columns better):

$$1.5A = 0.45SV_a - 0.25SV_b - 0V_c$$

$$0 = -0.25SV_a + 0.8SV_b - 0.5V_c$$

$$1A = -0V_a - 0.5SV_b + 0.6V_c$$

Check for diagonal symmetry: we have pairs of −0.25, 0, and −0.5 straddling the diagonal.

Now the system of equations can be solved for the three node voltages and verified using KCL summations at each node. This is left as an exercise.

Converting Sources and Other Simplifications

Given circuits with voltage sources, it may be easier to convert them to current sources and then apply the inspection technique rather than using the general approach outlined initially. There is one trap to watch out for when using source conversions: the voltage across or current through a converted component will most likely not be the same as the voltage or current in the original circuit. This is because the location of the converted component will have changed. For example, the circuit in Figure 7.2.9 could be solved using nodal analysis by converting the voltage source and the associated resistance into a current source. That is, E/R_1 would be converted into a source I_3 with a parallel resistor R_1. This is shown in Figure 7.2.10 .

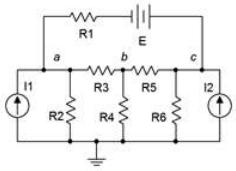


Figure 7.2.9 : Circuit with both current and voltage sources.

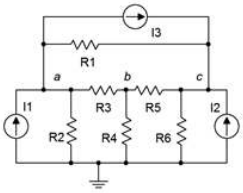


Figure 7.2.10 : Circuit converted to current sources only.

Here is the trap: in the converted circuit, although R_1 still connects to node a, the other end no longer connects to the voltage source. Rather, the right side now connects to node c. Therefore, the voltage drop across R_1 in the converted circuit is not likely to equal the voltage drop seen across R_1 in the original circuit (the only way they would be equal is if E was 0 volts). In the converted circuit, nodes a and c have not changed from the original, so the original voltage across R_1 can be determined via V_a, V_c and E in the original circuit.

If a circuit uses voltage sources exclusively, or even a very large proportion of voltage sources, an alternate technique called mesh analysis may be a better choice than using numerous conversions. Mesh analysis will be detailed later in this chapter.

A final item to consider is to simplify resistor networks in order to reduce the number of nodes. Fewer nodes means fewer equations and a quicker solution. For example, a portion of a network might consist of two parallel resistors which are in series with a third resistor. Ordinarily, the common junction of these three resistors would constitute a current summing node and an equation would need to be derived for it. It may be easier to simply replace the trio with a single resistor that equals the series-parallel combination value. This would remove the node and simplify creating the system of equations, but not alter the remainder of the circuit.

Supernode

On occasion you may come across a circuit like the one shown in Figure 7.2.11 that has a voltage source without a series resistance associated with it. Without that resistance, it becomes impossible to create an expression for the current passing through the source using the general method, and impossible to convert the voltage source into a current source in order to use the inspection method. One possible way out of this quandary is to simply add a very small resistor in series with it so that a source conversion is possible. The resistor in question would have to be much smaller than any surrounding resistors in order to have minimal impact on the results. A reduction by two orders of magnitude would generally yield a variation smaller than that produced by resistor tolerances in all but high precision circuits. The other way out is to use a supernode.

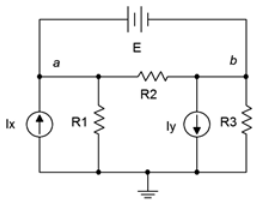


Figure 7.2.11 : Circuit for supernode analysis.

A supernode is, in effect, the combination of two nodes. It relies on a simple observation. If we examine the circuit of Figure 7.2.11 , the path of the voltage source produces identical currents flowing into and out of nodes a and b. As a consequence, if we treat the two nodes as one big node, then when we write a KCL summation, these two terms will cancel. To see just how this works, refer to Figure 7.2.12 .

In this version we have replaced the voltage source with its ideal internal resistance, a short. We have also labeled the two nodes of interest, a and b, and labeled the currents, drawn with convenient directions.

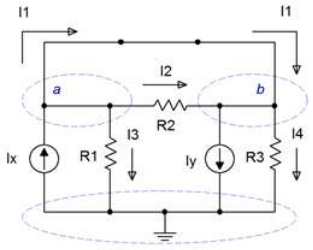


Figure 7.2.12 : Circuit modified for supernode analysis.

Due to the shorted voltage source, nodes a and b are now the same node. Consider the currents entering and exiting this combined or “super” node. On the left side (formerly node a) we see a constant current I_x entering while I_1, I_2 and I_3 are exiting. On the right side (formerly node b) we see I_y exiting along with I_4, and entering we see I_1 and I_2. Now let's set up the entering currents on the left side of the equals sign with the exiting currents on the right:

$$\sum I_{in} = \sum I_{out}$$

$$I_x + I_1 + I_2 = I_y + I_1 + I_2 + I_3 + I_4$$

This can be simplified to:

$$I_x - I_y = I_3 + I_4$$

Writing this in terms of Ohm's law we have:

$$I_x - I_y = \frac{1}{R_1}V_a + \frac{1}{R_3}V_b$$

We also know that $V_a - V_b = E$ from the original circuit. Assuming all sources and resistors are known, that makes two equations with two unknowns, solvable using simultaneous equation techniques. This is illustrated in the following example.

Example 7.2.4

Find V_a and V_b for the circuit of Figure 7.2.13 .

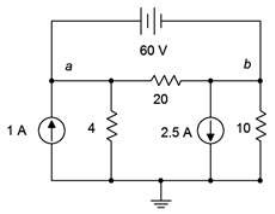


Figure 7.2.13 : Circuit for Example 7.2.4 .

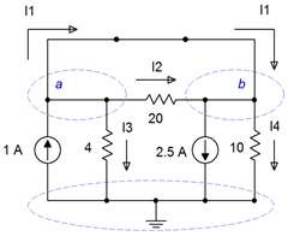


Figure 7.2.14 : Circuit modified for supernode analysis.

As shown in Figure 7.2.14 , we short the 60 volt source and write a current summation at the a b supernode:

$$\sum I_{in} = \sum I_{out}$$

$$1A + I_1 + I_2 = 2.5A + I_1 + I_2 + I_3 + I_4$$

This can be simplified to:

$$-1.5A = I_3 + I_4$$

Writing this in terms of Ohm's law we have:

$$-1.5A = \frac{1}{4\Omega}V_a + \frac{1}{10\Omega}V_b$$

$$-1.5A = 0.25SV_a + 0.1SV_b$$

We also know that $V_a - V_b = 60$ volts. Therefore $V_b = V_a - 60$ volts. Substituting this into the prior equation yields:

$$-1.5A = 0.25SV_a + 0.1S(V_a - 60V)$$

$$-1.5A = 0.25SV_a + 0.1SV_a - 6A$$

$$4.5A = 0.35SV_a$$

$$V_a = 12.857V$$

We know that V_b is 60 volts below V_a, so $V_b = -47.143$ volts.

To verify, we will perform a KCL summation at each node. For node a, assuming I_1 exits as drawn:

$$I_1 = 1A - \frac{V_a}{4\Omega} - \frac{V_a - V_b}{20\Omega}$$

$$I_1 = 1A - \frac{12.857V}{4\Omega} - \frac{12.857V - (-47.143V)}{20\Omega}$$

$$I_1 = -5.2143A \text{ (negative exit means it's entering)}$$

Doing likewise for node b, and assuming I_1 enters as drawn:

$$I_1 = 2.5A + \frac{V_b}{10\Omega} - \frac{V_a - V_b}{20\Omega}$$

$$I_1 = 2.5A + \frac{-47.143V}{10\Omega} - \frac{12.857V - (-47.143V)}{20\Omega}$$

$$I_1 = -5.2143A \text{ (negative enter means it's exiting)}$$

These currents match meaning that the current through the voltage source is verified to be the same at both terminals, as it must be.

An alternative to the basic supernode technique is to simply describe one node voltage in terms of another at the outset. This is illustrated in the example following.

Example 7.2.5

Find V_a, V_b and V_c for the circuit of Figure 7.2.15 .

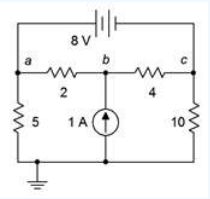


Figure 7.2.15 : Circuit for Example 7.2.5 .

Once again we have a situation of a voltage source without a series resistance. Without having to short it and thus treating nodes a and c as an explicit supernode, we can instead note by observation that the currents entering and exiting the voltage source must be identical.

The circuit is redrawn in Figure 7.2.16 with currents labeled and using equivalent conductances in place of the resistances. This time the voltage source is left in.

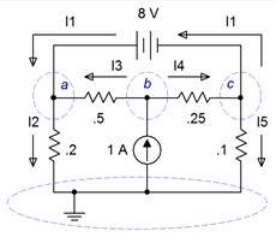


Figure 7.2.16 : Circuit of Example 7.2.5 with currents labeled and using conductances.

We start with the observation that $V_c = V_a - 8$ V. In other words, V_c is locked to V_a and if we find one of them, we can determine the other. Therefore, instead of writing equations using three nodes, we shall instead refer to node c in reference to node a, that is we shall write $V_a - 8$ V wherever we need V_c. Thus, this three node circuit will only need two equations.

We begin at node a and apply KCL as usual.

$$\sum I_{in} = \sum I_{out}$$

$$I_1 + I_3 = I_2$$

This is expanded using Ohm's law and we solve for I_1:

$$I_1 = I_2 - I3$$

$$I_1 = 0.2SV_a - 0.5S(V_b - V_a)$$

$$I_1 = 0.7SV_a - 0.5SV_b$$

On to node b:

$$1A = I_3 + I_4$$

$$1A = 0.5S(V_b - V_a) + 0.25S(V_b - V_c)$$

$$1A = 0.5S(V_b - V_a) + 0.25S(V_b - (V_a - 8V))$$

$$1A = 0.5S(V_b - V_a) + 0.25S(V_b - V_a + 8V)$$

$$-1A = -0.75V_a + 0.75V_b$$

And finally node c:

$$I_4 = I_1 + I_5$$

$$I_1 = I_4 - I_5$$

$$I_1 = 0.25S(V_b - V_c) - 0.1SV_c$$

$$I_1 = 0.25S(V_b - (V_a - 8V)) - 0.1S(V_a - 8V)$$

$$I_1 = 0.25S(V_b - V_a + 8V) - 0.1S(V_a - 8V)$$

$$I_1 = -0.35V_a + 0.25V_b + 2.8A$$

The final equations for nodes a and c both equal I_1, meaning they equal each other. Thus,

$$0.7SV_a - 0.5SV_b = -0.35SV_a + 0.25SV_b + 2.8A$$

$$2.8A = 1.05SV_a - 0.75SV_b$$

The final equations are:

$$2.8A = 1.05SV_a - 0.75SV_b$$

$$-1A = -0.75SV_a + 0.75SV_b$$

The solution is $V_a = 6$ volts and $Vb \approx 4.6667$ volts. As V_c is 8 volts less than V_a, then $V_c = -2$ volts.

Computer Simulation

In order to verify the result of Example 7.2.5 , the circuit is entered into a simulator as shown in Figure 7.2.17 .

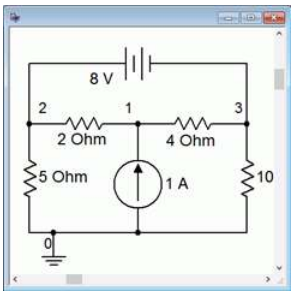


Figure 7.2.17 : Circuit of Example 7.2.5 in the simulator.

A DC operating point simulation is run. The results are shown in Figure 7.2.18 and match the calculated values perfectly. Node 1 corresponds to V_b, node 2 corresponds to V_a and node 3 corresponds to V_c.

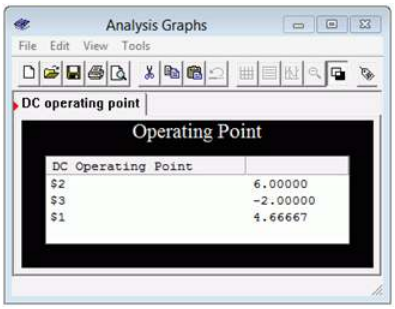


Figure 7.2.18 : Simulation results for the circuit of Example 7.2.5 .

This page titled 5.2: Nodal Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.2: Nodal Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

5.3: Mesh Analysis

In some respects mesh analysis is a mirror of nodal analysis. While nodal analysis leverages KCL to create a series of node equations that are used to solve for node voltages, mesh analysis uses KVL to create a series of loop equations that can be solved for mesh currents. A mesh current should not be confused with a branch current. While branch currents represent the current flowing through a particular component, mesh currents combine to create branch currents. That is, the current through any particular component may be an individual mesh current or a combination of two mesh currents. Circuits using complex series-parallel arrangement with multiple voltage and/or current sources may be solved using this technique. It is, however, limited to planar circuits. Planar circuits are those that can be drawn on a flat plane without having any of their wires cross. In essence, these circuits can be drawn so that they appear like a series of window panes. Non-planar circuits may be 3D in appearance and in such a circuit it becomes impossible to define the loop equations as any given component might have more than two meshing currents.

General Method

Consider the circuit of Figure 7.3.1 . We begin by designating a series of loops. These loops should be minimal in size and cover all components at least once. By convention, the loops are drawn clockwise. There is nothing magic about them being clockwise, it is just a matter of consistency. The current path along a loop is referred to as a mesh current. In the circuit of Figure 7.3.1 we have two loops, and thus, two mesh currents, I_1 and I_2. Note that all components exist in at least one loop (and sometimes in more than one loop, like R_3). Depending on circuit values, one or more of these loop's directions may in fact be opposite of reality. This is not a problem. If this is the case, the currents will show up as negative values, and thus we know that they're really flowing counter-clockwise.

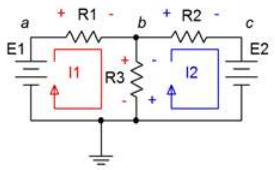


Figure 7.3.1 : Circuit for mesh analysis.

We begin by writing KVL equations for each loop.

$$\text{Loop 1: } E_1 = \text{ voltage across } R_1 + \text{ voltage across } R_3$$

$$\text{Loop 2: } -E_2 = \text{ voltage across } R_2 + \text{ voltage across } R_3$$

(E_2 is negative as I_2 is drawn flowing out of its negative terminal.)

Expand the voltage terms using Ohm's law.

$$\text{Loop 1: } E_1 = I_1 \cdot R_1 + (I_1 - I_2)R_3$$

$$\text{Loop 2: } -E_2 = I_2 \cdot R_2 + (I_2 - I_1)R_3$$

Expanding and collecting terms yields:

$$\text{Loop 1: } E_1 = (R_1 + R_3)I_1 - R_3 \cdot I_2$$

$$\text{Loop 2: } : -E_2 = R_3 \cdot I_1 + (R_2 + R_3)I_2$$

Assuming that the resistor values and source voltages are known, we have two equations with two unknowns. These can be solved for I_1 and I_2 using simultaneous equation solution techniques such as determinants or Gauss-Jordan elimination.

Example 7.3.1

Find V_b and V_{bc} for the circuit of Figure 7.3.2 .

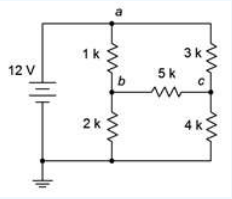


Figure 7.3.2 : Circuit for Example 7.3.1 .

The first step is to define a set of clockwise loops that are of minimum size and which cover all components. These loops are drawn in Figure 7.3.3 . Also, the polarities of the voltage drops produced by these currents are drawn next to the components in the same color for easy identification. Note that some components see only one current, such as the 3 kΩ resistor that sees only I_2; and some see two currents in opposing directions, such the 5 kΩ resistor that sees both I_2 and I_3. With multiple current directions flowing through a single component, obeying the resulting voltage polarities becomes very important.

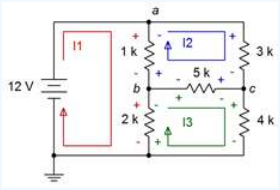


Figure 7.3.3 : Circuit of Example 7.3.1 with current loops and polarities added.

We now write KVL summations around each loop. If the current passing through a component sees a polarity of + to − then this is counted as a voltage drop while the reverse counts as a voltage rise. This is true even for voltage sources (i.e., a voltage source may appear negative in one loop and positive in another).

$$\sum V_{rises} = \sum V_{drops}$$

$$\text{Loop 1: } 12V = V_{1k} + V_{2k}$$

$$\text{Loop 2: } 0V = V_{3k} + V_{5k} + V_{1k}$$

$$\text{Loop 3: } 0V = V_{5k} + V_{4k} + V_{2k}$$

These are expanded using Ohm's law. The current for the loop under consideration is deemed positive while any opposing (i.e., meshing) current is deemed negative:

$$\text{Loop 1: } 12V = 1k(I_1 - I_2) + 2k(I_1 - I_2)$$

$$\text{Loop 2: } 0V = 3k(I_2) + 5k(I_2 - I_3) + 1k(I_2 - I_1)$$

$$\text{Loop 3: } 0V = 5k(I_3 - I_2) + 4k(I_3) + 2k(I_3 - I_1)$$

The terms are expanded:

$$\text{Loop 1: } 12V = 1kI_1 - 1kI_2 + 2kI_1 - 2kI_3$$

$$\text{Loop 2: } 0V = 3kI_2 + 5kI_2 - 5kI_3 + 1kI_2 - 1kI_1$$

$$\text{Loop 3: } 0V = 5kI_3 - 5kI_2 + 4kI_3 + 2kI_3 - 2kI_1$$

Like terms are grouped, once again creating aligned columns for the current coefficients:

$$\text{Loop 1: } 12V = (1k + 2k)I_1 - 1kI_2 - 2kI_3$$

$$\text{Loop 2: } 0V = -1kI_1 + (3k + 5k + 1k)I_2 - 5kI_3$$

$$\text{Loop 3: } 0V = -2kI_1 - 5kI_2 + (5k + 4k + 2k)I_3$$

Which simplifies to:

$$\text{Loop 1: } 12V = 3kI_1 - 1kI_2 - 2kI_3$$

$$\text{Loop 2: } 0V = -1kI_1 + 9kI_2 - 5kI_3$$

$$\text{Loop 3: } 0V = -2kI_1 - 5kI_2 + 11kI_3$$

Note that we have diagonal symmetry. The solution is I_1 = 5.729 mA, I_2 = 1.6258 mA and I_3 = 1.7806 mA. To find V_b we need to find the net current through the 2 kΩ resistor. That's $I_1 - I_3$, or 5.729 mA − 1.7806 mA, which is 3.9484 mA. When multiplied by 2 kΩ we find V_b = 7.8968 volts. Similarly, $V_{bc} = (I_3 - I_2)$ 5 kΩ = (1.7806 mA − 1.6258 mA) 5 kΩ, or 774 mV.

Computer Simulation

In order to verify the results of the analysis, the circuit is entered into a simulator and a virtual voltmeter is placed across the 5 kΩ resistor. This is shown in Figure 7.3.4 . The results agree nicely with the original analysis.

As nice as this is, in a practical circuit we need to be concerned about the effects of component tolerance. It would be extraordinarily odd if during production each resistor was equal to precisely the nominal value specified. In reality, each resistor will have a stated tolerance. Thus, with all five resistors changing slightly from unit to unit, we should expect the currents and voltages to vary as well. The question is, what would be a typical spread? This can be addressed via a Monte Carlo analysis. This analysis is, in fact, a series of simulations. Each simulation uses randomized component values that fall within the stated tolerance spread. For example, if we specify that the 1 kΩ resistor has a 5% tolerance, then the actual resistor used for a simulation will be a randomly generated value between 950 Ω and 1050 Ω. We then state how many trials we'd like, each one using their own unique randomized values for each component.

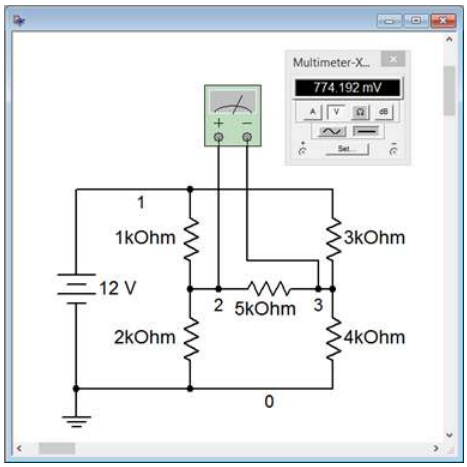


Figure 7.3.4 : Circuit of Figure 7.3.2 in a simulator with a virtual voltmeter.

The results of a Monte Carlo analysis using the DC operating point simulation is shown in Figure 7.3.5 . Ten randomized trials were generated for the value of V_b. Each of the five resistors was assigned a tolerance of 5%. As can be seen, voltage values both above and below the nominal simulation can result.

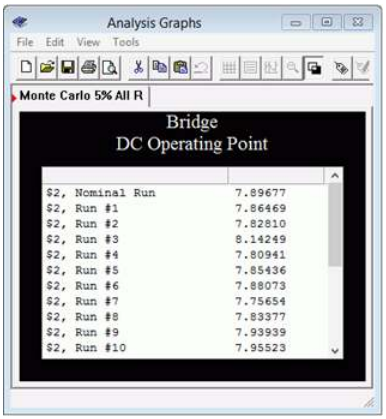


Figure 7.3.5 : Monte Carlo analysis results for the circuit of Figure 7.3.2 .

Inspection Method

Like nodal analysis, there is a method to generate the equations by inspection. Simply focus on one loop, which we'll call the loop under inspection, and ask the following questions: what is the total source voltage in this loop? This yields the voltage constant for the equation. Then sum all of the resistance values in the loop under inspection. This yields the coefficient for that current term. For the remaining current coefficients in this equation, sum the resistances that are in common between the loop under inspection and the other loops, respectively. These values will always be negative. Repeat this process for all loops, with each loop in turn becoming the loop under inspection. As was the case with nodal analysis, the set of equations produced must exhibit diagonal symmetry, that is, if a diagonal is drawn from the upper left to the lower right through the I_R pairs, then the coefficients found above the diagonal will have to match those found below the diagonal.

While it is possible to extend this technique to include current sources, it is often easier and less error-prone to convert the current sources into voltage sources and continue with the direct inspection method outlined above. Either way, it is important to remember that the number of loops determines the number of equations to be solved.

Example 7.3.2

Find V_c for the circuit of Figure 7.3.6 .

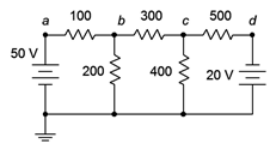


Figure 7.3.6 : Circuit for Example 7.3.2 .

First, we'll label the loops, as shown in Figure 7.3.7 .

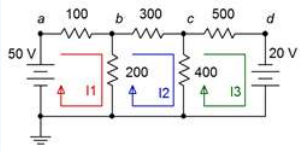


Figure 7.3.7 : Circuit of Example 7.3.2 with current loops identified.

Now, beginning with loop 1, sum all of the voltage sources in this loop. That's +50 volts. Secondly, sum all of the resistors in this loop. That's 100 Ω + 200 Ω, or 300 Ω. This is the coefficient for the I_1 term. The coefficient for the I_2 term is the sum of all resistances that are in common between loops 1 and 2. That's just the 200 Ω resistor. The coefficient for the I_3 term would be the sum of all resistances that are in common between loop 1 and loop 3. In this circuit, there aren't any so the coefficient is zero. The first equation is:

$$\text{Loop 1: } 50V = 300I_1 - 200I_2 - 0I_3$$

Repeat the process for loops 2 and 3:

$$\text{Loop 2: } 0V = -200I_1 + 900I_2 - 400I_3$$

$$\text{Loop 3: } 20V = -0I_1 - 400I_2 + 900I_3$$

Note that the 20 volt source shows up positive because of the direction of I_3 (into the negative terminal and out of the positive terminal).

As required, we have diagonal symmetry. The solution is I_1 = 214.47 mA, I_2 = 71.698 mA and I_3 = 54.088 mA. V_c is the net current through the 400 Ω resistor times 400 Ω. The net current is $I_2 - I_3$, or 71.698 mA − 54.088 mA, which is 17.61 mA. When multiplied by 400 Ω we find V_b = 7.044 volts. Note that we use $I_2 - I_3$ and not $I_3 - I_2$. The reason is because we are

trying to find the voltage from node c to ground, and I_2 is the current flowing in that direction. In contrast, using $I_3 - I_2$ would yield the voltage from ground to node c; the same magnitude but opposite sign.

As a crosscheck, KVL states that $V_c + V_{500}$ has to equal the −20 volt source. The drop across the 500 Ω is 54.088 mA times 500 Ω, or 27.044 volts from left to right. Starting at node c of 7.044 volts and then dropping 27.044 volts yields − 20 volts as expected.

Supermesh

Sometimes you may run across a current source which has no associated internal resistance, such as found in the circuit of Figure 7.3.8 . This is similar to the situation seen previously under nodal analysis where a voltage source does not have a specified internal resistance. There are two ways of solving this predicament. The first is add a very large resistance in parallel with the current source and then perform a source conversion on the pair so that the inspection method of mesh can be followed. The larger the value of this resistor, the more accuracy that is obtained. As a general rule it should be, at minimum, at least a couple of orders of magnitude larger than any surrounding resistor, and preferably larger. The second method is to use supermesh. A supermesh is a larger mesh loop than contains other mesh loops inside of it.

Consider the circuit shown in Figure 7.3.8 . In the center we have a current source, I_s, which lacks an associated internal resistance. Two traditional mesh loops, I_1 and I_2, are labeled as usual. The problem here is that we cannot use an Ohm's law based IR voltage drop for V_b. We have no way to express this as the voltage across I_s is an unknown. On the other hand, what we do know is that I_s must equal the combination of the traditional mesh currents I_1 and I_2. That is, from the perspective of the first loop, $I_s = I_1 - I_2$. Remember, one or both of the mesh currents could be negative, and thus rotating counterclockwise.

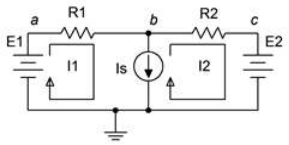


Figure 7.3.8 : Circuit for supermesh.

At this point we invoke the idea of a supermesh loop. First, we replace the offending current source with its ideal internal resistance (an open). The supermesh loop is drawn which encompasses the original two loops. This is shown in Figure 7.3.9 . The supermesh loop is drawn in red and labeled.

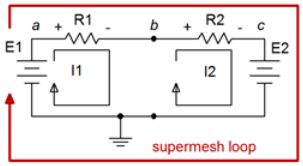


Figure 7.3.9 : Supermesh labeled.

We now perform a KVL summation around the supermesh loop, similar to what we have done previously. The difference this time is that we need to recognize that components each see one of the original mesh currents; namely I_1 or I_2 in this case. We do not solve for a supermesh current, we simply use the supermesh to define the loop for the KVL summation. The summation follows:

$$\sum V_{rises} = \sum V_{drops}$$

$$E_1 = V_{R1} + V_{R2} + E_2$$

The voltage drops across the resistors can be expanded using Ohm's law, using the original mesh current associated with each resistor.

$$E_1 - E_2 = I_1 R_1 + I_2 R_2$$

By inspection,

$$I_s = I_1 - I_2 \text{ or}$$

$$I_2 = I_1 - I_s$$

We now have two equations with two unknowns and can solve for I_1 and I_2. This procedure is illustrated in the following example.

Example 7.3.3

Find V_b for the circuit of Figure 7.3.10 .

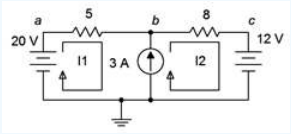


Figure 7.3.10 : Circuit for Example 7.3.3 .

First, we'll label the loops, as shown in Figure 7.3.11 .

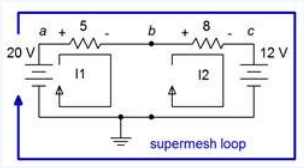


Figure 7.3.11 : Circuit of Figure 7.3.10 with supermesh labeled.

Next, we'll perform a KVL summation around the supermesh loop.

$$\sum V_{rises} = \sum V_{drops}$$

$$20V = V_{R1} + V_{R2} + 12V$$

Expand using Ohm's law and rearrange:

$$20V - 12V = 5\Omega I_1 + 8\Omega I_2$$

$$8V = 5\Omega I_1 + 8\Omega I_2$$

By inspection we know

$$3A = I_2 - I_1 \text{ or}$$

$$I_2 = I_1 + 3A$$

We can substitute this expression into the prior supermesh expression and solve for I1:

$$8V = 5\Omega I_1 + 8\Omega I_2$$

$$8V = 5\Omega I_1 + 8\Omega(I_1 + 3A)$$

$$8V = 5\Omega I_1 + 8\Omega I_1 + 24V$$

$$-16V = 13\Omega I_1$$

$$I_1 \approx -1.231A$$

Thus, I_2 = −1.231 A + 3 A, or 1.769 A. To determine V_b we simply subtract the drop across the 5 Ω resistor from the 20 volt source.

$$V_b = 20V - I_1 5\Omega$$

$$V_b = 20V - (-1.231A)5\Omega$$

$$V_b \approx 26.15V$$

As a crosscheck, we could also add the voltage across the 8 Ω resistor to the 12 volt source:

$$V_b = 12V + I_2 8\Omega$$

$$V_b = 12V + 1.769A8\Omega$$

$$V_b \approx 26.15V$$

Comparison of Nodal and Mesh

Compared to nodal analysis, mesh analysis has the advantage of dealing with resistances rather than conductances when writing the system of equations. Further, the mesh inspection method works with voltage sources which tends to be convenient for many circuits, while the nodal inspection method requires current sources. On the down side, the resulting set of mesh currents requires further processing in order to find either branch currents or node voltages, while nodal analysis produces node voltages directly with no further processing. Mesh also has the disadvantage of being limited to planar circuits while there are no such limits to nodal.

This page titled 5.3: Mesh Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.3: Mesh Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

5.4: Dependent Sources

A dependent source is a current or voltage source whose value is not fixed (i.e., independent) but rather which depends on some other circuit current or voltage. The general form for the value of a dependent source is $Y = kX$ where X and Y are currents and/or voltages and k is the proportionality factor. For example, the value of a dependent voltage source may be a function of a current, so instead of the source being equal to, say, 10 volts, it could be equal to twenty times the current passing through a particular resistor, or $V = 20I$.

There are four possible dependent sources: the voltage-controlled voltage source (VCVS), the current-controlled voltage source (CCVS), the voltage-controlled current source (VCCS), and the current-controlled current source (CCCS). The source and control parameters are the same for both the VCVS and the CCCS so k is unitless (although it may be given as volts/volt and amps/amp, respectively). For the VCCS and CCVS, k has units of amps/volt and volts/amp, respectively. These are referred to as the transresistance and transconductance of the sources with units of ohms and siemens.

The schematic symbols for dependent or controlled sources are usually drawn using a diamond. Also, there may be a secondary connection for the controlling current or voltage. Examples of a voltage-controlled voltage source, current-controlled voltage source, voltage-controlled current source and a current-controlled current source are shown in Figure 7.4.1 , left-to-right. On each of these symbols, the control element is shown to the left of the source. This portion is not always drawn on a schematic. Instead, the source simply may be labeled as a function, for example, $V = 0.02I_X$ where I_X is the controlling current.

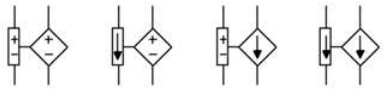


Figure 7.4.1 : Dependent sources (L to R): VCVS, CCVS, VCCS, CCCS.

Dependent sources are not “off-the-shelf” items in the same way that a battery is. Rather, dependent sources usually are used to model the behavior of more complex devices. For example, a bipolar junction transistor commonly is modeled as a CCCS while a field effect transistor may be modeled as a VCCS[1]. Similarly, many op amp circuits are modeled as VCVS systems. Solutions for circuits using dependent sources follow along the lines of those established for independent sources (i.e., the application of Ohm's law, KVL, KCL, etc.), however, the sources are now dependent on the remainder of the circuit which tends to complicate the analysis.

In general, there are two possible configurations: isolated and coupled. A example of the isolated form is shown in Figure 7.4.2 .

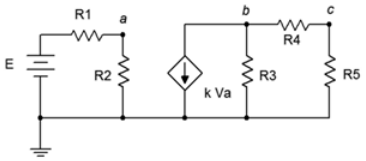


Figure 7.4.2 : Dependent source: isolated configuration.

In this configuration, the dependent source (center) does not interact with the subcircuit on the left driven by the independent source, thus it can be analyzed as two separate circuits. Solutions for this form are relatively straightforward in that the control value for the dependent source can be computed directly. This value is then substituted into the dependent source and the analysis continues as is usual. Sometimes it is convenient if the solution for a particular voltage or current is defined in terms of the control parameter rather than as a specific value (e.g., the voltage across a particular resistor might be expressed as 8 V_A instead of 12 volts, where V_A is 1.5 volts).

The second type of circuit (coupled) is somewhat more complex in that the dependent source can affect the parameter that controls the dependent source. In other words, the dependent source(s) will contribute terms that include the controlling parameter(s), thus partly controlling itself. Some additional effort will be in order to solve these circuits. To illustrate, consider the circuit of Figure 7.4.3 .

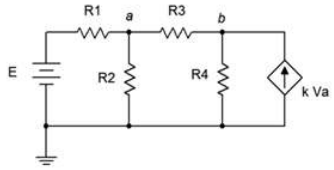


Figure 7.4.3 : Dependent source: coupled configuration.

In this example it should be obvious that the current from the dependent source can affect the voltage at node a, and it is this very voltage that in turn sets up the value of the current source. Circuits of this type can be analyzed using mesh or nodal analysis. Nodal analysis works well here and is illustrated following.

We begin by defining current directions. Assume that the currents through R_1 and R_3 are flowing into node a, the current through R_2 is flowing out of node a, and the current through R_4 is flowing out of node b. We shall number the branch currents to reflect the associated resistor. The resulting KCL equations are:

$$\sum I_{in} = \sum I_{out}$$

$$\text{Node } a : I_1 + I_3 = I_2$$

$$\text{Node } b : kV_a = I_3 + I_4$$

The currents are then described by their Ohm's law equivalents:

$$\text{Node } a : \frac{E - V_a}{R_1} + \frac{V_b - V_a}{R_3} = \frac{V_a}{R_2}$$

$$\text{Node } b : kV_a = \frac{V_b - V_a}{R_2} + \frac{V_b}{R_4}$$

Expanding terms yields:

$$\text{Node } a : \frac{E}{R_1} - \frac{V_a}{R_1} + \frac{V_b}{R_3} - \frac{V_a}{R_3} = \frac{V_a}{R_2}$$

$$\text{Node } b : kV_a = \frac{V_b}{R_2} - \frac{V_a}{R_2} + \frac{V_b}{R_4}$$

Collecting terms and simplifying yields:

$$\text{Node } a : \frac{E}{R_1} = \left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}\right) V_a - \frac{1}{R_3} V_b$$

$$\text{Node } b : 0 = -\left(k + \frac{1}{R_2}\right) V_a + \left(\frac{1}{R_2} + \frac{1}{R_4}\right) V_b$$

Values for the resistors, k, and E normally are known, so the analysis proceeds directly.

Also, it is worth remembering that it is possible to perform source conversions on dependent sources, within limits. The same procedure is followed as for independent sources. The new source will remain a dependent source (e.g., converting VCVS to VCCS). This process is not applicable if the control parameter directly involves the internal impedance (i.e., is its voltage or current).

Time for an example, this one utilizing a simplified model of a transistor amplifier.

Example 7.4.1

Find V_b and V_c for the circuit of Figure 7.4.4 .

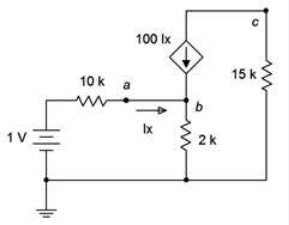


Figure 7.4.4 : Circuit for Example 7.4.1 .

This CCCS would be typical of a simple model of a bipolar junction transistor (nodes a, b, and c). Ideally, the output voltage, V_c, would be equal to the input voltage (1 V) times the resistor ratio of 15 kΩ / 2 kΩ and inverted, or approximately −7.5 volts. In reality, it usually comes up a little short of this. Let's see how well this works out.

This circuit is a good candidate for nodal analysis using the general method. Note that the points labeled a and b are the same node so we can write just two KCL summations. Using the current directions as draw on the schematic, for node b we have:

$$\sum I_{in} = \sum I_{out}$$

$$I_x + 100I_x = \frac{V_b}{2k\Omega}$$

$$101\frac{1V - V_b}{10k\Omega} = \frac{V_b}{2k\Omega}$$

$$10.1mA = \left(\frac{1}{2k\Omega} + \frac{101}{10k\Omega}\right)V_b$$

$$10.1mA = 10.6mSV_b$$

$$V_b = 0.95283V$$

For node c we have:

$$\sum Iin = \sum Iout$$

$$-\frac{V_c}{15k\Omega} = 100I_x$$

$$-\frac{V_c}{15k\Omega} = 100\frac{1V - V_b}{10k\Omega}$$

$$V_c = -7.0755V$$

Alternately, instead of writing the second KCL summation we could have used V_b to determine I_x, i.e., $(1 - V_b)/10$ kΩ. As the current through the 15 kΩ resistor is $100I_x$, we could then use Ohm's law to find V_c. In any case, we do see that V_c is inverted and just shy of the 7.5 volt estimate.

Computer Simulation

For verification, the dependent source circuit of Example 7.4.1 is entered into a simulator as shown in Figure 7.4.5 . A DC operating point analysis is run, the results of which are shown in Figure 7.4.6 . The output voltage shows roughly −7.08 volts and a V_b of just under 1 volt, thus achieving excellent agreement with the manual calculation.

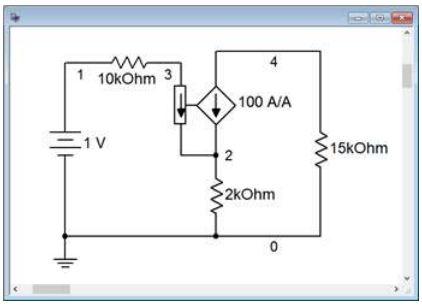


Figure 7.4.5 : Circuit of Figure 7.4.4 in a simulator.

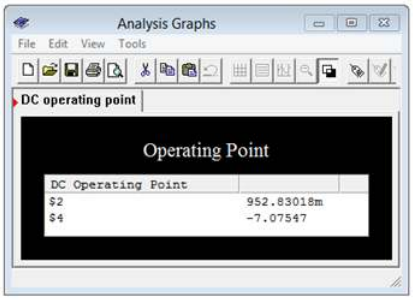


Figure 7.4.6 : Simulation results for the circuit of Figure 7.4.4 .

While this simulation is effective in this case, just using a dependent source to stand in for a transistor is rather limited. There are many other, perhaps more subtle, elements to a proper transistor model that will ensure both greater accuracy and proper results under a wide range of operating conditions. Any quality simulator will include a parts library using simulation models that are fine-tuned to particular manufacturer part numbers.

References

[1]For details, see Fiore, J.M., Semiconductor Devices: Theory and Application, and Operational Amplifiers and Linear Integrated Circuits: Theory and Application, both free OER titles.

This page titled 5.4: Dependent Sources is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.4: Dependent Sources** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

5.5: Summary

Nodal analysis can be used to solve virtually any complex multi-source DC electrical circuit. It is based on KCL, writing expressions involving each node in the circuit. A system of equations results, there being as many equations as there are nodes in the circuit, minus the reference node (which usually is taken as ground). The set of equations will exhibit diagonal symmetry, which can be used as a crosscheck before setting out to solve them. The solution will be a complete set of node voltages. From these, any branch current may be determined as needed.

There are two different methods of creating the system of equations. The first method is deemed the general method and will work for a mix of current sources and voltage sources. Individual currents are defined based on the node voltages and any known current sources. KCL is then applied at each node, followed by simplification and combination of terms to arrive at the end equations. The second approach uses the inspection method. If the circuit contains only current sources (or if the voltage sources are converted), this method allows direct generation of the system of equations without the need for simplification and thus is less prone to error.

Mesh analysis can be used to solve any planar complex multi-source DC electrical circuit. It is based on KVL, writing expressions involving each closed loop in the circuit. The loops are minimally sized and the set of loops must cover every component in the circuit. A system of equations results, there being as many equations as there are loops. As with nodal analysis, the set of equations will exhibit diagonal symmetry. The solution will be a complete set of mesh currents. From these, any node voltage may be determined.

Like nodal, mesh offers two different methods of creating the system of equations. The general method will work for a mix of current sources and voltage sources. Individual loops are defined based on the meshing currents passing through each component. KVL is then applied around each loop, followed by simplification and combination of terms to arrive at the end equations. In contrast, if the circuit contains only voltage sources (or if the current sources are converted), then the inspection method may be used. This method allows direct generation of the system of equations and is faster and less error prone.

Dependent sources are current or voltage sources whose value depends on the current or voltage developed in some other part of the circuit. There are four types: current controlled current source, current controlled voltage source, voltage controlled current source and voltage controlled voltage source. These sources are used commonly to model the characteristics of active devices such bipolar and field effect transistors. Techniques for solution tend to be a bit more involved than when using constant sources, however, nodal analysis in particular tends to work well.

Review Questions

1. Describe the practical differences between nodal analysis and mesh analysis.
2. What is diagonal symmetry? Of what use is it?
3. What are the differences between the general method and the inspection method of nodal analysis?
4. What are the differences between the general method and the inspection method of mesh analysis?
5. What is a supernode?
6. What is a supermesh?
7. Describe the concept of dependent sources and how they differ from independent or constant sources.

This page titled 5.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

5.6: Exercises

Analysis

1. Given the circuit in Figure 7.6.1 , write the mesh loop equations.

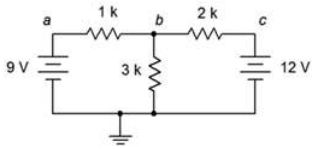


Figure 7.6.1

2. Using mesh analysis, determine the value of V_b for the circuit shown in Figure 7.6.1 .

3. For the circuit shown in Figure 7.6.1 , use mesh analysis to determine the current through the 1 kΩ resistor.

4. Given the circuit in Figure 7.6.2 , write the mesh loop equations and the associated determinants.

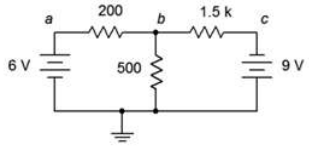


Figure 7.6.2

5. Using mesh analysis, determine the value of V_b for the circuit shown in Figure 7.6.2 .

6. For the circuit shown in Figure 7.6.2 , use mesh analysis to determine the current through the 500 Ω resistor.

7. Given the circuit in Figure 7.6.3 , write the mesh loop equations.

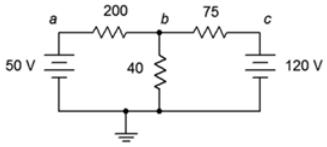


Figure 7.6.3

8. Using mesh analysis, determine the value of V_b for the circuit shown in Figure 7.6.3 .

9. For the circuit shown in Figure 7.6.3 , use mesh analysis to determine the current through the 75 Ω resistor.

10. Given the circuit in Figure 7.6.4 , write the mesh loop equations and the associated determinants.

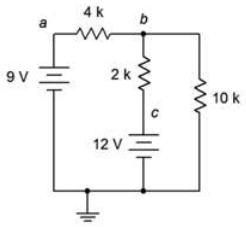


Figure 7.6.4

11. Using mesh analysis, determine the value of V_{bc} for the circuit shown in Figure 7.6.4 .

12. For the circuit shown in Figure 7.6.4 , use mesh analysis to determine the current through the 10 kΩ resistor.

13. Given the circuit in Figure 7.6.5 , write the mesh loop equations.

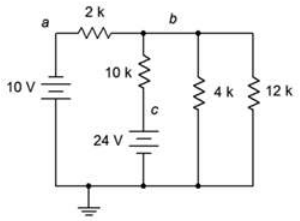


Figure 7.6.5

14. Using mesh analysis, determine the value of V_{ac} for the circuit shown in Figure 7.6.5 .
15. For the circuit shown in Figure 7.6.5 , use mesh analysis to determine the current through the 4 kΩ resistor.
16. Given the circuit in Figure 7.6.6 , write the mesh loop equations and the associated determinants.

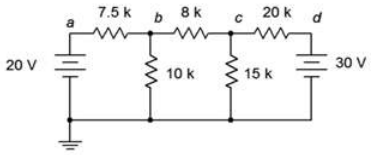


Figure 7.6.6

17. Using mesh analysis, determine the value of V_c for the circuit shown in Figure 7.6.6 .
18. For the circuit shown in Figure 7.6.6 , use mesh analysis to determine the current through the 10 kΩ resistor.
19. Given the circuit in Figure 7.6.7 , write the mesh loop equations.

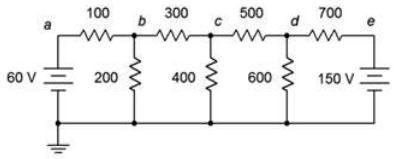


Figure 7.6.7

20. Using mesh analysis, determine the value of V_{bd} for the circuit shown in Figure 7.6.7 .
21. For the circuit shown in Figure 7.6.7 , use mesh analysis to determine the current through the 500 Ω resistor.
22. Given the circuit in Figure 7.6.8 , write the mesh loop equations.

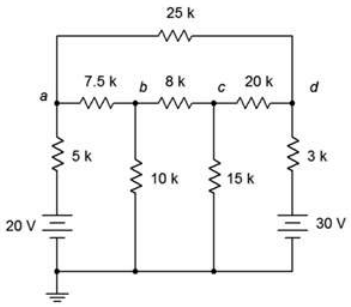


Figure 7.6.8

23. Using mesh analysis, determine the value of V_{ad} for the circuit shown in Figure 7.6.8 .
24. For the circuit shown in Figure 7.6.8 , use mesh analysis to determine the current through the 3 kΩ resistor.
25. Given the circuit in Figure 7.6.9 , write the mesh loop equations.

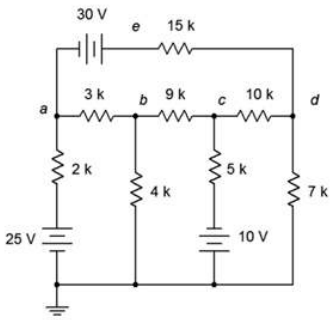


Figure 7.6.9

26. Using mesh analysis, determine the value of V_e for the circuit shown in Figure 7.6.9 .

27. For the circuit shown in Figure 7.6.9 , use mesh analysis to determine the current through the 9 kΩ resistor.

28. Given the circuit in Figure 7.6.10 , write the mesh loop equations and the associated determinants.

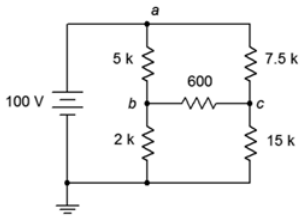


Figure 7.6.10

29. Using mesh analysis, determine the value of V_{bc} for the circuit shown in Figure 7.6.10 .

30. Given the circuit shown in Figure 7.6.10 , use mesh analysis to determine the current through the 600 Ω resistor.

31. Given the circuit in Figure 7.6.11 , write the mesh loop equations.

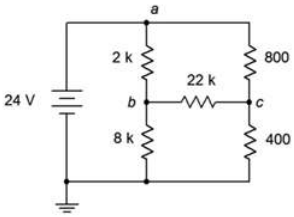


Figure 7.6.11

32. Using mesh analysis, determine the value of V_{bc} for the circuit shown in Figure 7.6.11 .

33. Given the circuit shown in Figure 7.6.11 , use mesh analysis to determine the current through the 800 Ω resistor.

34. For the circuit in Figure 7.6.12 , write the mesh loop equations.

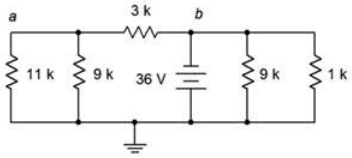


Figure 7.6.12

35. Using mesh analysis, determine the value of V_a for the circuit shown in Figure 7.6.12 .

36. For the circuit shown in Figure 7.6.12 , use mesh analysis to determine the current through the 3 kΩ resistor.

37. Given the circuit in Figure 7.6.13 , write the mesh loop equations.

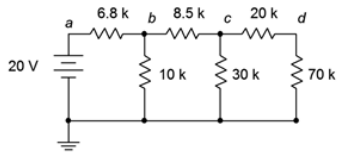


Figure 7.6.13

38. Using mesh analysis, determine the value of V_c for the circuit shown in Figure 7.6.13 .

39. For the circuit shown in Figure 7.6.13 , use mesh analysis to determine the current passing through the 8.5 kΩ resistor.

40. Given the circuit in Figure 7.6.14 , write the mesh loop equations (consider using source conversion).

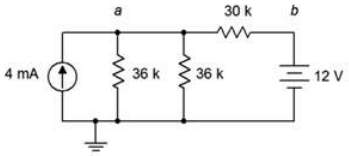


Figure 7.6.14

41. Using mesh analysis, determine the value of V_a for the circuit shown in Figure 7.6.14 .

42. For the circuit shown in Figure 7.6.14 , use mesh analysis to determine the current passing through the 30 kΩ resistor.

43. Given the circuit in Figure 7.6.15 , write the mesh loop equations (consider using source conversion).

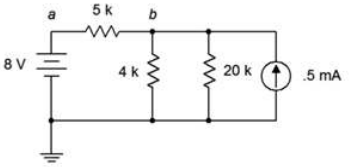


Figure 7.6.15

44. Using mesh analysis, determine the value of V_b for the circuit shown in Figure 7.6.15 .

45. For the circuit shown in Figure 7.6.15 , use mesh analysis to determine the current through the 5 kΩ resistor.

46. Given the circuit in Figure 7.6.16 , write the node equations.

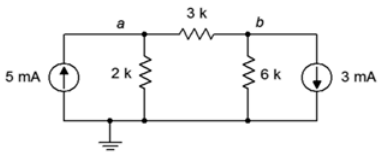


Figure 7.6.16

47. Using nodal analysis, determine the value of V_b for the circuit shown in Figure 7.6.16 .

48. For the circuit shown in Figure 7.6.16 , use nodal analysis to determine the current through the 3 kΩ resistor.

49. Given the circuit in Figure 7.6.17 , write the node equations.

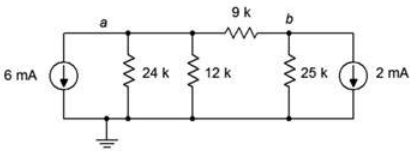


Figure 7.6.17

50. Using nodal analysis, determine the value of V_b for the circuit shown in Figure 7.6.17 .

51. For the circuit shown in Figure 7.6.17 , use nodal analysis to determine the current passing through the 12 kΩ resistor.

52. Given the circuit in Figure 7.6.18 , write the node equations.

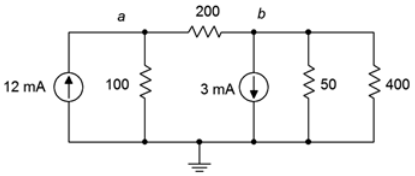


Figure 7.6.18

53. Using nodal analysis, determine the value of V_{ba} for the circuit shown in Figure 7.6.18 .

54. For the circuit shown in Figure 7.6.18 , use nodal analysis to determine the current passing through the 100 Ω resistor.

55. Given the circuit in Figure 7.6.19 , write the node equations.

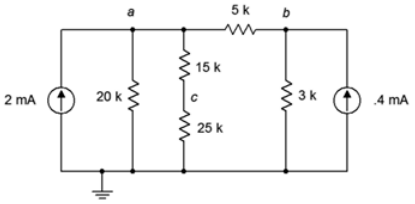


Figure 7.6.19

56. Using nodal analysis, determine the value of V_{ac} for the circuit shown in Figure 7.6.19 .

57. For the circuit shown in Figure 7.6.19 , use nodal analysis to determine the current passing through the 20 kΩ resistor.

58. Given the circuit in Figure 7.6.20 , write the node equations.

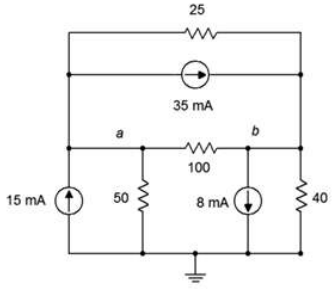


Figure 7.6.20

59. Using nodal analysis, determine the value of V_b for the circuit shown in Figure 7.6.20 .

60. For the circuit shown in Figure 7.6.20 , use nodal analysis to determine the current through the 3 kΩ resistor.

61. Given the circuit in Figure 7.6.21 , write the node equations.

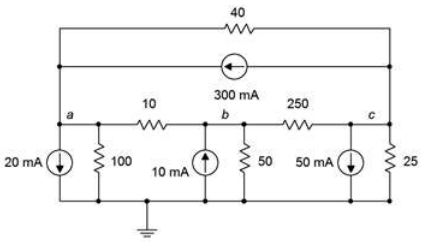


Figure 7.6.21

62. Using nodal analysis, determine the value of V_b for the circuit shown in Figure 7.6.21 .

63. For the circuit shown in Figure 7.6.21 , use nodal analysis to determine the current through the 40 Ω resistor.

64. Given the circuit in Figure 7.6.13 , write the node equations.

65. Using nodal analysis, determine the value of V_d for the circuit shown in Figure 7.6.13 .

66. For the circuit shown in Figure 7.6.13 , use nodal analysis to determine the current passing through the 20 kΩ resistor.

67. Given the circuit in Figure 7.6.14 , write the node equations using the general approach. Do not use source conversions.

68. Using nodal analysis, determine the value of V_{ab} for the circuit shown in Figure 7.6.14 .

69. For the circuit shown in Figure 7.6.14 , use nodal analysis to determine the current passing through the 30 kΩ resistor.

70. Given the circuit in Figure 7.6.15 , write the node equations.

71. Using nodal analysis, determine the value of V_b for the circuit shown in Figure 7.6.15 .

72. For the circuit shown in Figure 7.6.15 , use nodal analysis to determine the current through the 4 kΩ resistor.

73. For the circuit of Figure 7.6.22 , determine V_b.

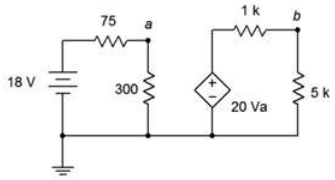


Figure 7.6.22

74. For the circuit of Figure 7.6.23 , determine V_c.

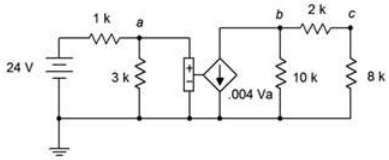


Figure 7.6.23

75. Given the circuit of Figure 7.6.24 , determine V_b.

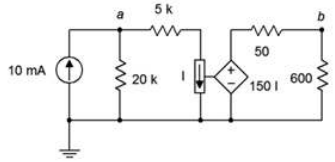


Figure 7.6.24

76. Find the current through the 10 kΩ resistor given the circuit of Figure 7.6.25 .

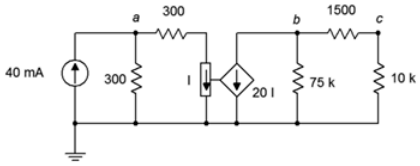


Figure 7.6.25

77. Given the circuit of Figure 7.6.26 , determine V_c.

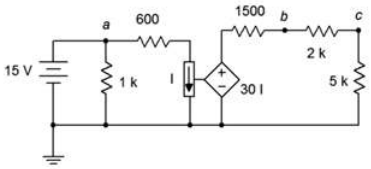


Figure 7.6.26

78. In the circuit of Figure 7.6.27 , determine V_a.

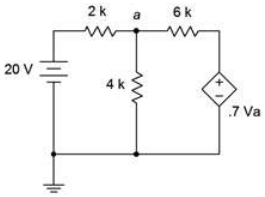


Figure 7.6.27

79. For the circuit of Figure 7.6.28 , determine V_a.

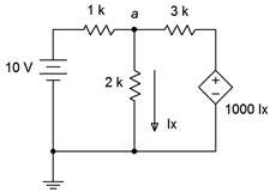


Figure 7.6.28

80. For the circuit of Figure 7.6.29 , determine V_a.

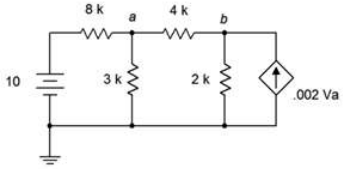


Figure 7.6.29

Challenge

81. Given the circuit in Figure 7.6.8 , write the node equations.

82. Using nodal analysis, determine the value of V_b for the circuit shown in Figure 7.6.8 .

83. For the circuit shown in Figure 7.6.8 , use nodal analysis to determine the current through the 3 kΩ resistor.

84. Given the circuit in Figure 7.6.6 , write the node equations.

85. Using nodal analysis, determine the value of V_c for the circuit shown in Figure 7.6.6 .

86. For the circuit shown in Figure 7.6.6 , use nodal analysis to determine the current through the 8 kΩ resistor.

87. Given the circuit in Figure 7.6.10 , write the node equations and the associated determinants.

88. Using nodal analysis, determine the value of V_{bc} for the circuit shown in Figure 7.6.10 .

89. For the circuit shown in Figure 7.6.10 , use nodal analysis to determine the current through the 2 kΩ resistor.

90. Given the circuit of Figure 7.6.30 , determine V_c.

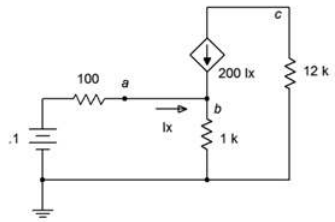


Figure 7.6.30

91. Find the current through the 10 kΩ resistor in the circuit of Figure 7.6.31 .

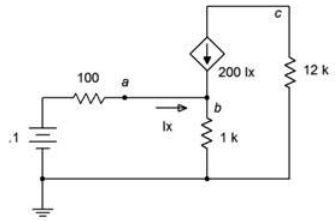


Figure 7.6.31

92. Given the circuit of Figure 7.6.32 , determine V_c.

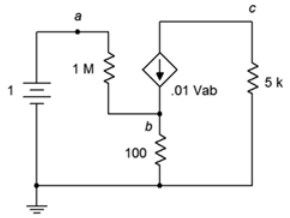


Figure 7.6.32

93. Given the circuit of Figure 7.6.33 , determine the current through the 5 kΩ resistor.

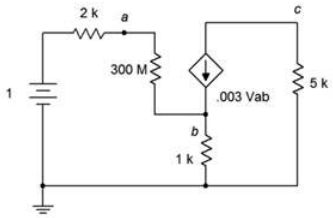


Figure 7.6.33

94. For the circuit of Figure 7.6.34 , determine V_b.

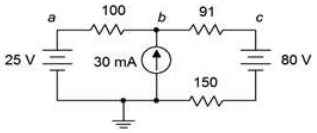


Figure 7.6.34

95. For the circuit of Figure 7.6.35 , determine V_c.

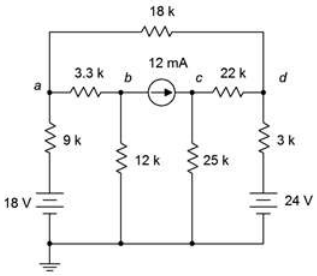


Figure 7.6.35

96. For the circuit of Figure 7.6.36 , determine V_a.

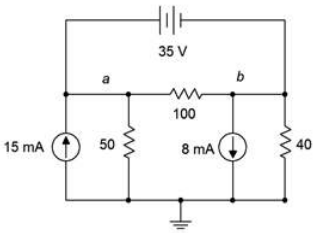


Figure 7.6.36

97. For the circuit of Figure 7.6.37 , determine V_b.

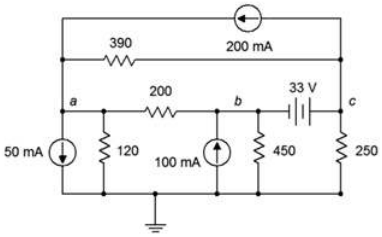


Figure 7.6.37

Simulation

98. Perform a DC bias simulation on the circuit depicted in Figure 7.6.7 to verify the component currents.

99. Perform a DC bias simulation on the circuit depicted in Figure 7.6.9 to verify the component currents.

100. Perform a DC bias simulation on the circuit depicted in Figure 7.6.7 to verify the loop currents and node voltages.

101. Perform a DC bias simulation on the circuit depicted in Figure 7.6.16 to verify the node voltages.

102. Perform a DC bias simulation on the circuit depicted in Figure 7.6.19 to verify the node voltages.

103. Perform a DC bias simulation on the circuit depicted in Figure 7.6.20 to verify the node voltages.

This page titled 5.6: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

6: Capacitors and Inductors

6.1: Capacitors

6.1.1: Introduction

6.1.2: Capacitance and Capacitors

6.1.3: Summary

6.1.4: Exercises

6.2: Inductors

6.2.1: Introduction

6.2.2: Inductance and Inductors

6.2.3: Summary

6.2.4: Exercises

This page titled 6: Capacitors and Inductors is shared under a not declared license and was authored, remixed, and/or curated by James M. Fiore.

SECTION OVERVIEW

6.1: Capacitors

Learning Objectives

After completing this chapter, you should be able to:

- Describe the theoretical and practical aspects of capacitor construction.
- Describe the current-voltage characteristic behavior of capacitors.
- Utilize component data sheets to determine operating characteristics of capacitors.
- Determine the initial and steady-state equivalents of resistor-capacitor networks.
- Determine the transient response of basic RC networks.

6.1.1: Introduction

6.1.2: Capacitance and Capacitors

6.1.3: Summary

6.1.4: Exercises

This page titled 6.1: Capacitors is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

6.1.1: Introduction

This chapter introduces another passive device, the capacitor. Capacitors are fundamentally different from resistors in terms of both their construction and their operation. For starters, when placed in DC circuits, capacitors are not ohmic, unlike resistors. Their current-voltage characteristic does not respond to Ohm's law. Instead, their current-voltage characteristic is dynamic in nature. Further, in the ideal case, capacitors do not dissipate power. Indeed, capacitors are energy storage devices. In a way, you could imagine them to be a little like rechargeable batteries, but unlike batteries these devices could be charged or discharged in tiny fractions of a second and last for billions upon billions of cycles.

Capacitors are an integral part of modern electronic systems. They are used in AC-to-DC power supplies to help smooth and stabilize the output voltage. In audio and communications systems they are used in filters, for example to control the high and low frequency response of amplifiers and similar equipment, or for tuning purposes. Essentially, in any application that needs to smooth out a varying voltage, store electric charge or filter a signal; a capacitor is likely to be used.

Much of the original work on capacitors was done in the 18th century. One of the earliest examples of a capacitor was the Leyden jar, which consists of a glass bottle or jar lined with metal foil on the inside and the outside. Benjamin Franklin's famous kite experiment made use of a Leyden jar. In the early days, capacitors were known as condensers (not to be confused with the condensers used in air conditioning systems).

This page titled 6.1.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

6.1.2: Capacitance and Capacitors

A capacitor is a device that stores energy. Capacitors store energy in the form of an electric field. At its most simple, a capacitor can be little more than a pair of metal plates separated by air. As this constitutes an open circuit, DC current will not flow through a capacitor. If this simple device is connected to a DC voltage source, as shown in Figure 8.2.1 , negative charge will build up on the bottom plate while positive charge builds up on the top plate. This process will continue until the voltage across the capacitor is equal to that of the voltage source. In the process, a certain amount of electric charge will have accumulated on the plates.

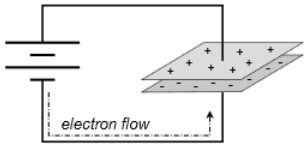


Figure 8.2.1 : Basic capacitor with voltage source.

The ability of this device to store charge with regard to the voltage appearing across it is called capacitance. Its symbol is C and it has units of farads (F), in honor of Michael Faraday, a 19th century English scientist who did early work in electromagnetism. By definition, if a total charge of 1 coulomb is associated with a potential of 1 volt across the plates, then the capacitance is 1 farad.

$$1 \text{ farad } \equiv 1 \text{ coulomb } /1 \text{ volt} \tag{6.1.2.1}$$

or more generally,

$$C = \frac{Q}{V} \tag{6.1.2.2}$$

Where

C is the capacitance in farads,

Q is the charge in coulombs,

V is the voltage in volts.

From Equation 6.1.2.2 we can see that, for any given voltage, the greater the capacitance, the greater the amount of charge that can be stored. We can also see that, given a certain size capacitor, the greater the voltage, the greater the charge that is stored. These observations relate directly to the amount of energy that can be stored in a capacitor.

Unsurprisingly, the energy stored in capacitor is proportional to the capacitance. It is also proportional to the square of the voltage across the capacitor.

$$W = \frac{1}{2}CV^2 \tag{6.1.2.3}$$

Where

W is the energy in joules,

C is the capacitance in farads,

V is the voltage in volts.

The basic capacitor consists of two conducting plates separated by an insulator, or dielectric. This material can be air or made from a variety of different materials such as plastics and ceramics. This is depicted in Figure 8.2.2 .

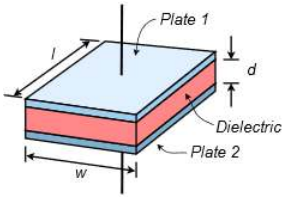


Figure 8.2.2 : Components of a generic capacitor.

For practical capacitors, the plates may be stacked alternately or even made of foil and formed into a rolled tube. However it is constructed, the characteristics of the dielectric will play a major role in the performance of the device, as we shall see.

In general, capacitance increases directly with plate area, A, and inversely with plate separation distance, d. Further, it is also proportional to a physical characteristic of the dielectric; the permittivity, ε. Thus, capacitance is equal to:

$$C = \varepsilon \frac{A}{d} \tag{6.1.2.4}$$

Where

C is the capacitance in farads,

A is the plate area in square meters,

d is the plate separation distance in meters,

ε is permittivity of the dielectric between the plates.

It should be noted that the effective plate area is somewhat larger than the precise physical area of the plates. This is due to a phenomenon called fringing. Essentially, the electric field lines bulge outward at the plate edges rather than maintain uniform parallel orientation. This is illustrated in Figure 8.2.3

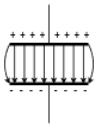


Figure 8.2.3 : Capacitor electric field with fringing.

From Equation 6.1.2.4 it is obvious that the permittivity of the dielectric plays a major role in determining the volumetric efficiency of the capacitor, in other words, the amount of capacitance that can be packed into a given sized component. Some dielectrics are notably more efficient than others. To make comparisons easier, relative permittivity is often used, that is, the ratio of the dielectric's permittivity to that of a vacuum, ε_0.

A table of relative permittivity for a variety of dielectrics is shown in Table 8.2.1 . A number of common dielectrics, such as various poly plastic films and mica, exhibit permittivities two to six times that of air, but there are also ceramic dielectrics whose dielectrics are hundreds to thousands of times that of air.

Table 8.2.1 : Relative permittivity of various dielectrics. Data derived from Wikipedia and other sources.

Material	Relative permittivity, $\varepsilon_r = \varepsilon/\varepsilon_0$
Vacuum	1 (ε_0 =8.85E−12 farads/meter)
Air	1.00058986 (at STP)
PTFE/Teflon	2.1
Polyethylene/XLPE	2.25
Polyimide	3.4
Polypropylene	2.2-2.36
Polystyrene	2.4-2.7
Polyester (Mylar)	3.1
Paper	1.4
Mica	3-6
Silicon dioxide	3.9
Rubber	7
Diamond	5.5-10
Silicon	11.68
Titanium dioxide	86-173
Strontium titanate	310
Calcium copper titanate	>250,000

At quick glance, it might seem that choosing the dielectric with the highest permittivity would be the best choice but this is not necessarily the case. There are several other factors that go into this decision including temperature stability, leakage resistance (effective parallel resistance), ESR (equivalent series resistance) and breakdown strength. For an ideal capacitor, leakage resistance would be infinite and ESR would be zero.

Unlike resistors, capacitors do not have maximum power dissipation ratings. Instead, they have maximum voltage ratings. The breakdown strength of the dielectric will set an upper limit on how large of a voltage may be placed across a capacitor before it is damaged. Breakdown strength is measured in volts per unit distance, thus, the closer the plates, the less voltage the capacitor can withstand. For example, halving the plate distance doubles the capacitance but also halves its voltage rating. Table 8.2.2 lists the breakdown strengths of a variety of different dielectrics. Comparing the tables of Tables 8.2.1 and 8.2.2 hints at the complexity of the situation. For instance, consider polystyrene versus polypropylene. Polystyrene offers modestly increased permittivity yet polypropylene has a considerable advantage in terms of breakdown strength. As a consequence, the plates can be placed much closer together when using polypropylene while achieving the same voltage rating as a capacitor using polystyrene. Therefore, the polypropylene capacitor will require less volume for the same capacitance. As an added benefit, polypropylene exhibits high temperature stability and low moisture absorption, among other characteristics. Comparing polypropylene to polyester, we find that polyester's improved permittivity along with a similar breakdown strength yields improved volumetric efficiency over polypropylene. Unfortunately, polyester suffers from greater temperature dependence.

Table 8.2.2 : Dielectric strength of various dielectrics. Data derived from Wikipedia and other sources.c

Substance	Breakdown Strength (kV/mm)
Air	3.0
Borosilicate glass	20-40
PTFE (Teflon, insulating film)	60-173
Polyethylene	19-160
Polypropylene	650
Polystyrene	19.7
PEEK (Polyether ether ketone)	23
Polyester (Mylar)	580
Neoprene rubber	15.7-26.7
Distilled water	65-70
Waxed paper	40-60
Mica	118
Diamond	2,000
PZT (ceramic)	10-25

Capacitor Styles and Packaging

Capacitors are available in a wide range of capacitance values, from just a few picofarads to well in excess of a farad, a range of over 10^{12}. Unlike resistors, whose physical size relates to their power rating and not their resistance value, the physical size of a capacitor is related to both its capacitance and its voltage rating (a consequence of Equation 6.1.2.4. Modest surface mount capacitors can be quite small while the power supply filter capacitors commonly used in consumer electronics devices such as an audio amplifier can be considerably larger than a D cell battery. A sampling of capacitors is shown in Figure 8.2.4 .



Figure 8.2.4 : A variety of capacitor styles and packages.

Toward the front and left side of the photo are a variety of plastic film capacitors. The disk-shaped capacitor uses a ceramic dielectric. The small square device toward the front is a surface mount capacitor, and to its right is a teardrop-shaped tantalum capacitor, commonly used for power supply bypass applications in electronic circuits. The medium sized capacitor to the right with folded leads is a paper capacitor, at one time very popular in audio circuitry. A number of capacitors have a crimp ring at one side, including the large device with screw terminals. These are aluminum electrolytic capacitors. These devices tend to exhibit high volumetric efficiency but generally do not offer top performance in other areas such as absolute accuracy and leakage current. They usually are polarized, meaning that the leads must match the polarity of the applied voltage. Inserting them into a circuit backwards can result in catastrophic failure. The polarity is usually identified by a series of minus signs and/or a stripe that indicates the

negative lead. Tantalum capacitors are also polarized but are typically denoted with a plus sign next to the positive lead. A variable capacitor used for tuning radios is shown in Figure 8.2.5 . One set of plates is fixed to the frame while an intersecting set of plates is affixed to a shaft. Rotating the shaft changes the amount of plate area that overlaps, and thus changes the capacitance.



Figure 8.2.5 : A variable capacitor.

For large capacitors, the capacitance value and voltage rating are usually printed directly on the case. Some capacitors use “MFD” which stands for “microfarads”. While a capacitor color code exists, rather like the resistor color code, it has generally fallen out of favor. For smaller capacitors a numeric code is used that echoes the color code. Typically it consists of a three digit number such as “152”.

The first two digits are the precision portion and the third digit is the power of ten multiplier. The result is in picofarads. Thus, 152 is 1500 pf.

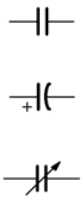


Figure 8.2.6 : Capacitor schematic symbols (top-bottom): non-polarized, polarized, variable.

The schematic symbols for capacitors are shown in Figure 8.2.6 . There are three symbols in wide use. The first symbol, using two parallel lines to echo the two plates, is for standard non-polarized capacitors. The second symbol represents polarized capacitors. In this variant, the positive lead is drawn with a straight line for that plate and often denoted with a plus sign. The negative terminal is drawn with a curved line. The third symbol is used for variable capacitors and is drawn with an arrow through it, rather like a rheostat.



Figure 8.2.7 : An LCR meter, designed to read capacitance, resistance and inductance.

In order to obtain accurate measurements of capacitors, an LCR meter, such as the one shown in Figure 8.2.7 , may be used. These devices are designed to measure the three common passive electrical components: resistors, capacitors and inductors[1]. Unlike a simple digital multimeter, an LCR meter can also measure the values at various AC frequencies instead of just DC, and also determine secondary characteristics such as equivalent series resistance and effective parallel leakage resistance.

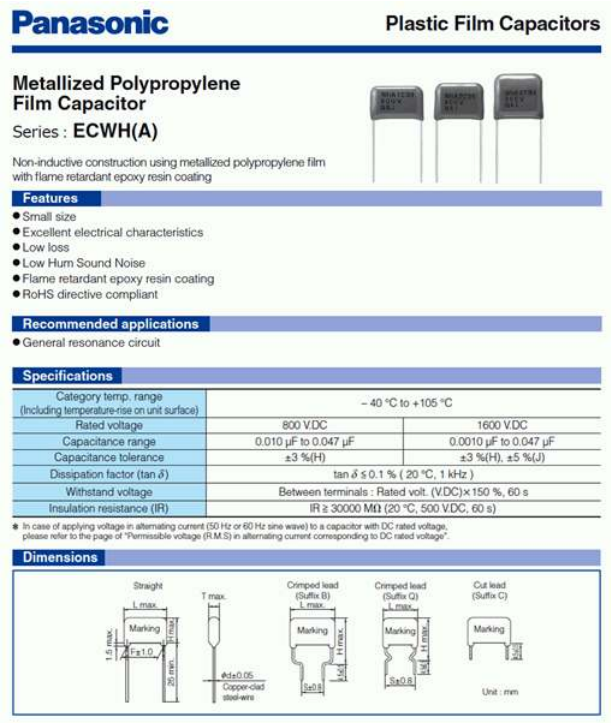
Capacitor Data Sheet

A portion of a typical capacitor data sheet is shown in Figure 8.2.8 . This is for a series of through-hole style metallized film capacitors using polypropylene for the dielectric. First we see a listing of general features. For starters, we find that the capacitors use a flame retardant epoxy coating and are also RoHS compliant. We then move to a set of electrical performance specifications. For example, we see that this series is available in two variants, one rated at 800 volts DC and the other rated at 1600 volts DC. Further, tolerance is available as either $\pm 3\%$ or $\pm 5\%$. Dissipation factor $(\tan\delta)$ is a measure of particular importance for AC

operation and is proportional to the ESR (equivalent series resistance, ideally 0), smaller being better. The insulation resistance indicates the value of an effective parallel leakage resistance (higher is better), here, some 30,000 MΩ. Finally, we see physical size data, essential for printed circuit board layouts.

Capacitors in Series and in Parallel

Multiple capacitors placed in series and/or parallel do not behave in the same manner as resistors. Placing capacitors in parallel increases overall plate area, and thus increases capacitance, as indicated by Equation 6.1.2.4. Therefore capacitors in parallel add in value, behaving like resistors in series. In contrast, when capacitors are placed in series, it is as if the plate distance has increased, thus decreasing capacitance. Therefore capacitors in series behave like resistors in parallel. Their value is found via the reciprocal of summed reciprocals or the product-sum rule.



Panasonic — Plastic Film Capacitors

Metallized Polypropylene Film Capacitor

Series : **ECWH(A)**

Non-inductive construction using metallized polypropylene film with flame retardant epoxy resin coating

Features

- Small size
- Excellent electrical characteristics
- Low loss
- Low Hum Sound Noise
- Flame retardant epoxy resin coating
- RoHS directive compliant

Recommended applications

- General resonance circuit

Specifications

Category temp. range (Including temperature-rise on unit surface)	− 40 °C to +105 °C	
Rated voltage	800 V.DC	1600 V.DC
Capacitance range	0.010 µF to 0.047 µF	0.0010 µF to 0.047 µF
Capacitance tolerance	±3 %(H)	±3 %(H), ±5 %(J)
Dissipation factor (tan δ)	tan δ ≤ 0.1 % (20 °C, 1 kHz)	
Withstand voltage	Between terminals : Rated volt. (V.DC)×150 %, 60 s	
Insulation resistance (IR)	IR ≥ 30000 MΩ (20 °C, 500 V.DC, 60 s)	

* In case of applying voltage in alternating current (50 Hz or 60 Hz sine wave) to a capacitor with DC rated voltage, please refer to the page of "Permissible voltage (R.M.S) in alternating current corresponding to DC rated voltage".

Dimensions

Straight, Crimped lead (Suffix B), Crimped lead (Suffix Q), Cut lead (Suffix C)

Unit : mm

Figure 8.2.8 : Capacitor data sheet. Courtesy of Panasonic

Example 8.2.1

Find the equivalent capacitance of the network shown in Figure 8.2.9 .

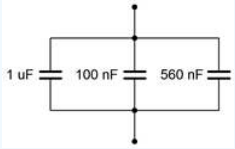


Figure 8.2.9 : Circuit for Example 8.2.1 .

These capacitors are all in parallel, and thus, the equivalent value is the sum of the three capacitances:

$$C_{Total} = C_1 + C_2 + C_3$$

$$C_{Total} = 1\mu F + 100nF + 560nF$$

$$C_{Total} = 1.66\mu F$$

Example 8.2.2

Find the equivalent capacitance of the network shown in Figure 8.2.10 .

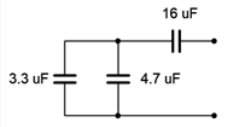


Figure 8.2.10 : Circuit for Example 8.2.2 .

In this circuit, we find that the left and middle capacitors are in parallel. This combination is in series with the capacitor to the right:

$$C_{left} = C_1 + C_2$$

$$C_{left} = 3.3\mu F + 4.7\mu F$$

$$C_{left} = 8\mu F$$

$$C_{Total} = \frac{C_{left}C_3}{C_{left} + C_3}$$

$$C_{Total} = \frac{8\mu F 16\mu F}{8\mu F + 16\mu F}$$

$$C_{Total} \approx 5.33\mu F$$

If a circuit contains nothing but a voltage source in parallel with a group of capacitors, the voltage will be the same across all of the capacitors, just as it is in a resistive parallel circuit. If the circuit instead consists of multiple capacitors that are in series with a voltage source, as shown in Figure 8.2.11 , the voltage will divide between them in inverse proportion. In other words, the larger the capacitance, the smaller its share of the applied voltage. The voltages can also be found by first determining the series equivalent capacitance. The total charge may then be determined using the applied voltage. Finally, the individual voltages are computed from Equation 6.1.2.2, $V = Q/C$, where Q is the total charge and C is the capacitance of interest. This is illustrated in the following example.

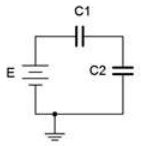


Figure 8.2.11 : A simple capacitors-only series circuit.

Example 8.2.3

Find the voltages across the capacitors in Figure 8.2.12 .

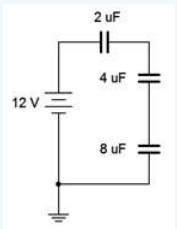


Figure 8.2.12 : Circuit for Example 8.2.3 .

The first step is to determine the total capacitance. As these are in series, we can use the reciprocal rule:

$$C_{Total} = \frac{1}{\frac{1}{C_1} + \frac{1}{C_2} + \frac{1}{C_3}}$$

$$C_{Total} = \frac{1}{\frac{1}{2\mu F} + \frac{1}{4\mu F} + \frac{1}{8\mu F}}$$

$$C_{Total} \approx 1.143\mu F$$

From here we determine the total charge:

$$Q = VC$$

$$Q = 12V1.143\mu F$$

$$Q = 13.71\mu C$$

Charge is constant across all of the series capacitors, therefore:

$$V_{2uF} = \frac{Q}{C}$$

$$V_{2uF} = \frac{13.71\mu C}{2\mu F}$$

$$V_{2uF} = 6.855V$$

$$V_{4uF} = \frac{Q}{C}$$

$$V_{4uF} = \frac{13.71\mu C}{4\mu F}$$

$$V_{4uF} = 3.427V$$

$$V_{8uF} = \frac{Q}{C}$$

$$V_{8uF} = \frac{13.71\mu C}{8\mu F}$$

$$V_{8uF} = 1.714V$$

The sum of the three voltages is 12 volts (within rounding error) and verifies KVL as expected.

Practical Tip

While it may be tempting to try, do not attempt to verify the operation of Example 8.2.3 in the laboratory using a standard DMM. The reason is because the internal resistance of a typical digital voltmeter is many orders of magnitude lower than the leakage resistance of the capacitors. As a result, charge will be transferred to the meter, ruining the measurement. It would be akin to trying to measure the voltages across a string of resistors, each in excess of 100 MΩ, with a meter whose internal resistance is 1 MΩ. The meter's resistance dominates the parallel combination and causes excessive loading which ruins the measurement. A special sort of voltmeter, an electrostatic voltmeter or electrometer, is needed for these types of measurements. These are sometimes referred to as non-charge transfer meters.

Current-Voltage Relationship

The fundamental current-voltage relationship of a capacitor is not the same as that of resistors. Capacitors do not so much resist current; it is more productive to think in terms of them reacting to it. The current through a capacitor is equal to the capacitance times the rate of change of the capacitor voltage with respect to time (i.e., its slope). That is, the value of the voltage is not important, but rather how quickly the voltage is changing. Given a fixed voltage, the capacitor current is zero and thus the capacitor behaves like an open. If the voltage is changing rapidly, the current will be high and the capacitor behaves more like a short. Expressed as a formula:

$$i = C\frac{dv}{dt} \tag{6.1.2.5}$$

Where

i is the current flowing through the capacitor,

C is the capacitance,

dv/dt is the rate of change of capacitor voltage with respect to time.

A particularly useful form of Equation 6.1.2.5 is:

$$\frac{dv}{dt} = \frac{i}{C} \tag{6.1.2.6}$$

An alternate way of looking at Equation 6.1.2.5 indicates that if a capacitor is fed by a constant current source, the voltage will rise at a constant rate (dv/dt). It is continuously depositing charge on the plates of the capacitor at a rate of I, which is equivalent to Q/t. As long as the current is present, feeding the capacitor, the voltage across the capacitor will continue to rise. A good analogy is if we had a pipe pouring water into a tank, with the tank's level continuing to rise. This process of depositing charge on the plates is referred to as charging the capacitor. For example, considering the circuit in Figure 8.2.13 , we see a current source feeding a single capacitor. If we were to plot the capacitor's voltage over time, we would see something like the graph of Figure 8.2.14 .

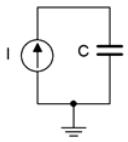


Figure 8.2.13 : Capacitor with current source.

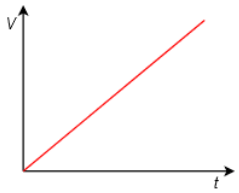


Figure 8.2.14 : Capacitor voltage versus time.

As time progresses, the voltage across the capacitor increases with a positive polarity from top to bottom. With a theoretically perfect capacitor and source, this would continue forever, or until the current source was turned off. In reality, this line would either begin to deflect horizontally as the source reached its limits, or the capacitor would fail once its breakdown voltage was reached. The slope of this line is dictated by the size of the current source and the capacitance.

Example 8.2.4

Determine the rate of change of voltage across the capacitor in the circuit of Figure 8.2.15 . Also determine the capacitor's voltage 10 milliseconds after power is switched on.

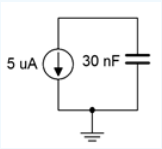


Figure 8.2.15 : Circuit for Example 8.2.4 .

First, note the direction of the current source. This will produce a negative voltage across the capacitor from top to bottom. The rate of change is:

$$\frac{dv}{dt} = \frac{i}{C}$$

$$\frac{dv}{dt} = \frac{-5\mu A}{30nF}$$

$$\frac{dv}{dt} \approx -166.7 \text{ volts per second}$$

Thus, for every second, the voltage rises another −166.7 volts. Assuming it is completely uncharged when power is applied, after 10 milliseconds it will have risen to −166.7 V/s times 10 ms, or −1.667 volts.

Equation 6.1.2.6 provides considerable insight into the behavior of capacitors. As just noted, if a capacitor is driven by a fixed current source, the voltage across it rises at the constant rate of i/C. There is a limit to how quickly the voltage across the capacitor can change. An instantaneous change means that dv/dt is infinite, and thus, the current driving the capacitor would also have to be infinite (an impossibility). This is not an issue with resistors, which obey Ohm's law, but it is a limitation of capacitors. Therefore we can state a particularly important characteristic of capacitors:

$$\text{The voltage across a capacitor cannot change instantaneously.} \tag{6.1.2.7}$$

This observation will be key to understanding the operation of capacitors in DC circuits.

References

[1]Inductors are the subject of the next chapter.

This page titled 6.1.2: Capacitance and Capacitors is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.2: Capacitance and Capacitors** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

6.1.3: Summary

The capacitor is a device that is used to store electric charge. Capacitance, C, is measured in farads, F. The idealized device consists of two conductive plates separated by some distance, that space being filled by an insulating dielectric. Capacitance is directly proportional to the plate area and the dielectric's permittivity, and inversely proportional to the plate distance. Another important characteristic of the dielectric is its breakdown strength. Along with the plate spacing, this will establish the capacitor's voltage rating. The permittivity will also help to determine the capacitor's volumetric efficiency, a measure of how much capacitance can be achieved within a given volume. Non-ideal parameters include the ESR, or equivalent series resistance, which is ideally zero; and the effective parallel leakage resistance, ideally infinity. Absolute accuracy, temperature stability and similar parameters round out the distinguishing features of one kind of capacitor against another. When placed in parallel, capacitors add in the same manner as resistors in series. When placed in series, capacitors behave like resistors in parallel.

Perhaps the most important operational characteristic regarding capacitors is that voltage across a capacitor cannot change instantaneously. It will take some finite amount of time before the charge on the capacitor builds, leading to a predictable rise in voltage across it. Because of this, for DC circuits capacitors initially behave like shorts, but after sufficient time has passed, they behave like opens. The amount of time required to reach steady-state is five time constants, where one time constant is defined as the product of the circuit's effective resistance and its capacitance. The current charge curve is of the shape ε^{-t}. The current starts at a maximum and eventually approaches zero as time passes. The corresponding voltage shape is of the form $1-\varepsilon^{-t}$. Here, the capacitor's voltage starts at zero and rises to some maximum value. The capacitor's voltage discharge curve effectively is swapped compared to the charge curve (e.g. voltage follows ε^{-t}).

Review Questions

1. What are the physical characteristics of capacitors and how do they affect capacitance?
2. Define the voltage-current characteristic for capacitors.
3. What is meant by a capacitor's volumetric efficiency?
4. How do the permittivity and breakdown strength of the dielectric affect the overall capacitance and voltage rating?
5. How do capacitors combine when placed in series and how do they combine when placed in parallel?
6. Define the initial and steady-state behavior of capacitors.
7. Define time constant for an RC circuit.
8. Describe the charge and discharge characteristics of RC circuits.

This page titled 6.1.3: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

6.1.4: Exercises

Analysis

1. For the circuit shown in Figure 8.6.1 , determine the effective capacitance.

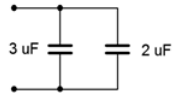


Figure 8.6.1

2. Determine the effective capacitance of the configuration shown in Figure 8.6.2 .

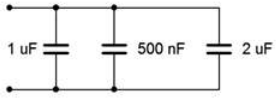


Figure 8.6.2

3. Given the capacitor network shown in Figure 8.6.3 , determine the effective value.



Figure 8.6.3

4. Determine the effective capacitance of network shown in Figure 8.6.4 .



Figure 8.6.4

5. Determine the voltage across each capacitor for the circuit shown in Figure 8.6.5 .

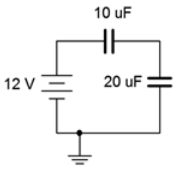


Figure 8.6.5

6. Determine the voltage across each capacitor for the circuit shown in Figure 8.6.6 .

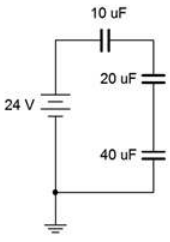


Figure 8.6.6

7. Determine the initial voltage across each component for the circuit shown in Figure 8.6.7 .

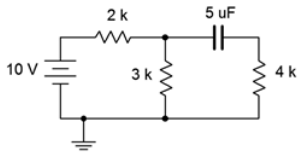


Figure 8.6.7

8. Given the network shown in Figure 8.6.7 , determine the steady-state voltage across each component.

9. For the circuit shown in Figure 8.6.8 , determine the capacitor voltage 3 microseconds after the power is switched on.

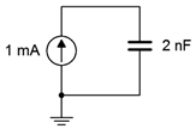


Figure 8.6.8

10. For the circuit shown in Figure 8.6.9 , determine the capacitor voltage 5 seconds after the power is switched on.

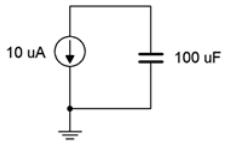


Figure 8.6.9

11. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.10 .

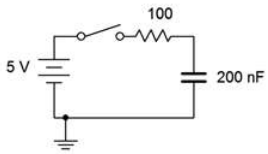


Figure 8.6.10

12. For the circuit shown in Figure 8.6.10 , determine the capacitor voltage and circulating current 20 microseconds and 100 milliseconds after the switch is thrown.

13. Given the circuit shown in Figure 8.6.11 , determine the capacitor voltage and circulating current 200 milliseconds and 10 seconds after the switch is thrown.

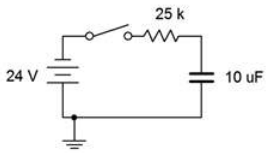


Figure 8.6.11

14. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.11 .

15. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.12 , switch position 1.

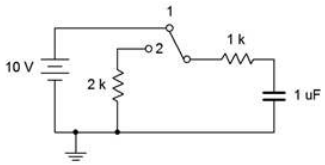


Figure 8.6.12

16. Determine the charge and discharge time constants for the circuit shown in Figure 8.6.13 .

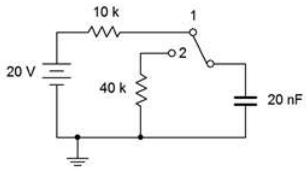


Figure 8.6.13

17. Given the circuit shown in Figure 8.6.12 , determine the capacitor voltage 10 milliseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

18. For the circuit shown in Figure 8.6.13 , determine the capacitor voltage 400 microseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

19. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.14 , switch position 1.

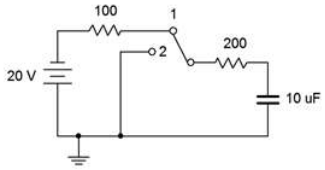


Figure 8.6.14

20. Given the circuit shown in Figure 8.6.14 , determine the capacitor voltage 12 milliseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

Design

21. Given the circuit of Figure 8.6.10 , determine a new resistor value such that steady-state is reached in 2 milliseconds.

22. Given the circuit of Figure 8.6.11 , determine a new resistor value such that steady-state is reached in 5 seconds.

23. When an audio amplifier is turned on, the power surge can cause an audible pop from the loudspeaker. To prevent this, amplifiers often connect to the loudspeaker via a relay. The relay is energized to connect the loudspeaker once the output has settled, typically a few seconds after power is applied. This delay may be created via an RC network. Suppose the driving circuit is 5 volts and the relay trips at 4 volts. Further, the associated charging resistance is 10 kΩ. Determine the capacitance required to achieve a 2 second delay time.

Challenge

24. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.15 .

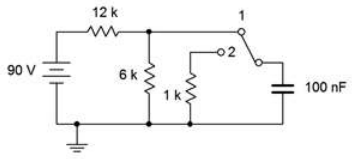


Figure 8.6.15

25. For the circuit shown in Figure 8.6.15 , determine the capacitor voltage 1 second after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

Simulation

26. Perform a transient analysis to verify the time to steady-state of Figure 8.6.10 (problem 11).

27. Perform a transient analysis to verify the time to steady-state of Figure 8.6.11 (problem 14).

28. Use a transient analysis to verify the design of problem 21.

29. Use a transient analysis to verify the design of problem 22.

30. Use a transient analysis to verify the operation of the circuit shown in Figure 8.6.15 as specified in problem 25. You may wish to do this as two separate simulations, one for each switch position, with the second position using a pre-charged capacitor.

This page titled 6.1.4: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

SECTION OVERVIEW

6.2: Inductors

Learning Objectives

After completing this chapter, you should be able to:

- Describe the theoretical and practical aspects of inductor construction.
- Describe the current-voltage characteristic behavior of inductors.
- Utilize component data sheets to determine operating characteristics of inductors.
- Determine the initial and steady-state equivalents of resistor-inductor networks.
- Determine the initial and steady-state equivalents of resistor-capacitor-inductor networks.
- Determine the transient response of basic RL networks.

6.2.1: Introduction

6.2.2: Inductance and Inductors

6.2.3: Summary

6.2.4: Exercises

This page titled 6.2: Inductors is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

6.2.1: Introduction

This chapter introduces our third passive device, the inductor. Inductors are fundamentally different from both resistors and capacitors in terms of their construction and their operation. Inductors do, however, share certain broad traits with capacitors. First, they are energy storage devices. In the case of the inductor, energy is stored in a magnetic field, similar to the case of the capacitor which utilizes an electric field. Further, in the ideal case, inductors do not dissipate power. Also like capacitors, when inductors are placed in DC circuits they are not ohmic, meaning that their current-voltage characteristic does not respond to Ohm's law. Instead, their currentvoltage characteristic is dynamic in nature. In some respects, though, their current-voltage behavior is opposite to the way in which capacitors behave, and thus they offer their own unique performance characteristics.

Inductors have a long history of use in electronic systems. In fact, one of the most common uses in the home is as an integral part of a typical loudspeaker system. They are used in modern switch-mode DC power supplies in products such as personal computers and televisions. In audio and communications systems they are used in filters and tuning circuits alongside capacitors. Like a capacitor, for any application that needs to smooth out a varying voltage, store energy or filter a signal; an inductor is a likely candidate.

Unfortunately, real-world inductors generally do not behave as close to their desired ideal operation as do realworld capacitors. The secondary effects of inductor construction limit their performance; perhaps the most notable factor being their potentially large equivalent series resistance. They are also susceptible to external magnetic fields which can introduce noise and interference, degrading signal quality. For these reasons, there are areas where, given a choice, capacitors would be preferred over inductors. But this is by no means a broad condemnation and there are areas where the use of inductors is essential. Beyond this, the very concept of inductance is important in that informs designers of the practical limits of performance of their circuits.

The concept of electromagnetic induction was first discovered by English scientist Michael Faraday in the early 19th century. He noticed that if he wrapped two wires around an iron ring and introduced a current in one of them, then a transient (shortlived) current would appear in the second coil of wire. He noticed a similar effect when he slid a bar magnet through a wire coil. Around the same time, American Joseph Henry discovered much the same independently of Faraday and performed considerable research in this area. In his honor, the unit of inductance, the henry (abbreviated H), is named after him. The symbol for inductance is L, named after physicist Heinrich Emil Lenz. Inductors are commonly referred to as coils or chokes, for reasons that will be apparent shortly

This page titled 6.2.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

6.2.2: Inductance and Inductors

To begin, we need to examine the interrelation between electric current and magnetic fields in a conductor. When a current passes through a conductor, such as a wire, a magnetic field is created around the conductor that is proportional to the strength of the current. This is illustrated in Figure 9.2.1 .

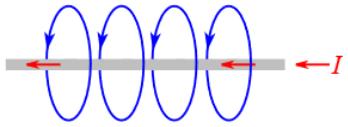


Figure 9.2.1 : Magnetic field around a conductor.

The magnetic field can be thought of as sets of concentric rings around the conductor, although for clarity only single loops are drawn in the Figure. The number of magnetic lines in a given area is known as the magnetic flux and is given the symbol Φ (the Greek letter phi). The unit of magnetic flux is the weber, Wb, named after Wilhelm Weber, a 19th century German physicist.

$$\text{Magnetic flux} \equiv \text{the number of magnetic lines enclosed in a given area.} \tag{6.2.2.1}$$

Note that the magnetic field runs the length of the conductor. The direction of the field lines follows the right hand rule: if you grasp the wire with your right hand such that your thumb is pointing in the direction of conventional current flow, then your fingers wrap in the direction of the magnetic field. This rule is illustrated in Figure 9.2.2 .



Figure 9.2.2 : Right hand rule. Image source (modified)

If we form the conductor into a loop, the field lines are corralled into the center of the loop. This is illustrated in Figure 9.2.3 . In this diagram it can be seen that the lines effectively collect in the center, going into the page.

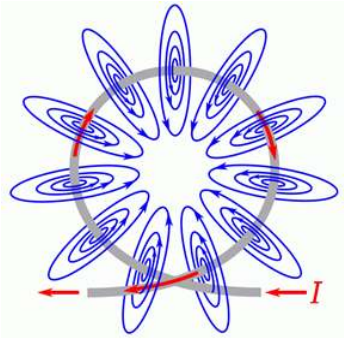


Figure 9.2.3 : Magnetic field around a loop.

The enhancement effect can be magnified by adding more loops in tandem. This is known as a solenoid and is shown in Figure 9.2.4 . It is the most basic form of an inductor.



Figure 9.2.4 : Solenoid.

The concentrating effect of the magnetic field is shown in Figure 9.2.5 . In this figure, the coil is shown from the side, as a cross-section of the individual loops. The dots inside the conductors indicate that the current is flowing toward you, out of the page; while the the crosses indicate that the current is flowing into the page. The lines of flux exit out the right, loop around, and reenter from the left. Due to limited space, the entire loop for each line is not drawn, and it is important to remember that magnetic flux

lines do not end, but always create a loop. Further, although it is shown as a plane here, this field is three dimensional, with lines looping back into the page as well as in front of it.

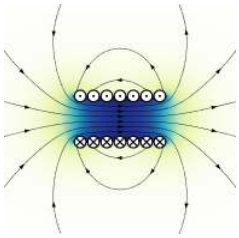


Figure 9.2.5 : Magnetic field in a solenoid. Image source

This is how electromagnets can be created[1]. The north pole is the exiting end (right side) of Figure 9.2.5 while the south pole is the entering end (left side).

If the current changes, there will be a commensurate change in the magnetic field. Further, this change in the field will induce a current in the conductor that creates a magnetic field that opposes the original change in the field. This is known as Lenz's law. Alternately, it can be stated that the induced current caused by a changing magnetic field will oppose the change in the original current that created that change in the original magnetic field.

At this point we can offer a proper definition of the weber:

$$\begin{aligned} &1 \text{ weber } \equiv \text{ the magnetic flux that, acting on a single loop of a conductor,} \\ &\text{produces a potential of 1 volt if the flux is reduced to zero} \\ &\text{at an even rate over 1 second.} \end{aligned} \tag{6.2.2.2}$$

In magnetic circuits we are also interested in the magnetic flux density which is the magnetic flux per unit area. The symbol for flux density is B and has units of teslas (T), named after Nicola Tesla, the Serbo-Croatian-American engineer and inventor. One tesla is defined as one weber per square meter.

$$1 \text{ tesla } \equiv 1 \text{ weber/meter}^2 \tag{6.2.2.3}$$

To provide a reference, the magnetic flux density of the Earth near the equator is approximately 31 μT, while the value of the voice coil gap in a loudspeaker is around 1 to 2 T, with medical MRI scanners being a little higher still.

Finally, we come to the definition of inductance and its unit, the henry:

$$\begin{aligned} &\text{Inductance is a measure of the tendency of a conductor to oppose a change in} \\ &\text{the current flowing through it.} \end{aligned} \tag{6.2.2.4}$$

$$1 \text{ henry } \equiv 1 \text{ weber/1 amp} \tag{6.2.2.5}$$

Unsurprisingly, the energy stored in the magnetic field of an inductor is proportional to the inductance. It is also proportional to the square of the current through the inductor.

$$W = \frac{1}{2} L I^2 \tag{6.2.2.6}$$

Where

W is the energy in joules,

L is the inductance in henries,

I is the current in amps.

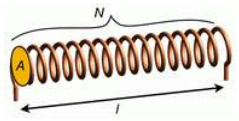


Figure 9.2.6 : Simple air-core inductor dimensions.

An inductor in its simplest form consists of a series of wire loops. These might be wound around an iron core, although a non-ferrous core might also be used. For a simple single layer inductor, such as the one drawn in Figure 9.2.6 , the inductance is described by the following formula:

$$L = \mu \frac{AN^2}{l} \tag{6.2.2.7}$$

Where

L is the inductance in henries,

μ is the permeability of the core material,

A is the cross sectional area of the coil,

N is the number of coils or turns,

l is the length of the coil.

Inductors may also be wound using multiple layers or around a toroidal core, and these designs utilize alternate formulas.

Inductor Styles and Packaging

Equation 6.2.2.7 indicates that, in order to achieve high inductance, we would like a core with high permeability, permeability being a measure of how easy it is to establish magnetic flux in said material. Substances such as iron or ferrite have a much greater permeability than air and are used commonly for cores. They do have a disadvantage in that they will saturate sooner than an air core, and this can lead to distortion.



Figure 9.2.7 : Electric bass guitar pickup.

Another approach is to pack as many turns as is possible within a given length. One way to do this is to minimize the thickness of the insulation around the wire[2]. This can be achieved by using a thin enamel coating instead of the typical plastic insulation. A second method is to use a very fine wire. This leads to two problems, namely an undesirable increase in equivalent series resistance (known colloquially as coil resistance or R_{coil}), and limited current carrying capacity. All of these effects have to be balanced in order to achieve the best performance for a given application.

Commercial inductors range in value from a fraction of a nanohenry for small surface mount “chip” inductors up to several henries. Some devices exhibit large internal inductances even though they are not specifically used as inductors. One common example is a transformer. Another example is an electric guitar or bass pickup, such as the one shown in Figure 9.2.7 with its cover removed. Units such as this may be constructed of several thousand turns of very fine wire (typically AWG 41 to 44) and achieve inductances in excess of one henry.

A variety of inductors is shown in Figure 9.2.9 , all of which are of the through-hole type (surface mount inductors do not appear considerably different from their surface mount resistor and capacitor cousins).

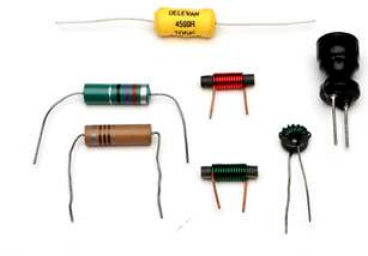


Figure 9.2.9 : A collection of inductors.

The two units toward the left are molded inductors and use a standard color code, similar to the ones used for resistors and capacitors. The unit at the top (yellow) is a high current inductor that features low R_{coil}. The three inductors in the center use obvious ferrite cores, two wound on straight cores and the third wound on a toroidal core. The unit to the right uses a high permeability material at the very top and is wrapped in a plastic sheath for protection. Variable inductors are also a possibility and can be made by using a ferrite core that can be slid within the coils, effectively changing the permeability of the core (part ferrite, part air).



Figure 9.2.10 : Inductor schematic symbols (top-bottom): standard, variable, iron/ferrite core.

The schematic symbols for inductors are shown in Figure 9.2.10 . The standard symbol is at the top. The variable inductor symbol is in the middle and is a twolead device, somewhat reminiscent of the symbol for a rheostat. At the bottom is the symbol for an inductor with an iron, ferrite, or similar high permeability core. In general, like resistors, single inductors are not polarized and cannot be inserted into a circuit backwards. There are, however, special applications where multiple coils can be wound on a common core, and for these, the polarity of their interconnection can make a difference.

Inductor Data Sheet

A portion of an inductor data sheet is shown in Figure 9.2.12 . This page lists the available sizes of this particular model, each with corresponding quantities. We can see this model is available in inductance values ranging from 1 μH up to 100 mH. Tolerance of the smaller values is $\pm$10% while values at and above 33 μH are at $\pm$5%. Q is the quality factor and is particularly important in AC circuits (higher being better), along with its associated frequency, f_Q. Continuing across we find I_R. This is the maximum rated current. For the smaller values, we find they can withstand in excess of 2 amps while the larger units can withstand only tens of milliamps.

TDK

RF chokes B82144A

LBC series, 5.2 x 12 (mm)

Characteristics and ordering codes

L_R µH	Tolerance[1]	Q_{min}	f_Q MHz	I_R mA	R_{max} Ω	$f_{res,min}$ MHz	Ordering code (reel packing)[2]
1.0	±10% ≙ K	40	7.96	2200	0.08	200	B82144A2102K000
1.5		40	7.96	2100	0.09	190	B82144A2152K000
2.2		40	7.96	1900	0.11	140	B82144A2222K000
3.3		40	7.96	1750	0.13	120	B82144A2332K000
4.7		40	7.96	1600	0.16	100	B82144A2472K000
6.8		40	7.96	1500	0.19	80	B82144A2682K000
10		60	2.52	1400	0.22	60	B82144A2103K000
15		60	2.52	1250	0.28	20	B82144A2153K000
22		50	2.52	1100	0.35	12	B82144A2223K000
33	±5% ≙ J	40	2.52	900	0.43	8.0	B82144A2333J000
47		40	2.52	800	0.50	5.0	B82144A2473J000
68		30	2.52	700	0.60	4.5	B82144A2683J000
100		50	0.796	600	0.70	3.5	B82144A2104J000
150		50	0.796	500	0.90	3.0	B82144A2154J000
220		50	0.796	400	1.60	2.4	B82144A2224J000
330		50	0.796	330	1.90	2.0	B82144A2334J000
470		40	0.796	280	2.50	1.5	B82144A2474J000
680		30	0.796	240	2.80	1.3	B82144A2684J000
1000		60	0.252	200	3.80	1.2	B82144A2105J000
1500		60	0.252	160	6.00	1.0	B82144A2155J000
2200		60	0.252	120	9.00	0.8	B82144A2225J000
3300		60	0.252	110	12.0	0.6	B82144A2335J000
4700		60	0.252	90	20.0	0.5	B82144A2475J000
6800		60	0.252	80	30.0	0.4	B82144A2685J000
10000		50	0.0796	60	42.0	0.35	B82144A2106J000
15000		50	0.0796	50	68.0	0.30	B82144A2156J000
22000		50	0.0796	40	120	0.26	B82144A2226J000
33000		50	0.0796	35	150	0.22	B82144A2336J000
47000		40	0.0796	30	230	0.18	B82144A2476J000
68000		40	0.0796	25	290	0.15	B82144A2686J000
100000		30	0.0796	20	420	0.12	B82144A2107J000

Figure 9.2.12 : Inductor data sheet. Courtesy of TDK

Finally, we come to R_{max}. This is also known as R_{coil}. It represents the equivalent series resistance of the inductor. In general, smaller is better. For this model, it ranges from a fraction of an ohm to a few hundred ohms. This trend is typical of inductors; all else being equal, the larger the inductance, the larger the value of the associated series resistance. In many circuits, the value of R_{coil} cannot be ignored.

Inductors in Series and in Parallel

Suppose we take two identical inductors and place them in series. This effectively doubles both the length and the number of loops. From Equation 6.2.2.7 we can see that doubling both the number of loops and the length would double the inductance. This is because N^2 goes up by a factor of four which is then halved be the increased length. Consequently, inductors in series add values just like resistors in series. By extension, inductors in parallel behave like resistors in parallel. The equivalent of parallel inductors can be found by using either the product-sum rule or by taking the reciprocal of the sum of their reciprocals.

Example 9.2.1

Find the equivalent inductance of the network shown in Figure 9.2.11 .

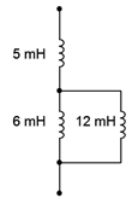


Figure 9.2.11 : Circuit for Example 9.2.1 .

The 6 mH and 12 mH inductors are in parallel. The equivalent value of the pair is:

$$L_{parallel} = \frac{L_2 L_3}{L_2 + L_3}$$

$$L_{parallel} = \frac{6mH12mH}{6mH + 12mH}$$

$$L_{parallel} = 4mH$$

This combination is in series with the 5 mH inductor. Therefore the total equivalent inductance is 4 mH + 5 mH, or 9 mH.

Current-Voltage Relationship

The fundamental current-voltage relationship of the inductor is the mirror image of that of the capacitor:

$$v = L\frac{di}{dt} \tag{6.2.2.8}$$

This states that the voltage across the inductor is a function of how quickly the current is changing. If the current is not changing (i.e., in steady-state), then the voltage across the inductor is zero. In this case, the inductor behaves like a short, or more accurately, like its R_{coil} value. In contrast, during a rapid initial current change, the inductor voltage can be large, and thus the inductor behaves like an open.

If we rearrange Equation 6.2.2.8 and solve for the rate of change of current, we find that:

$$\frac{di}{dt} = \frac{v}{L} \tag{6.2.2.9}$$

Thus if an inductor is fed by a constant voltage source, the current will rise at a constant rate equal to v/L. For example, considering the circuit in Figure 9.2.11 , we see a voltage source feeding a single inductor. If we were to plot the inductor's current over time, we would see something like the graph of Figure 9.2.12 .

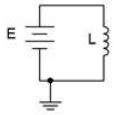


Figure 9.2.11 : Inductor with voltage source.

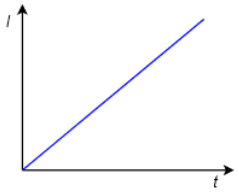


Figure 9.2.12 : Inductor current versus time.

As time progresses, the current through the inductor increases, flowing from top to bottom. With a theoretically perfect inductor and source, this would continue as long as the circuit was energized. In reality, this line would either begin to deflect horizontally as the source reached its limits, or the inductor would fail once its maximum current or power handling was reached. The slope of this line is dictated by the size of the applied voltage source and the inductance.

Example 9.2.2

Determine the rate of change of current through the inductor in the circuit of Figure 9.2.13 . Also determine the inductor's current 10 microseconds after power is switched on.

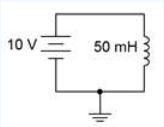


Figure 9.2.13 : Circuit for Example 9.2.2 .

From Equation 6.2.2.9, the rate of change of current is:

$$\frac{di}{dt} = \frac{v}{L}$$

$$\frac{di}{dt} = \frac{10V}{50mH}$$

$$\frac{di}{dt} = 200A \text{ per second}$$

In other words, for every second, the current rises another 200 amps. Thus, after just 10 microseconds it will have risen to 200 A/s times 10 μs, or 2 mA.

Equation 6.2.2.8 is key to understanding the behavior of inductors. As noted previously, if an inductor is driven by a fixed voltage source and ignoring R_{coil}, the current through it rises at the constant rate of v/L. This change in current through the inductor is not limitless. An instantaneous change requires that di/dt is infinite, and thus, the voltage driving the inductor would also have to be infinite, which is a clear impossibility. Therefore we can state a particularly important characteristic of capacitors:

$$\text{The current through an inductor cannot change instantaneously.} \tag{6.2.2.10}$$

This observation will be central to analyzing the operation of inductors in DC circuits.

References

[1]Truly one of the coolest inventions of all time: a magnet with an on/off switch.

[2]The wound wire must be insulated otherwise each loop will short to the loops next to it and we'd be left with a tube instead of a series of loops.

This page titled 6.2.2: Inductance and Inductors is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.2: Inductance and Inductors** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

6.2.3: Summary

The capacitor is a device that is used to store electric charge. Capacitance, C, is measured in farads, F. The idealized device consists of two conductive plates separated by some distance, that space being filled by an insulating dielectric. Capacitance is directly proportional to the plate area and the dielectric's permittivity, and inversely proportional to the plate distance. Another important characteristic of the dielectric is its breakdown strength. Along with the plate spacing, this will establish the capacitor's voltage rating. The permittivity will also help to determine the capacitor's volumetric efficiency, a measure of how much capacitance can be achieved within a given volume. Non-ideal parameters include the ESR, or equivalent series resistance, which is ideally zero; and the effective parallel leakage resistance, ideally infinity. Absolute accuracy, temperature stability and similar parameters round out the distinguishing features of one kind of capacitor against another. When placed in parallel, capacitors add in the same manner as resistors in series. When placed in series, capacitors behave like resistors in parallel.

Perhaps the most important operational characteristic regarding capacitors is that voltage across a capacitor cannot change instantaneously. It will take some finite amount of time before the charge on the capacitor builds, leading to a predictable rise in voltage across it. Because of this, for DC circuits capacitors initially behave like shorts, but after sufficient time has passed, they behave like opens. The amount of time required to reach steady-state is five time constants, where one time constant is defined as the product of the circuit's effective resistance and its capacitance. The current charge curve is of the shape ε^{-t}. The current starts at a maximum and eventually approaches zero as time passes. The corresponding voltage shape is of the form $1-\varepsilon^{-t}$. Here, the capacitor's voltage starts at zero and rises to some maximum value. The capacitor's voltage discharge curve effectively is swapped compared to the charge curve (e.g. voltage follows ε^{-t}).

Review Questions

1. What are the physical characteristics of capacitors and how do they affect capacitance?
2. Define the voltage-current characteristic for capacitors.
3. What is meant by a capacitor's volumetric efficiency?
4. How do the permittivity and breakdown strength of the dielectric affect the overall capacitance and voltage rating?
5. How do capacitors combine when placed in series and how do they combine when placed in parallel?
6. Define the initial and steady-state behavior of capacitors.
7. Define time constant for an RC circuit.
8. Describe the charge and discharge characteristics of RC circuits.

This page titled 6.2.3: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

6.2.4: Exercises

Analysis

1. For the circuit shown in Figure 8.6.1 , determine the effective capacitance.

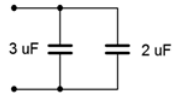


Figure 8.6.1

2. Determine the effective capacitance of the configuration shown in Figure 8.6.2 .

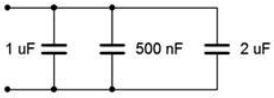


Figure 8.6.2

3. Given the capacitor network shown in Figure 8.6.3 , determine the effective value.



Figure 8.6.3

4. Determine the effective capacitance of network shown in Figure 8.6.4 .



Figure 8.6.4

5. Determine the voltage across each capacitor for the circuit shown in Figure 8.6.5 .

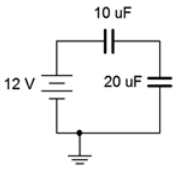


Figure 8.6.5

6. Determine the voltage across each capacitor for the circuit shown in Figure 8.6.6 .

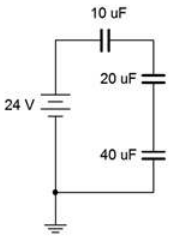


Figure 8.6.6

7. Determine the initial voltage across each component for the circuit shown in Figure 8.6.7 .

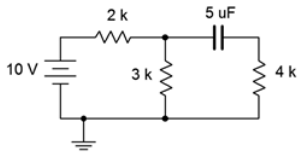


Figure 8.6.7

8. Given the network shown in Figure 8.6.7 , determine the steady-state voltage across each component.

9. For the circuit shown in Figure 8.6.8 , determine the capacitor voltage 3 microseconds after the power is switched on.

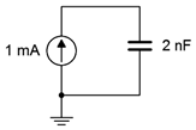


Figure 8.6.8

10. For the circuit shown in Figure 8.6.9 , determine the capacitor voltage 5 seconds after the power is switched on.

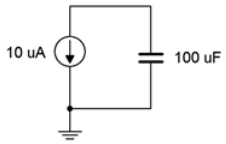


Figure 8.6.9

11. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.10 .

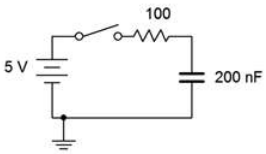


Figure 8.6.10

12. For the circuit shown in Figure 8.6.10 , determine the capacitor voltage and circulating current 20 microseconds and 100 milliseconds after the switch is thrown.

13. Given the circuit shown in Figure 8.6.11 , determine the capacitor voltage and circulating current 200 milliseconds and 10 seconds after the switch is thrown.

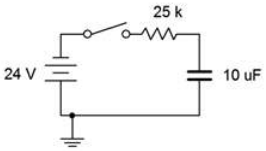


Figure 8.6.11

14. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.11 .

15. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.12 , switch position 1.

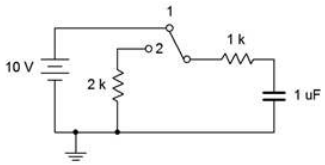


Figure 8.6.12

16. Determine the charge and discharge time constants for the circuit shown in Figure 8.6.13 .

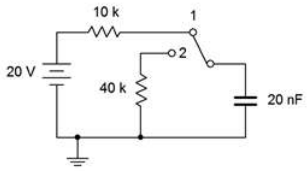


Figure 8.6.13

17. Given the circuit shown in Figure 8.6.12 , determine the capacitor voltage 10 milliseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

18. For the circuit shown in Figure 8.6.13 , determine the capacitor voltage 400 microseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

19. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.14 , switch position 1.

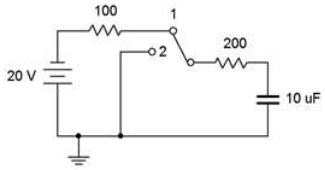


Figure 8.6.14

20. Given the circuit shown in Figure 8.6.14 , determine the capacitor voltage 12 milliseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

Design

21. Given the circuit of Figure 8.6.10 , determine a new resistor value such that steady-state is reached in 2 milliseconds.

22. Given the circuit of Figure 8.6.11 , determine a new resistor value such that steady-state is reached in 5 seconds.

23. When an audio amplifier is turned on, the power surge can cause an audible pop from the loudspeaker. To prevent this, amplifiers often connect to the loudspeaker via a relay. The relay is energized to connect the loudspeaker once the output has settled, typically a few seconds after power is applied. This delay may be created via an RC network. Suppose the driving circuit is 5 volts and the relay trips at 4 volts. Further, the associated charging resistance is 10 kΩ. Determine the capacitance required to achieve a 2 second delay time.

Challenge

24. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 8.6.15 .

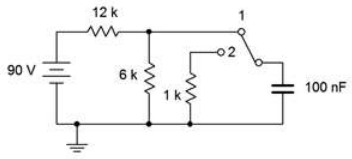


Figure 8.6.15

25. For the circuit shown in Figure 8.6.15 , determine the capacitor voltage 1 second after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the capacitor to discharge to nearly zero volts.

Simulation

26. Perform a transient analysis to verify the time to steady-state of Figure 8.6.10 (problem 11).

27. Perform a transient analysis to verify the time to steady-state of Figure 8.6.11 (problem 14).

28. Use a transient analysis to verify the design of problem 21.

29. Use a transient analysis to verify the design of problem 22.

30. Use a transient analysis to verify the operation of the circuit shown in Figure 8.6.15 as specified in problem 25. You may wish to do this as two separate simulations, one for each switch position, with the second position using a pre-charged capacitor.

This page titled 6.2.4: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

7: Advanced Topic- Operational Amplifiers

Learning Objectives

After completing this chapter, you should be able to:

- Convert between ordinary and decibel based power and voltage gains.
- Utilize decibel-based voltage and power measurements during circuit analysis.
- Define and graph a general Bode plot.
- Detail the differences between lead and lag networks, and graph Bode plots for each.
- Combine the effects of several lead and lag networks together in order to determine a system Bode plot.
- Describe the use of digital computers in the area of circuit simulation.
- Analyze differential amplifiers for a variety of AC characteristics, including single ended and differential voltage gains.
- Define common-mode gain and common-mode rejection.
- Describe a current mirror and note typical uses for it.

7.1: Introduction
7.2: The Decibel
7.3: Bode Plots
7.4: Combining the Elements - Multi-Stage Effects
7.5: The Differential Amplifier
7.6: Summary
7.7: Problems
7.8: Operational Amplifier Internals
7.8.1: Introduction to Operational Amplifiers
7.8.2: What Is An Op Amp?
7.8.3: Simple Op Amp Comparator
7.8.4: Op Amp Manufacture
7.8.5: Summary
7.8.6: Problems
7.9: Negative Feedback
7.9.1: Introduction to Negative Feedback
7.9.2: What Negative Feedback Is and Why We Use It
7.9.3: Basic Concepts
7.9.4: The Four Variants of Negative Feedback
7.9.5: Limitations On The Use of Negative Feedback
7.9.6: Summary
7.9.7: Problems
7.10: Basic Op Amp Circuits
7.10.1: Introduction
7.10.2: Inverting and Noninverting Amplifiers
7.10.3: Single Supply Biasing
7.10.4: Current Boosting
7.10.5: Summary

7.10.6: Problems
7.11: Integrators and Differentiators
7.11.1: Introduction
7.11.2: Integrators
7.11.3: Differentiators
7.11.4: Analog Computer
7.11.5: Alternatives to Integrators and Differentiators
7.11.6: Extended Topic- Other Integrator and Differentiator Circuits
7.11.7: Summary
7.11.8: Problems

Template:HideTOC

This page titled 7: Advanced Topic- Operational Amplifiers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.1: Introduction

Before we can begin our study of the operational amplifier, it is very important that certain background elements be in place. The purpose of this chapter is to present the very useful analysis concepts and tools associated with the decibel measurement scheme and the frequency domain. We will also be examining the differential amplifier that serves as the heart of most operational amplifiers. With a thorough working knowledge of these items, you will find that circuit design and analysis will proceed at a much quicker and more efficient pace. Consider this chapter as an investment in time, and treat it appropriately.

The decibel measurement scheme is in very wide use, particularly in the field of communications. We will be examining its advantages over the ordinary system of measurement, and how to convert values of one form into the other. One of the more important parameters of a circuit is its frequency response. To this end, we will be looking at the general frequency domain representations of a circuit's gain and phase. This will include both manual and computer generated analysis and graphing techniques. While our main emphasis will eventually concentrate on application with operational amplifiers, the techniques explored can be applied equally to discrete circuits. Indeed, our initial examples will use simple discrete or black box circuits exclusively.

Finally, we will examine the DC and AC operation of the differential amplifier. This is an amplifier that utilizes two active devices and offers dual inputs. It offers certain features that make it suitable as the first section of most operational amplifiers.

1.1.1: Variable Naming Convention

One item that often confuses students of almost any subject is nomenclature. It is important, then, that we decide upon a consistent naming convention at the outset. Throughout this text, we will be examining numerous circuits containing several passive and active components. We will be interested in a variety of parameters and signals. Although we will utilize the standard conventions, such as f_c for critical frequency and X_c for capacitive reactance, a great number of other possibilities exist. In order to keep confusion to a minimum, we will use the following conventions in our equations for naming devices and signals that haven't been standardized.

R	Resistor (DC, or actual circuit component)
r	Resistor (AC equivalent, where phase is 0 or ignored)
C	Capacitor
L	Inductor
Q	Transistor (Bipolar or FET)
D	Diode
V	Voltage (DC)
v	Voltage (AC)
I	Current (DC)
i	Current (AC)

Resistors, capacitors and inductors are differentiated via a subscript that usually refers to the active device it is connected to. For example, R_E is a DC bias resistor connected to the emitter of a transistor, while r_C refers to the AC equivalent resistance seen at a transistor's collector. C_E refers to a capacitor connected to a transistor's emitter lead (most likely a bypass or coupling capacitor). Note that the device related subscripts are always shown in upper case, with one exception: If the resistance or capacitance is part of the device model, the subscript will be shown in lower case to distinguish it from the external circuit components. For example, the AC dynamic resistance of a diode would be called r_d. If no active devices are present, or if several items exist in the circuit, a simple numbering scheme is used, such as R_1. In very complex circuits, a specific name will be given to particularly important components, as in R_{source}.

Voltages are normally given a two-letter subscript indicating the nodes at which it is measured. V_{CE} is the DC potential from the collector to the emitter of a transistor, while v_{BE} indicates the AC signal appearing across a transistor's base-emitter junction. A single-letter subscript, as in V_B, indicates a potential relative to ground (in this case, base to ground potential). The exceptions to this rule are power supplies, that are given a double letter subscript indicating the connection point (V_{CC} is the collector power supply), and particularly important potentials that are directly named, as in v_{in} (AC input voltage) and V_{R1} (DC voltage appearing across R_1). If an Equation for a specific potential is valid for both the AC and DC equivalent circuits, the uppercase form is preferred (this makes things much more consistent with the vast majority of op amp circuits that are directly coupled, and thus can amplify both AC and DC signals). Currents are named in a similar way, but generally use a single subscript referring to the measurement node (I_C is the DC collector current). All other items are directly named. By using this scheme, you will always be able to determine whether the item expressed in an Equation is a DC or AC equivalent, its approximate circuit location, and other factors about it.

This page titled 7.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.2: The Decibel

Most people are familiar with the term “decibel” in reference to sound pressure. It’s not uncommon to hear someone say something such as “It was 110 decibels at the concert last night, and my ears are still ringing.” This popular use is somewhat inaccurate, but does show that decibels indicate some sort of quantity - in this case, sound pressure level.

1.2.1: Decibel Representation of Power and Voltage Gains

In its simplest form, the decibel is used to measure some sort of gain, such as power or voltage gain. Unlike the ordinary gain measurements that you may be familiar with, the decibel form is logarithmic. Because of this, it can be very useful for showing ratios of change, as well as absolute change. The base unit is the Bel. To convert an ordinary gain to its Bel counterpart, just take the common log (base 10) of the gain. In Equation form:

$$\text{Bel gain } = \log_{10}(ordinary\ gain)$$

Note that on most hand calculators common log is denoted as “$\log$” while the natural log is given as “$\ln$”. Unfortunately, some programming languages use “$\log$” to indicate natural log and “$\log 10$” for common log. More than one student has been bitten by this bug, so be forewarned! As an example, if a circuit produces an output power of 200 milliwatts for an input of 10 milliwatts, we would normally say that it has a power gain of:

$$G = \frac{P_{out}}{P_{in}}$$
$$G = \frac{200\ mW}{10\ mW}$$
$$G = 20$$

For the Bel version, just take the log of this result.

$$G^{'} = \log_{10}\ G$$
$$G^{'} = \log_{10}\ 20$$
$$G^{'} = 1.301$$

The Bel gain is 1.3 Bels. The term “Bels” is not a unit in the strict sense of the word (as in “watts”), but is simply used to indicate that this is not an ordinary gain. In contrast, ordinary power and voltage gains are sometimes given units of W/W and V/V to distinguish them from Bel gains. Also, note that the symbol for Bel power gain is G' and not G. All Bel gains are denoted with the following prime (’) notation to avoid confusion. Because Bels tend to be rather large, we typically use one-tenth of a Bel as the norm. The result is the decibel (one-tenth Bel). To convert to decibels, simply multiply the number of Bels by 10. Our gain of 1.3 Bels is equivalent to 13 decibels. The units are commonly shortened to dB. Consequently, we may say:

$$G^{'} = 10 \log_{10} G \tag{7.2.1}$$

Where the result is in dB.

At this point, you may be wondering what the big advantage of the decibel system is. To answer this, recall a few log identities. Normal multiplication becomes addition in the log system, and division becomes subtraction. Likewise, powers and roots become multiplication and division. Because of this, two important things show up. First, ratios of change become constant offsets in the decibel system, and second, the entire range of values diminishes in size. The result is that a very wide range of gains may be

represented within a fairly small scope of values, and the corresponding calculations can become quicker. There are a couple of dB values that are useful to remember. With the aid of your hand calculator, it is very easy to show the following:

Factor	dB Value using $G^{'} = 10 \log_{10} G$
1	0 dB
2	3.01 dB
4	6.02 dB
8	9.03 dB
10	10 dB

We can also look at fractional factors (i.e., losses instead of gains.)

Factor	dB Value
0.5	−3.01 dB
0.25	−6.02 dB
0.125	−9.03 dB
0.1	−10 dB

If you look carefully, you will notice that a doubling is represented by an increase of approximately 3 dB. A factor of 4 is in essence, two doublings. Therefore, it is equivalent to 3 dB + 3 dB, or 6 dB. Remember, because we are using logs, multiplication turns into simple addition. In a similar manner, a halving is represented by approximately -3 dB. The negative sign indicates a reduction. To simplify things a bit, think of factors of 2 as $\pm$3 dB, the sign indicating whether you are increasing (multiplying), or decreasing (dividing). As you can see, factors of 10 work out to a very convenient 10 dB. By remembering these two factors, you can often estimate a dB conversion without the use of your calculator. For instance, we could rework our initial conversion problem as follows:

- The amplifier has a gain of 20
- 20 can be written as 2 times 10
- The factor of 2 is 3 dB, the factor of 10 is 10 dB
- The answer must be 3 dB + 10 dB, or 13 dB
- This verifies our earlier result

Time for a few examples.

Example 7.2.1

An amplifier has a power gain of 800. What is the decibel power gain?

$$G^{'} = 10 \ \log_{10} \ G$$
$$G^{'} = 10 \ \log_{10} \ 800$$
$$G^{'} = 10 \times 2.903$$
$$G^{'} = 29.03 \ dB$$

We could also use our estimation technique:

- G = 800 = $8 \cdot 10^2$
- 8 is equivalent to 3 factors of 2, or $2 \cdot 2 \cdot 2$, and can be expressed as 3 dB + 3 dB + 3 dB, which is, of course, 9 dB
- 10^2 is equivalent to 2 factors of 10, or 10 dB + 10 dB = 20 dB
- The result is 9 dB + 20 dB, or 29 dB

Note that if the leading digit is not a power of 2, the estimation will not be as precise. For example, if the gain is 850, you know that the decibel gain is just a bit over 29 dB. You also know that it must be less than 30 dB (1000 = 10^3 which is 3 factors of 10, or 30 dB.) As you can see, by using the dB form, you tend to concentrate on the magnitude of gain, and not so much on trailing digits.

Example 7.2.2

An attenuator reduces signal power by a factor of 10,000. What is this loss expressed in dB?

$$G' = 10\ log_{10}\ \frac{1}{10000}$$
$$G' = 10 \times (-4)$$
$$G' = -40\ dB$$

By using the approximation, we can say,

$$\frac{1}{10000} = 10^{-4}$$

The negative exponent tells us we have a loss (negative dB value), and 4 factors of 10.

$$G' = -10\ dB - 10\ dB - 10\ dB - 10\ dB$$
$$G' = -40\ dB$$

Remember, if an increase in signal is produced, the result will be a positive dB value. A decrease in signal will always result in a negative dB value. A signal that is unchanged indicates a gain of unity, or 0 dB.

To convert from dB to ordinary form, just invert the steps; that is, divide by ten and then take the antilog.

$$G = \log_{10}^{-1}\ \frac{G'}{10}$$

On most hand calculators, base 10 antilog is denoted as 10^x. In most computer languages, you just raise 10 to the appropriate power, as in G = 10.0^(Gprime / 10.0) (BASIC), or use an exponent function, as in pow(10.0, Gprime / 10.0) (C or Python).

Example 7.2.3

An amplifier has a power gain of 23 dB. If the input is 1 mW, what is the output?

In order find the output power, we need to find the ordinary power gain, G.

$$G = \log_{10}^{-1}\ \frac{G'}{10}$$
$$G = \log_{10}^{-1}\ \frac{23}{10}$$
$$G = 199.5$$

Therefore, $P_{out} = 199.5 \cdot 1$ mW, or 199.5 mW

You could also use the approximation technique in reverse. To do this, break up the dB gain in 10 dB and 3 dB chunks:

$$23\ dB = 3\ dB + 10\ dB + 10\ dB$$

Now replace each chunk with the appropriate factor, and multiply them together (remember, when going from log to ordinary form, addition turns into multiplication.)

$$3dB = 2X\ 10\ dB = 10X,\ \text{so,}$$
$$G = 2 \times 10 \times 10$$
$$G = 200$$

While the approximation technique appears to be slower than the calculator, practice will show otherwise. Being able to quickly estimate dB values can prove to be a very handy skill in the electronics field. This is particularly true in larger, multistage designs.

Example 7.2.4

A three-stage amplifier has gains of 10 dB, 16 dB, and 14 dB per section. What is the total dB gain?

Because dB gains are a log form, just add the individual stage gains to arrive at the system gain.

$$G_{total}^{'} = G_1^{'} + G_2^{'} + G_3^{'}$$
$$G_{total}^{'} = 10\ dB + 16\ dB + 14\ dB$$
$$G_{total}^{'} = 40\ dB$$

As you may have noticed, all of the examples up to this point have used power gain and not voltage gain. You may be tempted to use the same equations for voltage gain. In a word, DON'T. If you think back for a moment, you will recall that power varies as the square of voltage. In other words, a doubling of voltage will produce a quadrupling of power. If you were to use the same dB conversions, a doubling of voltage would be 3 dB, yet, because the power has quadrupled, this would indicate a 6 dB rise. Consequently, voltage gain (and current gain as well) are treated in a slightly different fashion. We would rather have our doubling of voltage work out to 6 dB, so that it matches the power calculation. The correction factor is very simple. Because power varies as the second power of voltage, the dB form should be twice as large for voltage (remember, exponentiation turns into multiplication when using logs). Applying this factor to Equation 7.2.1 yields:

$$A_v^{'} = 20\ \log_{10}\ A_v \tag{7.2.2}$$

Be careful though, the Bel voltage gain only equals the Bel power gain if the input and output impedances of the system are matched (you may recall from your earlier work that it is quite possible to design a circuit with vastly different voltage and power gains. A voltage follower, for example, exhibits moderate power gain with a voltage gain of unity. It is quite likely that the follower will not exhibit matched impedances.) If we were to recalculate our earlier table of common factors, we would find that a doubling of voltage gain is equivalent to a 6 dB rise, and a ten fold increase is equivalent to a 20 dB rise, twice the size of their power gain counterparts.

Note that current gain may be treated in the same manner as voltage gain (although this is less commonly done in practice).

Example 7.2.5

An amplifier has an output signal of 2 V for an input of 50 mV. What is $A_v^{'}$? First find the ordinary gain.

$$A_v = \frac{2}{0.05} = 40$$

Now convert to dB form.

$$A_v^{'} = 20\ \log_{10}\ 40$$
$$A_v^{'} = 20 \times 1.602$$
$$A_v^{'} = 32.04\ dB$$

The approximation technique yields $40 = 2 \cdot 2 \cdot 10$, or 6 dB + 6 dB + 20 dB = 32 dB

To convert $A^{'}v$ to A, reverse the process.

$$A_v = \log_{10}^{-1} \frac{A_v^{'}}{20}$$

Example 7.2.6

An amplifier has a gain of 26 dB. If the input signal is 10 mV, what is the output?

$$A_v = \log_{10}^{-1} \frac{A_v^{'}}{20}$$
$$A_v = \log_{10}^{-1} \frac{26}{20}$$
$$A_v = 19.95$$
$$V_{out} = A_v \ V_{in}$$
$$V_{out} = 19.95 \times 10 \ mV$$

The final point to note in this section is that, as in the case of power gain, a negative dB value indicates a loss. Therefore, a 2:1 voltage divider would have a gain of -6 dB.

1.2.2: Signal Representation in dBW and dBV

As you can see from the preceding section, it is possible to spend considerable time converting between decibel gains and ordinary voltages and powers. Because the decibel form does offer advantages for gain measurement, it would make sense to use a decibel form for power and voltage levels as well. This is a relatively straightforward process. There is no reason why we can't express a power or voltage in a logarithmic form. Because a dB value just indicates a ratio, all we need to do is decide on a reference (i.e., a comparative base for the ratio). For power measurements, a likely choice would be 1 watt. In other words, we can describe a power as being a certain number of dB above or below 1 watt. Positive values will indicate powers greater than 1 watt, while negative values will indicate powers less than 1 watt. In general Equation form:

$$P^{'} = 10 \ \log_{10} \frac{P}{reference} \tag{7.2.3}$$

The answer will have units of dBW, that is, decibels relative to 1 watt.

Example 7.2.7

A power amplifier has a maximum output of 120 W. What is this power in dBW?

$$P^{'} = 10 \ \log_{10} \frac{P}{1 \ Watt}$$
$$P^{'} = 10 \ \log_{10} \frac{120 \ W}{1 \ W}$$
$$P^{'} = 20.8 \ dBW$$

There is nothing sacred about the 1 watt reference, short of its convenience. We could just as easily choose a different reference. Other common reference points are 1 milliwatt (dBm) and 1 femtowatt (dBf). Obviously, dBf is used for very low signal levels, such as those coming from an antenna. dBm is in very wide use in the communications industry. To use these other references, just divide the given power by the new reference.

Example 7.2.8

A small personal audio tape player delivers 200 mW to its headphones. What is this output power in dBW, and in dBm?

For an answer in units of dBW, use the 1 watt reference

$$P^{'} = 10 \ \log_{10} \frac{P}{1 \ Watt}$$
$$P^{'} = 10 \ \log_{10} \frac{200 \ mW}{1 \ W}$$
$$P^{'} = -7 \ dBW$$

For units of dBm, use a 1 milliwatt reference

$$P^{'} = 10 \ \log_{10} \frac{P}{1 \ Watt}$$
$$P^{'} = 10 \ \log_{10} \frac{200 \ mW}{1 \ mW}$$
$$P^{'} = 23 \ dBW$$

200 mW, -7 dBW, and 23 dBm are three ways of saying the same thing. Note that the dBW and dBm values are 30 dB apart. This will always be true, because the references are a factor of 1000 (30 dB) apart.

In order to transfer a dBW or similar value into watts, reverse the process.

$$P = \log_{10}^{-1} \frac{P'}{10} \times reference$$

Example 7.2.9

A studio microphone produces a 12 dBm signal while recording normal speech. What is the output power in watts?

$$P' = \log_{10}^{-1} \frac{P'}{10} \times \ reference$$
$$P' = \log_{10}^{-1} \frac{12\ dBm}{10} \times \ 1\ mW$$
$$P = 15.8\ mW = 0.0158\ W$$

For voltages, we can use a similar system. A logical reference is 1 V, with the resulting units being dBV. As before, these voltage measurements will use a multiplier of 20 instead of 10.

$$V' = 20\ \log_{10} \frac{V}{reference} \tag{7.2.4}$$

Example 7.2.10

A test oscillator produces a 2 V signal. What is this value in dBV?

$$V' = 20\ \log_{10} \frac{V}{reference}$$
$$V' = 20\ \log_{10} \frac{2\ V}{1\ V}$$
$$V' = 6.02\ dB$$

When both circuit gains and signal levels are specified in dB form, analysis can be very quick. Given an input level, simply add the gain to it in order to find the output level. Given input and output levels, subtract them in order to find the gain.

Example 7.2.11

A computer disk read/write amplifier exhibits a gain of 35 dB. If the input signal is -42 dBV, what is the output signal?

$$V'_{out} = V'_{in} + A'_v$$
$$V'_{out} = -42\ dBV + 35\ dB$$
$$V'_{out} = -7\ dBV$$

Note that the final units are dBV and not dB, thus indicating a voltage and not merely a gain.

Example 7.2.12

A guitar power amp needs an input of 20 dBm to achieve an output of 25 dBW. What is the gain of the amplifier in dB?

First, it is necessary to convert the power readings so that they share the same reference unit. Because dBm represents a reference 30 dB smaller than the dBW reference, just subtract 30 dB to compensate.

$$20\ dBm = -10\ dBW$$

$$G' = P'_{out} - P'_{in}$$
$$G' = 25\ dBW - (-10\ dBW)$$
$$G' = 35\ dB$$

> Note that the units are dB and not dBW. This is very important! Saying that the gain is “so many” dBW is the same as saying the gain is “so many” watts. Obviously, gains are “pure” numbers and do not carry units such as watts or volts.

The usage of a dB-based system is shown graphically in Figure 7.2.1. Note how the stage gains are added to the input signal to form the output. Even large circuits can be quickly analyzed in this form. To make life in the lab even easier, it is possible to take measurements directly in dB form. By doing this, you need never convert while troubleshooting a design. For general-purpose work, voltage measurements are the norm, and therefore a dBV scale is often used.

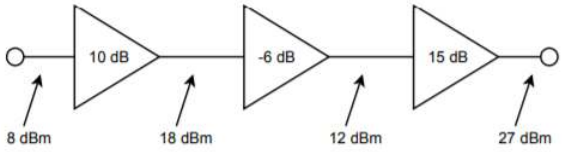


Figure 7.2.1: Multistage dB application.

1.2.3: Items Of Interest In The Laboratory

When using a digital meter on a dBV scale it is possible to “underflow” the meter if the signal is too weak. This will happen if you try to measure around zero volts, for example. If you attempt to calculate the corresponding dBV value, your calculator will probably show “error”. The effective value is negative infinite dBV. The meter will certainly have a hard time showing this value! Another item of interest revolves around the use of dBm measurements. It is common to use a voltmeter to make dBm measurements, in lieu of a wattmeter. While the connections are considerably simpler, a voltmeter cannot measure power. How is this accomplished then? Well, as long as the circuit impedance is known, power can be derived from a voltage measurement. A common impedance in communication systems (such as recording studios) is 600 Ω, so a meter can be calibrated to give correct dBm readings by using Power Law. If this meter is used on a non-600 Ω circuit, the readings will no longer reflect accurate dBm values (but will still properly reflect relative changes in dB).

This page titled 7.2: The Decibel is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.2: The Decibel** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.3: Bode Plots

The Bode plot is a graphical response prediction technique that is useful for both circuit design and analysis. It is named after Hendrik Wade Bode, an American engineer known for his work in control systems theory and telecommunications. A Bode plot is, in actuality, a pair of plots: One graphs the gain of a system versus frequency, while the other details the circuit phase versus frequency. Both of these items are very important in the design of well-behaved, optimal operational amplifier circuits.

Generally, Bode plots are drawn with logarithmic frequency axes, a decibel gain axis, and a phase axis in degrees. First, let's take a look at the gain plot. A typical gain plot is shown Figure 7.3.1.

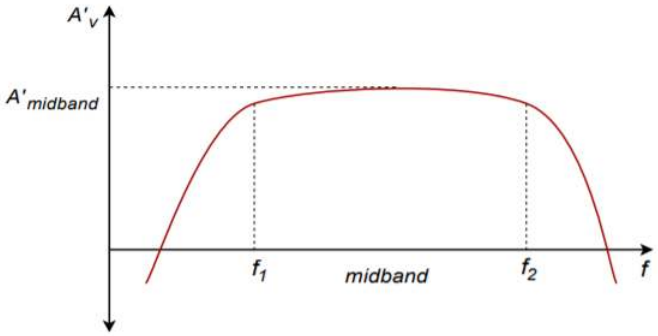


Figure 7.3.1: Gain plot.

Note how the plot is relatively flat in the middle, or midband, region. The gain value in this region is known as the midband gain. At either extreme of the midband region, the gain begins to decrease. The gain plot shows two important frequencies, f_1 and f_2. f_1 is the lower break frequency while f_2 is the upper break frequency. The gain at the break frequencies is 3 dB less than the midband gain. These frequencies are also known as the half-power points, or corner frequencies. Normally, amplifiers are only used for signals between f_1 and f_2. The exact shape of the rolloff regions will depend on the design of the circuit. It is possible to design amplifiers with no lower break frequency (i.e., a DC amplifier), however, all amplifiers will exhibit an upper break. The break points are caused by the presence of circuit reactances, typically coupling and stray capacitances. The gain plot is a summation of the midband response with the upper and lower frequency limiting networks. Let's take a look at the lower break, f_1.

1.3.1: Lead Network Gain Response

Reduction in low frequency gain is caused by lead networks. A generic lead network is shown in Figure 7.3.2. It gets its name from the fact that the output voltage developed across R leads the input. At very high frequencies the circuit will be essentially resistive. Conceptually, think of this as a simple voltage divider. The divider ratio depends on the reactance of C. As the input frequency drops, X_c increases. This makes V_{out} decrease. At very high frequencies, where $X_c \ll R$, V_{out} is approximately equal to V_{in}. This can be seen graphically in Figure 7.3.3. The break frequency (i.e., the frequency at which the signal has decreased by 3 dB) is found via the standard equation

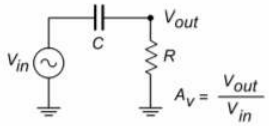


Figure 7.3.2: Lead network.

$$f_c = \frac{1}{2\pi RC}$$

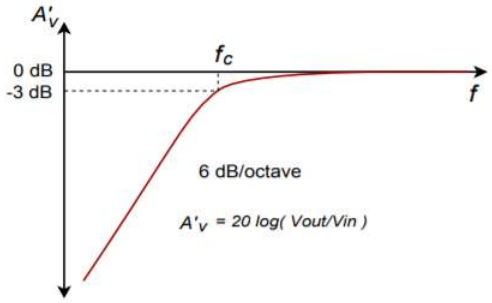


Figure 7.3.3: Lead gain plot (exact).

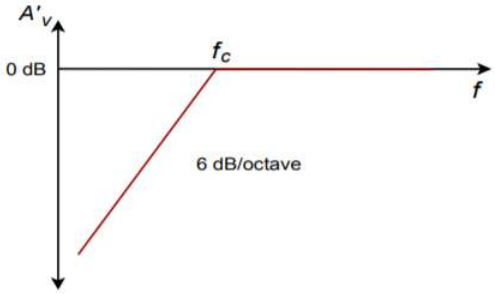


Figure 7.3.4: Lead gain plot (approximate).

The response below f_c will be a straight line if a decibel gain axis and a logarithmic frequency axis are used. This makes for very quick and convenient sketching of circuit response. The slope of this line is 6 dB per octave (an octave is a doubling or halving of frequency, e.g., 800 Hz is 3 octaves above 100 Hz).[1] This range covers a factor of two in frequency. This slope may also be expressed as 20 dB per decade, where a decade is a factor of 10 in frequency. With reasonable accuracy, this curve may be approximated as two line segments, called asymptotes, as shown in Figure 7.3.4. The shape of this curve is the same for any lead network. Because of this, it is very easy to find the approximate gain at any given frequency as long as f_c is known. It is not necessary to go through reactance and phasor calculations. To create a general response equation, start with the voltage divider rule to find the gain:

$$\frac{V_{out}}{V_{in}} = \frac{R}{R - j\ X_c}$$

$$\frac{V_{out}}{V_{in}} = \frac{R\angle 0}{\sqrt{R^2 + X_c^2}\angle - \arctan\frac{X_c}{R}}$$

The magnitude of this is,

$$|A_v| = \frac{R}{\sqrt{R^2 + X_c^2}} \tag{7.3.1}$$

$$|A_v| = \frac{1}{\sqrt{1 + \frac{X_c^2}{R^2}}}$$

Recalling that,

$$f_c = \frac{1}{2\pi R\ C}$$

we may say,

$$R = \frac{1}{2\pi f_c\ C}$$

For any frequency of interest, f,

$$X_c = \frac{1}{2\pi f_c\ C}$$

Equating the two preceding equations yields,

$$\frac{f_c}{f} = \frac{X_c}{R} \tag{7.3.2}$$

Substituting Equation 7.3.2 in Equation 7.3.1 gives,

$$A_v = \frac{1}{\sqrt{1+\frac{f_c^2}{f^2}}} \tag{7.3.3}$$

To express A_v in dB, substitute Equation 7.3.3 into Equation 1.2.2

$$A_v^{'} = 20\log_{10}\frac{1}{\sqrt{1+\frac{f_c^2}{f^2}}}$$

After simplification, the final result is:

$$A_v^{'} = -10\log_{10}\left(1+\frac{f_c^2}{f^2}\right) \tag{7.3.4}$$

Where

f_c is the critical frequency,

f is the frequency of interest,

$A_v^{'}$ is the decibel gain at the frequency of interest.

Example 7.3.1

An amplifier has a lower break frequency of 40 Hz. How much gain is lost at 10 Hz?

$$A_v^{'} = -10\log_{10}\left(1+\frac{f_c^2}{f^2}\right)$$
$$A_v^{'} = -10\log_{10}\left(1+\frac{40^2}{10^2}\right)$$
$$A_v^{'} = -12.3\ dB$$

In other words, the gain is 12.3 dB lower than it is in the midband. Note that 10 Hz is 2 octaves below the break frequency. Because the cutoff slope is 6 dB per octave, each octave loses 6 dB. Therefore, the approximate result is -12 dB, which double-checks the exact result. Without the lead network, the gain would stay at 0 dB all the way down to DC (0 Hz.)

1.3.2: Lead Network Phase Response

At very low frequencies, the circuit of Figure 7.3.2 is largely capacitive. Because of this, the output voltage developed across R leads by 90 degrees. At very high frequencies the circuit will be largely resistive. At this point V_{out} will be in phase with V_{in}. At the critical frequency, V_{out} will lead by 45 degrees. A general phase graph is shown in Figure 7.3.5. As with the gain plot, the phase plot shape is the same for any lead network. The general phase Equation may be obtained from the voltage divider:

$$\frac{V_{out}}{V_{in}} = \frac{R}{R - j\ X_c}$$
$$\frac{V_{out}}{V_{in}} = \frac{R\angle 0}{\sqrt{R^2+X_c^2}\angle -\arctan\frac{X_c}{R}}$$

The phase portion of this is,

$$\theta = \arctan\frac{X_c}{R}$$

By using Equation 7.3.2, this simplifies to,

$$\theta = \arctan \frac{f_c}{f} \tag{7.3.5}$$

Where

f_c is the critical frequency,

f is the frequency of interest,

θ is the phase angle at the frequency of interest.

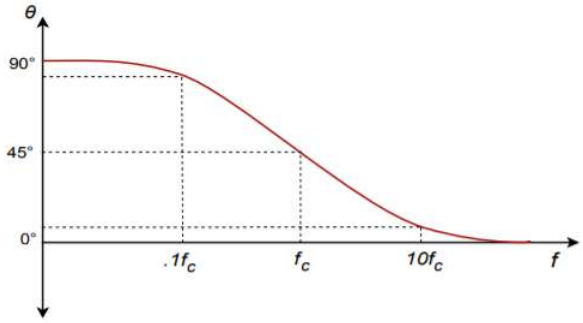


Figure 7.3.5: Lead phase (exact).

Often, an approximation such as Figure 7.3.6 used in place of 7.3.5. By using Equation 7.3.5, you can show that the approximation is off by no more than 6 degrees at the corners.

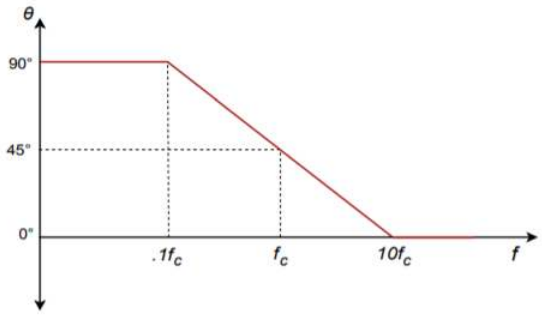


Figure 7.3.6: Lead phase (approximate).

Example 7.3.2

A telephone amplifier has a lower break frequency of 120 Hz. What is the phase response one decade below and one decade above?

One decade below 120 Hz is 12 Hz, while one decade above is 1.2 kHz.

$$\theta = \arctan \frac{f_c}{f}$$

$$\theta = \arctan \frac{120\ Hz}{12\ Hz}$$

$\theta = 84.3$ degrees one decade below f_c (i.e, approaching 90 degrees)

$$\theta = \arctan \frac{120}{1.2\ kHz}$$

$\theta = 5.71$ degrees one decade above f_c (i.e., approaching 0 degrees)

> Remember, if an amplifier is direct-coupled, and has no lead networks, the phase will remain at 0 degrees right back to 0 Hz (DC).

1.3.3: Lag Network Response

Unlike its lead network counterpart, all amplifiers will contain lag networks. In essence, it's little more than an inverted lead network. As you can see from Figure 7.3.7, it simply transposes the R and C locations. Because of this, the response tends to be inverted as well. In terms of gain, X_c is very large at low frequencies, and thus V_{out} equals V_{in}. At high frequencies, X_c decreases, and V_{out} falls. The break point occurs when X_c equals R. The general gain plot is shown in Figure 7.3.8. Like the lead network response, the slope of this curve is -6 dB per octave (or -20 dB per decade.) Note that the slope is negative instead of positive. A straight-line approximation is shown in Figure 7.3.9. We can derive a general gain Equation for this circuit in virtually the same manner as we did for the lead network. The derivation is left as an exercise.

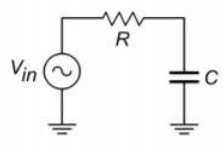


Figure 7.3.7: Lag network.

$$A_v' = -10 \log_{10} \left(1 + \frac{f^2}{f_c^2}\right) \tag{7.3.6}$$

Where

f_c is the critical frequency,

f is the frequency of interest,

A_v' is the decibel gain at the frequency of interest.

Note that this Equation is almost the same as Equation 7.3.4. The only difference is that f and f_c have been transposed.

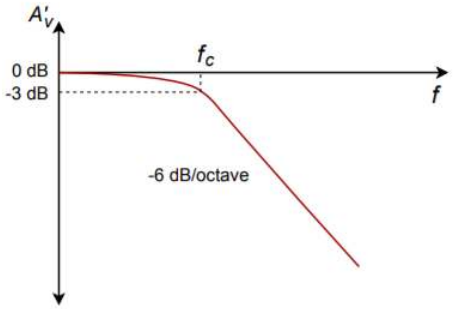


Figure 7.3.8: Lag gain (exact).

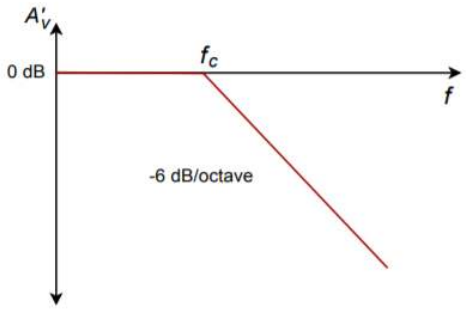


Figure 7.3.9: Lag gain (approximate).

In a similar vein, we may examine the phase response. At very low frequencies, the circuit is basically capacitive. Because the output is taken across C, V_{out} will be in phase with V_{in}. At very high frequencies, the circuit is essentially resistive. Consequently, the output voltage across C will lag by 90 degrees. At the break frequency the phase will be -45 degrees. A general phase plot is shown in Figure 7.3.10, with the approximate response detailed in Figure 7.3.11. As with the lead network,we may derive a phase equation. Again, the exact steps are very similar, and left as an exercise.

$$\theta = -90 + \arctan \frac{f_c}{f} \tag{7.3.7}$$

Where f_c is the critical frequency,

f is the frequency of interest,

θ is the phase angle at the frequency of interest.

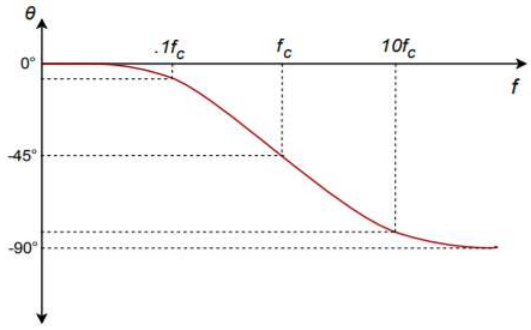


Figure 7.3.10: Lag phase (exact).

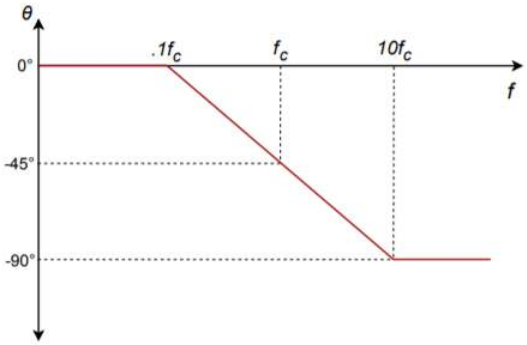


Figure 7.3.11: Lag phase (approximate).

Example 7.3.3

A medical ultra sound transducer feeds a lag network with an upper break frequency of 150 kHz. What are the gain and phase values at 1.6 MHz? Because this represents a little more than a 1 decade increase, the approximate values are -20 dB and -90 degrees, from Figures 7.3.9 and 7.3.11, respectively. The exact values are:

$$A_v^{'} = -10 \log_{10} \left(1 + \frac{f^2}{f_c^2}\right)$$
$$A_v^{'} = -10 \log_{10} \left(1 + \frac{1.6\ MHz^2}{150\ MHz^2}\right)$$
$$A_v^{'} = -20.6\ dB$$

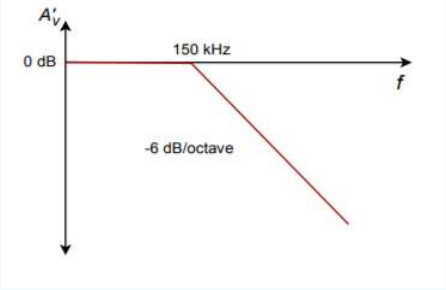


Figure 7.3.12: Bode plot for 150 kHz lag.

$$\theta = -90 + \arctan \frac{f_c}{f}$$
$$\theta = -90 + \arctan \frac{150\ kHz}{1.6\ MHz}$$
$$\theta = -84.6\ degrees$$

The complete Bode plot for this network is shown in Figure 7.3.12. It is very useful to examine both plots simultaneously. In this manner you can find the exact phase change for a given gain quite easily. This information is very important when the application of negative feedback is considered (Chapter Three). For example, if you look carefully at the plots of Figure 7.3.12, you will note that at the critical frequency of 150 kHz, the total phase change is -45 degrees. Because this circuit involved the use of a single lag network, this is exactly what you would expect.

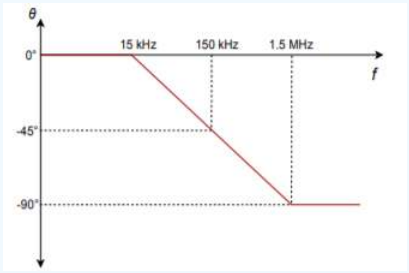


Figure 7.3.13: (continued) Bode plot for 150 kHz lag.

1.3.4: rise time versus Bandwidth

For pulse-type signals, the speed of an amplifier is often expressed in terms of its rise time. If a square pulse such as Figure 7.3.13a is passed into a simple lag network, the capacitor charging effect will produce a rounded variation, as seen in Figure 7.3.13b. This effect places an upper limit on the duration of pulses that a given amplifier can handle without producing excessive distortion.

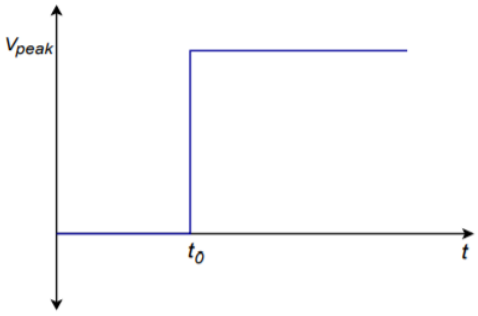


Figure 7.3.14a: Pulse rise time effect: Input to network.

By definition, rise time is the amount of time it takes for the signal to traverse from 10% to 90% of the peak value of the pulse. The shape of this pulse is defined by the standard capacitor charge Equation examined in earlier course work, and is valid for any system with a single clearly dominant lag network.

$$V_{out} = V_{peak}\left(1 - \epsilon^{\frac{-t}{RC}}\right) \tag{1.12}$$

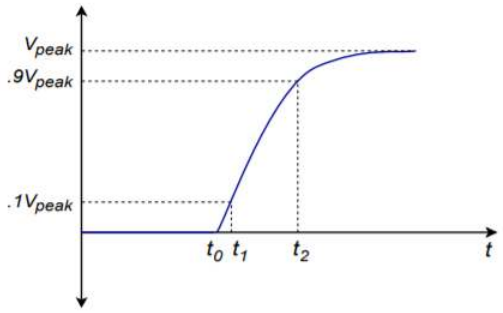


Figure 7.3.14*b*: Pulse rise time effect: Output of network.

In order to find the time internal from the initial starting point to the 10% point, set V_{out} to $0.1\ V_{peak}$ in Equation 1.12 and solve for t_1.

$$\begin{aligned} 0.1V_{peak} &= V_{peak}\left(1 - \epsilon^{\frac{-t_1}{RC}}\right) \\ 0.1V_{peak} &= V_{peak} - V_{peak}\ \epsilon^{\frac{-t_1}{RC}} \\ 0.9V_{peak} &= V_{peak}\ \epsilon^{\frac{-t_1}{RC}} \\ 0.9 &= \epsilon^{\frac{-t_1}{RC}} \\ \log\ 0.9 &= \frac{-t_1}{RC} \\ t_1 &= 0.105\ RC \end{aligned} \tag{7.3.8}$$

To find the interval up to the 90% point, follow the same technique using $0.9\ V_{peak}$. Doing so yields

$$t_2 = 2.303\ RC \tag{7.3.9}$$

The rise time, T_r, is the difference between t_1 and t_2

$$\begin{aligned} T_r &= t_1 - t_2 \\ T_r &= 2.303\ RC - 0.105\ RC \\ T_r &\approx 2.2\ RC \end{aligned} \tag{7.3.10}$$

Equation 7.3.10 ties the rise time to the lag network's R and C values. These same values also set the critical frequency f_2. By combining Equation 7.3.10 with basic critical frequency relationship, we can derive an Equation relating f_2 to T_r.

$$f_2 = \frac{1}{2\pi RC}$$

Solving 7.3.10 in terms of RC, and substituting yields

$$\begin{aligned} f_2 &= \frac{2.2}{2\pi T_r} \\ f_2 &= \frac{0.35}{T_r} \end{aligned} \tag{7.3.11}$$

Where f_2 is the upper critical frequency,

T_r is the rise time of the output pulse.

Example 7.3.4

Determine the rise time for a lag network critical at 100 kHz.

$$f_2 = \frac{0.35}{T_r}$$
$$T_r = \frac{0.35}{f_2}$$
$$T_r = \frac{0.35}{100\ kHz}$$
$$T_r = 3.5\ \mu s$$

References

[1]The term octave is borrowed from the field of music. It gets its name from the fact that there are eight notes in the standard western scale: do-re-me-fa-so-la-ti-do.

This page titled 7.3: Bode Plots is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.3: Bode Plots** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.4: Combining the Elements - Multi-Stage Effects

A complete gain or phase plot combines three elements: (1) the midband response, (2) the lead response, and (3) the lag response. Normally, a particular design will contain multiple lead and lag networks. The complete response is the summation of the individual responses. For this reason, it is useful to find the dominant lead and lag networks. These are the networks that affect the midband response first. For lead networks, the dominant one will be the one with the highest f_c. Conversely, the dominant lag network will be the one with the lowest f_c. It is very common to approximate the complete system response by drawing straight-line segments such as those given in Figures 1.3.4 and 1.3.9. The process goes something like this:

- Locate all f_cs on the frequency axis.
- Draw a straight line between the dominant lag and lead f_c s at the midband gain. If the system does not contain any lead networks, continue the midband gain line down to DC.
- Draw a 6 dB per octave slope between the dominant lead and the next lower lead network.
- Because the effects of the networks are cumulative, draw a 12 dB per octave slope between the second lead f_c and the third f_c. After the third f_c, the slope should be 18 dB per octave, after the fourth, 24 dB per octave, and so on.
- Draw a -6 dB per octave slope between the dominant lag f_c and the next highest f_c. Again, the effects are cumulative, so increase the slope by -6 dB at every new f_c.

Example 7.4.1

Draw the Bode gain plot for the following amplifier: A_v' midband = 26 dB, one lead network critical at 200 Hz, one lag network critical at 10 kHz, and another lag network critical at 30 kHz.

The dominant lag network is 10 kHz. There is only one lead network, so it's dominant by default.

- Draw a straight line between 200 Hz and 10 kHz at an amplitude of 26 dB.
- Draw a 6 dB per octave slope below 200 Hz. To do this, drop down one octave (100 Hz) and subtract 6 dB from the present gain (26 dB - 6dB = 20 dB.) The line will start at the point 200 Hz/26 dB, and pass through the point 100 Hz/20 dB. Because there are no other lead networks, this line may be extended to the left edge of the graph.
- Draw a -6 dB per octave slope between 10 kHz and 30 kHz. The construction point will be 20 kHz/20 dB. Continue this line to 30 kHz. The gain at the 30 kHz intersection should be around 16 dB. The slope above this second f_c will be -12 dB per octave. Therefore, the second construction point should be at 60 kHz/4 dB (one octave above 30 kHz, and 12 dB down from the 30 kHz gain). Because this is the final lag network, this line may be extended to the right edge of the graph.

A completed graph is shown in Figure 7.4.1.

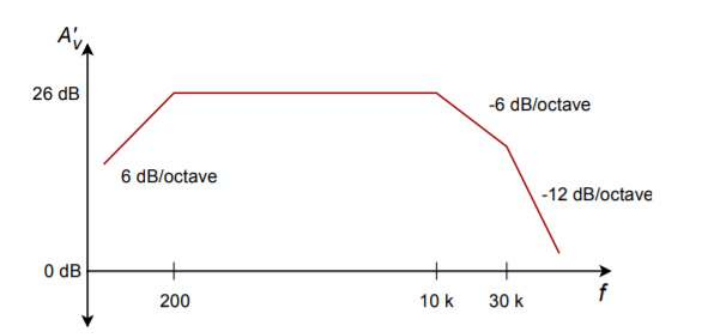


Figure 7.4.1: Gain plot of complete amplifier.

There is one item that should be noted before we leave this section, and that is the concept of narrowing. Narrowing occurs when two or more networks share similar critical frequencies, and one of them is a dominant network. The result is that the true -3 dB breakpoints may be altered. Here is an extreme example. Assume that a circuit has two lag networks, both critical at 1 MHz. A Bode plot would indicate that the breakpoint is 1 MHz. This is not really true. Remember, the effects of lead and lag networks are cumulative. Because each network produces a 3 dB loss at 1 MHz, the net loss at this frequency is actually 6 dB. The true -3 dB point will have been shifted. The Bode plot only gives you the approximate shape of the response.

This page titled 7.4: Combining the Elements - Multi-Stage Effects is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.4: Combining the Elements - Multi-Stage Effects** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.5: The Differential Amplifier

Most modern operational amplifiers utilize a differential amplifier front end. In other words, the first stage of the operational amplifier is a differential amplifier. This circuit is commonly referred to as a diff amp or as a long-tailed pair. A diff amp utilizes a minimum of 2 active devices, although 4 or more may be used in more complex designs. Our purpose here is to examine the basics of the diff amp so that we can understand how it relates to the larger operational amplifier. Therefore, we will not be investigating the more esoteric designs. To approach this in an orderly fashion, we will examine the DC analysis first, and then follow with the AC small signal analysis.

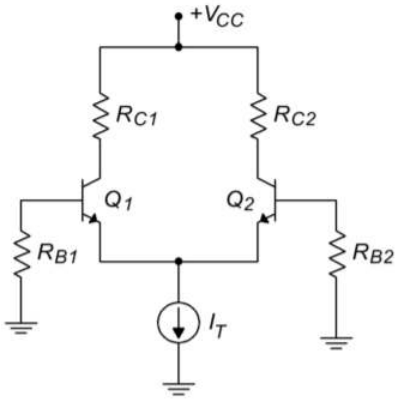


Figure 7.5.1: Simplified diff amp.

1.6.1: DC Analysis

A simplified diff amp is shown in Figure 7.5.1. This circuit utilizes a pair of NPN bipolar transistors, although the circuit could just as easily be built with PNPs or FETs. Note the inherent symmetry of the circuit. If you were to slice the circuit in half vertically, all of the components on the left half would have a corresponding component on the right half. Indeed, for optimal performance, we will see that these component pairs should have identical values. For critical applications, a matched pair of transistors would be used. In this case, the transistor parameters, such as β, would be very closely matched for the two devices.

In Figure 7.5.2, the circuit currents are noted, and the generalized current source has been replaced with a resistor/negative power supply combination. This is in essence, an emitter bias technique. Assuming that the base voltages are negligible and that V_{BE} is equal to 0.7 V, we can see that the emitter of each device is at approximately -0.7 V. Kirchhoff's Voltage Law indicates that the bulk of the negative supply potential must drop across R_T.

$$V_{RT} = | V_{EE} | - .7V$$

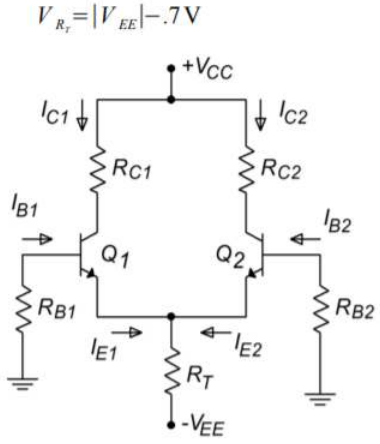


Figure 7.5.2: Diff amp analysis of Figure 7.5.1.

Knowing this, we may find the current through R_T, which is known as the tail current, I_T.

$$I_T = \frac{|\ V_{EE}\ | - 0.7V}{R_T}$$

If the two halves of the circuit are well matched, the tail current will split equally into two portions, I_{E1} and I_{E2}. Given identical emitter currents, it follows that the remaining currents and voltages in the two halves must be identical as well. These potentials and currents are found through the application of Kirchhoff's Voltage and Current Laws just as in any other transistor bias analysis.

Example 7.5.1

Find the tail current, the two emitter currents, and the two collector to ground voltages in the circuit of Figure 7.5.3. You may assume that the two transistors are very closely matched.

The first step is to find the tail current:

$$I_T = \frac{|\ V_{EE}\ | - 0.7V}{R_T}$$

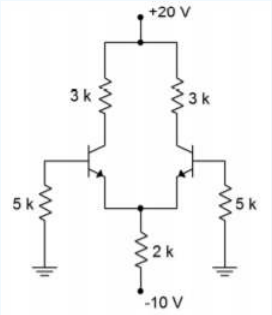


Figure 7.5.3: Diff amp for example 7.5.1.

The tail current is the combination of the two equal emitter currents, so

$$I_{EI} = I_{E2} = I_{T2}$$
$$I_{EI} = I_{E2} = \frac{4.65\ mA}{2}$$
$$I_{EI} = I_{E2} = 2.325\ mA$$

If we make the approximation that collector and emitter currents are equal, we may find the collector voltage by calculating the voltage drop across the collector resistor, and subtracting the result from the positive power supply.

$$V_c = V_{cc} - I_c\ R_c$$
$$V_c = 20\ V - 2.325\ mA \times 3k\Omega$$
$$V_c = 20\ V - 6.975\ V$$
$$V_c = 13.025\ V$$

Again, because we have identical values for both halves of the circuit, $V_{C1} = V_{C2}$. If we continue with this and assume a typical β of 100, we find that the two base currents are identical as well.

$$I_B = \frac{I_c}{\beta}$$
$$I_B = \frac{2.325\ mA}{100}$$
$$I_B = 23.25\ \mu A$$

Noting that the base currents flow through the 5 kΩ base resistors, we may find the base voltages. Note that this is a negative potential because the base current is flowing from ground into the transistor's base.

$$V_B = -I_b\ R_B$$
$$V_B = -23.25\ \mu A \times 5k\Omega$$
$$V_B = -116.25\ mV$$

This result indicates that the actual emitter voltage is closer to -0.8 V than -0.7 V, and thus, the tail current is actually a little less than our approximation of 4.65 mA. This error is probably within the error we can expect by using the 0.7 V junction potential approximation.

1.6.2: Input Offset Current and Voltage

As you have no doubt guessed, it is impossible to make both halves of the circuit identical, and thus, the currents and voltages will never be exactly the same. Even a small resistor tolerance variation will cause an upset. If the base resistors are mismatched, this will cause a direct change in the two base potentials. A variation in collector resistance will cause a mismatch in the collector potentials. A simple β or V_{BE} mismatch can cause variations in the base currents and base voltages, as well as smaller changes in emitter currents and collector potentials. It is desirable then to quantify the circuit's performance so that we can see just how well balanced it is. We can judge a diff amp's DC performance by measuring its input offset current and its input and output offset voltages. In simple terms, the difference between the two base currents is the input offset current. The difference between the two collector voltages is the output offset voltage. The DC potential required at one of the bases to counteract the output offset voltage is called the input offset voltage (this is little more than the output offset voltage divided by the DC gain of the amplifier). In an ideal diff amp all three of these factors are equal to 0. We will take a much closer look at these parameters and how they relate to operational amplifiers in later chapters. For now, it is only important that you understand that these inaccuracies exist, and what can cause them.

1.6.3: AC Analysis

Figure 7.5.4 shows a typical circuit with input and output connections. In order to minimize confusion with the DC circuit, AC equivalent values will be shown in lower case. Small emitter degeneration resistors, r_{E1} and r_{E2}, have been added to this

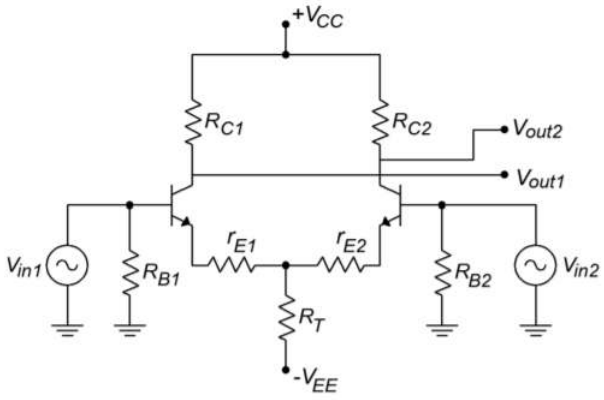


Figure 7.5.4: A typical diff amp with input and output connections.

diff amp. This circuit has two signal inputs and two signal outputs. It is possible to configure a diff amp so that only a single input and/or output is used. This means that there are four variations on the theme:

- Differential (also called dual- or double-ended) input, differential output.
- Differential input, single-ended output.
- Single-ended input, differential output.
- Single-ended input, single-ended output.

These variations are shown in Figure 7.5.5. For use in operational amplifiers, the differential input/single-ended output variation is the most common. We will examine the most general case, the differential input/differential output version.

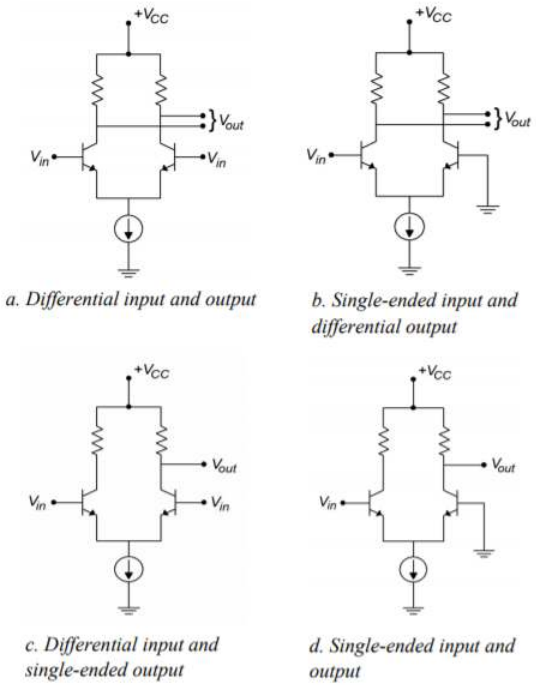


Figure 7.5.5: The four different diff amp input/output configurations.

Because the diff amp is a linear circuit, we can use the principle of Superposition to independently determine the output contribution from each of the inputs. Utilizing the circuit of Figure 7.5.4, we will first determine the gain Equation from V_{in1} to either output. To do this, we replace V_{in2} with a short circuit. The AC equivalent circuit is shown in Figure 7.5.6.

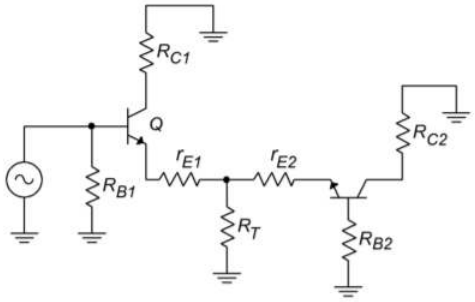


Figure 7.5.6: The circuit of Figure 7.5.4 redrawn for AC analysis.

For the output on collector 1, transistor 1 forms the basis of a common emitter amplifier. The voltage across r_{C1} is found via Ohm's Law.

$$v_{r_{C1}} = -i_{C1}\ r_{C1}$$

The negative sign comes from the fact that AC ground is used as our reference. (i.e., for a positive input, current flows from AC ground down through r_{C1}, and into the collector.) To a reasonable approximation, we can say that the collector and emitter currents are identical.

$$v_{r_d} = -i_{EI}\ r_{C1}$$

We must now determine the AC emitter current in relation to V_{in1}. In order to better visualize the process, the circuit of Figure 7.5.6 is altered to include simplified transistor models, as shown in Figure 7.5.7.

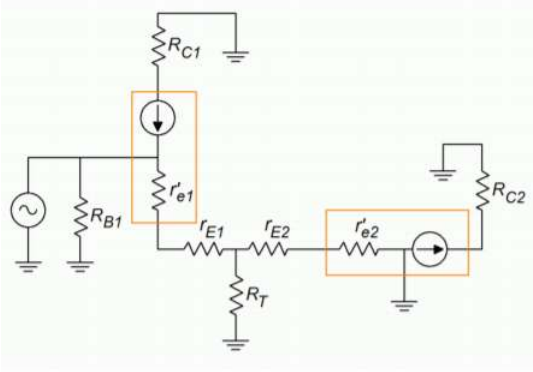


Figure 7.5.7: AC analysis.

$r_e^{'}$ is the dynamic resistance of the base emitter junctions and is inversely proportional to the DC emitter current. You may recall the following Equation from your prior course work:

$$r_e^{'} = \frac{26\ mV}{I_E}$$

Where

$r_e^{'}$ is the dynamic base-emitter junction resistance,

I_E is the DC emitter current.

For typical circuits, the values of $r_e^{'}$ and r_E are much smaller than the tail current biasing resistor, R_T. Because of its large size, we can ignore the parallel effect of R_T. By definition, the AC emitter current must equal the AC emitter potential divided by the AC resistance in the emitter section. If you trace the signal flow from the base of transistor 1 to ground, you find that it passes through $r_{e1}^{'}$,r_{E1}, $r_{e2}^{'}$ and r_{E2}. You will also notice that the magnitude of i_{E1} is the same as i_{E2}, although they are out of phase.

$$i_E = \frac{v_{in1}}{r_{e1}^{'} + r_{E1} + r_{e2}^{'} + r_{E2}}$$

Because the circuit values should be symmetrical for best performance, this Equation may be simplified to

$$i_E = \frac{v_{in}}{2(r_e^{'} + r_E)}$$

If we now solve for voltage gain,

$$A_v = -\frac{v_{out}}{v_{in}}$$

$$A_v = \frac{-i_E\ r_c}{v_{in}}$$

$$A_v = \frac{\frac{v_{in}}{2(r_e^{'}+r_E)} r_c}{v_{in}}$$

$$A_v = \frac{-r_c}{2(r_e^{'} + r_E)}$$

Where

A_v is the voltage gain,

r_C is the AC equivalent collector resistance,

r_E is the AC equivalent emitter resistance,

$r_e^{'}$ is the dynamic base-emitter junction resistance.

The final negative sign indicates that the collector voltage at transistor number 1 is 180 degrees out of phase with the input signal. Earlier, we noted that i_{E2} is the same magnitude as i_{E1}, the only difference being that it is out of phase. Because of this, the magnitude of the collector voltage at transistor number 2 will be the same as that on the first transistor. Because the second current is out of phase with the first, it follows that the second collector voltage must be out of phase with the first. This means that the voltage at the second collector is in phase with the first input signal. Its gain Equation is

$$A_v = \frac{r_c}{2(r_e^{'} + r_E)}$$

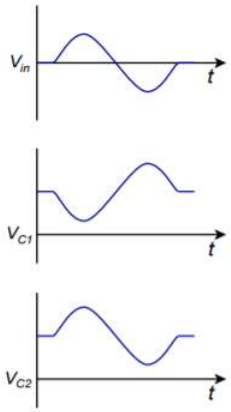


Figure 7.5.8: Waveforms for a single input.

The various waveforms are depicted in Figure 7.5.8. The preceding Equation is often referred to as the single-ended input/single-ended output gain Equation because it describes the single change from one input to one output. The output signal will be in phase if we are examining the opposite transistor, and out of phase if we are looking at the input transistor. Because the circuit is symmetrical, we will get similar results when we examine the second input. The voltage between the two collectors is 180 degrees apart. If we were to use a differential output, that is, derive the output from collector to collector rather than from one collector to ground, we would see an effective doubling of the output signal. If the reason for this is not clear to you, consider the following. Assume that each collector has a 1 V peak sine wave riding on it. When collector 1 is at +1 V, collector 2 is at -1 V, making +2 V total. Likewise, when collector 1 is at its negative peak, collector 2 is at its positive peak, producing a total of -2 V. The single ended input/differential output gain therefore is

$$A_v = \frac{r_c}{r_e^{'} + r_E}$$

Example 7.5.2

Using the circuit of Figure 7.5.4, determine the single-ended input/differential output and single-ended input/single-ended output voltage gains. Use the following component values: $V_{CC} = 15V$, $V_{EE} = -8V$, $R_T = 10k\Omega$, $R_C = 8k\Omega$, $r_E = 30\Omega$. In order to find $r_e^{'}$ we must find the DC current.

$$I_T = \frac{|\, V_{EE} \,| - 0.7V}{R_T}$$
$$I_T = \frac{7.3\ V}{10\ k\Omega}$$
$$I_T = 730\ \mu A$$

$$I_E = \frac{I_T}{2}$$
$$I_E = \frac{730\ \mu A}{2}$$
$$I_E = 365\ \mu A$$

$$r_e^{'} = \frac{26\ mV}{I_E}$$
$$r_e^{'} = \frac{26\ mV}{365\ \mu A}$$
$$r_e^{'} = 71.2\ \Omega$$

For the single ended output gain,

$$A_v = \frac{r_c}{2(r_e^{'} + r_E)}$$

$$A_v = \frac{8\ k\Omega}{2(71.2\ \Omega + 30\ \Omega)}$$

$$A_v = \frac{8\ k\Omega}{202.4\ \Omega}$$

$$A_v = 39.5$$

The differential output gain is twice this value, or 79.

Because it is possible to drive a diff amp with two distinct inputs, a wide variety of outputs may be obtained. It is useful to investigate two specific cases:

- Two identical inputs in both phase and magnitude.
- Two inputs with identical magnitude, but 180 degrees out of phase.

Let's consider the collector potentials for the first case. Assume that a diff amp has a single-ended input/single-ended output gain of 100 and a 10 mV signal is applied to both bases. Using Superposition, we find that the outputs due to each input are 100 times 10 mV, or 1 V in magnitude. For the first input, the voltages are sketched in Figure 7.5.9a (following page). For the second input, the voltages are sketched in Figure 7.5.9b. Note that each collector sees both a sine wave and an inverted sine wave, both of equal amplitude. When these two signals are added, the result is zero, as seen in Figure 7.5.9c. In Equation form,

$$v_{C1} = v_{in1}(-A_v) + v_{in2}\ A_v$$

$$v_{C1} = A_v(v_{in2} - v_{in1})$$

Because v_{in1} and v_{in2} are identical, the output is ideally zero given a perfectly matched and biased diff amp. The exact same effect is seen on the opposite collector. This last Equation is very important. It says that the output voltage is equal to the gain times the difference between the two inputs. This is how the differential amplifier got its name. In this case, the two inputs are identical, and thus their difference is zero. On the other hand, if we were to invert one of the input signals(case 2), we find a completely different result.

$$v_{in1} = -v_{in2}$$
$$v_{C1} = A_v(v_{in2} - v_{in1})$$
$$v_{C1} = A_v(v_{in2} - (-v_{in2}))$$
$$v_{C1} = 2\ A_v\ v_{in2}$$

Thus, if one input is inverted, the net result is a doubling of gain. This effect is shown graphically in Figures 7.5.9d through 7.5.9f. In short, a differential amplifier suppresses in phase signals while simultaneously boosting out of phase signals. This can be a very useful attribute, particularly in the area of noise reduction.

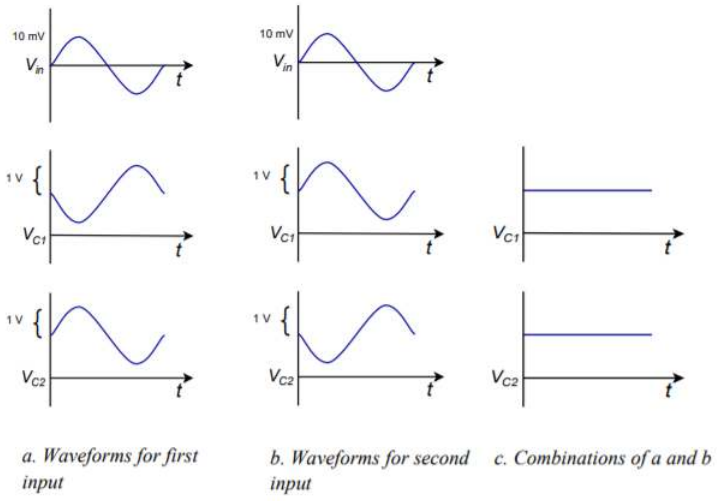


Figure 7.5.9: Input-output waveforms for common mode.

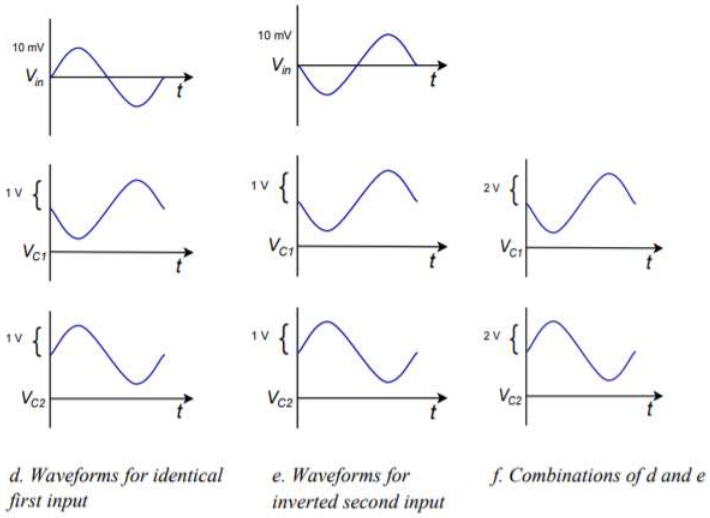


Figure 7.5.9: (continued) Input-output waveforms for common mode.

1.6.4: Common Mode Rejection

By convention, in phase signals are known as common-mode signals. An ideal differential amplifier will perfectly suppress these common-mode signals, and thus, its common-mode gain is said to be zero. In the real world, a diff amp will never exhibit perfect common-mode rejection. The common-mode gain may be made very small, but it is never zero. For a common-mode gain of zero, the two halves of the circuit have to be perfectly matched, and all circuit elements must be ideal. This is impossible to achieve as errors may arise from several sources. The most obvious error sources are resistor tolerance variations and transistor parameter spreads. The basic design of the circuit will also affect the common-mode gain. With some circuit rearrangements, it is possible to determine a common-mode gain for the circuits we have been using. The circuit of Figure 7.5.4 has been redrawn in Figure 7.5.10 in order to emphasize its parallel symmetry.

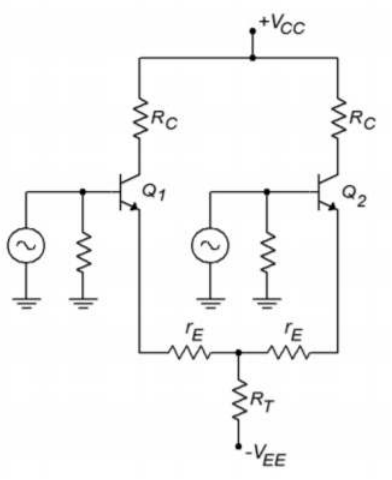


Figure 7.5.10: The circuit of Figure 7.5.4 redrawn for common mode rejection ratio (CMRR) analysis.

Because the DC potentials are identical in both halves, and identical signals drive both inputs, we can combine resistors in parallel in order to arrive at the circuit of Figure 7.5.11.

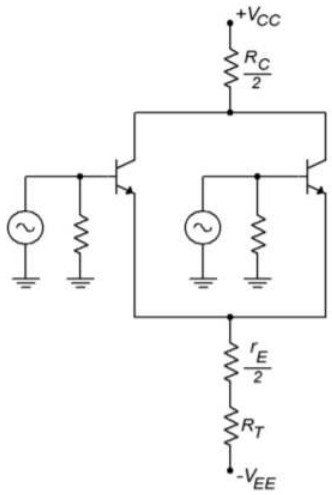


Figure 7.5.11: Common mode gain analysis.

Although it is not shown explicitly on the diagram, the internal dynamic resistances (r_e') may also be combined ($r_e'/2$). This circuit has been effectively reduced to a simple common emitter stage. Based on our earlier work, the gain for this circuit is

$$A_{v(cm)} = \frac{\frac{r_C}{2}}{R_T + \frac{r_e'}{2} + \frac{r_E}{2}}$$

This is the common-mode voltage gain. If R_T is considerably larger than r_C, then this circuit will exhibit good common mode rejection (assuming that the other parts are matched, naturally). R_T is the effective resistance of the tail current source. A very high internal resistance (i.e., an ideal current source) is desirable. There are many ways of creating a more ideal current source. One way is to use a third bipolar transistor as shown in Figure 7.5.12.

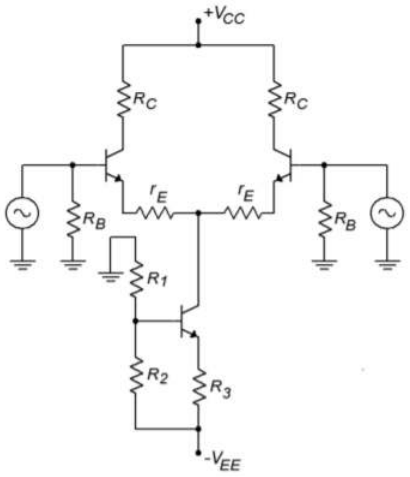


Figure 7.5.12: Improved current source.

The tail current is found by determining the potential across R_2 and subtracting the 0.7 V V_{BE} drop. The remaining potential appears across R_3. Given the voltage and resistance, Ohm's Law will let you find the tail current. In this circuit, R_2 is sometimes replaced with a Zener diode. This can help to reduce temperature induced current fluctuations. In any case, the effective resistance of this current source is considerably larger than the simple tail resistor variation. It is largely dependent on the characteristics of the tail current transistor, and can easily be in the megohm region.

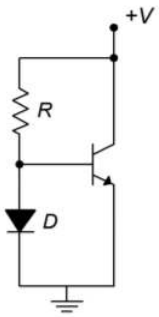


Figure 7.5.13: Current mirror.

1.6.5: Current Mirror

A very popular biasing technique in integrated circuits involves the current mirror. Current mirrors are also employed as active loads in order to optimize a circuit's gain. A simple current mirror is shown in Figure 7.5.13. This circuit requires that the transconductance curves of the diode and the transistor be very closely matched. One way to guarantee this is to use two transistors, and form one of them into a diode by shorting its collector to its base. If we use an approximate forward bias potential of 0.7 V and ignore the small base current, the current through the diode is

$$I_D = \frac{V_{CC} - 0.7\ V}{R}$$

In reality, the diode potential will probably not be exactly 0.7 V. This will have little effect on I_d though. Because the diode is in parallel with the transistor's base-emitter junction, we know that $V_d = V_{BE}$. If the two devices have identical transconductance curves, the transistor's emitter current will equal the diode current. You can think of the transistor as mirroring the diode's current, hence the circuit's name. If the two device curves are slightly askew, then the two currents will not be identical. This is shown graphically in Figure 7.5.14.

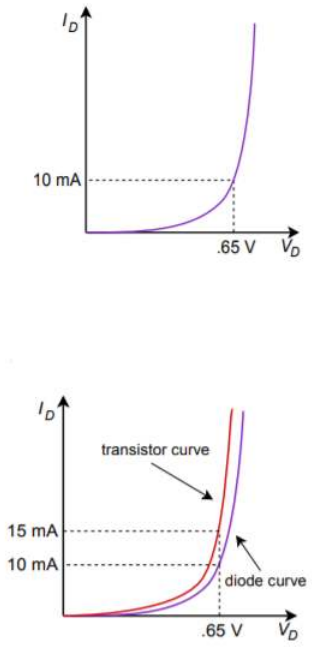


Figure 7.5.14: Transfer curve mismatch.

Example 7.5.3

A current mirror could be used in the circuit of Figure 7.5.12. The result is shown in Figure 7.5.15. If the positive power supply is 15 V, the negative supply is -10 V, and R is 10 k\Omega, the tail current will be

$$I_D = \frac{V_{CC} - V_{EE} - V_D}{R}$$

$$I_D = \frac{15\ V - (-10\ V) - 0.7\ V}{10k\Omega}$$

$$I_D = 2.43\ mA$$

Because the tail current is the mirror current,

$$I_T = I_D$$

$$I_T = 2.43\ mA$$

Biasing of this type is very popular in operational amplifiers. Another use for current mirrors is in the application of active loads. Instead of using simple resistors for the collector loads, a current mirror may be used instead. A PNP based current mirror suitable for use as an active load in our previous circuits is shown in Figure 7.5.16.

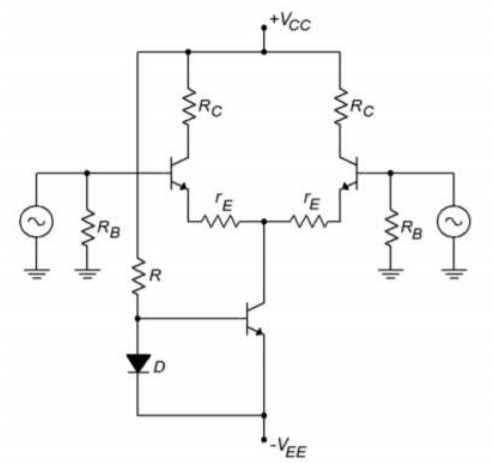


Figure 7.5.15: Current mirror bias.

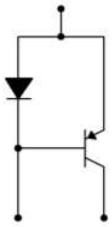


Figure 7.5.16: Active load current mirror.

To use this, we simply remove the two collector resistors from a circuit such as Figure 7.5.15, and drop in the current mirror. The result of this operation is shown in Figure 7.5.17. The current mirror active load produces a very high internal impedance, thus contributing to a very high differential gain. In effect, by using a constant current source in the collectors, all AC current is forced into the following stage. You may also note that the number of resistors used in the circuit has decreased considerably.

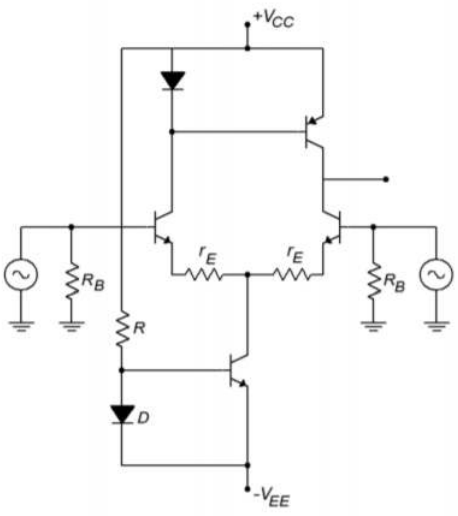


Figure 7.5.17: Current mirrors for bias and active load.

This page titled 7.5: The Differential Amplifier is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.6: The Differential Amplifier** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.6: Summary

We have seen how to convert gains and signals into a decibel form for both powers and voltages. This is convenient because what would require multiplication and division under the ordinary scheme only requires simple addition and subtraction in the dB scheme. Along with this, dB measurement is used almost exclusively for Bode gain plots. A Bode plot details a system's gain magnitude and phase response. For gain, the amplitude is measured in dB, while the frequency is normally presented in log form. For a phase plot, phase is measured in degrees, and again, the frequency axis is logarithmic. The changes in gain and phase at the frequency extremes are caused by lead and lag networks. Lead networks cause the low frequency gain to roll off. The roll off rate is 6 dB per octave per network. The phase will change from +90 degrees to 0 degrees per network. Lag networks cause the high frequency gain to roll off at a rate of -6 dB per octave per network. The phase change per lag network is from 0 degrees to -90 degrees. It was noted that computers may be used to quickly tabulate the response of complex networks such as these. Many computer circuit simulators are based on the original SPICE program. Among the popular commercial simulation programs are PSpice and Multisim. Freeware versions include LTspice and TINA-TI. Packages such as these offer graphical schematic capture tools and large component libraries.

Differential amplifiers are symmetrical circuits, employing a minimum of two active devices. They may be configured with single or dual inputs, and single or dual outputs. Diff amps are commonly used as the first stage of an operational amplifier. They tend to amplify the difference in the input signals while simultaneously suppressing in-phase, or common-mode, signals. Current mirrors are widely used for biasing purposes and as active loads. Active loads offer the advantage of producing higher gains than ordinary resistive loads.

This page titled 7.6: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.7: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.7: Problems

1.8.1: Review Questions

1. What are the advantages of using decibels over the ordinary scheme?

2. How do decibel power and voltage gain calculations differ?

3. What does the third letter in a decibel-based signal measurement indicate (as in dBV or dBm)?

4. What is a Bode plot?

5. What is a lead network? What general response does it yield?

6. What is a lag network? What general response does it yield?

7. What do the terms f_1 and f_2 indicate about a system's response?

8. What are the rolloff slopes for lead and lag networks?

9. What are the phase changes produced by individual lead and lag networks?

10. How is rise time related to upper break frequency?

11. How do multiple lead or lag networks interact to form an overall system response?

12. What is SPICE?

13. What is common-mode rejection?

14. What is a current mirror?

15. What is the advantage of using an active load?

1.8.2: Problems

Analysis Problems - dB emphasis

1. Convert the following power gains into dB form:

a) 10

b) 80

c) 500

d) 1

e) 0.2

f) 0.03.

2. Convert the following dB power gains into ordinary form:

a) 0 dB

b) 12 dB

c) 33.1 dB

d) 0.2 dB

e) −5.4 dB

f) −20 dB.

3. An amplifier has an input signal of 1 mW, and produces a 2 W output. What is the power gain in dB?

4. A hi-fi power amplifier has a maximum output of 50 W and a power gain of 19 dB. What is the maximum input signal power?

5. An amplifier with a power gain of 27 dB is driven by a 25 mW source. Assuming the amplifier doesn't clip, what is the output signal in watts?

6. Convert the following voltage gains into dB form:

a) 10

b) 40

c) 250

d) 1

e) 0.5

f) 0.004

7. Convert the following dB voltage gains into ordinary form:

a) 0.5 dB

b) 0 dB

c) 46 dB

d) 10.7 dB

e) −8 dB

f) −14.5 dB

8. A guitar pre-amp has a gain of 44 dB. If the input signal is 12 mV, what is the output signal?

9. A video amplifier has a 140 mV input and a 1.2 V output. What is the voltage gain in dB?

10. The pre-amp in a particular tape deck can output a maximum signal of 4 V. If this amplifier has a gain of 18 dB, what is the maximum input signal?

11. Convert the following powers into dBW:

a) 1 W

b) 23 W

c) 6.5 W

d) 0.2W

e) 2.3 mW

f) 1.2 kW

g) 0.045 mW

h) 0.3 μW

i) 5.6E−18 W.

12. Repeat Problem 11 for units of dBm.

13. Repeat Problem 11 for units of dBf.

14. Convert the following voltages into dBV:

a) 12.4 V

b) 1 V

c) 0.25 V

d) 1.414 V

e) 0.1 V

f) 10.6 kV

g) 13 mV

h) 2.78 μV.

15. A two stage power amplifier has power gains of 12 dB and 16 dB. What is the total gain in dB and in ordinary form?

16. If the amplifier of Problem 15 has an input of −18 dBW, what is the final output in dBW? in dBm? in watts?

17. Referring to Figure 1.2.1, what are the various stages' outputs if the input is changed to −4 dBm? to −34 dBW?

18. Which amplifier has the greatest power output?

a) 50 watts

b) 18 dBW

c) 50 dBm.

19. Which amplifier has the greatest power output?

a) 200 mW

b) −10 dBW

c) 22 dBm.

20. A three stage amplifier has voltage gains of 20 dB, 5 dB, and 12 dB respectively. What is the total voltage gain in dB and in ordinary form?

21. If the circuit of Problem 20 has an input voltage of −16 dBV, what are the outputs of the various stages in dBV? In volts?

22. Repeat Problem 21 for an input of 12 mV.

23. Which amplifier produces the largest output voltage?

a) 15 V

b) 16 dBV

Analysis Problems – Bode plot emphasis

24. Given a lead network critical at 3 kHz, what are the gain and phase values at 100 Hz, 3 kHz, and 40 kHz?

25. Given a lag network tuned to 700 kHz, what are the gain and phase values at 50 kHz, 700 kHz, and 10 MHz? What is the rise time?

26. A noninverting amplifier has a midband voltage gain of 18 dB and a single lag network at 200 kHz. What are the gain and phase values at 30 kHz, 200 kHz, and 1 MHz. What is the rise time?

27. Repeat Problem 26 for an inverting (−180 degrees) amplifier.

28. Draw the Bode plot for the circuit of Problem 26.

29. Draw the Bode plot for the circuit of Problem 27.

30. An inverting (−180 degrees) amplifier has a midband gain of 32 dB and a single lead network critical at 20 Hz (assume the lag network f_c is high enough to ignore for low frequency calculations). What are the gain and phase values at 4 Hz, 20Hz, and 100 Hz?

31. Repeat Problem 29 with a noninverting amplifier.

32. Draw the Bode plot for the circuit of Problem 30.

33. Draw the Bode plot for the circuit of Problem 31.

34. A noninverting amplifier used for ultrasonic applications has a midband gain of 41 dB, a lag network critical at 250 kHz, and a lead network critical at 30 kHz. Draw its gain Bode plot.

35. Find the gain and phase at 20 kHz, 100 kHz, and 800 kHz for the circuit of Problem 34.

36. If the circuit of Problem 34 has a second lag network added at 300 kHz, What are the new gain and phase values at 20 kHz, 100 kHz, and 800 kHz?

37. Draw the gain Bode plot for the circuit of Problem 36.

38. What are the maximum and minimum phase shifts across the entire frequency spectrum for the circuit of Problem 36?

39. A noninverting DC amplifier has a midband gain of 36 dB, and lag networks at 100 kHz, 750 kHz, and 1.2 MHz. Draw its gain Bode plot.

40. What are the maximum and minimum phase shifts across the entire frequency spectrum for the circuit of Problem 39?

41. What is the maximum rate of high frequency attenuation for the circuit of Problem 39 in dB/decade?

42. If an amplifier has two lead networks, what is the maximum rate of low frequency attenuation in dB/octave?

Analysis Problems – Differential amplifier emphasis

43. Given the circuit of Figure 1.6.4, determine the single-ended input/singleended output gain for the following values: $R_B = 5k\Omega, R_T = 7.5k\Omega, R_C = 12k\Omega, V_{CC} = 25V, V_{EE} = -9V, r_E = 50\Omega$.

44. Determine the differential voltage gain in the circuit of Figure 1.6.13 if $R_B = 15k\Omega, R_1 = 5k\Omega, R_2 = 7k\Omega, R_3 = 10k\Omega, R_C = 20k\Omega, V_{CC} = 22V, V_{EE} = -12V, r_E = 75\Omega$.

45. For the circuit of Problem 44, determine the output at collector 2 if $V_{in1}(t) = 0.001 \sin 2\pi 1000t$ and $V_{in2}(t) = -0.001 \sin 2\pi 1000t$.

46. Determine the differential voltage gain in the circuit of Figure 1.6.15 if $R_b = 8k\Omega, R_{mirror} = 22k\Omega, R_C = 10k\Omega, V_{CC} = 18V, V_{EE} = -15V, r_E = 25\Omega$.

47. For the circuit of Problem 46, determine the output at collector 1 if $V_{in1}(t) = -0.005 \sin 2\pi 2000t$ and $V_{in2}(t) = 0.005 \sin 2\pi 2000t$.

48. Determine the tail and emitter currents in the circuit of Figure 1.6.17 if $R_B = 6k\Omega, R_{mirror} = 50k\Omega, V_{CC} = 15V, V_{EE} = -15V, r_E = 0\Omega$.

Challenge Problems

49. You would like to use a voltmeter to take dBm readings in a 600 Ω system. What voltage should produce 0 dBm?

50. Assuming that it takes about an 8 dB increase in sound pressure level in order to produce a sound that is subjectively “twice as loud” to the human ear, can a hi-fi using a 100 W amplifier sound twice as loud as one with a 40 W amplifier (assuming the same loudspeakers)?

51. Hi-fi amplifiers are often rated with a “headroom factor” in dB. This indicates how much extra power the amplifier can produce for short periods of time, over and above its nominal rating. What is the maximum output power of a 250 W amplifier with 1.6 dB headroom?

52. If the amplifier of Problem 34 picks up an extraneous signal that is a −10 dBV sine wave at 15 kHz, what is the output?

53. If the amplifier of Problem 39 picks up a high frequency interference signal at 30 MHz, how much is it attenuated over a normal signal? If this input signal is measured at 2 dBV, what should the output be?

54. If an amplifier has two lag networks, and both are critical at 2 MHz, is the resulting f_2 less than, equal to, or greater than 2 MHz?

55. If an amplifier has two lead networks, and both are critical at 30 Hz, is the resulting f_1 less than, equal to, or greater than 30 Hz?

Computer Simulation Problems

56. Use a simulator to plot the Bode gain response of the circuit in Problem 39.

57. Use a simulator to plot the Bode phase response of the circuit in Problem 34.

58. Use a simulation program to generate a Bode plot for a lead network comprised of a 1 kΩ resistor and a 100 nF capacitor.

This page titled 7.7: Problems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.8: Problems** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

SECTION OVERVIEW

7.8: Operational Amplifier Internals

Learning Objectives

After completing this chapter, you should be able to:

- Describe the internal layout of a typical op amp.
- Describe a simple op amp computer simulation model.
- Determine fundamental parameters from an op amp data sheet.
- Describe an op amp based comparator, and note where it might be used.
- Describe how integrated circuits are constructed.
- Define monolithic planar construction.
- Define hybrid construction.

7.8.1: Introduction to Operational Amplifiers

7.8.2: What Is An Op Amp?

7.8.3: Simple Op Amp Comparator

7.8.4: Op Amp Manufacture

7.8.5: Summary

7.8.6: Problems

Template:HideTOC

This page titled 7.8: Operational Amplifier Internals is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.8.1: Introduction to Operational Amplifiers

In this chapter we introduce the fundamentals of the operational amplifier, or op amp, as it is commonly known. We will investigate the construction and usage of a typical general-purpose op amp. Specific along with generalized internal circuits and associated block diagrams are examined. An initial op amp data sheet interpretation is given as well. Toward the middle of the chapter, the first op amp circuit examples are presented. The chapter finishes with an explanation of semiconductor integration and construction techniques. After finishing this chapter, you should be familiar with the concepts of what an op amp is composed of, how it is manufactured, and a beginning idea of how it might be used in application circuit design.

This page titled 7.8.1: Introduction to Operational Amplifiers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.8.2: What Is An Op Amp?

An operational amplifier is, in essence, a multi-stage high gain amplifier treated as a single entity. Normally, op amps have a differential input and a single-ended output. In other words, one input produces an inverted output signal, and the other input produces a noninverted output signal. Often, the op amp is driven from a bipolar power supply (i.e., two supplies, one positive and one negative). Just about any sort of active amplifying device may be used for the individual stages. Op amps can be made entirely from vacuum tubes or discrete bipolar transistors (and of course, they were made that way some years ago). The advances in semiconductor manufacture in the late 1960's and early 1970's eventually made it possible to miniaturize the required components and place the whole affair on a single silicon chip (hence the term, integrated circuit). Through common use, this is what is generally meant by the term op amp today.

As seen in Figure 7.8.2.1, a typical op amp has at least five distinct connections; an inverting input (labeled “-”), a noninverting input (labeled “+”), an output, and positive and negative power supply inputs. These power supply connections are sometimes referred to as supply rails. Note that a ground connection is not directly given. Rather, a ground connection is implied through the other connections. This symbol and its associated connections are typical, but by no means absolute. There are a wide variety of devices available to the designer that offer such features as differential outputs or unipolar power supply operation. In any case, some form of triangle will be used for the schematic symbol.

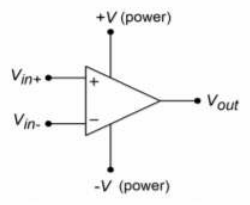


Figure 7.8.2.1: General op amp symbol.

It is best to think of op amps as general-purpose building blocks. With them, you can create a wide variety of useful circuits. For general-purpose work, designing with op amps is usually much quicker and more economical than an all-discrete approach. Likewise, trouble shooting and packaging constraints may be lessened. For more demanding applications, such as those requiring very low noise, high output current and/or voltage, or wide bandwidth, manufacturers have created specialized op amps. The final word in performance today is still dominated by discrete circuit designs though. Also, it is very common to see a mix of discrete and integrated devices in a given circuit. There is certainly no law that states that op amps can only be used with other op amps. Often, a judicious mix of discrete devices and op amps can produce a circuit superior to one made entirely of discretes or op amps alone.

Where might you find op amp circuits? In a word, anywhere. They're probably in use in your home stereo or TV where they help capture incoming signals, in electronic musical instruments where they can be used to create and modify tones, in a camera in conjunction with a light metering system, or in medical instruments where they might be used along with various bio-sensing devices. The possibilities are almost endless.

2.2.1: Block Diagram Of An Op Amp

At this point you may be asking yourself, “what's inside of the op amp?” The generic op amp consists of three main functional stages. A real op amp may contain more than three distinct stages, but can be reduced to this level for analysis. A generalized discrete representation is given in Figure 7.8.2.2. Since the op amp requires a differential input scheme, the first stage is most often a differential amplifier. As seen here, Q_1 and Q_2 comprise a PNP-based differential amplifier. The output of one collector (Q_2 here) is then fed to a high-gain second stage. This stage usually includes a lag network capacitor that plays a major roll in setting the op amp's AC characteristics (this is examined further in Chapter Five). Q_3 makes up the second stage in the example. It is set in a common emitter configuration for both current and voltage gain. The a-fore mentioned lag capacitor is positioned across Q_3's base-collector junction in order to take advantage of the Miller effect. The third and final section is a class B or class AB follower for the most effective load drive. Q_4 and Q_5 make up the final stage. The twin diodes compensate for the Q_4 and Q_5 V_{BE} drops, and produce a trickle bias current that minimizes distortion. This is a relatively standard class AB stage. Note that the entire circuit is

direct-coupled. There are no lead networks, and thus the op amp can amplify down to zero Hertz (DC). There are many possible changes that may be seen in a real world circuit, including the use of Darlington pairs or FETs for the differential amplifier, multiple high gain stages, and output current limiting for the class B section.

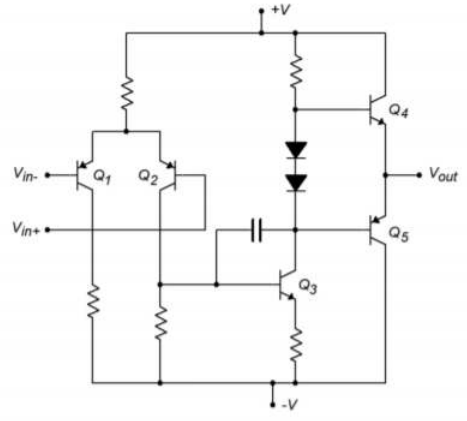


Figure 7.8.2.2: General op amp schematic.

The example discrete circuit uses only five transistors and two diodes. In contrast, an integrated version may use two to three dozen active devices. Because of the excellent device matching abilities of single chip integration, certain techniques are used in favor of standard discrete designs. Internal IC current sources are normally made through the use of current mirrors. Current mirror configurations are also employed to create active loads, in order to achieve maximum circuit gain.

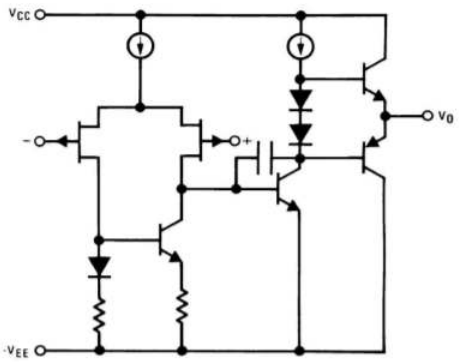


Figure 7.8.2.3: LF411 simplified schematic. Reprinted courtesy of Texas Instrutments

A typical integrated op amp will contain very few resistors, and usually only one or two lag network capacitors. Due to size limitations and other factors, inductors are virtually unseen in these circuits. A simplified equivalent circuit of the LF411 op amp is shown in Figure 7.8.2.3. Note that this device uses JFETs for the diff amp with an active load. The diff amp tail current source and the class AB trickle bias source are shown as simple current sources. In reality, they are a bit more complex, utilizing current mirror arrangements.

One of the most popular op amps over the years has been the 741. The specifications of this device seem rather lackluster by today's standards, but it was one of the first easy-to-use devices produced. As a result, it has found its way into a large number of designs. Indeed, it is still a wise choice for less demanding applications, or where parts costs are a major consideration. A complete schematic of the μA 741 is shown in Figure 7.8.2.4. Several different manufacturers make the 741. This version is manufactured by Signetics, and may be somewhat different than a 741 made by another company[1]. The circuit contains 20 active devices and about one dozen resistors.

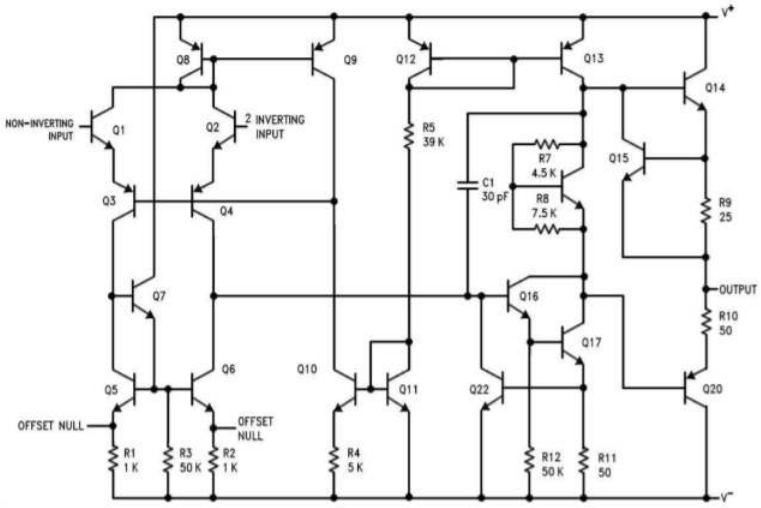


Figure 7.8.2.4: Schematic of μA741. Reprinted courtesy of Philips Semiconductors

At first glance, this circuit may look hopelessly confusing. A closer look reveals many familiar circuit blocks. First off, you will notice that a number of devices show a shorting connection between their base and collector terminals, such as Q_8, Q_{11}, and Q_{12}. In essence, these are diodes (they are drawn this way because they are manufactured as transistor junctions. It is actually easier to make diodes in this fashion). For the most part, these diodes are part of current mirror biasing networks. The bias setup is found in the very center of the schematic, and revolves around Q_9 through Q_{12}. The setup current is found by subtracting two diode drops (Q_{11}, \(Q_{12}) from the total power supply potential ($V_+ - V_-$), and dividing the result by R_5. For a standard ± 15 V power supply, this works out to

$$I_{bias} = \frac{V_+ - V_- - V_{BE-Q11} - V_{BE-Q12}}{R_5}$$

$$I_{bias} = \frac{30\ V - 1.4\ V}{39\ k\Omega}$$

$$I_{bias} = 733 \mu A$$

This current is reflected into Q_{13}. A close look at Q_{10} and Q_{11} reveals that this portion is not a simple current mirror. By including R_4, the voltage drop across the base emitter of Q_{10} is decreased, thus producing a current less than $733 \mu A$. This configuration is known as a Widlar current source. The derivation of the exact current Equation is rather involved, and beyond the scope of this chapter[2]. This current is reflected into Q_8 via Q_9, and establishes the tail current for the differential amplifier. The diff amp stage uses a total of four amplifying transistors in a common-collector/common-base configuration (Q_1 through Q_4). In essence, Q_1 and Q_2 are configured as emitter followers, thus producing high input impedance and reasonable current gain. Q_3 and Q_4 are configured as common base amplifiers, and as such, produce a large voltage gain. The gain is maximized by the active load comprised of Q_5 through Q_7. The output signal at the collector of Q_4 passes on to a dual transistor high-gain stage (Q_{16} and Q_{17}). Q_{16} is configured as an emitter follower and buffers Q_{17}, which is set as a common emitter voltage amplifier. Resistor R_{11} serves to stabilize both the bias and gain of this stage (i.e., it is an emitter degeneration or swamping resistor). Q_{17} is directly coupled to the class AB output stage (Q_{14} and Q_{20}). Note the use of a V_{BE} multiplier to bias the output transistors. The V_{BE} multiplier is formed from Q_{18} and resistors R_7 and R_8. Note that this section receives its bias current from Q_{13} which is part of the central current mirror complex.

Some transistors in this circuit are used solely for protection from overloads. A good example of this is Q_{15}. As the output current increases, the voltage across R_9 will increase proportionally. Note that this resistor is in parallel with the base emitter junction of Q_{15}. If this potential gets high enough, Q_{15} will turn on, shunting base drive current around the output device (Q_{14}). In this manner, current gain is reduced, and the maximum output current is limited to a safe value. This limiting value may be found via Ohm's Law:

$$I_{limit} = \frac{V_{BE}}{R_9}$$

$$I_{limit} = \frac{0.7\ V}{25\ \Omega}$$

$$I_{limit} = 28\ mA$$

In a similar fashion, Q_{20} is protected by R_{10}, R_{11}, and Q_{22}. If the output tries to sink too large of a current, Q_{22} will turn on, shunting current away from the base of Q_{16}. While individual op amp schematics will vary widely, they generally hold to the basic four-part theme presented here:

- A central current source/current mirror section to establish proper bias.
- A differential amplifier input stage with active load.
- A high voltage gain intermediate stage.
- A class B or AB follower output section.

Fortunately for the designer or repair technician, intimate knowledge of a particular op amp's internal structure is usually not required for successful application of the device. In fact, a few simple models can be used for the majority of cases. One very useful model is given in Figure 7.8.2.5.

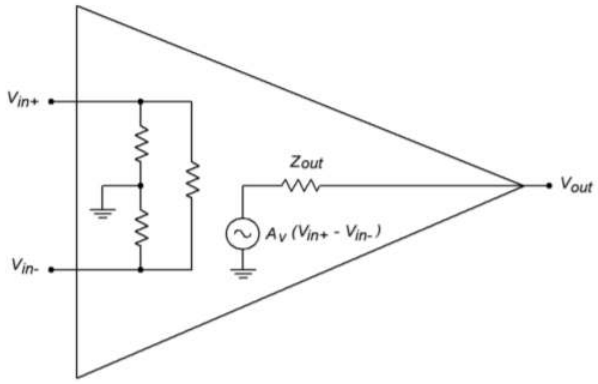


Figure 7.8.2.5: Simplified model.

Here the entire multistage op amp is modeled with a simple resistive input network, and a voltage source output. This output source is a dependent source. Specifically, it is a voltage-controlled voltage source. The value of this source is

$$E_{out} = A_v\ (V_{in+} - V_{in-})$$

The input network is specified as a resistance from each input to ground, as well as an input-to-input isolation resistance. For typical op amps these values are normally hundreds of kilo-ohms or more at low frequencies. Due to the differential input stage, the difference between the two inputs is multiplied by the system gain. This signal is presented to the output terminal through the final stage's output impedance. The output impedance will most likely be less than 100 Ω. System voltage gains in excess of 80 dB (10,000) are the norm.

2.2.2: A Simple Op Amp Simulation Model

It is possible to create a great variety of simulation models for any given op amp. Generally speaking, the more accurate the model is, the more likely it is to be complex. Due to the nature of most simulators, a more complex model requires a greater amount of time for an analysis to be completed. There is always a trade-off between model complexity and computation time. We can create a very simple model based on the previous section. This model is shown in Figure 7.8.2.6.

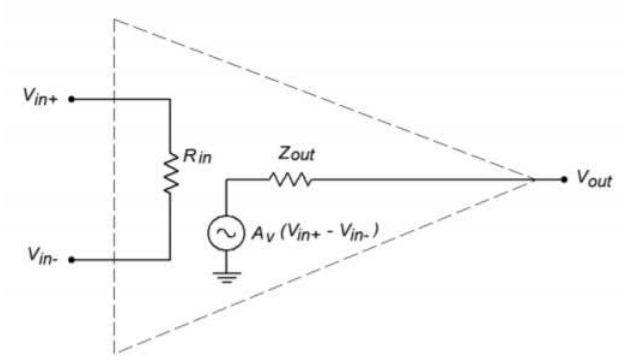


Figure 7.8.2.6: Simplified SPICE model.

It consists of just five nodes. The input section is modeled as a single resistor, R_{in}, between nodes 1 and 2. These two nodes are the noninverting and inverting inputs of the op amp, respectively. The second half of the model consists of a voltage controlled voltage source and an output resistor. The value of this dependent source is a function of the differential input voltage and the voltage gain. With a minimum of components, the simulation time for this model is very low. In order to use this model, you need only set three parameters, the input resistance, the output resistance, and the voltage gain. An example is shown in Figure 7.8.2.7 using Multisim.

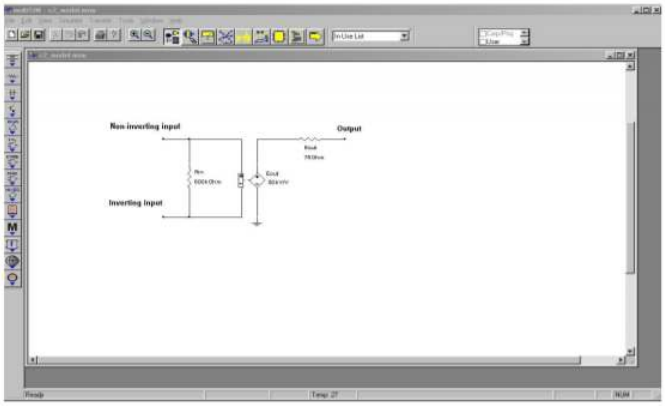


Figure 7.8.2.7: Simple op amp model in Multisim.

This model must be used with great care because it is so simplistic. It is useful as a learning tool for investigating the general operation of an op amp, but should never be considered as part of a true-to-life simulation. This model makes no attempt to consider the many limitations of the op amp. Because this model in no way imposes output signal swing limits, the effects of saturation will go unnoticed. Similarly, no attempt has been made at modeling the frequency response of the op amp. This is of great concern and we will spend considerable time on this subject in later chapters. Many other effects are also ignored. With so many limitations, you might wonder just where such a model may be used. This model is useful for non-critical simulations given low frequency inputs. You must also recognize the onset of saturation (clipping) yourself. Its primary advantage is that the circuit model is small, and thus computationally fast. Because of this, it is very efficient for students who are new to both op amps and circuit simulation. Perhaps of equal importance, is the fact that this model points out the fact that your simulation results can only be as good as the models you use. Many people fall into the trap that “because the simulation came from a computer, it must be correct”. Nothing could be further from the truth. Always remember the old axiom: GIGO (Garbage In = Garbage Out). It can be very instructive to simulate a circuit using differing levels of accuracy and complexity, and then noting how closely the results match the same circuit built in the laboratory.

2.2.3: Op Amp Data Sheet and Interpretation

Different manufacturers often use special codes and naming conventions to delineate their products from those of other manufacturers, as well as providing quality level and manufacturing information. A manufacturer's code is usually a letter prefix, while a quality or construction code is a suffix. Common prefix codes include μA (Fairchild), AD (Analog Devices), CS (Crystal),

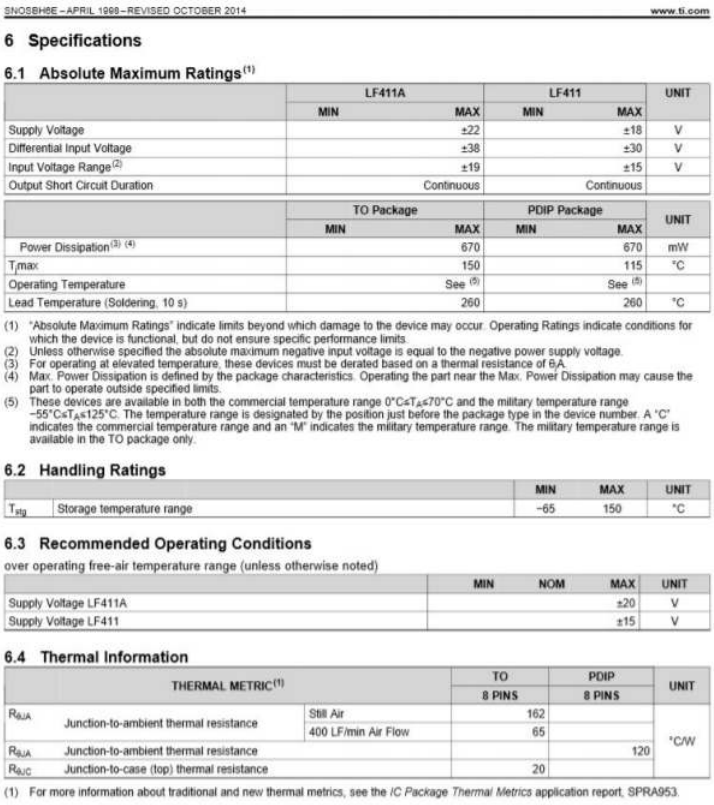
LM, LH, and LF (National Semiconductor, now owned by Texas Instruments, with M indicating monolithic construction, H indicating hybrid construction, and F indicating an FET device), LT (Linear Technology), MC (Motorola), NE and SE (Signetics), OPA (Burr-Brown), and TL (Texas Instruments).

Many manufacturers make a host of standard parts such as the 741. For example, Texas Instruments makes the LM741, while Fairchild makes the μA741. These parts are generally considered to be interchangeable, although they may vary in some ways. Some manufacturers will use the prefix code of the original developer of a part, and reserve their prefix for their own designs. As an example, Signetics produces their version of the 741, that they call μA741 because this op amp was first developed by Fairchild. (Signetics is then referred to as a second source for the μA741).

Suffix codes vary widely between manufacturers. Typical designations for consumer grade parts are C, and CN. The suffix N often means Not Graded. Interestingly, the lack of a final suffix often indicates a very high quality part, usually with an extended temperature range. Suffix codes are also used to indicate package styles. This practice is particularly popular among voltage regulators and other high-power linear ICs.

Finally, some manufacturers will use a “parallel” numbering system for high-grade parts. For example, the commercial grade device may have a “300 series” part number, with industrial grade given a “200 series” designation, and a military grade part given a “100 series” number. One possible example is the LM318 commercial grade op amp versus its high-grade counterpart, the LM118. Generally, militaryspecified parts will have a very wide temperature range, with industrial and commercial grades offering progressively less.

The data sheet for the LF411 op amp is shown in Figure 7.8.2.8. Let's take a look at some of the basic parameters and descriptions. The values given are typical of a modern op amp. A complete investigation of all parameters will be given in Chapter Five, once we've gotten a little more familiar with the device.



SNOSBH6E – APRIL 1999 – REVISED OCTOBER 2014 www.ti.com

6 Specifications

6.1 Absolute Maximum Ratings(1)

	LF411A		LF411		UNIT
	MIN	MAX	MIN	MAX	
Supply Voltage		±22		±18	V
Differential Input Voltage		±38		±30	V
Input Voltage Range(2)		±19		±15	V
Output Short Circuit Duration		Continuous		Continuous	

	TO Package		PDIP Package		UNIT
	MIN	MAX	MIN	MAX	
Power Dissipation(3) (4)		670		670	mW
$T_j$max		150		115	°C
Operating Temperature		See (5)		See (5)	
Lead Temperature (Soldering, 10 s)		260		260	°C

(1) “Absolute Maximum Ratings” indicate limits beyond which damage to the device may occur. Operating Ratings indicate conditions for which the device is functional, but do not ensure specific performance limits.
(2) Unless otherwise specified the absolute maximum negative input voltage is equal to the negative power supply voltage.
(3) For operating at elevated temperature, these devices must be derated based on a thermal resistance of θ_{JA}.
(4) Max. Power Dissipation is defined by the package characteristics. Operating the part near the Max. Power Dissipation may cause the part to operate outside specified limits.
(5) These devices are available in both the commercial temperature range $0°C \le T_A \le 70°C$ and the military temperature range $-55°C \le T_A \le 125°C$. The temperature range is designated by the position just before the package type in the device number. A “C” indicates the commercial temperature range and an “M” indicates the military temperature range. The military temperature range is available in the TO package only.

6.2 Handling Ratings

		MIN	MAX	UNIT
T_{stg}	Storage temperature range	−65	150	°C

6.3 Recommended Operating Conditions

over operating free-air temperature range (unless otherwise noted)

	MIN	NOM	MAX	UNIT
Supply Voltage LF411A			±20	V
Supply Voltage LF411			±15	V

6.4 Thermal Information

THERMAL METRIC(1)		TO	PDIP	UNIT
		8 PINS	8 PINS	
$R_{\theta JA}$ Junction-to-ambient thermal resistance	Still Air	162		°C/W
	400 LF/min Air Flow	65		
$R_{\theta JA}$ Junction-to-ambient thermal resistance			120	
$R_{\theta JC}$ Junction-to-case (top) thermal resistance		20		

(1) For more information about traditional and new thermal metrics, see the *IC Package Thermal Metrics* application report, SPRA953.

Figure 7.8.2.8: Data sheet for the LF411. Reprinted courtesy of Texas Instrutments

6.5 DC Electrical Characteristics (1)(2)

PARAMETER		TEST CONDITIONS		LF411A			LF411			UNIT
				MIN	TYP	MAX	MIN	TYP	MAX	
V_{OS}	Input Offset Voltage	R_S=10 kΩ, T_A=25°C			0.3	0.5		0.8	2.0	mV
$\Delta V_{OS}/\Delta T$	Average TC of Input Offset Voltage	R_S=10 kΩ			7			7		µV/°C
I_{OS}	Input Offset Current	V_S=±15V (2) (3)	T_j=25°C		25	100		25	100	pA
			T_j=70°C			2			2	nA
			T_j=125°C			25			25	nA
I_B	Input Bias Current	V_S=±15V (2) (3)	T_j=25°C		50	200		50	200	pA
			T_j=70°C			4			4	nA
			T_j=125°C			50			50	nA
R_{IN}	Input Resistance	T_j=25°C			10^{12}			10^{12}		Ω
A_{VOL}	Large Signal Voltage Gain	V_S=±15V, V_O=±10V, R_L=2k, T_A=25°C		50	200		25	200		V/mV
		Over Temperature		25	200		15	200		V/mV
V_O	Output Voltage Swing	V_S=±15V, R_L=10k		±12	±13.5		±12	±13.5		V
V_{CM}	Input Common-Mode Voltage Range			±16	+19.5		±11	+14.5		V
					−16.5			−11.5		V
CMRR	Common-Mode Rejection Ratio	R_S≤10k		80	100		70	100		dB
PSRR	Supply Voltage Rejection Ratio	See (4)		80	100		70	100		dB
I_S	Supply Current				1.8	2.8		1.8	3.4	mA

(1) RETS 411X for LF411MH and LF411MJ military specifications.
(2) Unless otherwise specified, the specifications apply over the full temperature range and for V_S=±20V for the LF411A and for V_S=±15V for the LF411. V_{OS}, I_B, and I_{OS} are measured at V_{CM}=0.
(3) The input bias currents are junction leakage currents which approximately double for every 10°C increase in the junction temperature, T_j. Due to limited production test time, the input bias currents measured are correlated to junction temperature. In normal operation the junction temperature rises above the ambient temperature as a result of internal power dissipation, P_D. T_j=T_A+θ_{jA} P_D where θ_{jA} is the thermal resistance from junction to ambient. Use of a heat sink is recommended if input bias current is to be kept to a minimum.
(4) Supply voltage rejection ratio is measured for both supply magnitudes increasing or decreasing simultaneously in accordance with common practice, from ±15V to ±5V for the LF411 and from ±20V to ±5V for the LF411A.

6.6 AC Electrical Characteristics

PARAMETER(1)(2)		TEST CONDITIONS	LF411A			LF411			UNIT
			MIN	TYP	MAX	MIN	TYP	MAX	
SR	Slew Rate	V_S=±15V, T_A=25°C	10	15		8	15		V/µs
GBW	Gain-Bandwidth Product	V_S=±15V, T_A=25°C	3	4		2.7	4		MHz
e_n	Equivalent Input Noise Voltage	T_A=25°C, R_S=100Ω, f=1 kHz		25			25		nV / √Hz
i_n	Equivalent Input Noise Current	T_A=25°C, f=1 kHz		0.01			0.01		pA / √Hz
THD	Total Harmonic Distortion	A_V=+10, R_L=10k, V_O=20 Vp-p, BW=20 Hz–20 kHz		<0.02 %			<0.02 %		

(1) Unless otherwise specified, the specifications apply over the full temperature range and for V_S=±20V for the LF411A and for V_S=±15V for the LF411. V_{OS}, I_B, and I_{OS} are measured at V_{CM}=0.
(2) RETS 411X for LF411MH and LF411MJ military specifications.

Figure 7.8.2.8 (cont.): Data sheet for the LF411. Reprinted courtesy of Texas Instrutments

First, note that two versions of the IC are given. We will examine the LF411 rather than the high-grade LF411A. At the very top of the data sheet is a listing of the absolute maximum ratings. The op amp should never operate at values greater than those presented as doing so may permanently damage it. Like most general-purpose op amps, the LF411 is powered by a bipolar power supply. The supply rails should never exceed $\pm$18 V DC. Normally, op amps will be used with $\pm$15 V supplies. Maximum power dissipation is given as 670 mW. Obviously then, this is a small signal device. In keeping with this, the operating temperature range and maximum junction temperatures are relatively low. We also see that the device can withstand differential input signals of up to 30 V, and single-ended inputs of up to 15 V, without damage. On the output, the LF411 is capable of withstanding a shorted load condition continuously. This makes the op amp a bit more "bullet-proof". The remainder of this section details soldering conditions. Excessive heat during soldering may damage the device.

The second section of the data sheet lists the DC characteristics of the op amp. This table is broken down into five major sections:

- The parameter symbol.
- The parameter name.
- The conditions under which the parameter is measured.
- The parameter values, either typical or min/max.
- The parameter units.

We will examine a few of these parameters right now. The fourth parameter given is I_B, the input bias current. I_B is the current drawn by the bases (or gates) of the input differential amplifier stage. Because the LF411 utilizes a JFET diff amp, we expect this value to be rather small. For an operating temperature of 25°C, a typical LF411 will draw 50 pA, and a worst-case LF411 no more than 200 pA. If we extend the temperature range out a bit, I_B can extend out to 50 nA. This is a sizable jump, but even 50 nA is a very small value for general-purpose work. Because larger bias currents are normally seen as undesirable, the maximum I_B is the worst case scenario, hence a minimum I_B is not reported. Along with this, we see a very high value for input resistance, some $10^{12}\Omega$, typically. Op amps utilizing bipolar input devices will show much higher values for I_B, and much lower values for R_{in}.

Next in line comes A_{vol}. This is the DC voltage gain. Note the test conditions. The power supply is set to $\pm$15 V, the load is 2 kΩ, and V_{out} is 10 V peak. Normally, we desire as much gain as possible, so the worst case scenario is the minimum A_{vol}. For 25°C operation this is specified as 25 V/mV, or 25,000. The average device will produce a gain of 200,000. As is typical, once the temperature range is expanded performance degrades. Over the operating temperature range, the minimum gain may drop to 15,000.

Because the op amp uses a class AB follower for its output stage, we should expect the output compliance to be very close to the power supply rails. The output voltage swing is specified for $\pm$15 V supplies with a 10 kΩ load. The typical device can swing out to $\pm$13.5 V, with a worst case swing of $\pm$12 V. A reduction in power supply value will naturally cause the maximum output swing to drop. A sizable reduction in the load resistance will also cause a drop in V_o, as we will see a bit later. These maximum output values are caused by the internal stages reaching their saturation limits. When this happens, the op amp is said to be clipping or in saturation. As a general rule of thumb, saturation may be approximated as 1.5 V less than the magnitude of the power supplies.

The last item in the list is the standby current draw, I_S. Note how small this is, only 1.8 mA, 3.4 mA worst case. This is the current the op amp draws from the supply under no signal conditions. When producing output signals, the current draw will rise.

The final section of the data sheet lists certain AC characteristics of the op amp that will be of great concern to us in later sections. Many device parameters change a great deal with frequency, temperature, supply voltage, or other factors. Because of this, data sheets also include a large number of graphs that further detail the op amp's performance. Finally, application hints and typical circuits may round out the basic data sheet.

References

[1]While the exact internal circuitry may be altered, the various manufacturers versions will have the same pinouts, and very similar performance specifications.

[2]A complete derivation of the Widlar current source may be found in Principles of Electronic Circuits, by S.G.Burns and P.R.Bond, 1987, West Publishing Company.

This page titled 7.8.2: What Is An Op Amp? is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.8.3: Simple Op Amp Comparator

Now that you have a feel for what an op amp is and what some typical parameters are, let's take a look at an application. The one thing that jumps to most people's attention is the very high gain of the average op amp. The typical LF411 showed A_{vol} at approximately 200,000. With gains this high, it is obvious that even very small input signals can force the output into saturation (clipping). Take a look at Figure 7.8.3.1. Here an op amp is being supplied by ±15 V, and is driving a 10 kΩ load. As seen in our model of Figure 2.2.5, V_{out} should equal the differential input voltage times the op amp's gain, A_{vol}.

$$V_{out} = A_{vol}(V_{in+} - V_{in-})$$

$$V_{out} = 200,000 \times (0.1\ V - 0\ V)$$

$$V_{out} = 20,000\ V$$

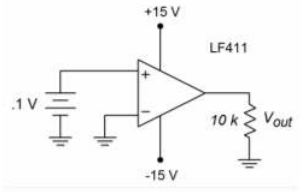


Figure 7.8.3.1: Comparator (single input).

The op amp cannot produce 20,000 V. The data sheet lists a maximum output swing of only ±13.5 V when using ±15 V supplies. The output will be truncated at 13.5 V. If the input signal is reduced to only 1 mV, the output will still be clipped at 13.5 V. This holds true even if we apply a signal to the inverting input, as in Figure 7.8.3.2.

$$V_{out} = A_{vol}(V_{in+} - V_{in-})$$

$$V_{out} = 200,000 \times (0.5\ V - 0.3\ V)$$

$$V_{out} = 40,000\ V$$

$$V_{out} = 13.5\ V, \text{due to clipping}$$

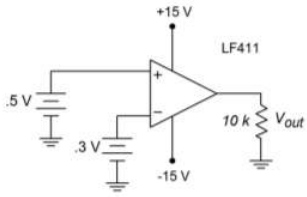


Figure 7.8.3.2: Comparator (dual input).

Computer Simulation

A simulation of Figure 7.8.3.2 using Multisim is shown in Figure 7.8.3.3. The LF411 op amp is selected from the component library, and you need not concern yourself with the model's internal make-up just yet. This particular model includes the effects of power supply limitations (i.e., output saturation) that the very simple dependent-source model presented earlier does not. Individual DC sources are used for the input signals. Although no AC signals are applied, it is perfectly valid to run a transient-mode simulation. The first millisecond of the output voltage is shown. It verifies the manual calculation, indicating a DC level of slightly more than 13.5 V.

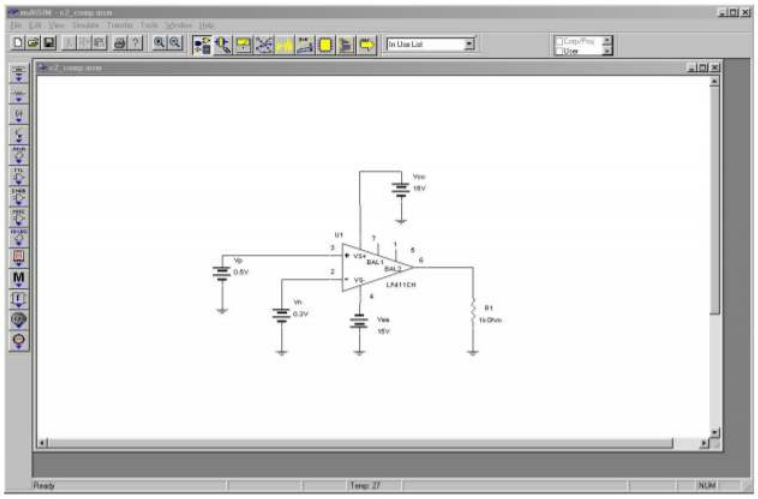


Figure 7.8.3.3a: Comparator example in Multisim.

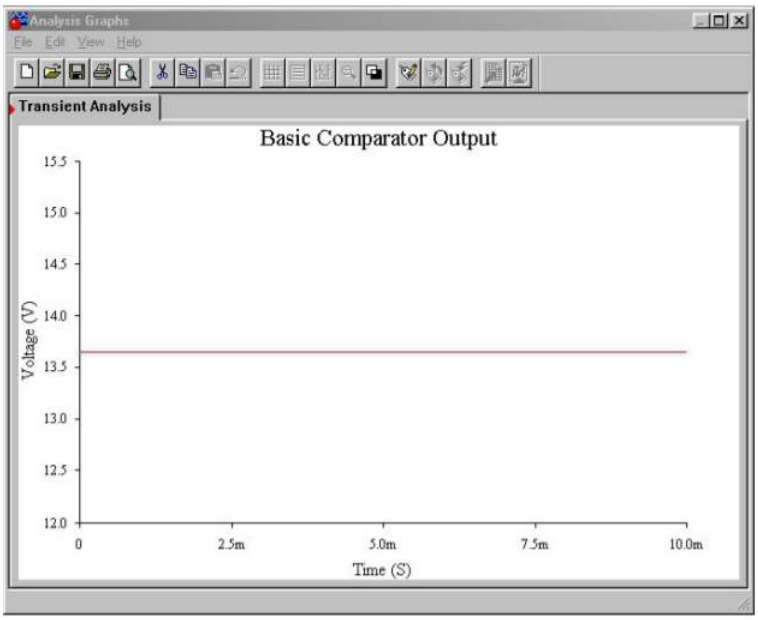


Figure 7.8.3.3b: Comparator output.

For any reasonable set of inputs, as long as the noninverting signal is greater than the inverting signal, the output will be positive saturation. If you trade the input signals so that the inverting signal is the larger, the converse will be true. As long as the inverting signal is greater than the noninverting signal, the output will be negative saturation. If the inverting and noninverting signals are identical, V_{out} should be 0 V. In the real world, this will not happen. Due to minute discrepancies and offsets in the diff amp stage, either positive or negative saturation will result. You have no quick way of knowing in which direction it will go. It is for this reason that it is impractical to amplify a very small signal, say around 10 μV. You might then wonder, “What is the use of this amplifier if it always clips? How can I get it to amplify a simple signal?” Well, for normal amplification uses, we will have to add on some extra components, and through the use of negative feedback (next chapter), we will create some very well-controlled, useful amplifiers. This is not to say that our barren op amp circuit is useless. Quite to the contrary, we have just created a comparator.

A comparator has two output states: High and low. In other words, it is a digital, logical output. Our comparator has a high state potential of 13.5 V, and a low state potential of -13.5 V. The input signals, in contrast, are continuously variable analog potentials. A comparator, then, is an interface between analog and digital circuitry. One input will be considered the reference, while the other

input will be considered the sensing line. Note that the differential input signal is the difference between the sensing input and the reference input. When the polarity of the differential input signal changes, the logical output of the comparator changes state.

Example 7.8.3.1

Figure 7.8.3.4 shows a light leakage detector that might be used in a photographer's dark room. This circuit utilizes a Cadmium Sulfide (CdS) cell that is used as a light sensitive resistor. The inverting input of the op amp is being used as the reference input, with a 1 V DC level. The noninverting input is being used as the sensing input. Under normal (no light) conditions, the CdS cell acts as a very high resistance, perhaps 1 MΩ. Under these conditions, a voltage divider is set up with the 10 kΩ resistor, producing about 150 mV at the noninverting input. Remember, no loading of the divider occurs because the LF411 utilizes a JFET input. Because the noninverting input is less than the inverting input, the comparator's output is negative saturation, or approximately -13.5 V. If the ambient light level rises, the resistance of the CdS cell drops, thus raising the signal applied to the noninverting input. Eventually, if the light level is high enough, the noninverting input signal will exceed the 1 V reference, and the comparator's output will move to positive saturation, about +13.5 V. This signal could then be used to trigger some form of audible alarm. A real world circuit would need the flexibility of an adjustable reference in place of the fixed 1 V reference. By swapping the CdS cell and the 10 kΩ resistor, and adjusting the reference, an inverse circuit (i.e., an alarm that senses darkness) may be produced.

Circuits of this type can be used to sense a variety of over level/under level conditions, including temperature and pressure. All that is needed is an appropriate sensing device. Comparators can also be used with AC input signals.

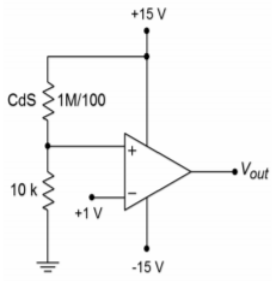


Figure 7.8.3.4: Light alarm.

Example 7.8.3.2

Sometimes, it is necessary to square up an AC signal for further processing. That is, we must turn it into an equivalent pulse waveform. One example of this might be a frequency counter. A frequency counter works by tallying the number of high-to-low or low-to-high transitions in the input signal over a specific length of time. For accurate counts, good edge transitions are required. Because a simple sine wave changes relatively slowly compared to an equal frequency square wave, some inaccuracy may creep into the readings. We can turn the input into a pulse-type output by running it through the comparator of Figure 7.8.3.5. Note that the reference signal is adjustable from −15 to +15 V. Normally, the reference is set for 0 V. Whenever the input is greater than the reference, the output will be positive saturation. When the input is less than the reference, the output will be negative saturation. By making the reference adjustable, we have control over the output duty cycle, and can also compensate for DC offsets on the input signal. A typical input/output signal set is given in Figure 7.8.3.6.

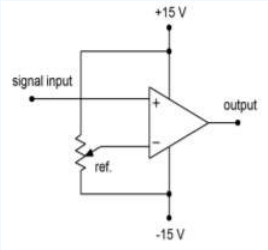


Figure 7.8.3.5: “Square-up” circuit.

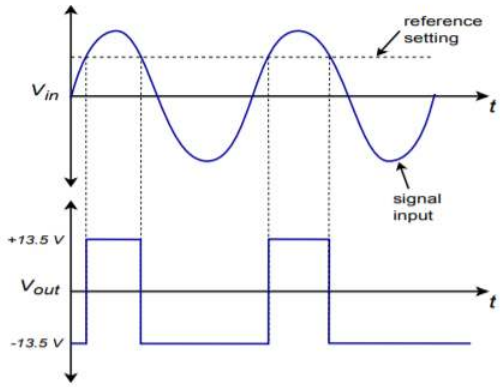


Figure 7.8.3.6: Output of “square-up” circuit.

There are a few limitations with our simple op amp comparator. For very fast signal changes, a typical op amp will not be able to accurately track its output. Also, the output signal range is rather wide and is bipolar. It is not at all compatible with normal TTL logic circuits. Extra limiting circuitry is required for proper interfacing. To help reduce these problems, a number of circuits have been specially optimized for comparator purposes. We will take a closer look at a few of them in Chapter Seven.

This page titled 7.8.3: Simple Op Amp Comparator is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.8.4: Op Amp Manufacture

Op amps and other linear integrated circuits are generally manufactured in one of two ways: The device is either a hybrid, or is monolithic. In either case, the circuit can contain hundreds of components. The resulting device will be packaged in a variety of styles, including plastic and ceramic dual in-line and single in-line types, multi-lead cans, flat packs, and surface mount forms. Some examples are shown in Figure 7.8.4.1. In each type, the circuitry is completely encased and not accessible to the designer or technician. If one of the components should fail, the entire op amp is replaced. The design and layout of the integrated circuit itself is normally carried out with the use of special computer workstations and software tools. These allow the designers to simulate portions of the circuit, and create the outlines and interconnections for the various components to be formed.

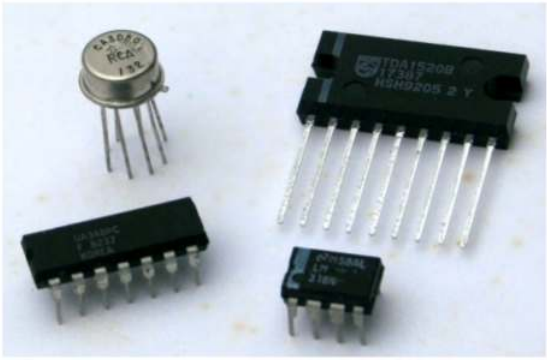


Figure 7.8.4.1: Package styles (through hole) clockwise from top-left: can, single in-line, mini dual in-line, dual in-line.

2.4.1: Monolithic Construction

The term monolithic is from the Greek, meaning literally “single stone”. In this process, all circuit elements are created and interconnected using a single slab of silicon (or other suitable material). Normally, several op amps are made from a single silicon wafer. Each wafer may be a few inches in diameter, with each op amp circuit chip comprising perhaps a square 1 millimeter by 1 millimeter in area. A single transistor can easily be smaller than 15 micrometers by 20 micrometers. Because the scale of construction is so small, special clean rooms are required in order to remove tiny air-borne particles of dust and grit that could interfere with the production of these super-small components. Workers in clean rooms are required to wear special suits as well.

Figure 7.8.4.2 outlines the major steps in the chip manufacturing process. This process starts with the preparation of a P-type silicon wafer. This is referred to as the substrate. After it has been cleaned and polished, an N-type epitaxial region is diffused into the P-type base. Epitaxial is from the Greek, roughly meaning “to arrange upon”. It is within this thin epitaxial region that the circuit elements will be formed, the remainder of the substrate lending mechanical support to the structure. The term diffusion refers to the manner in which the semiconductor material becomes doped. In essence, the base material is surrounded by a high concentration of doping material, usually gaseous, along with the application of heat. The low concentration wafer material will be infiltrated by the high concentration doping material. Diffusion is a relatively accurate and inexpensive means of controlling the semiconductor's properties.

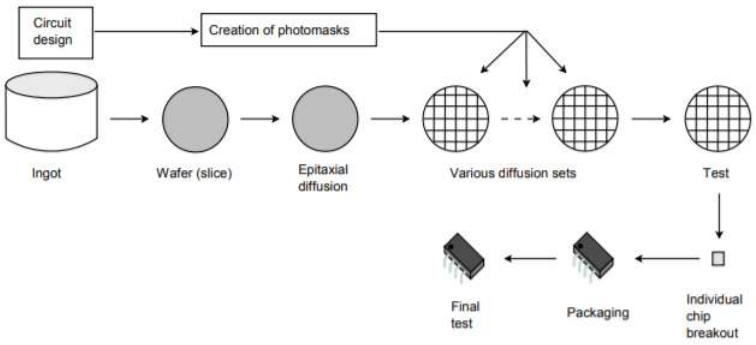


Figure 7.8.4.2: Chip manufacturing process.

Once the N-type region is produced, the wafer will undergo an oxidizing process that will leave the top surface covered with silicon dioxide. This layer prevents impurities from entering the N-type region. At this point, a series of steps will be used to create wells or deposits of alternate p and n material. These deposits will form the various active and passive components. Normally, this is done through a photolithographic process. This involves the use of light sensitive materials and masks. Conceptually, the process is not very much different from the way printed circuit boards are often made. In essence, specific areas of the silicon dioxide layer will be stripped away, thus exposing the epitaxial region, and allowing diffusion of other acceptor/donor impurities to take place. Because the silicon dioxide serves as an effective barrier to diffusion, only areas cleared of silicon dioxide will be effected by the diffusion process. In this manner, specific areas can be singled out, and selectively doped to create specific components. This is detailed below and in Figure 7.8.4.3.

In order to selectively remove the silicon dioxide, the top surface is coated with a light sensitive material called photoresist. Above this is placed a mask. This mask is much like a black and white negative; some areas are clear, and some areas are opaque. The resulting sandwich is then exposed to ultraviolet light. The clear areas of the mask will allow the light to pass through and cause a chemical change in the photoresist. A solution is then used to wash away the unexposed photoresist. At this point, a second solution is used to wash away the silicon dioxide. This solution will not effect the exposed photoresist, and thus, the silicon dioxide beneath it is unaffected. After this protecting layer of photoresist is removed, all that remains on the top surface of the wafer are alternating patches of silicon dioxide. The wafer can now be lead through another diffusion process.

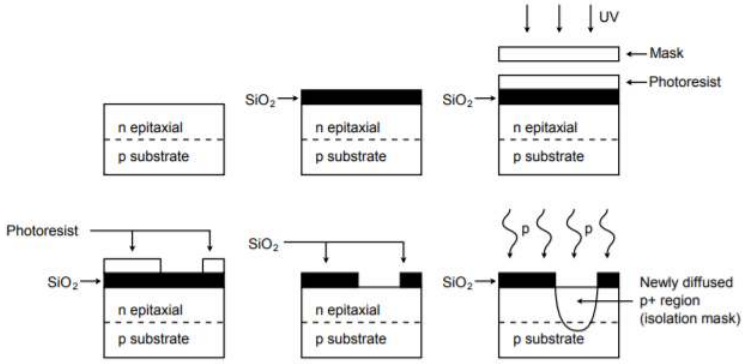


Figure 7.8.4.3: Diffusion process (one run).

The process of oxidizing, masking, and diffusing will be repeated several times. The initial run will be produced with an isolation mask. This is used to separate the various components. Normally, a base mask will be used next, followed by the emitter mask. The final masks will be used for contacts and interconnections. In this way, N-type material can be placed next to, or completely within, P-type material. The adjoining areas are, of course, PN junctions. Because all circuit elements are laid out length-wise on a thin strip, this form of manufacture is referred to as a planar process.

Once the final mask is completed, the wafer will be inspected. The individual chips will then be broken out of the wafer and mounted into the desired package. Leads will be connected to the chip with fine angel hair wire, and then the package will be sealed. It is now ready for final test and inspection. Part numbers and date codes will also be imprinted.

Virtually all general-purpose op amps today use a planar monolithic process. Some of the advantages of monolithic construction are its relative simplicity and low per-part cost.

2.4.2: Hybrid Construction

Hybrids are usually used where a complete monolithic solution is impractical. This is usually the case for special purpose devices, such as those requiring very high output current, very wide bandwidth, or that are very complex or sensitive. Hybrids, as the name suggests, are a collection of smaller circuit elements interconnected. A typical hybrid may contain two or three smaller monolithic chips and assorted miniaturized passive and/or power components. Passive components may be further integrated by using either a thin or thick film chip process. (A discussion of thin and thick film chip techniques is beyond the scope of this text). Due to the complexity of a hybrid chip, it is normally more expensive than its monolithic cousins. Although the IC itself may be more expensive, the complete application may very well wind up being less costly to produce because the cost of other components are

effectively absorbed within the hybrid IC. One place where hybrids are often used is in consumer stereo music systems. A hybrid power amplifier IC offers the convenience of a single IC solution with the capabilities of a discrete transistor approach. As an op amp user, it makes little difference whether the device is hybrid or monolithic when it comes to circuit analysis or design.

This page titled 7.8.4: Op Amp Manufacture is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.8.5: Summary

Op amps are presently in wide use in just about every aspect of linear electronics. An op amp is a multi-stage amplifier treated as a single entity. The first stage usually utilizes a differential amplifier that can be made with either bipolar or FET devices. The following stage(s) create a large voltage gain. The final stage is a class B voltage follower. The resulting op amp typically has a high input impedance, a low output impedance, and voltage gains in excess of 10,000. The op amp operates from a bipolar power supply, usually around $\pm$15 V. Externally, it has connections for the inverting and noninverting inputs, the single-ended output, and the power supplies. The op amp may be packaged in a variety of forms, including DIPs, SIPs, cans, flat packs and surface mount.

The general-purpose op amp is manufactured using a monolithic structure and a photolithographic process. Several chips are created from a single master wafer. Creation is a multi-step process involving the selective doping of specific areas on the chip through diffusion. The monolithic technique is relatively inexpensive and accurate. Because integration allows for very tight part matching and consistency, certain circuit design techniques are favored, including the use of current mirrors and active loads.

Finally, with very little supporting circuitry, simple op amps can make effective comparator circuits. A comparator is in essence, a bridge between the analog and digital worlds.

This page titled 7.8.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.8.6: Problems

Review Questions

1. What is an op amp?

2. Give several examples of where op amps might be used.

3. What is the typical stage layout of an op amp?

4. What comprises the first stage of a typical op amp?

5. What comprises the final stage of a typical op amp?

6. How does integrated circuit design differ from discrete design?

7. What is a comparator, and how might it be used?

8. What is meant by monolithic planar construction?

9. What is a mask, and how is it used in the construction of IC op amps?

10. What is the process of diffusion and how does it relate to the construction of IC op amps?

11. What are the advantages of monolithic IC construction?

12. How does hybrid construction differ from monolithic construction?

Problems

1. For the circuit in Figure 7.8.6.1, find V_{out} for the following inputs:

a) 0 V

b) -1 V

c) +2 V

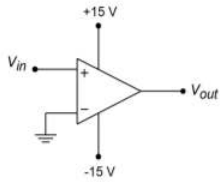


Figure 7.8.6.1

2. For the circuit in Figure 7.8.6.2, find V_{out} for the following inputs:

a) 0 V

b) -4 V

c) +5 V

d) -0.5 V

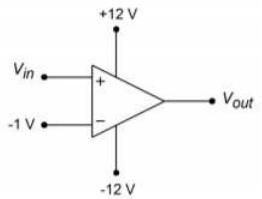


Figure 7.8.6.2

3. For the circuit in Figure 7.8.6.3, find V_{out} for the following inputs:

a) 0 V

b) -2 V

c) +1 V

d) -0.5 V

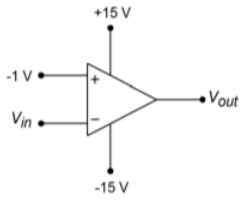


Figure 7.8.6.3

4. Sketch V_{out} if $V_{in}(t) = 1 \sin 60t$ in Figure 7.8.6.1.

5. Sketch V_{out} if $V_{in}(t) = 2 \sin 20t$ in Figure 7.8.6.2.

6. Sketch V_{out} if $V_{in}(t) = 3 \sin 10t$ in Figure 7.8.6.3.

7. A Temperature Dependent Resistor is used in the comparator of Figure 7.8.6.4. At what temperature will the comparator change state?

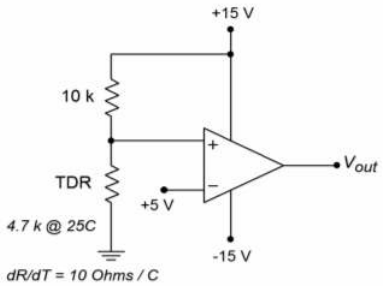


Figure 7.8.6.4

8. What is the value of the Light Dependent Resistor at the comparator trip point in Figure 7.8.6.5?

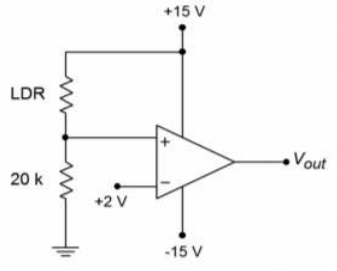


Figure 7.8.6.5

9. What reference voltage is required for a 50°C trip point in Figure 7.8.6.4?

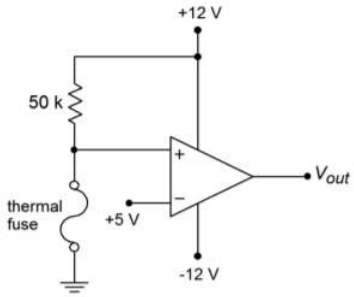


Figure 7.8.6.6

10. A thermal fuse is device found in such common items as coffee makers. Normally, its resistance is very low, ideally 0 Ω. With the application of excessive heat, the fuse opens, presenting a very high resistance, ideally infinite. Explain the operation of the circuit in Figure 7.8.6.6. Is the output voltage normally high or low?

11. A strain gauge is a device that can be used to measure the amount of bend or deflection in a part that it is attached to. It can be connected so that its resistance rises as the bend increases. What resistance is required to trip the comparator of Figure 7.8.6.7?

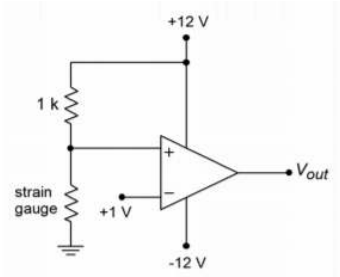


Figure 7.8.6.7

12. Determine the current flowing out of the collector of Q_{13} in Figure 2.2.4.

Computer Simulation Problem

13. Alter the simple simulation model presented in the chapter to include saturation effects. (Hint: Consider a device that limits voltage.)

This page titled 7.8.6: Problems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

SECTION OVERVIEW

7.9: Negative Feedback

Learning Objectives

After completing this chapter, you should be able to:

- Give examples of how negative feedback is used in everyday life.
- Discuss the four basic feedback connections, detailing their similarities and differences.
- Detail which circuit parameters negative feedback will alter, and how.
- Discuss which circuit parameters are not altered by negative feedback.
- Define the terms sacrifice factor, gain margin, and phase margin, and relate them to a Bode plot.
- Discuss in general, the limits of negative feedback in practical amplifiers.

7.9.1: Introduction to Negative Feedback

7.9.2: What Negative Feedback Is and Why We Use It

7.9.3: Basic Concepts

7.9.4: The Four Variants of Negative Feedback

7.9.5: Limitations On The Use of Negative Feedback

7.9.6: Summary

7.9.7: Problems

Template:HideTOC

This page titled 7.9: Negative Feedback is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.9.1: Introduction to Negative Feedback

As we saw in the last chapter, op amps are very useful devices. However, in many applications, the device's gain is simply too large and its bandwidth too narrow, for effective use. In this chapter we will explore the concept of negative feedback. This concept is realized by feeding a portion of the output signal back to the input of the system. The proper use of negative feedback will allow us to exercise fine control over the performance of electronic circuits. As a matter of fact, negative feedback is so useful to us that we will seldom use op amps without it. Negative feedback is not tied solely to op amps though, as almost any electronic circuit may benefit from its application. As with most things, there are disadvantages as well. A successful design will minimize the disadvantages and capitalize on the positive aspects. We will begin with the basic concepts of what negative feedback is and does, and then fine-tune our viewing by examining its four specific variants. We will look at specific examples of how negative feedback is applied to op amps, and finish off with a discussion of its practical limits.

This page titled 7.9.1: Introduction to Negative Feedback is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.1: Introduction to Negative Feedback** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.9.2: What Negative Feedback Is and Why We Use It

People use negative feedback every day of their lives. In fact, we probably couldn't get along without it. Simply put, negative feedback is a very rudimentary part of intelligence. In essence, negative feedback lets something correct for mistakes. It tends to stabilize operations and reduce change. Negative feedback relies on a loop concept. In human terms, it is akin to knowing what you are doing and being able to correct for mistakes as they happen. You are constantly evaluating and correcting your actions in order to achieve a desired goal. This may be stated as letting the input know what the output is doing. A good example of this is your ability to maintain a constant speed while driving along the highway. You have a desired result, or set-point, in mind, say 60 MPH. As you drive, you constantly monitor the speedometer. If you glance down and see that you're zipping along at 70 MPH, you think “Oops, I'm going a bit too fast” and lift your foot slightly off of the gas pedal. On the other hand, if you're only going 40 MPH, you will depress the gas pedal further. The faster and more accurate your updates are, the better you will be at maintaining an exact speed.

In contrast to negative feedback is positive feedback, which reinforces change. If you were to correct your speed by saying “Hmm, I'm going 70 MPH, I'd better step on the gas”, you'd be using positive feedback. Other examples of positive feedback include the “acoustic squeal” often heard over public address systems, and thermal runaway effects seen in discrete devices. When positive feedback is applied to normal amplifiers, they oscillate. That is, they produce their own signals without any input applied.

This page titled 7.9.2: What Negative Feedback Is and Why We Use It is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.2: What Negative Feedback Is and Why We Use It** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.9.3: Basic Concepts

Seeing the usefulness of negative feedback, it would be nice if we could apply the concept to our electronic circuits. The basic idea is quite simple, really. What we will do is sample a piece of the output signal, and then add it to the input signal out of phase (i.e., subtract it). By doing so, the circuit will see the difference between the input and the output. If the output signal is too large, the difference will be negative. Conversely, the difference signal will be positive if the output is too small. This signal is then multiplied by the circuit gain and cancels the output error. Thus, the circuit will be presented with the undesired errors in a way that will force the output to compensate (move in the opposite direction). This process is done continuously; the only time lags involved are the propagation delays of the circuits used.

Because the sampled output signal is effectively subtracted from the input signal, negative feedback is sometimes referred to as degenerative or destructive feedback. This subtraction can be achieved in a variety of ways. A differential amplifier is tailor made for this task because it has one inverting input and one noninverting input. (Note: If the error is presented in phase, the circuit magnifies the errors and positive feedback results.)

To see an example of how this works, refer to Figure 7.9.3.1. The triangle represents an amplifying circuit. It has a gain of A. The output signal is also presented to the input of the feedback network represented by the box. This network scales the output signal by a factor, β. The feedback network ranges from very simple to complex. It may contain several resistors, capacitors, diodes and what not, or it may be as simple as a single piece of wire. In any case, this scaled output signal is referred to as the feedback signal and is effectively subtracted from the input signal. This combination, called the error signal, is then fed to the amplifier where it is boosted and appears at the output. The process repeats like this forever (or at least until the power is switched off).

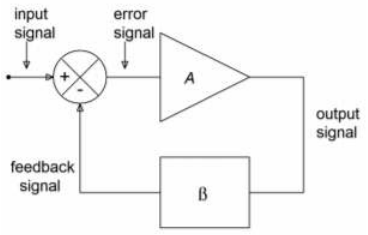


Figure 7.9.3.1: Negative feedback.

Let's assume that for some reason (perhaps a temperature change) the amplifier's gain were to rise. This should make the output signal increase by a similar percentage, but it doesn't. Here's why: As the output signal tries to rise, the feedback signal tracks with it. Now that there is a larger feedback signal, the error signal will become smaller (remember, error = input - feedback). This smaller signal is multiplied by the gain of the amplifier, thus producing a smaller output signal that almost completely offsets the original positive change. Note that if the output signal were too small, the error signal would increase, thus bringing the output back up to a normal level. When everything is working right, the feedback and input signals are almost the same size. (Actually, the feedback signal is somewhat smaller in magnitude.)

3.3.1: The Effects of Negative Feedback

Besides smoothing out gain anomalies, negative feedback can reduce the effect of device non-linearities, thus producing a reduction in static forms of distortion such as THD (Total Harmonic Distortion). Basically, these non-linearities can be viewed as a string of small gain errors. As such, they produce appropriate error signals and are compensated for in the above manner. Negative feedback can also increase the bandwidth of the system. It can increase the upper cutoff frequency f_2 and decrease the lower cutoff frequency f_1 (assuming the system has one). Also, we can exercise control over the input and output impedances of the circuit. It is possible to increase or decrease the impedances. As you might have guessed, we don't receive these benefits for nothing. The down side to negative feedback is that you lose gain. Effectively, you get to trade off gain for an increase in bandwidth, a decrease in distortion, and control over impedances. The more gain you trade off, the greater your rewards in the other three areas. In the case of our op amp, this is a wise tradeoff because we already have more gain than we need for typical applications. This give-and-take is a very important idea, so remember “BIG D”. That stands for Bandwidth, Impedance, Gain, and Distortion

At this point, we need to define a few terms. **Closed loop** refers to the characteristics of the system when feedback exists. For example, closed-loop gain is the gain of the system with feedback, whereas closed-loop frequency response} refers to the new system break points. Generalized closed-loop quantities will be shown with the subscript "cl". Similarly, we will denote impedances, gain and the like for specific feedback variants with a two- letter subscript abbreviating the exact feedback configuration. One possibility for closed-loop gain would be A_{sp}. Open loop refers to the characteristics of the amplifier itself. To remember this, think of disconnecting or opening the path through the feedback network. Once the path is broken, the amplifier is on its own. **Open-loop** gain then, refers to the gain of the amplifier by itself, with no feedback. All open-loop quantities will be shown with the subscript "ol". The symbol for open-loop gain would be A_{ol}. The term loop gain refers to the ratio between the open- and closed-loop gains (it may also be computed from their difference in decibels on a Bode plot). Loop gain indicates how much gain we have given up or sacrificed in order to enhance the operation of the system. Consequently, loop gain is often called sacrifice factor, and given the symbol S. Generally, tradeoffs are proportional to the sacrifice factor. For example, if we cut the gain in half, we will generally double the bandwidth, and halve the distortion. An example that illustrates this can be seen in Figure 7.9.3.2. Note how the loop gain decreases with increasing frequency. This means that the effects of feedback at higher frequencies are not as great.

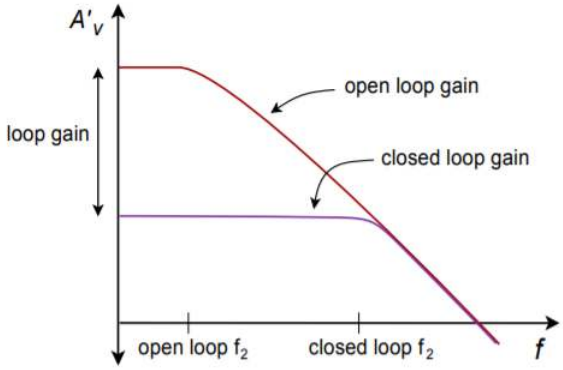


Figure 7.9.3.2: Response with and without feedback.

Up to this point we have made one simple assumption about our system; that it exhibits no “extra” phase change beyond the desired inversion. As we saw in the first chapter, however, all circuits do produce phase changes as the input frequency is increased. If this extra phase change were to reach -180° while the gain was greater than unity (0dB), our negative feedback will turn into positive feedback (the inversion = -180°, plus this extra -180°, places us at -360°. The net result is an inphase signal). If this were to happen, our amplifier would no longer be stable. In fact, it may very well turn into a high frequency oscillator. (You will see how to do this on purpose when you cover Chapter Nine). As the input frequency is raised, the phase will eventually exceed -180° and the gain will drop to a fraction (< 0dB). The real key here is making sure that the phase never reaches or exceeds -180° when the gain falls to 1. Stated another way, when the phase hits -180°, the gain should be a fraction. Generally, the farther you are from this “danger zone”, the better. In other words, you're better off if the extra phase at the unity gain point is -90° rather than -170°. Both are stable, but the first one gives you some breathing room.

Two measures of circuit safety are the gain and phase margins. Phase margin indicates the difference between the actual phase at unity gain and -180°. In the example above the first circuit would have a phase margin of 90° and the second would have a margin of 10°. Gain margin is the difference between the actual gain in dB at the -180° phase point, and 0dB. If our gain was -9 dB at -180 °, the gain margin would be 9 dB (i.e., we have 9 dB “to spare”). Reasonable values for gain and phase margin are > 6 dB and > 45 °. Gain and phase margins are depicted in Figure 7.9.3.3. It is possible to guarantee safe margins if the amplifier's open loop response maintains a 20 dB-per-decade rolloff up to the unity gain frequency, f_{unity}. This means that there is only one dominating lag network that will add a maximum phase shift of -90°. Even if the second network coincided with f_{unity}, it would add -45° at most. This would still leave us with a 45° phase margin. (Note that if we had several secondary networks critical at f_{unity} the phase could exceed -180°, however the slope would no longer be 20 dB-per-decade in reality.) It is for this reason that the general-purpose op amps examined in Chapter Two included a compensating capacitor. No matter how much feedback we wish to use, our circuits will always end up being stable. For the best circuit performance, it is possible to use amplifiers that do not have the

"constant rolloff" characteristic. The possibility exists that they may go into oscillation or become unstable if you are not careful and ignore the margins.

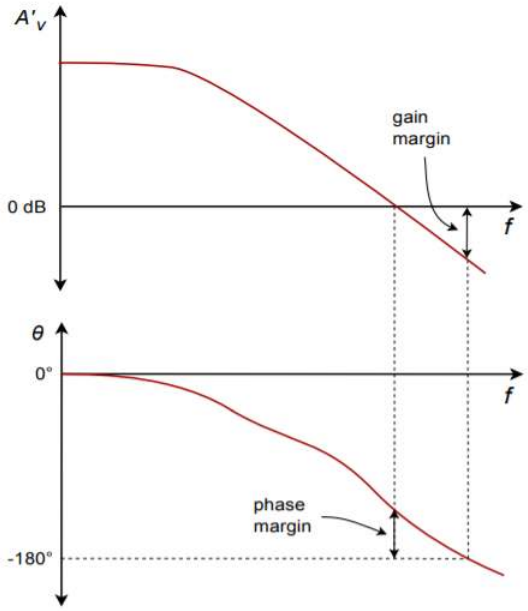


Figure 7.9.3.3: Gain and phase margin graphically determined from Bode plot.

This page titled 7.9.3: Basic Concepts is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.3: Basic Concepts** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.9.4: The Four Variants of Negative Feedback

Negative feedback can be achieved via four different forms. They differ in how the input and output impedances are changed. We have basically two choices when it comes to connecting the input and output of the amplifier to the output and input of the feedback network. We may produce either a series connection or a parallel connection. This yields four possibilities total. Each connection will produce a specific effect on the input or output impedance of the system. As you might guess, parallel connections decrease the impedance and series connections increase it. A high input impedance is desirable for maximum voltage transfer, whereas a low impedance is required for maximum current transfer. As a memory aid, think of volt meters and ammeters. For the smallest loading effect, volt meters should exhibit a high impedance and ammeters a very low one. In the case of the output connection, a low source impedance is required for the best voltage transfer, and a high source impedance for current transfer. You should think of the ideal voltage source (zero Z_{out}) and the ideal current source (infinite Z_{out}) here. Consequently, if we were to connect our feedback network in series with the amplifier's input, and in parallel with its output, we would have an increase in Z_{in} and a decrease in Z_{out}. This means that our system would be very good at sensing an input voltage, and ideal for producing a voltage. We will have created a voltage-controlled voltage source (VCVS), the ideal voltage amplifier. So that you can get a good idea of the possibilities, all four types are summarized in the following table.

Type (in-out)	Z_{in}	Z_{out}	Model	Idealization	Transfer Ratio
Series-Parallel	High	Low	VCVS	Voltage Amplifier	V_{out}/V_{in} Voltage gain
Series-Series	High	High	VCCS	Voltage to Current Transducer	I_{out}/V_{in} Transconductance
Parallel-Parallel	Low	Low	CCVS	Current to Voltage Transducer	V_{out}/I_{in} Transresistance
Parallel-Series	Low	High	CCCS	Current Amplifier	I_{out}/I_{in} Current Gain

Generally speaking, the input and output impedances will be raised or lowered from the non-feedback value by the sacrifice factor. Note that by using the proper form of feedback, we can achieve any of the possible models. This greatly enhances our ability to deal with specific applications. Much work in our field relies on optimal voltage transfer, therefore Series-Parallel (SP) is often used. A variation on Parallel Parallel (PP) is also used frequently, as we shall see.

3.4.1: Series-Parallel (SP)

The SP connection makes for the ideal voltage amplifier. A generalized block diagram is shown in Figure 7.9.4.1. You can tell that it has a series input because there is no input current node. Contrast this with the output: Note that the op amp's output current splits into two paths, one through the load and the other into the feedback network. This output node clearly denotes the feedback's parallel output connection.

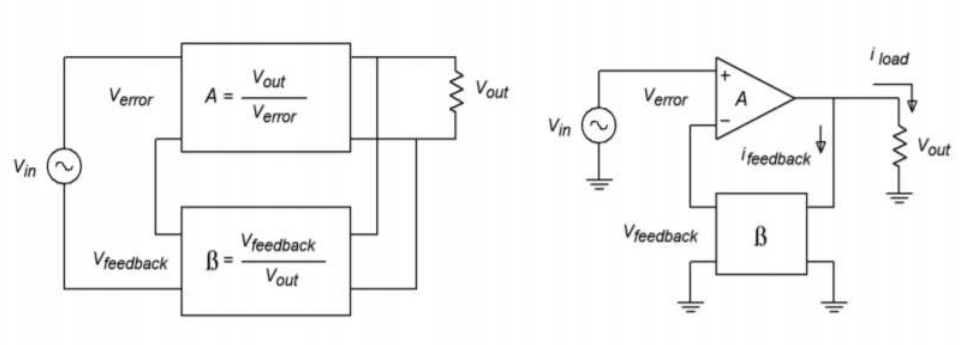


Figure 7.9.4.1: Series-parallel connection.

Let's take a look at exactly how SP negative feedback alters the system gain, impedances, and frequency response. For starters, let's examine the closed-loop gain (A_{sp}). Our amplifier block produces a gain A, and could be a diff amp, op amp, or other multi-stage

possibility. The feedback network is typically a voltage divider and produces a loss, β. The signal presented to the inverting input of the amp is the feedback signal and is equal to $V_{out}\beta$. Note that the source's signal, V_{in}, is applied to the non inverting input. Therefore, the differential input voltage (usually referred to as V_{error}), equals V_{in}–$V_{feedback}$. We also know from previous work with differential amplifiers that that $V_{out} = V_{error}A_{ol}$ (A_{ol} is the amplifiers open-loop gain). In other words we know:

$$V_{in} = V_{error} + V_{feedback} \tag{7.9.4.1}$$

$$V_{feedback} = V_{out}\beta \tag{7.9.4.2}$$

$$V_{out} = V_{error}A_{ol} \tag{7.9.4.3}$$

and by definition,

$$A_{sp} = \frac{V_{out}}{V_{in}} \tag{7.9.4.4}$$

Substituting Equation 7.9.4.3 into Equation 7.9.4.2,

$$V_{feedback} = V_{error}\beta A_{ol} \tag{7.9.4.5}$$

Substituting Equation 7.9.4.5 into Equation 7.9.4.1, and simplifying,

$$V_{in} = V_{error}(1 + \beta A_{ol}) \tag{7.9.4.6}$$

Finally, substituting Equation 7.9.4.6 and Equation 7.9.4.3 into Equation 7.9.4.4 and simplifying, yields,

$$A_{sp} = \frac{A_{ol}}{1 + \beta A_{ol}} \tag{7.9.4.7}$$

Since the fundamental definition of sacrifice factor, S, is A_{ol}/A_{cl}, we may also say $A_{sp} = A_{ol}/S$ and therefore, for SP,

$$S = 1 + \beta A_{ol} \tag{7.9.4.8}$$

Equation 7.9.4.7 is our general gain Equation but, if we can make $\beta A_{ol} \gg 1$ we may ignore the “+1” in the denominator and further simplify this as,

$$A_{sp} = \frac{1}{\beta} \tag{7.9.4.9}$$

This seemingly innocent Equation packs a rather hefty punch. What it is telling us is that the open-loop gain of the amplifier does not} play a role in setting the system gain, as long as the open-loop gain is very large. In other words, the system gain is controlled solely by the feedback network. Consequently, our amplifier can exhibit large gain changes in its open-loop response, but the closed-loop response will remain essentially constant. For this reason we will achieve identical closed-loop gains for op amps that exhibit sizable differences in their open-loop gains. Because signal distortion is produced by non-linearities that can be viewed as dynamic gain changes, our closed-loop distortion drops as well. Also, it is this very effect that extends our closed-loop frequency response. Imagine that our amplifier exhibits a gain of 10,000 at its upper break frequency of 100 Hz. If the feedback factor is equal to 0.1, our exact gain is:

$$A_{sp} = \frac{10,000}{1 + 0.1 \times 10,000} = 9.99$$

If we were to measure the amplifier's open-loop gain one decade up, at 1 kHz, it should be around 1,000 (assuming 20 dB/decade loss). The closed loop gain now equals:

$$A_{sp} = \frac{1,000}{1 + 0.1 \times 1,000} = 9.9$$

As you can see, the closed-loop gain changed only about 1 despite the fact that the open-loop gain dropped by a factor of 10. If we continue to raise the frequency, Asp would equal 9.09 at 10 kHz. Finally, at 100 kHz a sizable drop is seen because the gain falls to 5. At this point, our assumption of $\beta A_{ol} >> 1$ falls apart. Note however, that our loss relative to the midband gain is only a few dB. We have effectively stretched out the bandwidth of the system. Actually, this calculation is somewhat over-simplified as we have ignored the extra phase lag produced by the amplifier above the open-loop break frequency. If we assume that the open-loop

response is dominated by a single lag network (and it should be, in order to guarantee stability, remember?), a phase sensitive version of Equation 7.9.4.7 would be:

$$A_{sp} = \frac{-jA_{ol}}{1 - jA_{ol}\beta}$$

This extra phase will reach its maximum of -90 degrees approximately one decade above the open-loop break frequency. Consequently, when we find the magnitude of gain at 100 kHz, it's not

$$A_{sp} = \frac{10}{1 + 1}$$

but rather

$$A_{sp} = \frac{10}{\sqrt{1^2 + 1^2}}$$

which equals 7.07, for a -3 dB relative loss

A simpler way of stating all of this is: The new upper break frequency is equal to the open-loop upper break times the sacrifice factor, S. Because S is the loop gain, it is equal to A_{ol}/A_{sp}. Note that our low frequency S = 10,000/10, or 1,000. Therefore, our closed-loop break equals 1,000 times 100 Hz, or 100 kHz. A very important item to notice here is that there is an inverse relation between closed-loop gain and frequency response. Systems with low gains will have high upper-breaks, while high gain systems will suffer from low upper-breaks. This sort of trade-off is very common. Although most diff amps and op amps do not have lower break frequencies, circuits that do will see an extension of their lower response in a similar manner. (i.e., the lower break will be reduced by S). In order to achieve both high gain and wide bandwidth, it may be necessary to cascade multiple low gain stages.

Example 7.9.4.1

Assume that you have an amplifier connected as in Figure 7.9.4.1. The open loop gain (A_{ol}) of the amp is 200 and its open loop upper break frequency (f_{2-ol}) is 10 kHz. If the feedback factor (β) is 0.04, what are the closed loop gain (A_{sp}) and break frequency (f_{2-sp})

For A_{sp},

$$A_{sp} = \frac{A_{ol}}{1 + \beta A_{ol}}$$

$$A_{sp} = \frac{200}{1 + 0.04 \times 200}$$

$$A_{sp} = 22.22$$

The approximation says,

$$A_{sp} = \frac{1}{\beta}$$

$$A_{sp} = \frac{1}{0.04}$$

$$A_{sp} = 25$$

That is reasonably close to the general equation's answer (note that there is no need to include phase effects as we are looking for the midband gain). The approximation is more accurate when A_{ol} is larger.

For f_{2-sp}, first find the sacrifice factor, S

$$S = \frac{A_{ol}}{A_{sp}}$$

$$S = \frac{200}{22.22}$$

$$S = 9$$

$$f_{2-sp} = f_{2-ol}S$$

$$f_{2-sp} = 10kHz \times 9$$

$$f_{2-sp} = 90kHz$$

One interesting thing to note is that the product of the gain and upper break frequency will always equal a constant value, assuming a 20 dB per decade roll off. Our open loop product is 200 times 10 kHz, or 2 MHz. Our closed loop product is 22.22 times 90 kHz, which is 2 MHz. If we choose any other feedback factor, the resulting A_{sp} and f_{2-sp} will also produce a product of 2 MHz (try it and see). The reason for this is simple. A 20 dB per decade rolloff means that the gain drops by a factor of 10 when the frequency is increased by a factor of 10. There is a perfect 1:1 inverse relationship between the two parameters. No matter how much you increase one parameter, the other one will decrease by the same proportion. Thus, the product is a constant.

At this point you may be asking yourself, "What exactly is in that feedback network and how do I figure out β?" Usually, the feedback network just needs to produce a loss - it has to scale V_{out} down to $V_{feedback}$. The simplest item for the job would be a resistive voltage divider. (It is possible to have complex frequency dependent or non-linear elements in the network as we shall see in the future). An example is presented in Figure 7.9.4.2. If you study this diagram for a moment, you will notice that the feedback factor β is really nothing more than the voltage divider loss. V_{out} is the input to the feedback network and appears across $R_f + R_i||Z_{in}$. The output of the network is $V_{feedback}$, which appears across R_i. Simply put, the ratio is

$$\beta = \frac{R_i||Z_{in}}{R_f + R_i||Z_{in}}$$

If the Z_{in} is large enough to ignore, as in most op amps, this simplifies to

$$\beta = \frac{R_i}{R_f + R_i}$$

By substituting this Equation into our approximate gain Equation 7.9.4.9, we find

$$A_{sp} = R_f + R_i Ri = \frac{R_f}{R_i} + 1 \tag{7.9.4.10}$$

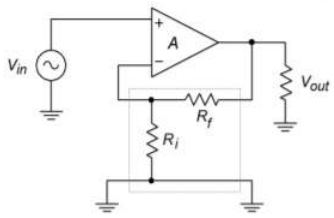


Figure 7.9.4.2: Simple voltage divider for β.

Note that the values of R_f and R_i are not really important, rather, their ratio is. We would arrive at the same gain if $R_f = 10k\Omega$ and $R_i = 1k\Omega$, or $R_f = 20k\Omega$ and $R_i = 2k\Omega$. We obviously have quite a bit of latitude when designing circuits for a specific gain, but we do face a few practical limits. If the resistors are too small we will run into problems with op amp output current. On the other hand, if the resistors are too large, excessive noise, offset, drift and loading effects will result. As a guideline for general-purpose circuits, $R_f + R_i$ is usually in the range of 10 k to 100 kΩ.

Computer Simulation

As evidenced earlier, if the open loop gain is very high, its precise value does not matter. We'll examine this effect using Example 7.9.4.1, but with much higher open loop gains. The simulation is shown in Figure 7.9.4.3 using Multisim and the basic op amp model presented in Chapter Two. The circuit uses $R_f = 24k\Omega$ and $R_i = 1k\Omega$ for an ideal closed loop voltage gain of 25. The input signal is set to 40 millivolts.

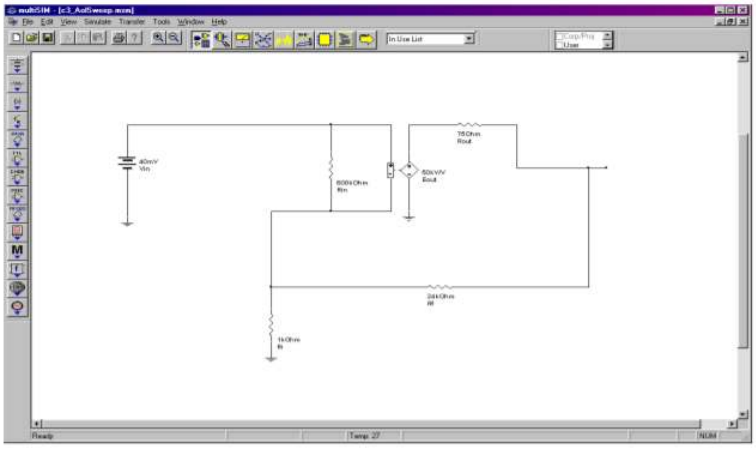


Figure 7.9.4.3a: Multisim schematic for gain sweep.

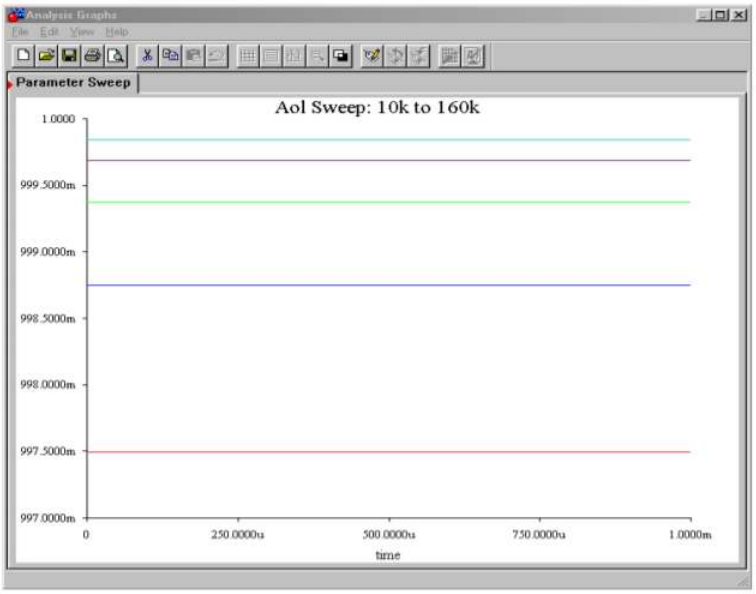


Figure 7.9.4.3b: Output variation due to open-loop gain change.

Using the Parameter Sweep option, the open loop gain is initialized at 10,000 and is progressively doubled to a maximum of 160,000. These are reasonable gain values for a production op amp. The output transient analysis shows that in spite of a 16 to 1 variation in open loop gain, all output voltages are approximately 1 V, achieving a closed loop gain of nearly 25 in all cases.

Example 7.9.4.2

Let's say that the microphone that you use for acoustic instruments produces a signal that is just too weak for you to record without excessive noise. After a little experimentation in lab, you discover that you need about 20 dB of voltage gain before you can successfully capture a softly picked guitar. Using Figure 7.9.4.4 as a guide, design this amplifier with the following device: A_{ol} = 50,000, $Z_{in-ol} = 600k\Omega$.

First, note that our A_{ol} and Z_{in-ol} values are more than sufficient for us to use the approximation formulas. Because our formulas all deal with ordinary gain, we must convert 20 dB.

$$A_{sp} = log^{-1}\frac{A_{sp}^{'}}{20}$$

$$A_{sp} = log^{-1}\frac{20dB}{20}$$

https://eng.libretexts.org/@go/page/53003

$$A_{sp} = 10$$

By rearranging Equation 7.9.4.10,

$$\frac{R_f}{R_i} = A_{sp} - 1$$

$$\frac{R_f}{R_i} = 9$$

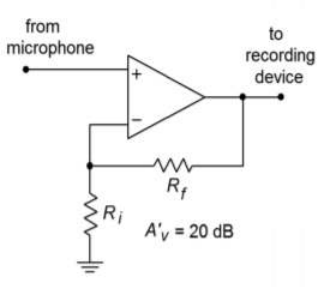


Figure 7.9.4.4: Microphone amplifier.

We see that R_f must be 9 times larger than R_i. There is no single right answer here; there are many possibilities.

Computer Simulation

One viable solution for Example 7.9.4.2 is simulated using Multisim in Figure 7.9.4.5. This circuit uses $R_f = 9k\Omega$ and $R_i = 1k\Omega$. The op amp model is the simple dependent source version examined in Chapter Two. The input signal is set to 0.1 V DC for simplicity. Both the output and feedback potentials are presented. The results of this simulation verify our hand calculations. In order to note the sensitivity of the design, you can alter certain parameters of the input file and rerun the simulation. Two of the more interesting areas are the absolute values of the feedback resistors and the open loop gain of the op amp. You will note that as these quantities are lowered, our approximation formulas become less accurate. With the given values, the approximations deviate from the simulation results by less than 1.

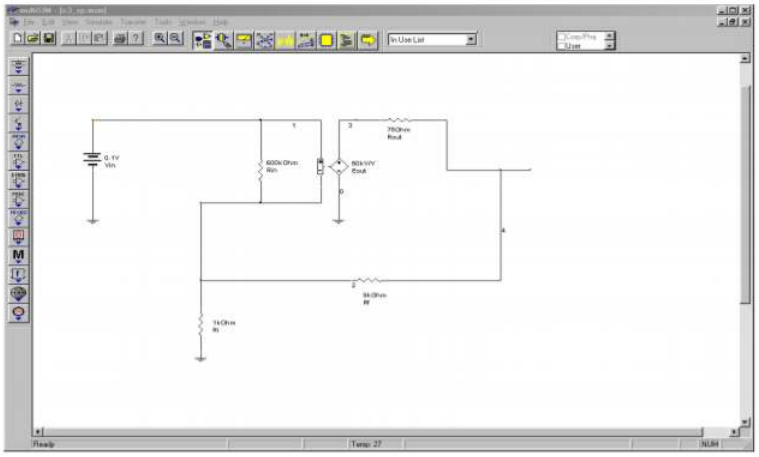


Figure 7.9.4.5a: Multisim schematic for analysis of the simple op amp model of Example 7.9.4.2.

https://eng.libretexts.org/@go/page/53003

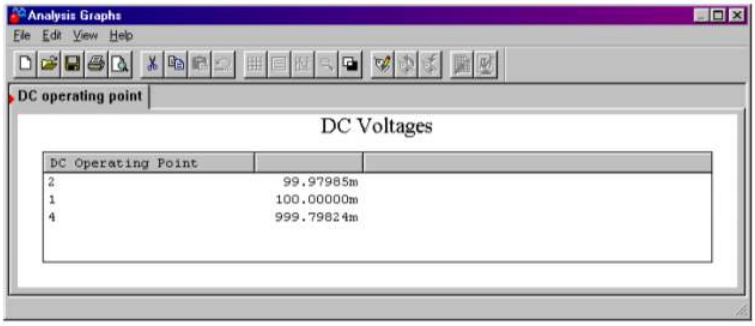


Figure 7.9.4.5b: Multisim DC outputs.

3.4.2: SP Impedance Effects

As noted earlier, negative feedback affects the closed-loop input and output impedances of our system. Series connections increase the impedance and parallel connections decrease the impedance.

Let's see exactly how this works in the SP case. First, we must distinguish between the Z_{in} of the amplifier itself and the Z_{in} of the system with feedback. We shall call them Z_{in-ol} and Z_{in-sp} respectively. Figure 7.9.4.6 shows this difference by using a simple model of the amplifier. By definition,

$$Z_{in-sp} = \frac{V_{in}}{I_{in}}$$

The idea here is to notice that the source only needs to supply enough signal current to develop the V_{error} drop across the amplifier's Z_{in-ol}. As far as the V_{in} signal source is concerned, $V_{feedback}$ is a voltage source, not a voltage drop. Therefore, $I_{in} = V_{error}/Z_{in-ol}$.

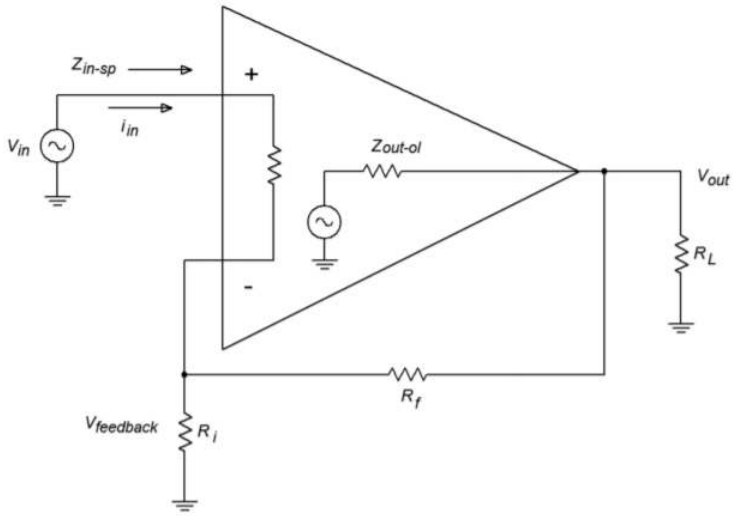


Figure 7.9.4.6: Series-parallel input impedance.

We may now say,

$$Z_{in-sp} = \frac{V_{in}}{\left(\frac{V_{error}}{Z_{in-ol}}\right)} = \frac{Z_{in-ol}V_{in}}{V_{error}}$$

Because V_{error} ideally equals V_{out}/A_{ol}

$$Z_{in-sp} = \frac{Z_{in-ol}A_{ol}V_{in}}{V_{out}} = \frac{Z_{in-ol}A_{ol}}{A_{sp}}$$

Sacrifice factor S, is defined as A_{ol}/A_{sp}, so,

$$Z_{in-sp} = Z_{in-ol}S$$

This is our ideal SP input impedance. Obviously, even moderate open-loop $Z_{in}s$ with moderate sacrifice factors can yield high closed-loop $Z_{in}s$. The upper limit to this will be the impedance seen from each input to ground. In the case of a typical op amp, this is sometimes referred to as the common-mode input impedance, Z_{in-cm}, and can be very high (perhaps hundreds of mega ohms). This is the impedance presented to common mode signals. This value effectively appears in parallel with our calculated Z_{in-sp}, above. An example is shown in Figure 7.9.4.7. Note that because Z_{in-cm} is measured with the inputs of the op amp in parallel, each input has approximately twice the value to ground. In the case of a discrete amplifier, you would be most concerned with the noninverting input's Z_{in}. In any case, because S drops as the frequency increases, Z_{in-sp} decreases as well. At very high frequencies, input and stray capacitances dominate, and the real system input impedance may be a small fraction of the low frequency value. Negative feedback cannot reduce effects that live outside of the loop.

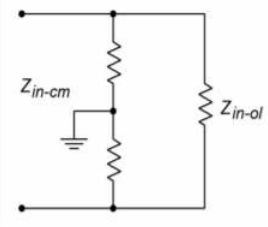


Figure 7.9.4.7: Common -mode input impedance.

Now for our Z_{out-sp}. Refer to Figure 7.9.4.8. Z_{out-sp} is the Thevenin output impedance. In order to find this, we will drive the amplifier's output with a voltage source and reduce all other independent voltage sources to zero (we have no current sources. Remember, this is a “paper” analysis technique and may not work in lab due to other factors). By figuring out the resulting output current, we can find Z_{out-sp} (by definition, $Z_{out-sp} = V_{out}/I_{out}$).

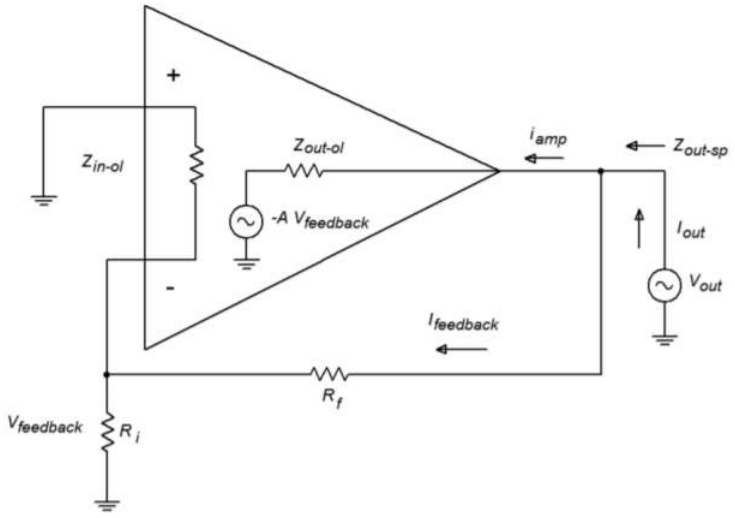


Figure 7.9.4.8: Series-parallel output impedance.

First, notice that I_{out} is made of two pieces, $I_{feedback}$ and I_{amp}. If we can find the two impedances associated with these parts, we can simply perform a parallel equivalent in order to determine Z_{out-sp}. The $I_{feedback}$ portion is very easy to determine. Ignoring the inverting input's loading effects on R_i, this impedance is just $R_f + R_i$. Finding the output impedance of the amplifier itself is a little more involved. I_{amp} is found by taking the drop across Z_{out-ol} and using Ohm's Law. The voltage across Z_{out-ol} is the difference between V_{out} and the signal created by the feedback path to the inverting input. This signal is $-A_{ol}V_{feedback}$.

$$I_{amp} = \frac{V_{out} - (-A_{ol}V_{feedback})}{Z_{out-ol}}$$

Since $V_{feedback} = V_{out}\beta$

$$I_{amp} = \frac{V_{out} + A_{ol}\beta V_{out}}{Z_{out-ol}}$$

$$I_{amp} = \frac{V_{out} + (1 + A_{ol}\beta)}{Z_{out-ol}}$$

By using Equation 7.9.4.8, this may be simplified to,

$$I_{amp} = \frac{SV_{out}}{Z_{out-ol}}$$

Because

$$Z_{out-amp} = \frac{V_{out}}{I_{amp}}$$

This is the part of Z_{out-sp} contributed by I_{amp}. To find Z_{out-sp}, just combine the two pieces in parallel:

$$Z_{out-sp} = \frac{Z_{out-ol}}{S} || (R_f + R_i)$$

With op amps, $R_f + R_i$ is much larger than the first part and can be ignored. For example, a typical device may have $Z_{out-ol} = 75\Omega$. Even a very modest sacrifice factor will yield a value many times smaller than a typical $R_f + R_i$ combo (generally over $1k\Omega$). Discrete circuits using common emitter or common base connections will have larger Z_{out-ol} values, and therefore the feedback path may produce a sizable effect. As in the case of input impedance, Z_{out-sp} is a function of frequency. As S decreases as the frequency rises, Z_{out-sp} will increase.

Example 7.9.4.3

An op amp has the following open-loop specs: $Z_{in} = 300k\Omega$ $Z_{out} = 100\Omega$ and A = 50,000. What are the low frequency system input and output impedances if the closed loop gain is set to 100?

First we must find S

$$S = \frac{A_{ol}}{A_{sp}}$$

$$S = \frac{50,000}{100}$$

$$S = 500$$

We may now find the approximate solutions.

$$Z_{in-sp} = SZ_{in-ol}$$

$$Z_{in-sp} == 500 \times 300\ k\Omega = 150\ M\Omega$$

$$Z_{out-sp} = \frac{Z_{out-ol}}{S}$$

$$Z_{out-sp} = \frac{100}{500} = 0.2\Omega$$

The effects are quite dramatic. Note that with such high Z_{in} values, op amp circuits may be used in place of FETs in some applications. One example would be the front-end amplifier/buffer in an electrometer (electrostatic voltmeter).

3.4.3: Distortion Effects

As noted earlier, negative feedback lowers static forms of distortion, such as THD. The question, as always, is “How much?” Something like harmonic distortion is internally generated, so we may model it as a voltage source in series with the output source. An example of this model can be seen in Figure 7.9.4.9. V_{dist} is the distortion generator. In the case of a simple input sine wave, V_{dist} will contain harmonics at various amplitudes (sine waves at integer multiples of the input frequency). These amplitudes are directly related to the input signal's amplitude. If we assume that Z_{out} is small enough to ignore, V_{dist} will appear at the output of the open-loop circuit. Thus, the total output voltage is the desired AV_{in} plus V_{dist}.

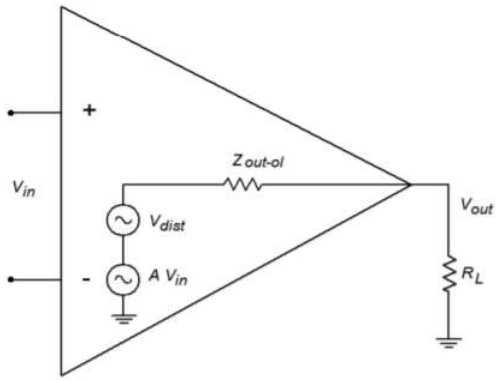


Figure 7.9.4.9: Distortion model (open-loop)

When we add feedback, as in Figure 7.9.4.10, this distortion signal is fed back to the inverting input, and because it is now out of phase, it partially cancels the internally generated distortion. Thus the SP distortion signal (V_{dis-sp}) is much smaller.

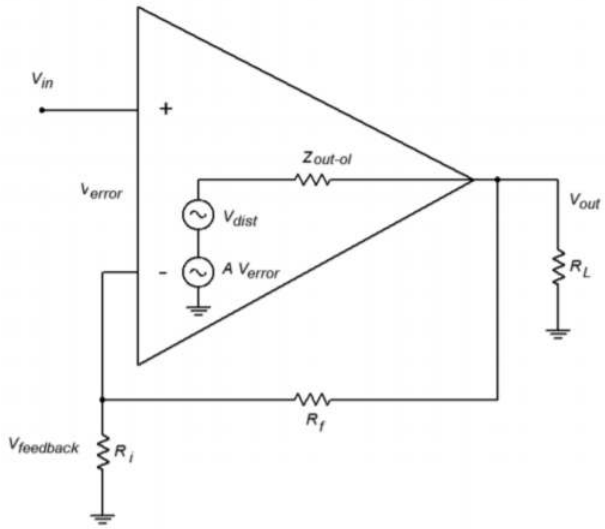


Figure 7.9.4.10: Distortion model (closed-loop)

The SP output signal is

$$V_{out} = A_{ol}V_{error} + V_{dist}$$

$$V_{out} = A_{ol}(V_{in} - V_{feedback}) + V_{dist}$$

$$V_{out} = A_{ol}(V_{in} - \beta V_{out}) + V_{dist}$$

We now perform some algebra in order to get this into a nicer form and solve for V_{out}

$$V_{out} = A_{ol}V_{in} - A_{ol}\beta V_{out} + V_{dist}$$
$$V_{out} + A_{ol}\beta V_{out} = A_{ol}V_{in} + V_{dist}$$
$$V_{out}(1 + A_{ol}\beta) = A_{ol}V_{in} + V_{dist}$$

Remember that $1 + A_{ol}\beta = S$, so,

$$V_{out}S = A_{ol}V_{in} + V_{dist}$$
$$V_{out} = A_{ol}\frac{V_{in}}{S} + \frac{V_{dist}}{S}$$

Because A_{ol}/S is just A_{sp}, this reduces to,

$$V_{out} = A_{sp}V_{in} + \frac{V_{dist}}{S}$$

The internally generated distortion is reduced by the sacrifice factor. As you can see, large sacrifice factors can drastically reduce distortion. An amplifier with 10% THD and a sacrifice factor of 100 produces an effective distortion of only 0.1%. This analysis does assume that the open-loop distortion is not overly grotesque. If the distortion is large, we cannot use this superposition approach (remember, superposition assumes that the circuit is essentially linear). Also, we are ignoring any additional distortion created by feeding this distortion back into the amplifier. For any reasonably linear amplifier, this extra distortion is a second-order effect, and thus constitutes only a small part of the total output signal.

Computer Simulation

An example of the reduction of distortion is simulated using the Distortion Analyzer from Multisim. A basic amplifier is shown in Figure 7.9.4.11. The value for R_f is changed from 999 kΩ to 99 kΩ to 9 kΩ. V_{in} is scaled accordingly so that the output of the amplifier remains at approximately 10 volts. The test frequency was set to 1 kHz and a total of 20 harmonics (up to 20 kHz) were used in the analysis. Using the LF411 op amp, the high gain version shows a THD of 0.09%. Reducing the gain by a factor of 10 (and thus increasing sacrifice factor by 10 fold) yields a THD of 0.011%. Finally, a further 10 fold reduction of gain yields a THD of 0.001%. Although the reduction in distortion is not exactly a factor of 10 each time, the trend can be seen clearly. The precise distortion values will depend on the accuracy of the op amp model used, the test frequency, and the number of harmonics kept in the analysis. 20 kHz was chosen here because that represents the upper limit of human hearing, and thus it would be appropriate for an audio amplifier.

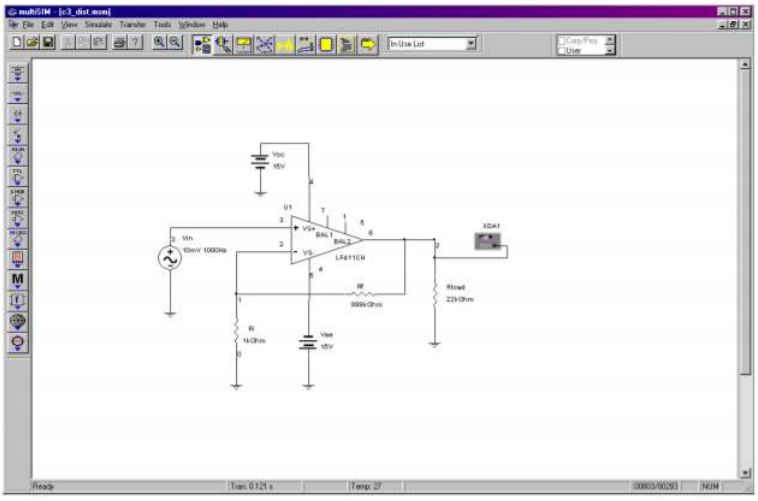


Figure 7.9.4.11a: Multisim distortion analysis

https://eng.libretexts.org/@go/page/53003

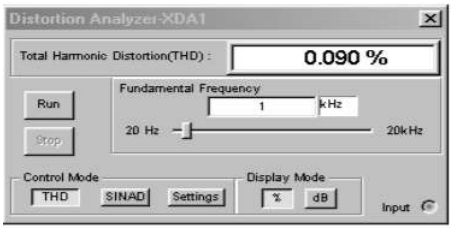


Figure 7.9.4.11*b*: Distortion analyzer results for high-gain version.

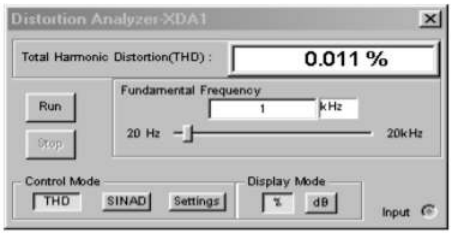


Figure 7.9.4.11*c*: Distortion analyzer results for medium-gain version.

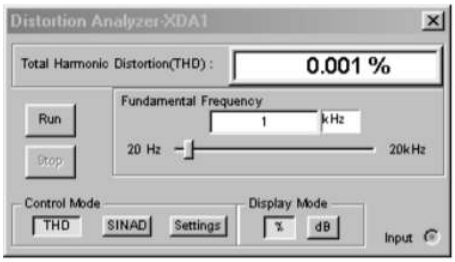


Figure 7.9.4.11*d*: Distortion analyzer results for low-gain version.

As we have seen, the sacrifice factor is a very useful item. Our gain, distortion, and Z_{out} are all reduced by S, while f_2 and Z_{in} are increased by S.

3.4.4: Noise

It is possible to model noise effects in much the same way as we just modeled distortion effects. By doing so, you will discover that noise can also be decreased by a large amount. Unfortunately, there is one major flaw. Unlike our distortion generator, a noise generator will produce a signal that is not dependent on the input signal. The net result is that although the noise level does drop by the sacrifice factor, so does the desired output signal. Thus, the signal-to-noise ratio at the output is unchanged. In contrast, the distortion signal is proportional to the input signal, so that when the desired signal is cut by S, the distortion signal sees a further cut by S (i.e., the distortion drops by S relative to the desired signal). As a matter of fact, it is quite possible that the noise produced in following stages may add up to more noise than the circuit had without feedback. Sad but true, negative feedback doesn't help us when it comes to signal-to-noise ratio.

3.4.5: Parallel-Series (PS)

The Parallel-Series connection is the opposite of the Series-Parallel form. PS negative feedback is used to make the ideal current amplifier. Its gain is dimensionless, but for convenience, it is normally given the units A/A (amps per amp). It produces a low Z_{in} (perfect for sensing I_{in}) and a high Z_{out} (making for an ideal current source). An example of PS is shown in Figure 7.9.4.12.

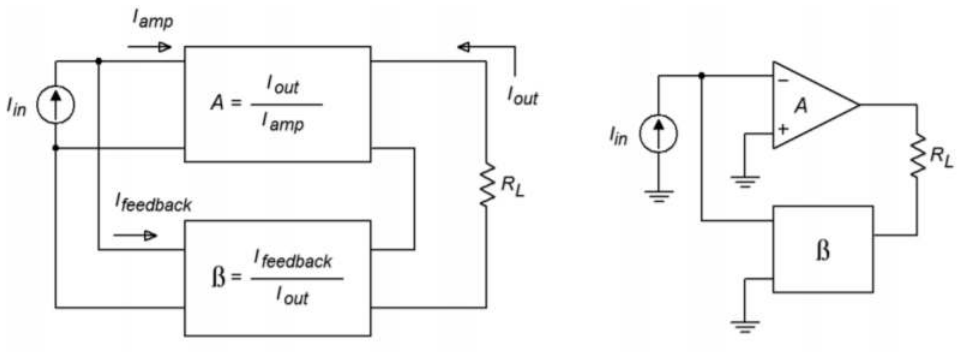


Figure 7.9.4.12: Parallel-series connection.

The signal source's current splits in two, with part going into the amplifier, and part going through the feedback network. This is how you know that you have a parallel input connection. The output current on the other hand, passes through the load and then enters the feedback network, indicating a series output connection. Note that this general model is an inverting type, and that the load is floating (i.e., not ground referenced). It is possible though to use ground referenced loads with some additional circuitry. As PS is used for current amplification, let's see how we can find the current gain. Figure 7.9.4.13 will help us along. PS current gain is defined as:

$$A_{ps} = \frac{I_{out}}{I_{in}} \tag{7.9.4.11}$$

The signal source's current splits into two paths, so

$$I_{in} = I_{amp} + I_{feedback} \tag{7.9.4.12}$$

Because I_{amp} times the amplifier's open-loop current gain is I_{out}, we can also say,

$$I_{amp} = \frac{I_{out}}{A_{ol}} \tag{7.9.4.13}$$

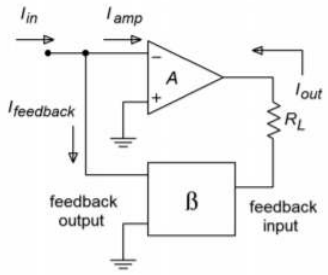


Figure 7.9.4.13: Parallel-series analysis.

The feedback network is nothing more than a current divider, where the feedback network's output ($I_{feedback}$) is β times smaller than its input (I_{out}). Don't let the arbitrary current direction fool you - the feedback “flow” is still from right to left, as always. In this case the amplifier is sinking current instead of sourcing it. As β is just a fraction, we may say

$$\begin{aligned} \beta &= \frac{I_{feedback}}{I_{out}} \text{or,} \\ I_{feedback} &= \beta I_{out} \\ I_{feedback} &= A_{ol}\beta I_{amp} \end{aligned} \tag{7.9.4.14}$$

By substituting Equation 7.9.4.14 into Equation 7.9.4.12 we see that

$$\begin{aligned} I_{in} &= I_{amp} + A_{ol}\beta I_{amp} \text{ or,} \\ I_{in} &= I_{amp}(1 + A_{ol}\beta) \end{aligned} \tag{7.9.4.15}$$

After substituting this into Equation 7.9.4.11 we find that

$$A_{ps} = \frac{I_{out}}{I_{amp}(1 + A_{ol}\beta)}$$

or, with the help of Equation 7.9.4.13

$$A_{ps} = \frac{I_{out}}{1 + A_{ol}\beta} \tag{7.9.4.16}$$

To make a long story short, the open-loop gain is reduced by the sacrifice factor. (Where have we seen this before?) The one item that you should note is that we have used only current gains in our derivation (compared to voltage gains in the SP case).

It is possible to perform a derivation using the open-loop voltage gain; however, the results are basically the same, as you might have guessed. Once again, our approximation for gain can be expressed as $1/\beta$.

Example 7.9.4.4

A co-worker asks you to measure the output current of a circuit that she's just built. If all is working correctly, this circuit should produce 100 microamps. Unfortunately, your hand-held DMM will only accurately measure down to 1 milliamp. It is 10 times less sensitive than it needs to be. In order to be read accurately, the current will need to be boosted. Does the amplifier of Figure 7.9.4.14 have the current gain you need?

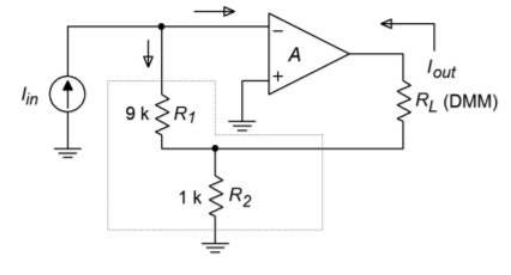


Figure 7.9.4.14: Current amplifier for Example 7.9.4.4.

If we are using an op amp, we can assume that $A_{ps} = 1/\beta$. The question then becomes, "what is β?" β is the current divider ratio. The resistors R_1 and R_2 make up the current divider. The output current splits between R_1 and R_2, where the R_1 path is $I_{feedback}$. According to the current divider rule,

$$\beta = \frac{R_2}{R_1 + R_2}$$

Consequently the gain magnitude is

$$A_{ps} = \frac{R_1 + R_2}{R_2} \text{or,}$$

$$A_{ps} = \frac{R_1}{R_2} + 1$$

This Equation looks a lot like the one in the SP example! Solving for A_{ps} yields

$$A_{ps} = \frac{9\ k}{1\ k} + 1$$

$$A_{ps} = 10$$

Yes, this circuit has just the gain you need. All you have to do is connect your co-worker's circuit to the input, and replace the load resistor with your hand-held DMM. Note that most DMMs are floating instruments and are not tied to ground (unlike an oscilloscope), so using it as a floating load presents no problems. The accuracy of the gain (and thus your measurement) depends on the accuracy of the resistors and the relative size of the op amp's input bias current. Therefore, it would be advisable to use a bi-FET type device (i.e., an op amp with an FET diff amp) and precision resistors.

3.4.6: PS Impedance Effects

As you have seen, the gain derivation for PS is similar to that for SP. The same is true for the impedance equations. First, we'll take a look at input impedance with Figure 7.9.4.15.

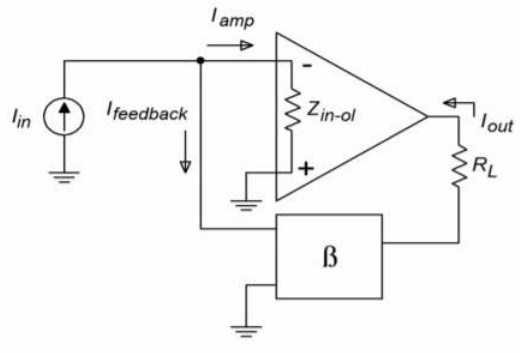


Figure 7.9.4.15: Parallel-series input impedance.

As always, we start with our base definition,

$$Z_{in-ps} = \frac{V_{in}}{I_{in}}$$

Recalling Equation 7.9.4.15, this can be rewritten as,

$$Z_{in-ps} = \frac{V_{in}}{I_{amp}(1 + A_{ol}\beta)} \tag{7.9.4.17}$$

V_{in} is merely the voltage that appears from the inverting input to ground. By using Ohm's Law, we can say,

$$V_{in} = I_{amp} Z_{in-ol}$$

Where Z_{in-ol} is the open-loop input impedance. Finally, substituting this into Equation 7.9.4.17 gives,

$$Z_{in-sp} = \frac{I_{amp} Z_{in-ol}}{I_{amp}(1 + A_{ol}\beta)}$$

$$Z_{in-sp} = \frac{Z_{in-ol}}{(1 + A_{ol}\beta)}$$

The input impedance is lowered by the sacrifice factor. If you're starting to wonder whether everything is altered by the sacrifice factor, the answer is, yes. This is, of course, an approximation. What do you think is going to happen to the output impedance? At this point, it shouldn't be too surprising. For this proof, refer to Figure 7.9.4.16. We shall use the same general technique to find Z_{out} as we did with the SP configuration. In this case we replace the load with a current source, and then determine the resulting output voltage.

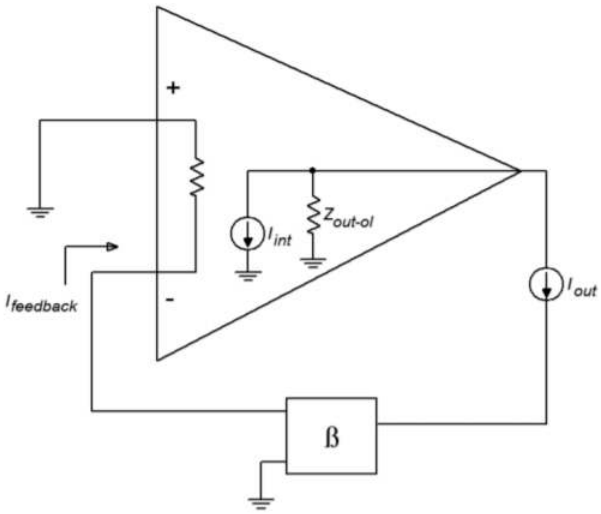


Figure 7.9.4.16: Parallel-series output impedance.

Note that the input signal current source is opened. The output current drives the feedback network and produces the feedback current. The feedback current is then multiplied by the amplifier's open-loop current gain. Because the feedback current enters the inverting input, the internal source is sinking current. Because we are driving the circuit from the output,

$$Z_{out-ps} = \frac{V_{out}}{I_{out}}$$

V_{out} is found through Ohm's Law

$$V_{out} = Z_{out-ol}(I_{out} + I_{int})$$

We now expand on our currents.

$$I_{int} = I_{feedback}A_{ol}$$

$$I_{int} = I_{out}\beta A_{ol}$$

$$V_{out} = Z_{out-ol}(I_{out} + I_{out}\beta A_{ol})$$

$$V_{out} = Z_{out-ol}I_{out}(1 + A_{ol}\beta)$$

$$V_{out} = Z_{out-ol}I_{out}S$$

Finally, we see that,

$$Z_{out-ps} = \frac{Z_{out-ol}I_{out}S}{I_{out}}$$

$$Z_{out-ps} = Z_{out-ol}S$$

As expected, the series output connection increased Z_{out} by the sacrifice factor. The remainder of the PS equations are essentially those used in our earlier SP work. Once again, the bandwidth will be increased by S, and the distortion will be reduced by S. This is also true for the Parallel-Parallel and Series-Series connections. As a matter of fact, the Z_{in} and Z_{out} relations are just what you might expect. The proofs are basically the same as those already presented, so we won't go into them. Suffice to say that parallel connections reduce the impedance by S, and series connections increase it by S.

3.4.7: Parallel-Parallel (PP) and Series-Series (SS)

Unlike our two earlier examples, the concept and modeling of gain is not quite as straight forward in the Parallel-Parallel and Series-Series cases. These forms do not produce gain, so to speak. They are neither voltage amplifiers, nor current amplifiers.

Instead, these connections are used as transducers. Parallel-Parallel turns an input current into an output voltage. It is shown in Figure 7.9.4.17.

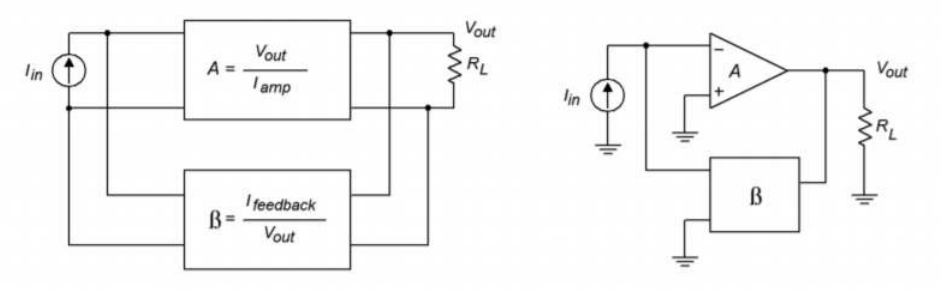


Figure 7.9.4.17: Parallel-parallel connection.

Series-Series turns an input voltage into an output current. It is shown in Figure 7.9.4.18. Our normal gain and feedback factors now have units associated with them. Because our output versus input quantities are measured in volts-per-amp (PP) or amps-per-volt (SS), the appropriate units for our factors are Ohms and Siemens. To be specific, we refer to our PP gain as a transresistance value, and the SS gain as a transconductance value. Although it is quite possible (and useful) to derive formulas for gain based on these, it is often done only for discrete designs. Quite simply, you cannot find transresitance or transconductance values on typical op amp data sheets.

Fortunately, we can make a few approximations and create some very useful circuits utilizing PP and SS feedback with op amps. These design and analysis shortcuts are presented in the next chapter, along with practical applications.

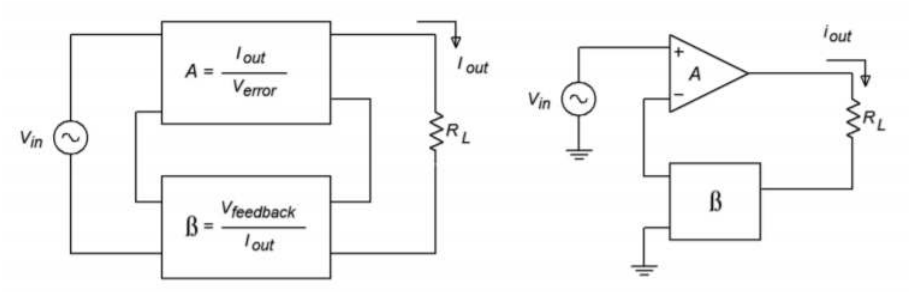


Figure 7.9.4.18: Series-series connection.

This page titled 7.9.4: The Four Variants of Negative Feedback is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.4: The Four Variants of Negative Feedback** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.9.5: Limitations On The Use of Negative Feedback

From the fore-going discussion, you may well think that negative feedback can do just about anything, short of curing a rainy day. Such is not the case. Yes, negative feedback can drastically lower distortion and increase bandwidth. Yes, it can have a very profound effect on input and output impedance. And yes, it certainly does stabilize our gains. What then, is the problem? Like all things, negative feedback has specific limitations. The first thing you should note is that S is a function of frequency. This was graphically depicted back in Figure 3.3.2. The amount of change seen in impedances and distortion is a function of S, it follows that these changes must be a function of frequency. Because S drops as the frequency increases, the effects of negative feedback diminish as well. For example, if an SP amplifier has an open-loop Z_{in} of 200 kΩ and the low frequency S is 500, the resulting Z_{in} with feedback is 100 MΩ. If we increase the input frequency past the open-loop f_2, the open-loop gain drops and thus, S drops. One decade up, S will only be 50, so the Z_{in} with feedback will only be 10 MΩ. If this amplifier has a lower break frequency (f_1), S will drop as the frequency is reduced (below f_1). The same sort of thing occurs with distortion, however, the harmonics each see a different S, so the calculation is a bit more involved. Along with the reduction in gain, there is also a change in phase. If the phase around the feedback loop varies from -180°, incomplete cancellation takes place, and thus, the effects of feedback are lessened. The bottom line is that the effects of negative feedback weaken as we approach the frequency extremes.

The other item that must be kept in mind is the fact that negative feedback does not change specific fundamental characteristics of the amplifier. Negative feedback cannot get a circuit to do something beyond its operational parameters. For example, feedback has no effect on clipping level (saturation point). Further, feedback has no effect on slew rate (the maximum rate of output signal change, and an item that we will examine in a later chapter). Actually, when an amplifier slews, feedback is effectively blocked. An accurate output signal is no longer sent back to the input, but the amplifier can't correct for the errors any faster than it already is. In a similar manner, even though feedback can be used to lower the output impedance of a system, this does not imply that the system can produce more output current.

One possible “problem” with negative feedback is really the fault of the designer. It can be very tempting to sloppily design an amplifier with poor characteristics and then correct for them with large amounts of feedback. No matter how much feedback you use, the result will never be as good as a system that was designed carefully from the start. An example of this effect can be seen with TIMD (Transient Inter-Modulation Distortion). TIMD is a function of non-linearities in the first stages of an amplifier, and the excessive application of negative feedback will not remove it. On the other hand, if the initial stages of the system are properly designed, TIMD is not likely to be a problem.[1]

References

[1]See E.M.Cherry and K.P.Dabke, “Transient Intermodulation Distortion- Part 2: Soft Nonlinearity”, Journal of the Audio Engineering Society, Vol.34 No.1/2 (1986): 19–35

This page titled 7.9.5: Limitations On The Use of Negative Feedback is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.5: Limitations On The Use of Negative Feedback** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.9.6: Summary

We have seen that negative feedback can enhance the performance of amplifier circuits. This is done by sampling a portion of the output signal and summing it out of phase with the input signal. In order to maintain stability, the gain of the amplifier must be less than unity by the time its phase reaches -180°. There are four basic variants of negative feedback: Series-Parallel, Parallel-Series, Parallel-Parallel, and Series-Series. In all cases, gain and distortion are lowered by the sacrifice factor S, and the bandwidth is increased by S. Parallel connections reduce impedance by S, whereas series connections increase the impedance by S. At the input, parallel connections are current sensing, and series connections are voltage sensing. At the output, parallel connections produce a voltage-source model, and series connections produce a current-source model. The sacrifice factor is the ratio between the openand closed-loop gains. It is a function of frequency, and therefore, the effects of negative feedback lessen at the frequency extremes.

This page titled 7.9.6: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.6: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.9.7: Problems

3.7.1: Review Questions

1. Give two examples of how negative feedback is used in everyday life.

2. What circuit parameters will negative feedback alter, and to what extent?

3. What is meant by the term “Sacrifice Factor”?

4. What is the usage of Gain and Phase Margin?

5. Name the different negative feedback connections (i.e., variants or forms).

6. How might negative feedback accidentally turn into positive feedback?

7. What circuit parameters won't negative feedback effect?

8. In practical amplifiers, when does negative feedback “stop working”, and why?

3.7.2: Problems

Analysis Problems

1. An amplifier's open-loop gain plot is given in Figure 7.9.7.1. If the amplifier is set up for a closed-loop gain of 100, what is the sacrifice factor (S) at low frequencies? What is S at 1 kHz?

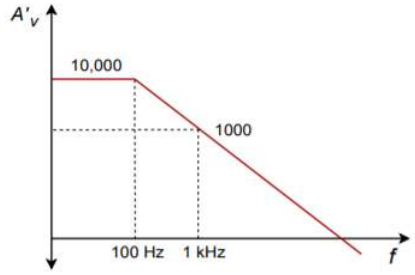


Figure 7.9.7.1

2. If the amplifier in Problem 1 has an open-loop THD of 5%, what is the closed-loop THD at low frequencies? Assuming that the open-loop THD doesn't change with frequency, what is the closed-loop THD at 1 kHz?

3. If an amplifier has an open-loop response as given in Figure 7.9.7.1, and a feedback factor (β) of 0.05 is used, what is the exact low frequency closed loop gain? What is the approximate low frequency gain? What is the approximate gain at 1 kHz?

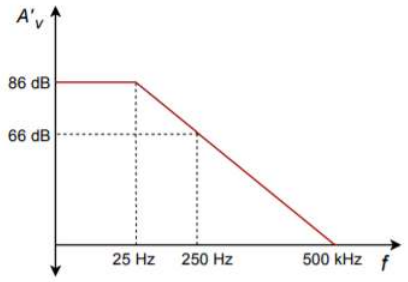


Figure 7.9.7.2

4. Using the open loop response curve in Figure 7.9.7.2, determine exact and approximate values of β for a closed-loop gain of 26 dB.

5. Determine the closed-loop f_2 for the circuit of Problem 4.

6. What is the maximum allowable phase shift at 500 kHz for stable usage of negative feedback in Figure 7.9.7.2?

7. Determine the gain and phase margins for the amplifier response given in Figure 7.9.7.3. Is this amplifier a good candidate for negative feedback?

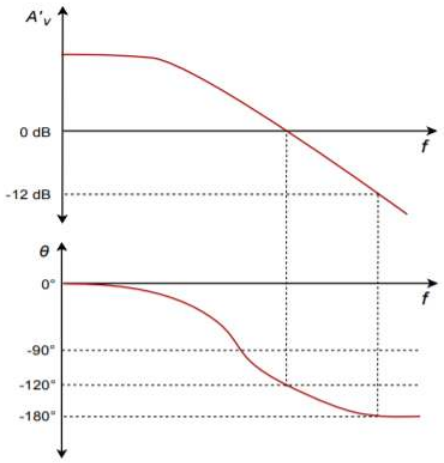


Figure 7.9.7.3

8. Determine the closed-loop (mid-band) gain in Figure 7.9.7.4.

9. What is the closed-loop Z_{in} in Figure 7.9.7.4? What is Z_{out}?

10. What is the low-frequency sacrifice factor in Figure 7.9.7.4?

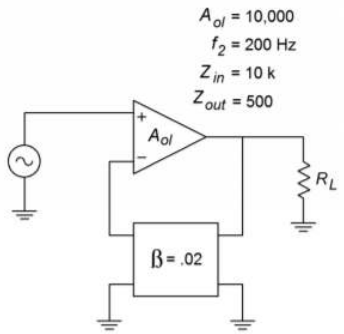


Figure 7.9.7.4

11. How much distortion reduction can we hope for in Figure 7.9.7.4?

12. How much of a signal/noise improvement can we expect in Figure 7.9.7.4?

13. Assuming $V_{in}(t) = 0.1 \sin 2\pi 500t$, in Figure 7.9.7.4, what is $V_{out}(t)$?

14. If the circuit in Figure 7.9.7.4 had an open-loop f_1 of 10 Hz, what would the closed-loop f_1 be?

15. Determine an appropriate pair of resistors to set β to 0.1 in Figure 7.9.7.4.

Challenge Problems

16. If the feedback network of Figure 7.9.7.4 produces a phase shift of -200° at 4 kHz, what effect will this have on circuit operation in Problem 10?

17. Consider the circuit of Figure 3.4.4. In general, what effect will the following alterations to the feedback network have on the closed loop system response? A) Placing a capacitor across R_f B) Placing a capacitor across R_i C) Placing a rectifying diode across R_f (both polarities).

Computer Simulation Problems

18. Rerun the simulation of Figure 3.4.5 using the following open loop gains: 1 k, 10 k, and 100 k. What can you conclude from the results?

19. Verify the results of Problem 13 using a circuit simulator. It is possible to extend the basic op amp model with a lag network in order to mimic f_{2-ol}.

20. Verify the stability of the circuit used in Problem 13 with regard to the open loop gain. Run simulations with the given A_{ol}, and with values one decade above and below. Compare the resulting simulations and determine the maximum deviation of A_{sp}.

This page titled 7.9.7: Problems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.7: Problems** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

SECTION OVERVIEW

7.10: Basic Op Amp Circuits

Learning Objectives

After completing this chapter, you should be able to:

- Relate each op amp circuit back to its general feedback form.
- Detail the general op amp circuit analysis idealizations.
- Solve inverting and noninverting voltage amplifier circuits for a variety of parameters, including gain and input impedance.
- Solve voltage/current transducer circuits for a variety of parameters.
- Solve current amplifier circuits for a variety of parameters.
- Define the term virtual ground.
- Analyze and design differential amplifiers.
- Analyze and design inverting and noninverting summing amplifiers.
- Discuss how output current capability may be increased.
- Outline the circuit modifications required for operation from a single polarity power supply.

7.10.1: Introduction

7.10.2: Inverting and Noninverting Amplifiers

7.10.3: Single Supply Biasing

7.10.4: Current Boosting

7.10.5: Summary

7.10.6: Problems

Template:HideTOC

This page titled 7.10: Basic Op Amp Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.10.1: Introduction

In this chapter we will be examining some common uses of op amps. Where Chapter Three focused on the theory of negative feedback in a more abstract way, this chapter zeroes in on the practical results of using negative feedback with op amps. Now that you know what negative feedback is and how it works, you can reap the benefits. The main theme is in the design and analysis of simple, small-signal, linear amplifiers. We will also be looking at some convenient approximation methods of analysis that can prove quite efficient. By the end of the chapter, you should be able to design single and multistage voltage amplifiers, voltage followers, and even amplifiers that sense current and/or produce constant current output. The voltage amplifiers offer the possibilities of inverting or not inverting the signal. Simple differential amplifiers are examined too. The chapter wraps up with a section on using op amps with a single polarity power supply and how to increase the available output current.

This is an introductory design section so the details of high frequency response, noise, offsets, and other important criteria are ignored. These items await a detailed analysis in Chapter Five. For the most part then, all calculations and circuit operations are assumed to be in the midband region.

This page titled 7.10.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.10.2: Inverting and Noninverting Amplifiers

As noted in our earlier work, negative feedback can be applied in one of four ways. The parallel input form inverts the input signal, and the series input form doesn't. Because these forms were presented as current-sensing and voltage-sensing respectively, you might get the initial impression that all voltage amplifiers must be noninverting. This is not the case. With the simple inclusion of one or two resistors, for example, we can make inverting voltage amplifiers or noninverting current amplifiers. Virtually all topologies are realizable. We will look at the controlledvoltage source forms first (those using SP and PP negative feedback).

For analysis, you can use the classic treatment given in Chapter Three; however, due to some rather nice characteristics of the typical op amp, approximations will be shown. These approximations are only valid in the midband and say nothing of the high frequency performance of the circuit. Therefore, they are not suitable for general-purpose discrete work. The idealizations for the approximations are:

- The input current is virtually zero (i.e., Z_{in} is infinite).
- The potential difference between the inverting and noninverting inputs is virtually zero (i.e., loop gain is infinite). This signal is also called the error signal.

Also, note for clarity that the power supply connections are not shown in most of the diagrams.

4.2.1: The Noninverting Voltage Amplifier

The noninverting voltage amplifier is based on SP negative feedback. An example is given in Figure 7.10.2.1. Note the similarity to the generic SP circuits of Chapter Three. Recalling the basic action of SP negative feedback, we expect a very high Z_{in}, a very low Z_{out}, and a reduction in voltage gain. Idealization 1 states that Z_{in} must be infinite. We already know that op amps have low Z_{out}, the second item is taken care of. Now let's take a look at voltage gain.

$$A_v = \frac{V_{out}}{V_{in}}$$

Because ideally $V_{error} = 0$

$$V_{in} = V_{Ri}$$

Also,

$$V_{out} = V_{Ri} + V_{Rf}$$

$$A_v = \frac{V_{Ri} + V_{Rf}}{V_{Ri}}$$

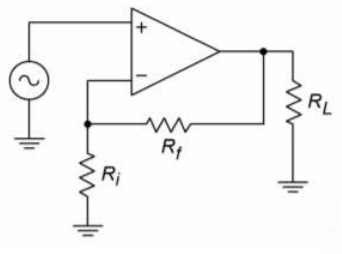


Figure 7.10.2.1: Noninverting voltage amplifier.

Expansion gives

$$A_v = \frac{R_i I_{Ri} + R_f I_{Rf}}{Ri I_{Ri}}$$

Because $I_{in} = 0$, $I_{Rf} = I_{Ri}$, and finally we come to

$$A_v = \frac{R_i + R_f}{R_i} \text{or}$$

$$A_v = 1 + \frac{R_f}{R_i} \tag{7.10.2.1}$$

Now that's convenient. The gain of this amplifier is set by the ratio of two resistors. The larger R_f is relative to R_i, the more gain you get. Remember, this is an approximation. The closed loop gain can never exceed the open loop gain, and eventually, A_v will fall off as frequency increases. Note that the calculation ignores the effect of the load impedance. Obviously, if R_l is too small, the excessive current draw will cause the op amp to clip.

Example 7.10.2.1

What are the input impedance and gain of the circuit in Figure 7.10.2.2?

First off, Z_{in} is ideally infinite. Now for the gain:

$$A_v = 1 + \frac{R_f}{R_i}$$

$$A_v = 1 + \frac{10k}{1k}$$

$$A_v = 11$$

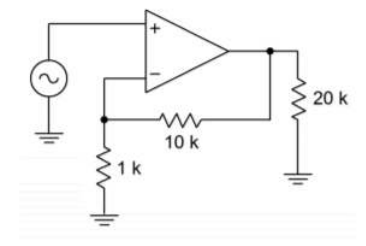


Figure 7.10.2.2: Noninverting circuit for Example 7.10.2.1.

The opposite process of amplifier design is just as straightforward.

Example 7.10.2.2

Design an amplifier with a gain of 26 dB and an input impedance of 47 kΩ. For the gain, first turn 26 dB into ordinary form. This is a voltage gain of about 20.

$$A_v = 1 + \frac{R_f}{R_i}$$

$$\frac{R_f}{R_i} = A_v - 1$$

$$\frac{R_f}{R_i} = 19$$

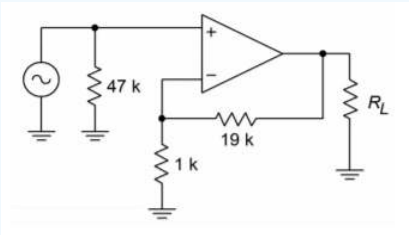


Figure 7.10.2.3: Noninverting design for Example 7.10.2.2.

At this point, choose a value for one of the resistors and solve for the other one. For example, the following would all be valid:

$$R_i = 1k\Omega,\ R_f = 19k\Omega$$

$$R_i = 2k\Omega,\ R_f = 38k\Omega$$

$$R_i = 500\Omega,\ R_f = 9.5k\Omega$$

Most of these are not standard values, though, and will need slight adjustments for a production circuit (see Appendix B). A reasonable range is $100k\Omega > R_i + R_f > 10k\Omega$. The accuracy of this gain will depend on the accuracy of the resistors. Now for the Z_{in} requirement. This is deceptively simple. Z_{in} is assumed to be infinite, so all you need to do is place a 47 kΩ in parallel with the input. The resulting circuit is shown in Figure 7.10.2.3.

If a specific Z_{in} is not specified, a parallel input resistor is not required. There is one exception to this rule. If the driving source is not directly coupled to the op amp input (e.g., it is capacitively coupled), a resistor will be required to establish a DC return path to ground. Without a DC return path, the input section's diff amp stage will not be properly biased. This point is worth remembering, as it can save you a great deal in future headaches. For example, in the lab a circuit like the one in Figure 7.10.2.2 may work fine with one function generator, but not with another. This would be the case if the second generator used an output coupling capacitor and the first one didn't.

Example 7.10.2.3

Design a voltage follower (i.e., ideally infinite Z_{in} and a voltage gain of 1).

The Z_{in} part is straightforward enough. As for the second part, what ratio of R_f to R_i will yield a gain of 1?

$$A_v = 1 + \frac{R_f}{R_i}$$

$$\frac{R_f}{R_i} = A_v - 1$$

$$\frac{R_f}{R_i} = 0$$

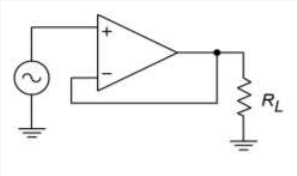


Figure 7.10.2.4: Voltage follower for Example 7.10.2.3.

This says that R_f must be 0 Ω. Practically speaking, that means that R_f is replaced with a shorting wire. What about R_i? Theoretically, almost any value will do. As long as there's a choice, consider infinite. Zero divided by infinite is certainly zero. The practical benefit of choosing $R_i = \infty$ is that you may delete R_i. The resulting circuit is shown in Figure 7.10.2.4. Remember, if the source is not directly coupled, a DC return resistor will be needed. The value of this resistor has to be large enough to avoid loading the source.

As you can see, designing with op amps can be much quicker than its discrete counterpart. As a result, your efficiency as a designer or repair technician can improve greatly. You are now free to concentrate on the system, rather than on the specifics of an individual biasing resistor. In order to make multi-stage amplifiers, just link individual stages together.

Example 7.10.2.4

What is the input impedance of the circuit in Figure 7.10.2.5? What is V_{out}? As in any multi-stage amplifier, the input impedance to the first stage is the system Z_{in}. The DC return resistor sets this at 100 kΩ.

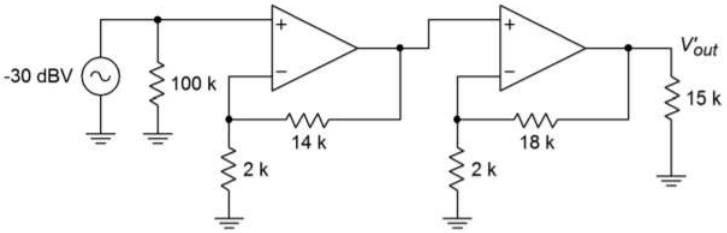


Figure 7.10.2.5: Multistage circuit for Example 7.10.2.4.

To find V_{out}, we need to find the gain (in dB).

$$A_{v1} = 1 + \frac{R_f}{R_i}$$

$$A_{v1} = 1 + \frac{14k}{2k}$$

$$A_{v1} = 8$$

$$A_{v1}^{'} = 18dB$$

$$A_{v2} = 1 + \frac{R_f}{R_i}$$

$$A_{v2} = 1 + \frac{18k}{2k}$$

$$A_{v2} = 10$$

$$A_{v2}^{'} = 20dB$$

$$A_{vt}^{'} = 18dB + 20dB = 38dB$$

$$V_{out}^{'} = A_{vt}^{'} + V_{in}^{'}$$

$$V_{out}^{'} = 38dB + (-30dBV)$$

$$V_{out}^{'} = 8dBV$$

Because 8 dBV translates to about 2.5 V, there is no danger of clipping either.

4.2.2: Inverting Voltage Amplifier

The inverting amplifier is based on the PP negative feedback model. The base form is shown in Figure 7.10.2.6. By itself, this form is current sensing, not voltage sensing. In order to achieve voltage sensing, an input resistor, R_i, is added. See Figure 7.10.2.7. Here's how the circuit works: V_{error} is virtually zero, so the inverting input potential must equal the noninverting input potential. This means that the inverting input is at a virtual ground. The signal here is so small that it is negligible. Because of this, we may also say that the impedance seen looking into this point is zero. This last point may cause a bit of confusion. You may ask, “How can the impedance be zero if the current into the op amp is zero?” The answer lies in the fact that all of the entering current will be drawn through R_f, thus bypassing the inverting input.

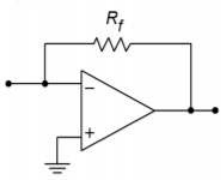


Figure 7.10.2.6: A basic parallel-parallel amplifier.

https://eng.libretexts.org/@go/page/53009

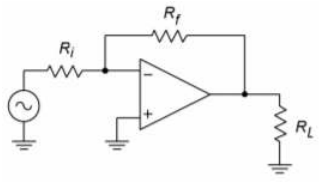


Figure 7.10.2.7: Inverting voltage amplifier.

Refer to Figure 7.10.2.8 for the detailed explanation. The right end of R_i is at virtual ground, so all of the input voltage drops across it, creating I_{in}, the input current. This current cannot enter the op amp and instead will pass through R_f. Because a positive signal is presented to the inverting input, the op amp will sink output current, thus drawing I_{in} through R_f. The resulting voltage drop across R_f is the same magnitude as the load voltage. This is true because R_f is effectively in parallel with the load. Note that both elements are tied to the op amp's output and to (virtual) ground. There is a change in polarity because we reference the output signal to ground. In short, V_{out} is the voltage across R_f, inverted.

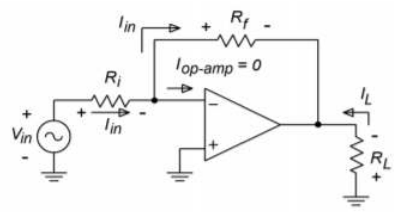


Figure 7.10.2.8: Analysis of the inverting amplifier from Figure 7.10.2.7.

$$A_v = \frac{V_{out}}{V_{in}}$$

$$V_{in} = I_{in}R_i$$

$$V_{out} = -V_{R_f}$$

$$V_{Rf} = I_{in}R_f$$

Substitution yields

$$A_v = -\frac{I_{in}R_f}{I_{in}R_i}$$

$$A_v = -\frac{R_f}{R_i} \tag{7.10.2.2}$$

Again, we see that the voltage gain is set by resistor ratio. Again, there is an allowable range of values.

The foregoing discussion points up the derivation of input impedance. Because all of the input signal drops across R_i, it follows that all the driving source “sees” is R_i. Quite simply, R_i sets the input impedance. Unlike the noninverting voltage amp, there is a definite interrelation between $Z_{in}(R_i)$ and $A_v(-R_f/R_i)$. This indicates that it is very hard to achieve both high gain and high Z_{in} with this circuit.

Example 7.10.2.5

Determine the input impedance and output voltage for the circuit in Figure 7.10.2.9.

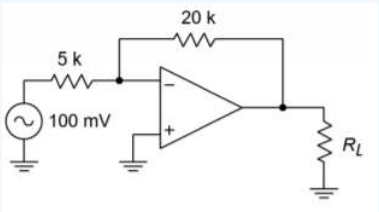


Figure 7.10.2.9: Inverting amplifier for Example 7.10.2.9.

The input impedance is set by R_i. $R_i = 5k\Omega$, therefore $Z_{in} = 5k\Omega$.

$$V_{out} = V_{in}A_v$$

$$A_v = -\frac{R_f}{R_i}$$

$$A_v = -\frac{20k}{5k}$$

$$A_v = -4$$

$$V_{out} = 100mV \times (-4)$$

$$V_{out} = -400mV, \text{ (i.e., inverted)}$$

Example 7.10.2.6

Design an inverting amplifier with a gain of 10 and an input impedance of 15 kΩ. The input impedance tells us what R_i must be

$$Z_{in} = R_i$$

$$R_i = 15k$$

Knowing R_i, solve for R_f:

$$A_v = -\frac{R_f}{R_i}$$

$$R_f = R_i(-A_v)$$

$$R_f = 15k \times (-(-10))$$

$$R_f = 150k$$

Computer Simulation

A Multisim simulation of the result of Example 7.10.2.6 is shown in Figure 7.10.2.10, along with its schematic. This simulation uses the simple dependent source model presented in Chapter Two. The input is set at 0.1V DC for simplicity. Note that the output potential is negative, indicating the inverting action of the amplifier. Also, note that the virtual ground approximation is borne out quite well, with the inverting input potential measuring in the μV region.

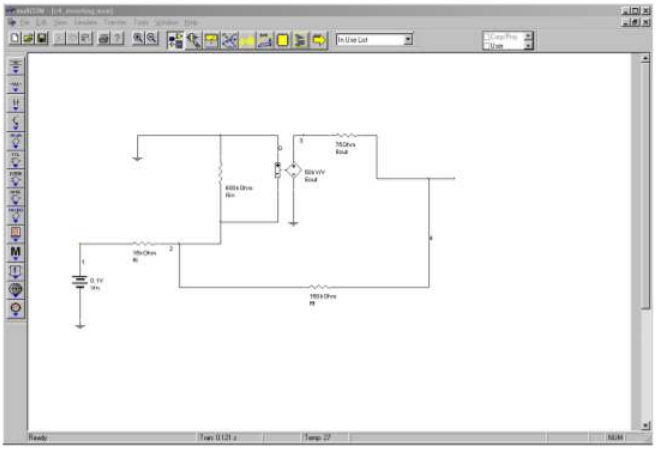


Figure 7.10.2.10a: Multisim simulation of the simple op amp model for Example 7.10.2.6. a. Schematic.

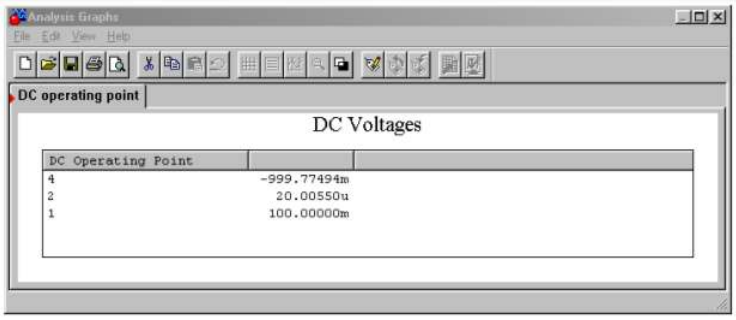


Figure 7.10.2.10*b*: Multisim simulation of the simple op amp model for Example 7.10.2.6. b. Output listing.

Example 7.10.2.7

The circuit of Figure 7.10.2.11 is a pre-amplifier stage for an electronic music keyboard. Like most musicians' pre-amplifiers, this one offers adjustable gain. This is achieved by following the amplifier with a pot. What are the maximum and minimum gain values?

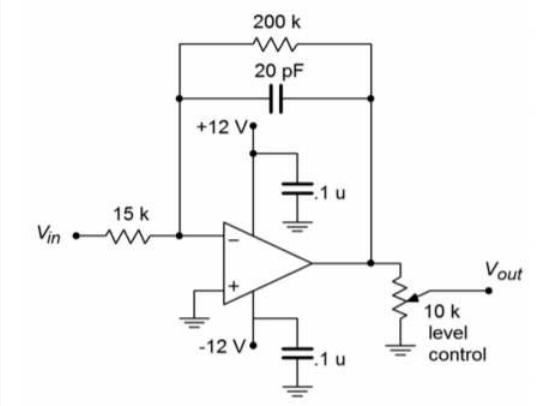


Figure 7.10.2.11: Musical instrument preamplifier for Example 7.10.2.7.

Note that the gain for the pre-amp is the product of the op amp gain and the voltage divider ratio produced by the pot. For maximum gain, use the pot in its uppermost position. Because the pot acts as a voltage divider, the uppermost position provides no divider action (i.e., its gain is unity). For midband frequencies, the 20 pF may be ignored.

$$A_{v-max} = -\frac{R_f}{R_i}$$

$$A_{v-max} = -\frac{200k}{15k}$$

$$A_{v-max} = -13.33$$

$$A_{v-max}^{'} = 22.5dB$$

For minimum gain, the pot is dialed to ground. At this point, the divider action is infinite, and thus the minimum gain is 0 (resulting in silence).

Z_{in} for the system is about 15 kΩ. As far as the extra components are concerned, the 20 pF capacitor is used to decrease high frequency gain. The two 0.1 μF bypass capacitors across the power supply lines are very important. Virtually all op amp circuits use bypass capacitors. Due to the high gain nature of op amps, it is essential to have good AC grounds at the power supply pins. At higher frequencies the inductance of power supply wiring may produce a sizable impedance. This impedance may create a positive feedback loop that wouldn't exist otherwise. Without the bypass capacitors, the circuit may oscillate or produce spurious output signals. The precise values for the capacitors are usually not critical, with 0.1 to 1 μF being typical.

4.2.3: Inverting Current-to-Voltage Transducer

As previously mentioned, the inverting voltage amplifier is based on PP negative feedback, with an extra input resistor used to turn the input voltage into a current. What happens if that extra resistor is left out, and a circuit such as Figure 7.10.2.6 is used? Without the extra resistor, the input is at virtual ground, thus setting Z_{in} to 0 Ω. This is ideal for sensing current. This input current will pass through R_f and produce an output voltage as outlined above. The characteristic of transforming a current to a voltage is measured by the parameter transresistance. By definition, the transresistance of this circuit is the value of R_f. To find V_{out}, multiply the input current by the transresistance. This circuit inverts polarity as well.

$$V_{out} = -I_{in}R_f \tag{7.10.2.3}$$

Example 7.10.2.8

Design a circuit based on Figure 7.10.2.6 if an input current of -50 μA should produce an output of 4 V.

The transresistance of the circuit is R_f

$$R_f = -\frac{V_{out}}{I_{in}}$$

$$R_f = -\frac{4V}{-50\mu A}$$

$$R_f = 80k$$

The input impedance is assumed to be zero.

At first glance, the circuit applications of the topology presented in the prior example seem very limited. In reality, there are a number of linear integrated circuits that produce their output in current form[1]. In many cases, this signal must be turned into a voltage in order to properly interface with other circuit elements. The current-to-voltage transducer is widely used for this purpose.

4.2.4: Noninverting Voltage-to-Current Transducer

This circuit topology utilizes SS negative feedback. It senses an input voltage and produces a current. A conceptual comparison can be made to the FET (a voltage controlled current source). Instead of circuit gain, we are interested in transconductance. In other words, how much input voltage is required to produce a given output current? The op amp circuit presented here drives a floating load. That is, the load is not referenced to ground. This can be convenient in some cases, and a real pain in others. With some added circuitry, it is possible to produce a grounded load version, although space precludes us from examining it here.

A typical voltage-to-current circuit is shown in Figure 7.10.2.12. Because this uses series-input type feedback, we may immediately assume that Z_{in} is infinite. The voltage to current ratio is set by feedback resistor R_i. As V_{error} is assumed to be zero, all of V_{in} drops across R_i, creating current I_{Ri}. The op amp is assumed to have zero input current, so all of I_{Ri} flows through the load resistor, R_l. By adjusting R_i, the load current may be varied.

$$I_{load} = I_{R_i}$$

$$I_{R_i} = \frac{V_{in}}{R_i}$$

$$I_{load} = \frac{V_{in}}{R_i}$$

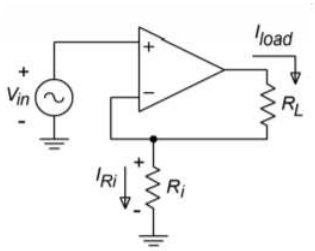


Figure 7.10.2.12: Voltage-to-current transducer.

By definition,

$$g_m = \frac{I_{load}}{V_{in}}$$

$$g_m = \frac{1}{R_i} \tag{7.10.2.4}$$

So, the transconductance of the circuit is set by the feedback resistor. As usual, there are practical limits to the size of R_i. If R_i and R_l are too small, the possibility exists that the op amp will "run out" of output current and go into saturation. At the other extreme, the product of the two resistors and the I_{load} cannot exceed the power supply rails. As an example, if R_i plus R_l is 10 kΩ, I_{load} cannot exceed about 1.5 mA if standard $\pm$15 V supplies are used.

Example 7.10.2.9

Given an input voltage of 0.4 V in the circuit of Figure 7.10.2.13, what is the load current?

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

$$g_m = \frac{1}{20k}$$

$$g_m = 50\mu S$$

$$I_{load} = g_m V_{in}$$

$$I_{load} = 50\mu S \times 0.4V$$

$$I_{load} = 20\mu A$$

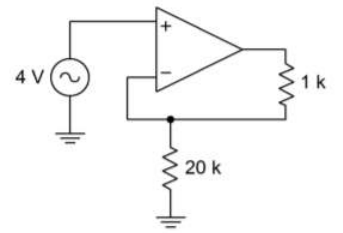


Figure 7.10.2.13: Voltage-to-current transducer for Example 7.10.2.9.

There is no danger of current overload here as the average op amp can produce about 20 mA, maximum. The output current will be 20 μA regardless of the value of R_l, up to clipping. There is no danger of clipping in this situation either. The voltage seen at the output of the op amp to ground is

$$V_{max} = (R_i + R_l) I_{load}$$

$$V_{max} = (20k + 1k) \times 20\mu A$$

$$V_{max} = 420mV$$

That's well below clipping level.

Computer Simulation

A simulation of the circuit of Example 7.10.2.9 is shown in Figure 7.10.2.14. Multisim's ideal op amp model has been chosen to simplify the layout. The load current is exactly as calculated at 20 μA. An interesting trick is used here to plot the load current, as many simulators only offer plotting of node voltages. Using Multisim's Post Processor, the load current is computed by taking the difference between the node voltages on either side of the load resistor and then dividing the result by the load resistance.

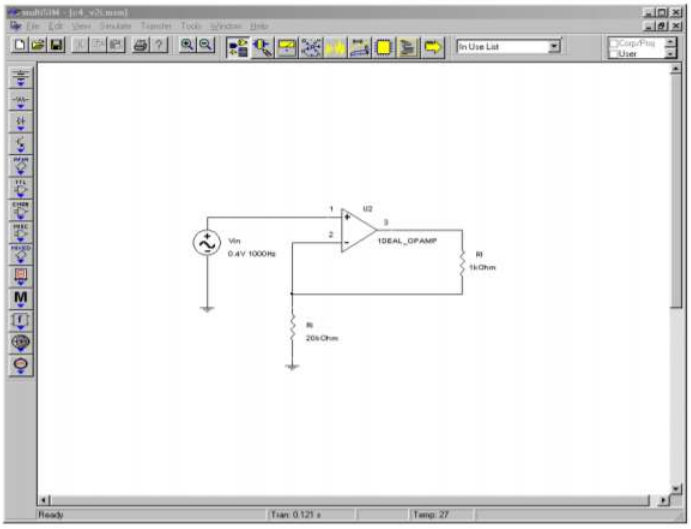


Figure 7.10.2.14a: Voltage-to-current transducer simulation schematic.

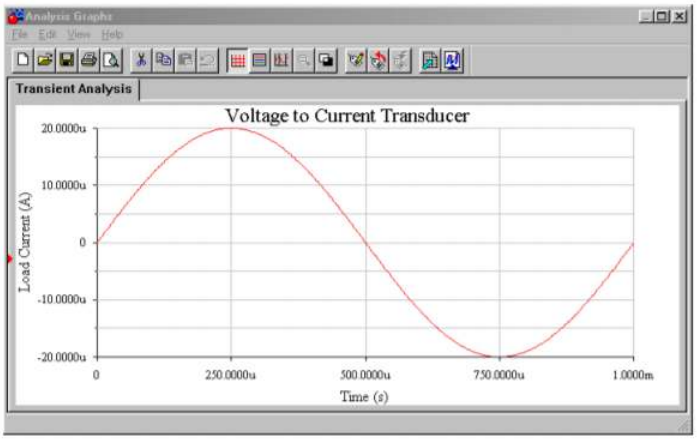


Figure 7.10.2.14b: Simulation results.

Example 7.10.2.10

The circuit of Figure 7.10.2.15 can be used to make a high-input-impedance DC voltmeter. The load in this case is a simple meter movement. This particular meter requires 100 μA for full-scale deflection. If we want to measure voltages up to 10 V, what must R_i be?

First, we must find the transconductance.

$$g_m = \frac{I_{load}}{V_{in}}$$

$$g_m = \frac{100\mu A}{10V}$$

$$g_m = 10\mu S$$

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

$$R_i = \frac{1}{10\mu S}$$

$$R_i = 100k$$

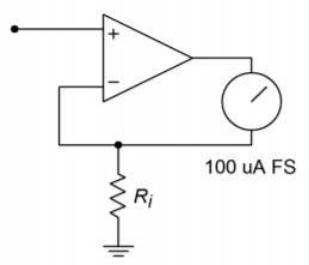


Figure 7.10.2.15: DC voltmeter for Example 7.10.2.10.

The meter deflection is assumed to be linear. For example, if the input signal is only 5 V, the current produced is halved to 50 μ A. 50 μA should produce half-scale deflection. The accuracy of this electronic voltmeter depends on the accuracy of R_i, and the linearity of the meter movement. Note that this little circuit can be quite convenient in a lab, being powered by batteries. In order to change scales, new values of R_i can be swapped in with a rotary switch. For a 1 V scale, R_i equals 10 kΩ. Note that for higher input ranges, some form of input attenuator is needed. This is due to the fact that most op amps may be damaged if input signals larger than the supply rails are used.

4.2.5: Inverting Current Amplifier

The inverting current amplifier uses PS negative feedback. As in the voltage-to-current transducer, the load is floating. The basic circuit is shown in Figure 7.10.2.16. Due to the parallel negative feedback connection at the input, the circuit input impedance is assumed to be zero. This means that the input point is at virtual ground. The current into the op amp is negligible, so all input current flows through R_i to node A. Effectively, R_i and R_f are in parallel (they both share node A and ground; actually virtual ground for R_i). Therefore, V_{Ri} and V_{Rf} are the same value. This means that a current is flowing through R_f, from ground to node A. These two currents join to form the load current. In this manner, current gain is achieved. The larger I_{Rf} is relative to I_{in}, the more current gain there is. As the op amp is sinking current, this is an inverting amplifier.

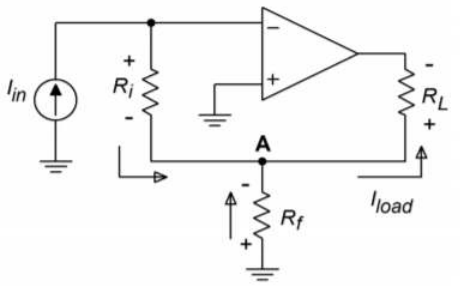


Figure 7.10.2.16: Inverting current amplifier

$$A_i = -\frac{I_{out}}{I_{in}}$$

$$I_{out} = I_{Rf} + I_{Ri} \tag{7.10.2.5}$$

$$I_{Ri} = I_{in}$$

$$I_{Rf} = \frac{V_{Rf}}{R_f}$$

Because V_{Rf} is the same value as V_{Ri},

$$I_{Rf} = \frac{V_{Ri}}{R_f} \tag{7.10.2.6}$$

$$V_{Ri} = I_{in} R_i \tag{7.10.2.7}$$

Substitution of 7.10.2.7 into 7.10.2.6 yields

$$I_{Rf} = \frac{I_{in} R_i}{R_f}$$

Substituting into 7.10.2.5 produces

$$I_{out} = I_{in} + \frac{I_{in} R_i}{R_f}$$

$$I_{out} = I_{in} \left(1 + \frac{R_i}{R_f}\right)$$

$$A_i = -\left(1 + \frac{R_i}{R_f}\right) \tag{7.10.2.8}$$

As you might expect, the gain is a function of the two feedback resistors. Note the similarity of this result with that of the noninverting voltage amplifier.

Example 7.10.2.11

What is the load current in Figure 7.10.2.17?

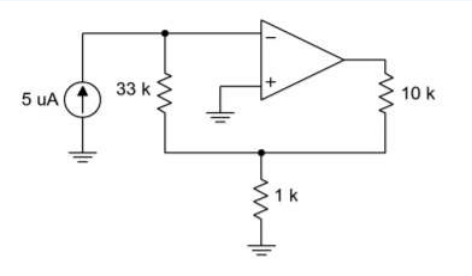


Figure 7.10.2.17: Current amplifier for Example 7.10.2.11.

$$I_{out} = -A_i I_{in}$$

$$A_i = -(1 + \frac{R_i}{R_f})$$

$$A_i = -(1 + \frac{33k}{1k})$$

$$A_i = -34$$

$$I_{out} = -34 \times 5\mu A$$

$$I_{out} = -170\mu A \text{ (sinking)}$$

We need to check to make sure that this current doesn't cause output clipping. A simple Ohm's Law check is all that's needed.

$$V_{max} = I_{out} R_{load} + I_{in} R_i$$

$$V_{max} = 170\mu A \times 10k + 5\mu A \times 33k$$

$$V_{max} = 1.7V + .165V$$

$$V_{max} = 1.865V \text{ (no problem)}$$

Example 7.10.2.12

Design an amplifier with a current gain of -50. The load is approximately 200 kΩ. Assuming a typical op amp ($I_{out-max}$ = 20 mA with ± 15 V supplies), what is the maximum load current obtainable?

$$A_i = -\left(1 + \frac{R_i}{R_f}\right)$$

$$\frac{R_i}{R_f} = -Ai - 1$$

$$\frac{R_i}{R_f} = 50 - 1$$

$$\frac{R_i}{R_f} = 49$$

Therefore, R_i must be 49 times larger than R_f. Possible solutions include:

$$R_i = 49k\Omega, R_f = 1k\Omega$$

$$R_i = 98k\Omega, R_f = 2k\Omega$$

$$R_i = 24.5k\Omega, R_f = 500\Omega$$

As far as maximum load current is concerned, it can be no greater than the op amp's maximum output of 20 mA, but it may be less. We need to determine the current at clipping. Due to the large size of the load resistance, virtually all of the output potential will drop across it. Ignoring the extra drop across the feedback resistors will introduce a maximum of 1% error (that's worst case, assuming resistor set number two).

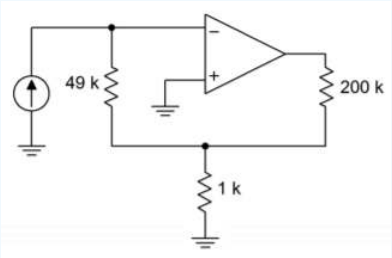


Figure 7.10.2.18: Current amplifier design for Example 7.10.2.12.

With 15 V rails, a typical op amp will clip at 13.5 V. The resulting current is found through Ohm's Law:

$$I_{max} = \frac{13.5V}{200k}$$

$$I_{max} = 67.5\mu A$$

Another way of looking at this is to say that the maximum allowable input current is 67.5 μA/50, or 1.35 μA. One possible solution is shown in Figure 7.10.2.18.

4.2.6: Summing Amplifiers

It is very common in circuit design to combine several signals into a single common signal. One good example of this is in the broadcast and recording industries. The typical modern music recording will require the use of perhaps dozens of microphones, yet the final product consists of typically two output signals (stereo left and right). If signals are joined haphazardly, excessive interference, noise, and distortion may result. The ideal summing amplifier would present each input signal with an isolated load not affected by other channels.

The most common form of summing amplifier is really nothing more than an extension of the inverting voltage amplifier. Because the input to the op amp is at virtual ground, it makes an ideal current summing node. Instead of placing a single input resistor at this point, several input resistors may be used. Each input source drives its own resistor, and there is very little affect from neighboring inputs. The virtual ground is the key. A general summing amplifier is shown in Figure 7.10.2.19.

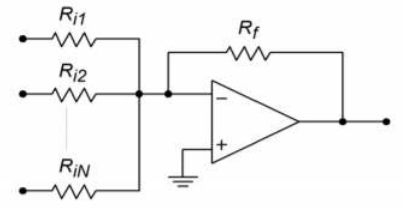


Figure 7.10.2.19: Summing amplifier.

The input impedance for the first channel is R_{i1}, and it's voltage gain is $-R_f/R_{i1}$. For channel 2, the input impedance is R_{i2}, with a gain of $-R_f/R_{i2}$. In general, then for channel N we have

$$Z_{inN} = R_{iN}$$

$$A_{vN} = -\frac{R_f}{R_{iN}}$$

The output signal is the sum of all inputs multiplied by their associated gains.

$$V_{out} = V_{in1}A_{v1} + V_{in2}A_{v2} + \cdots + V_{inN}A_{vN}$$

which is written more conveniently as

$$V_{out} = \sum_{i=1}^{n} V_{in_i}A_{v_i} \tag{7.10.2.9}$$

A summing amplifier may have equal gain for each input channel. This is referred to as an equal-weighted configuration.

Example 7.10.2.13

What is the output of the summing amplifier in Figure 7.10.2.20, with the given DC input voltages?

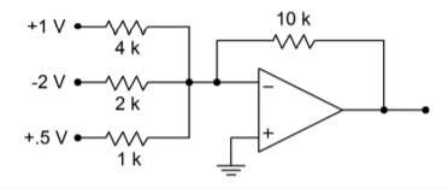


Figure 7.10.2.20: Summing amplifier for Example 7.10.2.13.

The easy way to approach this is to just treat the circuit as three inverting voltage amplifiers, and then add the results to get the final output.

Channel 1:

$$A_v = -\frac{R_f}{R_i}$$

$$A_v = -\frac{10k}{4k}$$

$$A_v = -2.5$$

$$V_{out} = -2.5 \times 1V$$

$$V_{out} = -2.5V$$

Channel 2:

$$A_v = -\frac{R_f}{R_i}$$

$$A_v = -\frac{10k}{2k}$$

$$A_v = -5$$

$$V_{out} = -5 \times -2V$$

$$V_{out} = 10V$$

Channel 3:

$$A_v = -\frac{R_f}{R_i}$$

$$A_v = -\frac{10k}{1k}$$

$$A_v = -10$$

$$V_{out} = -10 \times .5V$$

$$V_{out} = -5V$$

The final output is found via summation:

$$V_{out} = -2.5V + 10V + (-5V)$$

$$V_{out} = 2.5V$$

If the inputs were AC signals, the summation is not quite so straightforward. Remember, AC signals of differing frequency and phase do not add coherently. You can perform a calculation similar to the preceding to find the peak value, however, an RMS calculation is needed for the effective value (i.e., square root of the sum of the squares).

For use in the broadcast and recording industries, summing amplifiers will also require some form of volume control for each input channel and a master volume control as well. This allows the levels of various microphones or instruments to be properly balanced. Theoretically, individual-channel gain control may be produced by replacing each input resistor with a potentiometer. By adjusting R_i, the gain may be directly varied. In practice, there are a few problems with this arrangement. First of all, it is impossible to turn a channel completely off. The required value for R_i would be infinite. Second, because R_i sets the input impedance, a variation in gain will produce a Z_{in} change. This change may overload or alter the characteristics of the driving source. One possible solution is to keep R_i at a fixed value and place a potentiometer before it, as in Figure 7.10.2.21. The pot produces a gain from 1 through 0. The R_f/R_i combo is then set for maximum gain. As long as R_i is several times larger than the pot's value, the channel's input impedance will stay relatively constant. The effective Z_{in} for the channel is R_{pot} in parallel with R_i, at a minimum, up to R_{pot}.

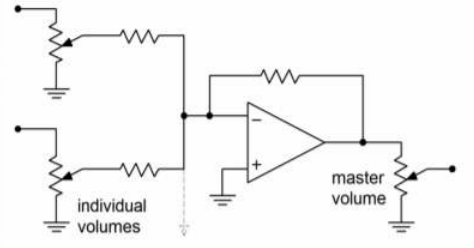


Figure 7.10.2.21: Audio mixer.

As far as a master volume control is concerned, it is possible to use a pot for R_f. Without a limiting resistor though, a very low master gain runs the risk of over driving the op amp due to the small effective R_f value. This technique also causes variations in offset potentials and circuit bandwidth. A technique that achieves higher performance involves using a stage with a fixed R_f value, followed with a pot, as in Figure 7.10.2.21.

Still another application of the summing amplifier is the level shifter. A level shifter is a two-input summing amplifier. One input is the desired AC signal, and the second input is a DC value. The proper selection of DC value lets you place the AC signal at a

desired DC offset. There are many uses for such a circuit. One possible application is the DC offset control available on many signal generators.

4.2.7: Noninverting Summing Amplifier

Besides the inverting form, summing amplifiers may also be produced in a noninverting form. Noninverting summers generally exhibit superior high frequency performance when compared to the inverting type. One possible circuit is shown in Figure 7.10.2.22. In this example, three inputs are shown, although more could be added. Each input has an associated input resistor. Note that it is not possible to simply wire several sources together in hopes of summing their respective signals. This is because each source will try to bring its output to a desired value that will be different from the values created by the other sources. The resulting imbalance may cause excessive (and possibly damaging) source currents. Consequently, each source must be isolated from the others through a resistor.

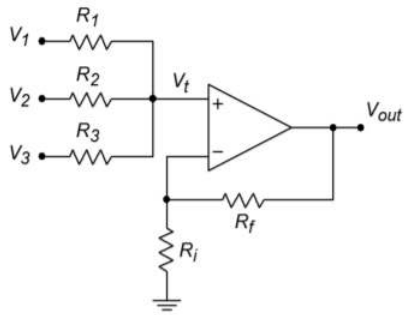


Figure 7.10.2.22: Noninverting summing amplifier.

In order to understand the operation of this circuit it is best to break it into two parts: the input source/resistor section and the noninverting amplifier section. The input signals will combine to create a total input voltage, V_t. By inspection, you should see that the output voltage of the circuit will equal V_t times the noninverting gain, or

$$V_{out} = V_t \left(1 + \frac{R_f}{R_i}\right)$$

All that remains is to determine V_t. Each of the input channels contributes to V_t in a similar manner, so the derivation of the contribution from a single channel will be sufficient.

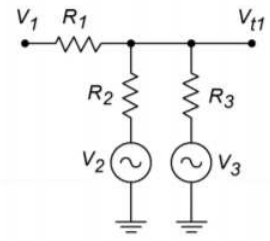


Figure 7.10.2.23: Channel 1 input equivalent circuit.

Unlike the inverting summer, the noninverting summer does not take advantage of the virtual-ground summing node. The result is that individual channels will affect each other. The equivalent circuit for channel 1 is redrawn in Figure 7.10.2.23. Using superposition, we would first replace the input generators of channels 2 and 3 with short circuits. The result is a simple voltage divider between V_1 and V_{t1}.

$$V_{t1} = V_1 \frac{R_2 || R_3}{R_1 + R_2 || R_3}$$

In a similar manner, we can derive the portions of V_t due to channel 2

$$V_{t2} = V_2 \frac{R_1||R_3}{R_2 + R_1||R_3}$$

and due to channel 3

$$V_{t3} = V_3 \frac{R_1||R_2}{R_3 + R_1||R_2}$$

V_t is the summation of these three portions.

$$V_t = V_{t1} + V_{t2} + V_{t3}$$

Thus, by combining these elements, we find that the output voltage is

$$V_{out} = \left(1 + \frac{R_f}{R_i}\right)\left(V_1 \frac{R_2||R_3}{R_1 + R_2||R_3} + V_2 \frac{R_1||R_3}{R_2 + R_1||R_3} + V_3 \frac{R_1||R_2}{R_3 + R_1||R_2}\right)$$

For convenience and equal weighting, the input resistors are often all set to the same value. This results in a circuit that averages together all of the inputs. Doing so simplifies the Equation to

$$V_{out} = \left(1 + \frac{R_f}{R_i}\right)\frac{V_1 + V_2 + V_3}{3}$$

or in a more general sense,

$$V_{out} = \left(1 + \frac{R_f}{R_i}\right)\frac{\sum_{i=1}^{n} V_n}{n} \tag{7.10.2.10}$$

where n is the number of channels.

One problem still remains with this circuit, and that is inter-channel isolation or crosstalk. This can be eliminated by individually buffering each input, as shown in Figure 7.10.2.24.

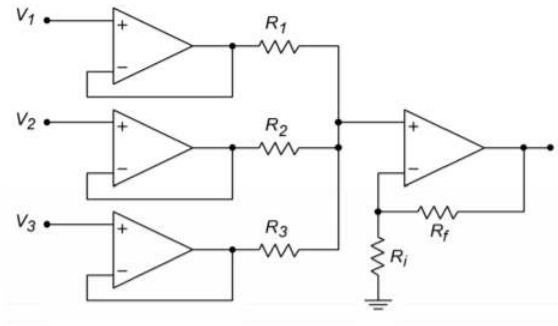


Figure 7.10.2.24: Buffered and isolated noninverting summing amplifier.

Example 7.10.2.14

A noninverting summer such as the one shown in Figure 7.10.2.22 is used to combine three signals. V_1 = 1 V DC, V_2 = -0.2 V DC, and V_3 is a 2 V peak 100 Hz sine wave. Determine the output voltage if $R_1 = R_2 = R_3 = R_f$ = 20 kΩ and R_i = 5 kΩ.

Because all of the input resistors are equal, we can use the general form of the summing equation.

$$V_{out} = \left(1 + \frac{R_f}{R_i}\right)\frac{V_1 + V_2 + \dpts + V_n}{\text{Number of channels}}$$

$$V_{out} = \left(1 + \frac{20k}{5k}\right)\frac{1VDC + (-0.2VDC) + 2\sin 2\pi 100t}{3}$$

$$V_{out} = 5\frac{0.8VDC + 2\sin 2\pi 100t}{3}$$

$$V_{out} = 1.33VDC + 3.33\sin 2\pi 100t$$

So we see that the output is a 3.33 V peak sine wave riding on a 1.33 V DC offset.

4.2.8: Differential Amplifier

As long as the op amp is based on a differential input stage, there is nothing preventing you from making a diff amp with it. The applications of an op amp based unit are the same as the discrete version examined in Chapter One. In essence, the differential amplifier configuration is a combination of the inverting and noninverting voltage amplifiers. A candidate is seen in Figure 7.10.2.25. The analysis is identical to that of the two base types, and Superposition is used to combine the results. The obvious problem for this circuit is that there is a large mismatch between the gains if lower values are used. Remember, for the inverting input the gain magnitude is R_f/R_i, whereas the noninverting input sees R_f/R_i + 1. For proper operation, the gains of the two halves should be identical. The noninverting input has a slightly higher gain, so a simple voltage divider may be used to compensate. This is shown in Figure 7.10.2.26. The ratio should be the same as the R_f/R_i ratio. The target gain is R_f/R_i, the present gain is 1 + R_f/R_i, which may be written as $(R_f + R_i)/R_i$. To compensate, a gain of $R_f/(R_f + R_i)$ is used.

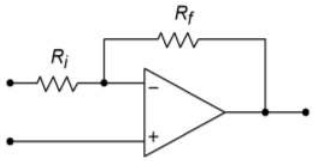


Figure 7.10.2.25: Differential amplifier candidate.

$$A_{v+} = \frac{R_f + R_i}{R_i}\frac{R_f}{R_f + R_i}$$

$$A_{v+} = \frac{R_f(R_f + R_i)}{R_i(R_f + R_i)}$$

$$A_{v+} = \frac{R_f}{R_i} \tag{7.10.2.11}$$

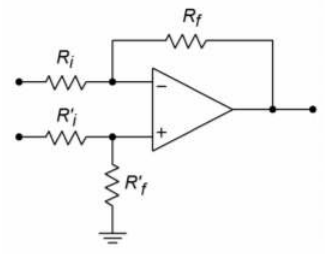


Figure 7.10.2.26: Differential amplifier with compensation for mismatched gains.

For a true differential amplifier, $R_i^{'}$ is set to R_i, and $R_f^{'}$ is set to R_f. A small potentiometer is typically placed in series with $R_f^{'}$ in order to compensate for slight gain imbalances due to component tolerances. This makes it possible for the circuit's common-mode rejection ratio to reach its maximum value. Another option for a simple difference amplifier is to set $R_i^{'}$ plus $R_f^{'}$ equal to R_i. Doing so will maintain roughly equal input impedance between the two halves if two different input sources are used.

Once the divider is added, the output voltage is found by multiplying the differential input signal by R_f/R_i.

Example 7.10.2.15

Design a simple difference amplifier with an input impedance of 10 kΩ per leg, and a voltage gain of 26 dB.

First of all, converting 26 dB into ordinary form yields 20. Because R_i sets Z_{in}, set R_i = 10 kΩ, from the specifications.

$$A_v = \frac{R_f}{R_i}$$

$$R_f = \frac{A_v}{R_i}$$

$$R_f = 20 \times 10k$$

$$R_f = 200k$$

For equivalent inputs,

$$R_i^{'} + R_f^{'} = R_i$$

$$R_i^{'} + R_f^{'} = 10k$$

Given that A_v = 20,

$$R_f^{'} = 20 \times R_i^{'}$$

Therefore,

$$21 \times R_i^{'} = 10k$$

$$R_i^{'} = 476$$

$$R_f^{'} = 20 \times R_i^{'}$$

$$R_f^{'} = 9.52k$$

The final result is shown in Figure 7.10.2.27. As you will see later in Chapter Six, the differential amplifier figures prominently in another useful circuit, the instrumentation amplifier.

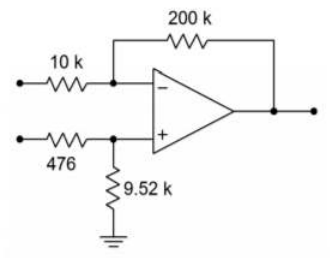


Figure 7.10.2.27: Difference amplifier for Example 7.10.2.15.

4.2.9: Adder/Subtractor

If inverting and noninverting summing amplifiers are combined using the differential amplifier topology, an adder/subtractor results. Normally, all resistors in an adder/subtractor are the same value. A typical adder/subtractor is shown in Figure 7.10.2.28.

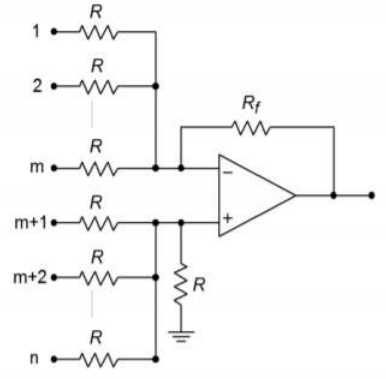


Figure 7.10.2.28: Adder-subtractor.

The inverting inputs number from 1 through m, and the noninverting inputs number from $m+1$ through n. The circuit can be analyzed by combining the preceding proofs of Equations 7.10.2.9 through 7.10.2.11 via the Superposition Theorem. The details are left as an exercise (**Problem 4.45**). When all resistors are equal, the input weightings are unity, and the output is found by:

$$V_{out} = \sum_{i=m+1}^{n} V_{in_i} - \sum_{j=1}^{m} V_{in_j} \tag{7.10.2.12}$$

In essence, you can think of the output voltage in terms of subtracting the inverting input summation from the noninverting input summation.

4.2.10: Adjustable Inverter/Noninverter

A unique adjustable gain amplifier is shown in Figure 7.10.2.29. What makes this circuit interesting is that the gain is continuously variable between an inverting and noninverting maximum. For example, the gain might be set for a maximum of 10. A full turn of the potentiometer would swing the gain from +10 through -10. The exact middle setting would produce a gain of 0. In this way, a single knob controls both the phase and magnitude of the gain.

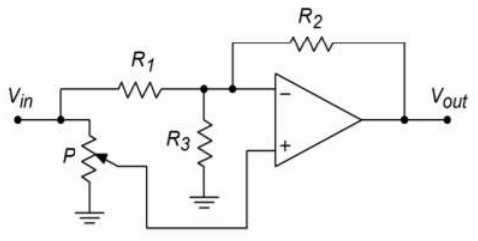


Figure 7.10.2.29: Adjustable inverter/ noninverter.

For the analysis of the circuit, refer to Figure 7.10.2.30.

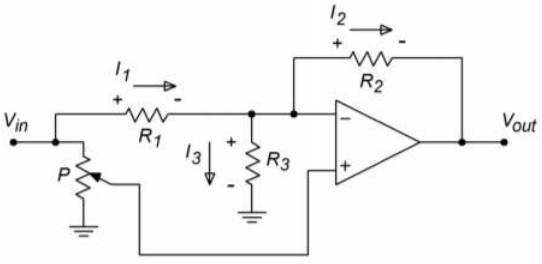


Figure 7.10.2.30: Inverter/noninverter analysis.

As you would expect, the gain of the circuit is defined as the ratio of the output to input voltages. It is important to note that unlike a normal inverting amplifier, the magnitude of the output voltage is not necessarily equal to the voltage across R_2. This is because

the inverting terminal of the op amp is not normally a virtual ground. Instead, the voltage across R_3 must also be considered. Because the two inputs of the op amp must be at approximately the same potential (i.e., V_{error} must be 0), the voltage at the inverting terminal must be the same as the voltage tapped off of the potentiometer. Representing the potentiometer's voltage divider factor as k, we find:

$$V_{out} = kV_{in} - V_{R2} \tag{7.10.2.13}$$

The drop across R_2 is simply I_2R_2. I_2 is found by Kirchhoff's Current Law and the appropriate voltage-resistor substitutions:

$$I_2 = I_1 - I_3$$

$$I_2 = \frac{V_{in} - kV_{in}}{R_1} - \frac{kV_{in}}{R_3}$$

$$I_2 = V_{in}\left(\frac{1-k}{R_1} - \frac{k}{R_3}\right)$$

Thus, V_{R2} is found to be

$$V_{R2} = V_{in}\left((1-k)\frac{R_2}{R_1} - k\frac{R_2}{R_3}\right) 4.14) \tag{7.10.2.14}$$

Combining Equations 7.10.2.13 and ??? and then solving for gain, we find

$$A_v = k - \left((1-k)\frac{R_2}{R_1} - k\frac{R_2}{R_3}\right)$$

$$A_v = k - \left(\frac{R_2}{R_1} - k\frac{R_2}{R_1} - k\frac{R_2}{R_3}\right)$$

$$A_v = -\frac{R_2}{R_1} + k\left(1 + \frac{R_2}{R_1} + k\frac{R_2}{R_3}\right)$$

The value of R_3 is chosen so that $R_1 = R_2||R_3$. This means that

$$R_3 = \frac{1}{\frac{1}{R_1} - \frac{1}{R_2}}$$

Substituting this into our gain Equation and simplifying yields

$$A_v = \frac{R_2}{R_1}(2k-1)$$

In essence, the resistors R_1 and R_2 set the maximum gain. The potentiometer sets k from 0 through 1. If $k = 1$, then $A_v = R_2/R_1$, or maximum noninverting gain. When $k = 0$, then $A_v = -R_2/R_1$, or maximum inverting gain. Finally, when the potentiometer is set to the mid-point, $k = 0.5$ and $A_v = 0$.

References

[1]Most notably, operational transconductance amplifiers and digital to analog converters, which we will examine in Chapters Six and Twelve, respectively.

This page titled 7.10.2: Inverting and Noninverting Amplifiers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.2: Inverting and Noninverting Amplifiers** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.10.3: Single Supply Biasing

Up to this point, all of the example circuits have used a bipolar power supply, usually $\pm$ 15 V. Sometimes this is not practical. For example, a small amount of analog circuitry may be used along with a predominantly digital circuit that runs off a unipolar supply. It may not be economical to create an entire negative supply just to run one or two op amps. Although it is possible to buy op amps that have been specially designed to work with unipolar supplies[1], the addition of simple bias circuitry will allow almost any op amp to run from a unipolar supply. This supply can be up to twice as large as the bipolar counterpart. In other words, a circuit that normally runs off a $\pm$ 15 V supply can be configured to run off of a +30 V unipolar supply, producing similar performance. We will look at examples using both the noninverting and inverting voltage amplifiers

The idea is to bias the input at one-half of the total supply potential. This can be done with a simple voltage divider. A coupling capacitor may be used to isolate this DC potential from the driving stage. For proper operation, the op amp's output should also be sitting at one-half of the supply. This fact implies that the circuit gain must be unity. This may appear to be a very limiting factor, but in reality, it isn't. The thing to remember is that the gain need only be unity for DC. The AC gain can be just about any gain you'd like.

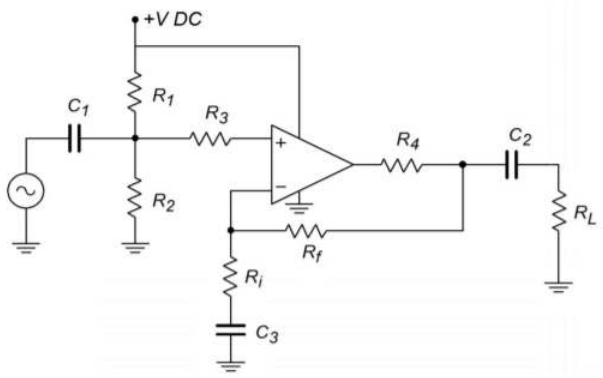


Figure 7.10.3.1: Single-supply bias in a noninverting amplifier.

An example using the noninverting voltage amplifier is shown in Figure 7.10.3.1. In order to set DC gain to unity without affecting the AC gain, capacitor C_3 is placed in series with R_i. R_1 and R_2 establish the 50% bias point. Their parallel combination sets the input impedance too. Resistors R_3 and R_4 are used to prevent destructive discharge of the coupling capacitors C_1 and C_2 into the op amp. They may not be required, but if present, typically run around 1 kΩ and 100 Ω respectively.

The inclusion of the capacitors produces three lead networks. A standard frequency analysis and circuit simplification shows that the approximate critical frequencies are

$$f_{in} = \frac{1}{2\pi C_1 R_1 || R_2}$$

$$f_{out} = \frac{1}{2\pi C_2 R_{load}}$$

$$f_{fdbk} = \frac{1}{2\pi C_3 R_i}$$

The input bias network can be improved by using the circuit of Figure 7.10.3.2. This reduces the hum and noise transmitted from the power supply into the op amp's input. It does so by creating a low impedance at node A. This, of course, does not affect the DC potential. R_5 now sets the input impedance of the circuit.

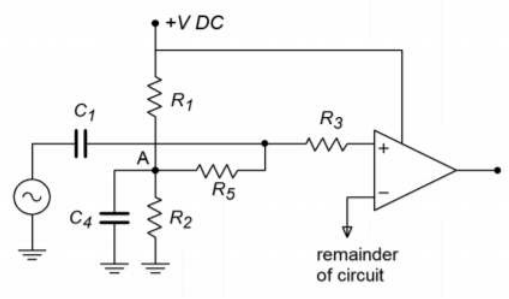


Figure 7.10.3.2: Improved bias for the circuit in Figure 7.10.3.1.

The important points to remember here are that voltage gain is still $1 + R_f/R_i$ in the midband, Z_{in} is now set by the biasing resistors R_1 and R_2, or R_5 (if used), and that frequency response is no longer flat down to zero Hertz.

A single-supply version of the inverting voltage amplifier is shown in Figure 7.10.3.3. It uses the same basic techniques as the noninverting form. The bias setup uses the optimized low-noise form. Note that there is no change in input impedance, it is still set by R_i. The approximate lead network critical frequencies are found through

$$f_{in} = \frac{1}{2\pi C_1 R_i}$$

$$f_{out} = \frac{1}{2\pi C_2 R_{load}}$$

$$f_{bias} = \frac{1}{2\pi C_3 R_1 || R_2}$$

Note the general similarity between the circuits of Figures 7.10.3.3 and 7.10.3.1. A simple redirection of the input signal creates one form from the other.

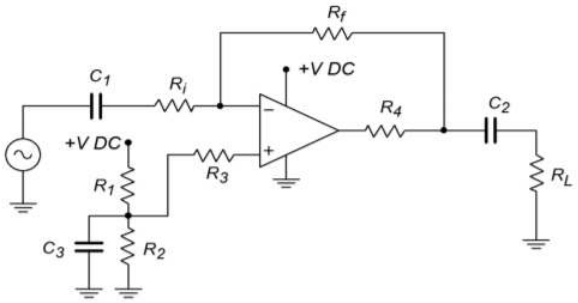


Figure 7.10.3.3: Single-supply inverting amplifier.

This page titled 7.10.3: Single Supply Biasing is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.3: Single Supply Biasing** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.10.4: Current Boosting

As previously noted, general-purpose op amps produce a maximum output current of around 20 mA. This is sufficient for a wide variety of uses. If the load is less than about 1 kΩ, the op amp will start to clip on the higher output signals. The average op amp cannot drive low impedance loads. A few examples of applications whose loads are inappropriate include distribution amplifiers, small audio power amplifiers such as a headphone amplifier, and small motors. This is most unfortunate, as we have already seen how useful these devices can be. There is a way out, though. It is possible to include a current gain stage right after the op amp. All that is needed is a simple class B or class AB push-pull follower. This follower will be able to produce the higher current required by low impedance loads. The op amp only needs to drive the follower stage. In order to increase system linearity and lower distortion, the follower can be placed inside of the op amp's feedback loop. Because the follower is noninverting, there is no problem with maintaining correct feedback (this assumes that the power devices used have a wider bandwidth than the op amp). An example of this is shown in Figure 7.10.4.1.

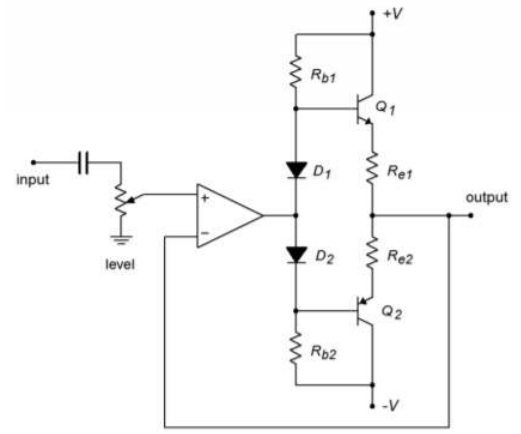


Figure 7.10.4.1: Current boosting.

This circuit is typical of an electronic crossover or distribution amplifier (an electronic crossover design example is given in Chapter Eleven). This output circuit needs to drive relatively low impedances through long cable runs of perhaps several hundred feet. The excessive capacitance resulting from long cable runs increases the current demand above that of a purely resistive load.

Circuits like the one in Figure 7.10.4.1 can produce currents of several hundred milliamps or more. Many times, small resistors are placed in the emitter or collector as a means of limiting maximum current or reducing distortion. The maximum output current limitation is a function of the class B devices. Some manufacturers offer current-boosting ICs to further simplify the design. The current-booster is a drop-in replacement for the class B follower. For very high current demands, Darlingtons or multi-stage designs may be required. It is even possible to provide voltage gain stages. Indeed, several consumer audio power amplifiers have been designed in exactly this way. In essence, the designers produce a discrete power amplifier and then “wrap it” within an op amp feedback loop.

Computer Simulation

A basic current booster is simulated using Multisim in Figure 7.10.4.2. In order to see secondary effects, the LF411 model has been chosen instead of the ideal device model.

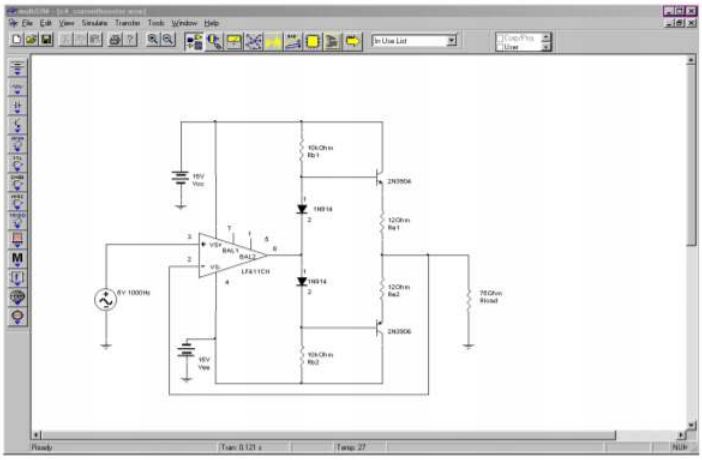


Figure 7.10.4.2*a*: Current booster simulation schematic.

The circuit is configured for a voltage gain of unity, thus the 5 volt input signal should yield a 5 volt output. According to its data sheet, the short-circuit current of the LF411 is approximately 25 mA at room temperature. It is not capable of driving a 75 Ω load to 5 volts by itself. The Transient Analysis shows a full 5 volt output signal, indicating the effectiveness of the current boosting stage. Also, a close inspection of the output waveform shows no obvious forms of distortion such as the cross-over distortion typical of simple class B stages. This shows that keeping the class B stage within the feedback loop does indeed minimize distortion.

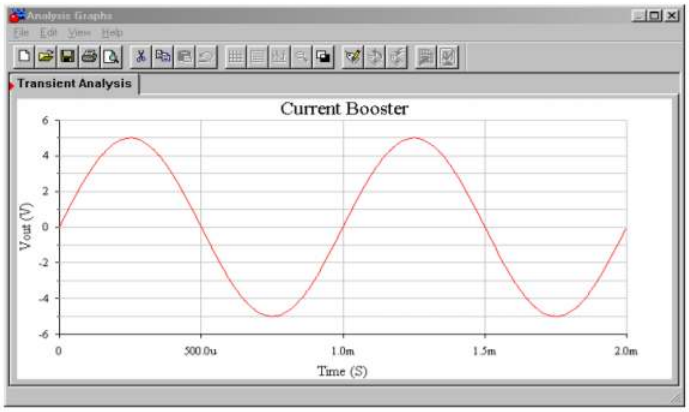


Figure 7.10.4.2*b*: Simulation of output waveform.

This page titled 7.10.4: Current Boosting is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.4: Current Boosting** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.10.5: Summary

In this chapter we have explored a variety of basic op amp circuits and learned a few analysis shortcuts. The basic assumptions are that the error voltage (differential input voltage) is zero and that the op amp's input current is zero. In all circuits, the gain or transfer parameter is a function of just one or two resistors. Circuits can be made that produce voltage gain, current gain, voltage-to-current conversion, or current-to-voltage conversion. The most popular op amp circuits are the noninverting voltage amplifier and the inverting voltage amplifier. These are based on SP and PP negative feedback, respectively. The noninverting type shows an ideally infinite input impedance, whereas the inverting type has its input impedance set by one of the feedback resistors. A variation of the inverting voltage amplifier is the summing amplifier. This adds its several input channels together in order to arrive at its single output signal. The input node is at virtual ground. The differential amplifier is basically the simultaneous use of both the inverting and noninverting voltage amplifier forms.

The voltage-to-current transducer is based on SS feedback. Its transconductance is set by a single feedback resistor. In a similar manner, the current-to-voltage transducer is based on PP feedback and has a single feedback resistor to set its transresistance. The current amplifier is based on PS feedback.

Although op amps are designed to run off bipolar power supplies, they can be used with unipolar supplies. Extra circuitry is needed for the proper bias. There is no restriction on AC gain; however, DC gain must be set to unity. Because lead networks are introduced, the system gain cannot be flat down to zero Hertz.

Finally, if higher output current requirements need to be met, it is possible to boost the op amp's capabilities with a discrete output stage. This stage is typically a class B or class AB push-pull follower. In order to lower system distortion, the follower is kept within the op amp's feedback loop.

This page titled 7.10.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.10.6: Problems

4.6.1: Review Questions

1. What forms of feedback are used for the inverting and noninverting voltage amplifiers?
2. What forms of feedback are used for the current-to-voltage and voltage-to-current transducers?
3. What form of feedback is used for the inverting current amplifier?
4. What are the op amp analysis idealizations?
5. What is virtual ground?
6. What is a summing amplifier?
7. How can output current by increased?
8. What circuit changes are needed in order to bias an op amp with a unipolar supply?
9. What operational parameters change when a circuit is set up for single supply biasing?
10. How might a circuit's gain be controlled externally?
11. What is meant by the term “floating load”?

4.6.2: Problems

Analysis Problems

1. What is the voltage gain in Figure 7.10.6.1? What is the input impedance?

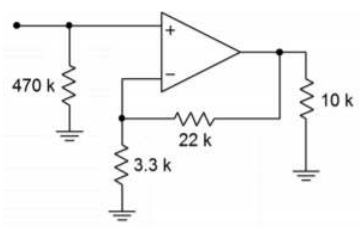


Figure 7.10.6.1

2. What is the voltage gain for the first stage of Figure 7.10.6.2? What is the input impedance?

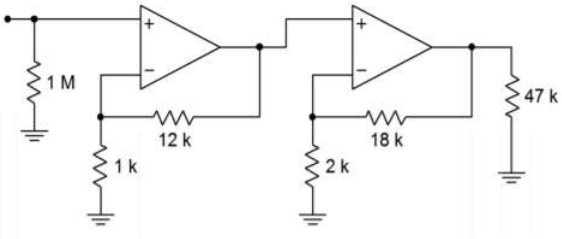


Figure 7.10.6.2

3. What is the voltage gain for the second stage of Figure 7.10.6.2? What is the input impedance?
4. What is the system voltage gain in Figure 7.10.6.2? What is the input impedance?
5. If the input to Figure 7.10.6.2 is -52 dBV, what is $V_{out}^{'}$?
6. What is the voltage gain in Figure 7.10.6.3? What is the input impedance?
7. If the input voltage to the circuit of Figure 7.10.6.3 is 100 mV, what is V_{out}?

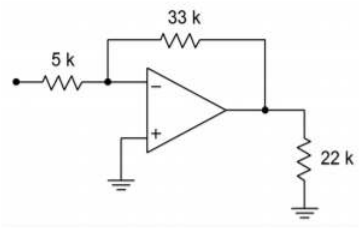


Figure 7.10.6.3

8. What is the system input impedance in Figure 7.10.6.4? What is the system gain?

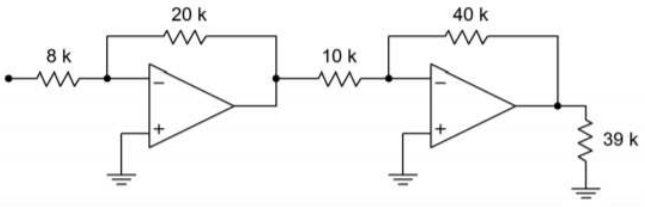


Figure 7.10.6.4

9. Redesign Figure 7.10.6.4 for an input impedance of 20 kΩ.

10. Given an input current of 2 μA, what is the output voltage in Figure 7.10.6.5?

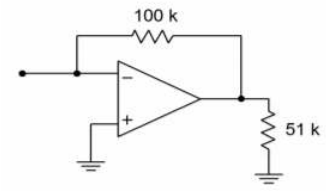


Figure 7.10.6.5

11. What is the meter deflection in Figure 7.10.6.6 if the input voltage is 1 V?

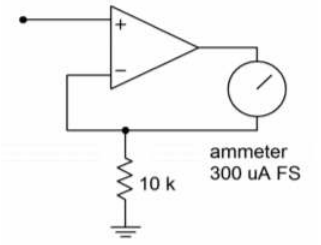


Figure 7.10.6.6

12. What input voltage will produce full-scale deflection in figure 7.10.6.6?

13. Determine a new value for the 10 kΩ resistor in Figure 7.10.6.6 such that a 0.1 V input will produce full-scale deflection.

14. What is the current gain in Figure 7.10.6.7?

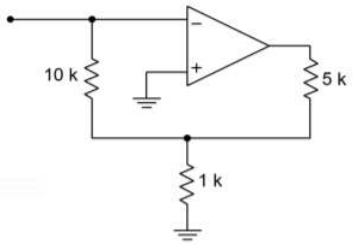


Figure 7.10.6.7

15. What is the maximum input current in Figure 7.10.6.7, assuming the circuit is running off of ±15 V supplies, and the op amp has a maximum output current of 25 mA?

16. If the differential input signal is 300 mV in Figure 7.10.6.8, what is V_{out}?

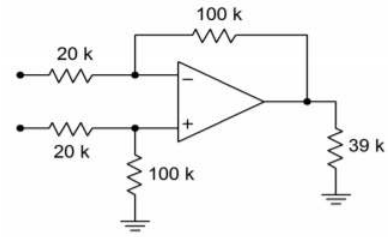


Figure 7.10.6.8

17. Determine new values for the voltage divider resistors in Figure 7.10.6.8, such that the resulting input impedance is balanced.

Design Problems

18. Design a noninverting amplifier with a voltage gain of 32 dB and an input impedance of 200 kΩ.

19. Design a voltage follower with a gain of 0 dB.

20. Design an inverting amplifier with a voltage gain of 14 dB and an input impedance of 15 kΩ.

21. Design a current-to-voltage transducer such that a 20 μA input current will produce a -1 V output.

22. Design a voltage-to-current transducer such that a 100 mV input will produce a 1 mA output.

23. Design a current amplifier with a gain of -20.

24. Design a differential amplifier with a gain of 18 dB and a balanced input impedance of 25 kΩ per input.

25. Design a voltage to current transducer with a transconductance of 1 mS. If V_{in} is 200 mV, what is I_{out}?

26. Design a current to voltage transducer with a transresistance of 10 kΩ. If the input current is 500 μA, what is V_{out}?

27. Redesign the circuit of Figure 7.10.6.1 for single supply operation (don't bother calculating capacitor values).

28. Redesign the circuit of Figure 7.10.6.3 for single supply operation (don't bother calculating capacitor values).

29. Design a summing amplifier such that channel 1 has a gain of 10, channel 2 has a gain of 15, and channel 3 has a gain of 5. The minimum channel input impedance should be 1 kΩ.

30. Determine capacitor values for Problem 27 if the lower break frequency f_1, is set to 20 Hz.

31. Determine capacitor values for Problem 28 if the lower break frequency f1, is set to 10 Hz.

Challenge Problems

32. Design a three channel summing amplifier such that: channel 1 $Z_{in} \geq 10k\Omega$, A_v = 6 dB; channel 2 $Z_{in} \geq 22k\Omega$, A_v = 10 dB; and channel 3 $Z_{in} \geq 5k\Omega$, A_v = 16 dB

33. Assuming 10% resistor values, determine the production gain range for Figure 7.10.6.1.

34. Assuming 5% resistor values, determine the highest gain produced in Figure 7.10.6.2.

35. Design an inverting amplification circuit with a gain of at least 40 dB, and an input impedance of at least 100 kΩ. No resistor used may be greater than 500 kΩ. Multiple stages are allowed.

36. Redesign the circuit of Figure 7.10.6.3 as a voltmeter with 500 mV, 2 V, 5 V, 20 V, and 50 V ranges.

37. Assuming 1% precision resistors and a meter accuracy of 5%, what range of input values may produce a full-scale reading of 2 V, for the circuit of Problem 36?

38. Design an amplifier with a gain range from -10 dB to +20 dB, with an input impedance of at least 10 kΩ.

39. Design an amplifier with a gain range from 0 to 20. The input impedance should be at least 5 kΩ.

40. What is the input impedance in Figure 7.10.6.9? What is A_v?

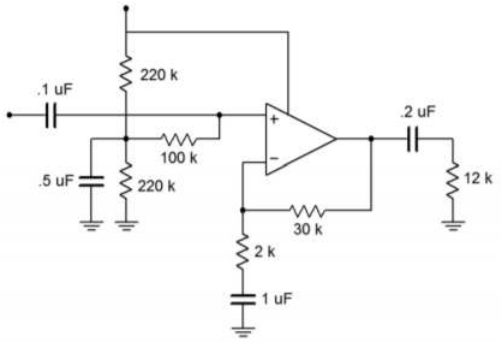


Figure 7.10.6.9

41. How much power supply ripple attenuation does the input biasing network of Figure 7.10.6.9 produce (assume f_{ripple} = 120 Hz)?

42. Assume that the circuit of Figure 7.10.6.10 utilizes a standard 20 mA output op amp. If the output devices are rated for a maximum collector current of 5 amps and a Beta of 50, what is the maximum load current obtainable?

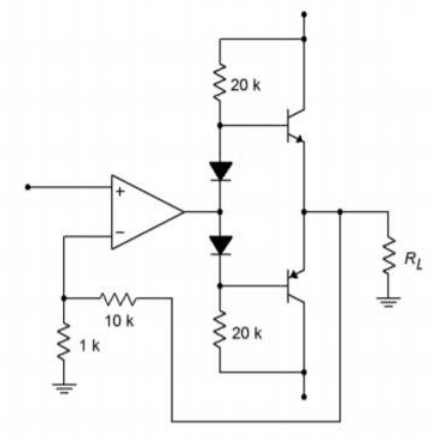


Figure 7.10.6.10

43. What are the voltage gain and input impedance in Figure 7.10.6.10?

44. Given a summer based on Figure 4.2.22, sketch the output waveform if $R_1 = R_2$ = 10 kΩ, $R_3 = R_f$ = 30 kΩ, R_i = 15 kΩ, V_1 = 0.3 V DC, $V_2 = 0.1 \sin 2\pi 50t$ and $V_3 = -0.2 \sin 2\pi 200t$.

45. Prove Equation 4.2.15 for the case when all resistors are of equal value.

Computer Simulation Problems

46. Simulate the operation of the circuit in Figure 4.2.9. Verify the output voltage and the virtual ground at the inverting input.

47. Use a simulator to verify the maximum and minimum gains of the circuit in Figure 4.2.11.

48. Use a simulator to verify the load current and the voltage of the circuit in Figure 4.2.17.

49. Verify the output potential of the circuit in Figure 4.2.20.

50. Simulate the output voltage of the circuit of Figure 7.10.6.4 for the following inputs:

A) $V_{in} = 0.1$ VDC,

B) $V_{in}(t) = 1 \sin 2\pi 10t$,

C) $V_{in} = 5$ VDC.

Also, note the potential at the output of the first stage. How might your op amp model affect the results?

51. Simulate the circuit in Figure 4.2.27. Determine the output potential for the following inputs:

A) $V_{in+}(t) = 0.1 \sin 2\pi 10t$, $V_{in-}(t) = 0.1 \sin 2\pi 10t$,

B) $V_{in+}(t) = 0.1 \sin 2\pi 10t$, $V_{in-}(t) = -0.1 \sin 2\pi 10t$.

52. Simulate the circuit of Figure 7.10.6.10, and determine the output of the circuit and op amp for inputs of 0.1 V DC and 1 V DC.

Use the space below creatively. Draw a cartoon. Compose a song. Write a limerick. Whatever works for you. If you're stuck, try starting with this: ♫

This page titled 7.10.6: Problems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **4.6: Problems** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

SECTION OVERVIEW

7.11: Integrators and Differentiators

Learning Objectives

After completing this chapter, you should be able to:

- Describe the fundamental usefulness and operation of an integrator.
- Describe the fundamental usefulness and operation of a differentiator.
- Detail the modifications required in order to make a practical op amp integrator or differentiator.
- Plot the useful frequency range of a given integrator or differentiator.
- Analyze the operation of integrator circuits using both time-continuous and time-discrete methods.
- Analyze the operation of differentiator circuits using both time-continuous and time-discrete methods.
- Explain the operation of an analog computer.

7.11.1: Introduction

7.11.2: Integrators

7.11.3: Differentiators

7.11.4: Analog Computer

7.11.5: Alternatives to Integrators and Differentiators

7.11.6: Extended Topic- Other Integrator and Differentiator Circuits

7.11.7: Summary

7.11.8: Problems

Template:HideTOC

This page titled 7.11: Integrators and Differentiators is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

7.11.1: Introduction

Up to this point, we have examined a number of different op amp circuits and applications. Viewed from a purely mathematical perspective, the circuits perform basic functions. Amplifiers multiply an input quantity by a constant. Voltage-controlled or transconductance amplifiers can be used to multiply a quantity by a variable. Absolute value and logarithmic functions can also be produced. There is no reason to stop with just these tools; higher mathematical functions may be enlisted. In this chapter we shall examine circuits that perform integration and differentiation. Although these circuits may appear to be somewhat esoteric at first glance, they can prove to be quite useful. Integrators perform the function of summation over time. They may be used whenever an integration function is required, for example, to solve differential equations. The differentiator is the mirror of the integrator and may be used to find rates of change. One possible application is finding acceleration if the input voltage represents a velocity.

Integrators and differentiators may be combined with summing amplifiers and simple gain blocks to form analog computers, that can be used to model physical systems. This can be a valuable aid in the initial design and testing of such things as mechanical suspension systems or loudspeakers. Unlike their digital counterparts, analog computers do not require the use of a programming language, per se. Also, they can respond in real-time
to an input stimulus.

Besides the obvious use as a direct mathematical tool, integrators and differentiators can be used for wave shaping purposes. As an example, the square/triangle generator of Chapter Nine used an integrator. With the exception of simple sinusoids, both the integrator and differentiator will change the fundamental shape of a waveform. This might be contrasted with a simple amplifier that only makes a wave larger and does not alter the basic shape. For practical work, certain alterations will be required to the ideal circuits. The effects of these changes, both good and bad, will also be studied.

This page titled 7.11.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.2: Integrators

The basic operation of an integrator is shown in Figure 7.11.2.1. The output voltage is the result of the definite integral of V_{in} from time = 0 to some arbitrary time t. Added to this will be a constant that represents the output of the network at $t = 0$. Remember, integration is basically the process of summation. You can also think of this as finding the "area under the curve". The output of this circuit always represents the sum total of the input values up to that precise instant in time. Consequently, if a static value (non-zero) is used as an input, the output will continually grow over time. If this growth continues unchecked, output saturation will occur. If the input quantity changes polarity, the output may also change in polarity. Having established the basic operation of the circuit, we are left with the design realization and limitations to be worked out.

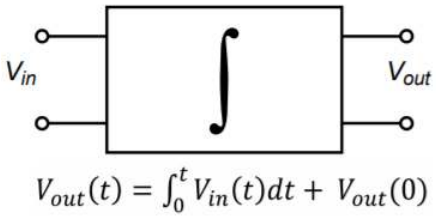


Figure 7.11.2.1: A basic integrator.

As noted in earlier work, the response of an op amp circuit with feedback will reflect the characteristics of the feedback elements. If linear elements are used, the resulting response will be linear. If a logarithmic device is used in the feedback loop, the resulting response will have a log or anti-log character. In order to achieve integration, then, the feedback network requires the use of an element that exhibits this characteristic. In other words, the current through the device must be proportional to either the integral or differential of the voltage across it. Inductors and capacitors answer these requirements, respectively. It should be possible then, to create an integrator with either an inductor or a capacitor. Capacitors tend to behave in a more ideal fashion than do inductors. Capacitors exhibit virtually no problems with stray magnetic field interference, do not have the saturation limitations of inductors, are relatively inexpensive to manufacture, and operate reliably over a wide frequency range. Because of these factors, the vast majority of integrators are made using capacitors rather than inductors. The characteristic Equation for a capacitor is

$$i(t) = C\frac{dv(t)}{dt} \tag{7.11.2.1}$$

This tells us that the current charging the capacitor is proportional to the differential of the input voltage. By integrating Equation 7.11.2.1, it can be seen that the integral of the capacitor current is proportional to the capacitor voltage.

$$v(t) = \frac{1}{C}\int_0^t i(t)dt \tag{7.11.2.2}$$

Assume for a moment that the capacitor voltage is the desired output voltage, as in Figure 7.11.2.2. If the capacitor current can be derived from the input voltage, the output voltage will be proportional to the integral of the input. The circuit of Figure 7.11.2.3 will satisfy our requirements. Note that it is based on the parallel-parallel inverting amplifier studied earlier. The derivation of its characteristic Equation follows.

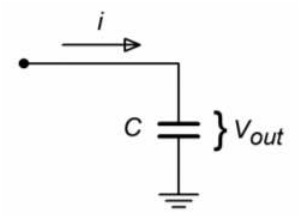


Figure 7.11.2.2: Capacitor as integration element.

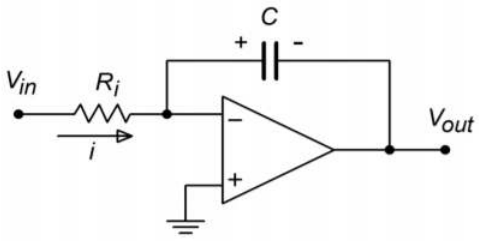


Figure 7.11.2.3: A simple op amp integrator.

First, note that the voltage across the capacitor is equal to the output voltage. This is due to the virtual ground at the inverting op amp terminal. Given the polarities marked, the capacitor voltage is of opposite polarity.

$$V_{out}(t) = -V_c(t) \tag{7.11.2.3}$$

Due to the virtual ground at the op amp's inverting input, all of V_in drops across R_i. This creates the input current, i.

$$i(t) = \frac{V_{in}(t)}{R_i} \tag{7.11.2.4}$$

Because the current drawn by the op amp's inputs is negligible, all of the input current flows into the capacitor, C. Combining Equations 7.11.2.2 and 7.11.2.4 yields

$$V_c(t) = \frac{1}{C}\int \frac{V_{in}(t)}{R_i}dt \tag{7.11.2.5}$$
$$V_c(t) = \frac{1}{R_iC}\int V_{in}(t)dt$$

Combining Equation 7.11.2.5 with Equation 7.11.2.3 produces the characteristic Equation for the circuit.

$$V_{out}(t) = -\frac{1}{R_iC}\int V_{in}(t)dt \tag{7.11.2.6}$$

Note that this Equation assumes that the circuit is initially relaxed (i.e., there is no charge on the capacitor at time = 0). The only difference between Equation 7.11.2.6 and the general Equation as presented in Figure 7.11.2.1 is the preceding constant $-1/R_iC$. If the inversion or magnitude of this constant creates design problems, it can usually be corrected by gain/attenuation networks and/or inverting buffers. Finally, it is worth noting that due to the virtual ground at the inverting input, the input impedance of this circuit is approximately equal to R_i, as is the case with the ordinary parallel-parallel inverting amplifier.

10.2.1: Accuracy and Usefulness of Integration

As long as the circuit operation follows the model presented above, the accuracy of Equation 7.11.2.6 is very high. Small errors occur due to the approximations made. One possible source is the op amp's input bias current. If the input signal current is relatively low, the idealization that all of the input current will bypass the op amp and flow directly into the capacitor is no longer realistic. One possible solution is to use an FET input op amp. The limited frequency response and noise characteristics of the op amp will also play a role in the ultimate circuit accuracy. Generally, slew rate performance is not paramount, as integrators tend to “smooth out” signal variations. As long as the input signal stays within the op amp's frequency limits, well above the noise floor, and the output does not saturate, accurate integration results.

Circuits of this type can be used to model any number of physical processes. For example, the heat stress on a given transducer might be found by integrating the signal across it. The resulting signal could then be compared to a predetermined maximum. If the temperature proved to be too high, the circuit could be powered down as a safety measure. Although the initial reaction might be to directly measure the temperature with some form of thermal sensor, this is not always practical. In such a case, simulation of the physical process is the only reasonable route.

10.2.2: Optimizing the Integrator

For best performance, high-quality parts are a must. Low offset and drift op amps are needed, as the small DC values that they produce at the input will eventually force the circuit into saturation. Although they are not shown in Figure 7.11.2.3, bias and offset

compensation additions are usually required.

Very low offset devices such as chopper-stabilized (or commutating auto-zero) op amps may be used for lower frequency applications. For maximum accuracy, a small input bias current is desired, and thus, FET input op amps should be considered. Also, the capacitor should be a stable, low-leakage type, such as polypropylene film. Electrolytic capacitors are generally a poor choice for these circuits.

Even with high-quality parts, the basic integrator can still prove susceptible to errors caused by small DC offsets. The input offset will cause the output to gradually move towards either positive or negative saturation. When this happens, the output signal is distorted, and therefore, useless. To prevent this it is possible to place a shorting switch across the capacitor. This can be used to re-initialize the circuit from time to time. Although this does work, it is hardly convenient. Before we can come up with a more effective cure, we must first examine the cause of the problem.

We can consider the circuit of Figure 7.11.2.3 to be a simple inverting amplifier, where R_f is replaced by the reactance of the capacitor, X_C. In this case, we can see that the magnitude of the voltage gain is

$$A_v = -\frac{X_C}{R_i} = -\frac{1}{2\pi f C R_i}$$

Because X_C is frequency-dependent, it follows that the voltage gain must be frequency-dependent. Because X_C increases as the frequency drops, the voltage gain must increase as the frequency decreases. To be more specific, the gain curve will follow a −6 dB per octave slope. Eventually, the amplifier's response reaches the open-loop response. This is shown in Figure 7.11.2.4.

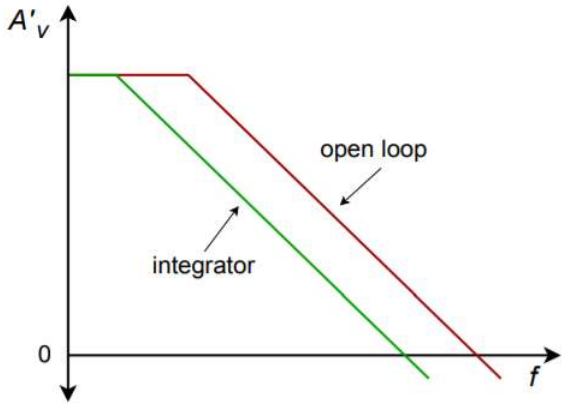


Figure 7.11.2.4: Response of a simple integrator.

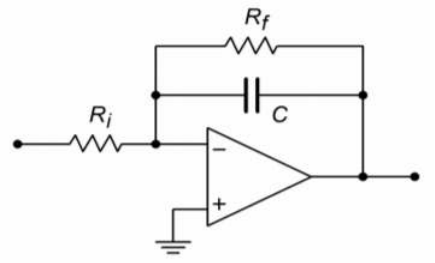


Figure 7.11.2.5: A practical integrator.

The DC gain of the system is at the open loop-level, thus it is obvious that even very small DC inputs can play havoc with the circuit response. If the low-frequency gain is limited to a more modest level, saturation problems can be reduced, if not completely eliminated. Gain limiting may be produced by shunting the integration capacitor with a resistor, as shown in Figure 7.11.2.5. This resistor sets the upper limit voltage gain to

$$A_{max} = -\frac{R_f}{R_i}$$

As always, there will be a trade-off with modification. By definition, integration occurs where the amplitude response rolls off at −6 dB per octave, as this is the response of our idealized capacitor model. Any alteration to the response curve will impact the ultimate accuracy of the integration. By limiting the low frequency gain, the Bode amplitude response will no longer maintain a constant −6 dB per octave rolloff. Instead, the response will flatten out below the critical frequency set by C and R_f, as shown in Figure 7.11.2.6. This critical frequency, f_{low}, is found in the standard manner,

$$f_{low} = \frac{1}{2\pi R_f C} \tag{7.11.2.7}$$

This is a very important frequency. It tells us where the useful integration range starts. If the input frequency is less than f_{low}, the circuit acts like a simple inverting amplifier and no integration results. The input frequency must be higher than f_{low} for useful integration to occur. The question then, is how much higher? A few general rules of thumb are useful. First, if the input frequency equals f_{low}, the resulting calculation will only be about 50% accurate[1]. This means that significant signal amplitude and phase changes will exist relative to the ideal. Second, at about 10 times f_{low}, the accuracy is about 99%. At this point, the agreement between the calculated and actual measured values will be quite solid.

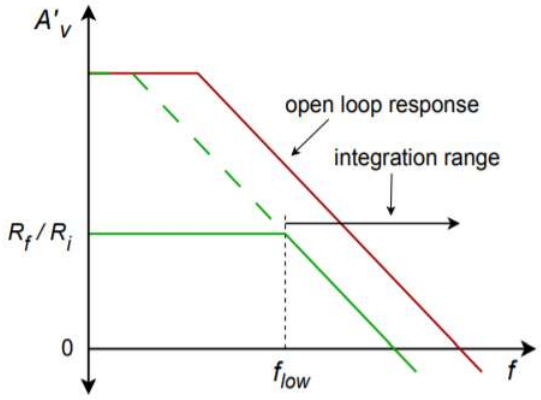


Figure 7.11.2.6: Response of a practical integrator.

Example 7.11.2.1

Determine the Equation for V_{out}, and the lower frequency limit of integration for the circuit of Figure 7.11.2.7.

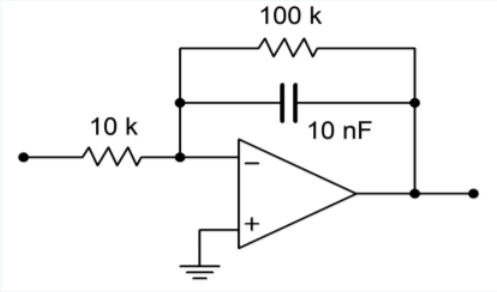


Figure 7.11.2.7: Integrator for Example 7.11.2.1.

The general form of the output Equation is given by Equation 7.11.2.6.

$$V_{out}(t) = -\frac{1}{R_i C}\int V_{in}(t)dt$$
$$V_{out}(t) = -\frac{1}{10\text{ k}\times 10\text{ nF}}\int V_{in}(t)dt$$
$$V_{out}(t) = -10^4\int V_{in}(t)dt$$

The lower limit of integration is set by f_{low}.

https://eng.libretexts.org/@go/page/53016

$$f_{low} = \frac{1}{2\pi R_f C}$$
$$f_{low} = \frac{1}{2\pi 100\ \text{k} \times 10\ \text{nF}}$$
$$f_{low} = 159\ \text{Hz}$$

This represents our 50% accuracy point. For 99% accuracy, the input frequency should be at least one decade above f_{low}, or 1.59 kHz. Accurate integration will continue to higher and higher frequencies.

The upper limit to useful integration is set by two factors: the frequency response of the op amp, and the signal amplitude versus noise. Obviously, at much higher frequencies, the basic assumptions of the circuit's operation will no longer be valid. The use of wide-bandwidth op amps reduces this limitation but cannot eliminate it. All amplifiers will eventually reach an upper frequency limit. Perhaps not so obvious is the limitation caused by signal strength. As noted earlier, the response of the integrator progresses at −6 dB per octave, which is equivalent to −20 dB per decade. In other words, a tenfold increase in input frequency results in a tenfold reduction in output amplitude. For higher frequencies, the net attenuation can be very great. At this point, the output signal runs the risk of being lost in the output noise. The only way around this is to ensure that the op amp is a low-noise type, particularly if a wide range of input frequencies will be used. For integration over a narrow band of frequencies, the RC integration constant may be optimized to prevent excessive loss.

Now that we have examined the basic structure of the practical integrator, it is time to analyze its response to different input waveforms. There are two basic ways of calculating the output: time-continuous or time-discrete. The time-continuous method involves the use of the indefinite integral, and is well suited for simple sinusoidal inputs. Waveforms that require a more complex time domain representation, such as a square wave, may be analyzed with the time-discrete method. This corresponds to the definite integral.

10.2.3: Analyzing Integrators with the Time-Continuous Method

The time-continuous analysis approach involves finding a time-domain representation of the input waveform and then inserting it into Equation 7.11.2.6. The indefinite integral is taken, and the result is the time-domain representation of the output waveform. The constant term produced by the indefinite integral is ignored. As long as the input waveform can be written in a time-domain form, this method may be used. Although any waveform may be expressed in this way, it is not always practical or the most expedient route. Indeed, relatively common waveforms such as square waves and triangle waves require an infinite series time-domain representation. We shall not deal with these waveforms in this manner. The time-continuous integration of these functions is left as an exercise in the Challenge Problems at the end of this chapter.

Example 7.11.2.2

Using the circuit of Figure 7.11.2.7, determine the output if the input is a 1 V peak sine wave at 5 kHz. First, write the input signal as a function time.

$$V_{in}(t) = 1 \sin 2\pi 5000t$$

Substitute this input into Equation 7.11.2.6:

$$V_{out}(t) = -\frac{1}{R_i C} \int V_{in}(t)dt$$
$$V_{out}(t) = -\frac{1}{10\ \text{k} \times 10\ \text{nF}} \int 1 \sin 2\pi 5000t dt$$
$$V_{out}(t) = -10^4 \int 1 \sin 2\pi 5000t dt$$
$$V_{out}(t) = -10^4 \frac{1}{2\pi 5000t} \int 2\pi 5000t \sin 2\pi 5000t dt$$
$$V_{out}(t) = -0.318(-\cos 2\pi 5000t)$$
$$V_{out}(t) = 0.318 \cos 2\pi 5000t$$

Thus, the output is a sinusoidal wave that is leading the input by 90° and is 0.318 V peak. Again, note that the constant produced by the integration is ignored. In Example 1 it was noted that f_{low} for this circuit is 159 Hz. Because the input frequency is over 10 times f_{low}, the accuracy of the result should be better than 99%. Note that if the input frequency is

changed, both the output frequency and amplitude change. For example, if the input were raised by a factor of 10 to 50 kHz, the output would also be at 50 kHz, and the output amplitude would be reduced by a factor of 10, to 31.8 mV. The output would still be a cosine, though.

Computer Simulation

Figure 7.11.2.8 shows the Multisim simulation of the circuit of Example 7.11.2.2. The steady-state response is graphed in Figure 7.11.2.8*b*..

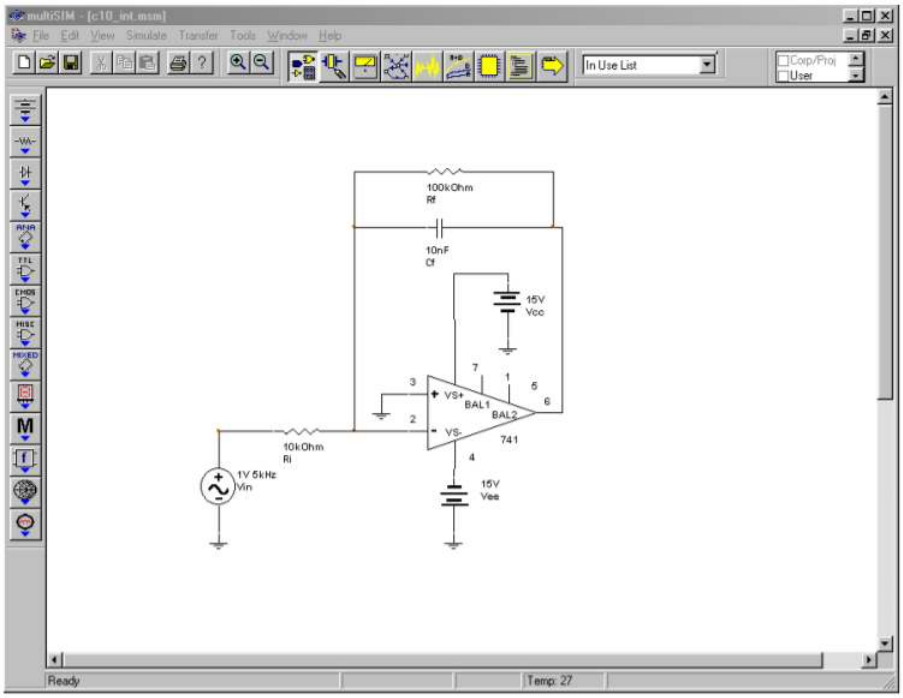


Figure 7.11.2.8*a*: Integrator in Multisim.

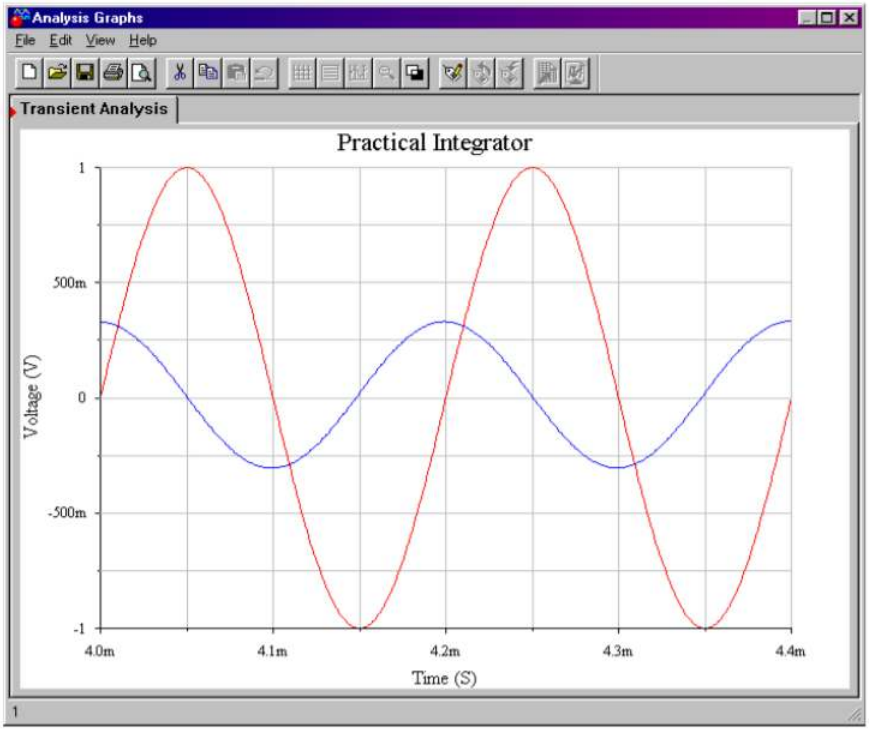


Figure 7.11.2.8*b*: Integrator input and output waveforms.

Note that in order to achieve steady-state results, the output is plotted only after several cycles of the input have passed through. Even after some 20 cycles, the output is not perfectly symmetrical. In spite of this, the accuracy of both the amplitude and phase is quite good when compared to the calculated results. This further reinforces the fact that the circuit is operating within its useful range. It can be instructive to rerun the simulation to investigate the initial response as well.

10.2.4: Analyzing Integrators with the Time-Discrete Method

Unlike the time-continuous approach, this method uses the definite integral and is used to find an output level at specific instances in time. This is useful if the continuous time-domain representation is somewhat complex, and yet the wave shape is relatively simple, as with a square wave. Often, a little logic may be used to ascertain the shape of the resulting wave. The integral is then used to determine the exact amplitude. The time-discrete method also proves useful for more complex waves when modeled within a computer program. Here, several calculations may be performed per cycle, with the results joined together graphically to form the output signal.

The basic technique revolves around finding simple time-domain representations of the input for specific periods of time. A given wave might be modeled as two or more sections. The definite integral is then applied over each section, and the results joined.

Example 7.11.2.3

Sketch the output of the circuit shown in Figure 7.11.2.7 if the input signal is a 10 kHz, 2 V peak square wave with no DC component.

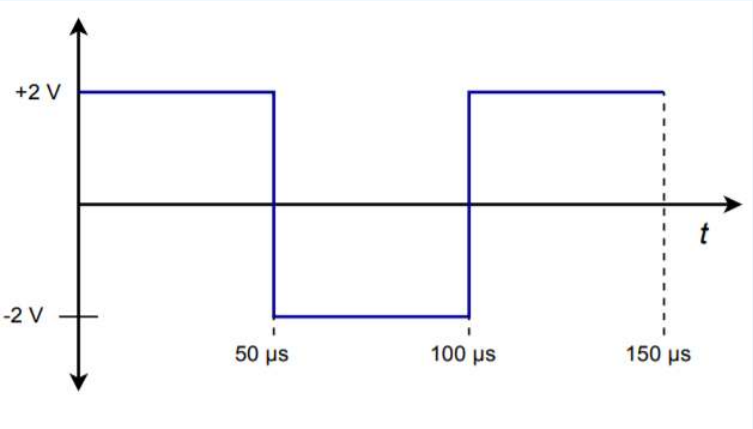


Figure 7.11.2.9: Input waveform for Example 7.11.2.3.

The first step is to break down the input waveform into simple-to-integrate components and ascertain the basic shape of the result. The input is sketched in Figure 7.11.2.9. The input may be broken into two parts: a non-changing 2 V potential from 0 to 50 μs, and a non-changing −2 V potential from 50 μs to 100 μs. The sequence repeats after this.

$$V_{in}(t) = 2 \text{ from } t = 0, \text{ to } t = 50\mu s$$
$$V_{in}(t) = -2 \text{ from } t = 50\mu s, \text{ to } t = 100\mu s$$

In essence, we are saying that the input may be treated as either $\pm$2 V DC for short periods of time. Because integration is a summing operation, as time progresses, the area swept out from underneath the input signal increases, and then decreases due to the polarity changes. It follows that the output will grow (or shrink) in a linear fashion, as the input is constant. In other words, straight ramps will be produced during these time periods. The only difference between them will be the polarity of the slope. From this we find that the expected output waveform is a triangle wave. All we need to do now is determine the peak value of the output. To do this, perform the definite integral using the first portion of the input wave.

$$V_{out}(t) = -\frac{1}{R_i C}\int V_{in}(t)dt$$
$$V_{out}(t) = -\frac{1}{10\text{ k} \times 10\text{ nF}}\int_0^{50\mu s} 2dt$$
$$V_{out}(t) = -10^4 \times 2 \times t|_{t=0}^{t=50\mu s}$$
$$V_{out} = -20000 \times 50\mu s$$
$$V_{out} = -1V$$

This represents the total change over the 50 μs half-cycle interval. This is a peak-to-peak change, saying that the output is 1 V negative relative to its value at the end of the preceding half-cycle. Integration for the second half cycle is similar, and produces a positive change. The resulting waveform is shown in Figure 7.11.2.10.

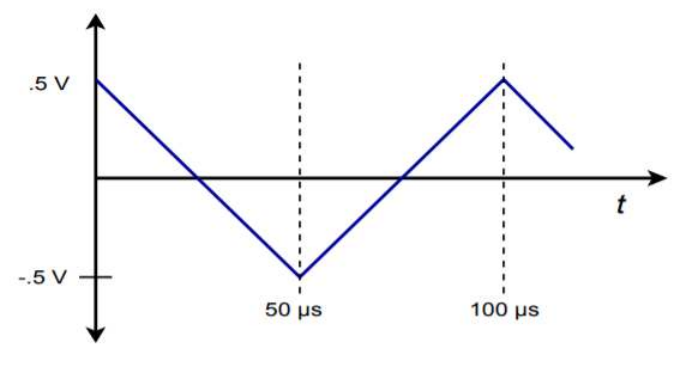


Figure 7.11.2.10: Output waveform for Example 7.11.2.3.

Note that the wave is effectively inverted. For positive inputs, the output slope is negative. Because the input frequency is much higher than f_{low}, we can once again expect high accuracy. Although the solution seems to imply that the output voltage should swing between 0 and −1 V, a real world integrator does indeed produce the indicated $\pm$ 0.5 V swing. This is due to the fact that the integration capacitor in this circuit will not be able to maintain the required DC offset indefinitely.

Example 7.11.2.4

Figure 7.11.2.11a shows an integrator connected to an accelerometer. This device produces a voltage that is proportional to the acceleration it experiences[2]. Accelerometers may be fastened to a variety of physical devices in order to determine how the devices respond to various mechanical inputs. This would be useful, for example, in experimentally determining the mechanical resonance characteristics of a surface. Another possibility is the determination of the lateral acceleration of an automobile as it travels through a corner. By integrating this signal, it is possible to determine velocity, and a further integration will produce position. If the accelerometer produces the voltage shown in Figure 7.11.2.11, determine the shape of the velocity curve (assume a non-repetitive input wave).

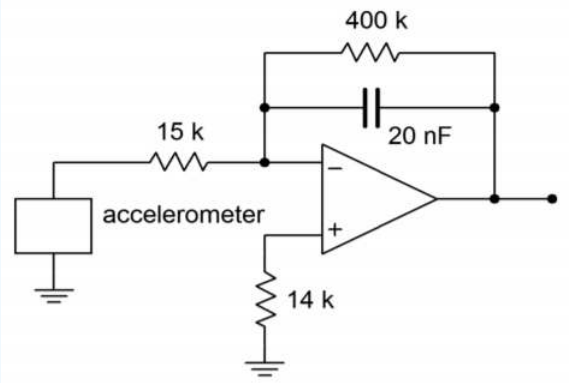


Figure 7.11.2.11a: Accelerometer with integrator circuit.

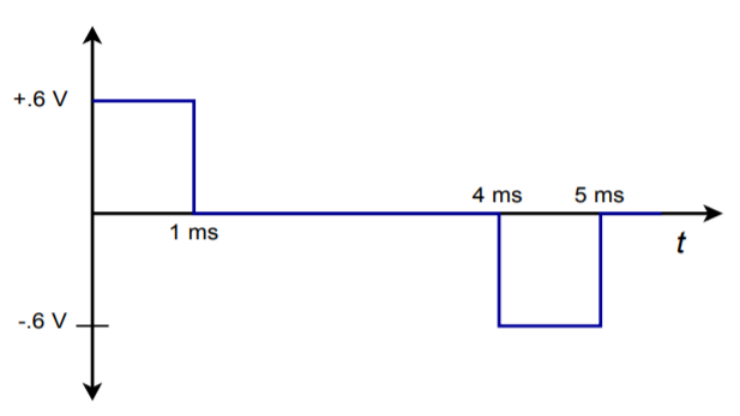


Figure 7.11.2.11b: Signal produced by accelerometer.

First, check the frequency limit of the integrator in order to see if high accuracy may be maintained. If the input waveform was repetitive, it would be approximately 200 Hz (1/ 5 ms).

$$f_{low} = \frac{1}{2\pi R_f C}$$

$$f_{low} = \frac{1}{2\pi 400 \text{ k} \times 20 \text{ nF}}$$

$$f_{low} = 19.9 \text{ Hz}$$

The input signal is well above the lower limit. Note that the input waveshape may be analyzed in piece-wise fashion, as if it were a square wave. The positive and negative portions are both 1 ms in duration, and 0.6 V peak. The only difference is the

polarity. As these are square pulses, we expect ramp sections for the output. For the positive pulse ($t = 0$ to $t = 1$ ms),

$$V_{out}(t) = -\frac{1}{R_i C}\int V_{in}(t)dt$$
$$V_{out}(t) = -\frac{1}{15\text{ k}\times 20\text{ nF}}\int_0^{10^{-3}} .6dt$$
$$V_{out}(t) = -3333\times 0.6\times t|_{t=0}^{t=10^{-3}}$$
$$V_{out} = -2000\times 10^{-3}$$
$$V_{out} = -2V$$

This tells us that we will see a negative going ramp with a −2 V change. For the time period between 1 ms and 4 ms, the input signal is zero, and thus the change in output potential will be zero. This means that the output will remain at −2 V until $t = 4$ ms.

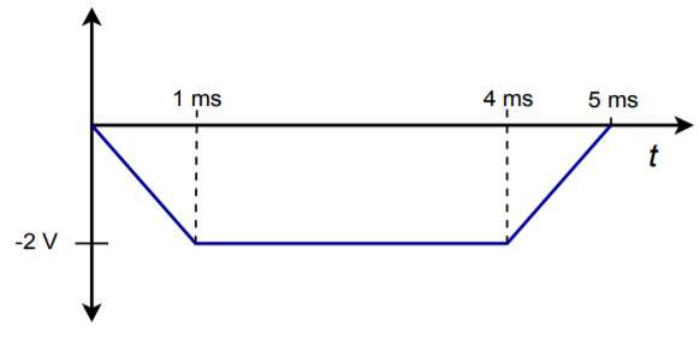


Figure 7.11.2.12: Integrator output.

For the period between 4 ms and 5 ms, a positive going ramp will be produced. Because only the polarity is changed relative to the first pulse, we can quickly find that the change is +2 V. The input waveform is nonrepetitive, so the output waveform appears as shown in Figure 7.11.2.12. (A repetitive input would naturally cause the integrator to “settle” around ground over time, producing the same basic waveshape, but shifted positively.) This waveform tells us that the velocity of the device under test increases linearly up to a point. After this point, the velocity remains constant until a linear decrease in velocity occurs. (Remember, the integrator inverts the signal, so the output curve is effectively upside down. If the input wave indicates an initial positive acceleration, the actual output is an initial positive velocity.) On a different time scale, these are the sort of waveforms that might be produced by an accelerometer mounted on an automobile that starts at rest, smoothly climbs to a fixed speed, and then smoothly brakes to a stop.

References

[1]The term 50% accurate is chosen for convenience. The actual values are 50% of the calculated for power and phase angle and 70.7% of calculated for voltage

[2]A mechanical accelerometer consists of a small mass, associated restoring springs, and some form of transducer that is capable of reading the mass's motion. More recent designs may be fashioned using micro-machined chips, where the deflection of a mass-supporting beam is measured indirectly.

This page titled 7.11.2: Integrators is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.2: Integrators** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.3: Differentiators

Differentiators perform the complementary function to the integrator. The base form of the differentiator is shown in Figure 7.11.3.1. The output voltage is the differential of the input voltage. This is very useful for finding the rate at which a signal varies over time. For example, it is possible to find velocity given distance and acceleration given velocity. This can be very useful in process control work.

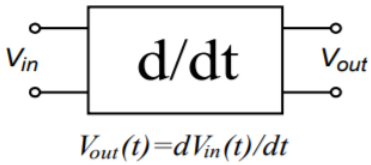


Figure 7.11.3.1: A basic differentiator

Essentially, the differentiator tends to reinforce fast signal transitions. If the input waveform is non-changing, (i.e., DC), the slope is zero, and thus the output of the differentiator is zero. On the other hand, an abrupt signal change such as the rising edge of a square wave produces a very large slope, and thus the output of the differentiator will be large. In order to create the differentiation, an appropriate device needs to be associated with the op amp circuit. This was the approach taken with the integrator, and it remains valid here. In fact, we are left with the same two options: using either an inductor, or a capacitor. Again, capacitors tend to be somewhat easier to work with than inductors and are preferred. The only difference between the integrator and the differentiator is the position of the capacitor. Instead of placing it in the R_f position, the capacitor will be placed in the R_i position. The resulting circuit is shown in Figure 7.11.3.2.

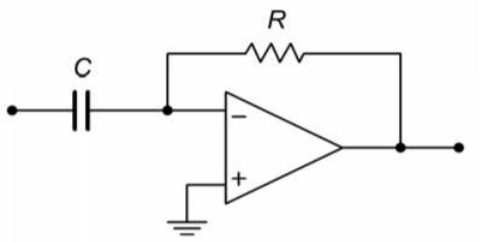


Figure 7.11.3.2: A simple op amp differentiator.

The analysis starts with the basic capacitor Equation (Equation 10.2.1):

$$i(t) = C\frac{dv(t)}{dt}$$

We already know from previous work that the output voltage appears across R_f, though inverted.

$$V_{out} = -V_{Rf}$$

Also, by Ohm's law,

$$V_{Rf} = iR_f$$

By using the approximation that all input current flows through R_f (since the op amp's input current is zero), and then substituting Equation 10.2.1 for the current, we find

$$V_{out}(t) = -R_f C\frac{dv(t)}{dt}$$

A quick inspection of the circuit shows that all of the input voltage drops across the capacitor, because the op amp's inverting input is a virtual ground. Bearing this in mind, we arrive at the final output voltage equation,

$$V_{out}(t) = -R_f C\frac{dV_{in}(t)}{dt} \tag{7.11.3.1}$$

As with the integrator, a leading constant is added to the fundamental form. Again, it is possible to scale the output as required through the use of gain or attenuation networks.

10.3.1: Accuracy and Usefulness of Differentiation

Equation 7.11.3.1 is an accurate reflection of the circuit response so long as the base assumptions remain valid. As with the integrator, practical considerations tend to force limits on the circuit's operating range. If the circuit is analyzed at discrete points in the frequency domain, it can be modeled as an inverting amplifier with the following gain equation:

$$A_v = -\frac{R_f}{X_C} = -2\pi f C R_f$$

Note that as the frequency decreases, X_C grows, thus reducing the gain. Conversely, as the input frequency is raised, X_C falls in value, causing the gain to rise. This rise will continue until it intersects the open-loop response of the op amp. The resulting amplitude response is shown in Figure 7.11.3.3.

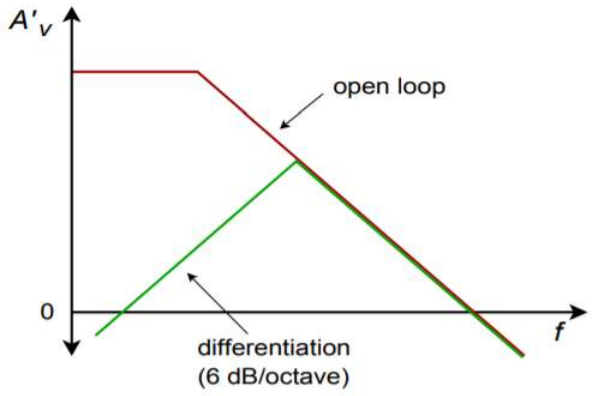


Figure 7.11.3.3: Response of a simple differentiator.

This response is the mirror image of the basic integrator response and exhibits a slope of 6 dB per octave. Note that DC gain is zero, and therefore the problems created by input bias and offset currents are not nearly as troublesome as in the integrator. Because of this, there is no limit as to how low the input frequency may be, excluding the effects of signal-to-noise ratio. Things are considerably different on the high end though. Once the circuit response breaks away from the ideal 6 dB per octave slope, differentiation no longer takes place.

10.3.2: Optimizing the Differentiator

There are a couple of problems with the general differentiator of Figure 7.11.3.2. First of all, it is quite possible that the circuit may become unstable at higher frequencies. Also, the basic shape of the amplitude response suggests that high frequencies are accentuated, thus increasing the relative noise level. Both of these problems may be reduced by providing an artificial upper-limit frequency, f_{high}. This tailoring may be achieved by shunting R_f with a small capacitor. This reduces the high frequency gain, and thus reduces the noise. The resulting response is shown in Figure 7.11.3.4.

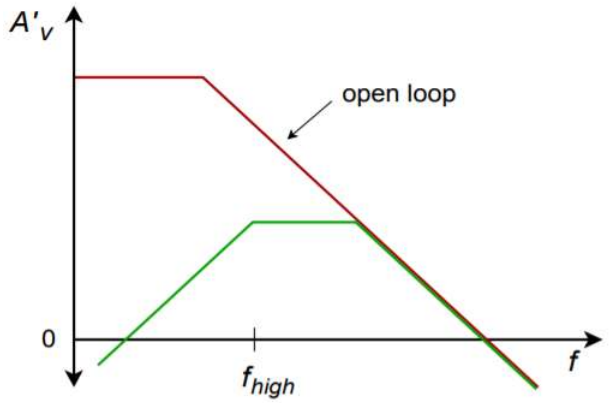


Figure 7.11.3.4: Response of a partially optimized differentiator.

We find f_{high} in the standard manner:

$$f_{high(fdbk)} = \frac{1}{2\pi R_f C_f} \tag{7.11.3.2}$$

f_{high} represents the highest frequency for differentiation. It is the 50% accuracy point. For higher accuracy, the input frequency must be kept well below f_{high}. At about 0.1 f_{high}, the accuracy of Equation 7.11.3.1 is about 99%. Generally, you have to be somewhat more conservative in the estimation of accuracy than with the integrator. This is because complex waves contain harmonics that are higher than the fundamental. Even though the fundamental may be well within the high accuracy range, the upper harmonics may not be.

The other major problem of the basic circuit is that the input impedance is inversely proportional to the input frequency. This is because X_C is the sole input impedance factor. This may present a problem at higher frequencies because the impedance will approach zero. To circumvent this problem, a resistor may be placed in series with the input capacitor in order to establish a minimum impedance value. Unfortunately, this will also create an upper break frequency, f_{high}.

$$f_{high(in)} = \frac{1}{2\pi R_f C} \tag{7.11.3.3}$$

The resulting response is shown in Figure 7.11.3.5. The effective f_{high} for the system will be the lower of Equations 7.11.3.2 and 7.11.3.3.

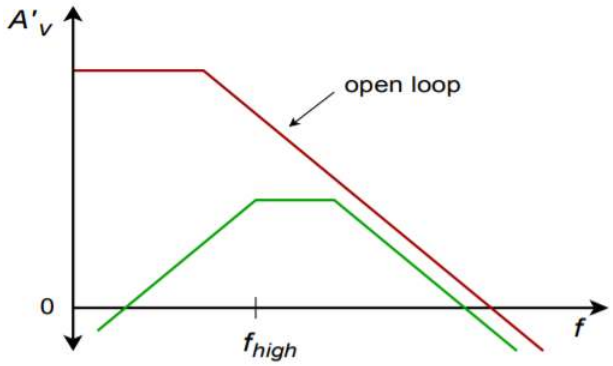


Figure 7.11.3.7: Response of a practical differentiator.

The completed, practical differentiator is shown in Figure 7.11.3.8. Note that a bias compensation resistor may be required at the noninverting input, although it is not shown.

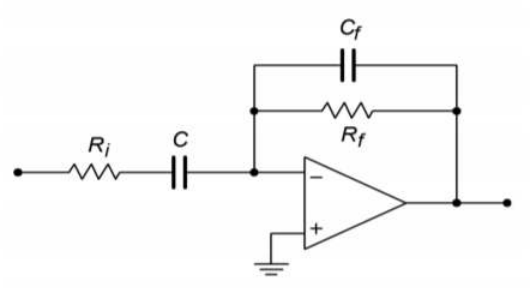


Figure 7.11.3.8: A practical differentiator.

10.3.3: Analyzing Differentiators with the Time-Continuous Method

The time-continuous method will be used when the input signal may be easily written in the time-domain (e.g., sine waves). For more complex waveforms, such as a triangle wave, a discrete time method will be used. The continuous method will lead directly to a time-domain representation of the output waveform. Specific voltage/time coordinates will not be evaluated.

Example 7.11.3.1

Determine the useful range for differentiation in the circuit of Figure 7.11.3.9. Also determine the output voltage if the input signal is a 2 V peak sine wave at 3 kHz.

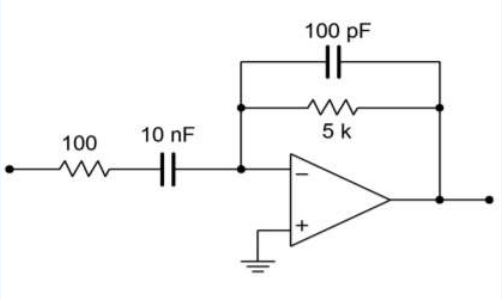


Figure 7.11.3.9: Differentiator for Example 7.11.3.1.

The upper limit of the useful frequency range will be determined by the lower of the two RC networks.

$$f_{high(fdbk)} = \frac{1}{2\pi R_f C_f}$$
$$f_{high(fdbk)} = \frac{1}{2\pi \times 5k \times 100pF}$$
$$f_{high(fdbk)} = 318.3kHz$$
$$f_{high(in)} = \frac{1}{2\pi R_f C}$$
$$f_{high(in)} = \frac{1}{2\pi \times 100 \times 10nF}$$
$$f_{high(in)} = 159.2kHz$$

Therefore, the upper limit is 159.2 kHz. Remember, the accuracy at this limit is relatively low, and normal operation will typically be several octaves lower than this limit. Note that the input frequency is 3 kHz, so high accuracy should result. First, write V_{in} as a time-domain expression:

$$V_{in}(t) = 2\sin 2\pi 3000t$$
$$V_{out}(t) = -R_f C\frac{dV_{in}(t)}{dt}$$
$$V_{out}(t) = -5k \times 10nF\frac{d2\sin 2\pi 3000t}{dt}$$
$$V_{out}(t) = -10^{-4}\frac{d\sin 2\pi 3000t}{dt}$$
$$V_{out}(t) = -1.885\cos 2\pi 3000t$$

This tells us that the output waveform is also sinusoidal, but it lags the input by 90°. Note that the input frequency has not changed, but the amplitude has. The differentiator operates with a 6 dB per octave slope, thus it can be seen that the output amplitude is directly proportional to the input frequency. If this example is rerun with a frequency of 6 kHz, the output amplitude will be double the present value.

Computer Simulation

The Multisim simulation for the circuit of Example 7.11.3.1 is shown in Figure 7.11.3.10. Note the excellent correlation for both the phase and amplitude of the output. As was the case with the integrator simulation, the Transient Analysis output plot is started after the initial conditions have settled.

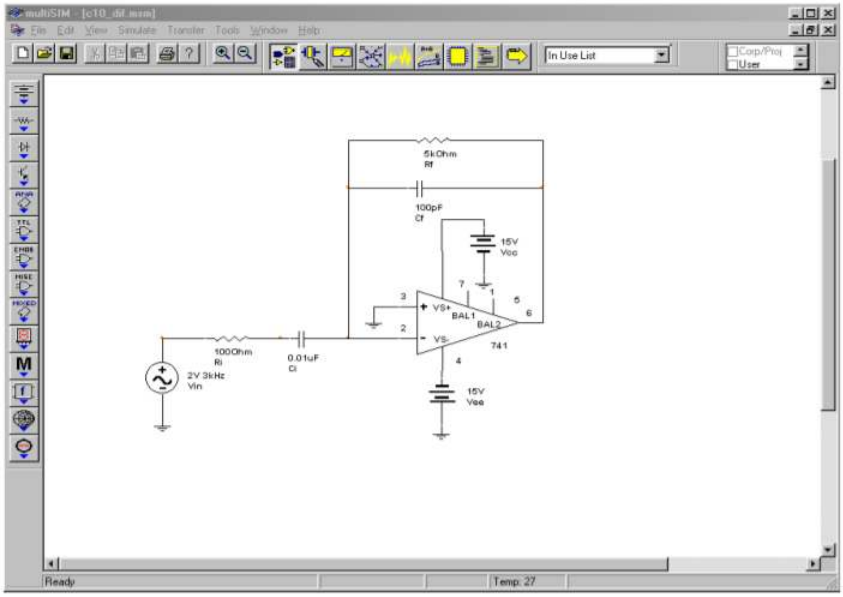


Figure 7.11.3.10a: Differentiator in Multisim.

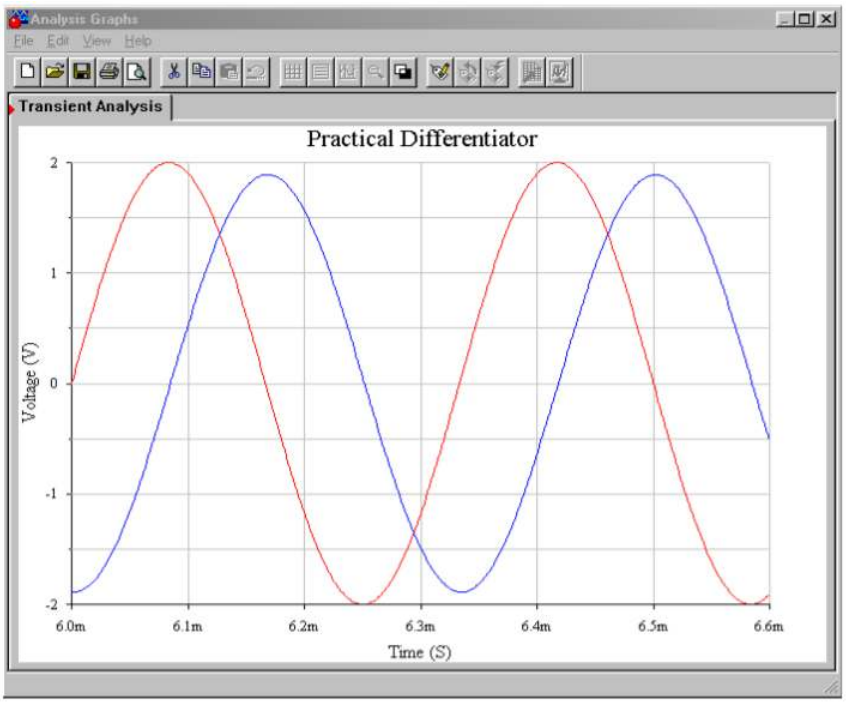


Figure 7.11.3.10b: Differentiator input and output waveforms.

10.3.4: Analyzing Differentiators with the Time-Discrete Method

For more complex waveforms, it is sometimes expedient to break the waveform into discrete chunks, differentiate each portion, and then combine the results. The idea is to break the waveform into equivalent straight-line segments. Differentiation of a straight-line segment will result in a constant (i.e., the slope, which does not change over that time). The process is repeated until one cycle of the input waveform is completed. The resultant levels are then joined together graphically to produce the output waveform.

Often, waveforms are symmetrical, and only part of the calculation need be performed: a sign change is all that will be needed for the mirror image portions. As an example, a triangle wave may be broken into a positive-going line segment and a negative going-line segment. The slopes should be equal, only the direction (i.e., sign) has changed. A square wave may be broken into four parts: a positive-going edge, a static positive value, a negative-going edge, and a static negative value. The "flat" portions have a slope of zero, so only one calculation must be performed, and that's the positive-going edge. We shall take a look at both of these waveforms in the next two examples.

Example 7.11.3.2

Sketch the output waveform for the circuit of Figure 7.11.3.9 if the input is a 3 volt peak triangle wave at 4 kHz.

First, note that the input frequency is well within the useful range of this circuit, as calculated in Example 7.11.3.1. (Note that the highest harmonics will still be out of range, but the error introduced will be minor.)

The triangle wave may be broken into a positive-going portion and a negative-going portion. In either case, the total voltage change will be 6 V in one half-cycle. The period of the waveform is

$$T = \frac{1}{4kHz}$$
$$T = 250\mu s$$

Therefore, for the positive going portion, a 6 volt change will be seen in 125 μs (−6 V in 125 μs for the negative going portion). The slope is

$$Slope = \frac{6V}{125\mu s}$$
$$Slope = 48000V/s$$

Which, as a time-domain expression is

$$V_{in}(t) = 48000t$$

Substituting this Equation into Equation 7.11.3.1 yields

$$V_{out}(t) = -R_f C\frac{dV_{in}(t)}{dt}$$
$$V_{out}(t) = -5k \times 10nF\frac{d48000t}{dt}$$
$$V_{out}(t) = -2.4V$$

During $t = 0$ through $t = 125\mu$s, the output is −2.4 V. Differentiation of the second half of the wave is similar, but produces a positive output, +2.4 V. The result is a 4 kHz square wave that is 2.4 V peak. The resulting waveform is shown in Figure 7.11.3.11.

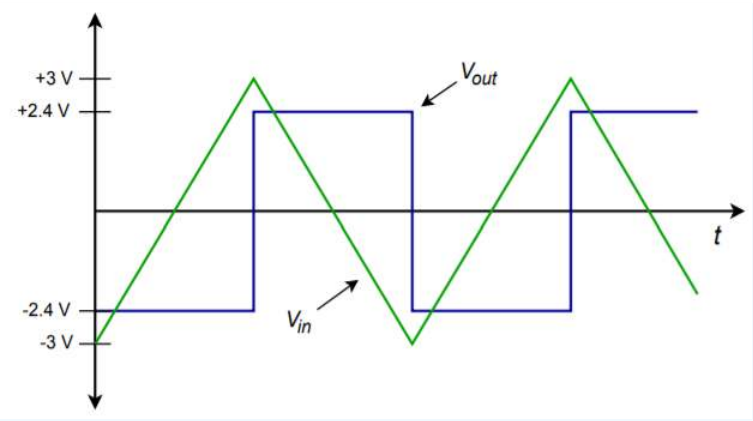


Figure 7.11.3.11: Input/output waveforms.

Computer Simulation

Some circuits are more sensitive to the choice of op amp than others are, and the effects of an improper choice may not always be immediately apparent without first building or simulating the circuit. A good example of this is shown in Figure 7.11.3.12. Multisim was used to create the Transient Analysis for the circuit of Example 7.11.3.2 with two different op amps. Accurate differentiation requires excellent high frequency response from the op amp. In the first simulation, a 741 op amp is used. This device is not particularly fast. As a result, the output waveform suffers from excessive overshoot and ringing. Also, slew rate limiting is fairly obvious, slowing the transitions of the output waveform. In contrast, using an LF411 in the same circuit yields far superior response. Some overshoot still exists, but its magnitude has been restrained, as has the ringing. Also, slew rate limiting is reduced by a wide margin. Clearly, the second result is much closer to the ideal calculation than the first run.

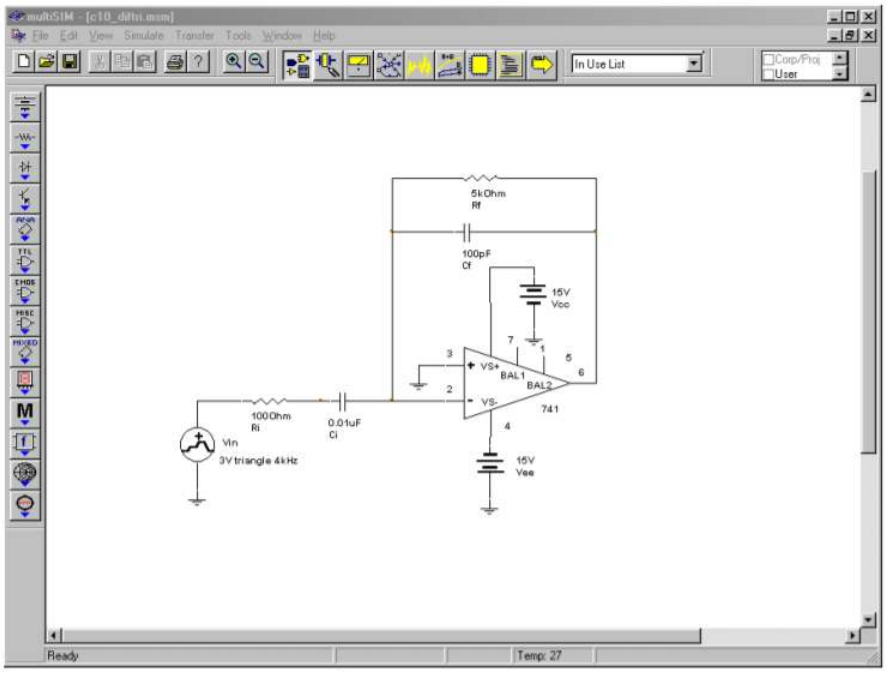


Figure 7.11.3.12*a*: Multisim schematic of differentiator.

https://eng.libretexts.org/@go/page/53017

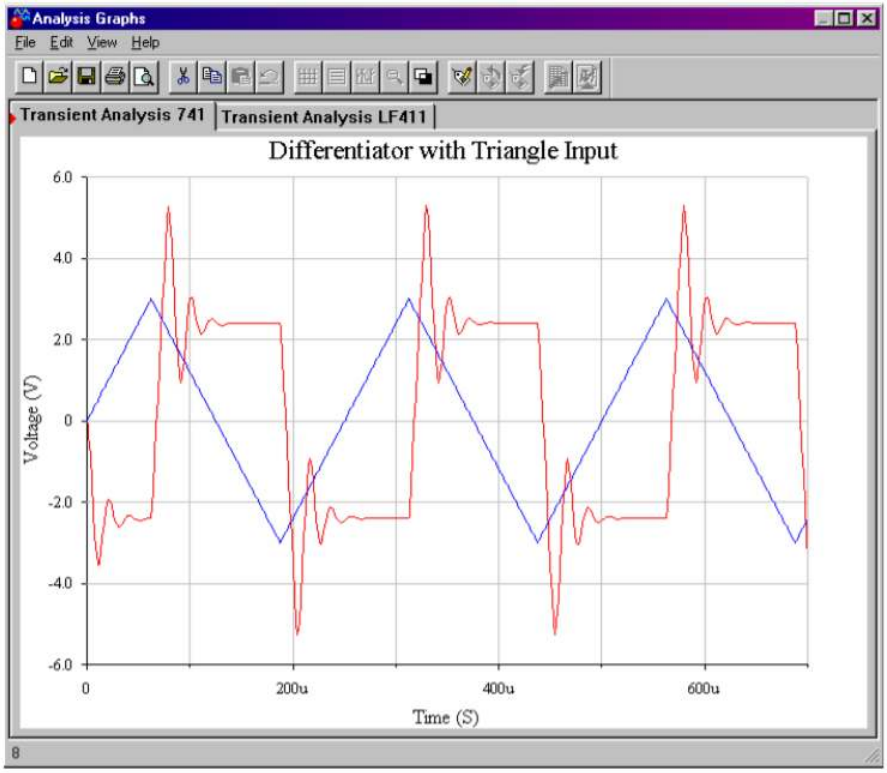


Figure 7.11.3.12*b*: Input and output waveforms using 741.

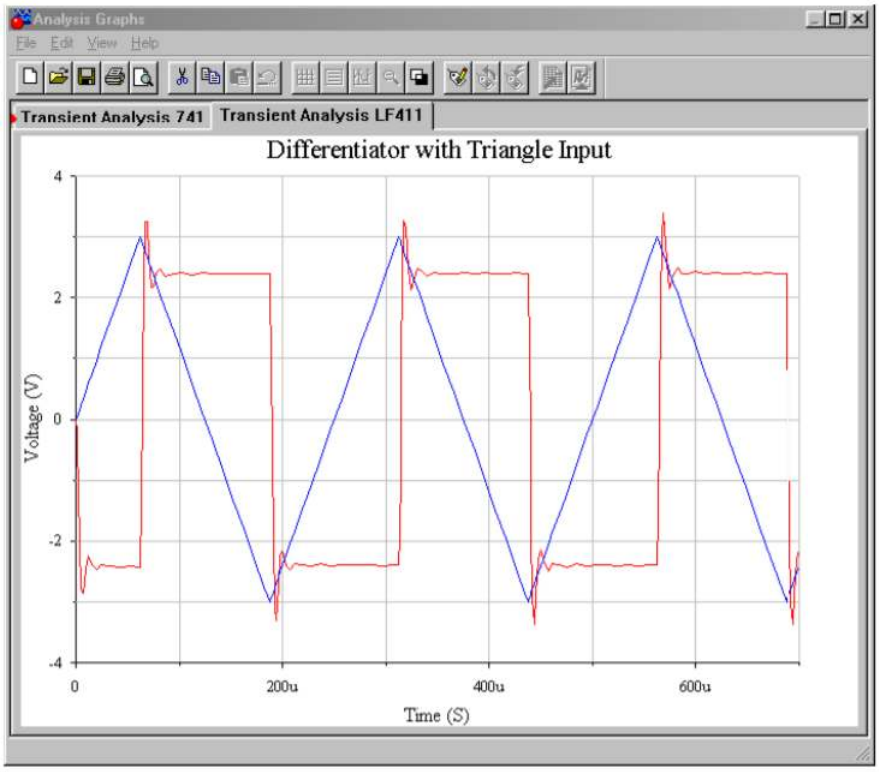


Figure 7.11.3.12*c*: Input and output waveforms using LF411.

Example 7.11.3.3

Repeat Example 7.11.3.2 with a 3 V peak, 4 kHz square as the input. Assume that the rising and falling edges of the square wave have been slew limited to 5 V/μs.

During the time periods that the input is at ± 3 V, the output will be zero. This is because the input slope is, by definition, zero when the signal is “flat”. An output is only noted during the transitions between the ± 3 V levels. Therefore, we need to find the slope of the transitions. It was stated that due to slew rate limiting (perhaps from some previous amplifier stage) the transitions run at 5 V/μs. As a time-domain expression, this is

$$V_{in}(t) = 5 \times 10^6 t$$

Substituting this Equation into Equation 7.11.3.1 yields,

$$V_{out}(t) = -R_f C \frac{dV_{in}(t)}{dt}$$
$$V_{out}(t) = -5k10nF \frac{d5 \times 10^6 t}{dt}$$
$$V_{out}(t) = -250V$$

Obviously, when using a standard op amp and ± 15 V power supply, clipping will occur in the vicinity of −13.5 V. For the negative going edge, a similar result will be seen (+250 V calculated, with clipping at +13.5 V). The resulting waveform is shown in Figure 7.11.3.13. Note that the output waveform spikes will also be limited by the slew rate of the differentiator's op amp.

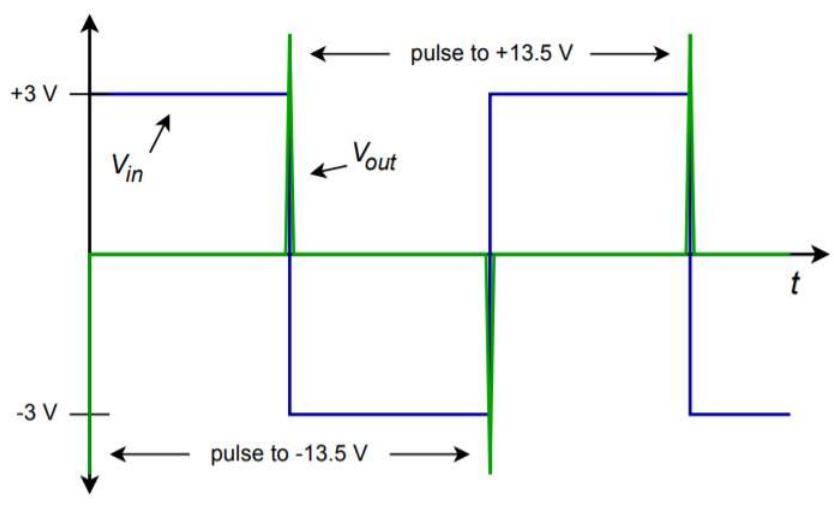


Figure 7.11.3.13: Differentiated square wave (note output clipping).

Example 7.11.3.4

Figure 7.11.3.14 shows a differentiator receiving a signal from an LVDT, or linear variable differential transformer[1]. An LVDT can be used to accurately measure the position of objects with displacements of less than one-thousandth of an inch. This could be useful in a computer-aided manufacturing system. By differentiating this position signal, a velocity signal may be derived. A second differentiation will produce acceleration. If the LVDT produces the wave shown in Figure 7.11.3.14, determine the velocity/time curve for the object being tracked.

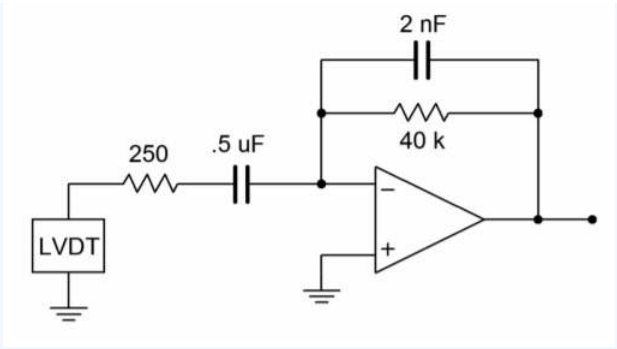


Figure 7.11.3.14a: Differentiator circuit with LVDT.

First, check the upper frequency limit for the circuit.

$$f_{high(fdbk)} = \frac{1}{2\pi R_f C_f}$$
$$f_{high(fdbk)} = \frac{1}{2\pi \times 40k \times 2nF}$$
$$f_{high(fdbk)} = 1.99kHz$$
$$f_{high(in)} = \frac{1}{2\pi R_i C}$$
$$f_{high(in)} = \frac{1}{2\pi \times 250 \times 500nF}$$
$$f_{high(in)} = 1.273kHz$$

The limit will be the lower of the two, or 1.273 kHz. This is well above the slowly changing input signal, and therefore, high accuracy should be possible.

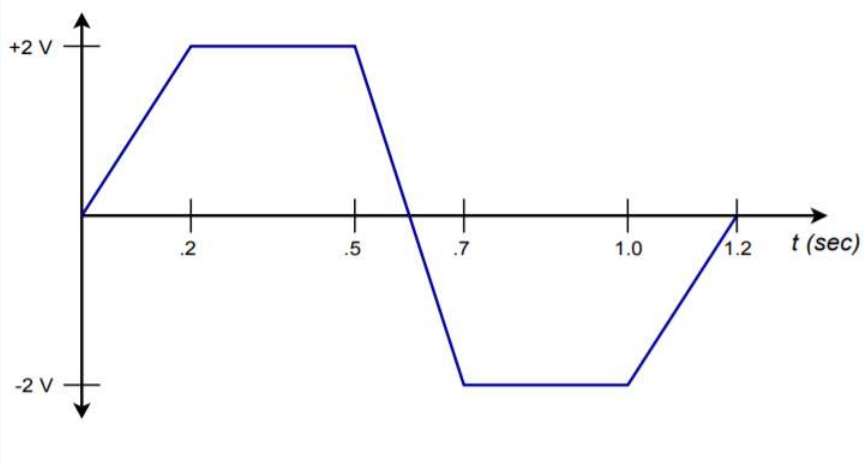


Figure 7.11.3.14b: Signal produced by LVDT.

This wave can be analyzed in piece-wise fashion. The ramp portions will produce constant output levels and the flat portions will produce an output of 0 V (i.e., the rate of change is zero).

For the first section,

$$Slope = \frac{2V}{0.2s}$$
$$Slope = 10V/s$$

Which, as a time-domain expression, is

$$V_{in}(t) = 10t$$
$$V_{out}(t) = -R_f C \frac{dV_{in}(t)}{dt}$$
$$V_{out}(t) = -40k \times 500nF \frac{d10t}{dt}$$
$$V_{out}(t) = -0.2V$$

So, during $t = 0$ through $t = 0.2$ s, the output is −0.2 V. The time period between 0.2 s and 0.5 s will produce an output of 0 V. For the negative-going portion,

$$Slope = \frac{-4V}{0.2s}$$
$$Slope = -20V/s \quad \text{thus,}$$
$$V_{in}(t) = -20t$$
$$V_{out}(t) = -R_f C \frac{dV_{in}(t)}{dt}$$
$$V_{out}(t) = -40k500nF \frac{d20t}{dt}$$
$$V_{out}(t) = 0.4V$$

The output is 0.4 V between 0.5 s and 0.7 s. The third section has the same slope as the first section, and will also produce a −0.2 V level. The output waveform is drawn in Figure 7.11.3.15.

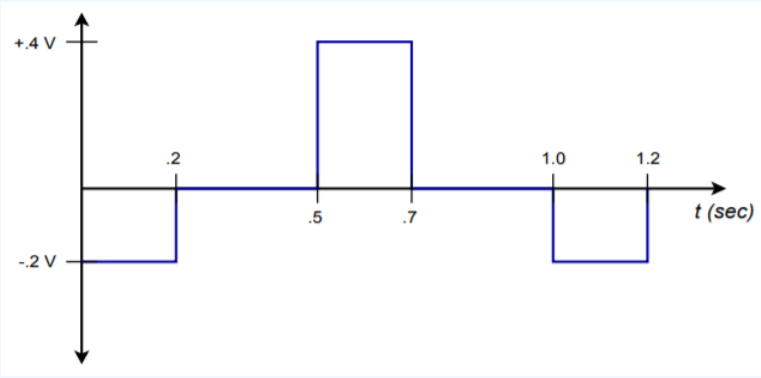


Figure 7.11.3.15: Differentiator output.

References

[1]An LVDT is a transformer with dual secondary windings and a movable core. The core is connected to a shaft, that is in turn actuated by some external object. The movement of the core alters the mutual inductance between primary and secondary. A carrier signal is fed into the primary, and the changing mutual inductance alters the strength of the signal induced into the secondaries. This signal change is turned into a simple DC voltage by a demodulator. The resulting DC potential is proportional to the position of the core, and thus, proportional to the position of the object under measurement.

This page titled 7.11.3: Differentiators is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.3: Differentiators** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.4: Analog Computer

Analog computers are used to simulate physical systems. These systems may be electrical, mechanical, acoustical, or what have you. An analog computer is basically a collection of integrators, differentiators, summers, and amplifiers. Due to their relative stability, integrators are favored over differentiators. It is not uncommon for analog computers to be made without any differentiators. Because physical systems may be described in terms of differential equations, analog computers may be used to solve these equations, thus producing as output some system parameter.

The basic advantage of simulation is that several variations of a given system may be examined in real time without actually constructing the system. For a large project this is particularly cost-efficient. The process starts by writing a differential Equation (first-, second-, or third-order) that describes the system in question. The Equation is then solved for its highest-order element, and the result used to create a circuit.

Example 7.11.4.1

Let's investigate the system shown in Figure 7.11.4.1. This is a simple mechanical system that might represent (to a rough approximation) a variety of physical entities, including the suspension of an automobile.

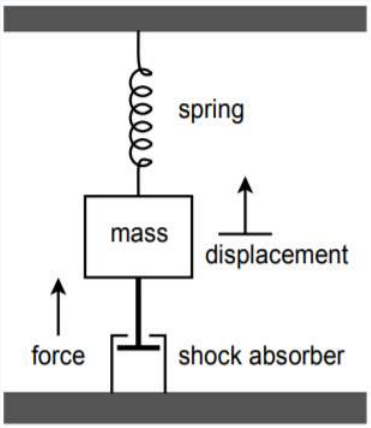


Figure 7.11.4.1: Mechanical system.

This system is comprised of a body with mass M, that is suspended from a spring. The spring has a spring constant, K. The mass is also connected to a shock absorber that produces damping, R. If an external force, F, excites the mass, it will move, producing some displacement, X. This displacement depends on the mass, force, spring constant, and damping. Essentially, the spring and shock absorber will create reactionary forces. From basic physics, $F = MA$, where A is the acceleration of the body. If X is the position of the body, then dX/dt is its velocity, and d^2X/dt^2 is its acceleration. Therefore, we can say

$$F = M\frac{d^2X}{dt^2}$$

In this system, the total force is comprised of the excitation force F, and the forces produced by the spring and shock absorber.

$$F - F_{spring} - F_{shock} = M\frac{d^2X}{dt^2}$$

The spring's force is equal to the displacement times the spring constant:

$$F_{spring} = KX$$

The shock absorber's force is equal to the damping constant times the velocity of the body:

$$F_{shock} = R\frac{dX}{dt}$$

By substituting and rearranging the above elements we find that

$$F = M\frac{d^2X}{dt^2} + R\frac{dX}{dt} + KX$$

Here F is seen as the input signal, and X as the output signal. A somewhat less busy notation form is the dot convention. A single dot represents the first derivative with respect to time, two dots represent the second derivative, and so on. The above Equation may be rewritten as

$$F = M\ddot{X} + R\dot{X} + KX$$

This is the final differential equation. Note how it contains only derivatives and no integrals. The last step is to solve the Equation for the highest-order differential. By setting it up in this form, the simulation circuit may be realized without using differentiators. This will indicate how many integrators will be required.

$$\ddot{X} = \frac{F}{M} - \frac{R}{M}\dot{X} - \frac{K}{M}X$$

This says that the second differential of X is the sum of three components. To realize the circuit, start with a summing amplifier with the three desired signals as inputs. This is shown in block form in Figure 7.11.4.2. Note that two of the inputs use X and the first derivative of X.

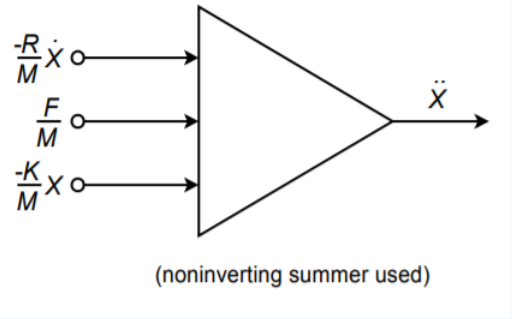


Figure 7.11.4.2: Circuit realization (block form).

These elements may be produced by integrating the output of Figure 7.11.4.2. Appropriate constants may be used to achieve the desired signal levels. This is shown in Figure 7.11.4.3. Certain elements may be combined; for example, a weighted summing amplifier may be used to eliminate unneeded amplifiers.

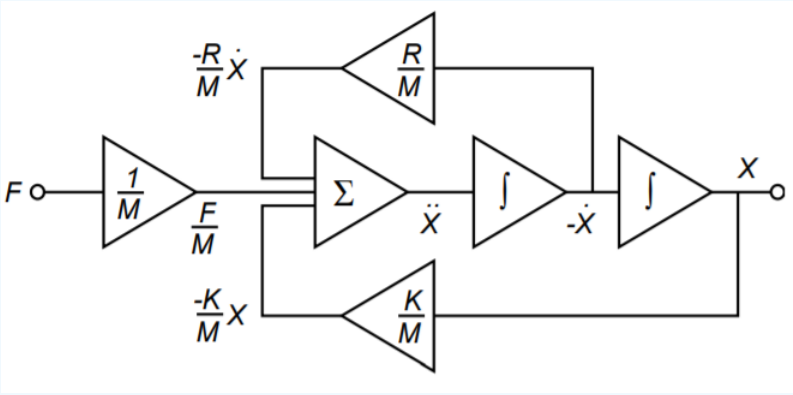


Figure 7.11.4.3: Circuit realization using function blocks.

In use, the constants R, K, and M are set by potentiometers (they are essentially nothing more than scaled gain factors). A voltage representing the excitation force is applied to the circuit, and the desired output quantity is recorded. Note that the output of interest could be the acceleration, velocity, or displacement of the body. In order to test the system with a new spring or damping constant, all that is needed is to adjust the appropriate potentiometer. In this manner, a large number of combinations may be tried quickly. The most successful combinations may then be built and tested for the final design. An analog computer such as this would be very useful in testing such items as the suspension of an automobile or a loudspeaker

system. To ease the design of the simulation circuit, commercial analog computers are available. Construction (or programming) of the circuit involves wiring integrator, amplifier, and summer blocks, together with the appropriate potentiometers. In this way, the details of designing and optimizing individual integrators or amplifiers is bypassed.

This page titled 7.11.4: Analog Computer is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.4: Analog Computer** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.5: Alternatives to Integrators and Differentiators

There are alternatives to using op amp based integrators and differentiators. As long as systems can be described by a reasonable set of equations, simulation using digital computers is possible. The primary advantages of a digital computer-based simulation scheme is that it is very flexible and potentially very accurate. Before the rise of affordable and powerful desktop digital computers, the speed of response of the digital technique was a severe limitation. Analog computers can be configured as real-time or faster-than-real-time devices: they respond at the same speed or faster than the system that is being simulated. The typically heavy computation load of the digital computer requires prodigious calculation speed to keep up with reasonably fast processes. Fortunately, this is no longer the problem it once was. Another advantage of the analog computer is its immediacy. It is by nature, interactive. This means that an operator can change simulation parameters and immediately see the result. Finally, there are some applications for which the added cost of the computing hardware cannot be justified. If these are not issues, then a digital-based simulation will most likely be preferred. The general idea is to calculate the response for several closely spaced points in time. The results may be used in a variety of ways. For example, graphs may be created that can then be studied at leisure or the resulting digital signal may be used elsewhere in the system. An extreme (although generally impractical) possibility is to design an op amp based simulator and then simulate its response by using a digital circuit simulator! As you might guess, this double simulation is not particularly efficient, especially with large systems.

This page titled 7.11.5: Alternatives to Integrators and Differentiators is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.5: Alternatives to Integrators and Differentiators** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.6: Extended Topic- Other Integrator and Differentiator Circuits

The basic integrator and differentiator circuits examined earlier may be extended into other forms. Perhaps the most obvious extension is to add multiple inputs, as in an ordinary summing amplifier. In complex systems, this concept may save the use of several op amps. A summing integrator is shown in Figure 7.11.6.1. Note its similarity to a normal summing amplifier. In this circuit, the input currents are summed at the inverting input of the op amp.

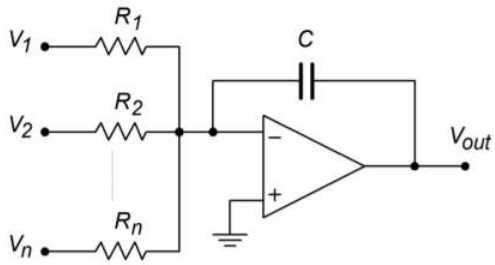


Figure 7.11.6.1: Summing integrator.

If the resistors are all set to the same value, we can quickly derive the output Equation by following the original derivation. The result is

$$V_{out} = -\frac{1}{RC}\int V_1 + V_2 + \cdots + V_n dt$$

The output is the negative integral of the sum of the inputs.

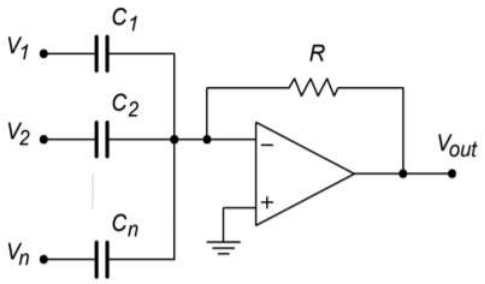


Figure 7.11.6.2: Summing differentiator.

In a similar vein, a summing differentiator may be formed. This is shown in Figure 7.11.6.2. Again, the proof of its output Equation follows the original differentiator derivation.

$$V_{out} = -RC_1\frac{dV_1}{dt} - RC_2\frac{dV_2}{dt} - \cdots - RC_n\frac{dV_n}{dt}$$

Another interesting adaptation of the integrator is the augmenting integrator. This circuit adds a constant gain portion to the output equation. An augmenting integrator is shown in Figure 7.11.6.3. The addition of the feedback resistor R_f provides the augmenting action. As you might surmise, the gain portion is directly related to R_f and R_i.

$$V_{out} = -V_{in}\frac{R_f}{R_i} - \frac{1}{RC}\int V_{in} dt$$

The augmenting integrator can also be turned into a summing/augmenting integrator by adding extra input resistors as in Figure 7.11.6.1. Note that the gain portion will be the same for all inputs if the input summing resistors are of equal value.

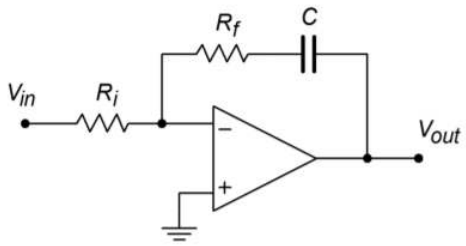


Figure 7.11.6.3: Augmenting integrator.

The final variant that we shall note is the double integrator. This design requires two reactive portions in order to achieve double integration. One possibility is shown in Figure 7.11.6.4. In this circuit, a pair of RC "Tee" networks are used. The output Equation is

$$V_{out} = -\frac{4}{(RC)^2}\int\int V_{in}dt$$

When properly used, the double integrator can cut down the parts requirement of larger circuit designs.

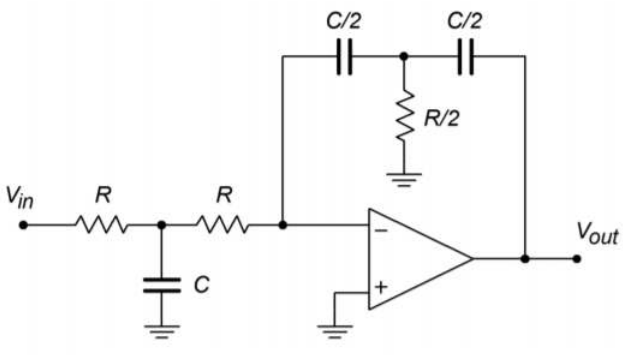


Figure 7.11.6.4: Double integrator.

This page titled 7.11.6: Extended Topic- Other Integrator and Differentiator Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.6: Extended Topic: Other Integrator and Differentiator Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.7: Summary

In this chapter we have examined the structure and use of integrator and differentiator circuits. Integrators produce a summing action whereas differentiators find the slope of the input. Both types are based on the general parallel-parallel inverting voltage feedback model. In order to achieve integration and differentiation, a capacitor is used in the feedback network in place of the standard resistor. Because the capacitor current is proportional to the rate of change of the capacitor's voltage, a differential or integral response is possible. For the integrator, the capacitor is placed in the normal R_f position, and for the differentiator, the capacitor is placed in the normal R_i position. As a result, the integrator exhibits a −6 dB per octave slope through the useful integration range. The differentiator exhibits the mirror image, or +6 dB per octave slope, throughout its useful range.

In both circuits, practical limitations require the use of additional components. In the case of the integrator, small DC offsets at the input can force the output into saturation. To avoid this, a resistor is placed in parallel with the integration capacitor in order to limit the low frequency gain. This has the unfortunate side effect of limiting the useful integration range to higher frequencies. In the case of the differentiator, noise, stability, and input impedance limits can pose problems. In order to minimize noise and aid in stability, a small capacitor may be placed in parallel with R_f. This reduces the high frequency gain. In order to place a lower limit on the input impedance, a resistor may be placed in series with the differentiation capacitor. The addition of either component will limit the upper range of differentiation.

We examined two general techniques for determining the output signal. The first form is referred to as the time-continuous method and although it may be used with virtually any waveform, our use was with simple sine waves only. In the case of the integrator, it corresponds to the indefinite integral - a time-domain Equation for the output is the result. The second form is the time-discrete method and is useful for waves that may be easily broken into segments. Each segment is analyzed, and the results joined graphically. This has the advantage of an immediate graphical result, whereas the shape of the time-continuous result may not be immediately apparent. The time-discrete method is also useful for producing output tables and graphs with a digital computer. In the case of the integrator, the time-discrete method corresponds to the definite integral.

Integrators and differentiators may be used in combination with summers and amplifiers to form analog computers. Analog computers may be used to model a variety of physical systems in real time. Unlike their digital counterparts, programming an analog computer only requires the proper interconnection of the various building blocks and appropriate settings for the required physical constants.

Finally, it is noted that integrators and differentiators may be used as wave shaping circuits. Integrators may be used to turn square waves into triangle waves. Differentiators can be used to turn triangle waves into square waves. Indeed, as differentiators tend to “seek” rapid changes in the input signal, they can be quite useful as edge detectors.

This page titled 7.11.7: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.7: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

7.11.8: Problems

10.7.1: Review Questions

1. What is the basic function of an integrator?

2. What is the basic function of a differentiator?

3. What is the function of the capacitor in the basic integrator and differentiator?

4. Why are capacitors used in favor of inductors?

5. What practical modifications need to be done to the basic integrator, and why?

6. What practical modifications need to be done to the basic differentiator, and why?

7. What are the negative side effects of the practical versus basic integrator and differentiator?

8. What is an analog computer, and what is it used for?

9. What are some of the advantages and disadvantages of the analog computer versus the digital computer?

10. How might integrators and differentiators be used as wave-shaping circuits?

10.7.2: Problems

Analysis Problems

1. Sketch the output waveform for the circuit of Figure 7.11.8.1 if the input is a 4 V peak square wave at 1 kHz.

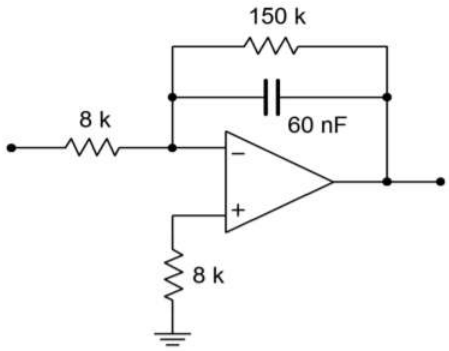


Figure 7.11.8.1

2. Repeat Problem 1 if $V_{in}(t) = 5 \sin 2\pi 318t$.

3. Repeat Problem 1 if $V_{in}(t) = 20 \cos 2\pi 10000t$.

4. Using the circuit of Problem 1, if the input is a ramp with a slope of 10 V/s, find the output after 1 ms, 10 ms, and 4 s. Sketch the resulting wave.

5. Determine the low frequency gain for the circuit of Figure 7.11.8.2.

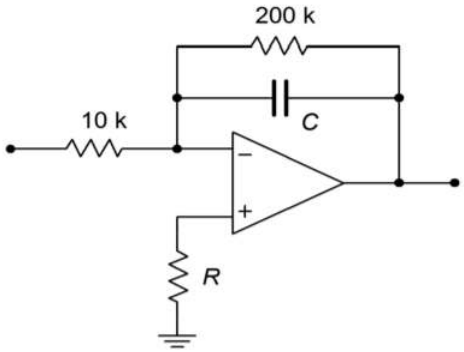


Figure 7.11.8.2

6. If $C = 33$ nF in Figure 7.11.8.2, determine V_{out} if V_{in} is a 200 mV peak sine wave at 50 kHz.

7. Repeat Problem 6 using a 200 mV peak square wave at 50 kHz.

8. Sketch the output of the circuit of Figure 10.2.7 with the input signal given in Figure 7.11.8.3.

9. Assume that the input to the circuit of Figure 10.2.7 is 100 mV DC. Sketch the output waveform, and determine if it goes to saturation.

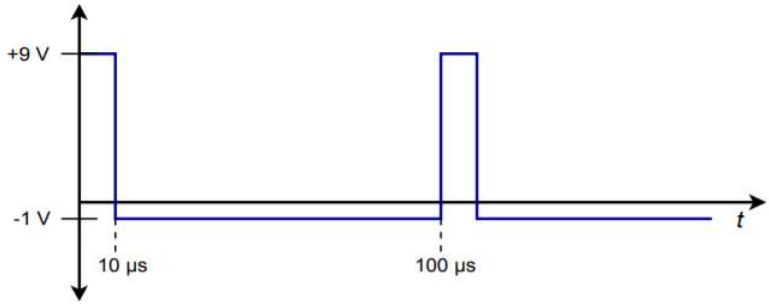


Figure 7.11.8.3

10. Sketch the output waveform for Figure 10.2.7, given an input of: $V_{in}(t) = 0.5 \cos 2\pi 9000t$.

11. Given the circuit of Figure 7.11.8.4, sketch the output waveform if the input is a 100 Hz, 1 V peak triangle wave.

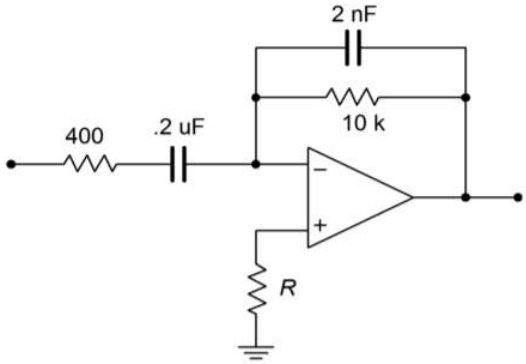


Figure 7.11.8.4

12. Repeat Problem 11 if the input is a 2 V peak square wave at 500 Hz. Assume that the rise and fall times are 1 μs and are linear (vs. exponential).

13. Repeat Problem 11 for the following input: $V_{in}(t) = 3 \cos 2\pi 60t$.

14. Repeat Problem 11 for the following input: $V_{in}(t) = 0.5 \sin 2\pi 1000t$.

15. Repeat Problem 11 for the following input: $V_{in}(t) = 10t^2$.

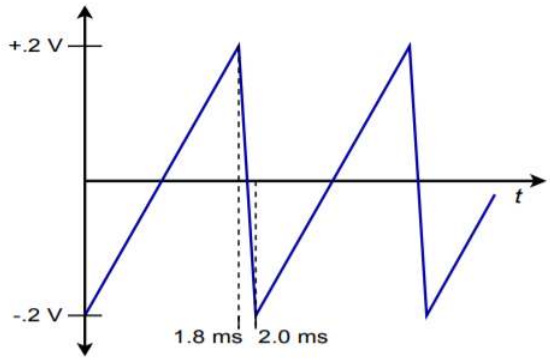


Figure 7.11.8.5

16. Given the input shown in Figure 7.11.8.5, sketch the output of the circuit of Figure 10.19.

17. Given the input shown in Figure 7.11.8.5, sketch the output of the circuit of Figure 10.19.

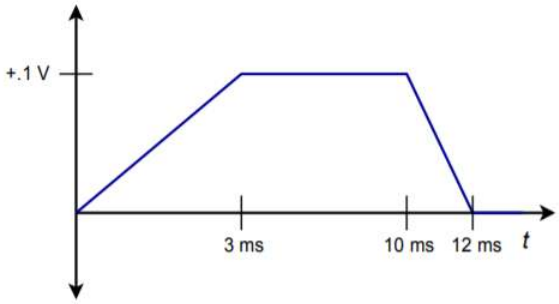


Figure 7.11.8.6

18. Sketch the output waveform for Figure 10.3.9, given an input of $V_{in}(t) = 0.5 \cos 2\pi 4000t$.

19. Sketch the output waveform for Figure 10.3.9, if V_{in} is a 300 mV peak triangle wave at 2500 Hz.

Design Problems

20. Given the circuit of Figure 7.11.8.1,

A) Determine the value of C required in order to yield an integration constant of −2000.

B) Determine the required of value of R for minimum integration offset error.

C) Determine f_{low}.

D) Determine the 99% accuracy point.

21. Given the differentiator of Figure 7.11.8.7, determine the value of R that will set the differentiation constant to -10^3.

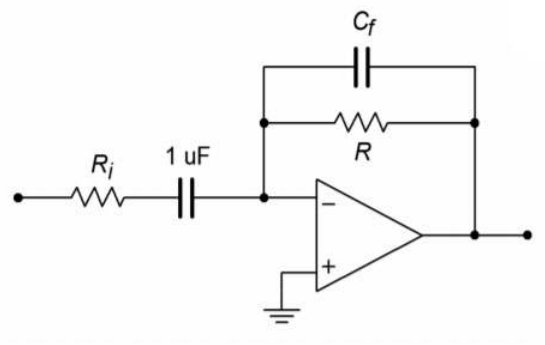


Figure 7.11.8.7

22. Determine the value of R_i such that the maximum gain is 20 for the circuit of Figure 7.11.8.7. (Use the values of Problem 21.)

23. Determine the value of C_f in Problem 22 such that noise above 5 kHz is attenuated.

24. Design an integrator to meet the following specifications: integration constant of −4500, f_{low} no greater than 300 Hz, Z_{in} at least 6 kΩ, and DC gain no more than 32 dB.

25. Design a differentiator to meet the following specifications: differentiation constant of $-1.2E-4$, f_{high} at least 100 kHz, and a minimum Z_{in} of 50 Ω.

Challenge Problems

26. Sketch the output waveform for the circuit of Figure 10.2.7 if the input is the following damped sinusoid: $V_{in}(t) = 3\varepsilon - 200t \sin 2\pi 1000t$.

27. A 1.57 V peak, 500 Hz square wave may be written as the following infinite series: $V_{in}(t) = 2\sum_{n=1}^{\infty} \frac{1}{2n-1} \sin 2\pi 500(2n-1)t$

Using this as the input signal, determine the infinite series output Equation for the circuit of Figure 10.2.7.

28. Sketch the output for the preceding problem. Do this by graphically adding the first few terms of the output.

29. Remembering that the voltage across an inductor is equal to the inductance times the rate of change of current, determine the output Equation for the circuit of Figure 7.11.8.8.

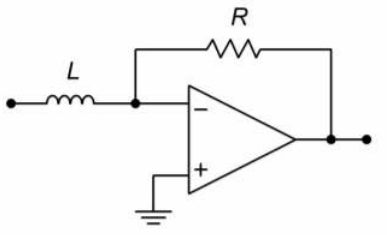


Figure 7.11.8.8

30. Repeat the preceding Problem for the circuit of Figure 7.11.8.9.

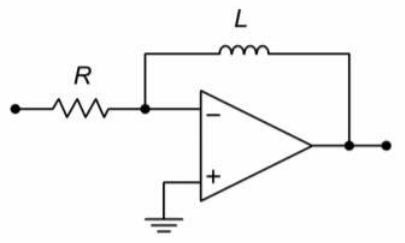


Figure 7.11.8.9

31. A 2.47 V peak, 500 Hz triangle wave may be written as the following infinite series: $V_{in}(t) = 2\sum_{n=1}^{\infty} \frac{1}{(2n-1)^2}\cos 2\pi 500(2n-1)t$

Using this as the input signal, determine the infinite series output Equation for the circuit of Figure 10.3.9.

32. Sketch the output for the preceding problem. Do this by graphically adding the first few terms of the output.

Computer Simulation Problems

33. Simulate Problem 1 and determine the steady-state response.

34. Model Problem 8 using a simulator. Determine both the steady-state and initial outputs.

35. Simulate Problem 14. Determine the steady-state output.

36. Model Problem 22 using a simulator and determine the output signal.

37. Compare the resulting waveform produced by the circuit of Problem 8 using both the 741 op amp and the medium-speed LF411. Does the choice of op amp make a discernible difference in this application?

This page titled 7.11.8: Problems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.8: Problems** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/OperationalAmplifiersAndLinearICs_3E.pdf.

CHAPTER OVERVIEW

8: Transient Behavior- First and Second Order Circuits

8.1: Introduction
8.2: Initial and Steady-State Analysis of RC Circuits
8.3: Transient Response of RC Circuits
8.4: Initial and Steady-State Analysis of RL Circuits
8.5: Transient Response of RL Circuits
8.6: Initial and Steady-State Analysis of RLC Circuits
8.7: Summary
8.8: Exercises

This page titled 8: Transient Behavior- First and Second Order Circuits is shared under a not declared license and was authored, remixed, and/or curated by James M. Fiore.

SECTION OVERVIEW

8.1: Introduction

This page titled 8.1: Introduction is shared under a not declared license and was authored, remixed, and/or curated by James M. Fiore.

8.2: Initial and Steady-State Analysis of RC Circuits

When analyzing resistor-capacitor circuits, always remember that capacitor voltage cannot change instantaneously. If we assume that a capacitor in a circuit is not initially charged, then its voltage must be zero. The instant the circuit is energized, the capacitor voltage must still be zero. If there is no voltage across the device, then it is behaving like a short circuit. We call this the initial state. Thus, we have our first rule regarding RC circuits:

$$\text{For DC analysis, initially capacitors appear as shorts.} \tag{8.2.1}$$

Consider the circuit of Figure 8.3.1 . Assume that C_1 and C_2 are initially uncharged and there is no voltage across them.

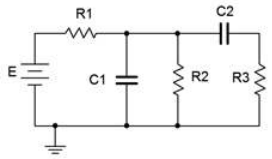


Figure 8.3.1 : A basic resistor-capacitor (RC) circuit.

The instant power is applied, the two capacitors appear as short circuits. If we redraw the circuit for this instant in time, we arrive at the equivalent circuit shown in Figure 8.3.2 .

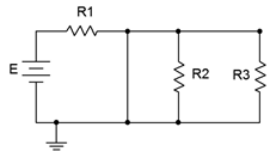


Figure 8.3.2 : A basic RC circuit, initial state.

Given this equivalent, we can see that shorting C_2 places R_2 and R_3 in parallel, however, they are both shorted out by C_1. This leaves only R_1 left in the circuit along with the source, E. At this point, currents will begin to flow, and thus begin charging up the capacitors. As the capacitor voltages rise, the current will begin to decrease, and eventually the capacitors will stop charging. At that point no further current will be flowing, and thus the capacitor will behave like an open. We call this the steadystate condition and we can state our second rule:

$$\text{At steady-state, capacitors appear as opens.} \tag{8.2.2}$$

Continuing with the example, at steady-state both capacitors behave as opens. This is shown in Figure 8.3.3 . This leaves E to drop across R_1 and R_2. This will create a simple voltage divider. The steady-state voltage across C_1 will equal that of R_2. As C_2 is also open, the voltage across R_3 will be zero while the voltage across C_2 will be the same as that across R_2.

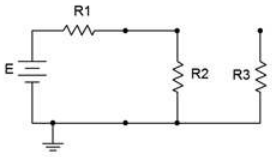


Figure 8.3.3 : A basic RC circuit, steady-state.

In reality, practical capacitors can be thought of as an ideal capacitance in parallel with a very large (leakage) resistance, so there will be a limit to this performance.

Example 8.3.1

Given the circuit of Figure 8.3.4 , find the voltage across the 6 kΩ resistor for both the initial and steady-state conditions assuming the capacitor is initially uncharged.

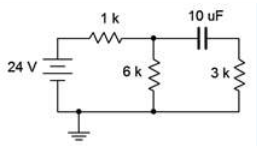


Figure 8.3.4 : Circuit for Example 8.2.4.

For the initial state the capacitor is treated as a short. The initial state equivalent circuit is drawn below in Figure 8.3.5 . Immediately apparent is the parallel connection between the 6 kΩ and 3 kΩ resistors. This combination is equivalent to 2 kΩ. Therefore, we can perform a voltage divider to find the potential across the 6 kΩ (i.e., the 2 kΩ combo).

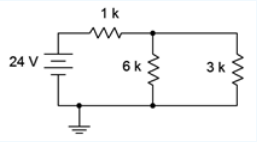


Figure 8.3.5 : Circuit of Figure 8.3.3 , initial state.

$$V_{6k} = E\frac{R_x}{R_x + R_y}$$

$$V_{6k} = 24V\frac{2k\Omega}{2k\Omega + 1k\Omega}$$

$$V_{6k} = 16V$$

For the steady-state condition the capacitor will be fully charged, its current will be zero, and we treat it as an open. The steady-state equivalent circuit is drawn below in Figure 8.3.6 .

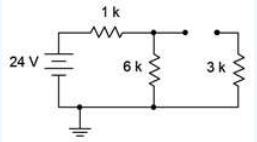


Figure 8.3.6 : Circuit of Figure 8.3.3 , steady-state.

The 3 kΩ resistor is now out of the picture, leaving us with the 6 kΩ in series with the 1 kΩ resistor. Once again, a voltage divider may be used to determine the voltage across the 6 kΩ.

$$V_{6k} = E\frac{R_x}{R_x + R_y}$$

$$V_{6k} = 24V\frac{6k\Omega}{6k\Omega + 1k\Omega}$$

$$V_{6k} = 20.57V$$

This page titled 8.2: Initial and Steady-State Analysis of RC Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.3: Initial and Steady-State Analysis of RC Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

8.3: Transient Response of RC Circuits

The question remains, “What happens between the time the circuit is powered up and when it reaches steady-state?” This is known as the transient response. Consider the circuit shown in Figure 8.4.1 . Note the use of a voltage source rather than a fixed current source, as examined earlier.

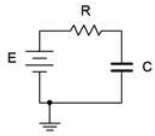


Figure 8.4.1 : A simple RC circuit.

The key to the analysis is to remember that capacitor voltage cannot change instantaneously. Assuming the capacitor is uncharged, the instant power is applied, the capacitor voltage must be zero. Therefore all of the source voltage drops across the resistor. This creates the initial current, and this current starts to charge the capacitor (the initial rate being equal to i/C as dictated by Equation 8.2.6). According to Kirchhoff's voltage law, as the capacitor voltage begins to increase, the resistor voltage must decrease because the sum of the two must equal the fixed source voltage. This means that the circulating current must also decrease. This, in turn, means that the rate of capacitor voltage increase begins to slow. As the capacitor voltage continues to increase, less voltage is available for the resistor, causing further reductions in current, and a further slowing of the rate of capacitor voltage change. Eventually, the capacitor voltage will be nearly equal to the source voltage. This will result in a very small potential across the resistor and an equally small current, slowing subsequent capacitor voltage increases to a near standstill. Theoretically, the capacitor voltage approaches the source voltage but never quite equals it. Similarly, the current drops to near zero, but never completely turns off. This is illustrated in Figure 8.4.2 .

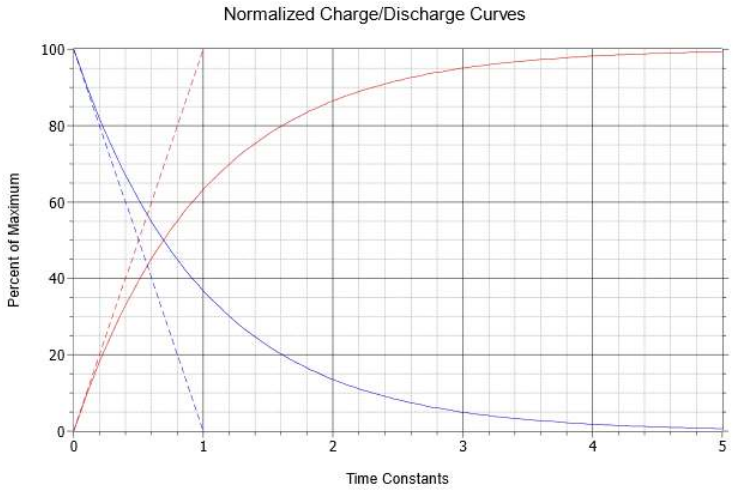


Figure 8.4.2 : Normalized charge and discharge curves.

The dashed red line represents the initial rate of change of capacitor voltage. This trajectory is what would be expected if an ideal current source drove the capacitor, as in Example 8.2.4. As noted previously, the rate of voltage change versus times is equal to i/C, and therefore in this case, E/RC. If the initial rate of change were to continue unabated, the source voltage, E, would be reached in RC seconds. Consequently, RC is referred to as the charge time constant and is denoted by τ (Greek letter tau). Thus,

$$\text{Time constant, } \tau = RC \tag{8.3.1}$$

As noted, once the capacitor begins to charge, the current begins to decrease and the capacitor voltage curve begins to fall away from the initial trajectory. The solid red curve represents the capacitor voltage. Notice that after five time constants the capacitor is nearly fully charged and the circuit is considered to be in steady-state (i.e., the capacitor behaves as an open).

$$\text{Steady-state is reached in approximately five time constants.} \tag{8.3.2}$$

This sort of recursively dependent operation is characteristic of exponential functions. The equation for the capacitor's voltage charging curve is:

$$V_C(t) = E\left(1 - \epsilon^{-\frac{t}{\tau}}\right) \tag{8.3.3}$$

Where

$V_C(t)$ is the capacitor voltage at time t,

E is the source voltage,

t is the time of interest,

τ is the time constant,

ε (also written e) is the base of natural logarithms, approximately 2.718.

The dashed blue line shows the initial slope of current change. The solid blue curve shows the circulating current (and by extension of Ohm's law, the resistor voltage). The equation for this curve, which follows the general shape ε^{-t}, is:

$$I(t) = \frac{E}{R}\epsilon^{-\frac{t}{\tau}} \tag{8.3.4}$$

and

$$V_R(t) = E\epsilon^{-\frac{t}{\tau}} \tag{8.3.5}$$

Once power is removed or bypassed, the stored charge on the capacitor will dissipate through any associated resistor(s) creating a discharge current which will end with the capacitor voltage drained back to zero. During the discharge phase, both the capacitor's voltage and current will follow the solid blue curve; Equations 8.3.4 and 8.3.5 being appropriate. The discharge time constant may be different from the charge times constant, depending on the associated resistances.

A precise derivation of the exponential charge/discharge equations is given in Appendix C.

Example 8.4.1

Given the circuit of Figure 8.4.3 , assume the switch is closed at time $t = 0$. Determine the charging time constant, the amount of time after the switch is closed before the circuit reaches steady-state, and the capacitor voltage at $t = 0$, $t = 50$ milliseconds and $t = 1$ second. Assume the capacitor is initially uncharged.

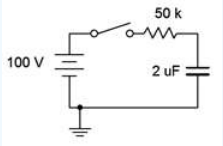


Figure 8.4.3 : Circuit for Example 8.4.1 .

First, the time constant:

$$\tau = RC$$

$$\tau = 50k\Omega 2\mu F$$

$$\tau = 100ms$$

Steady-state will be reached in five time constants, or 500 milliseconds. Therefore we know that $V_C(0) = 0$ volts and $V_C(1) = 100$ volts. To find $V_C(50ms)$ we simply solve Equation 8.3.3.

$$V_C(t) = E\left(1 - \epsilon^{-\frac{t}{\tau}}\right)$$

$$V_C(50ms) = 100V\left(1 - \epsilon^{-\frac{50ms}{100ms}}\right)$$

$$V_C(50ms) \approx 39.35V$$

https://eng.libretexts.org/@go/page/52937

This value can also be determined graphically from Figure 8.4.2 . The time of 50 milliseconds represents one-half time constant. Find this value on the horizontal axis and then track straight up to the solid red curve that represents the charging capacitor voltage. The point of intersection is at approximately 40% of the maximum value on the vertical axis. The maximum value here is the source voltage of 100 volts. Therefore the capacitor will have reached approximately 40% of 100 volts, or just about 40 volts.

Computer Simulation

The circuit of Figure 8.4.3 is entered into a simulator, as shown in Figure 8.4.4 . In order to reflect the notion of a time-varying circuit with a switch, the 100 volt DC voltage source has been replaced with a rectangular pulse voltage source. This source starts at 0 volts and then immediately steps up to 100 volts. It stays at this level for 500 milliseconds before dropping back to 0 volts.

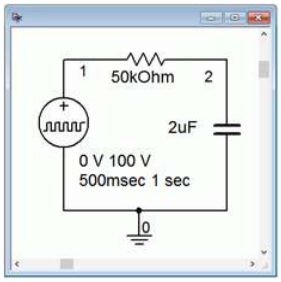


Figure 8.4.4 : Circuit of Figure 8.4.3 in a simulator.

A transient (i.e., time domain) simulation is run, plotting the capacitor voltage over time for the first 500 milliseconds. This is seen in Figure 8.4.5 . There are several items to note here. First, we see that the overall shape perfectly echoes the generic curve presented in Figure 8.4.2 . Second, we see that after the predicted five time constants, or 500 milliseconds, the capacitor voltage has plateaued, indicating steady-state. Third, at steady-state the capacitor voltage has virtually reached the maximum value set be the source, or 100 volts. Finally, at 50 milliseconds, we see that the capacitor voltage has reached roughly 40 volts, just as predicted.

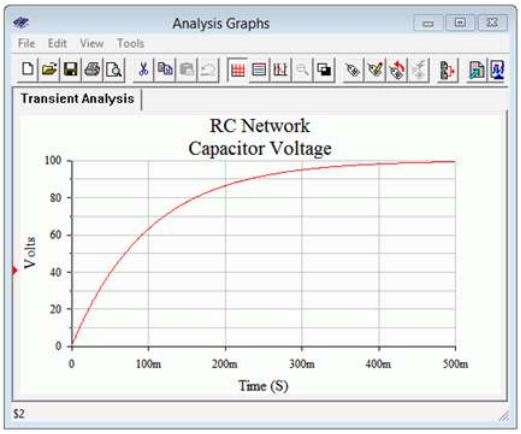


Figure 8.4.5 : Simulation results for the circuit of Figure 8.4.2 .

As mentioned previously, it is possible for a circuit to have different charge and discharge time constants. This could occur if a switch introduces or removes component(s) between the charge versus discharge phases. Further, because the capacitor is discharging, its current direction will be opposite to that of the charging current. KVL still must be satisfied, but because the capacitor is now behaving as a source, the magnitude of the discharge resistance's voltage must equal the capacitor voltage magnitude. Therefore, its curve will take the same shape (the solid blue curve of Figure 8.4.2). These issues will be illustrated in the following example.

Example 8.4.2

For the circuit of Figure 8.4.6 , assume the capacitor is initially uncharged. At time $t = 0$ the switch contacts position 1. The switch is thrown to position 2 at time $t = 50$ milliseconds.

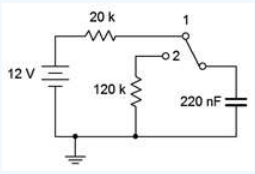


Figure 8.4.6 : Circuit for Example 8.4.2 .

Determine the charging time constant, the amount of time after the switch is closed before the circuit reaches steady-state, the maximum charging and discharging currents, and the capacitor voltage at $t = 0$, $t = 50$ milliseconds, $t = 90$ milliseconds, and $t = 1$ second.

We begin with the charge time constant:

$$\tau_{charge} = RC$$

$$\tau_{charge} = 20k\Omega 220nF$$

$$\tau_{charge} = 4.4ms$$

Steady-state will be reached in 5 times 4.4 milliseconds, or 22 milliseconds. The capacitor is initially uncharged, so $V_C(0) = 0$ volts. As the capacitor will have reached steady-state in 22 milliseconds, $V_C(50ms) = 12$ volts. The maximum charging current will occur at $t = 0$ when all of the 12 volt source drops across the 20 kΩ resistor, or 600 μamps, flowing left to right.

At 50 milliseconds the switch is thrown to position 2. The 12 volt source and 20 kΩ resistor are no longer engaged. At this point the capacitor has 12 volts across it, positive to negative, top to bottom. As the capacitor voltage cannot change instantaneously, the capacitor now acts as a voltage source and discharges through the 120 kΩ resistor. Note that the discharge current is flowing counterclockwise, the opposite of the charging current. The discharge time constant is:

$$\tau_{discharge} = RC$$

$$\tau_{discharge} = 120k\Omega 220nF$$

$$\tau_{discharge} = 26.4ms$$

The capacitor will fully discharge down to 0 volts in 5 time constants, or some 132 milliseconds after the switch is thrown to position 2. Thus steady-state occurs at $t = 182$ milliseconds. The maximum discharge current occurs the instant the switch is thrown to position 2 when all of the capacitor's 12 volts drops across the 120 kΩ resistor, yielding 100 μ amps, flowing top to bottom.

Clearly, at $t = 90$ milliseconds the capacitor is in the discharge phase. The capacitor's voltage and current during the discharge phase follow the solid blue curve of Figure 8.4.2 . The elapsed time for discharge is 90 milliseconds minus 50 milliseconds, or 40 milliseconds net. We can use a slight variation on Equation 8.3.5 to find the capacitor voltage at this time.

$$V_C(t) = E\epsilon^{-\frac{t}{\tau}}$$

$$V_C(40ms) = 12V - \epsilon^{-\frac{40ms}{26.4ms}}$$

$$V_C(40ms) \approx 2.637V$$

The shape of the capacitor's voltage will appear somewhat like a rounded pulse, rising with a curve and then falling back to zero with a complementary curve (the red and then blue curves of Figure 8.4.2).

Basic single resistor-capacitor circuits prove to be fairly easy to solve given a little practice, but what if a more complex circuit is used? In this situation the section feeding the capacitor may be simplified using Thévenin's theorem to determine the effective source voltage and charging resistance. The circuit then reverts back to a simple RC network which may be solved directly.

If power is interrupted before the capacitor is fully charged, the equations presented previously may be used to determine the precise voltage(s) and current(s) reached. The capacitor will then behave as a voltage source and begin to discharge, its voltage curve following the blue plot line of Figure 8.4.2 , with its maximum voltage being what the capacitor charged to, not the associated driving voltage. The following example and simulations address these issues.

Example 8.4.3

For this example we shall revisit the circuit of Example 8.3.1. The circuit is redrawn in Figure 8.4.7 for convenience. Assume the capacitor is initially uncharged.

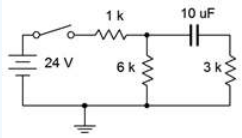


Figure 8.4.7 : Circuit for Example 8.4.3 .

Determine the charging time constant, the amount of time after the switch is closed before the circuit reaches steady-state, and the capacitor voltage at $t = 0$, 100 milliseconds, and 200 milliseconds. At 200 milliseconds, the switch is opened. Determine how long it takes for the capacitor to fully discharge and the voltage across the 6 kΩ resistor at $t = 275$ milliseconds (i.e., 75 milliseconds after the switch is opened).

The first step is to determine the Thévenin equivalent driving the capacitor. If we remove the capacitor and determine the open circuit voltage at those points, we see that it is just a voltage divider between the 24 volt source, the 6 kΩ resistor and the 1 kΩ resistor (the 3 kΩ resistor has no current through it and thus produces no voltage drop). This works out to 20.57 volts. The Thévenin resistance will be 3 kΩ in series with 1 kΩ || 6 kΩ, or roughly 3.857 kΩ. The equivalent charging circuit is drawn in Figure 8.4.8 .

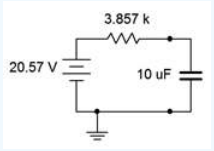


Figure 8.4.8 : Thévenin equivalent for the circuit of Figure 8.4.7 driving the capacitor.

We can now determine the charging time constant:

$$\tau_{charge} = RC$$

$$\tau_{charge} = 3.857k\Omega 10\mu F$$

$$\tau_{charge} = 38.57ms$$

Steady-state will be reached in 5 time constants, or 192.8 ms. Thus, we know that $V_C(0) = 0$ volts and $V_C(200ms) = 20.57$ volts. For the capacitor voltage at 100 milliseconds, we simply use the charge equation.

$$V_C(t) = E\left(1 - \epsilon^{-\frac{t}{\tau}}\right)$$

$$V_C(100ms) = 20.57V\left(1 - \epsilon^{-\frac{100ms}{38.57ms}}\right)$$

$$V_C(100ms) \approx 19.03V$$

For the discharge phase, we need to determine the time constant. With the voltage source removed, the capacitor will discharge through the now series combination of the 3 kΩ resistor and 6 kΩ resistor.

$$\tau_{discharge} = RC$$

$$\tau_{discharge} = 9k\Omega 10\mu F$$

$$\tau_{discharge} = 90ms$$

Steady-state will be reached 450 milliseconds later at $t = 650$ milliseconds. To find V_{6k} at $t = 275$ milliseconds, we can find the voltage across the capacitor and then perform a voltage divider between the 6 kΩ and 3 kΩ resistors. Remembering that $t = 275$ milliseconds is 75 milliseconds into the discharge phase, we have:

$$V_C(t) = E\epsilon^{-\frac{t}{\tau}}$$

$$V_C(75ms) = 20.57V\left(1 - \epsilon^{-\frac{75ms}{90ms}}\right)$$

$$V_C(75ms) \approx 8.94V$$

Finally, this voltage splits between the 6 kΩ and 3 kΩ resistors. Using the voltage divider rule, we find:

$$V_{6k} = V_C\frac{R_x}{R_x + R_y}$$

$$V_{6k} = 8.94V\frac{6k\Omega}{6k\Omega + 3k\Omega}$$

$$V_{6k} = 5.96V$$

Computer Simulation

In order to verify the analysis of Example 8.4.3 , the circuit of Figure 8.4.7 is entered into a simulator, as shown in Figure 8.4.9 . In place of a DC source, a pulse generator is used to mimic the on-off nature of the switch. This starts starts at zero volts and then immediately jumps up to 24 volts. It stays at this level for 200 milliseconds before returning back to zero. This is sufficient time to check whether or not the capacitor voltage has reached steady-state (predicted to take 192.8 milliseconds).

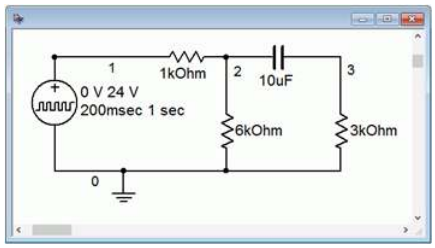


Figure 8.4.9 : Circuit of Figure 8.4.7 in a simulator.

A transient analysis is run on this circuit, plotting the capacitor voltage (i.e., the difference between the node 2 and node 3 voltages). The result is shown in Figure 8.4.10 . This plot confirms nicely the charge phase of the capacitor. After approximately 200 milliseconds, the voltage has leveled out at just over 20 volts, precisely as predicted.

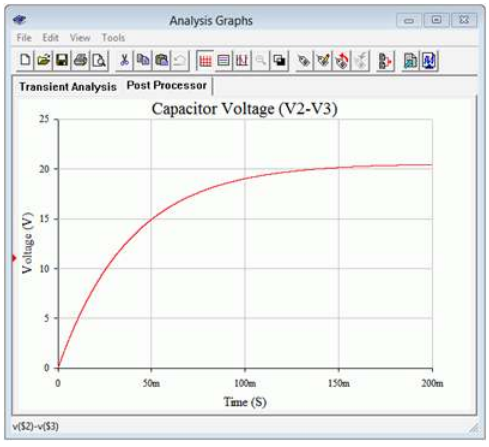


Figure 8.4.10 : Simulation results for the charge phase of the circuit of Figure 8.4.7 .

What about the discharge phase? In order to investigate that portion, the simulation circuit is modified. The pulse voltage source is disconnected from the remainder of the circuit, just as it would be if the switch in Figure 8.4.7 had been opened again. Also, the capacitor is modified to have an initial voltage of 20.57 volts, the precise value at had reached after it attained steady-state. A second transient analysis is run, again plotting the capacitor voltage. The results of this simulation are shown in Figure 8.4.11 . It is worth noting that the time axis is relative to the switch being opened, not the original timing. That is, the horizontal origin of 0 milliseconds corresponds to $t = 200$ milliseconds.

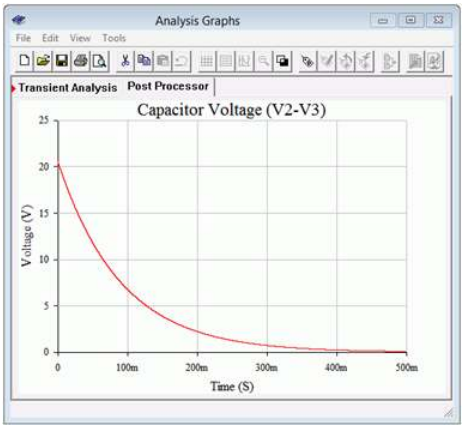


Figure 8.4.11 : Simulation results for the discharge phase of the circuit of Figure 8.4.7 .

Note that the time required to reach the new steady-state value of zero volts has stretched out to some 450 milliseconds after the switch is opened, precisely as predicted.

At this point, a fair question to ask is, “Couldn't we leave the pulse source in place in order to investigate the discharge phase?” Although the pulse source does go back down to zero volts at $t = 200$ milliseconds, that's not the same as opening the switch back in Figure 8.4.7 . If the source is still connected but producing zero volts, it becomes part of the Thévenin equivalent. As such, the 1 kΩ resistor is back in the circuit, producing a discharge time constant identical to the charge time constant.

To prove the point, the simulation is run again. The pulse remains at 24 volts for 200 milliseconds and then jumps down to zero, as before. The simulation time is extended out to 400 milliseconds total, enough to see both the charge and discharge phases. Further, instead of just plotting the capacitor voltage; the voltages of the source (blue trace, node 1), the 6 kΩ resistor (green trace, node 2) and the 3 kΩ resistor (red trace, node 5) are plotted. The results are shown in Figure 8.4.12 .

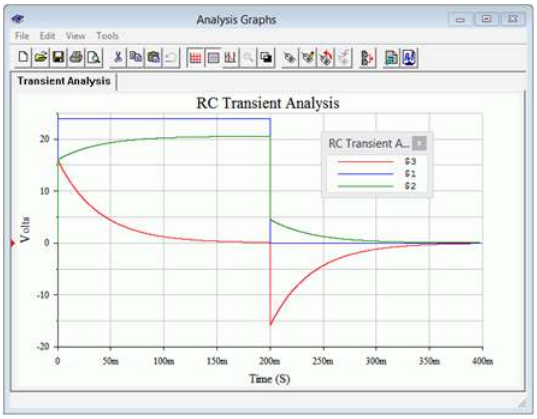


Figure 8.4.12 : Simulation results using pulse voltage source for discharge phase.

The first item to note is that steady-state appears to be reached in just under 200 milliseconds for both the charge and discharge phases, as expected. Second, note that the node 2 voltage (green) minus the node 3 voltage (red) starts at zero and winds up at a little over 20 volts at 200 milliseconds. This is, of course, the capacitor voltage, but what is interesting here is that this plot shows how the voltage across the 3 kΩ resistor shrinks as the capacitor voltage grows, the sum equaling the node 2 voltage. This makes

perfect sense because, as the capacitor voltage increases, the current through it must be decreasing, and as this same current is flowing through the 3 kΩ resistor, the resistor's voltage must also be decreasing due to Ohm's law.

The other interesting part of this plot is what happens at 200 milliseconds when the source goes back to zero. Note that the node 3 voltage immediately jumps to a negative value. This is because the voltage across the capacitor cannot change instantaneously. It must still have 20.57 volts across it the instant the source goes back to zero. In this situation, because the source is essentially a short, the capacitor winds up in series with the 3 kΩ resistor and the parallel combination of the 1 kΩ and 6 kΩ resistors, or about 857 Ω. Calculating the voltage divider between the 3 kΩ and 857 Ω resistors with 20.57 volt source shows the 3 kΩ resistor receiving approximately 16 volts. Further, the discharge current will be flowing out of the capacitor in a counterclockwise direction, meaning it flows from ground up through the 3 kΩ resistor. Thus, we expect node 3 to be at approximately −16 volts, which is precisely what the plot indicates. How cool is that?

What if we didn't wait for steady-state? How would these plots change? Essentially, the trajectories of the curves would not change. After all, how would the circuit “know” that the switch would open early or the pulse would flip prematurely?

What happens is that the curves are followed to the point in time where the circuit is interrupted. From there, the next phase occurs with the present voltages as the starting points. To test this, the simulation is run yet again, but this time the source pulse width is shortened to just 50 milliseconds, well short of steady-state. The results of the simulation are shown in Figure 8.4.13 .

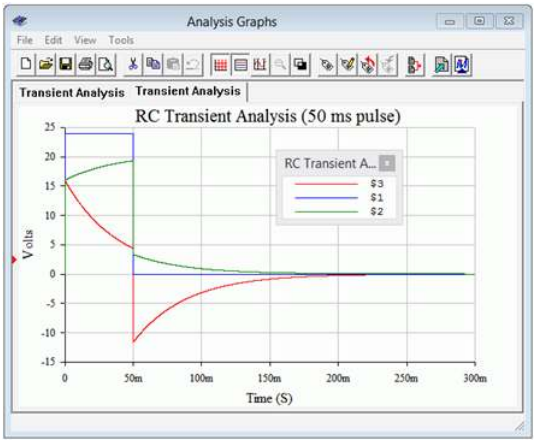


Figure 8.4.13 : Simulation results for interrupted discharge phase using pulse voltage source.

Comparing the plot of Figure 8.4.13 to that of Figure 8.4.12 shows that the two are identical up to 50 milliseconds. At that point, the input pulse returns to zero and the capacitor begins to discharge. The shapes and timings of the node 2 and node 3 voltages are the same as they were in Figure 8.4.12 , however, the amplitudes are reduced. This is because the capacitor did not have time to reach the steady-state voltage of 20.57 volts. In fact, it only reaches about 14.94 volts using Equation 8.3.3. Applying the voltage divider on this potential as before shows that the 3 kΩ resistor should jump to approximately −11.6 volts, which is confirmed by the simulation.

This page titled 8.3: Transient Response of RC Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **8.4: Transient Response of RC Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

8.4: Initial and Steady-State Analysis of RL Circuits

When analyzing resistor-inductor circuits, remember that current through an inductor cannot change instantaneously as this would require an infinite voltage source. When a circuit is first energized, the current through the inductor will still be zero, which is characteristic of opens. Once at steady-state, the current has leveled out and therefore the voltage across the inductor will approach zero, which is characteristic of shorts. Thus, we can state the general behavior of inductors at the beginning and ending of the charge cycle:

$$\text{For DC analysis, initially inductors appear as opens.} \tag{8.4.1}$$

$$\text{At steady-state, inductors appear as shorts.} \tag{8.4.2}$$

This is the opposite of what was seen with capacitors. For example, in the circuit of Figure 9.3.1 , initially L is open, leaving us with R_1 and R_2 in series with the source, E. At steady-state, L shorts out, leaving R_1 in series with the parallel combination of R_2 and R_3. All practical inductors will exhibit some internal resistance, so it is often best to think of an inductor as an ideal inductance with a small resistance (R_{coil}) in series with it.

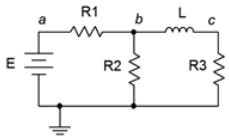


Figure 9.3.1 : Basic RL circuit.

Example 9.3.1

Assuming the initial current through the inductor is zero in the circuit of Figure 9.3.2 , determine the voltage across the 2 kΩ resistor when power is applied and after the circuit has reached steady-state. Draw each of the equivalent circuits.

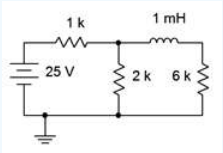


Figure 9.3.2 : Circuit for Example 9.3.1 .

First, we'll redraw the circuit for the initial-state equivalent. To do so, open the inductor. The new equivalent is shown in Figure 9.3.3 . By opening the inductor, the 6 kΩ resistor has been removed from the circuit and sees no voltage. What we are left with is a voltage divider between the source and the 1 kΩ and 2 kΩ resistors.

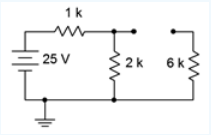


Figure 9.3.3 : Initial-state equivalent of the circuit of Figure 9.3.2 .

Using the voltage divider rule,

$$V_{2k} = E\frac{R_x}{R_x + R_y}$$

$$V_{2k} = 25V\frac{2k\Omega}{2k\Omega + 1k\Omega}$$

$$V_{2k} \approx 16.67V$$

For steady-state, we redraw using a short in place of the inductor, as shown in Figure 9.3.4 . Here we have another voltage divider, this time between the 1 kΩ resistor and the parallel combination of 2 kΩ and 6 kΩ, or 1.5 kΩ.

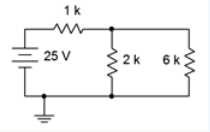


Figure 9.3.4 : Steady-state equivalent of the circuit of Figure 9.3.2 .

$$V_{2k} = E\frac{R_x}{R_x + R_y}$$

$$V_{2k} = 25V\frac{1.5k\Omega}{1.5k\Omega + 1k\Omega}$$

$$V_{2k} = 15V$$

Computer Simulation

To verify the results of Example 9.3.1 , the circuit is entered into a simulator as shown in Figure 9.3.5 . A DC operating point analysis is run and the results are shown in Figure 9.3.6 .

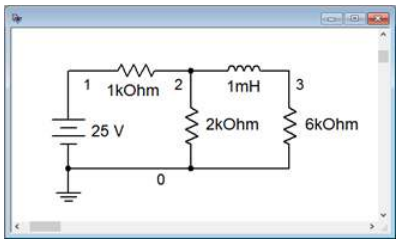


Figure 9.3.5 : Circuit of Figure 9.3.2 in a simulator.

The steady-state potential at node 2 corresponds to the voltage across the 2 kΩ resistor and agrees with the theoretical calculation of 15 volts. Note that node 3 is also 15 volts, indicating that the steady-state voltage across the inductor is zero, meaning it is behaving as a short, exactly as expected.

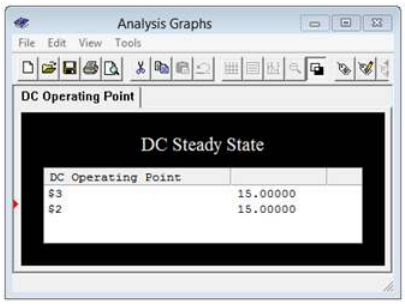


Figure 9.3.6 : Simulation results for the circuit of Figure 9.3.2 .

This page titled 8.4: Initial and Steady-State Analysis of RL Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.3: Initial and Steady-State Analysis of RL Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

8.5: Transient Response of RL Circuits

The transient response of RL circuits is nearly the mirror image of that for RC circuits. To appreciate this, consider the circuit of Figure 9.5.1 .

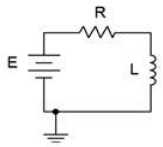


Figure 9.5.1 : RL circuit for transient response analysis.

Again, the key to this analysis is to remember that inductor current cannot change instantaneously. When power is first applied, the circulating current must remain at zero. Therefore no voltage drop is produced across the resistor, and by KVL, the voltage across the inductor must equal the source, E. This establishes the initial rate of change of current via Equation 9.2.9 $(di/dt = E/L)$ and is represented by the dashed red line in the graph of Figure 9.5.2 . As the current starts to increase, the voltage drop across the resistor begins to increase. This reduces the voltage available for the inductor, thus slowing the rate of change of current. This is depicted by the solid red curve on the graph. Meanwhile, the solid blue curve represents the decreasing inductor voltage. Thus, in the RL circuit, the inductor's voltage curve echoes the RC circuit's current curve (or resistor voltage curve), and the RL current curve echoes the RC circuit's capacitor voltage curve.

The curves presented in Figure 9.5.2 are identical to those presented in Chapter 8 when we discussed capacitors. They are reproduced here for your convenience.

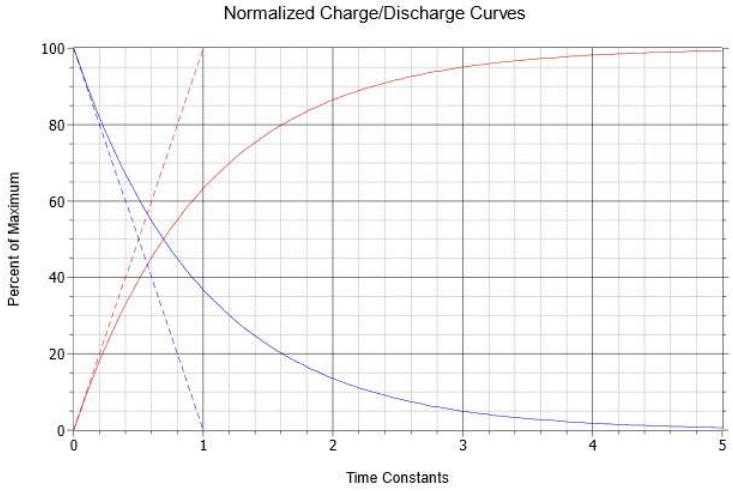


Figure 9.5.2 : Normalized charge and discharge curves.

As noted before, the rate of current change versus time is equal to v/L, and therefore in this case, E/L. If the initial rate of change were to continue unabated, the maximum (steady-state) current, E/R, would be reached in L/R seconds[1]. Therefore the time constant for an RL circuit is:

$$\tau = \frac{L}{R} \tag{8.5.1}$$

Once again, five constants will achieve steady-state.

Following the prior work on capacitors, the relevant equations for the RL circuit can be shown to be[2]:

$$V_L(t) = E\epsilon^{-\frac{t}{\tau}} \tag{8.5.2}$$

$$V_R(t) = E\left(1 - \epsilon^{-\frac{t}{\tau}}\right) \tag{8.5.3}$$

$$I(t) = \frac{E}{R}\left(1 - \epsilon^{-\frac{t}{\tau}}\right) \tag{8.5.4}$$

Where

$V_L(t)$ is the inductor voltage at time t,

$V_R(t)$ is the resistor voltage at time t,

$I(t)$ is the current at time t,

E is the source voltage,

R is the series resistance,

t is the time of interest,

τ is the time constant,

ε (also written e) is the base of natural logarithms, approximately 2.718.

Time for an example.

Example 9.5.1

Given the circuit of Figure 9.5.3 , assume the switch is closed at time $t = 0$. Determine the charging time constant, the amount of time after the switch is closed before the circuit reaches steady-state, and the inductor voltage and current at $t = 0$, $t = 2$ microseconds and $t = 1$ millisecond. Assume the inductor is initially uncharged.

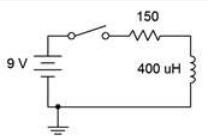


Figure 9.5.3 : Circuit for Example 9.5.1 .

First, the time constant:

$$\tau = \frac{L}{R}$$

$$\tau = \frac{400\mu H}{150\Omega}$$

$$\tau \approx 2.667\mu s$$

Steady-state will be reached in five time constants, or approximately 13.33 microseconds. Thus we know that $V_L(0) = 9$ volts and $V_L(1ms) = 0$ volts. Because the inductor is an open initially, $I_L(0) = 0$ amps. At I_L(1 ms), the circuit is in steady-state and the inductor acts like a short. Therefore, all of the 9 volt source drops across the 150 Ω resistor, for 60 mA.

To find V_L(2 \mu s) we simply solve Equation 8.5.2.

$$V_L(t) = E\epsilon^{-\frac{t}{\tau}}$$

$$V_L(2\mu s) = 9V\epsilon^{-\frac{2\mu s}{2.667\mu s}}$$

$$V_L(2\mu s) \approx 4.251V$$

This value can also be determined graphically from Figure 9.5.2 . The time of 2 microseconds represents 75% of a time constant. Find this value on the horizontal axis and then track straight up to the solid blue curve that represents the charging inductor voltage. The point of intersection is right around 47% of the maximum value on the vertical axis. The maximum value here is the source voltage of 9 volts. Therefore the inductor will have reached approximately 47% of 9 volts, or just over 4.2 volts.

The current can be found in a similar manner using Equation 8.5.4.

$$I_L(t) = \frac{E}{R}\left(1 - \epsilon^{-\frac{t}{\tau}}\right)$$

$$I_L(2\mu s) = \frac{9V}{150\Omega}\left(1 - \epsilon^{-\frac{2\mu s}{2.667\mu s}}\right)$$

$$I_L(2\mu s) = 60mA\left(1 - \epsilon^{-0.75}\right)$$

$$I_L(2\mu s) = 31.66mA$$

Computer Simulation

To verify our analysis, the circuit of Figure 9.5.3 is entered into a simulator, as shown in Figure 9.5.4 . In order to reflect the notion of a time-varying circuit with a switch, the 9 volt DC voltage source has been replaced with a rectangular pulse voltage source. This source starts at 0 volts and then immediately steps up to 9 volts. It stays at this level for 20 microseconds before dropping back to 0 volts.

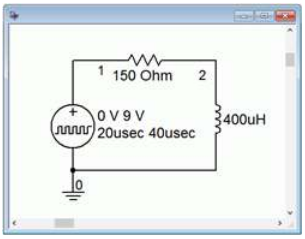


Figure 9.5.4 : The circuit of Figure 9.5.3 in a simulator.

The results of a transient analysis are shown in Figure 9.5.5 . The waveform shown tracks the inductor's voltage at node 2 with respect to ground.

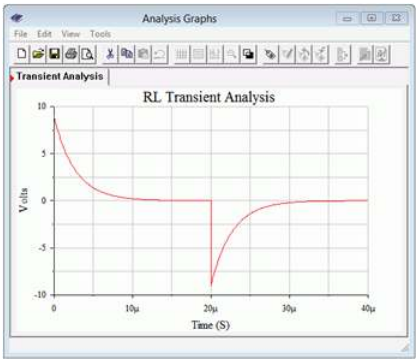


Figure 9.5.5 : Simulation results for the circuit of Figure 9.5.3 .

We can see that the voltage starts at 9 volts as expected. It then falls back to zero and is at steady-state in less than 15 microseconds, just as predicted. At 20 microseconds, the pulse source returns to zero volts. At this instant, the current through the inductor must still be the steady-state current of 60 milliamps. This current will still be flowing in a clockwise direction, thus it will produce a 9 volt drop across the 150 Ω resistor with a + to − polarity from left to right. This effectively places node 2 negative with respect to ground. The result is that the polarity of the inductor's voltage flips, with the inductor now acting as a short-lived source. We see this on the transient analysis as a negative 9 volt spike. The discharge time constant is identical to the charge constant, and thus we see the inductor's voltage fall back to zero in the same amount of time.

✔ Example 9.5.2

Given the circuit of Figure 9.5.6 , find V_L at $t = 1$ microsecond after the circuit is energized. Assume the inductor is initially uncharged.

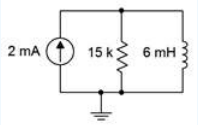


Figure 9.5.6 : Circuit for Example 9.5.2 .

First, the time constant:

$$\tau = \frac{L}{R}$$

$$\tau = \frac{6mH}{15k\Omega}$$

$$\tau = 400ns$$

Steady-state will be reached in five time constants, or 2 microseconds, at which point the inductor voltage will be zero as it will be behaving as a short. In contrast, as the inductor is initially an open $(I_L(0) = 0)$, all of the current from the source will flow into the 15 kΩ resistor, producing 30 volts across this parallel network. Therefore, we can state that $V_L(0) = 30$ volts and V_L(2 \mu s) = 0 volts, defining the extremes. In order to find V_L(1 \mu s), we can use Equation 8.5.2.

$$V_L(t) = E\epsilon^{-\frac{t}{\tau}}$$

$$V_L(1\mu s) = 30V\epsilon^{-\frac{1\mu s}{0.4\mu s}}$$

$$V_L(1\mu s) \approx 2.463V$$

There are a few different ways to crosscheck this value. For starters, we can determine the inductor current using a slight modification of Equation 8.5.4 (the current source value is used in place of E/R as the equation effectively requires the maximum or steady-state current).

$$I_L(t) = I\left(1 - \epsilon^{-\frac{t}{\tau}}\right)$$

$$I_L(1\mu s) = 2mA\left(1 - \epsilon^{-\frac{1\mu s}{0.4\mu s}}\right)$$

$$I_L(1\mu s) = 1.836mA$$

The inductor voltage of 2.463 volts would also have to appear across the parallel 15 kΩ resistor. This produces 2.463 V / 15 kΩ, or 0.164 mA of resistor current. By KCL, the remainder of the 2 mA source current must be flowing down through the inductor. This yields a net inductor current of 2 mA − 0.164 mA, or 1.836 mA, verifying our prior result.

Example 9.5.2 also reinforces the concept of the time constant being inversely proportional to the resistance, rather than directly proportional as in the RC case. In the circuit of Figure 9.5.6 , it should be obvious that the larger the resistance value, the larger the resulting initial-state voltage. From Equation 9.2.9 it can be seen that if the voltage across the inductor is increased, then the initial rate of change of current with respect to time will increase, and that implies a shorter time constant.

For more complex circuits, Thévenin's theorem may be used in order to determine the effective source voltage and charging resistance. As we saw with RC circuits, it's also possible that the discharge resistance may be considerably different from the charging resistance. In such a case, the charge and discharge curves can be highly asymmetric in both time and amplitude. Generally, the larger the discharge resistance is when compared to the charge resistance, the larger in voltage and the shorter in time the discharge spike will be (think in terms of the area under the curve staying constant).

Example 9.5.3

Assume the initial current through the inductor is zero in Figure 9.5.7 . Determine the time constant. Also, determine the inductor voltage and the voltage across the 6 kΩ resistor 200 nanoseconds after the switch is closed.

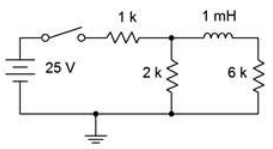


Figure 9.5.7 : Circuit for Example 9.5.3 .

This circuit is based on the circuit presented in Figure 9.3.2 as used in Example 9.3.1. In that analysis it was discovered that the steady-state voltage for the 6 kΩ and 2 kΩ resistors was 15 volts, the pair being in parallel. Further, the initial voltage across the 2 kΩ resistor and the inductor was 16.67 volts and for the 6 kΩ resistor, 0 volts.

For the Thévenin circuit, the open circuit voltage at the inductor would be the potential across the 2 kΩ resistor, which is obtained from a voltage divider between it and the 1 kΩ resistor, or 16.67 volts. The equivalent resistance is obtained by shorting the voltage source which leaves the 1 kΩ and 2 kΩ resistors in parallel, which is then in series with the 6 kΩ resistor, yielding approximately 6.667 kΩ. The equivalent is shown in Figure 9.5.8 .

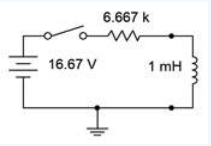


Figure 9.5.8 : Thévenin equivalent of the circuit of Figure 9.5.7 driving the inductor.

Immediately, we can see that the initial-state voltage across the inductor is confirmed in the equivalent circuit. We can now determine the time constant.

$$\tau = \frac{L}{R}$$

$$\tau = \frac{1mH}{6.667k\Omega}$$

$$\tau = 150ns$$

Steady-state will be reached in 750 nanoseconds. To find the inductor voltage we can use Equation 8.5.2.

$$V_L(t) = E\epsilon^{-\frac{t}{\tau}}$$

$$V_L(200ns) = 16.67V\epsilon^{-\frac{200ns}{150ns}}$$

$$V_L(200ns) \approx 4.39V$$

Referring back to the original circuit, in order to determine the voltage across the 6 kΩ resistor we can find the current through it and use Ohm's law. The current would be the same as the inductor's current as the two are in series. Thus, Equation 8.5.4 would do the trick.

$$I_L(t) = \frac{E}{R}\left(1 - \epsilon^{-\frac{t}{\tau}}\right)$$

$$I_L(200ns) = \frac{16.67V}{6.667k\Omega}\left(1 - \epsilon^{-\frac{200ns}{150ns}}\right)$$

$$I_L(200ns) = 2.5mA\left(1 - \epsilon^{-1.333}\right)$$

$$I_L(200ns) \approx 1.841mA$$

And finally,

$$V_{6k}(200ns) = IR$$

$$V_{6k}(200ns) = 1.841mA6k\Omega$$

$$V_{6k}(200ns) \approx 11.05V$$

Further, the voltage across the 2 kΩ resistor must be the sum, or approximately 15.44 volts.

Computer Simulation

The results of Example 9.5.3 are crosschecked in a simulator. Once again the circuit is built using a pulse generator, as shown in Figure 9.5.9 .

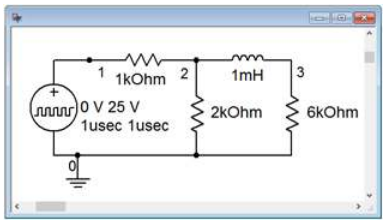


Figure 9.5.9 : Circuit of Figure 9.5.7 in a simulator.

A transient analysis is run out to 1 microsecond which is modestly into steady-state. Node voltages 2 and 3 are plotted, as shown in Figure 9.5.10 . The initial voltage across the 2 kΩ resistor (node 2) is as predicted, approximately 16.7 volts, and falls to 15 volts at steady-state, approximately 750 nanoseconds later. The voltage across the 6 kΩ resistor (node 2) starts at zero volts and also winds up at 15 volts in steadystate, just as predicted. Further, note that the predicted voltages across the 2 kΩ and 6 kΩ resistors at 200 nanoseconds are verified.

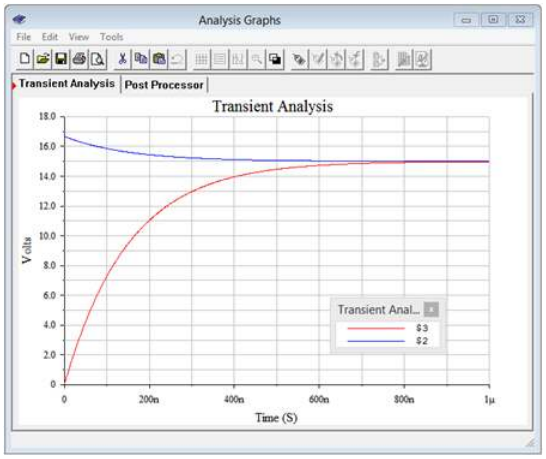


Figure 9.5.10 : Transient analysis simulation of the circuit of Figure 9.5.7 .

The voltage across the inductor is node 2 minus node 3. This differential is plotted separately in Figure 9.5.11 . The expected initial, steady-state and 200 nanosecond voltages are as predicted.

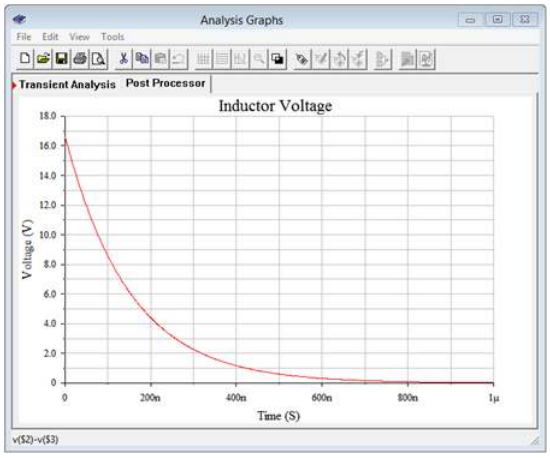


Figure 9.5.11 : Simulation of the inductor voltage versus time for the circuit of Figure 9.5.7 .

One very important observation is that if an RL circuit is abruptly altered or opened, very large voltage spikes may occur. This is due to the fact that inductor current cannot change instantaneously. If the circuit is opened, the open represents a very large resistance. Ohm's law indicates that the inductor current times this very large resistance may produce a very large voltage across the new open. In fact, the potential may be sufficient to cause a spark or arc. Note that because the current cannot change instantaneously (both magnitude and direction), the inductor now behaves as a voltage source of very high magnitude and with reverse polarity. This phenomenon is used to create the ignition spark in internal combustion engines. In short, the ignition coil is charged, creating some current flow. The circuit is then interrupted leaving just the coil in series with the spark plug, the spark plug being little more than a precisely sized gap between two electrodes. This results in a large voltage being developed across the spark plug gap, typically in the neighborhood of 20,000 volts, which is sufficient to create a small arc (i.e., the spark) that then ignites the air-fuel mixture in the piston.

An example of producing a discharge voltage spike that is considerably larger than the source voltage can be illustrated with the circuit of Figure 9.5.12 . We shall assume that the inductor is initially uncharged when power is applied, and that the switch is in position 1. In this case, the circuit consists of just the 12 volt source, the 2.2 kΩ resistor, and the inductor. The circuit reaches steady-state in roughly 227 nanoseconds. At that point the inductor behaves as a short, leaving the full 12 volt source to drop across the 2.2 kΩ resistor. This produces a clockwise current of approximately 5.455 mA.

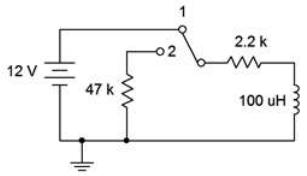


Figure 9.5.12 : Circuit illustrating large discharge voltage spike.

If we now move the switch to position 2, this current must be maintained because the current through the inductor cannot change instantaneously. The new discharge resistance is now the series combination of the two resistors, or 49.2 kΩ. Ohm's law and KVL dictate that the resulting inductor voltage must be 49.2 kΩ times 5.455 mA, or just beyond −268 volts. This potential is negative because the clockwise current is flowing up through the 47 kΩ resistor, producing a drop + to − from ground up. The increased resistance also shortens the time constant and now steady-state will be reached in only 10.1 nanoseconds. Thus, we see a much larger magnitude spike (over twenty times that of the source voltage) with a much shorter time duration (less than one-twentieth of the charge time).

Depending on the kind of switch that is used, things can be even more extreme than what has been just described. Switches come in two basic varieties: make-beforebreak and break-before-make. The former makes contact with the second position before it breaks contact with the first position, while the latter does the opposite. The behavior just described assumes that a make-before-break switch is being used. In contrast, if a break-before-make switch is used, we'd see a drastically different result.

Let us pick up back at steady-state with 5.455 mA flowing through the inductor. We now throw the switch to position 2. This new switch breaks contact with the wire leading back to voltage source prior to it making contact with the 47 kΩ resistor. For a short instant of time the switch is not making contact with anything and the resulting resistance in the loop is determined by the air gap between the switch contacts. Even if this was a mere 10 MΩ, the resulting potential would be over 50,000 volts. This will almost certainly create a spark and we will have inadvertently recreated the spark plug scenario. Hopefully, there are no combustible gases nearby.

References

[1]Which is to say, E/L amps per second times L/R seconds yields E/R amps.

[2]See also Appendix C.

This page titled 8.5: Transient Response of RL Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.5: Transient Response of RL Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

8.6: Initial and Steady-State Analysis of RLC Circuits

When analyzing resistor-inductor-capacitor circuits, remember that capacitor voltage cannot change instantaneously, thus, initially, capacitors behave as a short circuit. Once the capacitor has been charged and is in a steady-state condition, it behaves like an open. This is opposite of the inductor. As we have seen, initially an inductor behaves like an open, but once steady-state is reached, it behaves like a short. For example, in the circuit of Figure 9.4.1 , initially L is open and C is a short, leaving us with R_1 and R_2 in series with the source, E. At steady-state, L shorts out both C and R_2, leaving all of E to drop across R_1. For improved accuracy, replace the inductor with an ideal inductance in series with the corresponding R_{coil} value. Similarly, practical capacitors can be thought of as an ideal capacitance in parallel with a very large (leakage) resistance.

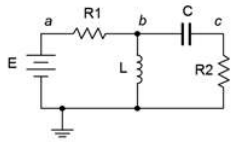


Figure 9.4.1 : Basic RLC circuit.

Example 9.4.1

Assuming the initial current through the inductor is zero and the capacitor is uncharged in the circuit of Figure 9.4.2 , determine the current through the 2 kΩ resistor when power is applied and after the circuit has reached steady-state. Draw each of the equivalent circuits.

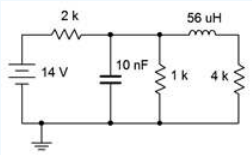


Figure 9.4.2 : Circuit for Example 9.4.1 .

For the initial-state equivalent we open the inductor and short the capacitor. The new equivalent is shown in Figure 9.4.3 . The shorted capacitor removes everything to its right from the circuit. All that's left is the source and the 2 kΩ resistor.

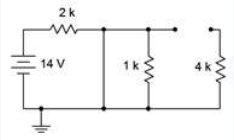


Figure 9.4.3 : Initial-state equivalent of the circuit of Figure 9.4.2 .

We can find the current through the 2 kΩ resistor using Ohm's law.

$$I_{2k} = \frac{E}{R}$$

$$I_{2k} = \frac{14V}{2k\Omega}$$

$$I_{2k} = 7mA$$

Steady-state is redrawn in Figure 9.4.4 , using a short in place of the inductor, and an open for the capacitor. We are left with a resistance of 2 kΩ in series with the parallel combination of 1 kΩ and 4 kΩ, or 2.8 kΩ in total.

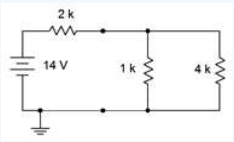


Figure 9.4.4 : Steady-state equivalent of the circuit of Figure 9.4.2 .

$$I_{2k} = \frac{E}{R}$$

$$I_{2k} = \frac{14V}{2.8k\Omega}$$

$$I_{2k} = 5mA$$

This page titled 8.6: Initial and Steady-State Analysis of RLC Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.4: Initial and Steady-State Analysis of RLC Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

8.7: Summary

The inductor is a device that stores energy in the form of a magnetic field. Inductance, L, is measured in henries, H. The idealized device consists of several loops or coils of wire. These may or may not be wrapped around a magnetic core material. Inductance is directly proportional to the permeability of the core material and the cross sectional area of the loops, and inversely proportional to the length of the coil. It is also proportional to the square of the number of loops. The most important non-ideal parameter is R_{coil}, or equivalent coil resistance. Ideally, this value is zero. Absolute accuracy, temperature stability and similar parameters round out the distinguishing features of one kind of inductor against another. When placed in series, inductors add in the same manner as resistors in series. When placed in parallel, inductors combine like resistors in parallel.

Perhaps the most important operational characteristic regarding inductors is that current through an inductor cannot change instantaneously. It will take some finite amount of time before the magnetic field reacts, leading to a predictable rise in current through it. Because of this, for DC circuits inductors initially behave like opens, but after sufficient time has passed, they behave like shorts. The amount of time required to reach steady-state is five time constants, where one time constant is defined as the inductance divided by the circuit's effective resistance. The current charge curve is of the shape $1-\varepsilon^{-t}$. The inductor's current starts at zero and rises to some maximum value. The corresponding voltage shape is of the form ε^{-t}. Here, the voltage starts at a maximum and eventually approaches zero as time passes.

Review Questions

1. What are the physical characteristics of inductors and how do they affect inductance?
2. Define the voltage-current characteristic for inductors.
3. What is R_{coil}?
4. How do inductors combine when placed in series? How do they combine when placed in parallel?
5. Define the initial and steady-state behavior of inductors.
6. Define time constant for an RL circuit.
7. How do the charge and discharge characteristics of RL circuits compare to those of RC circuits?

This page titled 8.7: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.6: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

8.8: Exercises

Analysis

1. For the circuit shown in Figure 9.7.1 , determine the effective inductance.

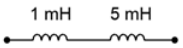


Figure 9.7.1

2. Determine the effective inductance of the network shown in Figure 9.7.2 .

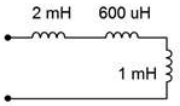


Figure 9.7.2

3. Given the inductor network shown in Figure 9.7.3 , determine the effective value.

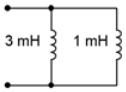


Figure 9.7.3

4. Determine the effective inductance of network shown in Figure 9.7.4 .

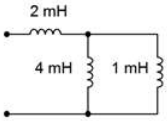


Figure 9.7.4

5. Determine the initial voltage across each component for the circuit shown in Figure 9.7.5 .

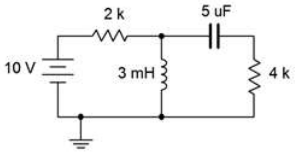


Figure 9.7.5

6. Given the network shown in Figure 9.7.5 , determine the steady-state voltage across each component.

7. Given the network shown in Figure 9.7.6 , determine the steady-state voltage across each component.

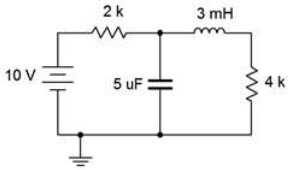


Figure 9.7.6

8. Determine the initial voltage across each component for the circuit shown in Figure 9.7.6 .

9. Given the network shown in Figure 9.7.7 , determine the steady-state voltage across each component.

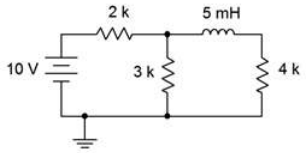


Figure 9.7.7

10. Determine the initial voltage across each component for the circuit shown in Figure 9.7.7 .

11. Determine the initial voltage across each component for the circuit shown in Figure 9.7.8 .

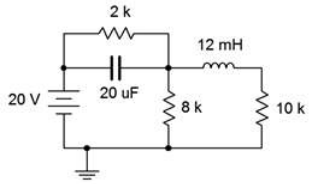


Figure 9.7.8

12. Given the network shown in Figure 9.7.8 , determine the steady-state voltage across each component.

13. Given the network shown in Figure 9.7.9 , determine the steady-state voltage across each component.

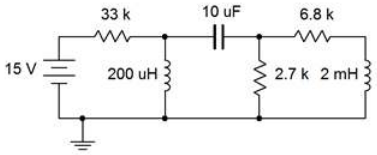


Figure 9.7.9

14. Determine the initial voltage across each component for the circuit shown in Figure 9.7.9 .

15. For the circuit shown in Figure 9.7.10 , determine the inductor current 20 microseconds after the power is switched on. Assume this is an ideal inductor with no internal resistance.

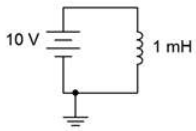


Figure 9.7.10

16. For the circuit shown in Figure 9.7.11 , determine the inductor current 100 milliseconds after the power is switched on. Assume this is an ideal inductor with no internal resistance.

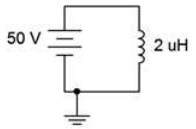


Figure 9.7.11

17. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 9.7.12 .

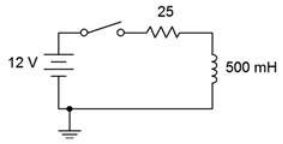


Figure 9.7.12

18. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 9.7.13 .

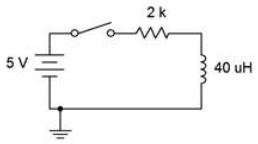


Figure 9.7.13

19. Given the circuit shown in Figure 9.7.2 , determine the inductor voltage and circulating current 200 milliseconds and 2 seconds after the switch is thrown. How does this change if we include 5 Ω of internal resistance to the inductor?

20. For the circuit shown in Figure 9.7.13 , determine the inductor voltage and circulating current 50 nanoseconds and 100 milliseconds after the switch is thrown.

21. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 9.7.14 , switch position 1.

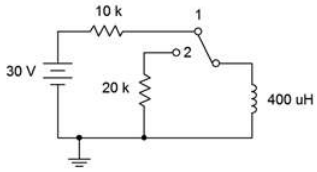


Figure 9.7.14

22. Determine the charge and discharge time constants for the circuit shown in Figure 9.7.15 .

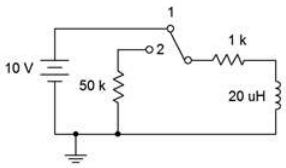


Figure 9.7.15

23. Given the circuit shown in Figure 9.7.14 , determine the inductor current 1 millisecond after the power is turned on. At this point, the switch is thrown to position 2. Determine the inductor voltage the instant of switch contact (assume ideal switch).

24. For the circuit shown in Figure 9.7.15 , determine the inductor current 400 microseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine the inductor voltage the instant of switch contact (assume ideal switch).

25. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 9.7.16 , switch position 1.

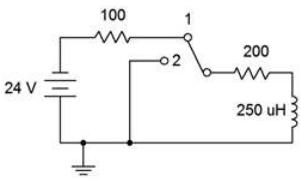


Figure 9.7.16

26. Given the circuit shown in Figure 9.7.16 , determine both the inductor current and voltage 10 milliseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the inductor to discharge to nearly zero amps. Assume ideal switch.

Design

27. For Figure 9.7.12 , determine a new resistor value such that steady-state is reached in 2 milliseconds.

28. For Figure 9.7.13 , determine a new inductor value such that steady-state is reached in 5 microseconds.

Challenge

29. Determine the time constant and the time required to reach steady-state for the circuit shown in Figure 9.7.17 .

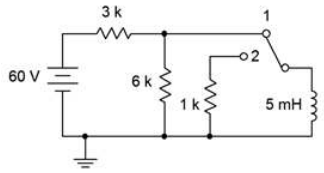


Figure 9.7.17

30. For the circuit shown in Figure 9.7.17 , determine the inductor voltage 400 microseconds after the power is turned on. At this point, the switch is thrown to position 2. Determine how long it will take the inductor to discharge to nearly zero amps. Assume ideal switch.

Simulation

31. Use a transient analysis to verify the time constant and time to steady-state of Figure 9.7.13 (problem 18).

32. Use a transient analysis to verify the inductor voltage waveform of Figure 9.7.12 as specified in problem 19.

This page titled 8.8: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **9.7: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

9: AC Signal Fundamentals

Learning Objectives

After completing this chapter, you should be able to:

- Describe sinusoidal waveforms in mathematical terms.
- Express complex waveforms using basic Fourier analysis.
- Perform mathematical operations using complex numbers.
- Draw phasor diagrams for complex numbers.
- Compute and plot capacitive and inductive reactance as a function of frequency.
- Compute and plot complex impedance as a function of frequency.
- Describe the relationships between resistance, reactance, impedance, conductance, susceptance and admittance

9.1: Introduction
9.2: Sinusoidal Waveforms
9.3: Basic Fourier Analysis
9.4: Complex Numbers
9.5: Reactance and Impedance
9.6: Phasor Diagrams
9.7: Summary
9.8: Exercises

This page titled 9: AC Signal Fundamentals is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

9.1: Introduction

In this chapter we begin our study of alternating current, or AC, electrical circuit analysis. As such, this chapter is filled with foundational material that will be used in the remainder of this text. It is critical that the concepts presented here be fully understood in order to achieve success in later chapters. We start with the mathematical description of the most simple AC waveform, the sine wave. This includes parameters such as amplitude, frequency, period, phase and DC offset. From there we discover how to describe other waveforms in terms of combinations of sine waves, and also how to determine the effective, or DC equivalent, value of a sine wave. AC analysis is practically impossible to perform without the use of complex numbers, and discussion follows with their description and proper manipulation. Finally, we introduce the concepts of reactance and impedance, and how they relate to resistance. This includes examination using both frequency domain plots and phasor diagrams. Phasor diagrams are vector plots and can be used to show the relationships between various voltages in a circuit, as well as between currents or resistive/reactive values.

Many of the topics in this text will echo your studies in DC circuit analysis, such as Ohm's law, Kirchhoff's voltage and current laws, series-parallel analysis, nodal analysis, and the like. Thus many concepts will be familiar. The major practical difference is that all quantities in DC systems are scalars, that is, they have only magnitude. In contrast, AC quantities are vectors, and thus have both magnitude and direction (or more properly, phase). Consequent;y, mathematical operations such as addition or multiplication have to follow vector rules instead of scalar rules. Forgetting to do so is a common error for students new to the subject and one that will bring no end of grief. These rules will be reviewed in the section covering complex numbers. Make sure that you have this material mastered before proceeding.

This page titled 9.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.2: Sinusoidal Waveforms

AC, or alternating current, is so named because the current alternates or flips back and forth between two polarities. In other words, the current (and consequently the voltage) is a function of time. This is fundamentally different from direct current that is fixed in polarity and generally constant over time. A laboratory DC voltage source, for example, ideally maintains a set voltage across its terminals and does not vary over time. In contrast, as an AC waveform swings back and forth through time, its shape can exhibit wide variations ranging from the simple, regular paths of laboratory standards such as sine waves, triangle waves and square waves, to the far more complex and undulating waveforms produced by musical instruments and the human voice.

The sine wave is the simplest wave that may be created. It represents the motion of a simple vector rotating at a constant speed, such as the vertical displacement of the second hand of a clock. An example is shown in Figure 9.2.1. The horizontal axis plots time. It increases as we move from left to right (i.e., if point A is to the right of point B, then A occurs later in time than does B). The vertical axis is represented here in general as a percentage of maximum but would ordinarily be a measurement of voltage, current, sound pressure, or the like.

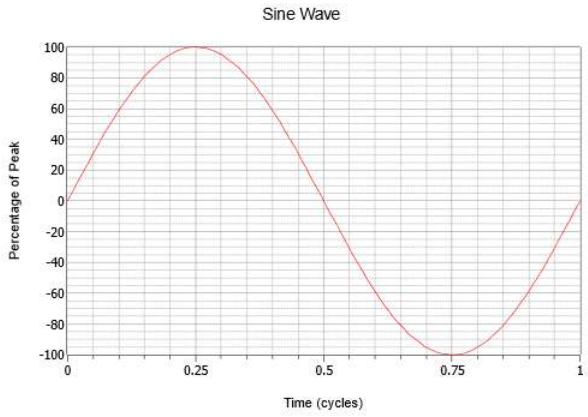


Figure 9.2.1: A sine wave.

Note the smooth variation that starts at zero, rises to a positive peak one quarter way through, falls back to zero when halfway through, continues to a negative peak three quarters through, and then rises again to where it started. This process then repeats. Each repeat is referred to as a cycle. In Figure 9.2.1, one complete cycle is shown.

Sine waves exhibit quarter wave symmetry. That is, each quarter (in time) of the wave is identical to any other if you simply flip it around the horizontal axis and/or rotate it upright about its peak. The time it takes to complete one cycle is called the period and is denoted with the symbol T (for Time). The reciprocal of the period is the frequency, f.

$$f = \frac{1}{T} \tag{9.2.1}$$

The frequency indicates how many cycles exist in one second. To honor one of the 19th century researchers in the field, instead of calling the unit “cycles per second”, we use Hertz, named after Heinrich Hertz and abbreviated Hz. In Figure 9.2.2 three sine waves are shown with different frequencies; the initial wave (green), a wave at twice the frequency (blue), and a third at half the frequency or twice the period (red).

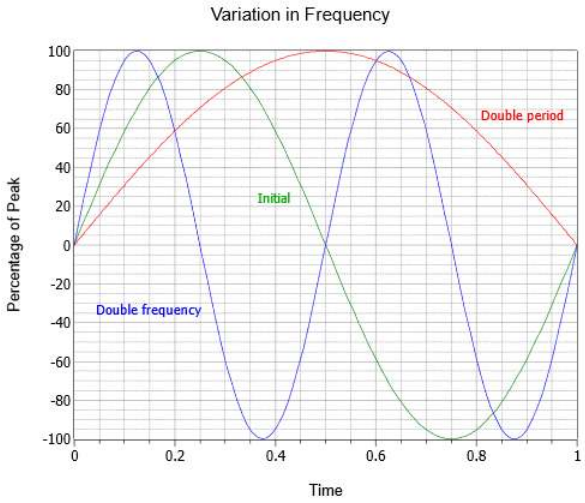


Figure 9.2.2: Sine wave frequency variation.

The amplitude (vertical) of the wave can be expressed as a peak quantity, which is the change from the center zero line up to the most positive value. Amplitude may also be expressed as peak-to-peak; the distance from the most negative to the most positive. For a sine wave this will always be twice the peak value, although that might not be the case for other waves which may be asymmetrical. A series of three sine waves with differing amplitudes are shown in Figure 9.2.3. Along side the initial (green) are double amplitude (blue) and half amplitude (red) versions.

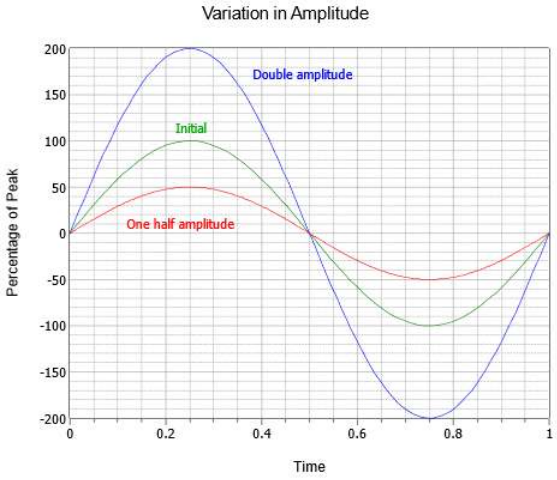


Figure 9.2.3: Sine wave amplitude variation.

Combining these parameters, consider the voltage waveform shown in Figure 9.2.4. Here we see two cycles of an AC voltage waveform.

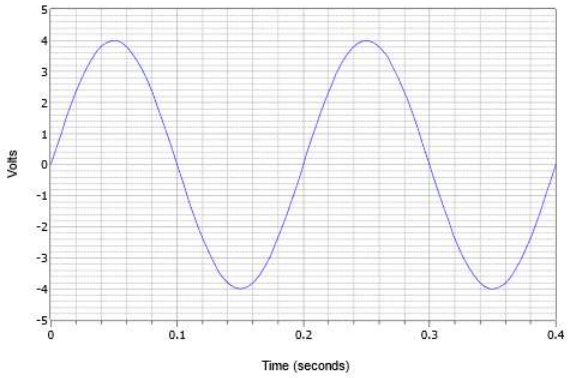


Figure 9.2.4: Basic sine wave example.

The peak value is 4 volts and the peak-to-peak value is 8 volts (typically abbreviated as “8 V pp”). The period of one cycle is 0.2 seconds, or $T = 200$ milliseconds. Further, the frequency, $f = 1/200$ milliseconds, or 5 Hz (5 cycles in one second).

AC waveforms may also be combined with a DC offset. Adding a positive DC level shifts the wave up vertically, while a negative DC level shifts the wave down vertically. This does not alter the frequency or AC portion of the amplitude (although the absolute peaks would shift by the DC value). Figure 9.2.5 shows the effect of various DC offsets. Above the initial wave (green) is an otherwise identical wave with a positive DC offset equal to 20% of the original peak value (blue). Below the original is a third wave (red) that exhibits a negative DC offset equal to half of the peak value of the original.

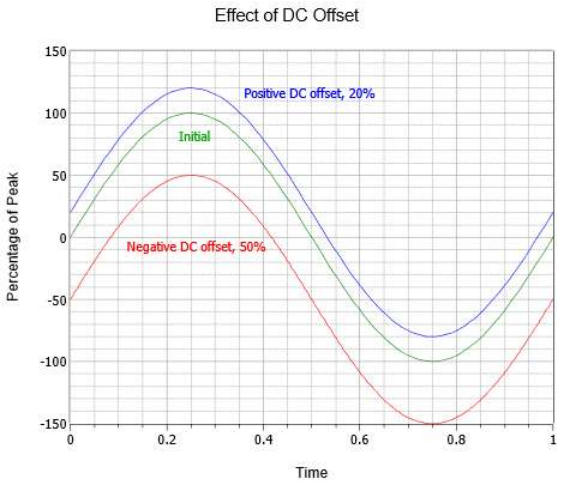


Figure 9.2.5: Sine wave DC offset variation.

Further, it is possible for a sine wave to be shifted in time compared to some other sine wave or reference. While it is possible to indicate this shift as an absolute time, it is more common to do so as a phase shift, that is, the time expressed as a portion of the period in degrees. For example, if one sine is ahead of another by one quarter of the period, it is said to be leading by 90° (i.e., 1/4 of 360°). If it is behind by ½ of the period, it is said to be lagging by 180° (i.e., later in time by 1/2 cycle). Another way of stating this is that leading waveforms start earlier in time and thus are drawn to the left of the reference, while lagging waveforms start later in time and are drawn to the right.

Figure 9.2.6 illustrates the effect of phase shift. Note that in this plot, $t = 0$ has been moved to the center of the horizontal axis. The middle curve is the initial, or reference, wave (green). To the left (red) is a wave leading the initial wave by one-eighth cycle, or 45 °. To the right (blue), is a lagging wave of half as much, or −22.5°.

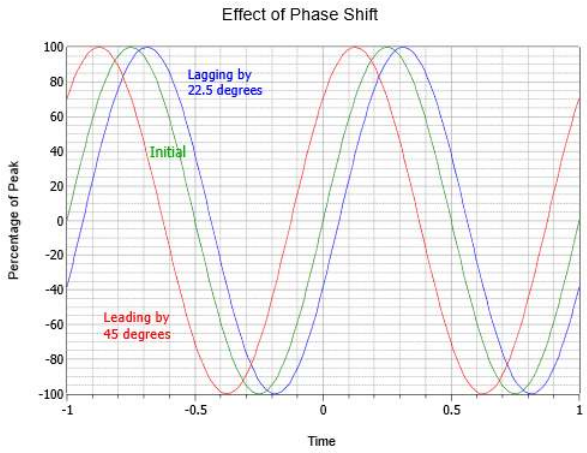


Figure 9.2.6: Sine wave phase variation.

Combining the foregoing elements allows us to develop a general format for a sine wave (voltage shown):

$$v(t) = V_{DC} + V_P \sin(2\pi ft + \theta) \tag{9.2.2}$$

where

- $v(t)$ is the voltage at some time t,
- V_{DC} is the DC offset, if any,
- V_P is the peak value,
- f is the frequency,
- θ is the phase shift (+ if leading and drawn to the left, − if lagging and drawn to the right).

For a quick and practical example, the waveform shown in Figure 9.2.4 has an amplitude of 4 volts peak, a frequency of 5 Hz, and no DC offset or phase shift. Thus, its expression is $v(t) = 4\sin(2\pi 5t)$

To compute a phase shift, first determine the time differential between the waveform and the reference, which we'll call Δt. The reference may be a fixed point in time (e.g., $t = 0$) or another waveform. Generally, the easy way to do this is to measure the difference at the zero-crossings, assuming there is no DC offset. If there is an offset, make the measurement where the zero crossing has been shifted to (i.e., at the DC offset level). Once the difference is found, divide it by the period to represent the shift as a fraction of a period. As one cycle represents one rotation of the vector, or 360 degrees, simply multiply the fraction by 360 degrees to find the phase shift in degrees. Expressed as a formula:

$$\theta = 360^\circ \frac{\Delta t}{T} \tag{9.2.3}$$

Remember, if the wave is shifted to the left then it is leading and positive, while a shift to the right is lagging or delayed in time, and thus negative.

Example 9.2.1

Write the expression for the waveform shown in Figure 9.2.7.

https://eng.libretexts.org/@go/page/52945

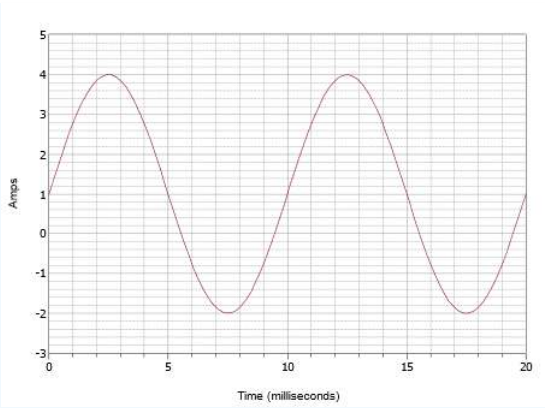


Figure 9.2.7: Waveform for Example 9.2.1.

This waveform superficially may look like the one in Figure 9.2.4 but don't let this fool you. First of all, the time scale is different. For this waveform, one cycle completes in 10 milliseconds. Therefore, the frequency is

$$f = \frac{1}{T} \tag{9.2.4}$$

$$= \frac{1}{10ms} \tag{9.2.5}$$

$$= 100Hz \tag{9.2.6}$$

The second issue is the DC offset. Note that the positive peak occurs at 4 amps while the negative peak occurs at −2 amps. This indicates a peak-topeak value of 6 amps. Without an offset, the positive peak would be at 3 amps, therefore there is a +1 amp DC offset. The vertical center of the waveform is shifted up from 0 amps to +1 amp. This point is at t = 0, therefore, there is no phase shift. The resulting expression is:

$$i(t) = 1 + 3\sin(2\pi 100t)$$

Example 9.2.2

Write the expression for the waveform shown in Figure 9.2.8.

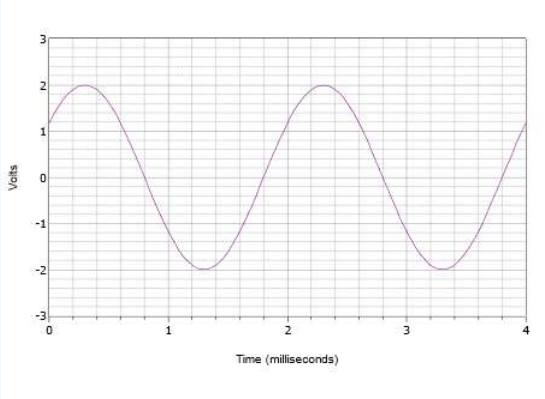


Figure 9.2.8: Waveform for Example 9.2.2.

First off, the positive peak is 2 volts and the peak-to-peak value is 4 volts. Therefore there is no DC offset. The vertical center of the wave does not start at $t = 0$, thus there must be a phase shift. The value at $t = 0$ is 1.2 volts. The wave hits this same amplitude at $t = 2$ milliseconds and begins to repeat another cycle. Consequently the period must be 2 milliseconds. The frequency is the reciprocal of this value, and thus $f = 500$ Hz.

The waveform is shifted to the left which indicates a positive or leading phase shift. If we examine the second cycle, we see that it hits zero volts at 1.8 milliseconds. Therefore the shift is 0.2 milliseconds. Expressed in degrees this is:

$$\theta = 360^\circ \frac{\Delta t}{T}$$

$$\theta = 360^\circ \frac{0.2ms}{2ms}$$

$$\theta = 36^\circ$$

The final expression is:

$$v(t) = 2\sin(2\pi 500t + 36^\circ)$$

Example 9.2.3

Draw the waveform corresponding to the following expression.

$$v(t) = -3 + 5\sin(2\pi 40000t - 72^\circ)$$

First, note that the −3 volt offset pushes the positive peak down from 5 volts to 2 volts, and the negative peak down from −5 volts to −8 volts. The frequency of 40 kHz dictates a period of:

$$\begin{aligned} T &= \frac{1}{f} \\ &= \frac{1}{40kHz} \\ &= 25\mu s \end{aligned}$$

The phase shift of -72° represents 72/360, or 0.2 cycles. This corresponds to a time delay (shifted right because it's negative) of 0.2 times 25 μs, or 5 μs.

Initially, it is often best to construct the plot via a series of discrete steps rather than trying to draw the entire thing in one go. First, draw a sine wave with a 5 volt peak amplitude and a period of 25 μs. Now, push the waveform down 3 volts so that the positive peak is only 2 volts and the negative peak is down at −8 volts. Finally, push the newly shifted waveform to the right by 5 μs. The result is shown in Figure 9.2.9.

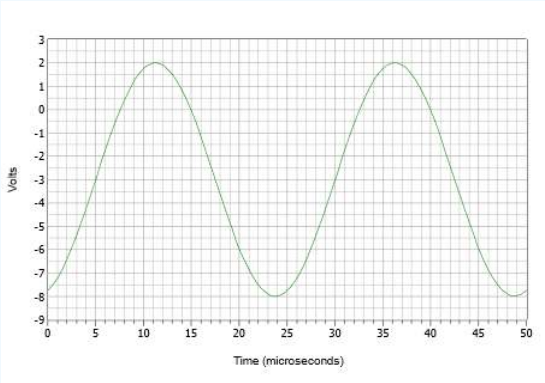


Figure 9.2.9: Waveform for Example 9.2.3.

Laboratory Measurements

In the laboratory, a function generator is used to generate sines and other waveshapes. These devices will allow precise control over both the amplitude and frequency of the wave along with adding a DC offset, if desired. An example is shown in Figure 9.2.10. The corresponding measurement tool is the oscilloscope, or just scope, for short.



Figure 9.2.10: Laboratory signal generator.

The oscilloscope is perhaps the most useful and versatile measurement device in the laboratory. Typically, they feature either two or four input channels, although more are possible. Each input channel has its own sensitivity adjustment and all channels share a common time reference. The display draws waveforms in the same manner as those seen in Figures 9.2.1 – 9.2.9. Also, they can plot one voltage versus another (X – Y mode). Modern oscilloscopes have additional features such as the automatic measurement of frequency, amplitude, phase shift, etc., cursor-based measurements, and the ability to save display images as graphics files. An example of a four channel digital oscilloscope is shown in Figure 9.2.11.



Figure 9.2.11: A digital oscilloscope.

Schematic Symbols

As far as schematics are concerned, the symbols for AC voltage and current sources are shown in Figure 9.2.12. The polarity and direction markings are not absolute; after all, these are AC sources whose polarity and directions flip back and forth. The markers are instead used to establish a timing reference, especially in circuits employing multiple sources.

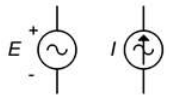


Figure 9.2.12: Schematic symbols for AC voltage source (left) and current source (right).

It is worth remembering that negating a source is the same as flipping its polarity. This was true for DC sources and remains true for AC sources. This is illustrated in Figure 9.2.13. Sometimes flipping or negating source will make analysis a little more obvious or easier to visualize.

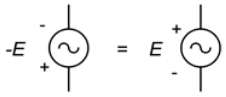


Figure 9.2.13: Polarity/sign equivalence.

Example 9.2.4

Assume an oscilloscope displays two waves as depicted in Figure 9.2.14. Determine the phase shift of the smaller 20 volt peak (blue) waveform relative to the larger 25 volt peak (red) waveform.

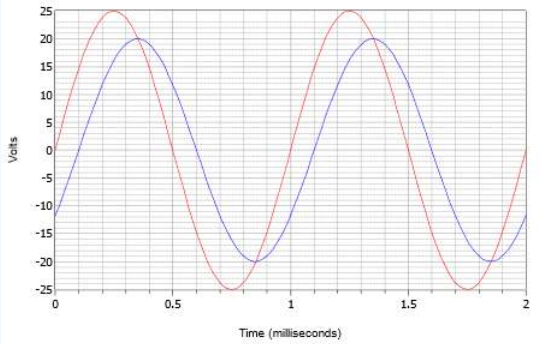


Figure 9.2.14: Waveforms for Example 9.2.4.

First, note that neither wave exhibits a DC offset. If one or both them had an offset, the wave(s) would have to be shifted vertically so that their normal zero-crossing points would be at the same level. Measuring either wave, the period is found to be 1 millisecond. The time shift most easily can be found at any of the zero-crossings (there are four locations to choose from). The delay is one small deviation, or 0.1 milliseconds, with the smaller wave delayed in time, or lagging the larger wave. This indicates a negative phase shift.

$$\theta = 360^\circ \frac{\Delta t}{T}$$

$$\theta = 360^\circ \frac{-0.1ms}{1ms}$$

$$\theta = -36^\circ$$

Sines and Cosines

There are a handful of specific phase shifts that are worth a closer look. If a sine wave is inverted, that is, flipped upside down, it is indistinguishable from a sine wave that has been shifted either +180 or −180 degrees. In other words, such a wave can be written three different ways: $-\sin(2\pi ft)$, $\sin(2\pi ft - 180^\circ)$, or $\sin(2\pi ft + 180^\circ)$. Further, if a sine wave is shifted by +90 degrees (i.e., leading and to the left), it may also be referred to as a cosine wave. Thus $\sin(2\pi ft + 90^\circ) = cos(2\pi ft)$. Finally, if a sine wave is shifted by −90 degrees (i.e., lagging and to the right), it may be referred to as a negative or inverted cosine wave. Thus $\sin(2\pi ft - 90^\circ) = -\cos(2\pi ft)$. The relationships of these four waves are illustrated in Figure 9.2.15.

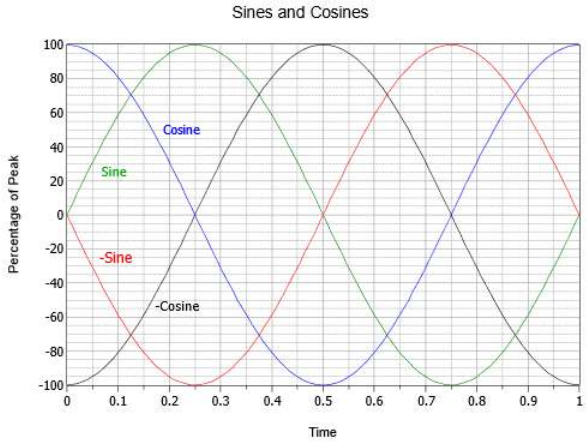


Figure 9.2.15: Time relationships between sines and cosines.

It is also worth noting that the cosine wave represents the first derivative, or slope, of the sine wave. As you may recall from other studies, the slope or “steepness” of a line is the ratio of the vertical change to the horizontal change, sometimes called “the rise over the run”. For a voltage, it would be the change in voltage over the change in time, or $\Delta V/\Delta t$. For a smooth, continuously changing curve like a sine wave, the slope at a given point is defined properly as the first derivative, or dv/dt in this case. To verify that this

is true visually, note that the steepest part of the sine wave (green) is where it crosses zero amplitude. As it crosses zero while moving positive (at $t = 0$ or $t = 1$ in Figure 9.2.15), the cosine (blue) is at its positive peak. As the sine cross zero while moving negative (at $t = 0.5$), the cosine is at its negative peak. Further, the sine wave is flat with a slope of zero at its positive and negative peaks (at at $t = 0.25$ and $t = 0.75$, respectively), and at those times the cosine's amplitude is also zero. It is also true that the sine wave is the slope of the negative cosine wave, the negative cosine is the slope of the negative sine, and the negative sine is the slope of the cosine. Moving in the reverse direction, we can say that the anti-derivative (indefinite integral) of a cosine wave is a sine wave, the integral of a sine wave is a negative cosine wave, and so forth. These relationships will prove most useful when we turn our attention to the response of capacitors and inductors in AC circuits.

– Root Mean Square Measurement

Along with peak and peak-to-peak, amplitude may be given as an RMS (Root Mean Square) value. In fact, if peak or peak-to-peak is not specified, the measurement is assumed to be RMS. RMS is a special calculation used for finding equivalent DC power (very common, for example, with audio power amplifiers). In other words, if we are interested in finding the power in a resistor, the calculation must be performed using RMS values for voltage or current, not peak or peak-to-peak values. Failure to do so will result in erroneous powers. This is true regardless of the waveshape; be it a sine wave, triangle wave or the complex waves of music signals. If a voltage is specified as RMS, it can be treated for power calculations just like an equivalently sized DC voltage. For example, a 1 volt RMS sine will produce the same power dissipation and heating in a given resistor as will 1 volt DC. For this reason, RMS is sometimes referred to as the effective value (i.e., effective DC value).

The name root-mean-square describes the process of determining the effective value. First, recall that power is proportional to the square of the voltage or current. Thus, our first step will be to square the input waveform. Of course, the waveform is a function of time and its square will yield some new shape. At this point, we need to find the average value of this new shape. The reason for this is simple, yet not necessarily obvious. Electrical and electronic components have mass, and thus do not heat or cool instantly. They exhibit a thermal time constant. Therefore, they respond to the average input over time. While we could compute some manner of “instantaneous peak power” at some specific instant in time, it does not represent the equivalent DC power. Once we have obtained the mean value of this squared waveform, the corresponding DC value is just the square root of the mean. The result is a fractional value between zero and one that is used as a scaling factor to turn a peak value into an RMS value. The value will be unique to the specific waveshape. That is, all sines (regardless of phase) have the same factor, all regular triangle waves have the same factor, and so on. As we mostly concern ourselves with sines, let's take a closer look at determining the RMS factor for them.

We begin with the basic expression for a sine wave without DC offset or phase shift, and with an amplitude of one:

$$v(t) = \sin(2\pi ft)$$

The first step is to square this waveform. A useful trigonometric identity is

$$(\sin x)^2 = \frac{1}{2} - \frac{1}{2}\cos 2x$$

Applying this to our wave yields:

$$v(t)^2 = \frac{1}{2} - \frac{1}{2}\cos(2\pi 2ft)$$

This expression describes an inverted cosine wave at twice the original frequency and half of the original amplitude, riding on a DC offset equal to its peak value. In other words, the negative peak of the cosine is at zero and the positive peak is at 1. The next step is to find the average or mean value of this intermediate result. The mean is equal to the offset of 0.5. This can be visualized as the area above the offset perfectly filling the “dip” below the offset. The final step is to take the square root of the mean. The square root of 0.5 is equal to one over the square root of two, or approximately 0.707. Therefore the RMS value is 0.707 times the peak. Alternately, you could divide the peak by square root of two, or approximately 1.414. This process is shown graphically in Figure 9.2.16.

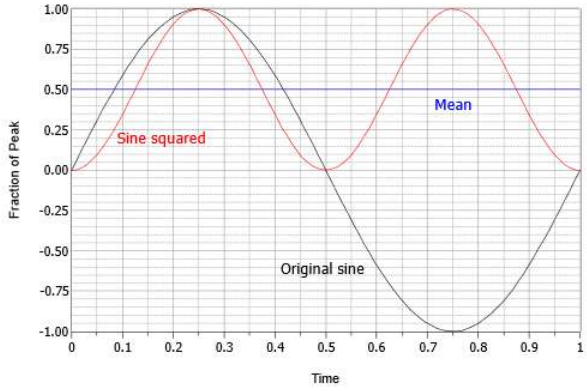


Figure 9.2.16: Process to find RMS factor for sines.

In summation, for sine waves, RMS is always the peak value times 0.707. We could also say the RMS value of any sine wave is its peak divided by approximately 1.414. Again, these ratios would not necessarily be true of non-sine waves. Details regarding other common shapes can be found in Appendix C. Finally, the ratio of the peak value to the RMS value is called the crest ratio. This is a fixed value for sine waves (again, about 1.414), but can be over 10:1 for some kinds of audio signals.

Wavelength

Another item of interest is the speed of propagation of the wave. This varies widely. In the case of light in a vacuum (or to a close approximation, an electrical current in a wire), the velocity is about 3E8 meters per second (i.e., 300,000 km/s) or about 186,000 miles per second.

Given a velocity and a period, we can imagine how far apart the peaks of the wave are. This distance is called the wavelength and is denoted by the Greek letter lambda λ. Wavelength is equal to the velocity divided by the frequency, $\lambda = v/f$. Thus, for a loudspeaker producing a 100 Hz sine, as the velocity of sound in air is 344 m/s, then $\lambda = 344 m/s$ / 100 Hz, or 3.44 meters (a little over 11 feet). Notice that the higher the frequency, the shorter the wavelength. Also, note that the faster the velocity, the longer the wavelength. Wavelength calculations are of particular importance in the fields of telecommunications and acoustics.

This page titled 9.2: Sinusoidal Waveforms is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.2: Sinusoidal Waveforms** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.3: Basic Fourier Analysis

The Fourier theorem, named after the French mathematician Jean-Baptiste Joseph Fourier, states that any repetitive waveform can be represented as a collection of sine and cosine waves of the proper amplitude and frequency. Alternately, it may be represented as a series of sine waves each with the proper amplitude, frequency and phase. This includes complex signals such as the human voice and musical instruments. Consequently, if a system is linear, the response of a system to a complex wave may be understood in terms of its response to individual sine waves, via superposition.

In this collection of waves, each component is known as a partial with the lowest frequency component known as the fundamental. All other partials are grouped together and referred to as overtones. “Regular” waveforms such as square waves and triangle waves feature a harmonic overtone sequence meaning that these overtones are integer multiples of the fundamental. As a shortcut, they are often referred to as just harmonics.

It might be hard to visualize initially, but like all waves, waves in the shape of a square or triangle are made up of a series of sines. The equation for a square wave is:

$$v(t) = \sum_{n=1}^{\infty} \frac{1}{2n-1} \sin((2n-1)2\pi ft) \tag{9.3.1}$$

This says that a square wave of frequency f is made up of an infinite series of sines at odd integer multiples of f, with an inverse amplitude characteristic. For example, a 100 Hz square consists of a 100 Hz sine, plus a 300 Hz sine at 1/3 amplitude, plus a 500 Hz sine at 1/5 amplitude, plus a 700 Hz sine at 1/7 amplitude, and so on.

A triangle wave is similar:

$$v(t) = \sum_{n=1}^{\infty} \frac{1}{(2n-1)^2} \cos((2n-1)2\pi ft) \tag{9.3.2}$$

Thus a triangle wave of frequency f is made up of an infinite series of cosines (sines with a 90 degree or one quarter cycle phase shift) at odd integer multiples of f, with an inverse square amplitude characteristic. For example, a 100 Hz triangle consists of a 100 Hz cosine, plus a 300 Hz cosine at 1/9 amplitude, plus a 500 Hz cosine at 1/25 amplitude, plus a 700 Hz cosine at 1/49 amplitude, and so on.

A series of graphs showing the construction of a square wave and a triangle wave follow. The square wave sequence begins with the fundamental and the first harmonic in Figure 9.3.1. The result is an oddly bumpy wave. The second graph of Figure 9.3.2 adds the next two harmonics. As more harmonics are added, the sides get steeper and the top/bottom start to flatten. They flatten because each additional harmonic partially cancels some of the peaks and valleys from the previous summation. This gives rise to a greater number of undulations with each undulation being smaller in amplitude. The sequence finishes with Figure 9.3.3 showing seven harmonics being added with the result approaching a reasonable square wave. As ever more harmonics are added, the wave would approach a flat top and bottom with vertical sides, the idealized square wave.

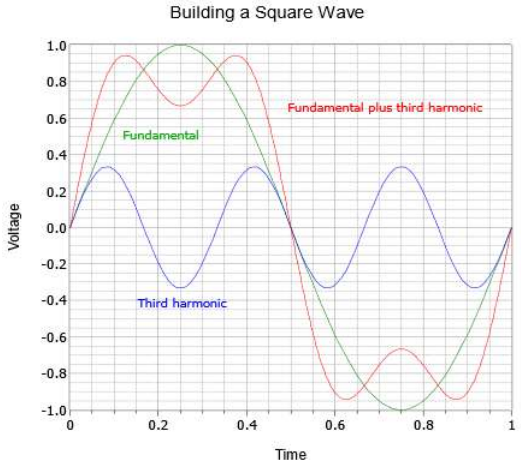


Figure 9.3.1: Building a square wave, part 1.

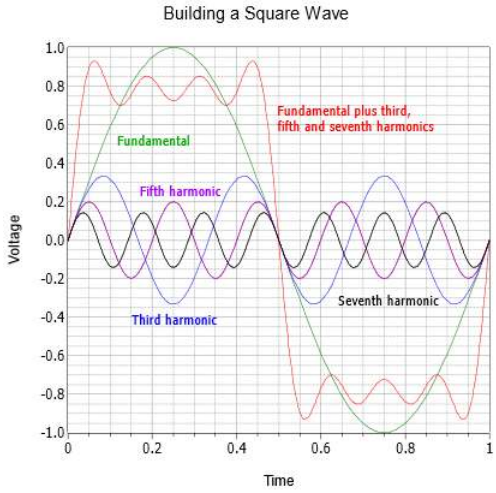


Figure 9.3.2: Building a square wave, part 2.

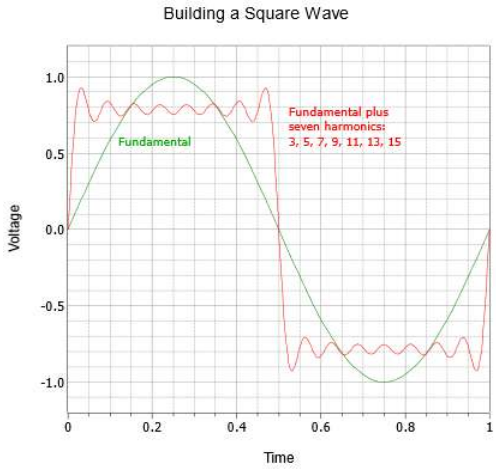


Figure 9.3.3: Building a square wave, part 3.

The triangle sequence begins with a fundamental and the first harmonic as shown in Figure 9.3.4. The resulting combination is already trending away from a simple sine shape. The second and final graph, Figure 9.3.5, shows a total of seven harmonics. The result is very close to a triangle, the only obvious deviation is the slight rounding at the very peaks. The addition of more harmonics would cause these to sharpen further.

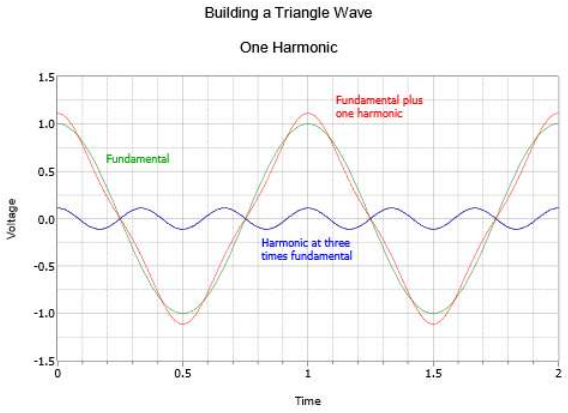


Figure 9.3.4: Building a triangle wave, part 1.

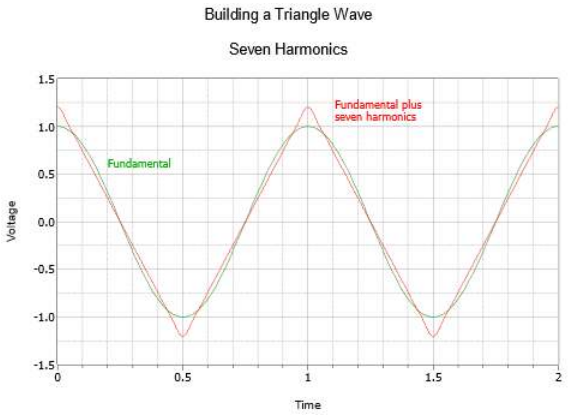


Figure 9.3.5: Building a triangle wave, part 2.

This page titled 9.3: Basic Fourier Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.3: Basic Fourier Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.4: Complex Numbers

In AC circuits, parameters such as voltage and current are vectors, that is, they have both a magnitude and a phase shift or angle. For example, a voltage might be “12 volts at an angle of 30 degrees” (or more compactly, $12\angle 30^\circ$). This is known as polar form or magnitude-angle form. Alternately, a vector can be broken into rectangular form, that is, its right angle components.

This can be visualized as a right triangle where the magnitude is the hypotenuse, the angle is the rotation above or below the horizontal, the horizontal component is the side adjacent to the angle and the vertical component is the side opposite of the angle. This is shown in Figure 9.4.1.

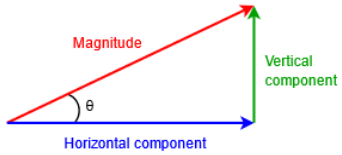


Figure 9.4.1: Polar and rectangular forms.

Properly, voltage and current vectors are complex numbers that lay on a complex plane consisting of a real part and an imaginary part. The horizontal axis is the real number axis and the vertical axis is the imaginary number axis. The imaginary axis denotes values times the imaginary operator j (and often referred to as i outside of electrical analysis). The j operator is the square root of -1. An example of such a value is $3 + j4$, in other words, 3 units along the horizontal real axis and 4 units up the vertical imaginary axis. This is depicted in Figure 9.4.2.

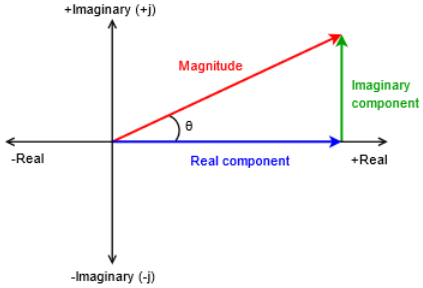


Figure 9.4.2: Polar and rectangular forms on the complex plane.

Converting from one form to another relies on basic trigonometric relations. For convenience, the relationships between the magnitude, angle, real component and imaginary component are reproduced below:

$$\text{Magnitude} = \sqrt{\text{Real}^2 + \text{Imaginary}^2} \tag{9.4.1}$$

$$\theta = \tan^{-1}\frac{\text{Imaginary}}{\text{Real}} \tag{9.4.2}$$

$$\text{Real} = \text{Magnitude}\ \cos\theta \tag{9.4.3}$$

$$j\text{Imaginary} = \text{Magnitude}\ \sin\theta \tag{9.4.4}$$

To add or subtract complex quantities, first put them into rectangular form and then combine the reals with the reals and the j terms with the j terms as in $(3 + j5) + (13 - j1) = 16 + j4$. These real and imaginary terms must be kept separate. $3 + j5$ does not equal 8 (or even $j8$). That would be like saying that moving 3 feet to your right and 5 feet forward puts you in the same location as moving 8 feet to your right (or 8 feet forward).

The direct way to multiply or divide complex values is to first put them in polar form and then multiply or divide the magnitudes. The angles are added together for multiplication and subtracted for division. For example, $12\angle 30^\circ$ times $2\angle 45^\circ$ is $24\angle 75^\circ$ while dividing them yields $6\angle -15^\circ$. The need for complex numbers will become more obvious as we move through the upcoming material. It is imperative that you have mastered the manipulation of complex values before moving on to subsequent chapters.

Example 9.4.1

Convert $15 + j20$ and $1k - j2k$ into polar form, and $10\angle 45^\circ$ and $0.2\angle -30^\circ$ into rectangular form.

For the first two conversions, use Equations 9.4.1 and 9.4.2.

$$\text{Magnitude } = \sqrt{15^2 + 20^2} = 25$$

$$\theta = \tan^{-1}\left(\frac{20}{15}\right) = 53.1^\circ$$

$$\text{Magnitude } = \sqrt{1k^2 + (-2k)^2} = 2.236k$$

$$\theta = \tan^{-1}\left(\frac{-2k}{1k}\right) = -63.4^\circ$$

The answers for the first part are $25\angle 53.1^\circ$ and $2.236k\angle -63.4^\circ$.

For the second pair of conversions use Equations 9.4.3 and 9.4.4.

$$\text{Real } = 10\cos 45^\circ = 7.07$$

$$j\text{Imaginary } = 10\sin 45^\circ = j7.07$$

$$\text{Real } = 0.2\cos -30^\circ = 0.173$$

$$j\text{Imaginary } = 0.2\sin -30^\circ = -j0.1$$

The answers for the second part are $7.07 + j7.07$ and $0.173 - j0.1$.

Example 9.4.2

a) Add $7 + j8to + j6$, b) Subtract $5\angle 53.1^\circ$ from $10\angle -45^\circ$, c) Divide $20\angle 90^\circ$ by $4\angle -50^\circ$, d) Multiply $90 + j75$ by $6 + j10$.

a) Add real to real and imaginary to imaginary: $7 + j14$.

b) First convert the values into rectangular: $3 + j4$ and $7.07 - j7.07$. Now subtract the first pair from the second pair: $4.07 - j11.07$ (or $11.8\angle -69.8^\circ$).

c) Divide the magnitudes and subtract the angles: $5\angle 140^\circ$.

d) First convert the values into polar: $117.15\angle 39.8^\circ$ and $11.66\angle 59^\circ$. Now multiply the magnitudes and add the angles: $1.366k\angle 98.8^\circ$.

This page titled 9.4: Complex Numbers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.4: Complex Numbers** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.5: Reactance and Impedance

Unlike a resistor, the voltage and current will not be in phase for an ideal capacitor or for an ideal inductor. For the capacitor, the current leads the voltage across the capacitor by 90 degrees. Recall that the voltage across a capacitor cannot change instantaneously, $i = C\,dv/dt$. For an inductor, the voltage leads the current by 90 degrees. Similarly, the current through an inductor cannot change instantaneously due to $v = L\,di/dt$. While ideal capacitors and inductors do not exhibit resistance, the voltage does react to the current. Unsurprisingly, we call this characteristic reactance and denote it with the letter X. Reactance, like resistance, is a ratio of voltage to current. We define capacitive reactance as:

$$X_C = \frac{v_c}{i_c} \tag{9.5.1}$$

Using a simple sine wave with a peak value of unity, $v_c = \sin 2\pi ft$, i_c can be found from $i = C\,dv/dt$. Recall from our prior work that the derivative or slope of a sine wave is a cosine wave, which is in turn equivalent to a sine wave shifted by +90°.

$$i_c = C\frac{dv_c}{dt}$$

$$i_c = C\frac{d\sin(2\pi ft)}{dt}$$

$$i_c = C\sin(2\pi ft + 90^\circ)$$

Substituting this result into Equation 1.7 yields:

$$X_C = \frac{v_c}{i_c}$$

$$X_C = \frac{\sin(2\pi ft)}{2\pi fC\sin(2\pi ft + 90^\circ)}$$

$$X_C = \frac{1}{2}\pi fC\angle -90^\circ$$

Or more directly:

$$X_C = -j\frac{1}{2\pi fC} \tag{9.5.2}$$

In summation, if we divide the capacitor's sinusoidal voltage by its current (Ohm's law), we obtain a value with a phase angle of −90°. While the resultant is an ohmic value, it cannot be classified as a resistance. Instead, it is referred to as a reactance and denoted with the letter X. Thus we can refer to a capacitor's reactance, X_C, as some number at an angle of −90°, or more conveniently, we simply prepend $-j$, as in $X_C = -j75\Omega$.

The case for the inductor is similar and left as an exercise. The inductive reactance, X_L, can be found using:

$$X_L = +j2\pi fL \tag{9.5.3}$$

An example would be $X_L = j68\Omega$. As stated, while a resistance cannot be added directly to a reactance, reactances can be added together so long as we heed the signs. For example, if we have a capacitive reactance of $-j60\Omega$ in series with an inductive reactance of $j100\Omega$, the result is $j40\Omega$. This is due to the fact that these two items are 180 degrees out of phase with each other, and so they partially cancel. Remember, always add or subtract like items: real (resistance) to real, and imaginary (reactance) to imaginary.

Equations 9.5.2 and 9.5.3 are notable because the reactance is not just a function of the capacitance or inductance, but also a function of frequency. The reactance of an inductor is directly proportional to frequency while the reactance of a capacitor is inversely proportional to frequency. The ohmic variations of a 20Ω resistor, a 500 μF capacitor and a 500 μH inductor across frequency are shown in Figure 9.5.1.

We can see that the value of resistance does not change with frequency while the inductive reactance increases with frequency and the capacitive reactance decreases.

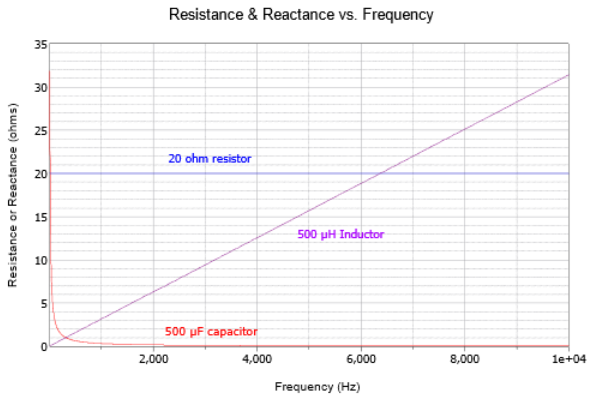


Figure 9.5.1: Resistance and reactance versus frequency (linear axis).

The linear frequency scale makes the capacitor change difficult to see. If this is plotted again but using a logarithmic frequency scale as in Figure 9.5.2, the symmetry becomes apparent.

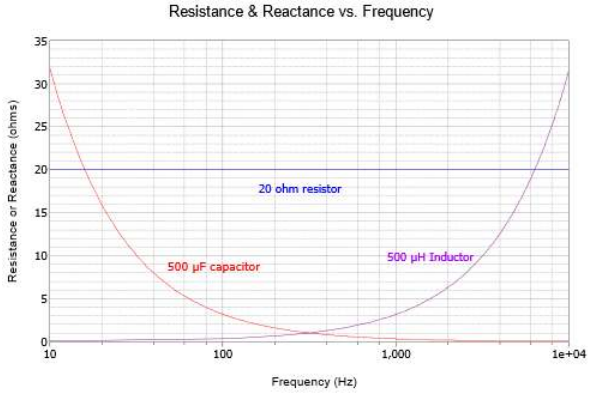


Figure 9.5.2: Resistance and reactance versus frequency (log axis).

The effect of both capacitor size and frequency is shown in Figure 9.5.3 using a log frequency axis: the smaller the capacitor, the larger the capacitive reactance at any particular frequency.

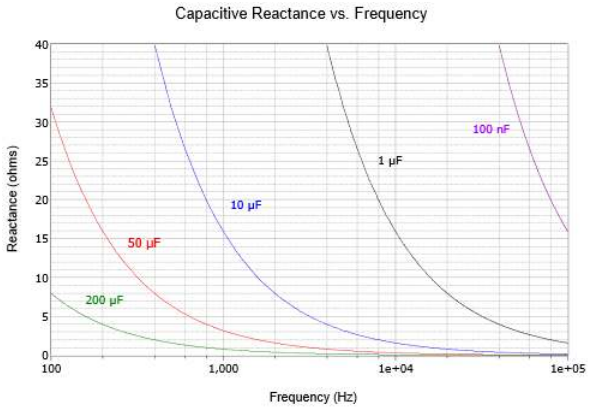


Figure 9.5.3: Variation of capacitive reactance with capacitance and frequency.

Similarly, the effect of both inductor size and frequency is shown in Figure 9.5.4 using a linear frequency axis: the larger the inductor, the larger the inductive reactance at any given frequency.

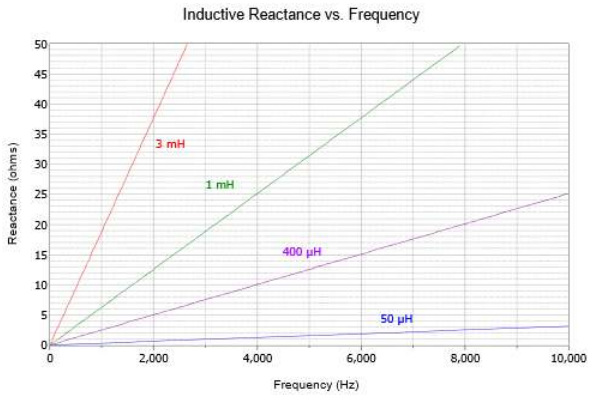


Figure 9.5.4: Variation of inductive reactance with inductance and frequency.

It is worth noting that the plots of Figures 9.5.3 and 9.5.4 are for ideal components. In reality, all components exhibit some resistive, capacitive and inductive effects due to their construction. For example, eventually, inductive and resistive effects will cause the capacitive reactance curves of Figure 9.5.3 to begin to rise at high frequencies. Similarly, resistive and capacitive effects will cause the curves Figure 9.5.4 to flatten at very low and very high frequencies.

An interesting observation to remember is that capacitors and inductors are a bit like a Chimera. That is, they look like different things to different sources, all mashed together at once. It would be improper to think of, say, a particular inductor as being "so many ohms". If the source signal is comprised of multiple sine waves, such as a square wave or a music waveform, the inductor "looks like" a different ohmic value to each of the different frequency components, simultaneously. This is an important concept and one that we can take great advantage of, for example, in the design of filter circuits.

Impedance

We now arrive at impedance. Impedance is a mixture of resistance and reactance, and is denoted by Z. This can be visualized as a series combination of a resistor and either a capacitor or an inductor. Examples include $Z = 100 - j50\Omega$, i.e., 100 ohms of resistance in series with 50 ohms of capacitive reactance; and $Z = 600\angle 45^{\circ}\Omega$, i.e., a magnitude of 600 ohms that includes resistance and inductive reactance (it must be inductive reactance and not capacitive reactance because the sign of the angle is positive).

To complete this system, we have susceptance and admittance. Susceptance, S, is the reciprocal of reactance. Admittance, Y, is the reciprocal of impedance. These are similar to the relation between conductance and resistance, and are convenient for parallel circuit combinations.

Example 9.5.1

Determine the reactances of a 1 mH inductor and a 2 μF capacitor to a sine wave of 2 kHz. Repeat for a frequency of 50 kHz.

Use Equations 9.5.2 and 9.5.3. For the capacitor at 2 kHz we have:

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 1kHz2\mu F}$$

$$X_C = -j79.6\Omega$$

For the second source, 50 kHz is 25 times larger than the original and capacitive reactance is inversely proportional to frequency. Therefore X_C is 25 times smaller, or $-j3.18\Omega$.

For the inductor at 2 kHz,

$$X_L = +j2\pi fL$$

$$X_L = +j2\pi 2kHz1mH$$

$$XL = +j12.57\Omega$$

Inductive reactance is directly proportional to frequency. Thus, increasing f by a factor of 25 increases X_L by the same factor, resulting in $j314.2\Omega$.

Example 9.5.2

Determine the susceptance of an inductor whose reactance is $j400\Omega$. Further, if this inductor is placed in series with a 1000Ω resistor, determine the resulting impedance in polar form, as well as the admittance.

Susceptance is the reciprocal of reactance.

$$S_L = \frac{1}{X_L}$$

$$S_L = \frac{1}{j400\Omega}$$

$$S_L = -j2.5 \text{ millisiemens}$$

The impedance, Z, is the vector sum, or $1000 + j400\Omega$. Converting this to polar form:

$$\text{Magnitude } = \sqrt{\text{Real}^2 + \text{Imaginary}^2}$$

$$\text{Magnitude } = \sqrt{1000^2 + 400^2}$$

$$\text{Magnitude } = 1077$$

$$\theta = \tan^{-1}\left(\frac{\text{Imaginary}}{\text{Real}}\right)$$

$$\theta = \tan^{-1}\left(\frac{400}{1000}\right)$$

$$\theta = 21.8^\circ$$

The result is $Z = 1077\angle 21.8^\circ\Omega$. The admittance is the reciprocal, yielding $Y = 928E-6\angle -21.8^\circ\mu S$.

This page titled 9.5: Reactance and Impedance is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.5: Reactance and Impedance** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.6: Phasor Diagrams

A time domain representation of sine waves as drawn earlier tells us everything we need to know about the waves, however it is not the most compact method of displaying them, nor are they easy to sketch by hand. Consider the the plot of Figure 9.6.1. Here, two sine waves (green and red) combine to create a third sine wave (blue). While the general idea of the two waves adding is apparent in the graph, it takes a moment of inspection to determine the wave magnitudes and the precise phase relationships between them. In contrast, phasor diagrams can be used to show the relationships of multiple sines waves in a simple and easy to read format. They can also be used to show how various voltages or currents combine.

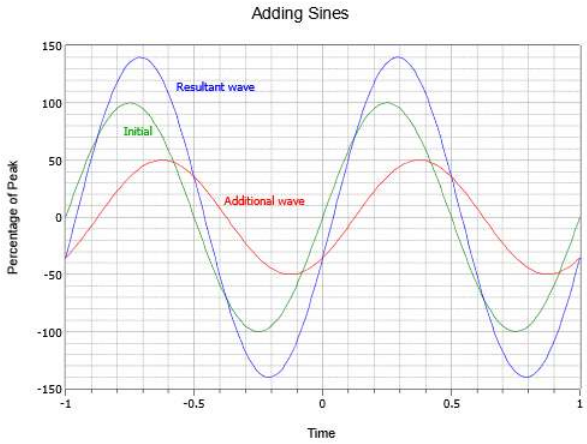


Figure 9.6.1: A time domain plot.

The phasor diagram is based on the complex plane discussed previously where the horizontal is the real axis and the vertical is the imaginary (j) axis. The magnitude and phase of each wave can then be drawn as a vector, and the relationships between the waves is shown directly. For manual plotting, it is convenient to convert from polar form to rectangular form. In the example phasor diagram of Figure 9.6.2, two vectors are shown: $8 + j6$ and $5 - j3$ (equivalent to $10\angle 36.9^\circ$ and $5.83\angle -31^\circ$). Phasor diagrams can be used to plot voltages, currents and impedances. We shall make considerable use of them in upcoming chapters.

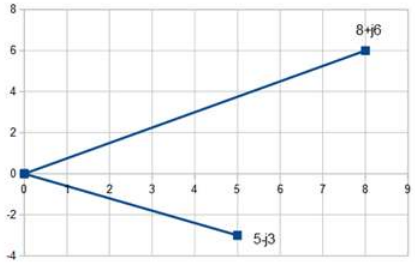


Figure 9.6.2: A phasor diagram.

This page titled 9.6: Phasor Diagrams is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.6: Phasor Diagrams** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.7: Summary

The most simple of AC waveforms is the sine wave. It can be thought of as the vertical displacement of a vector rotating at a constant speed, like the second hand of a clock. The length of the second hand represents the height or amplitude of the sine wave and the speed of rotation represents its period. As the speed tends to be very fast, it is more convenient for us to use frequency, which is just the reciprocal of the period. Sines can be displaced vertically, which is also called a DC offset, as well as having a horizontal or time shift. When expressed relative to a single cycle, this change is referred to as the phase shift. Finally, waveforms such as square waves, triangle waves and even more complex waveforms such as voice or music can be described in terms of combinations of sounds. Indeed, Fourier analysis tells us that any repeating waveform can be described as a series of sines each with the appropriate frequency, amplitude and phase shift. In order to determine the “effective DC value” of a sine wave, that is, the value that produces the same power dissipation, RMS values are used. The RMS value of a sine wave is its peak value divided by the square root of two (approximately equal to 0.707 of peak).

Complex numbers are used to describe AC voltages and currents, among other things. They consist of two parts: a real part and an imaginary part that is plotted perpendicular to the real part. All mathematical operations on complex numbers must follow vector rules. This includes basic trigonometric operations.

Reactance can be thought of as the imaginary axis version of resistance. That is, it restricts current flow. The difference is that there is a 90 degree phase shift between the current and voltage through a reactive element while the two are in phase for resistive elements. Capacitive reactance is inversely proportional to frequency while inductive reactance is directly proportional to frequency. The combination of resistance and reactance is known as impedance. Phasor diagrams may be used to plot the components of a complex impedance as well as show the relations between voltages or currents in a circuit.

Review Questions

1. What is a sine wave? Describe its constituent parameters (amplitude, frequency, offset, etc.).

2. What is the relationship between the frequency, period and phase of a sine wave?

3. What is the mathematical relationship between a sine wave and a cosine wave?

4. What is meant by the term RMS (root-mean-square) and what does it have to do with AC versus DC voltages, currents and powers?

5. Describe the difference between a scalar and a vector.

6. What is a complex number?

7. Detail the relationships between resistance, reactance, impedance, conductance, susceptance and admittance.

8. Describe how capacitive reactance varies with frequency.

9. Describe how inductive reactance varies with frequency.

10. Describe a phasor diagram.

This page titled 9.7: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.7: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

9.8: Exercises

Analysis

1. Determine the AC peak and RMS voltages, DC offset, frequency, period and phase shift for the following expression: $v(t) = 10 \sin 2\pi 1000t$

2. Determine the AC peak and RMS voltages, DC offset, frequency, period and phase shift for the following expression: $v(t) = 0.4 \sin 2\pi 5000t$

3. Determine the peak AC portion voltage, DC offset, frequency, period and phase shift for the following expression: $v(t) = -3 + 20 \sin 2\pi 50t$

4. Determine the peak AC portion voltage, DC offset, frequency, period and phase shift for the following expression: $v(t) = 12 + 2 \sin 2\pi 20000t$

5. Determine the AC peak and RMS voltages, DC offset, frequency, period and phase shift for the following expression: $v(t) = 10 \sin(2\pi 100t + 45^\circ)$

6. Determine the AC peak and RMS voltages, DC offset, frequency, period and phase shift for the following expression: $v(t) = 5 \sin(2\pi 1000t - 90^\circ)$

7. Determine the peak AC portion voltage, DC offset, frequency, period and phase shift for the following expression: $v(t) = 10 + 1 \sin(2\pi 400t - 45^\circ)$

8. Determine the peak AC portion voltage, DC offset, frequency, period and phase shift for the following expression: $v(t) = 10 + 10 \sin(2\pi 5000t + 30^\circ)$

9. A 1 kHz sine wave has a phase of 72°. Determine the time delay. Repeat for a 20 kHz sine wave.

10. A 2 kHz sine wave has a phase of 18°. Determine the time delay. Repeat for a 100 kHz sine wave.

11. An oscilloscope measures a time delay of 0.2 milliseconds between a pair of 500 Hz sine waves. Determine the phase shift.

12. An oscilloscope measures a time delay of −10 microseconds between a pair of 20 kHz sine waves. Determine the phase shift.

13. Convert the following from rectangular to polar form:

a) $10 + j10$

b) $5 - j10$

c) $-100 + j20$

d) $3k + j4k$

14. Convert the following from rectangular to polar form:

a) $2k + j1.5k$

b) $8 - j8$

c) $-300 + j300$

d) $-1k - j1k$

15. Convert these from polar to rectangular form:

a) $10\angle 45^\circ$

b) $0.4\angle 90^\circ$

c) $-9\angle 60^\circ$

d) $100\angle -45^\circ$

16. Convert these from polar to rectangular form:

a) $-4\angle 60^\circ$

b) $-0.9\angle 30^\circ$

c) $5\angle 120^\circ$

d) $6\angle -135^\circ$

17. Perform the following computations:

a) $(10+j10)+(5+j20)$

b) $(5+j2)+(-5+j2)$

c) $(80-j2)-(100+j2)$

d) $(-65+j50)-(5-j200)$

18. Perform the following computations:

a) $(100+j200)+(75+j210)$

b) $(-35+j25)+(15+j8)$

c) $(500-j70)-(200+j30)$

d) $(-105+j540)-(5-j200)$

19. Perform the following computations:

a) $(100+j200)\cdot(75+j210)$

b) $(-35+j25)\cdot(15+j8)$

c) $(500-j70)/(200+j30)$

d) $(-105+j540)/(5-j200)$

20. Perform the following computations:

a) $(10+j10)\cdot(5+j20)$

b) $(5+j2)\cdot(-5+j2)$

c) $(80-j2)/(100+j2)$

d) $(-65+j50)/(5-j200)$

21. Perform the following computations:

a) $(10\angle 0^\circ)\cdot(10\angle 0^\circ)$

b) $(5\angle 45^\circ)\cdot(-2\angle 20^\circ)$

c) $(20\angle 135^\circ)/(40\angle -10^\circ)$

d) $(8\angle 0^\circ)/(32\angle 45^\circ)$

22. Perform the following computations:

a) $(0.3\angle 0^\circ)\cdot(3\angle 180^\circ)$

b) $(5\angle -45^\circ)\cdot(-4\angle 20^\circ)$

c) $(0.05\angle 95^\circ)/(0.04\angle -20^\circ)$

d) $(500\angle 0^\circ)/(60\angle 225^\circ)$

23. Perform the following computations:

a) $(0.3\angle 0^\circ)+(3\angle 180^\circ)$

b) $(5\angle -45^\circ)+(-4\angle 20^\circ)$

c) $(0.05\angle 95^\circ) - (0.04\angle -20^\circ)$

d) $(500\angle 0^\circ) - (60\angle 225^\circ)$

24. Perform the following computations:

a) $(10\angle 0^\circ) + (10\angle 0^\circ)$

b) $(5\angle 45^\circ) + (-2\angle 20^\circ)$

c) $(20\angle 135^\circ) - (40\angle -10^\circ)$

d) $(8\angle 0^\circ) - (32\angle 45^\circ)$

25. Determine the capacitive reactance of a 1 μF capacitor at the following frequencies:

a) 10 Hz

b) 500 Hz

c) 10 kHz

d) 400 kHz

e) 10 MHz

26. Determine the capacitive reactance of a 220 pF capacitor at the following frequencies:

a) 10 Hz

b) 500 Hz

c) 10 kHz

d) 400 kHz

e) 10 MHz

27. Determine the capacitive reactance at 50 Hz for the following capacitors:

a) 10 pF

b) 470 pF

c) 22 nF

d) 33 μF

28. Determine the capacitive reactance at 1 MHz for the following capacitors:

a) 22 pF

b) 560 pF

c) 33 nF

d) 4.7 μF

29. Determine the inductive reactance of a 100 mH inductor at the following frequencies:

a) 10 Hz

b) 500 Hz

c) 10 kHz

d) 400 kHz

e) 10 MHz

30. Determine the inductive reactance of a 100 mH inductor at the following frequencies:

a) 10 Hz

b) 500 Hz

c) 10 kHz

d) 400 kHz

e) 10 MHz

31. Determine the inductive reactance at 1 kHz for the following inductors:

a) 10 mH

b) 500 mH

c) 10 μH

d) 400 μH

32. Determine the inductive reactance at 500 kHz for the following inductors:

a) 1 mH

b) 40 mH

c) 2 μH

d) 50 μH

33. Draw phasor diagrams for the following:

a) $5 + j2$

b) $-10 - j20$

c) $8\angle 45^\circ$

d) $2\angle -35^\circ$

34. Draw phasor diagrams for the following:

a) $60j - 20$

b) $-40 + j500$

c) $0.05\angle -45^\circ$

d) $-15\angle 60^\circ$

35. The fundamental of a certain square wave is a 5 volt peak, 1 kHz sine. Determine the amplitude and frequency of each of the next five harmonics.

36. The fundamental of a certain triangle wave is a 10 volt peak, 100 Hz sine. Determine the amplitude and frequency of each of the next five harmonics.

Design

37. Determine the capacitance required for the following reactance values at 1 kHz:

a) 560 Ω

b) 330 kΩ

c) 470 kΩ

d) 1.2 kΩ

e) 750 Ω

38. Determine the capacitance required for the following reactance values at 20 Hz:

a) 56 kΩ

b) 330 kΩ

c) 470 kΩ

d) 1.2 kΩ

e) 750 Ω

39. Determine the inductance required for the following reactance values at 100 MHz:

a) 560 Ω

b) 330 kΩ

c) 470 kΩ

d) 1.2 kΩ

e) 750 Ω

40. Determine the inductance required for the following reactance values at 25 kHz:

a) 56 Ω

b) 33 kΩ

c) 470 kΩ

d) 1.2 kΩ

e) 750 Ω

41. Which of the following have a reactance of less than 100 Ω for all frequencies below 1 kHz?

a) 2 mH

b) 99 mH

c) 470 pF

d) 10000 μF

42. Which of the following have a reactance of less than 8 Ω for all frequencies above 10 kHz?

a) 10 nH

b) 5 mH

c) 56 pF

d) 470 μF

43. Which of the following have a reactance of at least 1k Ω for all frequencies above 20 kHz?

a) 2 mH

b) 200 mH

c) 680 pF

d) 33 μF

44. Which of the following have a reactance of at least 75 Ω for all frequencies below 5 kHz?

a) 680 μH

b) 10 mH

c) 82 pF

d) 33 nF

Challenge

45. Determine the negative and positive peak voltages, RMS voltage, DC offset, frequency, period and phase shift for the following expression: $v(t) = -10\sin(2\pi 250t + 180^\circ)$

46. Determine the negative and positive peak voltages, DC offset, frequency, period and phase shift for the following expression: $v(t) = 1 - 100\sin 2\pi 50000t$

47. Assume you have a DC coupled oscilloscope set as follows: time base = 100 microseconds/division, vertical sensitivity = 1 volt/division. Sketch the display of this waveform: $v(t) = 2 + 3\sin 2\pi 2000t$

48. Assume you have a DC coupled oscilloscope set to the following: time base = 20 microseconds/division, vertical sensitivity = 200 millivolts/division. Sketch the display of this waveform: $v(t) = -0.2 + 0.4\sin 2\pi 10000t$

49. A 200 Ω resistor is in series with a 1 mH inductor. Determine the impedance of this combination at 200 Hz and at 20 kHz.

50. A 1 kΩ resistor is in series with an inductor. If the combined impedance at 10 kHz is $1.41k\angle 45^\circ$, determine the inductance in mH.

This page titled 9.8: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **1.8: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

10: Series RLC Circuits

Learning Objectives

After completing this chapter, you should be able to:

- Utilize Kirchhoff's voltage law, the voltage divider rule and Ohm's law to find node and component voltages in series RLC networks that utilize voltage sources or a single current source.
- Compute complex impedance and system current in series RLC circuits.
- Draw phasor diagrams for impedance and component voltages in series RLC circuits.

10.1: Introduction
10.2: The Series Connection
10.3: Series Impedance
10.4: Series Circuit Analysis
10.5: Concerning Practical Inductors
10.6: Summary
10.7: Exercises

This page titled 10: Series RLC Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

10.1: Introduction

In this chapter we introduce series RLC circuits for the AC case. There is much here that will be familiar from your prior studies with DC series circuits, however, there will be a few notable changes and perhaps a surprise or two lurking. The key to most of this is to remember that all computations involve vector quantities. In fact, DC can be thought of as a special case of AC; namely, the frequency drops to zero hertz causing X_C to approach an open and X_L to drop to zero. This leaves us with just resistors and scalar quantities because the phase angles in the remainder of the circuit collapse to zero for DC steady state.

Many of the solution techniques from DC analysis will be applicable here. This includes the use of Ohm's law and Kirchhoff's voltage law, along with the voltage divider rule. Generally, reactance values will need to be computed from capacitor and inductor values before the main analysis may begin. In this chapter, as in the most of the remaining chapters, we shall be concerned with determining the circuit response based on a source with a single frequency of excitation, in other words, a simple sine wave.

To clarify our analyses, we shall make considerable use of both time domain voltage plots as well as phasor diagrams.

This page titled 10.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

10.2: The Series Connection

A series connection is always characterized by a single loop or path for current flow. There are no junctions from which current can flow out of or into. Consequently,

The current is the same everywhere in a series connection.

Each component in such a loop will see the same current, regardless of whether it is a resistor, capacitor or inductor. Before component voltages can be determined, the capacitive and inductive reactance values must be computed from the capacitor and inductor values, based on the frequency of the driving source. Consequently, if this frequency were to change, the reactances would change, and this would in turn cause changes in the circulating current and component voltages.

This page titled 10.2: The Series Connection is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.2: The Series Connection** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

10.3: Series Impedance

Perhaps the first practical issue we face is determining the effective impedance of an RLC series loop. For starters, resistors in series simply add. Reactances also add but we must be careful of the sign. Inductive reactance and capacitive reactance will partially cancel each other. Thus, the impedance in rectangular form is the sum of the resistive components for the real portion, plus the sum of the reactances for the imaginary (j) portion. We will often find it convenient to express this value in polar form.

Example 10.3.1

What is the impedance of the network shown in Figure 10.3.1 at a frequency of 15 kHz?

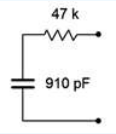


Figure 10.3.1: Circuit for Example 10.3.1.

First we need to find the capacitive reactance value.

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 15kHz910pF}$$

$$X_C = -j11.66k\Omega$$

As there is only one resistor and one capacitor, the result in rectangular form is $47k - j11.66k\Omega$. In polar form this is:

$$\text{Magnitude } = \sqrt{\text{Real}^2 + \text{Imaginary}^2}$$

$$\text{Magnitude } = \sqrt{47k^2 + (-11.66k)^2}$$

$$\text{Magnitude } = 48.42k$$

$$\theta = \tan^{-1}\left(\frac{\text{Imaginary}}{\text{Real}}\right)$$

$$\theta = \ tan^{-1}\left(\frac{-11.66k}{47k}\right)$$

$$\theta = -13.9^\circ$$

That is, in polar form $Z = 48.42E3\angle -13.9^\circ\Omega$.

Example 10.3.2

Determine the effective impedance of the circuit shown in Figure 10.3.2 if the source frequency is 2 kHz. Repeat this for source frequencies of 200 Hz and 20 kHz. Finally, express the results in both rectangular and polar form.

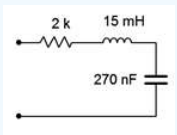


Figure 10.3.2: Circuit for Example 10.3.2.

The first step is to find the reactance values at 2 kHz.

$$X_L = j2\pi fL$$

$$X_L = j2\pi 2000Hz15mH$$

$$X_L = j188.5\Omega$$

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 2000Hz270nF}$$

$$X_C = -j294.7\Omega$$

Combine reals with reals and j terms with j terms.

$$Z = R + jX_L - jX_C$$

$$Z = 500 + j188.5 - j294.7\Omega$$

$$Z \approx 500 - j106.2\Omega = 511.2\angle -12^\circ\Omega$$

At 200 Hz X_C will ten times larger and X_L will be ten times smaller.

$$X_C = -j2947\Omega$$

$$X_L = j18.85\Omega$$

$$Z = 500 + j18.85 - j2947\Omega$$

$$Z \approx 500 - j2928\Omega = 2970\angle -80.3^\circ\Omega$$

At 20 kHz X_C will ten times smaller and X_L will be ten times larger.

$$X_C = -j29.47\Omega$$

$$X_L = j1885\Omega$$

$$Z = 500 + j1885 - j29.47\Omega$$

$$Z \approx 500 + j1856\Omega = 1922\angle 74.9^\circ\Omega$$

To help visualize this complex impedance, it is useful to construct a phasor plot as shown in Figure 10.3.3. We will do this for the initial case of 2 kHz. The resistive component is the horizontal vector of length 500 (yellow). X_L is straight up (blue) at $188.5\angle 90^\circ$, and X_C is straight down (red) at $294.7\angle -90^\circ$. By copying the X_L vector and then shifting it down and next to X_C, the difference between the two reactive components can be seen (purple component directly above the X_L copy). This Figure 10.3.2 Circuit for Example 10.3.2. reactive sum is then copied and shifted right to join the resistive component to show the final result. This is $511.2\angle -12^\circ\Omega$ (green), as expected.

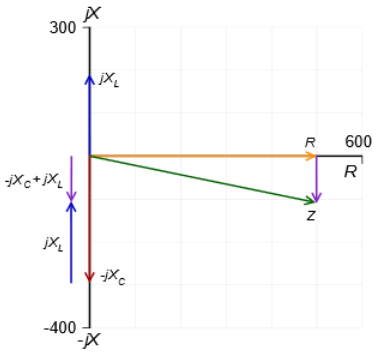


Figure 10.3.3: Construction of the impedance plot for the network of Figure 10.3.2.

Example 10.3.3

Determine the impedance of the network shown in Figure 10.3.4. If the input frequency is 1 kHz, determine the capacitor and inductor values.

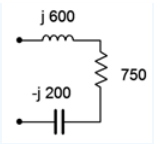


Figure 10.3.4: Circuit for Example 10.3.3.

The reactance values are already given, so we simply add them to determine the impedance in rectangular form. Combine reals with reals and j terms with j terms, and then convert to polar form.

$$Z = R + jX_L - jX_C$$

$$Z = 750 + j600 - j200\Omega$$

$$Z = 750 + j400\Omega = 850\angle 28.1^\circ\Omega$$

To find the capacitance and inductance, we simply rearrange the reactance formulas and solve. First, the inductor:

$$X_L = j2\pi fL$$

$$L = \frac{|X_L|}{2\pi f}$$

$$L = \frac{600\Omega}{2\pi 1kHz}$$

$$L \approx 95.5mH$$

And now for the capacitor:

$$X_C = -j\frac{1}{2\pi fC}$$

$$C = \frac{1}{2\pi f|X_C|}$$

$$C = \frac{1}{2\pi 1kHz200\Omega}$$

$$C = 796nF$$

A plot of the impedance vector summation is shown in Figure 10.3.5. Note how the three components combine graphically to arrive at Z.

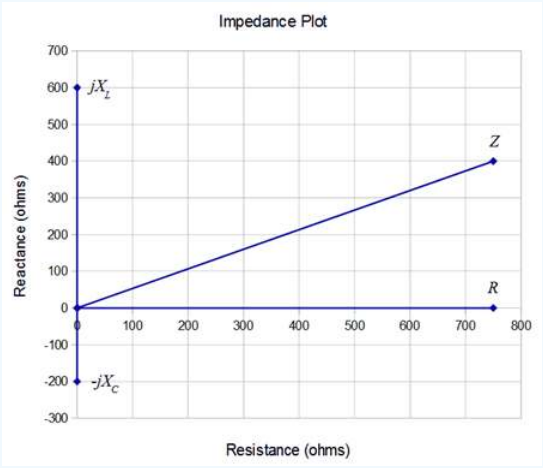


Figure 10.3.5: Impedance plot for the network of Example 10.3.3.

This page titled 10.3: Series Impedance is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.3: Series Impedance** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

10.4: Series Circuit Analysis

The techniques employed for series AC circuit analysis are the same as those used for DC. The key item to remember for series circuits, whether AC or DC, as that the current will be the same everywhere in a series connection. The major analysis tools are Ohm's law, Kirchhoff's voltage law (KVL), and optionally, the voltage divider rule.

KVL states that the sum of voltages around a loop must be zero, or that the sum of voltage rises around a loop must equal the sum of voltage drops:

$$\sum v \uparrow = \sum v \downarrow \tag{10.4.1}$$

The voltage divider rule for AC states that the voltage across any component or group of components is proportional to the ratio of the impedance of said component and the total series impedance. For some component A driven by source e,

$$v_A = e\frac{Z_A}{Z_{Total}} \tag{10.4.2}$$

If current is the desired quantity, the equivalent impedance of the network is found and then divided into the equivalent source voltage. If the voltages across specific components are desired, they can be found using the circulating current and Ohm's law. Alternately, they can be computed using the voltage divider rule. If a current source is driving the series circuit then the circulating current is the source current. Individual component voltages may be found using the source current and Ohm's law. The source voltage may be found by summing the component voltages via KVL, or by determining the equivalent series impedance and multiplying it by the source current.

Example 10.4.1

For the circuit of Figure 10.4.1, find the circulating current and the voltages across the capacitor and resistor.

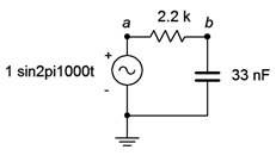


Figure 10.4.1: Circuit for Example 10.4.1.

The first step is to determine X_C and then find Z_{total}. Then use Ohm's law to find the circulating current.

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 1000Hz33nF}$$

$$X_C = -j4823\Omega$$

$$Z_{Total} = R + (-jX_C)$$

$$Z_{Total} = 2200\Omega - j4823\Omega$$

$$Z_{Total} = 5301\angle -65.5^\circ\Omega$$

$$i = \frac{v}{Z}$$

$$i = \frac{1V}{5301\angle -65.5^\circ\Omega}$$

$$i = 188.6\angle 65.5^\circ\mu A$$

Now use Ohm's law to find the component voltages.

$$v_R = i \times R$$

$$v_R = 188.6\angle 65.5^\circ\mu A \times 2200\angle 0^\circ\Omega$$

$$v_R = 0.4149\angle 65.5^\circ V$$

$$v_C = i \times X_C$$

$$v_C = 188.6\angle 65.5^\circ \mu A \times 4823\angle -90^\circ \Omega$$

$$v_C = 0.9098\angle -24.5^\circ V$$

To complete this Example, we shall generate a series of plots: First off, an impedance plot showing the vector combination of R and X_C leading to Z_{Total}. This is shown in Figure 10.4.2.

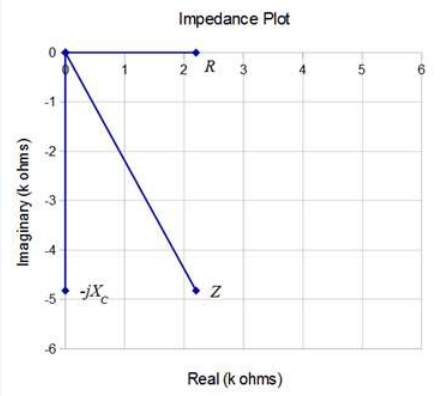


Figure 10.4.2: Impedance plot for Example 10.4.1.

Next, Figure 10.4.3 illustrates the phasor diagram of the three voltages. It may not be immediately apparent but an examination of this plot shows that it is just a counterclockwise rotation of the impedance plot. In fact, it is rotated by 65.5 degrees, the phase angle of the current. This should come as no surprise as the current is multiplied by each resistance, reactance or impedance to develop the voltage, and a vector multiplication simply adds the angles.

Also, note that if we simply summed the magnitudes of the component voltages, the result would be considerably larger than the input voltage, seeming to violate KVL. In contrast, once a proper vector summation is performed, all is right in paradise.

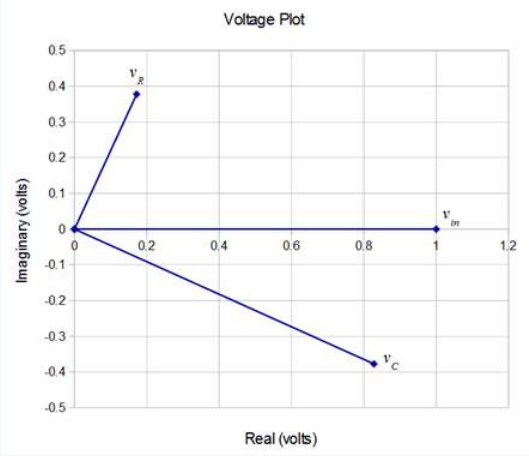


Figure 10.4.3: Voltage phasor diagram for Example 10.4.1.

For further clarity, Figure 10.4.4 plots the voltages in the time domain. Note how the component voltages add graphically resulting in the input voltage.

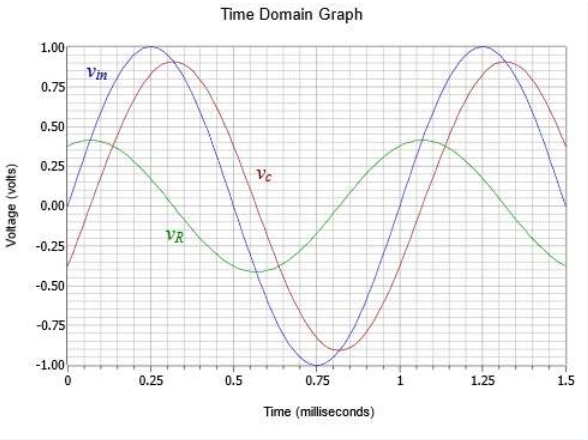


Figure 10.4.4: Time domain response for Example 10.4.1.

The time shifts may be computed for verification as follows:

$$\theta = 360^\circ \frac{\Delta t}{T}$$

$$\Delta t_R = T \frac{\theta_R}{360^\circ}$$

$$\Delta t_R = 1ms \frac{65.5^\circ}{360^\circ}$$

$$\Delta t_R = 181.9\mu s$$

$$\Delta t_C = T \frac{\theta_C}{360^\circ}$$

$$\Delta t_C = 1ms \frac{-24.5^\circ}{360^\circ}$$

$$\Delta t_C = -68\mu s$$

Thus we verify the resistor voltage leading (to the left) by 181.9 μs and the capacitor voltage lagging (to the right) by 68 μs, as seen in Figure 10.4.4.

Computer Simulation

The circuit of Example 10.4.1 is captured into a simulator as shown in Figure 10.4.5. Node voltage 1 corresponds to the input voltage and node 2 corresponds to the capacitor voltage. The resistor voltage is merely the difference between these two, or node 1 minus node 2.

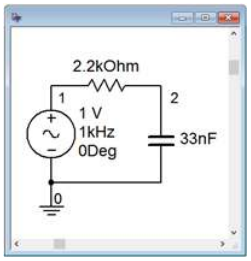


Figure 10.4.5: The circuit of Example 10.4.1 in a simulator.

A transient or time domain simulation is performed. The results of the simulation are shown in Figure 10.4.6. Note the tight agreement with the computed results as plotted in Figure 10.4.4.

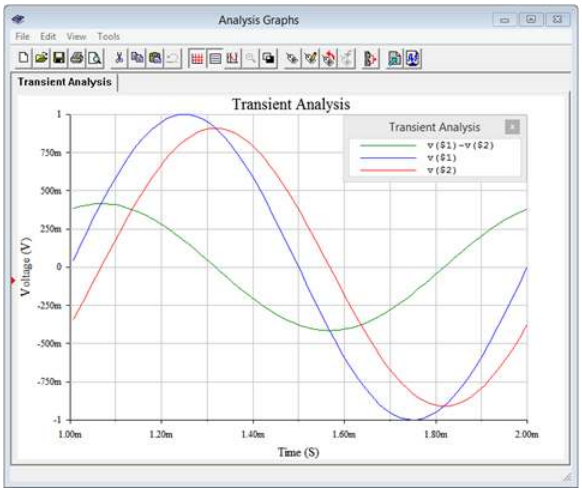


Figure 10.4.6: Time domain response of Example 10.4.1 from a simulator.

We will now examine a variety of different series circuits. To ease the computations, we will simply state values for the component reactances rather than specify a frequency along with inductor and/or capacitor values.

Example 10.4.2

Determine the component voltages for the circuit shown in Figure 10.4.7.

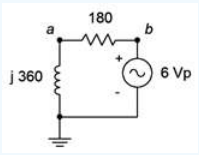


Figure 10.4.7: Circuit for Example 10.4.2.

Given the reference direction of the source (which produces a counterclockwise reference current), the voltage across the resistor will be defined as $v_b - v_a$. The first step is to find the equivalent series impedance. By inspection, this is $180 + j360\Omega$, which is equivalent to $402.5\angle 63.4^\circ\Omega$.

The voltages can be found via the voltage divider rule.

$$v_L = e\frac{X_L}{Z}$$

$$v_L = 6\angle 0^\circ V \frac{360\angle 90^\circ\Omega}{402.5\angle 63.4^\circ\Omega}$$

$$v_L = 5.366\angle 26.6^\circ Vp$$

$$v_R = e\frac{R}{Z}$$

$$v_R = 6\angle 0^\circ V \frac{180\angle 0^\circ\Omega}{402.5\angle 63.4^\circ\Omega}$$

$$v_R = 2.683\angle -63.4^\circ Vp$$

The rectangular versions of v_L and v_R are $4.8 + j2.4$ and $1.2 - j2.4$ respectively, summing up to the source of $6\angle 0^\circ$. This is illustrated in the phasor voltage plot of Figure 10.4.8.

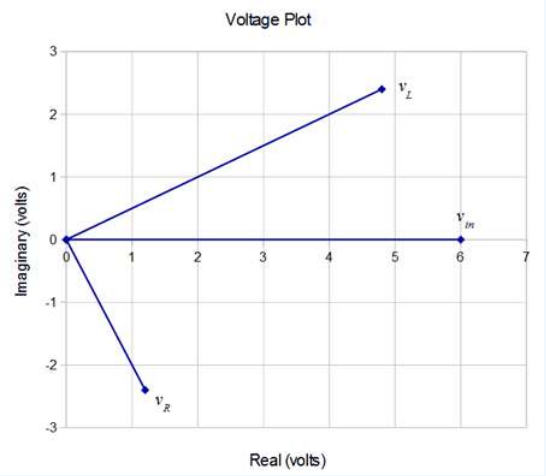


Figure 10.4.8: Voltage phasor diagram for Example 10.4.2.

Compare the plot of Figure 10.4.8 to that of the RC circuit in Figure 10.4.3. In this circuit, the impedance is inductive. This causes the current to lag the voltage, the opposite of the case for the RC circuit. Thus, the resistor voltage winds up below the real axis instead of above it (remember, the current and voltage for a resistor are in phase, and therefore the voltage across the resistor will have the same phase angle as the current through it). Once again, simply summing the magnitudes of the voltages yields a greater value than the source, however, a proper vector addition shows that KVL is satisfied.

A time domain plot of the source and component voltages is shown in Figure 10.4.9. Note the similarity with the plot of Figure 10.4.4. Further, note how the time positions of the component voltages have swapped.

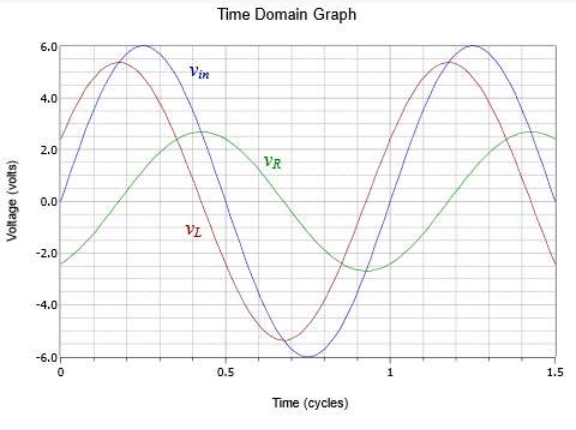


Figure 10.4.9: Time domain response for Example 10.4.2.

Finally, as a source frequency was not specified for this circuit, the time scale of Figure 10.4.9 is calibrated in terms of cycles rather than seconds.

Example 10.4.3

Find the voltages v_a and v_b in the circuit of Figure 10.4.10. Assume the current source is the reference of 0°.

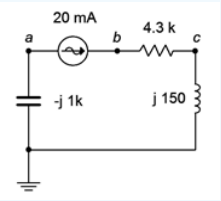


Figure 10.4.10: Circuit for Example 10.4.3.

Ohm's law may be used directly as the current is set by the source. Given the reference direction of the 20 mA source, the reference polarity for the capacitor voltage will be + to − from bottom to top. Thus, v_a is negative.

$$v_a = v_C = -i \times X_C$$

$$v_a = -20E-3\angle 0^\circ A \times 1000\angle -90^\circ \Omega$$

$$v_a = -20\angle -90^\circ V$$

Recalling that a negative magnitude is the same as a 180° phase shift, we may remove the magnitude's negative sign by adding 180° to the phase angle. Therefore, v_a may also be written as $20\angle 90^\circ$.

The node b voltage is found by multiplying the current by the impedance seen from node b to ground.

$$v_b = i \times (R + jX_L)$$

$$v_b = 20\angle 0^\circ mA \times (4300 + j150\Omega)$$

$$v_b = 86.05\angle 2^\circ V$$

Notice the small size of the phase angle. This is expected given that the inductive reactance is so much smaller than the resistive component. Finally, these voltage are RMS as the current is assumed to be RMS (not having been labeled as peak or peak-to-peak).

Example 10.4.4

Find the voltage v_{bd} in the circuit of Figure 10.4.11. $E_1 = 2\angle 0^\circ$ volts peak and $E_2 = 5\angle 60^\circ$ volts peak.

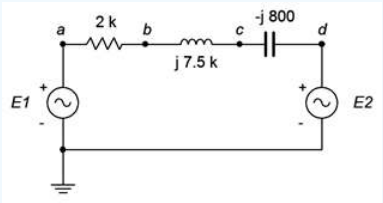


Figure 10.4.11: Circuit for Example 10.4.3.

The first step is to find the equivalent source voltage. Voltage sources in series add, however, note the reference polarity for E_2. If we take E_1 as the system reference, then the combined source is $E_1 - E_2$. Using E_1 as the reference source creates a clockwise reference current and thus a positive reference polarity for v_{bd}. If E_2 was taken as the reference then v_{bd} would be assumed to be negative due to the assumed counterclockwise reference direction of the resulting current. Either way will work.

$$E_{Total} = E_1 - E_2$$

$$E_{Total} = 2\angle 0^\circ Vp - 5\angle 60^\circ Vp$$

$$E_{Total} = 4.359\angle -96.6^\circ Vp$$

We can now use a voltage divider between the impedance seen from node b to node d versus the total series impedance. The impedance of the circuit is $2k + j7.5k - j800\Omega$, or $2k + j6.7k\Omega$. This is equivalent to $6992\angle 73.4^\circ\Omega$. The impedance of between nodes b and d is $j7.5k - j800\Omega$, or $j6.7k\Omega$. In polar form this is $6700\angle 90^\circ\Omega$.

The voltage divider yields:

$$v_{bd} = e\frac{Z_{bd}}{Z_{Total}}$$

$$v_{bd} = 4.359\angle -96.6^\circ V \frac{6700\angle 90^\circ\Omega}{6992\angle 73.4^\circ\Omega}$$

$$v_{bd} = 4.177\angle -80^\circ Vp$$

Although we have answered the question, at this point attempting to visualize the waveforms in your head to verify the results may be a little challenging. No problem! If we also find the voltage across the resistor, we can check to see if KVL is verified

using a phasor diagram. First, the resistor voltage may be found using the voltage divider rule.

$$v_R = e\frac{R}{Z_{Total}}$$

$$v_R = 4.359\angle -96.6^\circ V \frac{2000\angle 0^\circ \Omega}{6992\angle 73.4^\circ \Omega}$$

$$v_R = 1.247\angle -170^\circ Vp$$

We now plot all four voltages as seen in Figure 10.4.12 (note the unequal horizontal versus vertical scaling for ease of visual analysis).

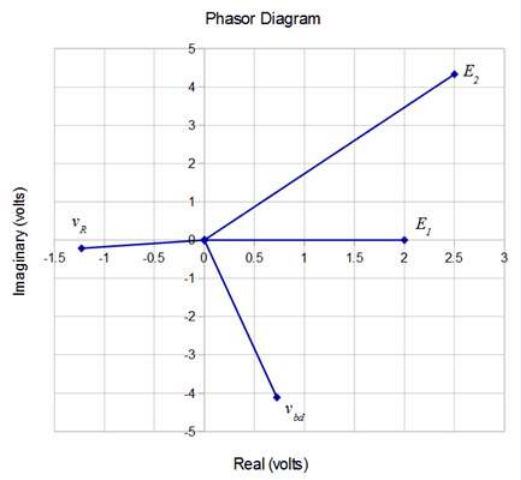


Figure 10.4.12: Voltage phasor diagram for Example 10.4.3.

KVL tells us that the sum of the voltage rises must equal the sum of the voltage drops. We had used E_1 as the reference and treated it as a rise. This made the reference direction clockwise for the circulating current. As a consequence, v_R and v_{bd} are voltage drops. E_2 is also seen as a drop given its reference polarity markings. In other words, E_1 should equal the sum of v_R, v_{bd} and E_2. Let's see if this is borne out graphically in the phasor diagram.

Looking at imaginary parts first, E_1 is strictly real so the other three vectors should sum to zero for their imaginary parts. The imaginary part of E_2 is about +4.3 while the other two come in around −4.1 and −0.2, so they do balance. For the real parts, E_2 is +2.5, v_{bd} is about +0.7 and v_R is about −1.2. These add up to +2, the real component of E_1. Everything balances. For more exacting results, you could convert each of the polar form results into rectangular form and add the reals to the reals, and the imaginaries to the imaginaries, and get the same result. In fact, using the prior recorded values, the three drops sum to $1.997\angle 0.0013^\circ$ volts versus precisely $2\angle 0^\circ$ volts. This small deviation is due to the accumulated rounding and truncation errors of the intermediate results.

Sometimes a problem concerns determining resistance or reactance values. The solution paths will require use of the analysis rules in reverse. For example, if a resistor value is needed to set a specific current, the total required impedance can be determined from this current and the given voltage supply. The values of the other series components can then be subtracted from the total (using rectangular form), yielding the required resistor or reactance value.

Another possibility is determining a specific capacitance or inductance to meet certain requirements. An example of this would be a simple crossover network for a loudspeaker system. Generally, a single transducer is incapable of reproducing music signals of all frequencies at sufficiently loud levels while maintaining low distortion. Consequently, the frequency range is split into two or more bands with each being covered by a transducer optimized to reproduce those frequencies. These transducers, along with a few electrical components, are then packaged into an appropriate enclosure commonly referred to as a loudspeaker[1]. In a basic system, the audible spectrum is split into two parts, as seen in Figure 10.4.13. The low frequencies are handled by a large transducer called a woofer, while the high frequencies are handled by a small, light transducer called a tweeter that can follow the rapidly changing high frequency content with accuracy. Woofers are not capable of reproducing high frequencies, and excessive low frequency content can physically damage a tweeter. Thus, some means of “steering” the high frequency content to the tweeter and the low

frequency content to the woofer must be employed. One way to do this is to place an inductor or capacitor in series with the transducer.



Figure 10.4.13: A two-way bookshelf loudspeaker system with tweeter (top) and woofer (bottom).

Figure 10.4.14 shows a simple way to block low frequencies from a tweeter. In this diagram, the loudspeaker is modeled as a resistor. Although this is not perfectly accurate, it is sufficient to illustrate the idea.

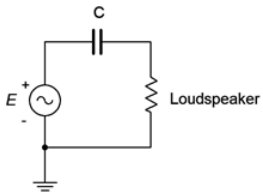


Figure 10.4.14: A basic loudspeaker crossover network.

To understand this concept, simply think of the circuit as a voltage divider between the loudspeaker and the reactance of the capacitor. Capacitive reactance is inversely proportional to frequency. The circuit is designed such that X_C is much larger than than the impedance of the loudspeaker at very low frequencies. This means that very little of a low frequency signal will reach the loudspeaker as most of that voltage will drop across the capacitor. In contrast, at high frequencies X_C is much smaller than the tweeter's impedance and thus virtually all of the source signal reaches the loudspeaker. The transition point between these two regions is called the crossover frequency and is usually defined as the point where X_C equals the magnitude of the loudspeaker impedance. For a woofer, the capacitor would be replaced by an inductor. At low frequencies the inductive reactance would be minimal, thereby allowing all of the low frequency content to reach the woofer. The opposite happens at high frequencies. The inductive reactance would rise with frequency and eventually prevent or “choke off” the signal to the woofer (hence, the slang term “choke” for an inductor).

Example 10.4.5

Referring to the circuit of Figure 10.4.14, assume the loudspeaker impedance is $8\angle 0^\circ\Omega$. Determine a capacitor value for a crossover frequency of 2.5 kHz. At this frequency the magnitude of X_C should be the same as that of the loudspeaker. For a 10 volt RMS input signal, determine the loudspeaker voltage at frequencies of 100 Hz, 2.5 kHz and 15 kHz.

The first step is to determine the value of the capacitor such that at 2.5 kHz X_C equals 8Ω. To do this, simply rearrange the capacitive reactance formula:

$$X_C = -j\frac{1}{2\pi fC}$$

$$C = 12\pi f|X_C|$$

$$C = \frac{1}{2\pi 2500Hz8\Omega}$$

$$C = 7.96\mu F$$

To find the loudspeaker voltage at the three specified frequencies, first find the reactance at that frequency and then use the voltage divider rule. We already know that at 2.5 kHz the reactance is $-j8\Omega$.

$$v_{loudspeaker} = e\frac{Z_{Load}}{Z_{Total}}$$

$$v_{loudspeaker} = 10\angle 0^{\circ} Vrms \frac{8\Omega}{8 - j8\Omega}$$

$$v_{loudspeaker} = 7.07\angle 45^{\circ} Vrms$$

At 100 Hz, the frequency has decreased by a factor of 25 so the reactance goes up by the same factor yielding $-j200\Omega$.

$$v_{loudspeaker} = e\frac{Z_{Load}}{Z_{Total}}$$

$$v_{loudspeaker} = 10\angle 0^{\circ} Vrms \frac{8\Omega}{8 - j200\Omega}$$

$$v_{loudspeaker} = 0.4\angle 87.7^{\circ} Vrms$$

At 15 kHz, the frequency has increased by a factor of 6 so the reactance goes down by the same factor yielding $-j1.333\Omega$.

$$v_{loudspeaker} = e\frac{Z_{Load}}{Z_{Total}}$$

$$v_{loudspeaker} = 10\angle 0^{\circ} Vrms \frac{8\Omega}{8 - j1.333\Omega}$$

$$v_{loudspeaker} = 9.86\angle 9.5^{\circ} Vrms$$

From this tally, it should be obvious that the lowest bass frequencies will experience attenuation while the highest frequencies will see little reduction in amplitude. This is just what we want.

The concept of a variation in output voltage with respect to frequency, or frequency response, will be examined in much greater detail in upcoming chapters.

References

[1]To add some confusion, the term “loudspeaker” can refer to either the individual transducers (engineer-speak) or the entire finished system (audiophile-speak).

This page titled 10.4: Series Circuit Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.4: Series Circuit Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

10.5: Concerning Practical Inductors

Up to this point, inductors have been treated as ideal components, that is, pure inductance. In reality, all inductors have some resistance associated with them due to the resistance of the wire used to make the coil. This is called ESR, or Equivalent Series Resistance. It is also denoted as R_{coil}. Ideally, this resistance will be small enough to ignore, but ultimately it will place a limit on the performance of any circuit that utilizes an inductor.

While it is possible to measure the resistance of an inductor using a DMM, this will not yield an accurate value at all frequencies. In fact, as frequency increases, ESR will also increase. This is due to skin effect. At higher frequencies, current is not distributed equally throughout the cross-section of a conductor. In fact, it tends to "hug" the outer surface or "skin" of the conductor. This reduces the effective crosssectional area and thereby increases the resistance[1]. In general, as frequency rises, so does R_{coil}. Unfortunately, the situation is further complicated by distributed capacitance that will become an issue at still higher frequencies. As a consequence, manufacturers will give a "Q plot", such as the one shown in Figure 10.5.1.

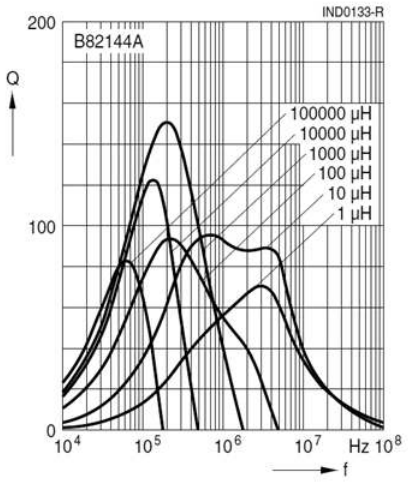


Figure 10.5.1: Inductor Q graph. Courtesy of TDK

The Q, or quality factor, of an inductor can be defined in terms of the peak energy stored in the device versus the energy dissipated per cycle. Keeping time constant, we can relate this to the power via the relation i^2R. The current necessarily is the same for both the reactive and resistive components, therefore the Q of the coil is equal to the ratio of its reactance to its resistance at the frequency of interest.

$$Q_{coil} = \frac{X_L}{R_{coil}} \tag{10.5.1}$$

Where

Q_{coil} is the quality factor of the inductor,

X_L is the magnitude of the inductive reactance at the frequency of interest,

R_{coil} is the resistance of the inductor at the frequency of interest.

In general, the higher the Q_{coil}, the better. As can be seen in Figure 10.5.1, Q_{coil} is not a constant. Indeed, it increases with frequency until a peak is reached, at which point it begins to fall.

When dealing with practical inductors, the effective Q_{coil} can be determined from a graph if the operating frequency is known. Once the Q_{coil} is found, the effective value of Q_{coil} can be found from Equation 10.5.1. This value can then be placed in series with the ideal inductor to create a more accurate result. For analysis, this pair is sometimes drawn with a box around it to denote that Q_{coil} is not a separate physical resistor, but is the effective resistance of the inductor.

Example 10.5.1

Find the voltage across the inductor in the circuit of Figure 10.5.2. Assume the source voltage is $20\angle 0^\circ$ peak-to-peak at a frequency of 20 kHz. L is equal to 10 mH, Q_{coil} is 50, and R_1 is 600Ω.

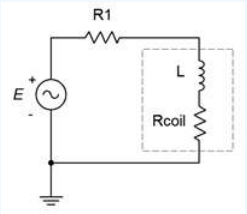


Figure 10.5.2: Circuit for Example 10.5.1.

Remember, the inductor consists of both elements within the dashed box. First, find the magnitude of the inductive reactance.

$$|X_L| = 2\pi fL$$

$$|X_L| = 2\pi 20kHz10mH$$

$$|X_L| = 1257\Omega$$

We can now find R_{coil} via Equation 10.5.1:

$$Q_{coil} = \frac{X_L}{R_{coil}}$$

Therefore,

$$R_{coil} = \frac{X_L}{Q_{coil}}$$

$$R_{coil} = \frac{1257}{50}$$

$$R_{coil} = 25.1\Omega$$

The impedance of the inductor is $25.1 + j1257\Omega$, or $1257\angle 88.9^\circ\Omega$. The total impedance is $600 + 25.1 + j1257\Omega$, or $1404\angle 63.6^\circ\Omega$. We can use a voltage divider to find the inductor's voltage.

$$v_{ind} = e\frac{Z_{ind}}{Z_{Total}}$$

$$v_{ind} = 20\angle 0^\circ Vpp\frac{1257\angle 88.9^\circ\Omega}{1404\angle 63.6^\circ\Omega}$$

$$v_{ind} = 17.9\angle 25.3^\circ Vpp$$

It is worth noting that, due to the coil resistance, the inductor's current and voltage are not precisely 90° out of phase, but rather 88.9°. That's a small but measurable deviation.

References

[1]Recall that $R = \rho l/A$. A decrease in area A results in increased resistance, R.

This page titled 10.5: Concerning Practical Inductors is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.5: Concerning Practical Inductors** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

10.6: Summary

A series connection is any connection in which the current through one component must be identical to the current flowing through any other component in that connection. This remains true for AC circuits employing resistors, capacitors and inductors. The equivalent impedance of a set of resistors, inductors and capacitors placed in series is the vector sum of their resistance and reactance values. Inductive and capacitive reactance have opposing signs and will partially cancel each other. While resistance remains constant across frequency, reactance does not. Capacitive reactance decreases with frequency while inductive reactance increases. Therefore, the precise impedance value will vary with frequency. If the impedance angle is positive, the circuit is said to be inductive. If the impedance angle is negative, the circuit is said to be capacitive.

Once inductance and capacitance values have been turned into their corresponding reactances at the frequency of interest, Ohm's law can be used to find the voltage across any component. The voltage will equal the product of the resistance or reactance and the current flowing through it. This is a vector computation. The current through a resistor will be in phase with the voltage across it. In contrast, the inductor voltage will lead the current and the resistor voltage by 90 degrees. Also, the capacitor voltage will lag the current and the resistor voltage by 90 degrees. From these observations it is apparent that the inductor and capacitor voltages in a series connection must be 180 degrees out of phase.

Kirchhoff's voltage law (KVL) states that the sum of voltage rises around any series loop must equal the sum of voltage drops around that loop. This remains true for AC analysis, however, it is imperative that a vector summation is performed and not just a simple summation of the magnitudes. If the phase angles are ignored, it is possible for the voltage magnitudes to sum to considerably more than the source voltage. The voltage divider rule (VDR) remains true for AC analysis, but again, it is a vector computation. Thus, a rendering using just magnitudes will not produce a correct result. For example, a divider between a resistor and inductive reactance of equal value will not yield the source voltage splitting in half across each component.

For highest accuracy, the coil resistance, or R_{coil}, of inductors should not be ignored. It is treated as a small resistance in series with and integral to the inductor. Coil resistance is computed from the inductor's Q, or quality factor. Q is defined as the ratio of X_L to R_{coil}.

Review Questions

1. How is the equivalent impedance for a string of series connected resistors, capacitors and inductors computed?
2. Will the impedance of an RLC circuit be the same at all frequencies? Why/why not?
3. How are series connected AC voltage sources combined?
4. Does Kirchhoff's voltage law change for AC circuit analysis?
5. Is it possible for the voltage magnitudes of an AC RLC circuit to sum to more than the source voltage? Why/why not?
6. How is a phasor impedance plot for a series network related to its phasor voltage plot?

This page titled 10.6: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.6: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

10.7: Exercises

Analysis

1. Determine the impedance of the circuit of Figure 10.7.1 for a 1 kHz sine.

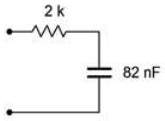


Figure 10.7.1

2. Determine the impedance of the circuit of Figure 10.7.1 for a 5 kHz sine.

3. Determine the impedance of the circuit of Figure 10.7.2 for a 10 kHz sine.

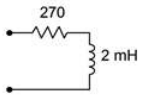


Figure 10.7.2

4. Determine the impedance of the circuit of Figure 10.7.2 for a 50 kHz sine.

5. Determine the impedance of the circuit of Figure 10.7.3 for a 1 kHz sine.

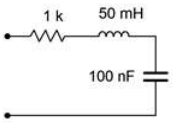


Figure 10.7.3

6. Determine the impedance of the circuit of Figure 10.7.3 for a 500 Hz sine.

7. Determine the impedance of the circuit of Figure 10.7.4.

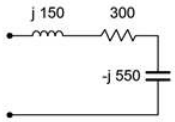


Figure 10.7.4

8. In the circuit of Figure 10.7.4, if the input frequency is 100 Hz, what is the value of the inductor, in mH?

9. In the circuit of Figure 10.7.4, if the input frequency is 200 Hz, what is the value of the capacitor, in μF?

10. Draw the voltage and current waveforms for the circuit of Figure 10.7.5.

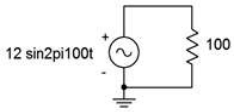


Figure 10.7.5

11. Draw the voltage and current waveforms for the circuit of Figure 10.7.6.

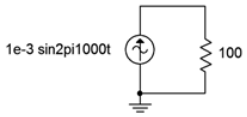


Figure 10.7.6

12. Draw the voltage and current waveforms for the circuit of Figure 10.7.7 if E is a one volt peak sine at a frequency of 10 kHz and C = 3.3 nF.

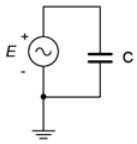


Figure 10.7.7

13. Draw the voltage and current waveforms for the circuit of Figure 10.7.8 if E is a two volt peak-peak sine at a frequency of 40 Hz and L = 33 mH.

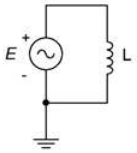


Figure 10.7.8

14. Draw the voltage and current waveforms for the circuit of Figure 10.7.9 if I is a 10 μA peak sine at a frequency of 2 kHz and C = 6.8 nF.

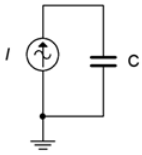


Figure 10.7.9

15. Draw the voltage and current waveforms for the circuit of Figure 10.7.10 if I is a two amp peak-peak sine at a frequency of 40 Hz and L = 33 mH.

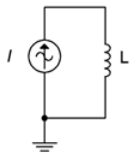


Figure 10.7.10

16. Determine the impedance of the circuit of Figure 10.7.11.

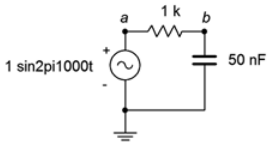


Figure 10.7.11

17. Determine the impedance of the circuit of Figure 10.7.11 using a frequency of 10 kHz.

18. For the circuit of Figure 10.7.11, determine the circulating current and the voltages across each component. Draw a phasor diagram of the three component voltages. Also find the time delay between the voltages of the components.

19. For the circuit of Figure 10.7.11 using a frequency of 10 kHz, determine the circulating current and the voltages across each component. Draw a phasor diagram of the three component voltages and determine the time delay between the capacitor and resistor voltages.

20. Determine the impedance of the circuit of Figure 10.7.12.

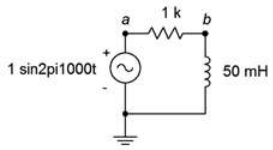


Figure 10.7.12

21. Determine the impedance of the circuit of Figure 10.7.12 using a frequency of 10 kHz.

22. For the circuit of Figure 10.7.12, determine the circulating current and the voltages across each component. Also find the time delay between the voltages of the components.

23. For the circuit of Figure 10.7.12 with a frequency of 3 kHz, determine the circulating current and the voltages across each component. Also find the time delay between the voltages of the components.

24. For the circuit of Figure 10.7.13, determine the circulating current.

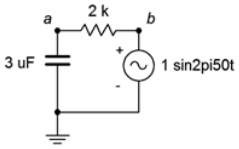


Figure 10.7.13

25. Determine the impedance of the circuit of Figure 10.7.13 using a frequency of 1.5 kHz.

26. For the circuit of Figure 10.7.13, determine the circulating current and the voltages across each component. Also find the time delay between the voltages of the components.

27. For the circuit of Figure 10.7.13 with a frequency of 1.5 kHz, determine the circulating current and the voltages across each component. Also find the time delay between the voltages of the components.

28. For the circuit of Figure 10.7.14, determine the circulating current and the voltages across each component. Draw a phasor diagram of the three component voltages and determine the time delay between the inductor and resistor voltages.

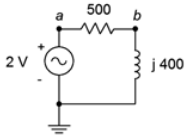


Figure 10.7.14

29. For the circuit of Figure 10.7.15, determine the circulating current and the voltages across each component.

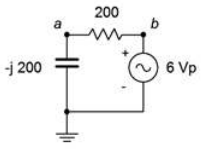


Figure 10.7.15

30. For the circuit of Figure 10.7.16, determine the circulating current and the voltages across each component.

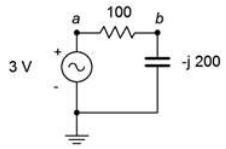


Figure 10.7.16

31. For the circuit of Figure 10.7.17, determine the applied voltage and the voltages across each component.

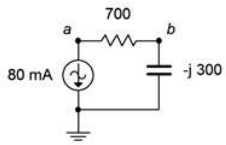


Figure 10.7.17

32. For the circuit of Figure 10.7.18, determine the applied voltage and the voltages across each component.

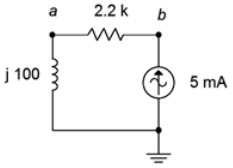


Figure 10.7.18

33. For the circuit of Figure 10.7.19, determine the circulating current and the voltages across each component.

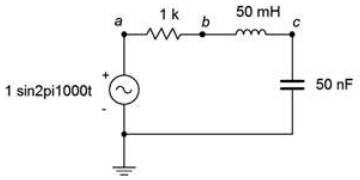


Figure 10.7.19

34. Repeat the previous problem using an input frequency of 10 kHz.

35. For the circuit of Figure 10.7.20, determine the circulating current and the voltages across each component. The source is a 10 volt peak sine at 20 kHz, $R = 200\Omega$, C = 100 nF and L = 1 mH.

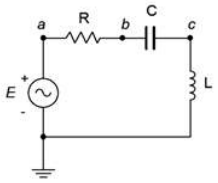


Figure 10.7.20

36. For the circuit of Figure 10.7.20, find v_b and v_{ac}.

37. For the circuit of Figure 10.7.21, find v_b and v_{ac}. The source is a 50 volt peakpeak sine at 10 kHz, $R = 100\Omega$, C = 200 nF and L = 1 mH.

https://eng.libretexts.org/@go/page/52959

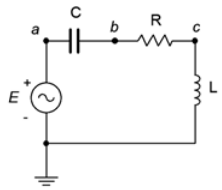


Figure 10.7.21

38. For the circuit of preceding problem, determine the circulating current and the voltages across each component.

39. For the circuit of Figure 10.7.22, determine the circulating current and the voltages across each component. E is a 1 volt peak 2 kHz sine. Also, draw a phasor diagram of the four component voltages.

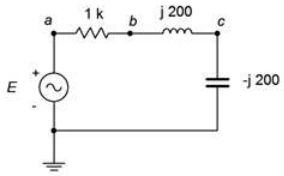


Figure 10.7.22

40. For the circuit of Figure 10.7.22, find v_b and v_{ca}. E is a 1 volt peak 2 kHz sine.

41. For the circuit of Figure 10.7.23, determine v_b, v_c and v_{ac}. E is a 10 volt peak 15 kHz sine.

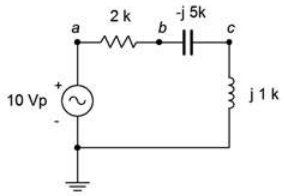


Figure 10.7.23

42. For the circuit of Figure 10.7.24, determine the circulating current and the voltages across each component. E is a 100 millivolt peak 250 Hz sine. Further, draw a phasor diagram of the four component voltages.

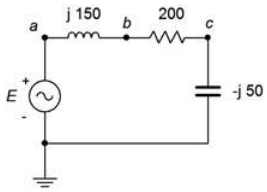


Figure 10.7.24

43. For the circuit of Figure 10.7.25, determine the circulating current and the voltages across each component. E is a 2 volt RMS 1 kHz sine. Also, draw a phasor diagram of the four component voltages.

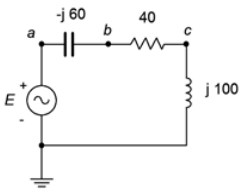


Figure 10.7.25

44. For the circuit of Figure 10.7.26, determine v_b, v_c and v_{ac}. E is a 1 volt peak 25 kHz sine.

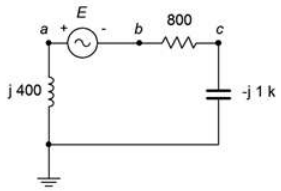


Figure 10.7.26

45. For the circuit of Figure 10.7.27, determine the voltages across each component. The source is a 50 mA peak sine at 15 kHz, R = 200Ω, C = 100 nF and L = 1.5 mH.

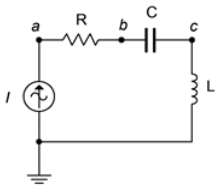


Figure 10.7.27

46. For the circuit of Figure 10.7.28, determine v_{ac}, v_b and v_c. The source is a 10 mA peak-peak sine at 50 kHz, R = 2kΩ, C = 10 nF and L = 800 μH.

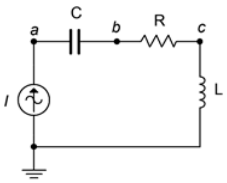


Figure 10.7.28

47. For the circuit of Figure 10.7.29, determine the voltages across each component. The source is a 2 mA RMS sine at 1 kHz, R = 1.2kΩ, C = 750 nF and L = 6.8 mH.

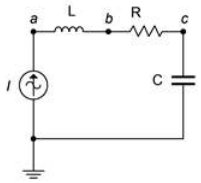


Figure 10.7.29

48. For the circuit of Figure 10.7.30, determine v_{ac}, v_b and v_a. The source is a 2 mA peak-peak sine at 300 kHz, R = 560Ω, C = 6.8 nF and L = 400 μH.

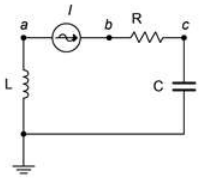


Figure 10.7.30

49. For the circuit of Figure 10.7.31, determine the voltages across each component.

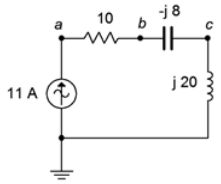


Figure 10.7.31

50. For the circuit of Figure 10.7.32, determine the voltages across each component. Further, draw a phasor diagram of the four component voltages.

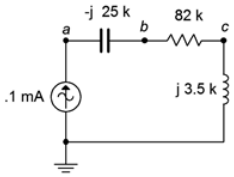


Figure 10.7.32

51. For the circuit of Figure 10.7.33, v_{ac}, v_b and v_c. The source is 5 mA peak at 8 kHz.

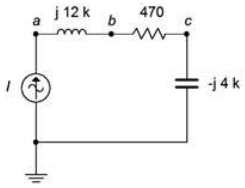


Figure 10.7.33

52. For the circuit of Figure 10.7.34, determine the voltages across each component. The source is 20 mA peak at 100 kHz.

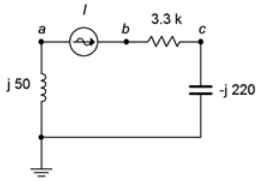


Figure 10.7.34

53. For the circuit of Figure 10.7.35, determine the voltages across each component.

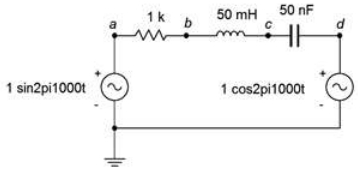


Figure 10.7.35

54. For the circuit of Figure 10.7.36, determine the voltages v_b and v_{db}. $E1 = 2\angle 0^\circ$ and $E2 = 5\angle 90^\circ$.

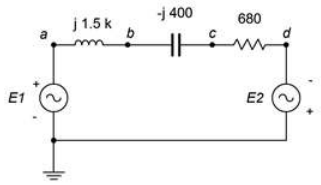


Figure 10.7.36

55. Determine the inductance and capacitance values for the circuit of problem 52.

56. For the circuit of Figure 10.7.36, determine the inductor and capacitor values if the source frequency is 12 kHz.

57. For the circuit of Figure 10.7.37, determine the voltages across each component. $E1 = 1\angle 0^\circ$ and $E2 = 8\angle 60^\circ$.

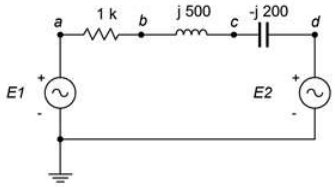


Figure 10.7.37

Design

58. Redesign the circuit of Figure 10.7.11 using a new capacitor such that the current magnitude from the source is 100 μA.

59. Redesign the circuit of Figure 10.7.12 using a new frequency such that the current magnitude from the source is 200 μA.

60. For the circuit of Figure 10.7.11, determine a new capacitor such that $|X_C| = R$.

61. For the circuit of Figure 10.7.12, determine a new frequency such that $|X_L| = R$.

Challenge

62. For the circuit of Figure 10.7.19, determine a new frequency such that $|X_C| = |X_L|$.

63. Determine the output voltage across the capacitor of Figure 10.7.11 at frequencies of 100 Hz, 5 kHz and 20 kHz. In light of this, if the input signal was a 1 kHz square wave instead of a sine wave as pictured, how would this circuit affect the shape of the output waveform (hint: consider superposition)?

64. Assume that you are troubleshooting a circuit like the one shown in Figure 10.7.20. E is a 2 volt peak sine at 2 kHz, $R = 390\Omega$, C = 100 nF and L = 25 mH. The circulating current measures approximately 4 mA with a lagging phase angle of just under −40 degrees. What is the likely problem?

65. Given the circuit shown in Figure 10.7.20, find the values for C and L if the source is a 6 volt sine wave at 1 kHz, $R = 2k\Omega$, v_R = 4 V and v_L = 5 V.

66. The circuit of Figure 10.7.38 can be used as part of a loudspeaker crossover network. The goal of this circuit is to steer the low frequency tones to the low frequency transducer (labeled here as “Loudspeaker” and often referred to as a woofer). A similar network substitutes a capacitor for the inductor to steer the high frequency tones to the high frequency transducer (AKA tweeter). These networks can be pictured as frequency sensitive voltage dividers. At very low frequencies, X_C is very large and blocks low frequency tones from reaching the tweeter. A mirror situation occurs with the inductor/woofer variant. The crossover frequency is the frequency at which the reactance magnitude equals the resistance. Assuming simple 8Ω resistances for the woofer and tweeter, determine capacitor and inductor values that would yield a 1.5 kHz crossover frequency. How might this concept be extended to a mid-range loudspeaker that only produces tones in the middle of the musical frequency spectrum? (Note, this concept will be revisited in the final simulation problem, below, and also in the Simulation portion of Chapter 4 which covers series-parallel circuits.)

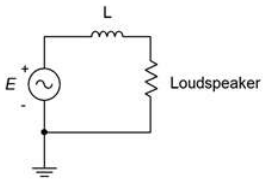


Figure 10.7.38

Simulation

67. Simulate the solution of design problem 58 and determine if the values produce the required results.

68. Simulate the solution of design problem 59 and determine if the values produce the required results.

69. Simulate the solution of design problem 60 and determine if the values produce the required results. Hint: if the reactance/resistance magnitudes are the same, then the voltage magnitudes will be identical.

70. Simulate the solution of design problem 61 and determine if the values produce the required results. Hint: if the reactance/resistance magnitudes are the same, then the voltage magnitudes will be identical.

71. Simulate the solution of challenge problem 62 and determine if the new frequency produces the required results. Hint: if the reactance magnitudes are the same, then the voltage magnitudes will be identical. Further, their phases will cause these voltages to cancel, leaving the resistor voltage equal to the source voltage.

72. Using a transient analysis, crosscheck the crossover design of the final challenge problem, above. Plot the resistor (loudspeaker) voltage across the range of 100 Hz to 20 kHz for both sections.

This page titled 10.7: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **2.7: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

11: Parallel RLC Circuits

Learning Objectives

After completing this chapter, you should be able to:

- Utilize Kirchhoff's current law, the current divider rule and Ohm's law to find branch currents in parallel RLC networks that utilize current sources or a single voltage source.
- Compute complex impedance and system voltage in parallel RLC circuits.
- Determine the equivalent value of multiple AC current sources in parallel.
- Draw phasor diagrams for impedance/susceptance and component currents in parallel RLC circuits.

11.1: Introduction
11.2: The Parallel Connection
11.3: Parallel Impedance
11.4: Parallel Circuit Analysis
11.5: Summary
11.6: Exercises

This page titled 11: Parallel RLC Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

11.1: Introduction

In this chapter we introduce the analysis of AC RLC circuits configured in parallel. AC parallel circuits echo the characteristic of their DC counterpart and many of the solution techniques from DC analysis will be applicable here. This includes the use of Ohm's law and Kirchhoff's current law, along with the current divider rule. Generally, as with the series circuits presented in the previous chapter, reactance values will need to be computed from capacitor and inductor values before the main analysis may begin. Here, as in the most of the remaining chapters, we shall be concerned with determining the circuit response based on a source with a single frequency of excitation, in other words, a simple sine wave.

Parallel circuits are in many ways the complement of series circuits. The most notable characteristic of a parallel circuit is that it has only two nodes and each component is connected from one node to the other. There are no other connections with which to create a voltage divider. Consequently, all components see the same voltage. Currents divide among the components in proportion to their conductance/susceptance (i.e., in inverse proportion to their resistance/reactance).

The key to this is to remember that all computations involve vector quantities. This can lead to some surprising results for the uninitiated. For example, due to the 180 degree phase differential between inductors and capacitors, it is possible for an individual branch current to be greater than the source current. This does not violate Kirchhoff's current law, as we shall see. Indeed, it is reminiscent of a similar situation in AC series circuits where an individual component voltage can be greater than the source voltage without violating Kirchhoff's voltage law.

To clarify our analyses, we shall make considerable use of both time domain plots of currents as well as phasor diagrams.

This page titled 11.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

11.2: The Parallel Connection

A parallel connection is always characterized by having all components share just two nodes. There are no other junctions from which current can flow out of or into, or for voltage to split. Consequently,

The voltage is the same everywhere in a parallel connection.

Each component in such a connection will see the same voltage, regardless of whether it is a resistor, capacitor or inductor. Before the branch currents can be determined, the capacitive and inductive reactance values must be computed. As seen from prior work, these are obtained from the capacitor and inductor values, and based on the frequency of the driving source. Consequently, if this frequency were to change, the reactances would change, and this would in turn cause changes in the various component currents as well as the total current delivered by the source.

This page titled 11.2: The Parallel Connection is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.2: The Parallel Connection** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

11.3: Parallel Impedance

Perhaps the first order of business is to determine equivalent impedance values for some collection of parallel components. Recall that the reciprocal of reactance is **susceptance**,

$$S = \frac{1}{X} \tag{11.3.1}$$

and that the reciprocal of impedance is **admittance**,

$$Y = \frac{1}{Z} \tag{11.3.2}$$

The units are siemens for each. It is also worth noting that, due to the division, the signs reverse. For example, a capacitive susceptance has an angle of +90 degrees and if a complex admittance has a negative angle, then the associated impedance is inductive.

The “conductance rule” for parallel combinations studied in the DC case remains valid for the AC case, although we generalize it for impedances:

$$Z_{total} = \frac{1}{\frac{1}{Z_1} + \frac{1}{Z_2} + \frac{1}{Z_3} + \cdots + \frac{1}{Z_N}} \tag{11.3.3}$$

Each of the individual impedances presented in Equation 11.3.3 (i.e., Z_1, Z_2, etc) can represent a simple resistance, a pure reactance or a complex impedance. Further, the product-sum rule shortcut for two components also remains valid for AC components:

$$Z_{total} = \frac{Z_1 \times Z_2}{Z_1 + Z_2} \tag{11.3.4}$$

There is one special case where Equation 11.3.4 can be “troublesome”, and that's when the two impedances consist of a pure capacitive reactance and a pure inductive reactance, both of the same magnitude. The two items will effectively cancel each other leaving a denominator of zero and an undefined result. While the theoretical combination “blows up” and approaches infinity, in reality it is limited by associated resistances such as R_{coil}, and arrives at some finite value This situation is studied in great depth in Chapter 8, which covers the concept of resonance.

Example 11.3.1

Determine the impedance of the network shown in Figure 11.3.1.

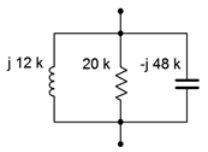


Figure 11.3.1: Circuit for Example 11.3.1.

Equation 11.3.3 would be best here.

$$Z_{total} = \frac{1}{\frac{1}{Z_1} + \frac{1}{Z_2} + \frac{1}{Z_3}}$$

$$Z_{total} = \frac{1}{\frac{1}{j12k\Omega} + \frac{1}{20k\Omega} + \frac{1}{-j48k\Omega}}$$

$$Z_{total} = 12.49E3\angle 51.3^{\circ}\Omega$$

This result might be a little surprising to the sharp-eyed. Notice that the magnitude of the total is larger than the magnitude of the smallest component (the inductor at $j12k\Omega$). This would never be the case if these three components were all resistors: the result would have to be smaller than the smallest element in the group.

The reason for this is that the capacitive reactance partially cancels the inductive reactance. If the product-sum rule (Equation 11.3.4) is used with these two components, the result is $16E3\angle 90^\circ$ or $j16k\Omega$. Placing that in parallel with the 20 kΩ resistor (again using Equation 11.3.4) leads to the result computed above.

An admittance diagram is illustrated in Figure 11.3.2. The vector summation of the component conductance and susceptances is verified nicely.

The individual component values are:

$$S_L = \frac{1}{j12k\Omega} \approx -j83.33E-6S$$

$$S_C = \frac{1}{-j48k\Omega} \approx j20.83E-6S$$

$$G = \frac{1}{20k\Omega} = 50E-6S$$

$$Y_{total} = \frac{1}{12.49E3\angle 51.3^\circ\Omega} \approx 80.1E-6\angle -51.3^\circ S$$

In rectangular form $Y_{total} = 50E-6-j62.5E-6S$.

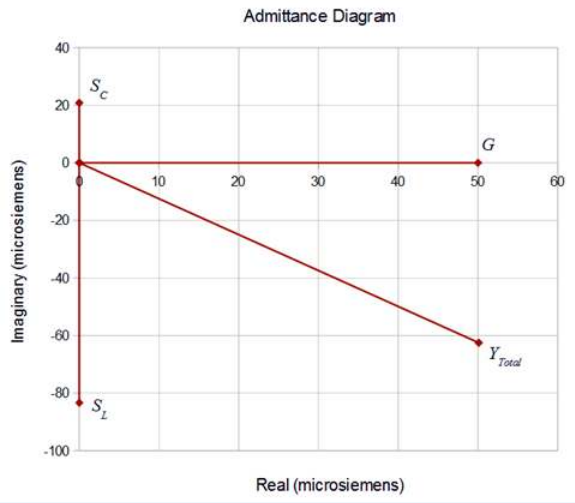


Figure 11.3.2: Admittance diagram for the network of Figure 11.3.1.

Example 11.3.2

Determine the impedance of the network shown in Figure 11.3.3 at a frequency of 10 kHz. Repeat this for a frequency of 1 kHz.

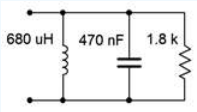


Figure 11.3.3: Circuit for Example 11.3.2.

First, find the reactances at 10 kHz. For the inductor we find:

$$X_L = j2\pi fL$$

$$X_L = j2\pi 10kHz680\mu H$$

$$X_L \approx j42.73\Omega$$

And for the capacitor:

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 10kHz 470nF}$$

$$X_C \approx -j33.86\Omega$$

Now use Equation 11.3.3 to combine the elements.

$$Z_{total} = \frac{1}{\frac{1}{Z_1} + \frac{1}{Z_2} + \frac{1}{Z_3}}$$

$$Z_{total} = \frac{1}{\frac{1}{j42.73\Omega} + \frac{1}{1.8k\Omega} + \frac{1}{-j33.86\Omega}}$$

$$Z_{total} = 162.6\angle -84.8^{\circ}\Omega$$

In rectangular form this is $14.68 - j161.9\Omega$. As the capacitor's reactance is the smallest of the three components, it dominates the equivalent impedance at this frequency. By working the capacitive reactance formula in reverse, it can be shown that the reactive portion of $-j161.9\Omega$ can achieved at this frequency by using a capacitance of 98.3 nF. That means that at 10 kHz, this parallel network has the same impedance as a 14.68 Ω resistor in series with a 98.3 nF capacitor. At any other frequency this will no longer be true, as will be illustrated momentarily.

At 1 kHz, the frequency is reduced by a factor of ten. Therefore, X_L will be ten times lower, or approximately $j4.273\Omega$. Further, X_C will be ten times higher, or about $-j338.6\Omega$. The inductive reactance will now dominate.

The new impedance is:

$$Z_{total} = \frac{1}{\frac{1}{Z_1} + \frac{1}{Z_2} + \frac{1}{Z_3}}$$

$$Z_{total} = \frac{1}{\frac{1}{j4.273\Omega} + \frac{1}{1.8k\Omega} + \frac{1}{-j338.6\Omega}}$$

$$Z_{total} = 4.328\angle 89.9^{\circ}\Omega$$

In rectangular form this is $10.4E-3 + j4.328\Omega$. The result is inductive, the opposite of what we saw at 10 kHz. Using the inductive reactance formula, it can be shown that at 1 kHz this parallel network has the same impedance as a 10.4 milliohm resistor in series with a 689 μH inductor.

This page titled 11.3: Parallel Impedance is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.3: Parallel Impedance** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

11.4: Parallel Circuit Analysis

Kirchhoff's current law (KCL) is the operative rule for parallel circuits. It states that the sum of all currents entering and exiting a node must equal zero. Alternately, it can be stated as the sum of currents entering a node must equal the sum of currents exiting that node. As a pseudo formula:

$$\sum I \rightarrow = \sum I \leftarrow \tag{11.4.1}$$

A similar situation occurs in AC parallel circuits to that seen AC series circuits; namely that it appears that KCL (like KVL) is “broken”. That is, a simplistic summing of the magnitudes of the currents might not balance. Once again, this is because the summation must be a vector summation, paying attention to the phase angles of each separate current.

It is possible to drive a parallel circuit with multiple current sources. These sources will add in much the same way that voltage sources in series add, that is, polarity and phase must be considered. Ordinarily, voltage sources with differing values are not placed in parallel as this violates the basic rule of parallel circuits (voltage being the same across all components).

The current divider rule remains valid for AC parallel circuits. Given two components, Z_1 and Z_2, and a current feeding them, I_T, the current through one of the components will equal the total current times the ratio of the opposite component over the sum of the impedance of the pair.

$$i_1 = i_T \frac{Z_2}{Z_1 + Z_2} \tag{11.4.2}$$

This rule is convenient in that the parallel equivalent impedance need not be computed, but remember, it is valid only when there are just two components involved.

When analyzing a parallel circuit, if it is being driven by a voltage source, then this same voltage must appear across each of the individual components. Ohm's law can then be used to determine the individual currents. According to KCL, the total current exiting the source must be equal to the sum of these individual currents. For example, in the circuit shown in Figure 11.4.1, the voltage E must appear across both R and L. Therefore, the currents must be $i_L = E/X_L$ and $i_R = E/R$, and $i_{total} = i_L + i_R$. i_{total} can also be found be determining the parallel equivalent impedance of R and X_L, and then dividing this into E. This technique can also be used in reverse in order to determine a resistance or reactance value that will produce a given total current: dividing the source by the current yields the equivalent parallel impedance. As one of the two is already known, the known component can be used to determine the value of the unknown component via Equation 3.3.3.

Example 11.4.1

Determine the currents in the circuit shown in Figure 11.4.1 if the source is $10\angle 0^\circ$ volts peak, $X_L = j2k\Omega$ and $R = 1k\Omega$

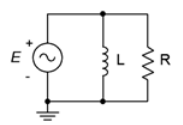


Figure 11.4.1: A simple parallel RL circuit.

The same voltage must appear across all elements in a parallel connection. In this case that's $10\angle 0^\circ$ volts peak. The two branch currents are found via Ohm's law:

$$i_L = \frac{v}{Z}$$

$$i_L = \frac{10\angle 0^\circ V}{j2k\Omega}$$

$$i_L = 5E - 3\angle -90^\circ A$$

$$i_R = \frac{v}{Z}$$

$$i_R = \frac{10\angle 0^\circ V}{1k\Omega}$$

$$i_R = 10E - 3\angle 0^\circ A$$

The source current is the sum of the these two currents, or $11.18E - 3\angle -26.6^\circ$ amps. This can be verified by determining the equivalent parallel impedance and then applying Ohm's law:

$$Z_T = \frac{Z_1 \times Z_2}{Z_1 + Z_2}$$

$$Z_T = \frac{1k\Omega \times j2k\Omega}{1k\Omega + j2k\Omega}$$

$$Z_T = 894\angle 26.6^\circ \Omega$$

As a side note, in rectangular form this is $800 + j400$, meaning that this parallel combination is equivalent to a series combination of an 800 ohm resistance and a 400 ohm inductive reactance. Continuing with Ohm's law, we have:

$$i_{Source} = \frac{v}{Z_T}$$

$$i_{Source} = \frac{10\angle 0^\circ V}{894\angle 26.6^\circ \Omega}$$

$$i_{Source} = 11.18E - 3\angle -26.6^\circ A$$

A phasor diagram of the currents is shown in Figure 11.4.2, below.

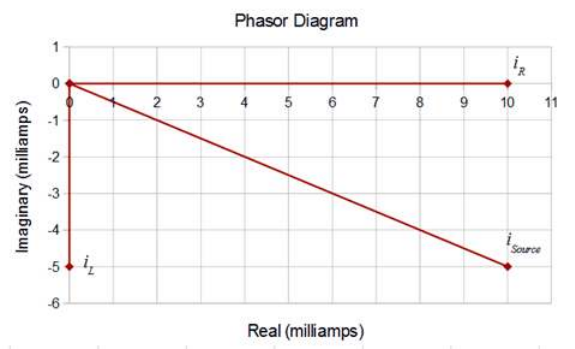


Figure 11.4.2: Phasor diagram of currents in the circuit of Figure 11.4.1.

Current Measurement in the Laboratory

At this point, a practical question arises; how do we verify these currents in the laboratory? After all, the preeminent measurement tool is the oscilloscope, and these are designed to measure voltage, not current. While it is possible to obtain current measurement probes, a simple method to measure current is to use a small current sensing resistor with a standard oscilloscope setup. We have seen already that the voltage across a resistor must be in phase with the current through it. Therefore,

whatever phase angle we see on a resistor's voltage, its current phase angle must be the same. The idea is to simply insert a resistor into the branch of which we wish to measure the current. As long as the resistance is much smaller than the impedance seen in the rest of that branch, it will have little impact on the precise value of the current. We can then place a probe across this sensing resistor to read its voltage. Ohm's law is then be used to determine the current's magnitude, while the phase angle is obtaining directly from the oscilloscope (again, because the sensing resistor's voltage must be in phase with its current, and thus the voltage phase angle is equal to the current phase angle). This technique will be illustrated in the simulation portion of the next example problem.

Example 11.4.2

Determine the branch currents in the circuit of Figure 11.4.3.

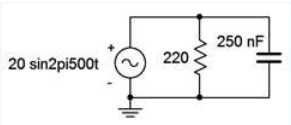


Figure 11.4.3: Circuit for Example 11.4.2.

The first step is to determine the capacitive reactance.

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 500Hz250nF}$$

$$X_C \approx -j1273\Omega$$

Both the resistor and capacitor will see 20 volts peak from the source. Their currents can be determined via Ohm's law:

$$i_C = \frac{v}{X_C}$$

$$i_C = \frac{20V}{1273\angle -90^\circ\Omega}$$

$$i_C \approx 15.71E-3\angle 90^\circ A$$

$$i_R = \frac{v}{R}$$

$$i_R = \frac{20V}{220\angle 0^\circ\Omega}$$

$$i_R \approx 90.91E-3\angle 0^\circ A$$

The source current is the sum of these two currents, or $92.26E-3\angle 9.8^\circ$ amps. The source current may also be determined by dividing the source voltage by the equivalent parallel impedance, as follows.

$$Z_T = \frac{Z_1 \times Z_2}{Z_1 + Z_2}$$

$$Z_T = \frac{220\Omega \times (-j1273\Omega)}{220\Omega - j1273k\Omega}$$

$$Z_T = 216.8\angle -9.8^\circ\Omega$$

$$i_{Source} = \frac{v}{Z_T}$$

$$i_{Source} = \frac{20\angle 0^\circ V}{216.8\angle -9.8^\circ\Omega}$$

$$i_{Source} = 92.26E-3\angle 9.8^\circ A$$

A time domain plot of the currents is illustrated in Figure 11.4.4 along with a phasor plot in Figure 11.4.5. Note that the source current is close in both amplitude and phase to the resistor current. By comparison, the capacitor current is considerably smaller and with an obvious leading phase shift.

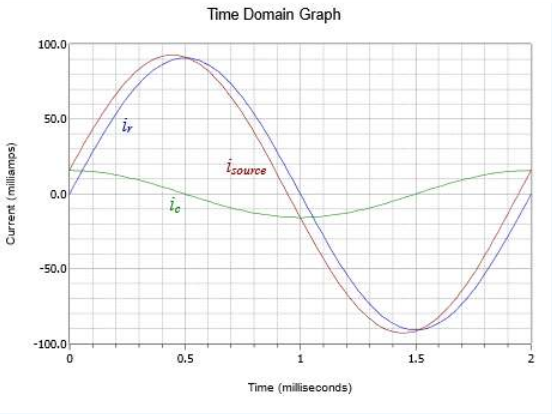


Figure 11.4.4: Current waveforms for the circuit of Figure 11.4.3.

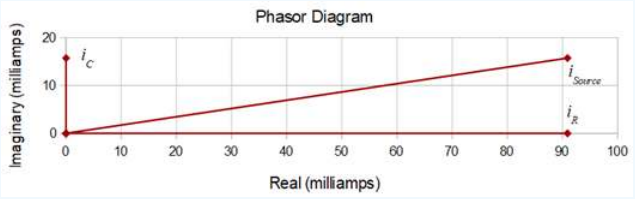


Figure 11.4.5: Phasor diagram for the circuit of Figure 11.4.3.

Computer Simulation

The circuit of Figure 11.4.3 is captured in a simulator as shown in Figure 11.4.6. Individual 2 ohm resistors are used to sense the currents in the resistor and capacitor branches. These sensing resistors are inserted directly above ground for convenience of measure. In this way a differential measurement is not needed.

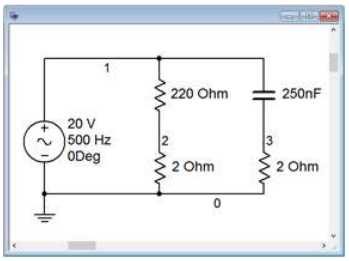


Figure 11.4.6: Circuit of Figure 11.4.3 in a simulator with added current sensing resistors.

The value of a sensing resistor needs to be considerably smaller than the impedance of the branch in which it is inserted. Two orders of magnitude (i.e., less than 1 %) would be a good place to start. While even smaller values will increase accuracy, from a practical standpoint in the lab, such tiny resistor values would produce extremely small voltages that would be difficult to measure with great accuracy.

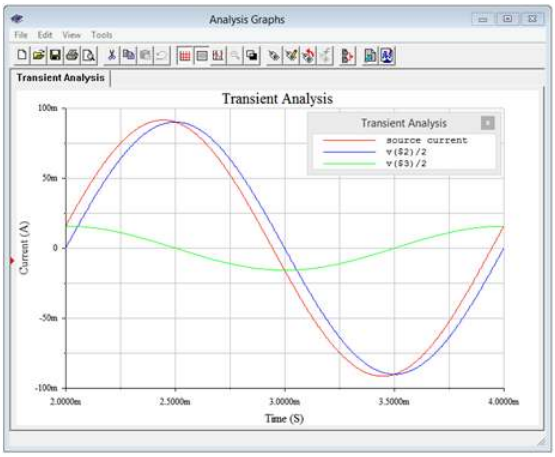


Figure 11.4.7: Simulation results for the circuit of Figure 11.4.6.

A basic transient analysis is then performed as shown in Figure 11.4.7. The results of the simulation are in tight agreement with the plot of Figure 11.4.4. Note that the plotted resistor and capacitor "currents" are, in fact, the voltages across the associated sensing resistors (nodes 2 and 3) divided by 2 (ohms), in direct application of Ohm's law. Not only does the simulation verify the results computed earlier, it also validates the concept of using small current sensing resistors in the laboratory.

Example 11.4.3

Determine the the branch currents in the circuit of Figure 11.4.8. Use the source voltage as the reference ($\angle 0^\circ$).

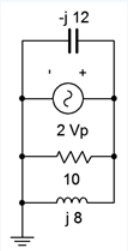


Figure 11.4.8: Circuit for Example 11.4.3.

Although this circuit is drawn a little differently than the prior examples, it remains a simple parallel circuit with just two nodes. Therefore, each element sees a 2 volt peak potential. Ohm's law will suffice to find the three component currents. These are then added to find the source current. Their reference directions are right-to-left given the source's polarity.

$$i_C = \frac{v}{X_C}$$

$$i_C = \frac{2\angle 0^\circ V}{12\angle -90^\circ \Omega}$$

$$i_C \approx 0.1667\angle 90^\circ A$$

$$i_L = \frac{v}{X_L}$$

$$i_L = \frac{2\angle 0^\circ V}{8\angle 90^\circ \Omega}$$

$$i_L = 0.25\angle -90^\circ A$$

$$i_R = \frac{v}{R}$$

$$i_R = \frac{2\angle 0^\circ V}{10\angle 0^\circ \Omega}$$

$$i_R = 0.2\angle 0^\circ A$$

The sum of these three currents is approximately $0.2167\angle -22.6^\circ$ amps. This value can be verified by finding the combined parallel impedance. Using Equation 3.3.3 it can be shown that this impedance is $9.231\angle 22.6^\circ \Omega$. Thus,

$$i_{Source} = \frac{v}{Z}$$

$$i_{Source} = \frac{2\angle 0^\circ V}{9.231\angle 22.6^\circ \Omega}$$

$$i_{Source} \approx 0.2167\angle -22.6^\circ A$$

A phasor diagram showing the vector summation of the currents is illustrated in Figure 11.4.9.

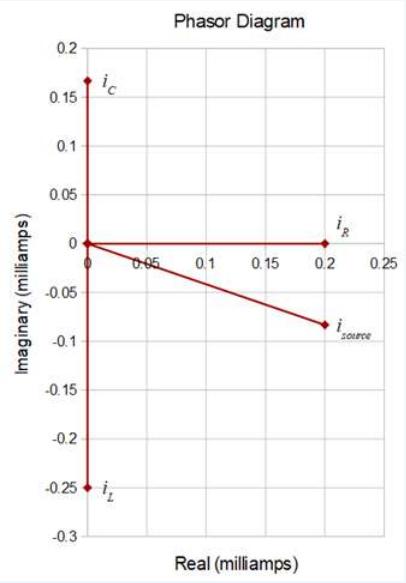


Figure 11.4.9: Current phasor diagram for the circuit of Figure 11.4.8.

Note that i_C and i_L are in perfect opposition as expected. Subtracting the magnitude of i_C from i_L leaves us with the precise vertical component of the source current.

If a parallel circuit is driven by a current source, as shown in Figure 11.4.10, there are two basic methods of solving for the component currents. The first method is to use the current divider rule. If desired, the component voltage can then be found using Ohm's law. An alternate method involves finding the parallel equivalent impedance first and then using Ohm's law to determine the voltage (remember, being a parallel circuit, there is only one common voltage). Given the voltage, Ohm's law can be used to find the current through one component. To find the current through the other, either Ohm's law can be applied a second time or KCL may be used; subtracting the current through the first component from the source current. If there are more than two components, usually the second method would be the most efficient course of action.

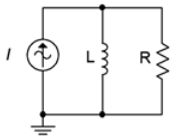


Figure 11.4.10: Parallel network driven by a current source.

It is worth noting that both methods described above will yield the correct answers. One is not “more correct” than the other. We can consider each of these as a separate solution path; that is, a method of arriving at the desired end point. In general, the more complex the circuit, the more solution paths there will be. This is good because one path may be more obvious to you than another. It also allows you a means of crosschecking your work.

Example 11.4.4

Determine the currents in the circuit of Figure 11.4.10 if the source is 300 milliamps peak at 10 kHz, R is 750Ω and L is 15 mH.

The first step is to determine the inductive reactance.

$$X_L = j2\pi fL$$

$$X_L = j2\pi 10kHz15mH$$

$$X_L \approx j942.4\Omega$$

The current divider rule can be used to find the current through the resistor.

$$i_R = i_{Source}\frac{X_L}{R + X_L}$$

$$i_R = 0.3\angle 0^\circ A\frac{942.4\angle 90^\circ\Omega}{750\angle 0^\circ\Omega + 942.4\angle 90^\circ\Omega}$$

$$i_R = 0.2347\angle 38.5^\circ A$$

KVL can be used to find the remaining current through the inductor as the source current must equal the sum of the inductor and resistor currents. Therefore:

$$i_L = i_{Source} - i_R$$

$$i_L = 0.3\angle 0^\circ A - 0.2347\angle 38.5^\circ A$$

$$i_L = 0.1868\angle -51.5^\circ A$$

This value could also have been obtained by applying the current divider rule a second time:

$$i_L = i_{Source}\frac{R}{R + X_L}$$

$$i_L = 0.3\angle 0^\circ A\frac{750\angle 0^\circ\Omega}{750\angle 0^\circ\Omega + 942.4\angle 90^\circ\Omega}$$

$$i_L = 0.1868\angle -51.5^\circ A$$

A time domain plot of the current waveforms is shown in Figure 11.4.11.

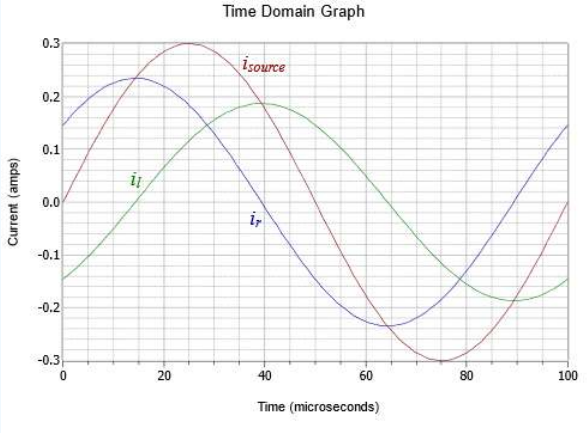


Figure 11.4.11: Current waveforms for the circuit of Example 11.4.4.

From the plot it is apparent that the inductor and resistor currents do indeed add up to the source current. Further, the current through the inductor is lagging, as expected.

Computer Simulation

The circuit of Example 11.4.4 is captured in a simulator as shown in Figure 11.4.12. A pair of 1 ohm current sensing resistors have been added. By using 1 ohm, the sensing voltage will have the same magnitude as the current, with no scaling required.

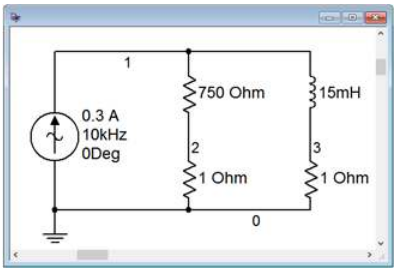


Figure 11.4.12: Circuit of Example 11.4.4 in a simulator.

A transient analysis is performed, plotting the voltages at nodes 2 and 3 along with their sum (the source current). The results are shown in Figure 11.4.3. The plot is delayed for a millisecond in order to avoid the initial power-up transient. The results are in full agreement with the plot of Figure 11.4.11.

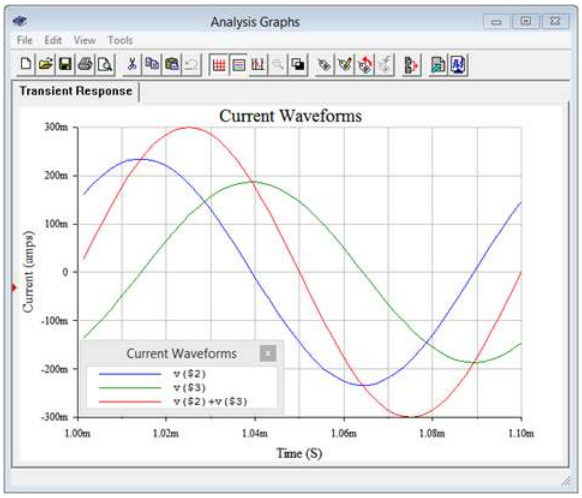


Figure 11.4.13: Results of the transient response simulation for the circuit of Example 11.4.5.

Example 11.4.5

Determine the branch currents in the circuit of Figure 11.4.14. Use the current source as the time reference (i.e., $\angle 0^\circ$).

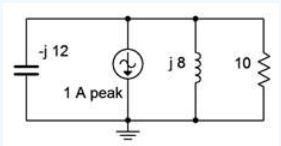


Figure 11.4.14: Circuit for Example 11.4.5.

Instead of using repeated current dividers here, we can instead find the equivalent impedance and then apply Ohm's law to determine the system voltage. Each branch current may be obtained by dividing this voltage by the impedance of that branch; a quick application of Ohm's law.

The system impedance is computed through Equation ??? as follows:

$$Z_{total} = \frac{1}{\frac{1}{Z_1} + \frac{1}{Z_2} + \frac{1}{Z_3}}$$

$$Z_{total} = \frac{1}{\frac{1}{-j12\Omega} + \frac{1}{j8\Omega} + \frac{1}{10\Omega}}$$

$$Z_{total} = 9.231\angle 22.6^\circ \Omega$$

https://eng.libretexts.org/@go/page/52964

The system voltage is found using Ohm's law. As the reference direction of the current source is flowing down into ground, the branch currents will be flowing up. This makes their voltage reference negative with respect to ground (i.e., + to − from bottom to top). We can deal with this by describing the current source as negative (i.e., flowing out of the top node). The source can be written as either $-1\angle 0^\circ$ or as $1\angle 180^\circ$, it makes no difference.

$$v = i_{source} \times Z_{total}$$

$$v = -1\angle 0^\circ A \times 9.231\angle 22.6^\circ$$

$$v \approx -9.231\angle 22.6^\circ V$$

This potential may also be written as $9.231\angle -157.4^\circ$ volts (remember, an inversion is the same as a 180 degree phase shift).

And now for the branch currents:

$$i_C = \frac{v}{X_C}$$

$$i_C = \frac{9.231\angle -157.4^\circ V}{12\angle -90^\circ \Omega}$$

$$i_C \approx 0.7692\angle -67.4^\circ A$$

$$i_L = \frac{v}{X_L}$$

$$i_L = \frac{9.231\angle -157.40^\circ V}{8\angle 90^\circ \Omega}$$

$$i_L = 1.154\angle 112.6^\circ A$$

$$i_R = \frac{v}{R}$$

$$i_R = \frac{9.231\angle -157.4^\circ V}{10\angle 0^\circ \Omega}$$

$$i_R = 0.9231\angle -157.4^\circ A$$

The sum of these three currents is approximately $1\angle 180^\circ$ amps (i.e., $-1\angle 0^\circ$), the value of the current source, as expected.

A phasor diagram of the currents is plotted in Figure 11.4.15.

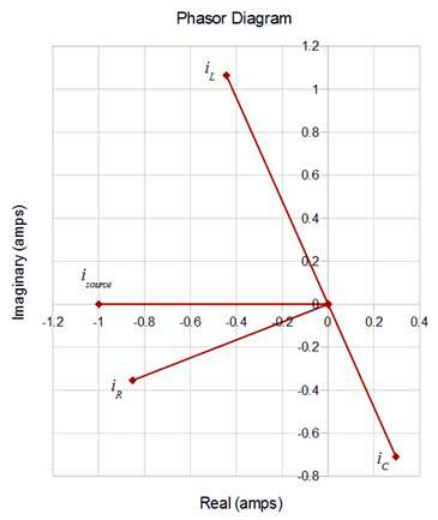


Figure 11.4.15: Current phasor diagram for the circuit of Figure 11.4.13.

Now for the sneaky bit. In case you didn't notice, the components in this circuit are identical to those of Figure 11.4.8 with the exception of the source. As a consequence, the phase angles and magnitudes of the branch currents through the three components will remain unchanged relative to each other. Due to the change in both magnitude and direction of the source, the

entire phasor plot has been rotated clockwise by 154.7 degrees and all of the magnitudes have been lengthened by the ratio of the voltages $(9.231/2)$. This is perhaps easiest to see by focusing on the vector for i_{source} in Figure 11.4.9. Rotate that vector clockwise, along with every other vector, until i_{source} lines up with the negative real axis. The result is the plot of Figure 11.4.15, albeit with newly lengthened magnitudes.

If a parallel circuit contains multiple current sources, they may be added together (a vector sum, of course) to generate an equivalent single current source. The analysis then proceeds as before. This is demonstrated in the following example.

Example 11.4.6

For the circuit of Figure 11.4.16, determine the currents through the capacitor, inductor and resistor, and also determine the system voltage. i_1 is $0.5\angle 0^\circ$ amps and i_2 is $3\angle 45^\circ$ amps.

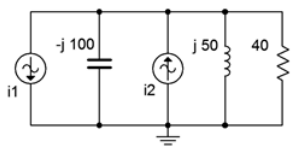


Figure 11.4.16: Circuit for Example 11.4.6.

The first step here is to determine the effective current driving the circuit. If we treat the top node as positive, i_2 is entering (positive) while i_1 is exiting (negative). The combination is:

$$i_{total} = i_1 + i_2$$

$$i_{total} = -0.5\angle 0^\circ A + 3\angle 45^\circ A$$

$$i_{total} = 2.67\angle 52.6^\circ A$$

This current has the same reference direction as source two. A phasor diagram of the process is illustrated in Figure 11.4.17 to help visualize the vector addition.

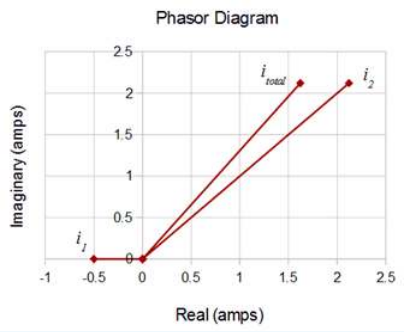


Figure 11.4.17: Phasor diagram of the combined current sources for the circuit for Example 11.4.6.

The next step is to find the combined impedance so that we can find the applied voltage.

$$Z_{total} = \frac{1}{\frac{1}{Z_1} + \frac{1}{Z_2} + \frac{1}{Z_3}}$$

$$Z_{total} = \frac{1}{\frac{1}{-j100\Omega} + \frac{1}{j50\Omega} + \frac{1}{40\Omega}}$$

$$Z_{total} = 37.14\angle 21.8^\circ \Omega$$

$$v = i_{source} \times Z_{total}$$

$$v = 2.67\angle 52.6^\circ A \times 37.14\angle 21.8^\circ$$

$$v \approx 99.16\angle 74.4^\circ V$$

We now apply Ohm's law to each of the components to determine the branch currents.

$$i_C = \frac{v}{X_C}$$

$$i_C = \frac{99.16\angle 74.4^\circ V}{100\angle -90^\circ \Omega}$$

$$i_C \approx 0.9916\angle 164.4^\circ A$$

$$i_L = \frac{v}{X_L}$$

$$i_L = \frac{99.16\angle 74.4^\circ V}{50\angle 90^\circ \Omega}$$

$$i_L = 1.983\angle -15.6^\circ A$$

$$i_R = \frac{v}{R}$$

$$i_R = \frac{99.16\angle 74.4^\circ V}{40\angle 0^\circ \Omega}$$

$$i_R = 2.479\angle 74.4^\circ A$$

The sum of these three currents is approximately $2.67\angle 52.6^\circ$ amps. This balances the net entering current from the two sources, verifying KVL.

This page titled 11.4: Parallel Circuit Analysis is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.4: Parallel Circuit Analysis** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

11.5: Summary

In this chapter we have examined parallel circuits using either a single voltage source, or one or more AC current sources, along with two or more resistors, capacitors and inductors. The defining characteristic of a parallel configuration is that all components are connected to just two nodes and that all elements in this configuration see the same voltage. If multiple AC current sources are present, they may be combined into a single equivalent current source via vector addition.

Unlike the purely resistive case, the equivalent impedance of a group of parallel RLC components will not always be smaller than the smallest resistance or reactance in the group due to cancellations between the inductance and capacitance. As voltage is identical for all components, then the currents through the capacitors must be 180 degrees out of phase with the currents through the inductors. Thus it is quite possible that the current through one of the reactance branches could be larger than the current supplied by the source. In general, the effective impedance is found by summing the individual conductances and susceptances to find the total admittance of the group, and then taking the reciprocal of this value. The product-sum rule may be used so long as the angles are taken into account (i.e., a vector process).

Kirchhoff's current law (KCL) states that the sum of currents entering a node must equal the sum of currents leaving that node. This remains true for AC circuit analysis, however, it must be remembered to always use a vector summation. A simple summation of current magnitudes will not achieve proper results.

Individual branch currents in a circuit driven by a voltage source may be determined by using Ohm's law: simply divide the source voltage by the individual resistance and reactance values. These branch currents must sum to the total current delivered by the voltage source thanks to KCL. If the parallel network is driven by current sources, the individual branch currents can be found by first determining the effective parallel impedance and then using Ohm's law to find the system voltage. Once the voltage is known, Ohm's law is used again on each component to find the associated branch current. Alternately, the current divider rule may be used repeatedly on successive pairings of components.

Review Questions

1. How is the equivalent impedance for a group of parallel connected resistors, inductors and capacitors computed?
2. How is the equivalent value for parallel connected AC current sources computed?
3. Is the product-sum rule still applicable for AC analysis? Can it be used for reactance and/or complex impedance?
4. Define Kirchhoff's current law for AC circuit analysis.
5. Is it possible for a branch current to be larger than the source current in an AC parallel circuit? Explain why/why not.

This page titled 11.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

11.6: Exercises

Analysis

1. Determine the effective impedance of the network shown in Figure 11.6.1 at 10 MHz.

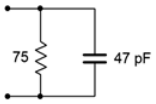


Figure 11.6.1

2. Determine the effective impedance of the network shown in Figure 11.6.2 at 100 Hz.

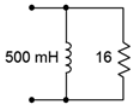


Figure 11.6.2

3. Determine the effective impedance of the network shown in Figure 11.6.3 at 5 kHz.

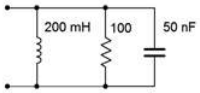


Figure 11.6.3

4. Determine the effective impedance of the network shown in Figure 11.6.4 at 20 kHz.

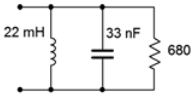


Figure 11.6.4

5. Determine the effective impedance of the network shown in Figure 11.6.5.

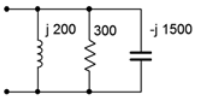


Figure 11.6.5

6. Determine the effective impedance of the network shown in Figure 11.6.5 if the frequency is halved and if the frequency is doubled.

7. For the network shown in Figure 11.6.1, determine the frequency below which the impedance is mostly resistive.

8. For the network shown in Figure 11.6.2, determine the frequency below which the impedance is mostly inductive.

9. Draw phasor impedance plot for problem 1.

10. Draw phasor impedance plot for problem 2.

11. Determine the three branch currents for the circuit shown in Figure 11.6.6 and draw their phasor diagram.

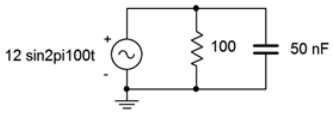


Figure 11.6.6

12. Determine the three branch currents for the circuit shown in Figure 11.6.7 and draw their phasor diagram.

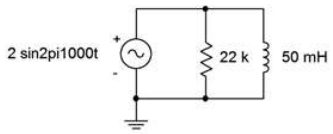


Figure 11.6.7

13. Determine the four branch currents for the circuit shown in Figure 11.6.8 and draw their phasor diagram.

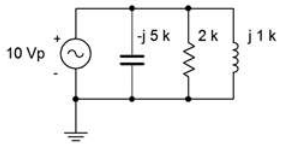


Figure 11.6.8

14. Determine all of the branch currents for the circuit shown in Figure 11.6.9 assuming E is a 1 volt RMS sine.

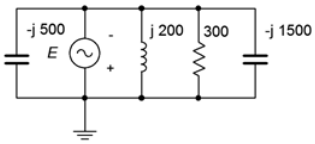


Figure 11.6.9

15. Determine all of the branch currents for the circuit shown in Figure 11.6.10 given E = 10 volt peak sine, R = 220, $X_C = -j500$. and $X_L = j1.5$ k.

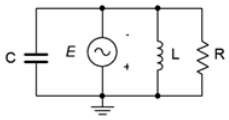


Figure 11.6.10

16. Determine all of the branch currents for the circuit shown in Figure 11.6.11 given E = 2 volt peak sine, R = 1 k, $X_C = -j2$ k. and $X_L = j3$ k.

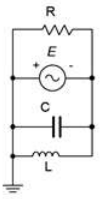


Figure 11.6.11

17. Determine the component currents for the circuit shown in Figure 11.6.12. Draw phasor diagram of the source and branch currents.

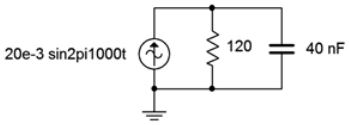


Figure 11.6.12

18. Determine the resistor and capacitor voltages for the circuit shown in Figure 11.6.12.

19. Determine the resistor and inductor voltages for the circuit shown in Figure 11.6.13.

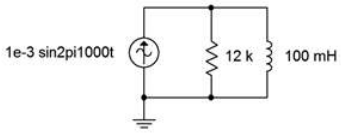


Figure 11.6.13

20. Determine the component currents for the circuit shown in Figure 11.6.13. Draw phasor diagram of the source and branch currents.

21. Determine the source voltage for the circuit shown in Figure 11.6.14.

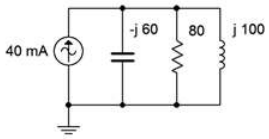


Figure 11.6.14

22. Determine the component currents for the circuit shown in Figure 11.6.14. Draw the phasor diagram of the source and branch currents.

23. Determine the component currents for the circuit shown in Figure 11.6.15. I is 20 mA at 0 degrees.

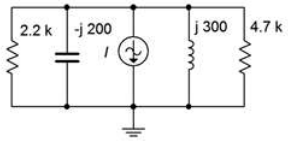


Figure 11.6.15

24. Determine the source voltage for the circuit shown in Figure 11.6.15. I is 20 mA at 0 degrees.

25. Determine the source voltage for the circuit shown in Figure 11.6.16. Assume $I1$ is 1 mA at 0 degrees and $I2$ is 2 mA at +90 degrees.

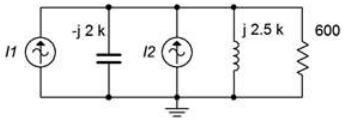


Figure 11.6.16

26. Determine the capacitor and inductor currents in the circuit of Figure 11.6.16. Assume $I1$ is 1 mA at 0 degrees and $I2$ is 2 mA at +90 degrees.

27. Determine the resistor and capacitor currents in the circuit of Figure 11.6.17. Assume $I1$ is $2\angle 0^\circ$ amps and $I2$ is $0.5\angle 45^\circ$.

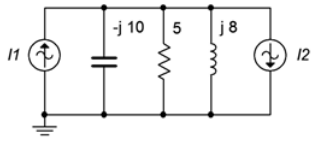


Figure 11.6.17

28. Determine the source voltage for the circuit shown in Figure 11.6.17. Assume $I1$ is $2\angle 0^\circ$ A and $I2$ is $0.5\angle 45^\circ$.

Design

29. For the network shown in Figure 11.6.18, determine a value of C such that the impedance magnitude of the circuit is 1 kΩ. The source is a 50 Hz sine and R is 2.2 kΩ.

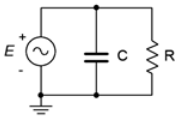


Figure 11.6.18

30. For the network shown in Figure 11.6.19, determine a value of L such that the impedance magnitude of the circuit is 2 kΩ. The source is a 2 MHz sine and R is 3.3 kΩ.

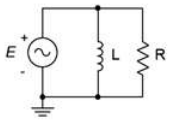


Figure 11.6.19

31. For the circuit shown in Figure 11.6.18, determine a value for C such that the magnitude of the source current is 1 mA. E is a 2 volt 10 kHz sine and R = 8 kΩ.

32. For the network shown in Figure 11.6.19, determine a value for L such that the magnitude of the source current is 10 mA. E is a 25 volt 100 kHz sine and R = 4 kΩ.

33. For the network shown in Figure 11.6.20, determine a value of C such that the impedance magnitude of the circuit is 10 k Ω. The source is a 440 Hz sine and R is 33 kΩ.

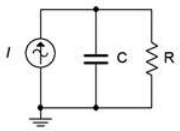


Figure 11.6.20

34. For the network shown in Figure 11.6.21, determine a value of L such that the impedance magnitude of the circuit is 200 Ω. The source is a 60 Hz sine and R is 680 Ω.

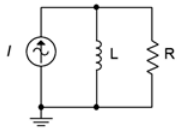


Figure 11.6.21

35. For the circuit shown in Figure 11.6.20, determine a value for C such that the magnitude of the circuit voltage is 200 volts. The source current is a 100 mA 1200 Hz sine and R = 15 kΩ.

36. For the circuit shown in Figure 11.6.21, determine a value for L such that the magnitude of the circuit voltage is 50 volts. The source current is a 2.3 A 60 Hz sine and R = 330 Ω.

37. Given the circuit shown in Figure 11.6.18, determine a value for C such that the impedance angle is −45 degrees. The source a 1 volt peak sine at 600 Hz and R = 680 Ω.

38. Given the circuit shown in Figure 11.6.19, determine a value for L such that the impedance angle is 45 degrees. The source a 10 volt peak sine at 100 kHz and R = 1.2 kΩ.

39. Determine a value for C such $|X_C| = |X_L|$ for the circuit shown in Figure 11.6.22. The source frequency is 1 kHz, R = 200 Ω and L = 50 mH.

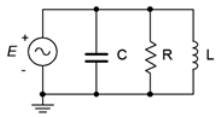


Figure 11.6.22

40. Determine a value for L such $|X_C| = |X_L|$ for the circuit shown in Figure 11.6.23. The source frequency is 22 kHz, R = 18 kΩ and C = 5 nF.

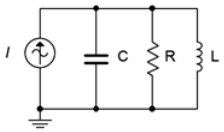


Figure 11.6.23

41. Add one or more components in parallel with the circuit of Figure 11.6.2 such that the resulting impedance at 20 Hz is 10 Ω with a phase angle of at least +30°.

Challenge

42. Determine a value for C such that the impedance angle for the circuit shown in Figure 11.6.22 is purely resistive (0 degrees). The source frequency is 1 kHz, R = 200 Ω and L = 50 mH.

43. Is it possible to change the value of the resistor in Figure 11.6.14 so that the system voltage is 4 volts? If so, what is the value? If not, why not?

44. Is it possible to change the value of the inductor and/or capacitor in Figure 11.6.14 so that the system voltage is 4 volts? If so, what is/are the values? If not, why not?

45. Assume you are troubleshooting a circuit like the one shown in Figure 11.6.23. I is a 10 mA peak sine at 2 kHz, R = 390 Ω, C = 200 nF and L = 25 mH. The measured resistor voltage is a little under 2.5 volts. What is the likely culprit?

46. Given the circuit shown in Figure 11.6.23, find the values for C and L if the source is a sine wave at 1 kHz, R = 4 kΩ, i_{Source} = 3 mA, i_R = 2 mA, i_L = 5 mA,

Simulation

47. Using a transient analysis simulation, verify that the source current magnitude is 1 mA using the capacitor value determined in design problem 31.

48. Using a transient analysis simulation, verify that the source current magnitude is 10 mA using the inductor value determined in design problem 32.

49. Using a transient analysis simulation, verify that the source voltage magnitude is 200 volts using the capacitor value determined in design problem 35.

50. Using a transient analysis simulation, verify that the source voltage magnitude is 50 volts using the inductor value determined in design problem 36.

51. Using a transient analysis simulation, verify the design solution for problem 39. This can be checked by seeing if the current magnitudes in C and L are identical.

52. Using a transient analysis simulation, verify the design solution for problem 40. This can be checked by seeing if the current magnitudes in C and L are identical.

53. Impedance magnitude as a function of frequency can be investigated by driving the circuit with a fixed amplitude current source across a range of frequencies. The resulting voltage will be proportional to the effective impedance. Investigate this effect by performing an AC analysis on the circuits shown in Figures 11.6.12 and 11.6.13. Use a frequency range of 10 Hz to 1 MHz. Before running the simulations, sketch your expected results.

54. Following the idea presented in the previous problem, investigate the impedance as a function of frequency of the circuit shown in Figure 11.6.23. Use R = 1 k, C = 10 nF, and L = 1 mH. Run the simulation from 100 Hz to 10 MHz. Make sure to sketch your expected results first.

This page titled 11.6: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **3.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

12: AC Circuit Analysis Theorems and Techniques

Learning Objectives

After completing this chapter, you should be able to:

- Find the voltage source equivalent of a current source and vice versa.
- Compute voltages and currents in multi-source RLC networks using superposition.
- Simplify RLC networks using Thévenin's and Norton's theorems.
- Determine conditions for maximum power transfer and compute the maximum power.
- Utilize delta-Y and Y-delta conversions for circuit simplification.

12.1: Introduction

12.2: Source Conversions

12.3: Superposition Theorem

12.4: Thévenin's and Norton's Theorems

12.5: Maximum Power Transfer Theorem

12.6: Advanced Topic- Delta-Y (Pi-T) Conversions

12.7: Summary

12.8: Exercises

This page titled 12: AC Circuit Analysis Theorems and Techniques is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

12.1: Introduction

In this chapter we shall examine a number of theorems and techniques to help us analyze complex circuits and address specialized applications. We will begin by examining the concept of source impedance in order to make more accurate models of our idealized constant voltage and current sources. This will be a step beyond using a simple resistance as found in the DC case. From there we will investigate how to convert from one type of source to another, such as creating a voltage source that is the functional equivalent of a current source. A functional equivalent is a source that can be swapped out for another while leaving all of the other circuit currents and voltages intact. In other words, all of the circuit's component voltage drops and branch currents will be identical to those found in the original configuration. This technique is useful in a number of ways, particularly in that it can help reduce more complex circuits to simplify analysis.

The concept of equivalence can be extended beyond just a single source to an entire network. For this we will examine Thévenin's and Norton's theorems. Using these theorems, entire circuits utilizing dozens of components can be modeled as a single source with an associated complex impedance. When coupled with the maximum power transfer theorem, these tools will allow us to determine component values that produce the maximum amount of load power.

We will also address a method of analyzing circuits that contain multiple current and/or voltage sources that are connected in a non-trivial fashion (i.e., not just series voltage sources or parallel current sources). This is called the superposition theorem and it can be applied to any circuit or parameter that meets certain requirements, including circuits that have a mix of current sources and voltage sources. Superposition can also be used to determine voltages and currents when sources use different frequencies. In fact, one of way of imagining a complex wave-shape is to treat it as a series of connected sources, each with a unique frequency, phase and amplitude. Superposition we give us a means to handle this new situation.

Finally, we will examine how to find equivalent circuits for certain component arrangements that use three connecting points, in other words, RLC combinations shaped like a triangle or like the letter Y. These are known as delta and Y configurations. These configurations are difficult to address with basic series-parallel simplification techniques. Converting from one configuration to the other will help solve that issue.

This page titled 12.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.2: Source Conversions

In DC analysis, we noted that real world sources have practical limits: voltage sources cannot produce infinite current and current sources cannot produce infinite voltage. A simple way of creating a more accurate model for independent sources is to include an internal resistance. For DC voltage sources, a resistance is added in series with the source, and for DC current sources a resistance is added in parallel with the source. While this works well enough for typical DC sources, the AC situation is a little more complicated.

Models for AC Sources

Just as we added a simple resistance to the DC sources to make improved models, we can add a complex impedance to AC sources to do likewise. Once again, it is possible to make even more involved models that will be more accurate, but for most work, this addition will suffice. Generally, there are wider variations in values for the AC case than the DC case.

We can think of AC sources as belonging in one of two broad categories. First, there are power generators, that is, systems designed to generate and deliver power for other electrical devices. This would include the AC mains system in a residence or a portable power generator. At the other end of the spectrum are signal sources such as transducers and sensors. These devices are generally low power and are not designed to produce particularly high currents, quite the opposite of generators.

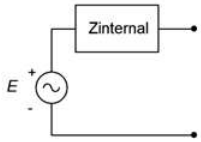


Figure 12.2.1: Practical AC voltage source model.

The model for an AC voltage source adds an impedance in series, as shown in Figure 12.2.1. This impedance sets an upper limit on the source's current output. Even if the output terminals are shorted, the maximum current will be dictated via Ohm's law to be the source voltage divided by the internal impedance, or $E/Z_{internal}$. Obviously, this internal impedance will create some voltage divider effect with the attached load. To minimize this effect, the impedance should be as small as practicably possible. Thus,

$$\text{The ideal internal impedance of a voltage source is zero ohms (a short).}$$

It is not always possible to get close to this ideal. In fact, in some situations the AC source is far removed from the ideal.

Consider the signal source shown in Figure 12.2.2. This is a pickup for an electric bass guitar. This device is used to translate the motions of guitar strings into a voltage that can be fed to an amplifier. It consists of a few thousand turns of very fine magnet wire wrapped around a magnet.



Figure 12.2.2: An electric bass guitar pickup (cover removed).

When the metal guitar strings vibrate, they alter the surrounding magnetic field. As the magnetic field changes, it induces a current, and hence a voltage, in the coil. The resulting signal is an electrical analog of the string vibrations. It is, quite literally, an AC signal source. But what of its internal impedance? The coil is made of wire between AWG 40 and 45, and may be in excess of 1000 feet (300 meters) in length. Consulting an AWG table[1], we find that this will yield 1000 or more ohms of resistance. Of course, a few thousand turns of wire will also create a hefty inductance and pickups such as this may exhibit an inductance of a few henries. Putting these pieces together, the equivalent internal impedance might look something like that shown in Figure 5.3.

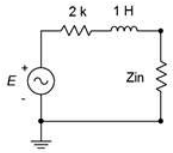


Figure 12.2.3: Example of internal impedance of guitar pickup.

Clearly, values like these are a far cry from the fractional ohm values found in power generation sources, and they present unique challenges. In the circuit of Figure 12.2.3, the component labeled Z_{in} represents the input impedance of the associated amplifier. Typically, it would be highly resistive but the magnitude can vary quite a bit depending on the application. A guitar or bass amplifier may exhibit 1 MΩ or more, while an auxiliary input on a home stereo might be 10 kΩ. What happens if you plug a guitar into your hi-fi? Obviously, this system creates a voltage divider between the internal impedance of the pickup and the input impedance of the amplifier, but more importantly, the divider is a function of frequency. At low frequencies like 100 Hz, the inductive reactance is small, around $j600\Omega$, and the signal loss to a 10 kΩ input is not that great. On the other hand, at a high frequency such as 10 kHz, the inductive reactance will be over $j60k\Omega$ resulting in considerable signal loss. Thus, the high frequencies are reduced relative to the low frequencies. The sonic effect is akin to turning down the treble control — everything will seem muffled. If the input impedance is increased considerably, say by a factor of 100, then the voltage divider effect at all audible frequencies will be negligible and the the instrument will sound true to form. Finally, although this example shows a substantial inductance, it is possible for AC sources to have an associated capacitance, or even have negligible reactance (i.e., be purely resistive).

For a current source, the improved model adds an impedance in parallel, as shown in Figure 12.2.4. This impedance sets an upper limit on the source's voltage output. If the output terminals are opened, the maximum voltage will no longer produce a huge voltage. Instead, it is dictated by Ohm's law to be the source current times the internal impedance, or $I \cdot Z_{internal}$. This internal impedance will create some current divider effect with the attached load. To minimize this effect, the internal impedance should be as large as practicably possible. Thus,

The ideal internal impedance of a current source is infinite ohms.

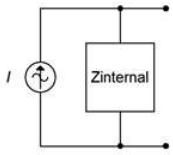


Figure 12.2.4: Practical AC current source model.

From here on, whenever we deal with practical voltage and current sources, we understand that these sources have some associated internal resistance, even if they're not shown explicitly in a schematic diagram. Further, whenever we talk about ideal sources, we simply use a short for the internal impedance of a voltage source and an open for the internal impedance of a current source.

Source Equivalences

For any voltage source consisting of an ideal voltage source with a series internal impedance, an equivalent current source may be created. Similarly, for any current source consisting of an ideal current source with a parallel internal impedance, an equivalent voltage source may be created. By “equivalent”, we mean that both circuits will produce the same voltage and current to identical loads. Consider the simple voltage source on the left side of Figure 12.2.5. Its equivalent current source is shown on the right.

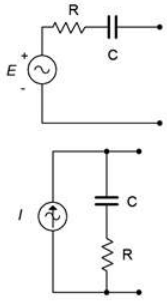


Figure 12.2.5: A simple AC voltage source (left) and corresponding current source (right).

For reasons that will become apparent under the section on Thévenin's theorem following, the internal impedances of these two circuits must be identical if they are to behave identically. Knowing that, it is a straightforward process to find the required values of the other source. The current/voltage characteristic is linear for these circuits, and a straight plot line can be defined by just two points. The two obvious points to use are the opened and shorted load cases. In other words, if the circuit is equivalent for these

two situations, it must work for any load. The shorted load case produces a large load current with zero load voltage, and the opened load case produces a large load voltage with zero load current. It would not make sense if the equivalent source could produce a greater current or voltage under the same extreme conditions as the original.

For example, given a voltage source, the current that can be developed when the load is shorted is $E/Z_{internal}$. Under that same load condition, all of the current from the current source version must be flowing through the load (otherwise the load isn't shorted). Therefore, the value of the equivalent current source must be the current of $E/Z_{internal}$. Note that the resulting source normally will not have the same phase angle as the original source due to the phase angle of the associated impedance.

To continue, if we look at the open load case for the voltage source, the load current would be zero and the load voltage would be the entire source voltage of E. For the current source, the load would also see no current and its voltage would be the voltage appearing across its internal resistance which is $Z_{internal}$ times the current $E/Z_{internal}$, or just E. Thus, the two behave identically at the load limits.

Similarly, if we start with a current source, an open load produces a load voltage of $I \cdot Z_{internal}$. Therefore, the equivalent voltage source must have a value of $I \cdot Z_{internal}$. For the current source, a shorted load would produce a load current equal to the source value, or I. The voltage source version would produce a current of $E/Z_{internal}$, where the value of E was just found to be equal to $I \cdot Z_{internal}$, and thus the load current would be $I \cdot Z_{internal}/Z_{internal}$, or just I. Once again, the two versions behave identically at the load limits.

Changing the source frequency results in different values for both the reactance and the converted source voltage or current, thus the equivalent is valid only for the frequency in question.

To summarize the process of source conversion:

- The internal impedance will be the same for both versions.
- If converting from a voltage source to a current source, the value of the current source will be the short circuit current available from the voltage source (i.e., shorted load case), and is equal to $E/Z_{internal}$.
- If converting from a current source to a voltage source, the value of the voltage source will be the open circuit voltage available from the current source (i.e., opened load case), and is equal to $I \cdot Z_{internal}$.
- The equivalent is unique to the frequency of the source.

If a multi-source is being converted (i.e., voltage sources in series or current sources in parallel), first combine the sources to arrive at the simplest source and then do the conversion. Do not convert the sources first and then try to combine them as you will wind up with series-parallel configurations rather than simple sources.

Judicious use of source conversions can sometimes simplify multi-source circuits by allowing converted sources to be combined, resulting in a single source.

Example 12.2.1

Convert the source of Figure 12.2.6 into its current source equivalent. $E = 2\angle 0^\circ$ volts RMS.

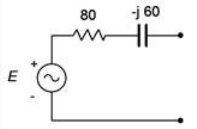


Figure 12.2.6: Circuit for Example 12.2.1.

First, the existing impedance of $80 - j60\Omega$ does not change, it is simply moved to a parallel position. To find the value of the current source, compute the short circuit current the existing source is capable of producing.

$$I = \frac{E}{Z}$$

$$I = \frac{2\angle 0^\circ V}{80 - j60\Omega}$$

$$I = 0.02\angle 36.9^\circ A$$

The result is shown in Figure 12.2.7. The current source is 20 mA RMS with a leading phase angle of 36.9 degrees.

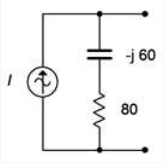


Figure 12.2.7: Current source equivalent of the source of Figure 12.2.6.

Example 12.2.2

Convert the source of Figure 12.2.8 into its voltage source equivalent. $I = 0.1\angle 0^{\circ}$amps peak.

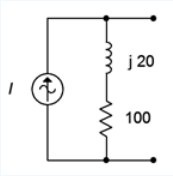


Figure 12.2.8: Circuit for Example 12.2.2.

Again, the internal impedance of $100 + j20\Omega$ does not change and we simply place it in series. To find the value of the voltage source, find the open circuit voltage the existing source is capable of producing.

$$E = I \times Z$$

$$E = 0.1\angle 0^{\circ} A \times (100 + j20\Omega)$$

$$E \approx 10.2\angle 11.3^{\circ} V$$

The equivalent is shown in Figure 12.2.9. The voltage source E is 10.2 volts peak with a leading phase angle of 11.3 degrees.

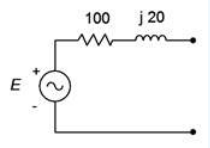


Figure 12.2.9: Voltage source equivalent of the source of Figure 12.2.8.

Computer Simulation

To verify this process, the circuits of Figures 12.2.8 and 12.2.9 are entered into a simulator. The original circuit simply specified an inductive reactance so a convenient frequency of 1 kHz was used and then an appropriate inductor was created that would yield the desired reactance of $j20\Omega$. This turned out to 3.183 mH. Further, a random load resistor (200 Ω) was picked and applied to both circuits. The result is shown in Figure 5.10.

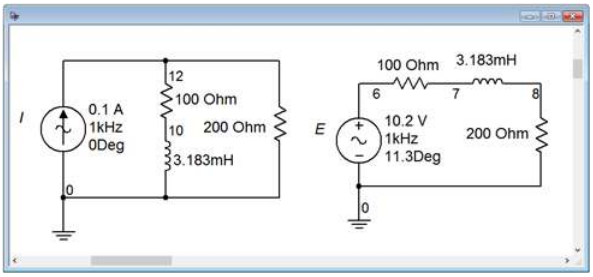


Figure 12.2.10: The circuits of Figures 12.2.8 and 12.2.9 in a simulator.

If these two circuits are equivalent, then the voltages seen across the 200 Ω load resistors should be identical. These voltages correspond to node voltages 12 and 8. A quick voltage divider computation shows that the load voltage should be approximately $6.785\angle 7.5^{\circ}$ volts peak.

https://eng.libretexts.org/@go/page/52969

A transient analysis is performed, plotting the load voltages. The results are shown in Figure 12.2.11. The plot itself is delayed by one millisecond in order to get past the initial turn-on transient. Further, the current source circuit is shifted by approximately 2 microseconds. Without this slight shift in time, the two voltages perfectly overlap so that it appears there is only one trace. By looking carefully at the graph it can be seen that there are indeed two traces. The amplitudes of these waveforms match each other perfectly and also match the computed result.

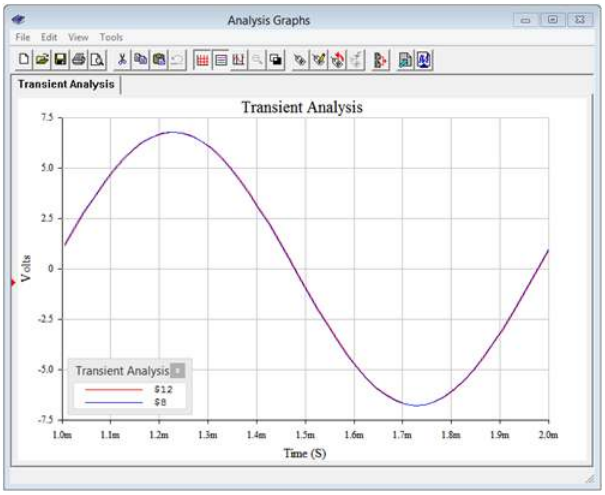


Figure 12.2.11: Simulation results for the source conversion.

Now let's turn our attention to using source conversion to simplify and solve a multisource circuit.

Example 12.2.3

For the circuit of Figure 12.2.12, determine v_b. $I = 2E - 3\angle 90^\circ$ amps peak and $E = 10\angle 0^\circ$ volts peak.

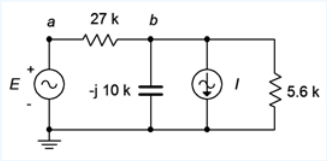


Figure 12.2.12: Circuit for Example 12.2.3.

One method of solution is to transform the voltage source into a current source. By doing so, the entire circuit is reduced to a parallel network. The two current sources can then be summed and the remaining three components can be combined into one equivalent parallel impedance. At that point Ohm's law can be used to find v_b.

The value of the current source is:

$$I_c = \frac{E}{Z}$$

$$I_c = \frac{10\angle 0^\circ V}{27k\Omega}$$

$$I_c = 0.3704E - 3\angle 0^\circ A$$

The converted circuit is shown in Figure 12.2.13 (in blue).

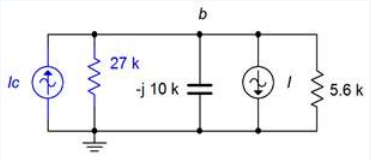


Figure 12.2.13: Circuit of Figure 12.2.12 with converted source.

Combining the two current sources and using an upward reference direction yields:

$$I_{total} = I_c - I$$

$$I_{total} = 0.3704E-3\angle 0^\circ A - 2E-3\angle 90^\circ A$$

$$I_{total} = 2.034E-3\angle -79.5^\circ A$$

The combined impedance is:

$$Z_{total} = \frac{1}{\frac{1}{X_C} + \frac{1}{R_1} + \frac{1}{R_2}}$$

$$Z_{total} = \frac{1}{\frac{1}{-j10k\Omega} + \frac{1}{27k\Omega} + \frac{1}{5.6k\Omega}}$$

$$Z_{total} = 4208\angle -24.9^\circ \Omega$$

And finally we compute v_b:

$$v_b = I_{total} \times Z_{total}$$

$$v_b = 2.034E-3\angle -79.5^\circ A \times 4208\angle -24.9^\circ \Omega$$

$$v_b = 8.56\angle -104.4^\circ V$$

We can perform a KCL crosscheck at node b to verify the results in the original circuit. Given the reference directions, all currents are flowing out of the node except for the current supplied by the voltage source.

$$i_{Esource} = \frac{E - v_b}{R_1}$$

$$i_{Esource} = \frac{10\angle 0^\circ V - 8.56\angle -104.4^\circ V}{27k\Omega}$$

$$i_{Esource} = 0.544E-3\angle 34.4^\circ A$$

The remaining three currents should add up to this value.

$$i_{capacitor} = \frac{v_b}{X_C}$$

$$i_{capacitor} = \frac{8.56\angle -104.4^\circ V}{-j10k\Omega}$$

$$i_{capacitor} = 0.856E-3\angle -14.4^\circ A$$

$$i_{R2} = \frac{v_b}{R_2}$$

$$i_{R2} = \frac{8.56\angle -104.4^\circ V}{5.6k\Omega}$$

$$i_{R2} = 1.53E-3\angle -104.4^\circ A$$

Finally, $0.856E-3\angle -14.4^\circ + 1.53E-3\angle -104.4^\circ + 2E-3\angle 90^\circ$ amps does indeed equal $0.544E-3\angle 34.4^\circ$ amps, within carried rounding error.

References

[1]Such as the one found in Chapter 2 of the companion text, DC Electrical Circuit Analysis, a companion free OER text by the author.

This page titled 12.2: Source Conversions is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.2: Source Conversions** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.3: Superposition Theorem

Superposition allows the analysis of multi-source AC series-parallel circuits. Superposition can only be applied to networks that are linear and bilateral. Fortunately, all of components we have discussed; resistors, capacitors and inductors, fall into that category. Further, superposition cannot be used to find values for non-linear functions, such as power, directly. This is not a limitation though because power can be computed from the resulting voltage or current values.

The basic idea is to determine the contribution of each source by itself, and then combine the results to get the final answer. The contributions are either all voltages or all currents, depending on need. We can state the superposition theorem as:

Theorem 12.3.1: Superposition Theorem

Any voltage or current in a multi-source linear bilateral network may be determined by summing the contributions caused by each source acting alone, with all other source replaced by their internal impedance.

The process generates a series of new single-source circuits, one for each source. These new circuits are then analyzed for the parameter(s) of interest.

Consider the circuit depicted in Figure 12.3.1.

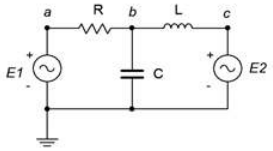


Figure 12.3.1: A basic multi-source circuit.

Here we see two voltage sources, $E1$ and $E2$, driving a three element series-parallel network. As there are two sources, two derived circuits must be created; one using only $E1$ and the other using only $E2$. When considering a given source, all other sources are replaced by their ideal internal impedance: for a voltage source, that's a short; and for a current source, an open. We start by considering $E1$. In the new circuit $E2$ is replaced with a short. This leaves a fairly simple network where X_C and X_L are in parallel. This combination is in series with R and $E1$. Using basic series-parallel techniques, we can solve for desired quantities such as the current flowing through R or the voltage v_b. It is important to indicate the reference current direction and voltage polarity with respect to the source being considered (here, that's left-to-right and positive, respectively). The process is then repeated for $E2$, shorting $E1$ and leaving us with R in parallel with X_C, which is in turn in series with X_L and $E2$. Note that although in this version V_b is still positive, the reference current direction for R is now right-to-left. The numerical results from this version are added to those of the $E1$ version (minding polarities and directions) to achieve the final result. If power is needed, it can be computed from these currents or voltages. Note that superposition can work with a mix of current sources and voltage sources. The practical downside is that for large circuits using many sources, numerous derived circuits will need to be analyzed. For example, if there are three voltage sources and two current sources, then a total of five derived circuits will be created.

It is also possible to use superposition to find the resulting currents or voltages in a circuit that uses sources with different frequencies. In this instance, the equivalent circuits will have different reactance values. In fact, a single non-sinusoidal source can be analyzed using this method by treating the source as a series of superimposed sine waves with each sine source producing a new circuit with its own unique reactance values.

To summarize the superposition technique:

- For every voltage or current source in the original circuit, create a new sub-circuit. The sub-circuits will be identical to the original except that all sources other than the one under consideration will be replaced by their ideal internal impedance. This means that all remaining voltage sources will be shorted and all remaining current sources will be opened.
- Indicate the reference current directions and voltage polarities on each of the new sub-circuits, as generated by the source under consideration.
- Solve each of the sub-circuits for the desired voltages and/or currents using standard series-parallel analysis techniques. Make sure to note the reference voltage polarities and current directions for these items.

- Add all of the contributions from each of the sub-circuits to arrive at the final values, being sure to account for current directions and voltage polarities in the process.

To illustrate the superposition technique, let's reexamine the the dual source circuit shown in Figure 5.2.12 (repeated in Figure 12.3.2 for ease of reference). We will solve this using superposition and compare the results to those of Example 5.2.3 which used source conversion.

Example 12.3.1

For the circuit of Figure 12.3.2, determine v_b using superposition. $I = 2E - 3\angle 90^\circ$ amps peak and $E = 10\angle 0^\circ$ volts peak.

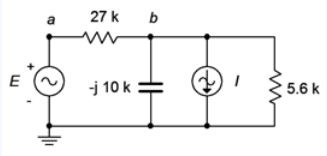


Figure 12.3.2: Circuit for Example 12.3.1.

As the circuit has two sources, it will require two sub-circuits. For the voltage source, the current source will be replaced with an open. For the second circuit utilizing the current source, the voltage source will be replaced with a short. First, using the voltage source we find:

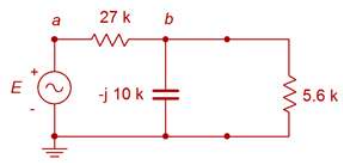


Figure 12.3.3: Circuit of Figure 12.3.2 considering voltage source.

For this circuit, v_b may be determined via a voltage divider. To proceed, we need the impedance of the parallel combo on the right.

$$Z_{right2} = \frac{R \times jX_C}{R - jX_C}$$

$$Z_{right2} = \frac{5.6k\Omega \times (-j10k\Omega)}{5.6k\Omega - j2k\Omega}$$

$$Z_{right2} = 4886\angle -29.2^\circ\Omega$$

Now for the voltage divider to find the contribution of the first source to v_b.

$$v_{b1} = E\frac{Z_{right2}}{Z_{right2} + R_1}$$

$$v_{b1} = 10\angle 0^\circ V\frac{4886\angle -29.2^\circ\Omega}{4886\angle -29.2^\circ\Omega + 27k\Omega}$$

$$v_{b1} = 1.558\angle -24.88^\circ V$$

We turn our attention to the current source's contribution. We short the voltage source and redraw:

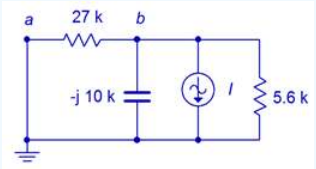


Figure 12.3.4: Circuit of Figure 12.3.4 considering current source.

This is a simple parallel circuit. We can find v_b by placing the resistors and capacitors in parallel, and then using Ohm's law. Note that the reference direction of the current source is downward, meaning that the component current is upward, which makes v_b negative (i.e., + to − bottom to top). The parallel impedance is:

$$Z_{total} = \frac{1}{\frac{1}{X_C} + \frac{1}{R_1} + \frac{1}{R_2}}$$

$$Z_{total} = \frac{1}{\frac{1}{-j10k\Omega} + \frac{1}{27k\Omega} + \frac{1}{5.6k\Omega}}$$

$$Z_{total} = 4208\angle -24.9^\circ\Omega$$

We apply Ohm's law to find this source's contribution to v_b.

$$v_{b2} = I \times Z_{total}$$

$$v_{b2} = -2E - 3\angle 90^\circ A \times 4208\angle -24.9^\circ\Omega$$

$$v_{b2} = 8.416\angle -114.9^\circ V$$

The final result is the sum of the two parts:

$$v_b = v_{b1} + v_{b2}$$

$$v_b = 1.558\angle -24.88^\circ V + 8.416\angle -114.9^\circ V$$

$$v_b = 8.558\angle -104.4^\circ V$$

This is virtually the same value obtained using the source conversion technique in the prior example.

As mentioned previously, superposition can be used to determine the results even when the sources use different frequencies. This will be explored in the next example.

Example 12.3.2

For the circuit of Figure 12.3.5, determine v_b.

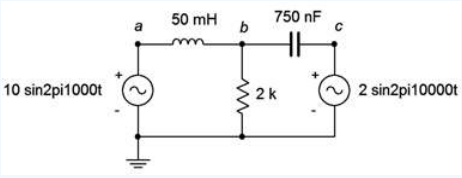


Figure 12.3.5: Circuit for Example 12.3.2.

Using superposition, we derive two new circuits, each with unique reactance values. The first circuit is shown in Figure 12.3.6.

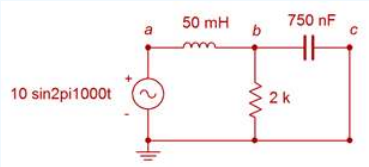


Figure 12.3.6: Circuit for Example 12.3.2, first source only.

The reactance values are:

$$X_L = j2\pi fL$$

$$X_L = j2\pi 1kHz50mH$$

$$X_L \approx j314.2\Omega$$

$$X_C = -j\frac{1}{2\pi fC}$$

$$X_C = -j\frac{1}{2\pi 1kHz750nF}$$

$$X_C \approx -j212.2\Omega$$

A voltage divider can be used to find this portion of v_b. The parallel combo of the 2 kΩ resistor and capacitor is $211\angle -83.9°\Omega$.

$$v_{b1} = E\frac{Z_{right2}}{Z_{right2} + X_L}$$

$$v_{b1} = 10\angle 0°V\frac{211\angle -83.9°\Omega}{211\angle -83.9°\Omega + 314.2\angle 90°\Omega}$$

$$v_{b1} = 19.78\angle -161.9°V$$

Remember, this waveform is at a frequency of 1 kHz. We can repeat this process for the second source which uses 10 kHz. At this new frequency the inductive reactance will be ten times larger, or $j3142\Omega$, and the capacitive reactance will be ten times smaller, or $-j21.22\Omega$. The new circuit is shown in Figure 12.3.7.

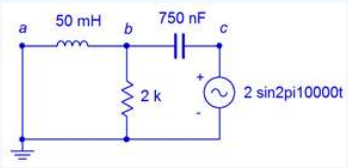


Figure 12.3.7: Circuit for Example 12.3.2, second source only.

Once again, a voltage divider can be used to find this portion of v_b. The parallel combo of the 2 kΩ resistor and inductor is $1687\angle 32.5°\Omega$.

$$v_{b2} = E\frac{Z_{left2}}{Z_{left2} + X_C}$$

$$v_{b2} = 2\angle 0°V\frac{1687\angle 32.5°\Omega}{1687\angle 32.5°\Omega + 21.22\angle -90°\Omega}$$

$$v_{b2} = 2.01\angle 0.6°V$$

This contribution is at a frequency of 10 kHz. Thus, the combination is a relatively small 10 kHz sine at about 2 volts peak riding on a 1 kHz sine that is nearly ten times larger in amplitude. This is shown in Figure 12.3.8.

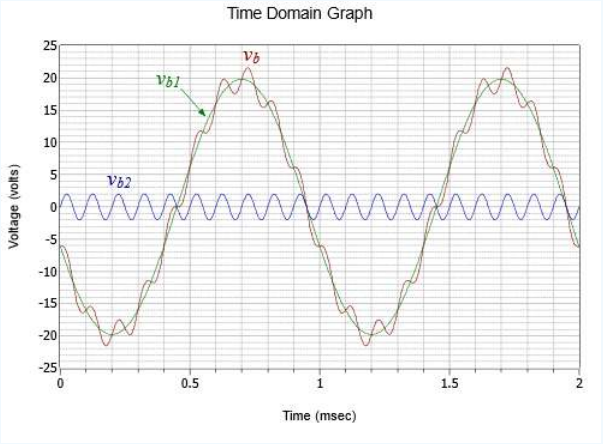


Figure 12.3.8: Voltage plot for Example 12.3.2.

Computer Simulation

In order to verify the two-component waveform of Example 12.3.2, the circuit of Figure 12.3.5 is captured in a simulator as shown in Figure 12.3.9.

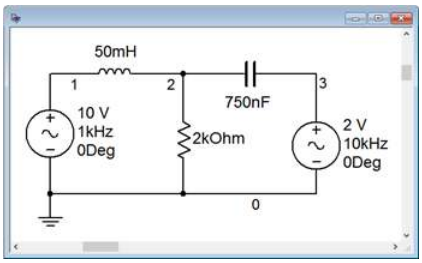


Figure 12.3.9: Circuit of Example 12.3.2 in a simulator.

A transient analysis is performed. The results are illustrated in Figure 12.3.10.

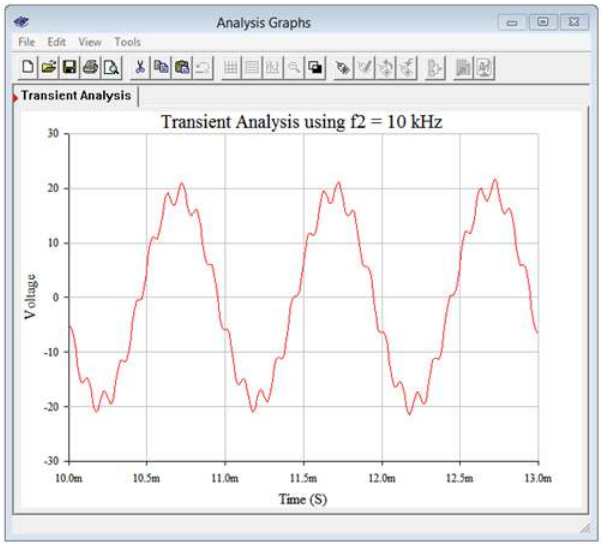


Figure 12.3.10: Simulation results for the circuit of Example 12.3.2.

The results match the computed values nicely. We can see the small amplitude high frequency sine wave effectively following the contour of the much larger 1 kHz sine wave.

This page titled 12.3: Superposition Theorem is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.3: Superposition Theorem** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.4: Thévenin's and Norton's Theorems

These theorems are related in that they allow complex linear networks to be simplified down to a single source with an associated internal impedance. They simplify analysis when checking a circuit with multiple possible loads.

Thévenin's Theorem

Thévenin's theorem is named after Léon Charles Thévenin. It states that:

> Any single port linear network can be reduced to a simple voltage source, E_{th}, in series with an internal impedance Z_{th}.

It is important to note that a Thévenin equivalent is valid only at a particular frequency. If the system frequency is changed, the reactance and impedance values will change and the resulting E_{th} and Z_{th} values will be altered. Consequently, these equivalents are generally not appropriate for a circuit using multiple sources with differing frequencies[1]. A generic example of a Thévenin equivalent is shown in Figure 12.4.1.

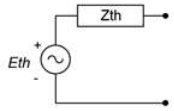


Figure 12.4.1: Generic Thévenin equivalent circuit.

The phrase “single port network” means that the original circuit is cut in such a way that only two connections exist to the remainder of the circuit. That remainder may be a single component or a large multi-component sub-circuit. E_{th} is the open circuit voltage at the port and Z_{th} is the impedance looking back into the port (i.e., the equivalent that now drives the remainder). As there are many ways to cut a typical circuit, there are many possible Thévenin equivalents. Consider the circuit shown in Figure 12.4.2.

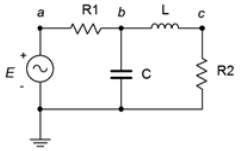


Figure 12.4.2: Circuit under consideration for a Thévenin equivalent.

Suppose we want to find the Thévenin equivalent that drives R_2. We cut the circuit immediately to the left of R_2. That is, The first step is to make the cut, removing the remainder of the circuit. In this case the remainder is just R_2. We then determine the open circuit output voltage at the cut points (i.e., at the open port). This voltage is called the Thévenin voltage, E_{th}. This is shown in Figure 12.4.3. In a circuit such as this, basic series-parallel analysis techniques may be used to find E_{th}. In this circuit, due to the open, no current flows through the inductor, L, and thus no voltage is developed across it. Therefore, E_{th} must equal the voltage developed across the capacitor, C.

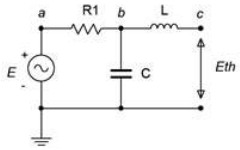


Figure 12.4.3: E_{th},the open circuit output voltage.

The second part is finding the Thévenin impedance, Z_{th}. Beginning with the “cut” circuit, replace all sources with their ideal internal impedance (thus shorting voltage sources and opening current sources). From the perspective of the cut point, look back into the circuit and simplify to determine its equivalent impedance. This is shown in Figure 12.4.4. Looking in from where the cut was made (right side), we see that R_1 and X_C are in parallel, and this combination is then in series with X_L. Thus, Z_{th} is equal to $jX_L + (R1||-jX_C)$.

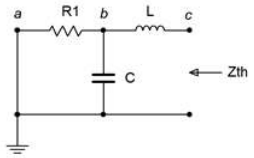


Figure 12.4.4: Finding Z_{th}.

As noted earlier, the original circuit could be cut in a number of different ways. We might, for example, want to determine the Thévenin equivalent that drives C in the original circuit of Figure 12.4.2. The new port location appears in Figure 12.4.5.

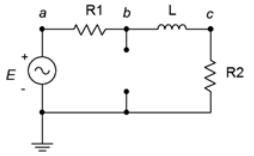


Figure 12.4.5: An alternate port location.

Clearly, this will result in different values for both E_{th} and Z_{th}. For example, Z_{th} is now $R_1||(R_2 + jX_L)$.

A common error is to find Z_{th} from the wrong perspective, namely, finding the impedance that the source drives. This is flatly incorrect. Remember, Z_{th} is found by looking into the port and simplifying whatever is seen from there. One way to remember this is that it is possible to create equivalents for multi-source circuits. In that instance, there isn't a single driving source, so finding its load impedance is nonsensical.

Measuring the Thévenin Equivalent in the Laboratory

In a laboratory situation, the Thévenin equivalent can be found quickly and efficiently with the proper tools. First, the circuit is "cut", leaving just the portion to be Thévenized. The voltage at the cut points is measured with an oscilloscope. This is E_{th}. All of the sources are then replaced with their internal impedance, ideally shorting voltage sources and opening current sources[2]. An LCR impedance meter can then be connected to the port to read Z_{th}. If an impedance meter is not available, then the source(s) are left in place and an LCR substitution box is placed at the cut points. The box is adjusted so that the voltage across it is equal to half of E_{th}. By the voltage divider rule, the value of the substitution box must be equal to Z_{th}. In this case, the substitution box will yield either an inductance or capacitance value which can then be turned into a reactance given the frequency.

Example 12.4.1

For the circuit of Figure 12.4.6, determine the Thévenin equivalent that drives the 300 Ω resistor and find v_c. Assume the source angle is 0°.

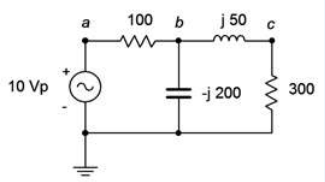


Figure 12.4.6: Circuit for Example 12.4.1.

First, let's find E_{th}, the open circuit output voltage. We cut the circuit so that the 300 Ω resistor is removed. Then we determine the voltage at the cut points. This circuit is shown in Figure 12.4.7.

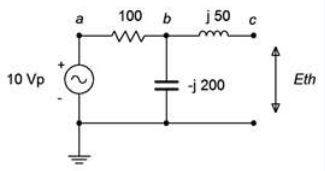


Figure 12.4.7: Circuit for finding E_{th} for Example 12.4.1.

There is no current flowing through the inductor due to the open. Therefore, the voltage across the inductor is zero. Consequently, E_{th} is the voltage across the capacitor, and that can be found with a voltage divider.

$$E_{th} = E\frac{X_C}{X_C + R_1}$$

$$E_{th} = 10\angle 0^\circ V\frac{-j200\Omega}{-j200\Omega + 100\Omega}$$

$$E_{th} = 8.944\angle -26.6^\circ V \text{ or } 8 - j4V$$

To find Z_{th}, we replace the source with a short and then look back in from the cut points. The equivalent circuit is shown in Figure 12.4.8. The inductor is in series with the parallel combination of the resistor and capacitor.

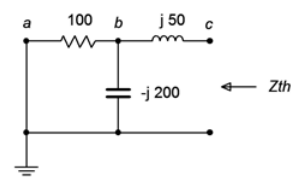


Figure 12.4.8: Circuit for finding Z_{th} for Example 12.4.1.

$$Z_{left2} = \frac{R \times jX_C}{R - jX_C}$$

$$Z_{left2} = \frac{100\Omega \times (-j200\Omega)}{100\Omega - j200\Omega}$$

$$Z_{left2} = 89.44\angle -26.6^\circ\Omega$$

$$Z_{th} = Z_{left2} + X_L$$

$$Z_{th} = 89.44\angle -26.6^\circ\Omega + j50\Omega$$

$$Z_{th} = 80.62\angle 7.12^\circ\Omega or 80 + j10\Omega$$

The completed equivalent is shown in Figure 12.4.9.

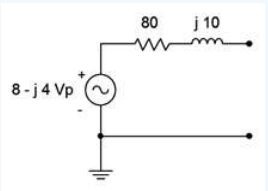


Figure 12.4.9: Completed equivalent for Example 12.4.1.

The voltage across the 300 Ω resistor can be found directly:

$$v_{R2} = E_{th}\frac{R_2}{Z_{th} + R_2}$$

$$v_{R2} = 8 - j4V\frac{300\Omega}{80 + j10\Omega + 300\Omega}$$

$$v_{R2} = 7.06\angle -28.1^\circ V$$

This value can be verified by following a standard series-parallel simplification. For example, the impedance of the three rightmost components $(181.4\angle -53.97^\circ\Omega)$ forms a voltage divider with the 100 Ω resistor and the 10 volt source. This leads to v_b $(7.156\angle -18.61^\circ$ volts). A second divider can then be used between v_b, the inductor and the 300 Ω resistor to find v_c, which is $7.06\angle -28.1^\circ$ volts as expected. The big advantage of using the Thévenin equivalent is that we can easily find v_c for any other value of load because we need only analyze the simpler equivalent circuit rather than the original.

Thévenin's theorem can also be used on multi-source circuits. The technique for finding Z_{th} does not change, however, finding E_{th} is a little more involved, as illustrated in the next example.

Example 12.4.2

Find v_b for the circuit of Figure 12.4.10 using Thévenin's theorem.

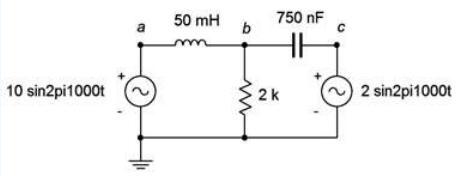


Figure 12.4.10: Circuit for Example 12.4.2.

This circuit is similar to the one used in Example 5.3.2 (Figure 5.3.5). The difference here is that the second source uses the same frequency as the first source. The reactance values are:

$$X_L = j314.2\Omega$$

$$X_C = -j212.2\Omega$$

The voltage across the 2 kΩ is v_b, so we'll treat that resistor as the load in order to define the equivalent circuit. This is redrawn in Figure 12.4.11.

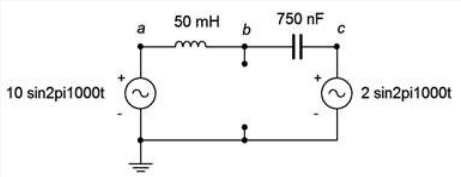


Figure 12.4.11: Circuit for finding equivalents for Figure 12.4.10.

To find Z_{th}, we short the two sources. We're left with the inductor and capacitor in parallel. If this is confusing, remember that we are looking from node b to ground (the cut points), so they are not in series. That is, if a sensing current entered at node b, it could split left and right, indicating parallel paths, not a series connection.

$$Z_{th} = \frac{-jX_C \times jX_L}{-jX_C + jX_L}$$

$$Z_{th} = \frac{-j212.2\Omega \times j314.2\Omega}{-j212.2\Omega + j314.2\Omega}$$

$$Z_{th} = -j653.7\Omega$$

We have a few options to find E_{th}. Superposition could be used, each circuit requiring a voltage divider. Alternately, the equivalent is basically a series loop as far as finding v_b is concerned. Thus, we could find the voltage across the inductor and subtract that from the left source (assuming a reference current direction of clockwise). Finding the inductor voltage requires either a voltage divider or finding the current. Neither approach is considerably less work than the other, and it's probably a good idea to get in a little more practice using superposition, so...

Considering the left source, we short the right source and find v_b.

$$v_{bR} = E_1 \frac{X_C}{X_C + X_L}$$

$$v_{bR} = 10\angle 0^\circ V \frac{-j212.2\Omega}{-j212.2\Omega + j314.2\Omega}$$

$$v_{bR} = 20.84\angle 180^\circ V \text{ or } -20.84\angle 0^\circ V$$

For the right source, we short the left source and find v_b. Then we add the two contributions to find the final voltage. Note that both sources will produce a reference polarity of + to − from top to bottom.

$$v_{bL} = E_2 \frac{X_C}{X_C + X_L}$$

$$v_{bL} = 2\angle 0^\circ V \frac{j314.2\Omega}{-j212.2\Omega + j314.2\Omega}$$

$$v_{bL} = 6.16\angle 0^\circ V$$

The sum of the two is $-20.84\angle 0^\circ + 6.16\angle 0^\circ$, or $14.68\angle 180^\circ$ volts. The Thévenin equivalent is a source of $14.68\angle 180^\circ$ volts in series with an impedance of $-j653.7\Omega$.

To find the voltage across the 2 kΩ resistor, we apply it to the equivalent circuit and solve.

$$v_b = E_{th} \frac{R}{R + Z_{th}}$$

$$v_b = 14.68\angle 180^\circ V \frac{2k\Omega}{2k\Omega + (-j653.7\Omega)}$$

$$v_b = 13.95\angle -161.9^\circ V$$

Computer Simulation

To verify the results of the preceding example, the circuit of Figure 12.4.10 is captured in a simulator, as shown in Figure 12.4.12.

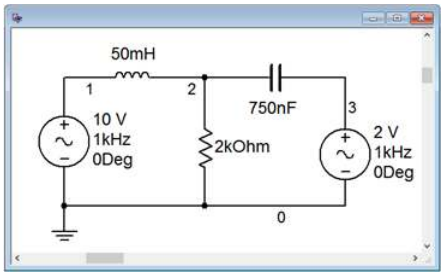


Figure 12.4.12: The circuit of Figure 12.4.10 captured in a simulator.

Next, a transient analysis is run, plotting the voltage at node 2, which corresponds to v_b in the original circuit. The result is shown in Figure 12.4.13. The plot is delayed by 0.1 seconds in order to get past the initial turn-on transient.

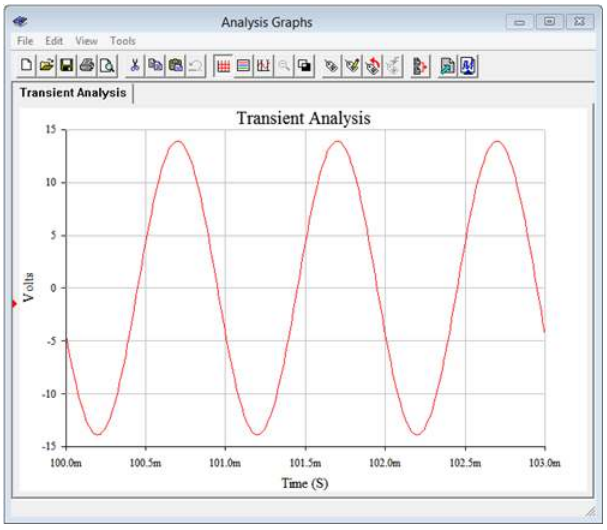


Figure 12.4.13: Simulation results for the circuit of Figure 12.4.10.

Both the amplitude and phase of the simulation waveform match the computed results.

Norton's Theorem

Norton's theorem is named after Edward Lawry Norton. It is the current source version of Thévenin's theorem. In other words, complex networks can be reduced to a single current source with a parallel internal impedance. Formally, Norton's theorem states:

Any single port linear network can be reduced to a simple voltage source, I_n, in parallel with an internal impedance Z_n.

The process of finding a Norton equivalent is very similar to finding a Thévenin equivalent. First, the Norton impedance is the same as the Thévenin impedance. Second, instead of finding the open circuit output voltage, the short circuit output current is found. This is the Norton current. Due to the equivalence afforded by source conversions, if a Thévenin equivalent for a network can be created, then it must be possible to create a Norton equivalent. Indeed, if a Thévenin equivalent is found, a source conversion can be performed on it to yield the Norton equivalent.

Example 12.4.3

Let's reexamine Example 12.4.1, this time creating a Norton equivalent circuit. For convenience, the original circuit of Figure 12.4.6 is repeated in Figure 12.4.14. Once again, the goal will be to determine the equivalent that drives the 300 Ω resistor and to find vc.

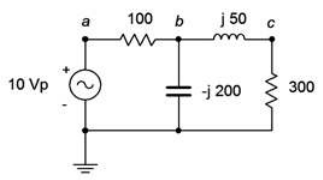


Figure 12.4.14: Circuit for Example 12.4.3.

As noted, the Norton impedance, Z_n, is the same as Z_{th}. That was $80 + j10\Omega$. The Norton current, I_n, is the short-circuit current through the cut points. We can think of this as replacing the load resistor with an ammeter. This is the same as the current through the inductor. In this situation, the capacitor and inductor are in parallel and yield an impedance of $j66.67\Omega$. Thus, the source current is:

$$i_{source} = \frac{E}{R + Z_{LC}}$$

$$i_{source} = \frac{10\angle 0^\circ}{100\Omega + j66.67\Omega}$$

$$i_{source} = 83.2E - 3\angle -33.7^\circ A$$

This splits between the capacitor and inductor. Using the current divider rule we find:

$$i_n = i_{inductor} = I_{source}\frac{X_C}{X_C + X_L}$$

$$i_n = 83.2\angle -33.7^\circ A\frac{-j200\Omega}{-j200\Omega + j50\Omega}$$

$$i_n = 0.1109\angle -33.7^\circ A$$

A source conversion can be applied to verify this value. The resulting voltage source is $8 - j4$ volts, precisely the value of the Thévenin equivalent.

Ultimately, deciding between using the Thévenin or Norton equivalents is a matter of personal taste and convenience. They work equally well.

References

[1]It is possible that an equivalent can be valid across a specified range of frequencies, but it will not hold for all frequencies.

[2]A typical laboratory signal generator has a 50 Ω internal impedance, and using this value would be more accurate than just replacing the source with a shorting wire.

This page titled 12.4: Thévenin's and Norton's Theorems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.4: Thévenin's and Norton's Theorems** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.5: Maximum Power Transfer Theorem

The concept of maximum power transfer in DC resistive circuits was presented in earlier work. While maximizing load power is not a goal of all circuit designs, it is a goal of a portion of them and thus worth a closer look. In short, given an AC voltage source with internal impedance, as seen in Figure 12.5.1, a useful question to ask is "What value of load impedance will yield the maximum amount of power in the load?" In the DC case, it was discovered that the load resistance must equal the source resistance in order to achieve maximum load power. In the AC case things appear to be much more complicated by the possible presence of reactances in both the source and load.

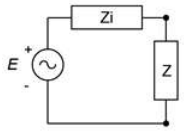


Figure 12.5.1: Defining maximum power transfer.

As a refresher of prior study, consider the basic circuit depicted in Figure 12.5.1 with source E, source internal resistance Z_i and load impedance Z. For the moment, we shall ignore the reactive portions and just describe the load power in terms of the load's resistive portion, R. To make the job easier, we may normalize the source resistance R_i to 1 Ω. By doing this, R also becomes a normalized value, that is, it no longer represents a simple resistance value but rather represents a ratio in comparison to R_i. In this way the analysis will work for any set of source values. Note that the value of E will equally scale the power in both R_i and R, so a precise value is not needed, and thus, we may as well chose 1 volt for convenience.

The power in the load can be determined by using I^2R where $I = E/(R_i + R)$, yielding

$$P = \left(\frac{E}{R_i + R}\right)^2 R$$

Using our normalized values of 1 volt and 1 Ω,

$$P = \left(\frac{1}{1 + R}\right)^2 R$$

After expanding we arrive at:

$$P = \frac{R}{R^2 + 2R + 1} \tag{12.5.1}$$

We now have an equation that describes the load power in terms of the load resistance. Before we go any further, take a look at what this equation tells you, in general. It is obvious that maximum power will not occur at the extremes. If $R = 0$ or $R = \infty$ (i.e., shorted or opened load) the load power is zero. The maximizing case occurs somewhere in the middle. To find the precise value that produces the maximum load power, the proof can be divided into two portions. The first involves graphing the function and the second requires differential calculus to solve for a precise value. We shall proceed with the graphing portion which will lead us to the answer. The more rigorous proof of the second portion is detailed in Appendix C.

The curve of Equation 12.5.1 is plotted in Figure 12.5.2. The normalized load resistance is set along the horizontal and the normalized power (i.e., for a source of 1 volt) is set along the vertical.

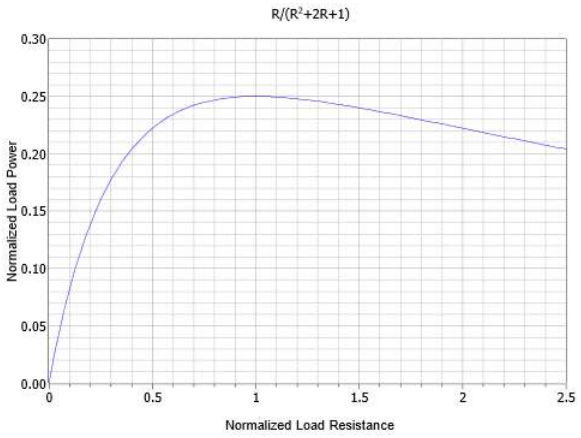


Figure 12.5.2: The resistive portion of the power equation plotted.

An examination of the power curve shows that the peak occurs at $R = 1$. In other words, the load must be equal to the source resistance. Thus, we can say that if no reactances are involved, maximum load power occurs when the load resistance equals the source resistance. It does not matter if the source is DC or AC.

The graph shown in Figure 12.5.2 is asymmetric but the concept of resistance ratios is key here. This is easier to see if we plot the completed power curve using a logarithmic horizontal axis and also scale the vertical axis to 100%, as shown in Figure 12.5.3. The peak is more apparent and the curve is symmetrical in shape rather than lopsided. This reinforces the idea that the ratio of the resistances is what matters.

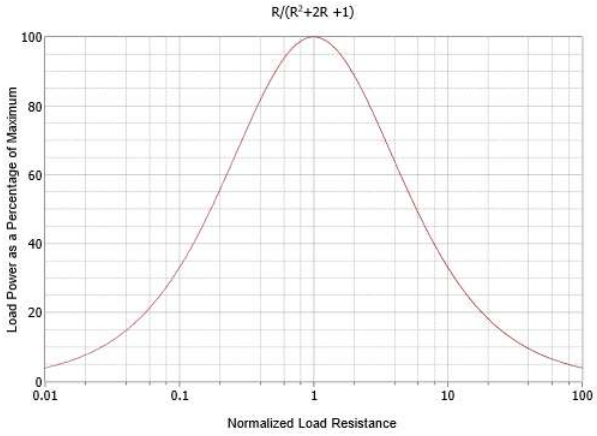


Figure 12.5.3: The load power curve with logarithmic axis showing symmetry.

At this point we may turn our attention to the possible presence of reactances in both the source and load. It turns out that this is not nearly as complicated as it might look. The key is that only resistors dissipate power, not inductors or capacitors[1]. Load power is proportional to i_{load}^2, so our immediate goal is to maximize load current for any set of source and load resistances.

We can modify the original power equation by adding a new term, X, which represents the net reactance in the circuit. In other words, X is equal to the sum of the reactances in the source impedance and load impedance. The power in the load is still determined by using I^2R, however, we must now include the X term when computing the current:

\[I = \frac{E}{\sqrt{((R_i+R)^2+X^2)} \nonumber \]

This leads is to a new load power expression:

\[P = \left(\frac{E}{\sqrt{((R_i+R)^2+X^2) \right)^2 R \label{5.2} \]

A cursory look at Equation ??? shows that to maximize P, X must be zero. A normalized plot of this equation is shown in Figure 12.5.4 for $R = R_i$.

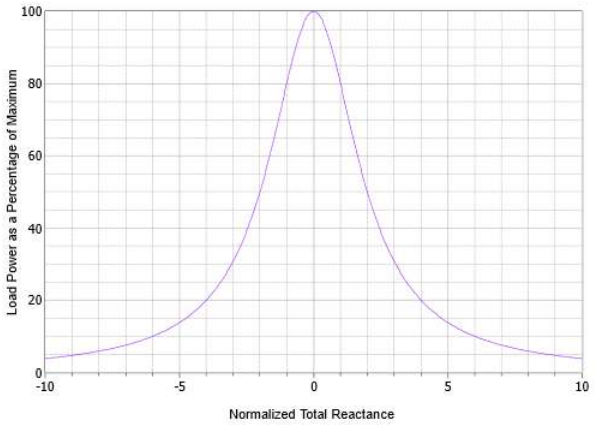


Figure 12.5.4: The reactive portion of the power equation plotted using matched resistance.

A single peak is evident when X is 0. This can be achieved by setting the load reactance equal in magnitude to the source reactance but with the opposite sign. In this manner, the reactances will cancel out, leaving a purely resistive circuit with a minimal value, and thus producing maximal current for that set of resistors.

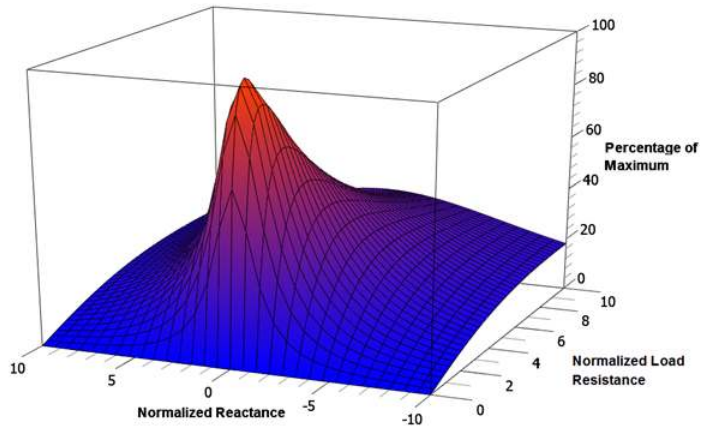


Figure 12.5.5: The load power surface showing variations with both load resistance and total reactance.

Two variables are involved here, so to further clarify the situation, a 3D surface plot of normalized power is shown in Figure 12.5.5. The vertical axis represents the percentage of maximum power while the front and side axes are the normalized total reactance and load resistance, respectively. A single peak is evident here and coincides with $X = 0$ and $R = 1$. This is more easily seen by viewing the surface from the back as shown in Figure 12.5.6. Note that the highest isocontour encircles the intersection of $X = 0$ and $R = 1$ (i.e., \R_{load} = R_i\)).

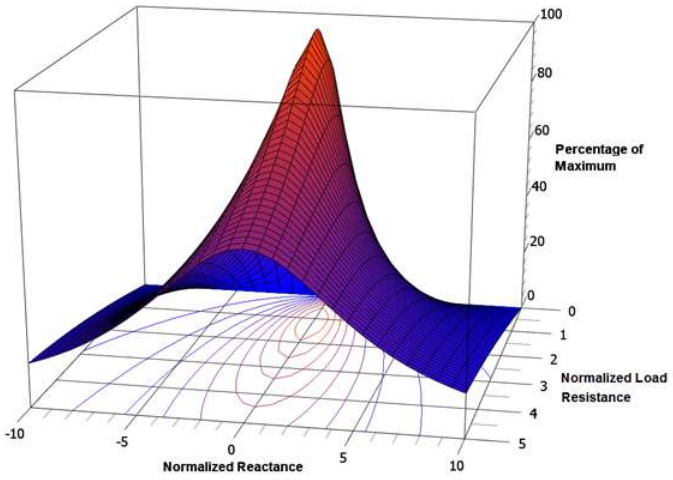


Figure 12.5.6: An alternate view of the load power surface.

In sum, we have verified that the resistive portions of the source and load impedance must be identical and that the reactive portions must be of the same magnitude but of opposite sign. This configuration is also known as the complex conjugate. Finally, we can state:

> Maximum load power will be achieved when the load impedance is equal to the complex conjugate of the internal impedance of the driving source.

No other value of load impedance will produce a higher load power. We can imagine two general cases, one with an inductive source impedance and another with a capacitive source impedance. These are shown with the proper loads in Figure 12.5.7.

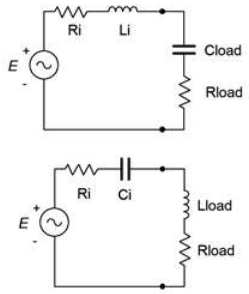


Figure 12.5.7: Configurations for complex conjugate loads.

To achieve maximum load power in these circuits, $R_{load} = R_i$ and $|jX_L| = |-jX_C|$. Note that X_L and X_C do not have to have the same magnitude as R_i.

While using the complex conjugate produces the maximum load power, it does not produce the largest possible load current or load voltage. In fact, this condition produces a load voltage and a load current that are half of their maximums. Their product, however, is at the maximum. Further, efficiency at maximum load power is only 50% (i.e., only half of all generated power goes to the load with the other half being wasted internally). Values of R greater than R_i will achieve higher efficiency but at reduced load power. Sometimes we favor efficiency over maximal load power.

As any linear single port network can be reduced to something like Figure 12.5.7 by using Thévenin's theorem, combining the two theorems allows us to determine maximum power conditions for any impedance in a complex circuit.

Example 12.5.1

Consider the circuit of Figure 12.5.8. What is the power generated in the load if it is equal to 40 Ω? Further, is that the maximum power that can be attained, and if not, what is the maximum load power and what value of load would be needed?

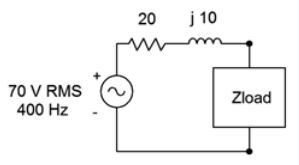


Figure 12.5.8: Circuit for Example 12.5.1.

To find the load power, first find the circulating current, then use power law. The total impedance seen by the source is $20 + j10\Omega + 40\Omega$, or $60 + j10\Omega$.

$$i = \frac{E}{Z_{total}}$$

$$i = \frac{70V}{60\Omega + j10\Omega}$$

$$i = 1.151\angle -9.5^{\circ}A$$

As the voltage and current are in phase for a resistor, we can ignore the angle for the power calculation.

$$P_{load} = i^2 \times R_{load}$$

$$P_{load} = (1.151A)^2 \times 40\Omega$$

$$P_{load} \approx 53W$$

This is not the maximum load power that can be achieved because this load is not the complex conjugate of the source impedance. The required load for maximum load power is shown in Figure 12.5.9.

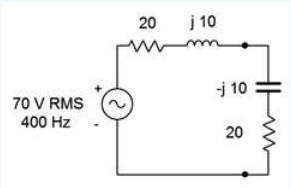


Figure 12.5.9: Circuit of Figure 12.5.8 with proper load configuration.

We shall repeat the process to find the new load power.

$$i = \frac{E}{Z_{total}}$$

$$i = \frac{70V}{40\Omega}$$

$$i = 1.75\angle 0^{\circ}A$$

$$P_{load} = i^2 \times R_{load}$$

$$P_{load} = (1.75A)^2 \times 20\Omega$$

$$P_{load} = 61.25W$$

An alternate method notes that the new circuit's total impedance is purely resistive and that the source and load resistances are identical. Therefore the voltage source must split evenly across them. In this case that's 35 volts RMS each.

$$P_{load} = \frac{v_R^2}{R_{load}}$$

$$P_{load} = \frac{(35V)^2}{20\Omega}$$

$$P_{load} = 61.25W$$

Example 12.5.2

For the circuit of Figure 12.5.10, determine the value of Z_{load} that will achieve maximum load power and also determine that power. The current source is $0.1\angle 0^{\circ}$ amps RMS at 50 kHz.

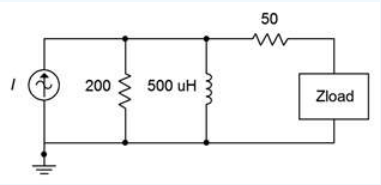


Figure 12.5.10: Circuit for Example 12.5.2.

The first job is determine the inductive reactance at 50 kHz. Recalling that $X_L = j2\pi fL$, this works out to $j157\Omega$. We now need to find the Thévenin equivalent. To find Z_{th} we open the current source and look back in from the load. We see the 50 Ω resistor in series with the parallel combination of the 200 Ω resistor and the inductor. The parallel combination is:

$$Z = \frac{R \times jX_L}{R + jX_L}$$

$$Z = \frac{200\Omega \times (j157\Omega)}{200\Omega + j157\Omega}$$

$$Z = 76.3 + j97.2\Omega$$

Therefore, $Z_{th} = 126.3 + j97.2\Omega$. The complex conjugate is $126.3 - j97.2\Omega$. The capacitive reactance formula may be used to determine the appropriate capacitance value to achieve $-j97.2\Omega$.

$$C = \frac{1}{2\pi f X_C}$$

$$C = \frac{1}{2\pi 50kHz97.2\Omega}$$

$$C = 32.8nF$$

The resulting circuit is shown in Figure 12.5.11.

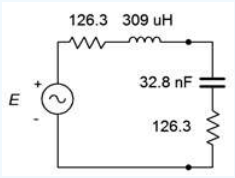


Figure 12.5.11: Thévenin equivalent of the circuit of Figure 12.5.10 with appropriate load.

To find the load power we need to find E_{th}. The open circuit output voltage is the potential appearing across the inductor/resistor pair in Figure 12.5.10. This is because there is no current flowing through the 50 Ω resistor, and therefore there is no voltage across it. E_{th} can be found via Ohm's law as we already know the impedance of the parallel branch.

$$E_{th} = i \times Z$$

$$E_{th} = 0.1\angle 0^{\circ} A \times (76.3 + j97.2\Omega)$$

$$E_{th} = 12.4\angle 51.9^{\circ} V$$

Again, using the complex conjugate, the voltage source splits evenly between the resistive components. As the current source was specified as RMS, so too will be the equivalent voltage.

$$P_{load} = \frac{v_R^2}{R_{load}}$$

$$P_{load} = \frac{(6.2V)^2}{126.3\Omega}$$

$$P_{load} = 304.4mW$$

This represents the maximum load power that can be achieved in this circuit. Do not forgot, though, that an equal amount of power is dissipated by the source. This produces an efficiency of just 50%.

In summation, we can say that maximum load power is achieved when the load impedance is equal to the complex conjugate of the internal impedance of the circuitry driving the load. Usually, this requires the application of either a Thévenin or Norton equivalent. Finally, although maximum power transfer is a desired outcome in some situations, it is not desirable in all situations. The reason is one of efficiency. At the maximum load power, efficiency is only 50%. In contrast, for load impedances that are greater than the source impedance, the load power will decrease, however, the efficiency will increase. Increased efficiency is particularly important when striving to minimize heat and extend battery life.

References

[1]Power in AC circuits is examined in great detail in Chapter 7.

This page titled 12.5: Maximum Power Transfer Theorem is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.5: Maximum Power Transfer Theorem** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.6: Advanced Topic- Delta-Y (Pi-T) Conversions

Certain component configurations, such as bridged networks, cannot be reduced to a single impedance using basic series-parallel conversion techniques. One method for simplification involves converting sections into more convenient forms. The configurations in question are networks with three external connection points. Due to the manner in which they drawn, they are referred to as delta networks and Y networks[1]. These configurations are shown in Figure 12.6.1. Note that the terminal designation of the delta version are upside down compared to those of the Y configuration.

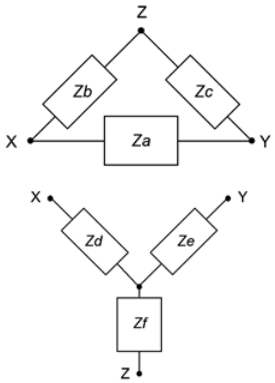


Figure 12.6.1: Delta and Y (Δ-Y) networks.

These networks can be redrawn without angles. In this form they are known as pi (also called "π") networks and T (also called "tee") networks. These configurations are shown in Figure 12.6.2.

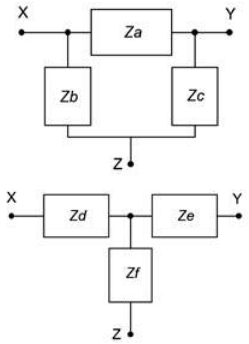


Figure 12.6.2: Alternate form: Pi and T (π-T) networks.

It is possible to convert back and forth between delta and Y networks in many cases. That is, for a given delta network, there may exist a Y network such that the impedances seen between the X, Y and Z terminals are identical, and vice versa. Consequently, one configuration can replace another in order to simplify a larger circuit. Unlike the DC version, certain AC networks cannot be converted using the following technique (see the final Challenge problem for an investigation of this).

Δ-Y Conversion

A true equivalent circuit would present the same impedance between any two terminals as the original circuit. Consider the circuits of Figure 12.6.1 for the unloaded case (i.e., just these networks with nothing else connected to them). The equivalent impedances seen between each pair of terminals for the delta and the Y respectively are:

$$Z_{XY} = Z_a||(Z_b + Z_c) = Z_d + Z_e \tag{12.6.1}$$

$$Z_{XZ} = Z_b||(Z_a + Z_c) = Z_d + Z_f \tag{12.6.2}$$

$$Z_{ZY} = Z_c||(Z_b + Z_a) = Z_e + Z_f \tag{12.6.3}$$

Let's assume that we have the delta network and are looking for the Y network equivalent. We start by focusing on the final set of terms for each of the three expressions (e.g., $Z_{XY} = Z_d + Z_e$). Note that we have three equations with three unknowns (Z_d, Z_e and Z_f). Thus, they can be solved using a term elimination process. If we subtract Equation 12.6.3 from Equation 12.6.1, we can

eliminate the second impedance (Z_e) and arrive at a difference between the first and third unknown impedance s ($Z_d - Z_f$). This quantity can then be added to Equation 12.6.2 to eliminate the third impedance (Z_f), leaving just the first unknown impedance (Z_d).

$$(Z_d + Z_e) - (Z_e + Z_f) = (Z_d - Z_f) = Z_a||(Z_b + Z_c) - Z_b||(Z_a + Z_c)$$

$$(Z_d + Z_f) + (Z - d - Z_f) = 2Z_d = 2(Z_b||(Z_a + Z_c) + Z_a||(Z_b + Z_c) - Z_c||(Z_a + Z_b))$$

Therefore,

$$Z_d = Z_b||(Z_a + Z_c) + Z_a||(Z_b + Z_c) - Z_c||(Z_a + Z_b)$$

which, after simplifying[2], is:

$$Z_d = \frac{Z_a Z_b}{Z_a + Z_b + Z_c} \tag{12.6.4}$$

Similarly, we can show that

$$Z_e = \frac{Z_a Z_c}{Z_a + Z_b + Z_c} \tag{12.6.5}$$

$$Z_f = \frac{Z_b Z_c}{Z_a + Z_b + Z_c} \tag{12.6.6}$$

Note that if the magnitudes and angles of three original impedances are identical, the magnitudes of the Y equivalent impedances will all be one-third of the original magnitude, and with the original phase angle.

Y-Δ Conversion

For the reverse process of converting from Y to delta, start by noting the similarities of the expressions for Z_d, Z_e and Z_f (i.e., Equations 12.6.4 through 12.6.6). If two of these expressions are divided, a single equation for Z_a, Z_b or Z_c will result. For example, using Equations 12.6.4 and 12.6.5:

$$\frac{Z_d}{Z_e} = \frac{\frac{Z_a Z_b}{Z_a + Z_b + Z_c}}{\frac{Z_a Z_c}{Z_a + Z_b + Z_c}}$$

$$\frac{Z_d}{Z_e} = \frac{Z_a}{Z_b}\frac{Z_a}{Z_c}$$

$$\frac{Z_d}{Z_e} = \frac{Z_b}{Z_c}$$

Therefore,

$$\frac{Z_b}{Z_c} = \frac{Z_d}{Z_e}$$

$$Z_b = \frac{Z_c Z_d}{Z_e}$$

This process can be repeated for Equations 12.6.4 and 12.6.6 to obtain an expression for Z_a. The two expressions for Z_a and Z_b can then be substituted into Equation 12.6.4 to obtain an expression for Z_c that utilizes only Z_d, Z_e and Z_f. A similar process is followed for Z_a and Z_b resulting in:

$$Z_a = \frac{Z_d Z_e + Z_e Z_f + Z_d Z_f}{Z_f} \tag{12.6.7}$$

$$Z_b = \frac{Z_d Z_e + Z_e Z_f + Z_d Z_f}{Z_e} \tag{12.6.8}$$

$$Z_c = \frac{Z_d Z_e + Z_e Z_f + Z_d Z_f}{Z_d} \tag{12.6.9}$$

If the Y network consists of three identical impedances, then the values of the delta equivalent will all be three times the original magnitude, the inverse of the situation when converting from delta to Y.

In summation, equations 12.6.4, 12.6.5 and 12.6.6 can be used to convert a delta network into a Y network, and equations 12.6.7, 12.6.8 and 12.6.9 can be used to convert a Y network into a delta network. Examples of how to apply this technique to tame up-to-now intractable series-parallel networks follow.

Example 12.6.1

Convert the network of Figure 12.6.3 into its delta configuration equivalent.

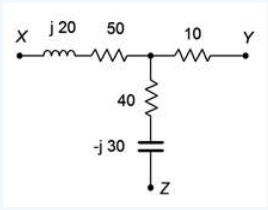


Figure 12.6.3: Network for Example 12.6.1.

Referring back to Figure 12.6.2, use Equation 12.6.7 to determine Z_a.

$$Z_a = \frac{Z_d Z_e + Z_e Z_f + Z_d Z_f}{Z_f}$$

$$Z_a = \frac{(50 + j20\Omega)(10\Omega) + (10\Omega)(40 - j30\Omega) + (50 + j20\Omega)(40 - j30\Omega)}{(40 - j30\Omega)}$$

$$Z_a = 65.6 + j29.2\Omega$$

Z_b and Z_c may be determined in similar manner using Equations 12.6.8 and 12.6.9:

$$Z_b = 350 - j80\Omega$$

$$Z_c = 54.8 - j37.9\Omega$$

The equivalent is shown in Figure 12.6.4.

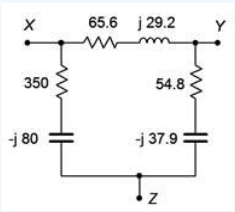


Figure 12.6.4: Equivalent delta network for the Y network of Figure 12.6.3.

Remember, a complex impedance can always be expressed in rectangular form. Rectangular form can be expressed directly as a series combination of a resistor and either an inductive or capacitive reactance. Even if the original impedances of a network are in a parallel form (or even a more complex form), the equivalent can be expressed as a series combination.

Example 12.6.2

Determine v_a in the circuit of Figure 12.6.5. Assume the source has a phase angle of zero degrees.

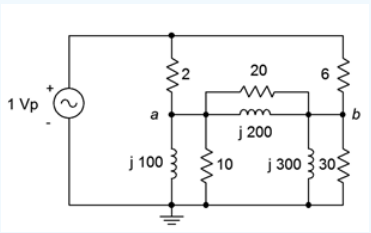


Figure 12.6.5: Circuit for Example 12.6.2.

This circuit cannot be simplified sufficiently using basic series-parallel techniques due to the bridge section. The components between and below nodes a and b comprise a delta network, as shown in Figure 12.6.6. If this network is replaced with a Y equivalent, the resulting circuit reduces to a simple series-parallel system.

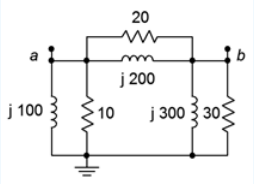


Figure 12.6.6: Delta network within the circuit of Figure 12.6.5.

Before continuing, it would be helpful to determine the impedance of each of the parallel sections. For the leftmost pair:

$$Z_{left2} = \frac{R \times jX_L}{R + jX_L}$$

$$Z_{left2} = \frac{10\Omega \times j100\Omega}{10\Omega + j100\Omega}$$

$$Z_{left2} = 9.9 + j0.99\Omega$$

In similar manner, the top pair is determined to be $19.8 + j1.98\Omega$ and the rightmost pair is $29.7 + j2.97\Omega$. Referring back to Figure 12.6.2, we can use Equation 12.6.4 to determine Z_d.

$$Z_d = \frac{Z_a Z_b}{Z_a + Z_b + Z_c}$$

$$Z_d = \frac{(19.8 + j1.98\Omega) \times (9.9 + j0.99\Omega)}{(19.8 + j1.98\Omega) + (9.9 + j0.99\Omega) + (29.7 + j2.97\Omega)}$$

$$Z_d = 3.3 + j0.33\Omega$$

Likewise, we can use Equations 12.6.5 and 12.6.6 to determine Z_e and Z_f.

$$Z_e = 9.9 + j0.99\Omega$$

$$Z_f = 4.95 + j0.495\Omega$$

Swapping the equivalent Y network into the original circuit leads us to the circuit of Figure 12.6.7 (Y network shown in blue). This circuit can be simplified directly to find v_a.

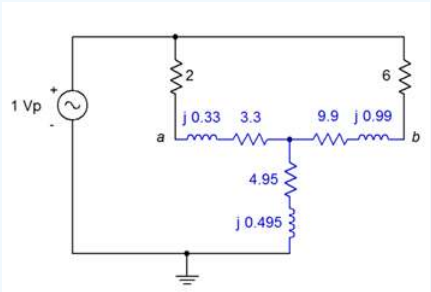


Figure 12.6.7: Equivalent circuit for the circuit of Figure 12.6.5.

In this equivalent circuit, v_a is simply the source voltage of $1\angle 0^\circ$ minus the voltage across the 2 Ω resistor. The immediate goal, then, is to find the current through that resistor. This can be achieved via a current divider once the source current is known. To find the source current, we need to find the total impedance of the network. On the upper left side, Z_d is in series with the 2 Ω resistor for a total of $5.3 + j0.33\Omega$. This is in parallel with the upper right side total of $15.9 + j0.99\Omega$.

$$Z_{upper} = \frac{Z_{upperleft} \times Z_{upperright}}{Z_{upperleft} + Z_{upperright}}$$

$$Z_{upper} = \frac{(5.3 + j0.33\Omega) \times (15.9 + j0.99\Omega)}{(5.3 + j0.33\Omega) + (15.9 + j0.99\Omega)}$$

$$Z_{upper} = 3.975 + j0.2475\Omega$$

This is in series with the lower section of $4.95 + j0.495\Omega$ for a total of $8.956\angle 4.76^\circ\Omega$. Using Ohm's law, we find the source current:

$$i_{source} = \frac{E}{Z_{total}}$$

$$i_{source} = \frac{1\angle 0^\circ V}{8.956\angle 4.76^\circ\Omega}$$

$$i_{source} = 0.1117\angle -4.76^\circ A$$

Now for the current divider and also Ohm's law for the 2 Ω resistor.

$$i_{2\Omega} = i_{source}\frac{Z_{upperright}}{Z_{upperright} + Z_{upperleft}}$$

$$i_{2\Omega} = 0.1117\angle -4.76^\circ A\frac{15.9 + j0.99\Omega}{(15.9 + j0.99\Omega) + (5.3 + j0.33\Omega)}$$

$$i_{2\Omega} = 83.7E - 3\angle -4.76^\circ A$$

$$v_{2\Omega} = i2\Omega \times R$$

$$v_{2\Omega} = 83.7E - 3\angle -4.76^\circ A \times 2\Omega$$

$$v_{2\Omega} = 0.1675\angle -4.76^\circ V$$

Finally, we subtract that potential from the source to find v_a.

$$v_a = E - v_{2\Omega}$$

$$v_a = 1\angle 0^\circ V - 0.1675\angle -4.76^\circ V$$

$$v_a = 0.833\angle 0.95^\circ V$$

References

[1]In some sources the capital Greek letter delta (Δ) is used instead of spelling out “delta” and the letter Y is spelled out as “wye”. Thus, you may come across discussion of “Δ-Y ”, “Δ-wye” or “delta-wye” networks. It's all the same stuff.

[2]This process, though not particularly difficult, is somewhat tedious. It is, as they say, “left as an exercise for the student”.

This page titled 12.6: Advanced Topic- Delta-Y (Pi-T) Conversions is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.6: Delta-Y (Pi-T) Conversions** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.7: Summary

In this chapter we have examined several techniques and theorems to assist with the analysis of AC electrical circuits. We began with more practical models for voltage and current sources by adding an internal impedance to set limits on the source's maximum output and make it sensitive to output frequency. For a voltage source, this impedance is in series, its ideal value being a short, just as it was for the DC case. For current sources, the impedance is in parallel, its ideal value being an open.

Source conversions allow us to create an equivalent voltage source for any practical current source and vice versa. An equivalent source is one that will create the same voltage across (and current into) the remaining circuit as did the original source. In some cases, this swap allows multiple sources to be combined into a single source, simplifying analysis. If the associated impedance does not have a zero degree phase angle, then the converted source will not be in phase with the original, but will instead be shifted by the impedance angle.

The superposition theorem states that, for any multi-source linear bilateral network, the contributions of each source may be determined independent of all other sources, the final result being the summation of the contributions. This remains true in the AC case, however, care must be taken regarding phase shifts when combining the various contributions. The original circuit of N sources generates N new circuits, one for each source under consideration and with all other sources replaced by their ideal internal impedance.

Thévenin's and Norton's theorems allow the simplification of complex linear single port (i.e., two connecting points) networks. The AC Thévenin equivalent consists of a voltage source with a series impedance while the AC Norton equivalent consists of a current source with a parallel impedance. These impedances can be represented in general as a resistance in series with a reactance, and given an operating frequency, the reactance can be turned into a capacitance or inductance. These equivalents, when replacing the original sub-circuit, will create the same voltage across the remainder of the circuit with the same current draw. In other words, the remainder of the circuit will see no difference between being driven by the original sub-circuit or by either the Thévenin or Norton equivalents.

The maximum power transfer theorem states that for a simple voltage source with an internal impedance driving a simple load, the maximum load power will be achieved when the load impedance equals the complex conjugate of the internal impedance. The complex conjugate has the same real or resistive value, however, the reactive portion is of the opposite sign. This results in a cancellation of the reactive components, leaving just the resistive portions and maximizing load current. At this point, efficiency will be 50%. If the load impedance is higher than the internal impedance, the load power will not be as great, however, the system efficiency may improve, depending on the phase angle.

Delta-Y conversions allow the generation of equivalent “three connection point” impedance networks. RLC networks with three elements in the shape of a triangle or delta (with one connection point at each corner) may be converted into a three element network in the shape of a Y or T, or vice versa. The two versions will behave identically to the remainder of the circuit. This allows the simplification of some circuits and eases analysis.

Review Questions

1. What are the ideal internal impedances of AC voltage and current sources?

2. Outline the process of converting an AC voltage source into an AC current source, and vice versa.

3. In general, describe the process of using superposition to analyze a multisource circuit.

4. What do Thévenin's and Norton's theorems state? How are they related?

5. What are the conditions to achieve maximum power transfer for AC circuits? How does this differ from the DC version?

6. What are delta and Y configurations? How are they related?

This page titled 12.7: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.7: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

12.8: Exercises

Analysis

1. For the circuit shown in Figure 12.8.1, use superposition to find v_b.

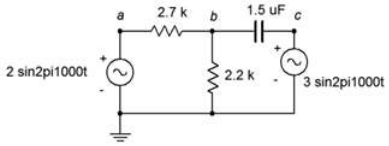


Figure 12.8.1

2. For the circuit shown in Figure 12.8.1, use superposition to find the current through the capacitor.

3. Use superposition to find the current through the 82 Ω resistor. For the circuit shown in Figure 12.8.2.

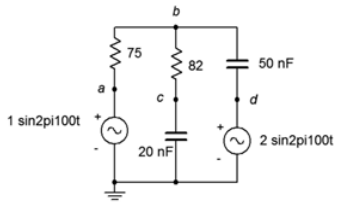


Figure 12.8.2

4. Use superposition to find v_b and v_{cd} for the circuit shown in Figure 12.8.2.

5. In the circuit of Figure 12.8.3, use superposition to find v_b. Source one is one volt peak and source two is two volts peak.

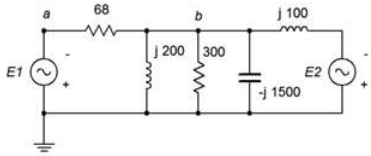


Figure 12.8.3

6. In the circuit of Figure 12.8.3, use superposition to find the currents through the two inductors. Source one is two volts peak and source two is three volts peak.

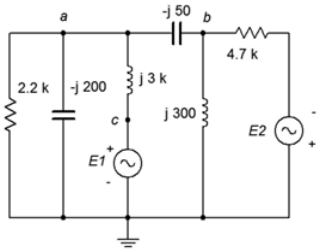


Figure 12.8.4

7. Use superposition to find the current through the 2.2 kΩ resistor for the circuit of Figure 12.8.4. $E1 = 1\angle 0^\circ$ and $E2 = 10\angle 90^\circ$.

8. Use superposition to find v_{ab} for In the circuit of Figure 12.8.4. $E1 = 1\angle 0^\circ$ and $E2 = 2\angle 45^\circ$.

9. In the circuit of Figure 12.8.5, use superposition to find v_b and v_{cd}. The sources are in phase.

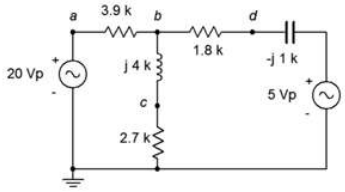


Figure 12.8.5

10. In the circuit of Figure 12.8.5, use superposition to find the current through the capacitor. The sources are in phase.

11. Use superposition to find the two source currents for In the circuit of Figure 12.8.6. Source one is 100 mV peak and source two is 500 mV peak (in phase).

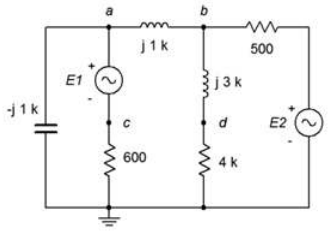


Figure 12.8.6

12. Use superposition to find v_{cd} for the circuit of Figure 12.8.6. Source one is 100 mV peak and source two is 1 V peak (in phase).

13. In the circuit of Figure 12.8.7, use superposition to find v_{ab}.

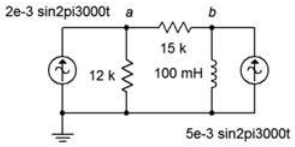


Figure 12.8.7

14. In the circuit of Figure 12.8.7, use superposition to find the current through the 15 kΩ resistor.

15. In the circuit of Figure 12.8.8, use superposition to find v_{ab}.

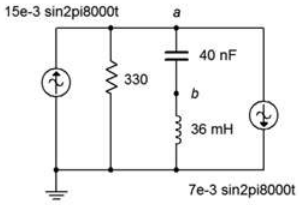


Figure 12.8.8

16. In the circuit of Figure 12.8.8, use superposition to find the current flowing through the resistor.

17. For the circuit of Figure 12.8.9, use superposition to find v_a and v_b. The sources are in phase.

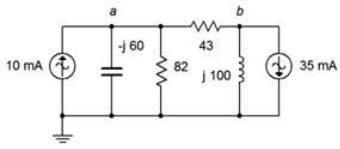


Figure 12.8.9

18. For the circuit of Figure 12.8.9, use superposition to find the currents through the inductor and capacitor. The sources are in phase.

19. Use superposition in the circuit of Figure 12.8.10 to find the currents through the inductor and capacitor. $I1 = 1\angle 45^\circ$ and $I2 = 2\angle 45^\circ$.

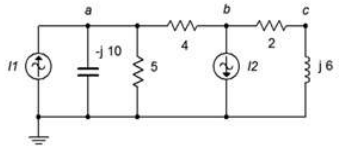


Figure 12.8.10

20. Use superposition in the circuit of Figure 12.8.10 to find v_{ab} and v_{bc}. $I1 = 1\angle 0^\circ$ and $I2 = 2\angle 90^\circ$.

21. In the circuit of Figure 12.8.11, Use superposition to find v_{bc}. $I1 = 10\angle 0^\circ$ and $I2 = 6\angle 0^\circ$.

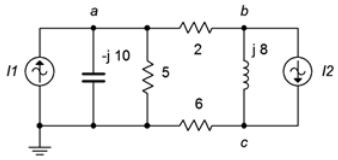


Figure 12.8.11

22. In the circuit of Figure 12.8.11, Use superposition to find the current flowing through the 2 Ω resistor. $I1 = 4\angle 120^\circ$ and $I2 = 6\angle 0^\circ$.

23. Use superposition to determine the current of source E in the circuit of Figure 12.8.12. $E = 40\angle 180^\circ$ and $I = 20E - 3\angle 0^\circ$.

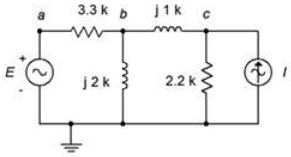


Figure 12.8.12

24. Use superposition to determine v_{ac} in the circuit of Figure 12.8.12. $E = 28\angle 0^\circ$ and $I = 8E - 3\angle -180^\circ$.

25. Use superposition to determine vb in the circuit of Figure 12.8.13. $I = 3E - 3\angle 0^\circ$ and $E = 9\angle 0^\circ$.

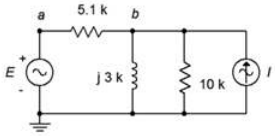


Figure 12.8.13

26. Use superposition to determine the inductor current in the circuit of Figure 12.8.13. $I = 4E - 3\angle 0^\circ$ and $E = 18\angle -45^\circ$.

27. For the circuit of Figure 12.8.14, use superposition to determine the inductor current. $I = 100E - 3\angle 0^\circ$ and $E = 26\angle 0^\circ$.

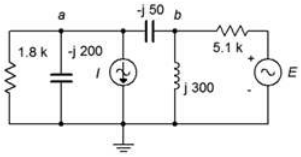


Figure 12.8.14

28. For the circuit of Figure 12.8.14, use superposition to determine v_{ab}. $I = 50E - 3\angle 0^\circ$ and $E = 18\angle 90^\circ$.

29. Use superposition to determine v_{ab} in the circuit of Figure 12.8.15. $I = 10E - 3\angle 0^\circ$ and $E = 12\angle 0^\circ$.

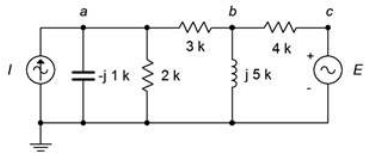


Figure 12.8.15

30. Use superposition to determine the capacitor current in the circuit of Figure 12.8.15. $I = 5E - 3\angle 0^\circ$ and $E = 18\angle 120^\circ$.

31. For the circuit of Figure 12.8.16, determine the Thévenin equivalent that drives the 20 nF capacitor.

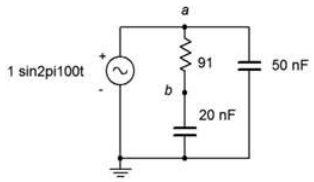


Figure 12.8.16

32. Given the circuit of Figure 12.8.16, determine the Norton equivalent that drives the 20 nF capacitor.

33. For the circuit of Figure 12.8.17, determine the Thévenin and Norton equivalents that drive the 600 Ω resistor if the source $E = 12\angle 0^\circ$.

34. Given the circuit of Figure 12.8.17, determine the Thévenin equivalent that drives the $j1$ kΩ inductive reactance if $E = 9\angle 0^\circ$.

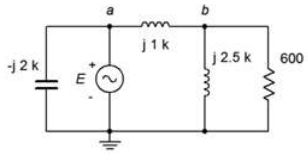


Figure 12.8.17

35. Given the circuit of Figure 12.8.17, determine the Norton equivalent that drives the $j2.5$ kΩ inductive reactance if $E = 24\angle 45^\circ$.

36. Use Thévenin's theorem to find v_b in the circuit of Figure 12.8.17 if $E = 18\angle 0^\circ$.

37. Use Thévenin's theorem to find v_b in the circuit of Figure 12.8.18.

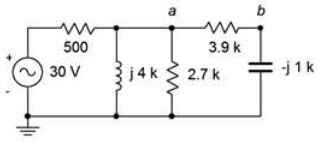


Figure 12.8.18

38. Determine the Thévenin equivalent that drives the $3.9k\Omega + j1k\Omega$ combo in the circuit of Figure 12.8.18. Does this combo's impedance achieve maximum load power? If not, what combo will achieve maximum power and what is the resulting power?

39. Determine the Norton equivalent that drives the 500 Ω resistor in the circuit of Figure 12.8.18. Determine the value of component(s) that when placed in series with the 500 Ω resistor will achieve maximum load power (i.e., for the combo as the load).

40. For the circuit of Figure 12.8.19, determine the Thévenin and Norton equivalents that drive the combo of $36\Omega + j100\Omega$. Does this combo achieve maximum load power? If not, what combo will achieve maximum power and what is the resulting power?

https://eng.libretexts.org/@go/page/52975

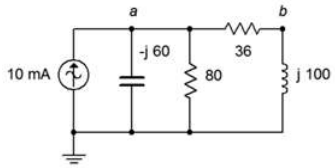


Figure 12.8.19

41. For the circuit of Figure 12.8.20, determine the Thévenin and Norton equivalents that drive the combo of 300 Ω in parallel with $-j1500$ Ω. Does this combo achieve maximum load power? If not, what combo will achieve maximum power and what is the resulting power? $E = 120\angle 0°$.

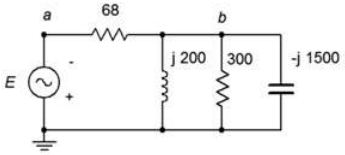


Figure 12.8.20

42. For the circuit of Figure 12.8.21, determine the Thévenin and Norton equivalents that drive the combo of 4.7 kΩ in parallel with $j300$ Ω. Does this combo achieve maximum load power? If not, what combo will achieve maximum power and what is the resulting power? $I = 200E - 3\angle 0°$.

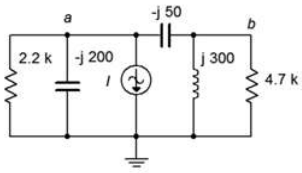


Figure 12.8.21

43. Determine the equivalent Y (T) network for the circuit of Figure 12.8.22. $R_1 = R_2 = R_3 = 10k\Omega$ and $X_{L1} = X_{L2} = X_{L3} = j10k\Omega$.

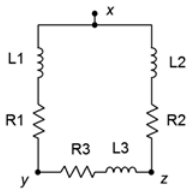


Figure 12.8.22

44. Determine the equivalent Y (T) network for the circuit of Figure 12.8.23.

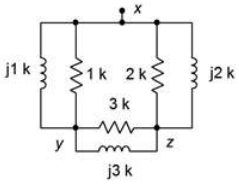


Figure 12.8.23

45. Determine the equivalent Y (T) network for the circuit of Figure 12.8.24. $R_1 = R_2 = R_3 = 4k\Omega$ and $X_{C1} = X_{C2} = X_{C3} = -j3k\Omega$.

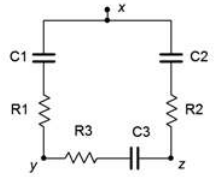


Figure 12.8.24

46. Determine the equivalent Y (T) network for the circuit of Figure 12.8.25.

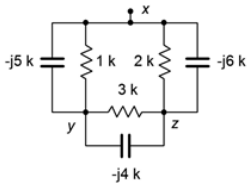


Figure 12.8.25

47. Determine the equivalent delta (pi) network for the circuit of Figure 12.8.26.

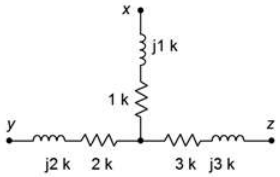


Figure 12.8.26

48. Determine the equivalent delta (pi) network for the circuit of Figure 12.8.27.

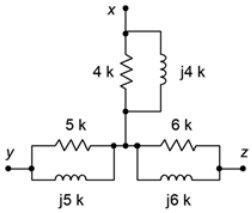


Figure 12.8.27

49. Determine the equivalent delta (pi) network for the circuit of Figure 12.8.28.

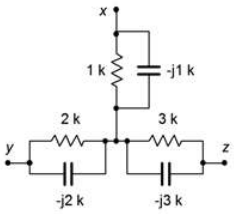


Figure 12.8.28

50. Determine the equivalent delta (pi) network for the circuit of Figure 12.8.29.

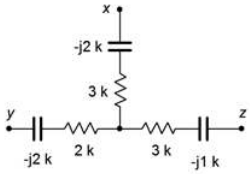


Figure 12.8.29

51. Find voltage v_{bc} in the circuit of Figure 12.8.30 through the use of one or more delta-Y conversions. $E = 10\angle 0°$, $R_1 = 1k\Omega$, $R_2 = 2k\Omega$, $R_3 = 3k\Omega$, $X_C = -j4k\Omega$ and $X_L = j8k\Omega$.

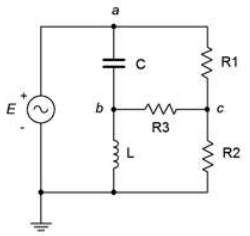


Figure 12.8.30

52. Find voltage v_{bc} in the circuit of Figure 12.8.31 through the use of one or more delta-Y conversions. $E = 20\angle 0°$, $R_1 = 1k\Omega$, $R_2 = 8k\Omega$, $R_3 = 3k\Omega$, $X_C = -j4k\Omega$ and $X_L = j2k\Omega$.

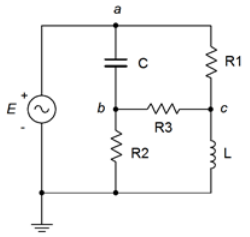


Figure 12.8.31

Design

53. Design an equivalent current source for Figure 12.8.32. $E = 12\angle 90°$, $R = 1k\Omega$ and $X_C = -j200\Omega$. The source frequency is 10 kHz.

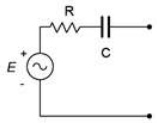


Figure 12.8.32

54. Design an equivalent current source for Figure 12.8.32 if $E = 10\angle 0°$, $R = 2.2k\Omega$ and C = 100 nF. The source frequency is 1 kHz.

55. Design an equivalent current source for Figure 12.8.33 if $E = 1\angle 0°$, R = 600 Ω and L = 2 mH. The source frequency is 20 kHz.

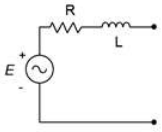


Figure 12.8.33

56. Design an equivalent current source for Figure 12.8.33. $E = 2\angle 90°$, R = 10 kΩ and $X_L = j900\Omega$.

57. Design an equivalent voltage source for Figure 12.8.34. $I = 300E-3\angle 0^\circ$, $R = 4.3k\Omega$ and $X_C = -j5k\Omega$.

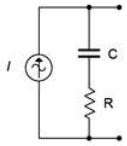


Figure 12.8.34

58. Design an equivalent voltage source for Figure 12.8.34 if $I = 100E-3\angle 120^\circ$, R = 75 Ω and L = 1 mH. The source frequency is 10 kHz.

59. Design an equivalent voltage source for Figure 12.8.35 if $I = 10E-3\angle 0^\circ$, R = 9.1 kΩ and L = 5 mH. The source frequency is 100 kHz.

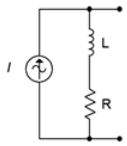


Figure 12.8.35

60. Design an equivalent voltage source for Figure 12.8.35 if $I = 50E-3\angle 0^\circ$, R = 560 Ω and $X_L = j350\Omega$.

61. Reconfigure the circuit of Figure 12.8.36 so that it uses only voltage sources. Express all new component and source values in terms of the original labels.

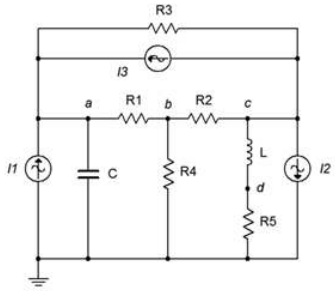


Figure 12.8.36

62. Reconfigure the circuit of Figure 12.8.37 so that it uses only current sources. Express all new component and source values in terms of the original labels.

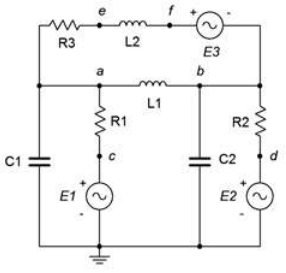


Figure 12.8.37

63. Redesign the circuit of Figure 12.8.1 so that it uses only current sources and produces the same node voltages as the original circuit.

64. Consider the 600 Ω resistor to be the load in Figure 12.8.17. Determine a new value for the load in order to achieve maximum load power. Also determine the maximum load power.

65. Using Thévenin's theorem with the circuit of Figure 12.8.18, determine a new value of capacitive reactance such that it cancels the Thévenin reactance.

66. Using Norton's theorem with the circuit of Figure 12.8.19, determine a new value of inductive reactance such that the inductor current is 1 mA in magnitude.

Challenge

67. Redesign the circuit of problem 7 (Figure 12.8.4) so that it uses only current sources and produces the same node voltages as the original circuit.

68. In the circuit of Figure 12.8.38, use any method or combination of methods to determine v_{ab}. $I1 = 0.05\angle 0^\circ$, $I2 = 0.1\angle 0^\circ$ and $I3 = 0.2\angle 90^\circ$.

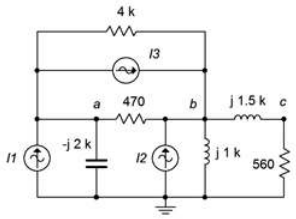


Figure 12.8.38

69. In the circuit of Figure 12.8.39, use any method or combination of methods to determine v_{ab}. $E1 = 5\angle 0^\circ$, $E2 = 10\angle 90^\circ$ and $E3 = 15\angle 0^\circ$.

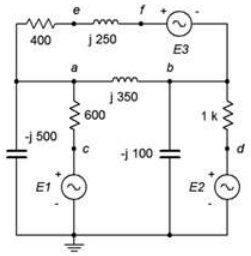


Figure 12.8.39

70. In the circuit of Figure 12.8.39, use any method or combination of methods to determine v_{ce}. $E1 = 15\angle 0^\circ$, $E2 = 30\angle 90^\circ$ and $E3 = 45\angle 0^\circ$.

71. Use any method or combination of methods in the circuit of Figure 12.8.40 to determine v_{ad}. $E1 = 90\angle 0^\circ$, $E2 = 120\angle 0^\circ$ and $I = 400E - 3\angle 180^\circ$.

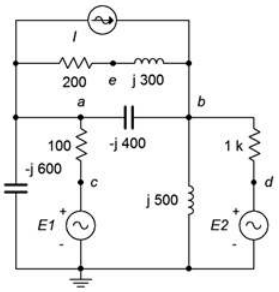


Figure 12.8.40

72. Consider the $2.7k\Omega + j4k\Omega$ combo to be the load in Figure 12.8.5. Determine if this value achieves maximum load power. If not, determine a new value for the load in order to achieve maximum load power. Also determine the maximum load power.

73. In the circuit of Figure 12.8.41, assume the source E is 120 volts RMS at 60 Hz. Determine the value for the load, Z, that will produce maximum load power. Express Z in terms of a resistor and either an inductor or capacitor. Further, specify both the series and parallel equivalents for the load.

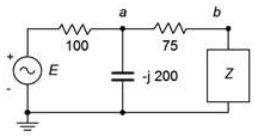


Figure 12.8.41

74. Convert the circuit of Figure 12.8.42 into the equivalent delta configuration.

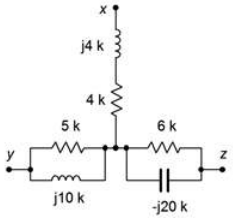


Figure 12.8.42

75. Find the Thévenin equivalent looking into nodes a and b for the circuit of Figure 12.8.43. $I = 4\angle 0^\circ$.

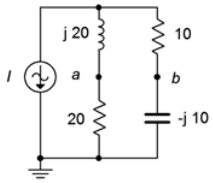


Figure 12.8.43

76. Find voltage v_{bc} in the circuit of Figure 12.8.44 through the use of one or more delta-Y conversions. $E = 100\angle 0^\circ$, $R_1 = R2 = 2k\Omega$, $R_3 = 3k\Omega$, $R_4 = 10k\Omega$, $R_5 = 5k\Omega$, $X_{C1} = X_{C2} = -j2k\Omega$.

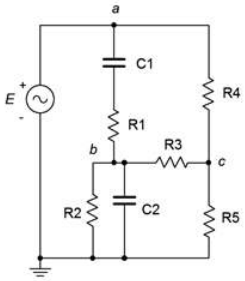


Figure 12.8.44

77. Given the circuit of Figure 12.8.45, determine an equivalent circuit using a single voltage source. $E1 = 100\angle 0^\circ$, $E2 = 60\angle 180^\circ$, $E3 = 40\angle 90^\circ$, $E4 = 75\angle 0^\circ$.

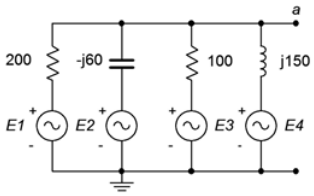


Figure 12.8.45

78. As mentioned earlier, it is possible that certain AC Y and delta networks cannot be converted. Consider the circuit of Figure 12.8.46. Can this circuit be converted with a practical outcome? Why/why not?

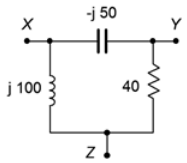


Figure 12.8.46

Simulation

79. Verify the voltage computed for problem 1 by running a transient analysis.

80. Verify the voltage computed for problem 4 by running a transient analysis.

81. Using multiple transient analysis simulations, compare the original circuit of problem 54 to its converted equivalent. Do this by connecting various components to the output terminals, trying several different impedance values and checking to see if the two circuits always produce the same voltage across this impedance.

82. Using multiple transient analysis simulations, compare the original circuit of problem 59 to its converted equivalent. Do this by connecting various components to the output terminals, trying several different impedance values and checking to see if the two circuits always produce the same voltage across this impedance.

83. Run a transient analysis to verify the design of problem 70.

84. Run a transient analysis to verify the design of problem 71.

This page titled 12.8: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **5.8: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

13: AC Power

Learning Objectives

After completing this chapter, you should be able to:

- Describe current, voltage and power relationships in AC RLC networks.
- Plot and make use of the power triangle to determine real, apparent and reactive power components in an AC power system.
- Compute the power factor of an RLC network.
- Determine necessary components for basic power factor correction.
- Perform basic power calculations for systems involving power factor and efficiency.

13.1: Introduction
13.2: Power Waveforms
13.3: Power Triangle
13.4: Power Systems
13.5: Summary
13.6: Exercises

This page titled 13: AC Power is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

13.1: Introduction

This chapter introduces the concept of power and power waveforms in AC systems. This is an important part of AC circuit analysis and turns out to have striking differences compared to the DC counterpart. While it remains true that power is the product of current and voltage, a naive application of that definition can lead to erroneous conclusions for the AC case. In Chapter 1, RMS (i.e., root-mean-square) values were defined and explained. As a general rule, RMS values are used for power calculations, not peak or peak-to-peak values. Further, while complex non-sinusoidal waveshapes are a decided possibility in electronic systems, we shall limit ourselves here to sinusoids.

One of the tools we shall use is the power triangle. This is a simple trigonometric device designed to illustrate the power relations between resistive and reactive components in a complex impedance. One of its parameters is the power factor, PF. As we shall see, ordinarily we like the power factor to be unity as this implies best use of the available current. It turns out that this is not the case in many systems. As a consequence, we shall also investigate a simple means of compensating or shifting the power factor back to unity. This is known as power factor correction.

As part of our discussion involving power factor, we shall examine typical applications such as motors. Here we shall consider the power factor of a motor along with its efficiency. The efficiency is defined as the useful output power relative to the supplied power and is always less than 100%. Finally, we shall consider basic power factor correction for this application.

One practical item to remember here is that, while the instantaneous power changes over time due to the sinusoidal cycling of the voltage and current, what matters to most electrical and electronic devices is the production of internal heat. Devices such as resistors, transistors and so forth, have mass, and thus exhibit a thermal time constant. That time constant tends to be much longer than the period of the wave. The effect is an averaging of the power waveform. In other words, components do not heat up and cool down instantaneously, any more than a hot fry pan would drop to room temperature the moment it was removed from its burner.

This page titled 13.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

13.2: Power Waveforms

Computation of power in AC systems is somewhat more involved than the DC case due to the phase between the current and voltage. It has been stated in prior work that power dissipation is characteristic of resistors, and that ideal inductors and capacitors do not dissipate power. We shall show precisely why this is the case by examining three distinct cases for AC circuits: purely resistive, purely reactive and complex impedance.

Resistive Load

First, consider the case of the purely resistive load, that is, a load with a phase angle of 0 degrees. To determine the power, we simply multiply the voltage by the current. Recall that the basic expression for a sine wave voltage without a DC offset is:

$$v(t) = V\sin(2\pi ft + \theta)$$

where

- $v(t)$ is the voltage at some time t,
- V is the peak value,
- f is the frequency,
- θ is the phase shift.

We know that the current and voltage are always in phase for a resistor, and thus θ is zero degrees. Thus, the expression for a sinusoidal current is similar, using I in place of V for the peak current. We multiply the current and voltage together to arrive at an expression for power[1]:

$$\begin{aligned} P(t) &= v(t) \times i(t) && (13.2.1)\\ &= V\sin(2\pi ft) \times I\sin(2\pi ft) && (13.2.2)\\ &= VI\left(\frac{1}{2} - \frac{1}{2}\cos(2\pi 2ft)\right) && (13.2.3)\\ &= \frac{VI}{2} - \frac{VI}{2}\cos(2\pi 2ft) && (13.2.4) \end{aligned}$$

The final expression is made of two parts; the first portion which is fixed (not a function of time) and the second portion which consists of a negative cosine wave at twice the original frequency. This can be visualized as a time shifted sine wave that is riding on a DC level which is equal to the peak value of the new sinusoid. This is shown in Figure 13.2.1 using current and voltage peaks normalized to unity. In this figure, the current waveform (green) is drawn just slightly above its true value so that it may be seen easily next to the otherwise identical red voltage waveform.

The power product is shown in blue. Unless the frequency is ridiculously low, the resistor's heating will respond to the average value of this waveform thanks to the device's thermal time constant. Due to the fact that sinusoids are symmetrical around zero, the effective power dissipation averaged over time will be the offset value, or $VI/2$. For example, a one volt peak source delivering a current of one amp peak, as shown here, should generate $VI/2$, or 0.5 watts. This crosschecks nicely with the RMS calculation of roughly 0.707 volts RMS times 0.707 amps RMS also yielding 0.5 watts.

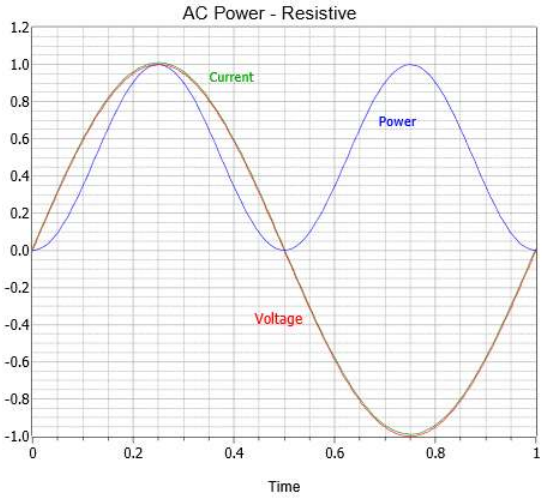


Figure 13.2.1: Waveforms for a resistive load (the current is shifted slightly positive to ease viewing).

Reactive Load

The situation is considerably different if the load is purely reactive. For a load consisting of just an inductor, the voltage leads the current by 90 degrees. This is equivalent to a cosine wave. Once again, we multiply the voltage by the current to arrive at an expression for power:

$$P(t) = V\cos 2\pi ft \times I\sin 2\pi ft \tag{13.2.5}$$

$$= VI\left(\frac{1}{2}\sin 2\pi 2ft\right) \tag{13.2.6}$$

$$= \frac{VI}{2}\sin 2\pi 2ft \tag{13.2.7}$$

Note that this expression does not contain a constant term and only contains a time-varying term. Consequently, without an offset, there is no net power dissipation. The result is shown in Figure 13.2.2. Here power is being alternately generated and dissipated (i.e., positive values indicate dissipation while negative values indicate generation). In this respect, the reactive element can be thought of as alternately storing and releasing energy in the manner of an ideal spring being compressed and then released.

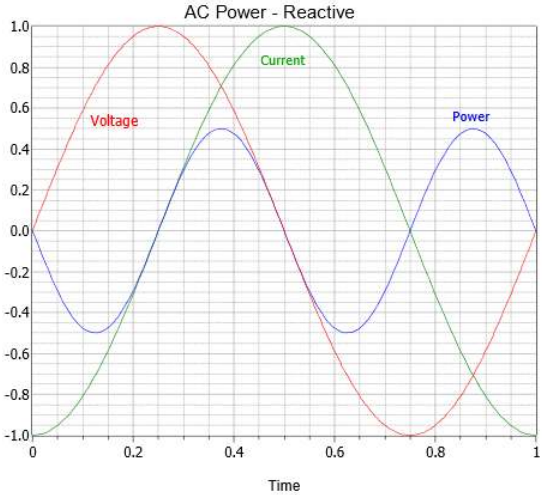


Figure 13.2.2: Waveforms for a purely reactive load.

Complex Impedance Load

Finally, we come to the case of a complex load, part resistive and part reactive. Given some phase angle, θ, we have:

$$P(t) = V \sin 2\pi ft \times I \sin(2\pi ft + \theta) \tag{13.2.8}$$

$$= VI\left(\frac{1}{2}\cos\theta - \frac{1}{2}\cos(2\pi 2ft + \theta)\right) \tag{13.2.9}$$

$$= \frac{VI}{2}\cos\theta - \frac{VI}{2}\cos(2\pi 2ft + \theta) \tag{13.2.10}$$

This expression contains both a constant term and a time varying term, like the case for the purely resistive load shown in Equation 13.2.4. There is, however, an important distinction. The constant term is multiplied by the cosine of the impedance angle, a value whose magnitude ranges from 0 up to 1. Therefore, unless θ is zero, the offset will not equal the peak value of the sinusoidal portion. This is a particularly important point which shall be amplified in a moment.

Example waveforms using $\theta = 45^\circ$ are shown graphically in Figure 13.2.3. The power waveform dips slightly below zero but is not symmetrical around the time axis. Consequently, there is some power dissipation but not as much as in the purely resistive case. In short, the long term power average is now a function of the phase angle, θ. As cosine θ may range between 0 and 1, the power for the complex impedance case will never be more than that of the purely resistive version. Indeed, we can see that Equation 13.2.10 is the general case. If the load is purely resistive then θ is zero, and Equation 13.2.10 reduces to Equation 13.2.4. Similarly, if the load is purely reactive then θ is $\pm$90 degrees, and Equation 13.2.10 reduces to Equation 13.2.7.

While this analysis used an inductive load, the same can be said regarding the capacitive case (simply swap the labels for the current and voltage waveforms). Finally, in the equations above, V and I are peak values. If RMS values are used, there is no need to divide VI by 2.

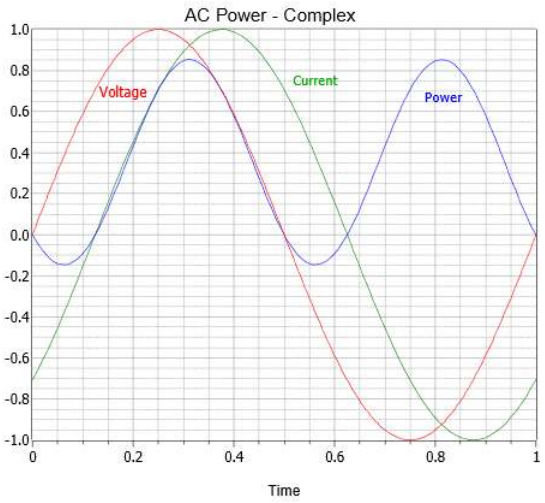


Figure 13.2.3: Waveforms for a complex impedance load.

At this point we can see that resistors dissipate true power but that reactive components do not. This raises a practical problem, namely, what to call the currentvoltage product for purely reactive or complex loads. That is, we can't lump together the current-voltage values for an inductor with those of a resistor any more than we would simply add the magnitudes of resistance and reactance. The practical solution is that we refer to the “power” in reactive components as reactive power. Reactive power uses the symbol Q. Further, the units are not watts, but volt-amps reactive, or more commonly, VAR[2]. Continuing, for a complex impedance we refer to apparent power. It uses the symbol S and has units of volt-amps, abbreviated VA. It is called apparent power because it appears to be the power if you naively multiply the value obtained from a voltmeter by the value obtained from an ammeter. Those devices would not account for the phase angle between the voltage and current, unlike a proper power meter, and their product would not be the true power. The various power terms are summarized in Figure 13.2.4.

Figure 13.2.4: (TABLE) Symbols and units for power quantities.

Quantity	Symbol	Unit, Abbreviation
Power	P	watts, W
Apparent Power	S	volt-amps, VA
Reactive Power	Q	volt-amps reactive, VAR

A few examples are in order to help solidify these concepts.

Example 13.2.1

Determine the power dissipated by the resistor in the circuit of Figure 13.2.5. Also find the apparent power drawn by the circuit and the reactive power of the inductor. The source frequency is 1 kHz.

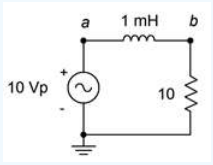


Figure 13.2.5: Circuit for Example 13.2.1.

The first item is to find the reactance of the inductor.

$$X_L = 2\pi f L$$

$$X_L = 2\pi 1kHz1mH$$

$$X_L = j6.283\Omega$$

There are several ways to find power. In a series loop like this, the most direct is to use the i^2R forms. The source current can be found via Ohm's law. As power calculations utilize RMS values, first find the RMS value of the source voltage.

$$v_{RMS} = \frac{v_{peak}}{\sqrt{2}}$$

$$v_{RMS} = \frac{10V}{\sqrt{2}}$$

$$v_{RMS} \approx 7.07V$$

$$i = \frac{v}{Z}$$

$$i = \frac{7.07V}{10 + j6.283\Omega}$$

$$i = 0.5986\angle -32.1^\circ A$$

For the power calculations, we shall only use the magnitudes of the voltage and current. Here, the symbol "||" refers to just the magnitude of the reactance or impedance.

$$P = i^2R$$

$$P = (0.5986A)^2 10\Omega$$

$$P \approx 3.58W$$

$$Q = i^2|X|$$

$$Q = (0.5986A)^2 6.283\Omega$$

$$Q \approx 2.25 \text{ VAR, inductive}$$

$$S = i^2|Z|$$

$$S = (0.5986A)^2|10 + j6.283\Omega|$$

$$S \approx 4.23 \text{ VA, inductive}$$

Computer Simulation

The circuit of Figure 13.2.5 is captured in a simulator as shown in Figure 13.2.6. Three different transient analysis simulations are run.

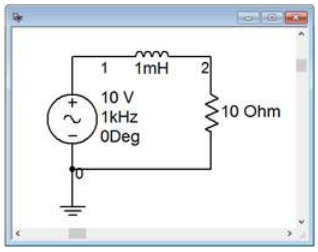


Figure 13.2.6: The circuit of Figure 13.2.5 in a simulator.

The first simulation plots the circulating current (green), the resistor voltage (red) and their product (the power, in blue). This is shown in Figure 13.2.7. We can see that current and voltage are perfectly in phase, as expected. Also, the power waveform ranges from zero up to about 7 watts. The average of this is approximately half of the peak-to-peak, or about 3.5 watts, just as calculated.

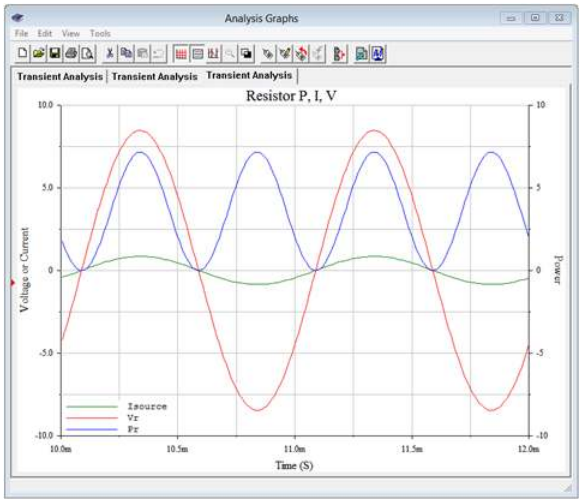


Figure 13.2.7: Transient analysis for the resistor of Figure 13.2.5.

The power value can also be computed from voltage times current as a crosscheck. The voltage across the resistor can be found via the voltage rule, and its magnitude is approximately 5.986 volts RMS. Multiplying this by the RMS current will also yield 3.58 watts.

It is instructive to compare these curves to those generated in Figure 13.2.1 for the general resistive case. The current and voltage values in Figure 13.2.1 were normalized to unity so they do not appear to be identical to those of Figure 13.2.7, however, the important part is that the phase relationships are the same along with the position of the power waveform. In both cases the power waveform ranges from a minimum of zero up to some maximum value. Consequently, its average value must be half of its peak-to-peak value.

A second set of plots is generated for the inductor. This is shown in Figure 13.2.8. Again, compare this set against the curves seen in Figure 13.2.2 for the general reactive case. We can see that the current (green) is 90 degrees out of phase with the inductor's voltage (red) and lagging, as expected. More importantly, we see that the power waveform (blue) is centered around zero. The full cycle average of this is zero, meaning that no power is dissipated. But how does this square with the 2.25 VAR reactive power that was calculated for the inductor? A close look at the power plot shows that that value corresponds to the maximum value of the reactive power waveform (i.e., half of its peak-to-peak value).

https://eng.libretexts.org/@go/page/52978

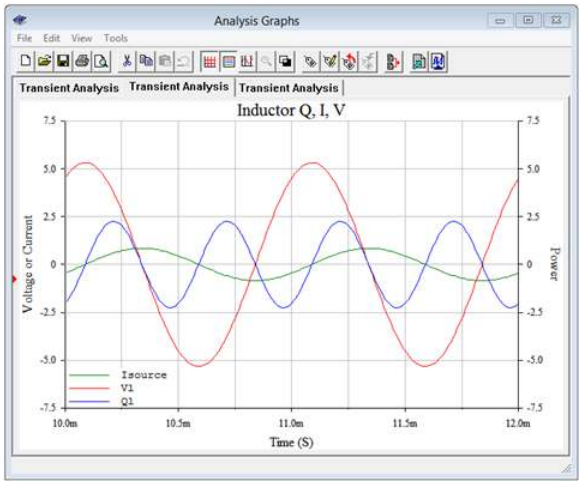


Figure 13.2.8: Transient analysis for the inductor of Figure 13.2.5.

Finally, a third set of curves are created for the circuit as a whole. In other words, now we're treating the series combination of the inductor and resistor as the load. The results are illustrated in Figure 13.2.9.

The computed impedance phase angle was lagging at 32.1 degrees. We can see this same shift between the voltage (red) and current (green) waveforms. The interesting bit here is the offset and amplitude of the power waveform (blue). The waveform has a peak-to-peak value of about 8.5 VA. Once again, the computed value for apparent power, S, works out to one-half of the plotted peak-to-peak value. This will be the case for P, Q and S. Further, it turns out that if we find the full cycle average of this waveform, those small negative peaks would subtract from the total area and reduce the value. The result would be the true power of 3.58 watts. We'll look at this more closely following another example.

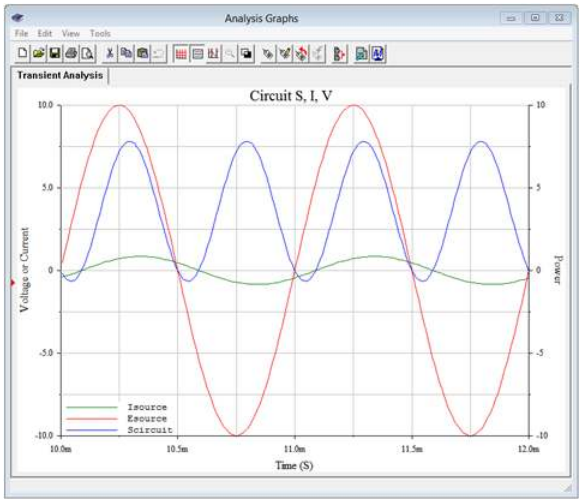


Figure 13.2.9: Transient analysis for the resistor and inductor together in Figure 13.2.5.

Example 13.2.2

Determine the power dissipated by the resistor in the circuit of Figure 13.2.10. Also find the apparent power drawn by the circuit and the reactive power of the capacitor. The source frequency is 1 kHz.

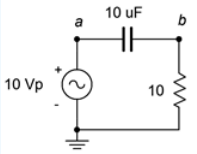


Figure 13.2.10: Circuit for Example 13.2.2.

The first item is to find the reactance of the capacitor.

$$X_C = \frac{1}{2\pi fC}$$

$$X_C = \frac{1}{2\pi 1kHz1\mu F}$$

$$X_C = -j15.92\Omega$$

Unlike the the previous example, we shall use the v^2/R forms for power as an alternative. The RMS value of the source voltage is 7.07 volts. First, find v_R.

$$v_R = E\frac{R}{R - jX_C}$$

$$v_R = 7.07V\frac{10\Omega}{10 - j15.92\Omega}$$

$$v_R = 3.761\angle 57.9^\circ V$$

The capacitor voltage is found via KVL:

$$v_C = E - v_R$$

$$v_C = 7.07\angle 0^\circ - 3.761\angle 57.9^\circ V$$

$$v_C = 5.99\angle -32.1^\circ V$$

For the powers, we just use the magnitude of the voltage.

$$P = \frac{v^2}{R}$$

$$P = \frac{(3.761V)^2}{10\Omega}$$

$$P \approx 1.414W$$

$$Q = \frac{i^2}{|X|}$$

$$Q = \frac{(5.99V)^2}{15.92\Omega}$$

$$Q \approx 2.25 \text{ VAR, capacitive}$$

$$S = \frac{E^2}{|Z|}$$

$$S = \frac{(7.07V)^2}{|10 - j15.92\Omega|}$$

$$S \approx 2.66 \text{ VA, capacitive}$$

Computer Simulation

In order to verify the results, the circuit of Figure 13.2.10 is captured in a simulator as shown in Figure 13.2.11. Once again, three different transient analysis simulations are run; one each for the resistor, the capacitor, and the pair together.

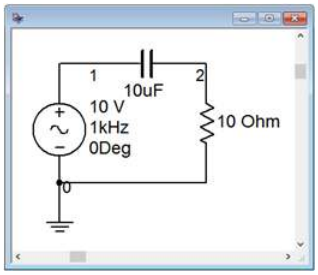


Figure 13.2.11: The circuit of Figure 13.2.10 in a simulator.

In Figure 13.2.12 we see the results of a transient analysis run on the resistor. We can see that the voltage and current are in phase. Also, the power waveform swings from zero up to around 2.8 watts or so. This corresponds to an average value of just under 1.5 watts, and this agrees nicely with the computed result.

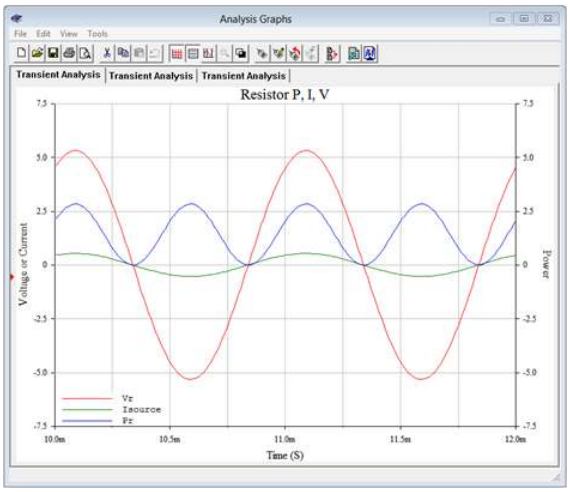


Figure 13.2.12: Transient analysis for the resistor of Figure 13.2.11.

Figure 13.2.13 illustrates the results from a transient analysis run on the capacitor. As expected, the current is leading the voltage by 90 degrees. We can also see that the power waveform is swinging symmetrically around zero, meaning that there is no net power dissipation. The peak value is just under 2.5 VAR, aligning nicely with the calculated value.

Finally, in Figure 13.2.14 we have the results of a transient analysis using both the capacitor and the resistor as the load. The current waveform is still leading the voltage, but by less than 90 degrees. In fact, it leads by about 2/3rds of a division, some 60° or so, which conforms nicely to the expected impedance angle of -57.9° (i.e. from $10 - j15.92\Omega$). The power waveform sits slightly below the horizontal axis indicating it is neither true power nor reactive power, but a combination. The peak-to-peak value is somewhat over 5 units, indicating an apparent power of just over 2.5 VA, again, just as expected.

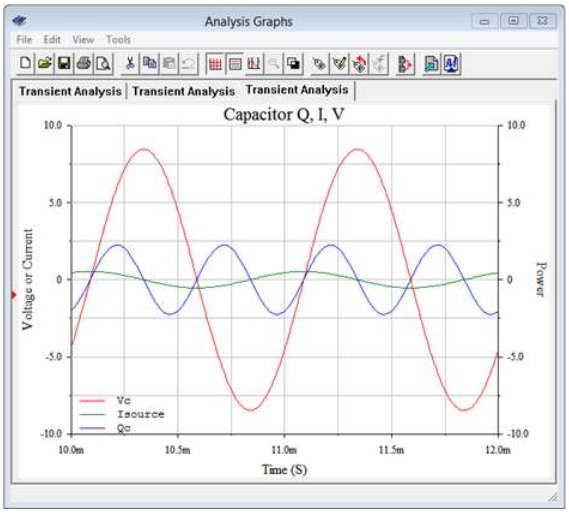


Figure 13.2.13: Transient analysis for the capacitor of Figure 13.2.11.

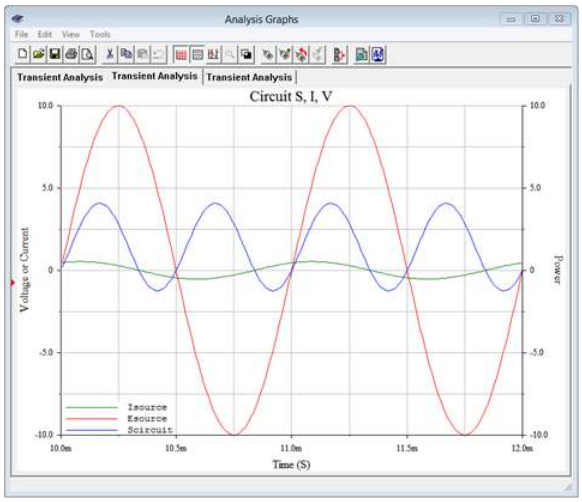


Figure 13.2.14: Transient analysis for the resistor and capacitor together in Figure 13.2.11.

References

[1]A useful trigonometric identity here is $(\sin x)^2 = 1/2 - 1/2 \cos 2x$

[2]For the plural form, some sources use “VARs” while others use “VAR”. We shall use the latter.

This page titled 13.2: Power Waveforms is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.2: Power Waveforms** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

13.3: Power Triangle

The prior section revealed that the phase angle between the current and voltage cannot be ignored when computing power. For example, if a 120 volt RMS source delivers 2 amps of current, it appears that it delivers 240 watts. This is only true if the load is purely resistive. For a complex load, the true power is somewhat less. In fact, as we've just seen, if the load is purely reactive, there will be no true load power at all.

Although plotting the current, voltage and power waveforms is instructive, it can be somewhat cumbersome. Instead, we use a power triangle as shown in Figure 13.3.1. The horizontal axis represents true power, P, in watts. The vertical axis represents reactive power, Q, in VAR. The vector combination of P and Q results in the apparent power, S, in VA. Remember, the apparent power is the product of the magnitudes of the current and voltage. This is what the power "appears to be" based on simple current and voltage measurements from a DMM, versus a proper power meter. In the resistive case, there is no reactive power and thus S and P are the same. Consequently, the S vector collapses onto the P vector. In a purely reactive case, there is no true power and S and Q are the same; both vectors identical and vertical. For the complex case, S is the vector sum of P and Q. This simple graphic nicely encapsulates the relationship between the three vectors. Further, given any two of the four parts (three vector magnitudes and θ) and with just a little trigonometry, the other two parts may be found.

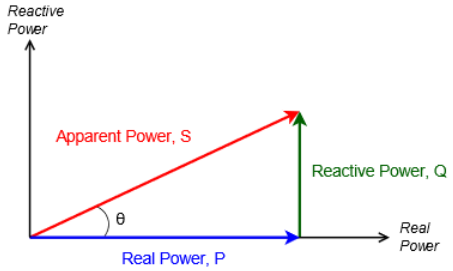


Figure 13.3.1: The power triangle.

For example, knowing the real and reactive powers, the apparent power can be found via the Pythagorean theorem. Similarly, if the apparent power and angle are known, the real and reactive powers may be found using sine and cosine. Remember, apparent power can be found from the product of the RMS voltage and current magnitudes for any complex impedance, and θ is the same as the impedance angle (i.e., the voltage angle minus the current angle). For your convenience, some useful power triangle relationships are listed following.

$$S = v_{RMS} \times i_{RMS} \tag{13.3.1}$$

$$S = \sqrt{P^2 + Q^2} \tag{13.3.2}$$

$$\theta = \tan^{-1}\frac{Q}{P} \text{ (i.e., the impedance angle)} \tag{13.3.3}$$

$$P = S\cos\theta \tag{13.3.4}$$

$$P = \sqrt{S^2 - Q^2} \tag{13.3.5}$$

$$Q = S\sin\theta \tag{13.3.6}$$

$$Q = \sqrt{S^2 - P^2} \tag{13.3.7}$$

Power Factor

As we are often interested in the true power, it is worth noting that a rearrangement of Equation 13.3.4 shows that the ratio of true power to apparent power is the cosine of the impedance angle, $P/S = \cos\theta$. This is known as the power factor and is abbreviated PF. Thus, $PF = \cos\theta$. Knowing the phase angle and the apparent power, true power can be calculated. If PF is positive it is said to be a lagging power factor. This is the case for inductive loads where the current is lagging the voltage. In contrast, a capacitive load results in a negative or leading PF. Recall that for capacitors, current leads the voltage. The sign is only used to indicate leading or lagging and will be useful when we examine power factor correction, shortly. For example, if a 100 volt RMS source delivers 1

amp for an apparent power of 100 VA and the phase angle is -30°, PF is $\cos(-30^\circ)$ or 0.866 leading and the true power is $P = 100\cos(-30^\circ) = 86.6$ watts.

$$PF = \frac{P}{S} = \cos\theta \text{ (positive is lagging and inductive)} \tag{13.3.8}$$

Example 13.3.1

Find S, P and Q in the circuit of Figure 13.3.2. $E = 120$ volts RMS. The source frequency is 60 Hz.

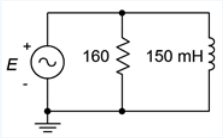


Figure 13.3.2: Circuit for Example 13.3.1.

The first step is to determine the inductive reactance.

$$X_L = 2\pi f L$$

$$X_L = 2\pi 60Hz 150mH$$

$$X_L = j56.55\Omega$$

From here we can determine the system impedance which will, in turn, allow us to determine the source current.

$$Z = \frac{R \times jX_L}{R + jX_L}$$

$$Z = \frac{160\Omega \times (j56.55\Omega)}{160\Omega + j56.55\Omega}$$

$$Z = 53.3\angle 70.5^\circ\Omega$$

$$i_{source} = \frac{e_{source}}{Z}$$

$$i_{source} = \frac{120\angle 0^\circ V}{53.3\angle 70.5^\circ\Omega}$$

$$i_{source} = 2.252\angle -70.5^\circ A$$

The apparent power is the product of the magnitudes of circuit voltage and current.

$$S = E \times i_{source}$$

$$S = 120V \times 2.252A$$

$$S = 270.1 \text{ VA, inductive}$$

$$P = S\cos\theta$$

$$P = 270.1VA\cos 70.5^\circ$$

$$P = 90W$$

$$Q = S\sin\theta$$

$$Q = 270.1VA\sin 70.5^\circ$$

$$Q = 254.7 \text{ VAR, inductive}$$

As a crosscheck, the true power can also be determined by squaring the RMS voltage and then dividing by the 160 Ω resistance. Similarly, dividing the squared voltage by X_L will generate Q.

$$P = \frac{v_R^2}{R}$$

$$P = \frac{120V^2}{160\Omega}$$

$$P = 90W$$

The power triangle for this circuit is shown in Figure 13.3.3.

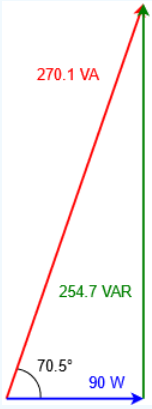


Figure 13.3.3: Power triangle for the circuit of Figure 13.3.2.

Power Factor Correction

One issue with a reactive load is that the current is higher than it needs to be in order to achieve a certain true load power. This is wasteful and would require larger conductors. To alleviate these issues, an opposite reactance can be added to the load such that the resulting load is purely resistive. This can be realized by determining the original Q value and then adding a sufficient reactance to produce an additional Q of the opposite sign resulting in cancellation. From there, it is short step to determine the required impedance. Then, knowing the frequency, the required capacitance or inductance can then be found using the appropriate reactance formula. This is illustrated in the following example.

Example 13.3.2

Find circuit PF, S, P and Q for Figure 13.3.4. $E = 20\angle 0^\circ$ volts peak at a frequency of 10 kHz. Also find an appropriate component which when placed from node a to ground brings PF to unity.

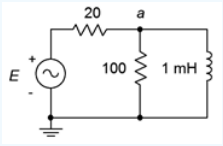


Figure 13.3.4: Circuit for Example 13.3.2.

The inductive reactance of 1 mH at 10 kHz is $j62.83\Omega$. This is in parallel with the 100 Ω resistor, which is then in series with the 20 Ω resistor.

$$Z = 20\Omega + (100\Omega || j62.83\Omega)$$

$$Z = 20\Omega + (28.3 + j45\Omega)$$

$$Z = 66\angle 43^\circ\Omega$$

This time we shall use peak values to illustrate the difference compared to using RMS values as in the previous example. The source current is:

$$i_{source} = \frac{E}{Z}$$

$$i_{source} = \frac{20\angle 0^\circ V}{66\angle 43^\circ\Omega}$$

$$i_{source} = 0.303\angle -43^\circ A \text{ peak}$$

To find S, multiply the magnitudes of source voltage and current. As these are peak values, multiply each by 0.707 to arrive at RMS, or just cut the answer in half (i.e., 0.707^2 is 0.5). This is the apparent power of the whole circuit.

$$S = \frac{E \times i_{source}}{2}$$

$$S = \frac{20V \times 0.303A}{2}$$

$$S = 3.03 \text{ VA, inductive}$$

$$P = S \cos \theta$$

$$P = 3.03VA \cos 43^\circ$$

$$P = 2.22W$$

To be clear, 2.22 watts is the combined power dissipation between the two resistors.

$$Q = S \sin \theta$$

$$Q = 3.03VA \sin 43^\circ$$

$$Q = 2.07 \text{ VAR, inductive}$$

The power triangle for this circuit is shown in Figure 13.3.5.

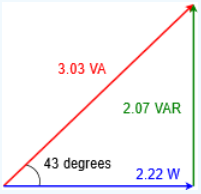


Figure 13.3.5: Power triangle for the circuit of Figure 13.3.4.

For the second part involving power factor correction, we need to add a reactive power equal in magnitude to the existing value but of the opposite sign. This means we need to add 2.07 VAR capacitive. The new power triangle is shown in Figure 13.3.6. The vertical components cancel, resulting in the apparent power equaling the true power with $PF = 1$.

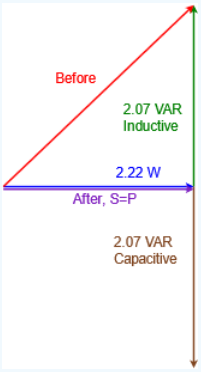


Figure 13.3.6: Power triangle for the circuit of Figure 13.3.4 with power factor correction.

We can place the correction capacitor where it's convenient in physical terms, just as long as we add 2.07 VAR capacitive. We don't have a physical circuit, so that's not a consideration here. A convenient location would be to place it across the existing resistor-inductor combo. Our goal then is to first find the required reactance, and from that, determined the required capacitance. We shall do this two different ways. In the first case, we note that the capacitor appears across the only other reactive element in the circuit. Therefore, in order for them to cancel they must have the same reactance magnitude, and thus X_C must equal $-j62.8\Omega$. But what if there were multiple reactive elements in the circuit or if it was impractical to locate the component there, for example, due to space restrictions? In that case, we would simply work the power relation backwards. For comparison, suppose we place the capacitor in the same location; from node a to ground. A voltage divider can be used to determine the present value of v_a. This works out to 11.4 volts RMS. Using the voltage form of the power rule:

$$Q = \frac{v_X^2}{X}$$

$$X = \frac{v_X^2}{Q}$$

$$X = \frac{11.4V^2}{2.07VAR}$$

$$X = 62.8\Omega$$

Either way we derive X_C, now we just solve the capacitive reactance formula to find C.

$$C = \frac{1}{2\pi f X_C}$$

$$C = \frac{1}{2\pi 10kHz62.8\Omega}$$

$$C = 253.3nF$$

The resulting circuit is shown in Figure 13.3.7.

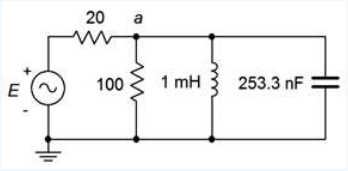


Figure 13.3.7: The circuit of Figure 13.3.4 with power factor correction.

Computer Simulation

It is useful to see the reduction in current demand caused by using power factor correction. To so so, the circuit of Figure 13.3.7 is captured in a simulator as shown in Figure 13.3.8.

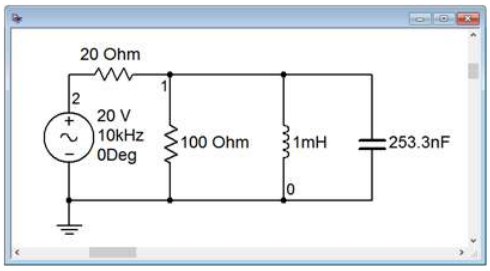


Figure 13.3.8: The circuit of Figure 13.3.7 in a simulator.

We will run two transient analyses to find the source current. The first version will be run using the original circuit configuration without the capacitor. The second run will include the capacitor. To plot the currents, we will make use of Ohm's law. First we obtain the voltages at nodes 1 and 2. Then, using the simulator's post processor, we subtract v_1 from v_2 which yields the voltage across the 20 Ω resistor. This quantity is then divided by 20 Ω to arrive at the input current. This is similar to the current sense resistor technique used in earlier chapters.

The resulting current for the original circuit is shown in Figure 13.3.9. Note that the peak current is just over 300 milliamps, as calculated in step two of the example.

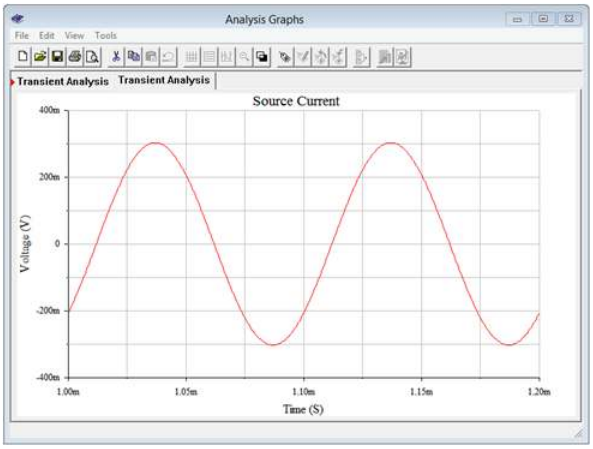


Figure 13.3.9: Simulation results for the circuit of Figure 13.3.8 without power factor correction.

The second simulation is shown in Figure 13.3.10, now with power factor correction.

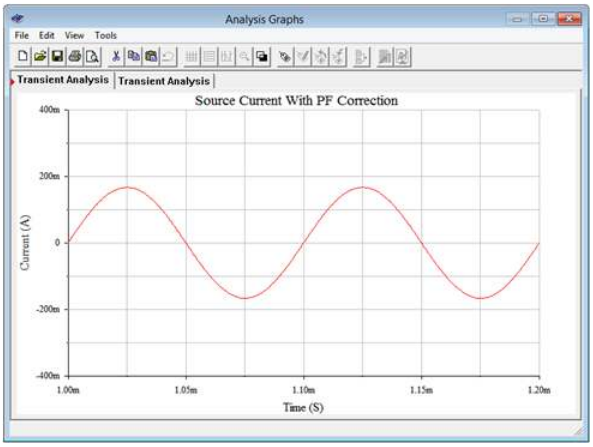


Figure 13.3.10: Simulation results for the circuit of Figure 13.3.8 with power factor correction.

The amplitude here is much reduced, in the range of 160 to 170 milliamps peak. By adding the capacitor, the two reactive currents cancel, leaving a parallel impedance of just 100 Ω. This is in series with the 20 Ω resistor. Dividing 120 Ω into the 20 volt peak source yields a peak current of approximately 167 mA, in agreement with the simulation. The cancellation of the reactive currents is also shown by the fact that the source current is no longer out of phase. In Figure 13.3.10 the current waveform is in phase and starts at zero, as expected. This implies a load that is the equivalent of a pure resistance. In contrast, the plot of Figure 13.3.9 shows a current that is lagging by around one-half division, or about 45 degrees, which unsurprisingly is the approximate value of the circuit's impedance angle. Obviously then, this combined impedance must be inductive.

To sum up, power factor correction is used to lower current demand while keeping load power constant. If the load is fixed, this can be achieved through the use of a capacitor (for inductive loads) or an inductor (for capacitive loads). If the load demand is dynamic, then a more complex system is required, for example, switching through a bank of capacitors to get a value close to the precise value needed for that particular load. More examples of power factor correction are in the next section.

References

[1]Hyperbole seems to work for the movie industry, anyway.

This page titled 13.3: Power Triangle is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.3: Power Triangle** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

13.4: Power Systems

In this section we shall change our focus and consider power systems at a more functional or abstract level rather than at the component level. Specifically, we would like to consider common loads such as motors, heating elements, lighting devices and the like, and how to analyze a system consisting of a variety of different loads. For the most part, we shall stick with the basic scheme of multiple loads being supplied by a common and ideally constant voltage source with each load configured in parallel with the others. Systems do not have to be configured this way but it is typical of residential and commercial applications.

Typical Loads

Three common loads are heating elements, lighting devices and motors or compressors. The first two are fairly obvious, but the the latter are also quite common, although often hidden from view inside devices such as refrigerators, air conditioners and heat pumps. We can classify loads in terms of their typical impedance. Devices such as baseboard electrical heating units, toasters, coffee makers and electric ovens are classified as resistive heating devices. The load they present is easily approximated as a simple resistance. The same is true for incandescent lights, however, there can be a very large change in resistance between the on and off states of incandescent lights. Motors, and other devices integrating motors such as compressors, present an inductive impedance. This is due to the large coils of wire, or windings, inside the motor. Any motorized electrical device falls into this category, such as a washing machine, drill press or the fan of Figure 13.4.1.



Figure 13.4.1: A small boxer fan for cooling electronic components.

Purely capacitive loads are not nearly as common but some everyday devices exhibit highly complex impedances that can vary widely in both amplitude and angle. A standard moving coil dynamic loudspeaker falls into this category. While you would never plug a loudspeaker directly into an AC power system[1], the amplifier that drives it must be designed to meet the challenges of an impedance whose magnitude can change by a factor of ten across the audible range of frequencies. To complicate matters, while the phase typically is inductive, at some frequencies it can be purely resistive or even capacitive.

Efficiency

Efficiency is defined as usable power output divided by applied power input and is denoted by the Greek letter eta, \eta . Normally it is expressed as a percentage and it can never be over 100%.

$$\eta = \frac{P_{output}}{P_{input}} \tag{13.4.1}$$

Some loads, such as typical heating elements, can be thought of as being 100% efficient, meaning that all of the electrical input is turned into useful output (in this case that's heat, although there are more effective ways of using that electrical input to generate heat, such as a heat pump[2]). In contrast, incandescent light bulbs turn very little of their input into the desired quantity (light) and thus suffer from low efficiency. Just a few percent of the electrical energy fed into an incandescent bulb turns into light while the vast majority turns into heat. From that perspective, incandescent lights are more efficient at producing heat than light. LED lighting, on the other hand, is perhaps an order of magnitude more efficient than incandescent lighting, meaning that we can get the same light output for a much smaller electrical power input while simultaneously producing less heat.

Motors lose power in the form of mechanical losses (e.g., friction) and electrical losses (e.g., resistance of windings). Motors are rated in terms of their output power, not the power they draw from the source. For example, a motor with a 1 HP rating might be said to generate 1 HP (approximately 745.7 W) at the shaft. If the motor is 90% efficient, the electrical draw would be 745.7 W/0.9 or 829 W. This situation is further complicated by the phase angle (i.e., power factor) of the motor due to reactive elements, as noted in the previous sections. Compared to motors, home loudspeakers are particularly inefficient, typically converting only about 1% of the electrical input into usable acoustic output.

Example 13.4.1

A certain 120 VAC RMS 60 Hz 1.2 HP motor has an efficiency of 90% and a lagging power factor of 0.85. Determine the apparent power and the current drawn from the system. Also draw the power triangle. 1 HP ≈ 745.7 watts.

The motor output is 1.2 horsepower which is equivalent to:

$$P_{watts} \approx 1.2HP \times 745.7W/HP$$

$$P_{watts} \approx 895W$$

The input required to achieve this is computed with Equation 13.4.1.

$$P_{input} = \frac{P_{output}}{\eta}$$

$$P_{input} = \frac{895W}{0.9}$$

$$P_{input} = 994W$$

From Equations 7.3.7 and 7.3.8, the remaining powers are:

$$S = \frac{P}{PF}$$

$$S = \frac{994W}{0.85}$$

$$S = 1170VA$$

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

$$Q = \sqrt{(1170VA)^2 - (994W)^2}$$

$$Q = 617VAR$$

The current is found via Equation 7.3.1, essentially power law:

$$S = v_{RMS} \times i_{RMS}$$

$$i_{RMS} = \frac{S}{v_{RMS}}$$

$$i_{RMS} = \frac{1170VA}{120V}$$

$$i_{RMS} = 9.75A$$

The power triangle is shown in Figure 13.4.2.

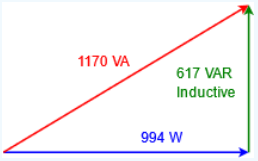


Figure 13.4.2: Power triangle for the motor described in Example 13.4.1.

Example 13.4.2

For the motor described in Example 13.4.1, determine an appropriate component that when placed in parallel will produce unity power factor. Draw the complete power triangle for the system and determine the new current draw.

In Example 13.4.1 the reactive power was determined to be 617 VAR and was inductive, so we'll need 617 VAR capacitive to compensate. The new power triangle is shown in Figure 13.4.3. The source voltage was stated to be 120 V. We can use a

variation on power law to determine the required reactance.

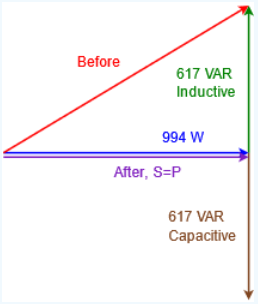


Figure 13.4.3: Power triangle for the corrected system of Example 13.4.2.

$$Q = \frac{v_{RMS}^2}{X_C}$$

$$X_C = \frac{v_{RMS}^2}{Q}$$

$$X_C = \frac{120V^2}{617VAR}$$

$$X_C = -j23.34\Omega$$

Now use the capacitive reactance formula to determine the capacitance value. The line frequency was specified as 60 Hz.

$$C = \frac{1}{2\pi f X_C}$$

$$C = \frac{1}{2\pi 60Hz23.34\Omega}$$

$$C = 114\mu F$$

After correction, the apparent power and real power are the same. Thus,

$$i_{RMS} = \frac{S}{v_{RMS}}$$

$$i_{RMS} = \frac{994VA}{120V}$$

$$i_{RMS} = 8.28A$$

The current draw has been reduced by nearly 1.5 amps, a considerable savings. The new current is 85% of what it used to be.

Now let's consider a larger system that contains several devices.

Example 13.4.3

The system shown in Figure 13.4.4 is supplied by a 240 VAC RMS 60 Hz source. Load 1 is 1000 watts of resistive heating elements. Load 2 is a 3 HP motor with an efficiency of 92% and a lagging power factor of 0.75. Load 3 is a capacitor bank equivalent to 75 μ F. Draw the system power triangle and determine whether or not load 3 is appropriate to bring the system power factor to unity.

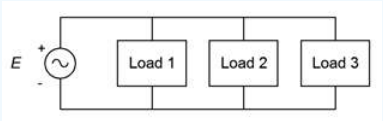


Figure 13.4.4: System block diagram for Example 13.4.3.

First, determine the power triangle for the motor. Then we can add the other loads to it to create a system power triangle. The motor output is 3 HP which is equivalent to:

$$P_{watts} \approx 3HP \times 745.7W/HP$$

$$P_{watts} \approx 2237W$$

The required input power is.

$$P_{input} = \frac{P_{output}}{\eta}$$

$$P_{input} = \frac{2237W}{0.92}$$

$$P_{input} = 2432W$$

The computations for apparent and reactive power follow.

$$S = \frac{P}{PF}$$

$$S = \frac{2432W}{0.75}$$

$$S = 3242VA$$

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

$$Q = \sqrt{(3242VA)^2 - (2432W)^2}$$

$$Q = 2144 \text{ VAR, inductive}$$

The power triangle for the motor is shown in Figure 13.4.5.

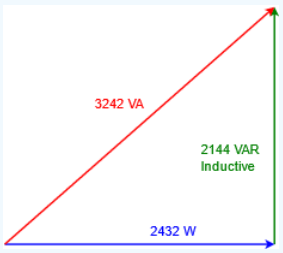


Figure 13.4.5: Power triangle for the motor of Example 13.4.3.

Now for the capacitor. First, we need to determine the reactance, then we can use the v^2/Z form of power law to determine Q.

$$X_C = \frac{1}{2\pi fC}$$

$$X_C = \frac{1}{2\pi 60Hz75\mu F}$$

$$X_C = -j35.37\Omega$$

$$Q = \frac{v^2}{X}$$

$$Q = \frac{(240V)^2}{35.37\Omega}$$

$$Q = 1629 \text{ VAR, capacitive}$$

There are no other reactive elements in the system to consider. We have 515 fewer capacitive VAR than inductive VAR, so the correction will not be ideal. We would need to increase the total capacitance by about 24 \mu F to achieve a power factor of unity. The system power diagram with all of the individual parts is shown in Figure 13.4.6. This is then simplified to the final system power triangle as shown in Figure 13.4.7. The system PF is 0.989 lagging.

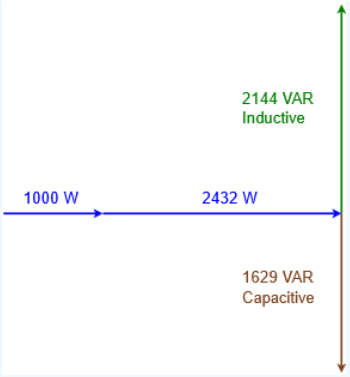


Figure 13.4.6: Power components for the system of Example 13.4.3.

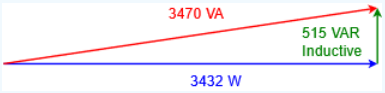


Figure 13.4.7: Final power triangle for the system of Example 13.4.3.

References

[1]That is, unless you enjoy hearing a short-lived blast of sound which is followed by a burst of flames. Plugging a nominal 8 Ω loudspeaker into a 120 VAC outlet would generate 1800 watts of power, well beyond the design limits of any common loudspeaker.

[2]A heat pump moves heat from one place to another. In a properly designed system it takes less energy to move heat than it takes to generate it, and thus it's much more effective than simple resistive heating when it comes to warming the interior of a building.

This page titled 13.4: Power Systems is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.4: Power Systems** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

13.5: Summary

Power is the product of current and voltage, however, the phase relationship between the two has a major impact on the result. The most simple and straightforward case is when the current and voltage are in phase, meaning the load is a pure resistance. In this case, the true power can be computed directly as the product of RMS current and voltage. At the other extreme, when the voltage and current are 90 degrees out of phase, as in the case of a purely capacitive or inductive load, power is alternately generated and dissipated. That is, in a reactive load, true power dissipation is zero. A mechanical analogy is the storage and release of energy in an ideal spring as it is alternately compressed and then allowed to expand. In between these two extremes, that is, when the load is a complex impedance, the true power dissipation is somewhere between zero and the resistive maximum.

The power triangle is used to make visual sense of this situation. It is a right triangle comprised of three legs. The horizontal leg represents the true, or resistive, power and is denoted by the letter P. It is measured in watts. The vertical leg represents the so-called “reactive power”. It is denoted by the letter Q and has units of VAR (volt-amps-reactive). Q can be either inductive or capacitive. The third leg, the hypotenuse, is the apparent power, S. It is measured in VA (volt-amps). It is called apparent power because that is what the power appears to be if it is naively measured with a voltmeter and an ammeter, ignoring the phase difference between them. The angle between the real and apparent powers, θ (theta), is the phase angle between the voltage and current. In other words, theta is the impedance angle. Knowing theta, real power can be determined using right angle trigonometry, namely, $P = S\cos\theta$. The cosine of theta is also know as the power factor, PF. It ranges from 0 (purely reactive) to 1 (purely resistive). Positive or inductive impedance angles are said to have lagging power factor while negative or capacitive impedance angles produce leading power factor.

Ideally, loads are purely resistive and have a power factor of unity. If this is not the case, then for a given voltage, a higher current is needed to create the same real power as in the purely resistive case. This is not advantageous. Power factor correction is the process of shifting the power factor back to unity for complex loads. This is done by inserting a reactance of the opposite sign to counterbalance the reactive portion of the load, for example, adding capacitive reactance to a system that is inductive. The added reactive power must have the same magnitude as the original reactive power but be of the opposite sign, resulting in cancellation.

Efficiency is the measure of usable output power to applied power. Ideally, electromechanical systems such as motors would be 100% efficient, meaning that there is no power loss, but this is not a practical possibility. For example, there will always be frictional losses and power losses in wires.

Review Questions

1. Define apparent power, real power and reactive power.
2. Describe the power triangle.
3. What is power factor, PF?
4. Describe the difference between leading and lagging power factor.
5. What is power factor correction?
6. Give examples of resistive loads and inductive loads.

This page titled 13.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

13.6: Exercises

Analysis

1. For the circuit shown in Figure 13.6.1, determine apparent power S, real power P, reactive power Q and power factor PF. Also, draw the power triangle.

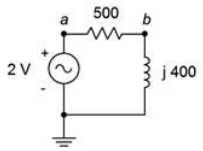


Figure 13.6.1

2. For the circuit shown in Figure 13.6.2, determine apparent power S, real power P, reactive power Q and power factor PF. Also, draw the power triangle.

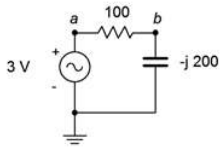


Figure 13.6.2

3. For the circuit shown in Figure 13.6.3, determine apparent power S, real power P, reactive power Q and power factor PF. Also, draw the power triangle. The source is 120 volts.

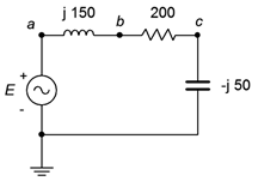


Figure 13.6.3

4. For the circuit shown in Figure 13.6.4, determine apparent power S, real power P, reactive power Q and power factor PF. Also, draw the power triangle. The source is 120 volts.

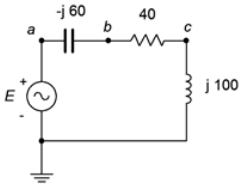


Figure 13.6.4

5. For the circuit shown in Figure 13.6.5, determine apparent power S, real power P, reactive power Q and power factor PF. The source is 90 volts, $X_L = j30\ \Omega$, $R = 50\ \Omega$.

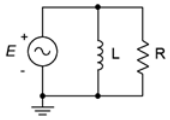


Figure 13.6.5

6. For the circuit shown in Figure 13.6.6, determine apparent power S, real power P, reactive power Q and power factor PF. Also, draw the power triangle. The source is 240 volts, $X_C = -j200\ \Omega$, R = 75 Ω.

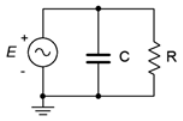


Figure 13.6.6

7. For the circuit shown in Figure 13.6.7, determine apparent power S, real power P, reactive power Q and power factor PF. The source is 120 volts, $X_L = j40\ \Omega$, $X_C = -j25\ \Omega$, R = 20 Ω.

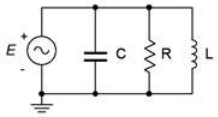


Figure 13.6.7

8. For the circuit shown in Figure 13.6.7, determine apparent power S, real power P, reactive power Q and power factor PF. The source is 120 volts, 60 Hz. R = 80 Ω, C = 20 μF, L = 400 mH.

9. An audio power amplifier delivers a 30 volt RMS 1 kHz sine to a loudspeaker. If the loudspeaker impedance at this frequency is $7\angle 45^\circ$, determine the RMS current delivered to the load and the true power.

10. An audio power amplifier delivers an 80 volt peak 35 Hz sine to a subwoofer. If the subwoofer impedance at this frequency is $4\angle -30^\circ$, determine the peak current delivered to the load and the true power.

11. A certain load is specified as drawing 8 kVA with a lagging power factor of 0.8. Determine the real power P, and the reactive power Q. Further, if the source is 120 volts at 60 Hz, determine the effective impedance of the load in both polar and rectangular form, and the requisite resistance/inductance/capacitance values.

12. A certain load is specified as drawing 20 kVA with a leading power factor of 0.9. Determine the real power P, the reactive power Q and draw the power triangle. If the source is 240 volts at 60 Hz, determine the effective impedance of the load in both polar and rectangular form, and the requisite resistance/inductance/capacitance values.

13. Consider the system shown in Figure 13.6.8. E is a standard 120 V input. If the three loads are 45 W, 60 W and 75 W incandescent light bulbs, respectively, determine the apparent power delivered to the system, the source current, the reactive power and the real power.

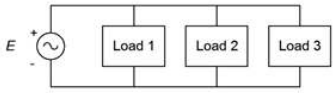


Figure 13.6.8

14. Given the system shown in Figure 13.6.8, determine the apparent power delivered to the system, the source current, the real power, the reactive power and the efficiency. E is 120 V. The three loads are resistive heating elements of 500 W, 1200 W and 1500 W, respectively.

15. Consider the system shown in Figure 13.6.9. E is 240 V. If the three loads are 200 W, 400 W and 1000 W resistive, respectively, determine the apparent power delivered to the system, the real power and the reactive power.

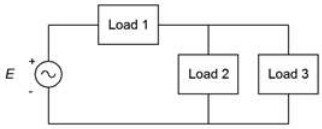


Figure 13.6.9

https://eng.libretexts.org/@go/page/52982

16. Given the system shown in Figure 13.6.9, determine the apparent power delivered to the system, the real power, the reactive power and the efficiency. E is 480 V. The three loads are resistive heating elements of 1500 W, 2000 W and 3500 W, respectively.

17. Consider the system shown in Figure 13.6.8. E is 120 V. Load 1 is 1 kW resistive, load 2 is 400 W resistive and load 3 is 600 VAR inductive. Determine the apparent power delivered to the system, the source current, the reactive power, the real power and the power factor.

18. Consider the system shown in Figure 13.6.8. E is 240 V. Load 1 is 2 kW resistive, load 2 is 800 W resistive and load 3 is 1200 VAR capacitive. Determine the apparent power delivered to the system, the source current, the real power, the reactive power and the power factor.

19. Given the system shown in Figure 13.6.8, determine the apparent power delivered to the system, the source current, the real power, the reactive power and the power factor. E is 120 V. Load 1 is 600 W of incandescent lighting, load 2 is 1200 W of heating elements and load 3 is 200 VAR capacitive.

20. Given the system shown in Figure 13.6.8, determine the apparent power delivered to the system, the source current, the real power, the reactive power and the power factor. E is 60 V. Load 1 is 90 W of incandescent lighting, load 2 is 800 W of heating elements from a dryer and load 3 200 VAR inductive.

21. A 120 V 3 HP motor draws a real power of 2500 W from the source. Determine its efficiency.

22. A 120 V 12 HP motor draws a real power of 10 kW from the source. Determine its efficiency.

23. An ideal 120 V 2 HP motor draws an apparent power of 1800 W from the source. Determine its power factor.

24. An ideal 120 V 0.3 HP motor draws an apparent power of 270 W from the source. Determine its power factor.

25. A 120 V motor is rated at 0.5 HP. It has an efficiency of 78% and a lagging power factor of 0.7. Determine the apparent power drawn from the source (S), the real power (P), and the reactive power (Q) supplied. Also draw the power triangle and find the delivered current.

26. A motor is rated at 10 HP. It has an efficiency of 92% and a lagging power factor of 0.8. Determine the apparent power drawn from the source (S), the real power (P), and the reactive power (Q) supplied. Also draw the power triangle. Finally, determined the current drawn from the 120 V source.

27. Consider the system shown in Figure 13.6.8. E is 120 V. Load 1 is 1 kW resistive, load 2 is 400 W resistive and load 3 is a 1 HP motor that is 80% efficient and has a 0.85 lagging power factor. For the system, determine the apparent power delivered, the source current, the real power, the reactive power and the power factor.

28. Consider the system shown in Figure 13.6.8. E is 120 V. Load 1 is 2.5 kW resistive, load 2 is 500 VAR capacitive and load 3 is a 2 HP motor that is 85% efficient and has a 0.9 lagging power factor. For the system, determine the total power delivered, the source current, the apparent power, the real power and the power factor.

29. For the system shown in Figure 13.6.10, E is 240 V. Load 1 is 1.2 kW resistive heating, load 2 is 400 W resistive lighting, load 3 is a 0.5 HP motor that is 80% efficient with a 0.7 lagging power factor, and load 4 is a 1 HP motor that is 85% efficient with a 0.8 lagging power factor. For the system, determine the apparent power delivered, the source current, the real power, the reactive power and the power factor.

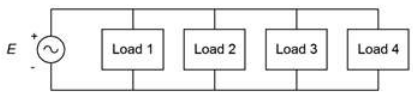


Figure 13.6.10

Design

30. A 120 V 60 Hz source drives a load equivalent to a 75 Ω resistor in parallel with a 25 μF capacitor. Determine the appropriate capacitance or inductance value to place across this load to produce unity power factor.

31. A 240 V 60 Hz source drives a load equivalent to a 10 Ω resistor in parallel with a 50 mH inductor. Determine the appropriate capacitance or inductance value to place across this load to produce unity power factor.

32. A load of $50\angle 30^\circ$ is driven by a 120 V 60 Hz source. Determine the appropriate capacitance or inductance value to place across this load to produce unity power factor.

33. A load of $50\angle -50^\circ$ is driven by a 240 V 60 Hz source. Determine the appropriate capacitance or inductance value to place across this load to produce unity power factor.

34. A certain load is specified as drawing 8 kVA with a lagging power factor of 0.8. The source is 120 volts at 60 Hz. Determine the appropriate capacitor or inductor to place in parallel with this load to produce unity power factor.

35. A certain load is specified as drawing 20 kVA with a leading power factor of 0.9. The source is 240 volts at 60 Hz. Determine the appropriate capacitor or inductor to place in parallel with this load to produce unity power factor.

36. A 240 V 60 Hz source is connected to a load consisting of heating elements of 10 kW along with a 15 HP motor with η = 90%, PF = 0.85. Determine an appropriate capacitor or inductor to place in parallel to produce unity power factor.

37. A 120 V 60 Hz source is connected to a load consisting of 350 W of resistive lighting along with a 1.5 HP motor with η = 70%, PF = 0.75. Determine an appropriate component to place in parallel to produce unity power factor.

Challenge

38. A power distribution system for a concert has the following specifications: Ten class D audio power amplifiers rated at 2 kW output each with 90% efficiency and unity power factor, 10 kW worth of resistive stage lighting to illuminate the musicians alongside a troupe of trained dancing kangaroos, a 3 HP motor used to continuously rotate the drum riser throughout the performance (η = 80%, PF = 0.75) and a 2 HP compressor which inflates and deflates a giant rubber T. rex during particularly exciting parts of the show (η = 85%, PF = 0.8). For the system, determine the total power delivered, the source current, the apparent power, the real power, and the power factor. Finally, make a sketch of this extravaganza with its entertainers in full regalia singing their latest tune “Maximum Volume”.

Simulation

39. Verify the design of problem 28 by performing a transient analysis. The design will have been successful if the source current and voltage are in phase.

40. Verify the design of problem 29 by performing a transient analysis. The design will have been successful if the source current and voltage are in phase.

This page titled 13.6: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **7.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/ACElectricalCircuitAnalysis.pdf.

CHAPTER OVERVIEW

14: Advanced Topic- Magnetic Circuits and Transformers

Learning Objectives

After completing this chapter, you should be able to:

- Describe the basic quantities of a magnetic circuit including magnetic flux, flux density and magnetizing force.
- Outline a BH curve and define aspects including hysteresis, coercivity and retentivity.
- Analyze basic magnetic circuits using BH curves.
- Describe the uses and operation of transformers.
- Analyze basic transformer circuits.

14.1: Introduction

14.2: Electromagnetic Induction

14.3: Magnetic Circuits

14.4: Transformers

14.5: Summary

14.6: Exercises

This page titled 14: Advanced Topic- Magnetic Circuits and Transformers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

14.1: Introduction

This chapter extends the material presented in the chapter on inductors and introduces the concept of magnetic circuits. Basic applications are also illustrated. Here we shall take a closer look at the parameters of magnetic circuits such as permeability and flux density, and introduce new parameters including reluctance and magnetizing force. We shall also examine a key graphic that helps define the magnetic properties of materials, namely the BH curve. In the process we shall define new terms including coercivity and retentivity, and introduce the concept of hysteresis.

As we shall see, there are certain parallels between magnetic circuits and electrical circuits. For example, there is an “Ohm's law for magnetic circuits”, sometimes referred to as Hopkinson's law. There is also a version of Kirchhoff's voltage law dealing with the magnetic equivalents of voltage rises and drops.

Perhaps the two most common applications of magnetic circuits for the new student involve the use of transformers and relays, both of which will be examined in this chapter. Other common applications include dynamic loudspeakers and microphones, magnetic resonance imaging (i.e., medical MRI), pick-up cartridges for vinyl phonographs and the various applications made available by magnetic Hall effect sensors such as advanced anti-lock braking systems, timing control systems for internal combustion engines, and other instances requiring position and motion sensing. These magnetic devices tend to be much more reliable and capable than simple devices such as mechanical switches. This does not mean that magnetic systems are in some way “taking over”, indeed, there are some applications where they have become obsolete. One example is the use of output transformers to couple a home audio power amplifier to a loudspeaker. At one time, these were an integral part of every design, however, today they generally are not used. With this in mind, it is perhaps best to think of magnetic circuits as another item in the engineer's or technician's toolkit to help solve practical problems.

This page titled 14.1: Introduction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.1: Introduction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

14.2: Electromagnetic Induction

Perhaps the most important observation regarding magnetic systems is Faraday's law of electromagnetic induction. Briefly, it states:

$$\text{If a conductor is cut by changing magnetic lines of force, a voltage will be induced in the conductor.} \tag{14.2.1}$$

More specifically,

$$e = -\frac{d\Phi}{dt} \tag{14.2.2}$$

Where

e is the induced voltage,

$d\Phi/dt$ is the rate of change of magnetic flux with respect to time.

Therefore, the larger the flux and the more quickly it fluctuates, the greater the induced voltage. What is important here is a relative change with respect to the conductor and field. This can be accomplished in two basic ways: by a magnetic field that is itself fluctuating around a fixed conductor, or by a conductor moving through a magnetic field.

The concept of electromagnetic induction is a key element of a variety of transducers. Simply put,

$$\text{A transducer is a device that transforms energy from one type to another.} \tag{14.2.3}$$

Although many people don't think of them as such, electric motors and generators are perfect examples of transducers. An electric motor transforms electrical energy into mechanical energy and a generator does the precise opposite, transforming mechanical energy into electrical energy. More commonly, the word transducer is associated with devices such as loudspeakers and microphones. A loudspeaker transforms its electrical input into an acoustic pressure wave (sound) whereas the microphone performs the complementary function of transforming an acoustic pressure wave into an electrical signal. While there are different ways of constructing these devices, the most common designs are the dynamic loudspeaker and dynamic microphone, both of which rely on the principle of electromagnetic induction. The designs of the two devices are strikingly similar, mostly varying in size and parts optimization. Let's take a closer look at how they operate.

Dynamic Loudspeakers and Microphones

Regardless of their output capabilities and frequency range, all dynamic loudspeakers share a common set of elements. Corresponding elements can be found in dynamic microphones. Indeed, the devices are so similar that in some applications, small loudspeakers might pull double duty and switch over to a microphone mode of operation. A cutaway view of a loudspeaker designed to reproduce low frequencies is shown in Figure 10.2.1 .

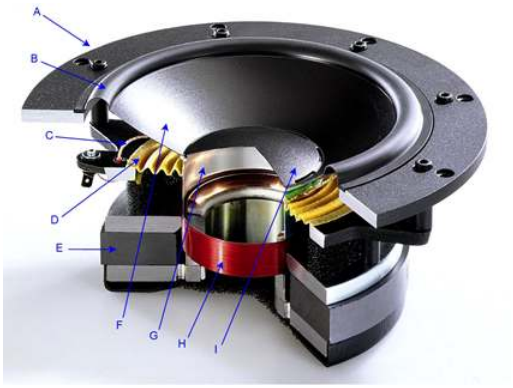


Figure 10.2.1 : Cutaway view of a dynamic loudspeaker. A. Frame B. Suspension C. Lead wire D. Spider E. Magnet F. Diaphragm G. Voice coil former H. Voice coil I. Dust cap Image courtesy of Audio Technology.

The heart of the system is a voice coil (H) which is a coil of tightly wound magnet wire. This sits inside of a magnetic field that is created by a powerful permanent magnet (E). The voice coil is connected to a diaphragm (F) that is is usually made of paper or plastic. This assembly is connected to the frame by the spider (D) at the top of the voice coil, and a suspension at the end of the diaphragm (B). The voice coil and diaphragm can move back and forth relative to the frame, like a piston.

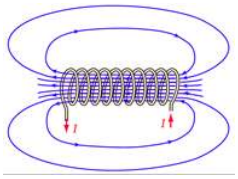


Figure 10.2.2 : Magnetic field around a coil. Image ©, courtesy of HyperPhysics

To understand the operation, recall that passing a current through a coil of wire creates a magnetic field around the coil, as illustrated in Figure 10.2.2 . This magnetic field is the same as is created by a normal magnet. In this case the north pole is on the left side (i.e., lines of force exiting). Consequently, this coil will interact with a permanent magnet just like any other magnet would. In the case of the loudspeaker, the voice coil is fed a current from the amplifier that echoes the music or voice signal. This current creates a magnetic field around the voice coil which interacts with the field of the permanent magnet. As the direction of the current changes, the poles of the voice coil's field reverse. Thus, sometimes the voice coil is pushed outward and sometimes it is pulled inward. Further, the larger the current, the greater the field it creates, and the greater the push or pull.

This results in a back-and-forth motion of the diaphragm that echoes the shape of the waveform being fed from the amplifier. As the diaphragm pushes against the air, it sets up a pressure wave, and we have sound.

A dynamic microphone runs the sequence in reverse. First, the diaphragm will move back and forth in accordance to an applied sound wave. This causes the voice coil to move back and forth within the field of the permanent magnet. We now have a coil of wire being cut by magnetic lines of force, and by Faraday's law of induction, this means that a voltage will be induced in the coil. As long as the diaphragm can respond to the subtle changes in the acoustic pressure wave, then the resulting induced voltage should be of high fidelity. Clearly, to do this with great accuracy, the diaphragm/voice coil/suspension assembly will need to very light and nimble, meaning that the components of the microphone will tend to be much smaller than those of a loudspeaker.

Electric Guitar Pickup

A set of pickups for an electric guitar is shown in Figure 10.2.3 . Although it might be tempting to think that the pickups are just microphones, they are not. If you have any doubt of this statement, just try screaming into a guitar pickup and listen to what comes out[1]. The operation of the guitar pickup does, however, rely on Faraday's law.



Figure 10.2.3 : Pickups on an electric guitar.

As discussed in the prior chapter covering inductance, an electric guitar pickup consists of thousands of turns of very fine wire around a permanent magnet. The pickup is placed immediately below the guitar strings. In this position, the metal guitar strings (generally various combinations of steel, nickel and/or cobalt), are within the field generated by the magnet. When a string is plucked, it vibrates in a complex pattern that depends on the note it is tuned to and the overtones that are present. This pattern distorts the magnetic field because the strings have a much higher permeability than the air around them. We now have a changing magnetic field, in the middle of which is large coil of wire. Faraday's law states that a voltage must be induced in this coil, and that it will follow the motions of the strings. This induced voltage is then fed to an amplifier, hopefully, set to 11.

The astute observer might ask, “Why are there multiple pickups?” Perhaps surprisingly, it is not to get a larger, and thus louder, signal. It has to do with the timbre, or tone quality, of the sound. When a guitar string vibrates, its motion can be thought of as containing the myriad motions of the fundamental pitch and all of the overtones that go with it. Towards the bridge, where the

strings are attached, the lowest pitched elements produce little motion, and thus their strength in the overall signal is reduced. The result is that a pickup placed closer to the bridge sounds “thin” or “cutting” while one placed further up sounds “full” or “thick”.

Bicycle Computer

Our third and final illustrative example is that of a bicycle computer. These handy little devices consist of a small sensor system on the front wheel which is connected to a display unit mounted on the handlebars. Typically, these units will display current speed, average speed, elapsed time, distance covered, and other attributes of interest.



Figure 10.2.4 : Bicycle computer sensor and magnet.

The sensing apparatus of a typical bicycle computer is shown in Figure 10.2.4 . This apparatus consists of two parts: a permanent magnet mounted to one of the wheel spokes, and a weatherproof sensing unit mounted to the front fork. The principle of operation is fairly straightforward: each time the while rotates, the magnet attached to the spoke swings by the sensor. The moving magnetic field creates a short-lived voltage spike in the sensor, an example shown in Figure 10.2.5 . The computer records these spikes over time. Knowing the circumference of the wheel, a simple multiplication yields the accumulated distance traversed. Given internal clocking circuitry, the time recorded between the pulses can be turned into a velocity. Given these data, other attributes such as average or maximum speed are easily obtained with further computation.

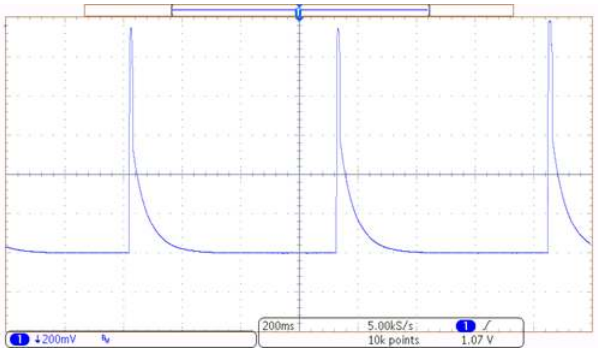


Figure 10.2.5 : Sensor signal feeding bicycle computer.

Using the oscilloscope trace shown in Figure 10.2.5 as an example, we can see that the time between the pulses is about 3.5 divisions, with each division being 200 milliseconds in length. This gives one rotation every 0.7 seconds, or roughly 5140 revolutions per hour. The sensor was mounted on a bike with 700 by 25 mm tires which yields a circumference of about 2.1 meters. Thus, the velocity would be 5140 revolutions per hour at 2.1 meters per revolution, or about 10.8 km/h ($\approx$ 6.73 MPH).[2]

References

[1]A little string resonance if you're lucky, and some irritated band-mates if you're not.

[2]In reality, the bike was mounted on a maintenance stand so its velocity was, in fact, zero. Running alongside a bike with an oscilloscope and probes is no easy feat, especially when the scope needs AC power.

This page titled 14.2: Electromagnetic Induction is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.2: Electromagnetic Induction** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

14.3: Magnetic Circuits

Magnetic circuits include applications such as transformers and relays. A very simple magnetic circuit is shown in Figure 10.3.1 .

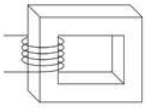


Figure 10.3.1 : Simple magnetic circuit.

First, it consists of a magnetic core. The core may be comprised of a single material such as sheet steel but can also use multiple sections and air gap(s). Around the core is at least one set of turns of wire, i.e., a coil formed around the core. Multiple sets of turns are used for transformers (in the simplest case, one for the primary and another for the secondary). As we have seen already, passing current through the windings generates a magnetic flux, Φ, in the core. As this flux is constrained within the cross sectional area of the core, A, we can derive a flux density, B.

$$B = \frac{\Phi}{A} \tag{14.3.1}$$

Where

B is the magnetic flux density in teslas,

Φ is the magnetic flux in webers,

A is the area in square meters.

Recall from Chapter 9 that one tesla is defined as one weber per square meter. An alternate unit that is sometimes used is gauss (cgs system of units), named after Carl Friedrich Gauss, the German mathematician and scientist.

$$1 \text{ tesla } = 10,000 \text{ gauss} \tag{14.3.2}$$

Example 10.3.1

A magnetic flux of 6E−5 webers exists in a core whose cross section has dimensions of 1 centimeter by 2 centimeters. Determine the flux density in teslas.

First, convert the dimensions to meters to find the area. There are 100 centimeters to the meter.

$$A = width \times height$$

$$A = 0.01m \times 0.02m$$

$$A = 2E-4m^2$$

$$B = \frac{\Phi}{A}$$

$$B = \frac{6E-5Wb}{2E-4m^2}$$

$$B = 0.3T$$

Ohm's Law for Magnetic Circuits (Hopkinson's or Rowland's Law)

There is a common parallel drawn between magnetic circuits and electrical circuits, namely Hopkinson's law (Rowland's law). For electrical circuits, Ohm's law states:

$$V = IR$$

In like manner, for magnetic circuits, we have:

$$\boldsymbol{F} = \Phi \boldsymbol{R} \tag{14.3.3}$$

Where

$\boldsymbol{F}$ is the magnetomotive force (or MMF) in amp-turns,

Φ is the magnetic flux in webers,

$\boldsymbol{R}$ is the reluctance of the material in amp-turns/weber.

The magnetomotive force compares to a source voltage or electromotive force (EMF), the magnetic flux is likened to the flow of current, and the reluctance stands in for resistance (i.e., on the one hand we have a material that resists the flow of current, and on the other, a material in which there is "reluctance" to establish magnetic flux). Further, magnetomotive force is the product of the current flowing through a coil and the number of loops or turns in the coil:

$$\boldsymbol{F} = NI \tag{14.3.4}$$

Where

$\boldsymbol{F}$ is the magneto-motive force in amp-turns,

N is the number of loops or turns in the coil,

I is the current in the coil in amps.

The equation for reluctance has a nice parallel with the equation for resistance (Equation 2.11 from Chapter 2):

$$R = \frac{\phi l}{A}$$

$$\boldsymbol{R} = \frac{l}{\mu A} \tag{14.3.5}$$

Where

$\boldsymbol{R}$ is the reluctance in amp-turns/weber,

l is the length of the material in meters,

A is the cross sectional area of the material in square meters,

μ is the permeability of the material in henries/meter.

Given the characteristics of the coil and the path length of the magnetic circuit, the magnetic flux gives rise to a magnetizing force, H.

$$H = \frac{NI}{l} \tag{14.3.6}$$

Where

H is the magnetizing force in amp-turns/meter,

N is the number of turns or loops in the coil,

I is the coil current in amps,

l is the length of the magnetic path in meters.

Equation 14.3.5 reveals that ferromagnetic materials (i.e., materials that have high permeability, such as steel) produce a low reluctance. The practical problem here is that μ, unlike the resistivity ρ of resistors, is not a constant for these sorts of materials. It can vary considerably, as seen in the general diagram presented in Figure 10.3.2 . As a result, it is impractical to find reluctance in same manner that we find resistance. All is not lost, though.

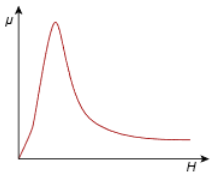


Figure 10.3.2 : Typical permeability curve for a high permeability material.

The flux density and corresponding magnetizing force for any given material are related by the following equation:

$$B = \mu H \tag{14.3.7}$$

Where

B is the flux density in teslas,

μ is the permeability of the material in henries/meter,

H is the magnetizing force in amp-turns/meter.

Once again, the tricky bit here is the permeability of the core material. For air, we can use the permeability of free space, μ_0.

$$\mu_0 = 4\pi \times 10^{-7} H/m \approx 1.257 \times 10^{-6} H/m \tag{14.3.8}$$

For other materials, such as sheet steel or cast steel, we shall take another path; namely an empirically derived curve that plots flux density B against magnetizing force H. Such graphs generically are referred to as "BH curves". An example is shown in Figure 10.3.3 . Clearly, this curve is not a nice straight line, or even an obvious, predictable function. The immediately apparent attributes are the initial steep rise which is followed by a flattening of the curve. This flattening corresponds to saturation of the magnetic material. In contrast, a plot for air would reveal a straight line with a very shallow slope. As we shall see, being able to achieve a high flux density for a given magnetizing force will result in an effective and efficient magnetic circuit. So while air has the positive attribute of not saturating, the resulting flux density is low, usually leading to lower performance.

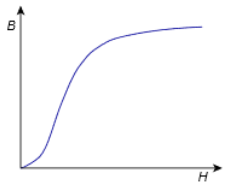


Figure 10.3.3 : Generic BH curve.

The BH Curve

The process of generating a BH curve is as follows. First, we create a core of the material to be investigated. A coil of wire is then wrapped around this core. An example is shown in Figure 10.3.4 . Here we have a basic toroid with a coil of N turns.

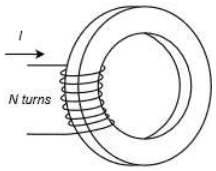


Figure 10.3.4 : Toroidal core with coil.

We begin with the system at rest and not energized. A small current, I, is applied to the coil. This produces a magnetizing force via Equation 14.3.6. There will be a corresponding flux density, as per Equation 14.3.7.

The current is then increased. This produces an increase in magnetizing force and a corresponding change in flux density. The current is increased further, until the curve flattens, indicating that saturation has been reached. This trajectory is shown as the dashed line in Figure 10.3.5 , starting at point a with point b indicating saturation. The current is then reduced. This causes a

reduction in magnetizing force, but while the flux density decreases, it does not trace back perfectly along the original trajectory. Instead, the curve is displaced above the original.

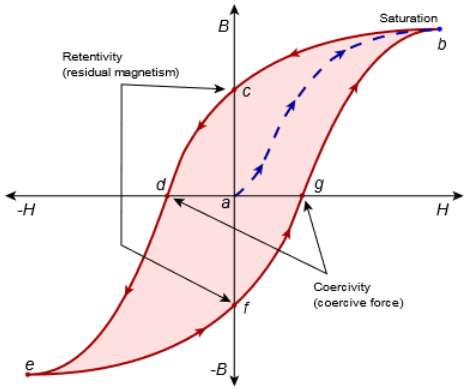


Figure 10.3.5 : Generation of BH curve.

Eventually, the current will be reduced to zero. This corresponds to point c on the graph of Figure 10.3.5 . At this point, even though there is no current in the coil, there is still flux in the core. The resulting flux density is referred to as retentivity and is a measure of residual magnetism. This phenomenon is what makes it possible to magnetize materials.

If we now reverse the direction of the coil current and begin to increase its magnitude, the flux density will continue to drop. At point d, it will have reached zero. As we have effectively coerced the flux back to zero, we call the magnetizing force required to do this the coercivity or coercive force.

As the current magnitude increases, the flux density also increases but with opposite sign. Eventually, saturation is reached again at point e. Once again, if the current's magnitude is decreased, the magnitude of the flux density will also decrease but will not perfectly trace back along the original trajectory. This time it will follow a lower path. When the current is reduced to zero at point f, a mirror retentivity is evident. Further positive increases in current show a mirror coercivity at point g. Finally, as current is increased to maximum, we again reach saturation at point b.

If the current is cycled in this manner again, the process is repeated, and the outer path indicated by the arrows is taken again. Thus, a specific value of magnetizing force can give rise to different values of flux density: it depends on the recent history of the material. This effect is known as hysteresis and is found in other areas as well.

Effectively, published BH curves follow the middle of the hysteresis loop. An example of BH curves for three different core materials is shown in Figure 10.3.6 [1]. Curve A is for sheet steel (which is common in transformers), curve B is for cast steel, and curve C is for cast iron. We shall make use of these in upcoming examples. Curves for other materials are also available.

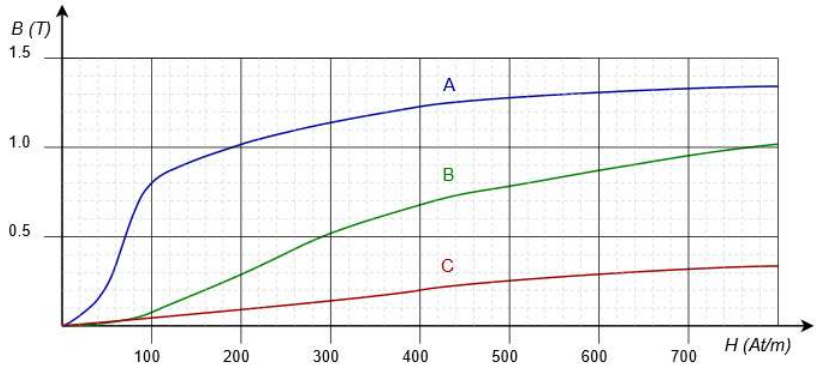


Figure 10.3.6 : BH curves for: A. Sheet steel B. Cast steel C. Cast iron

✔ Example 10.3.2

https://eng.libretexts.org/@go/page/53026

Assume the toroid of Figure 10.3.4 is made of cast steel, has a 500 turn coil, a cross section of 2 cm by 2 cm, and an average path length of 50 cm. Determine the flux in webers if a current of 0.3 amps feeds the coil.

We shall use Equation 14.3.6 to find the magnetizing force and from the BH curve, find the flux density.

$$H = \frac{NI}{l}$$

$$H = \frac{500 \text{ turns } \times 0.3A}{0.5m}$$

$$H = 300At/m \text{ (amp-turns per meter)}$$

Cast steel corresponds to curve B (green) in Figure 10.3.5 . A decent estimate for the flux density is 0.52 teslas. This is the corresponding flux density. In order to find the flux, we need to find the area of the core.

$$A = width \times height$$

$$A = 0.02m \times 0.02m$$

$$A = 4E-4m^2$$

$$\Phi = B \times A$$

$$\Phi = 0.52T \times 4E-4m^2$$

$$\Phi = 2.08E-4Wb$$

KVL for Magnetic Circuits

A cursory examination of Equations 14.3.4 and 14.3.6 shows that:

$$\boldsymbol{F} = Hl = NI \tag{14.3.9}$$

Continuing the Ohm's law analogy, the amp-turns product of the coil, NI, is analogous to a voltage rise. Further, the Hl product is analogous to a voltage drop. If we then extend the analogy to include the concept of Kirchhoff's voltage law, it should come as no surprise that the sum of NI "rises" must equal the sum of the Hl "drops". In the circuit of Figure 10.3.4 , there is a single "rise" and a single "drop". Thus, the magnetic circuit is analogous to the minimal electric circuit shown in Figure 10.3.7 . $\boldsymbol{F}$ is the magnetomotive force, NI, while $\boldsymbol{R}$ is the reluctance of the toroidal core. This reluctance will experience a "drop" of Hl.

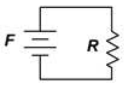


Figure 10.3.7 : Electrical circuit analogy for the magnetic system of Figure 10.3.4 .

The core could consist of two or more different materials, creating the equivalent of a series circuit. In this case, a table such as the one found in Figure 10.3.8 can be used to aid in computation. In this table, each section of the core gets its own row. The table is divided into two sides (note the thick separating line in the center). In general, we will be working through problems where we know the data on the left side and need to find something on the right side, or vice versa. The bridge between the two sides (i.e., traversing the thick line) is a BH curve for the material of that particular section. The exception to this rule is if the section is an air gap. In that case we can use Definition 14.3.8 which shows that for air, $H \approx 7.958E5B$ or alternately, $B \approx 1.257E-6H$.

Figure 10.3.8 : Table format used for magnetic circuit analysis.

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
1						
2						

Time for a few illustrative examples. We shall consider a simple system like that of Figure 10.3.4 , a two section core, a core with an air gap, and a core with two coils.

Example 10.3.3

Assume the core of Figure 10.3.9 is made of sheet steel, has a 200 turn coil, a cross section of 1 cm by 1 cm, and an average path length of 12 cm. Determine the coil current required to achieve a flux of 1E−4 webers.

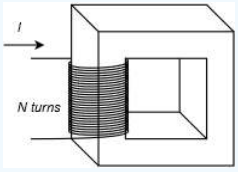


Figure 10.3.9 : Magnetic system for Example 10.3.3 .

The analogous circuit consists of a single source and reluctance, like that of Figure 10.3.7 . Thus,

$$NI = H_{sheet} l_{sheet}$$

We shall begin by filling in the portions of the table that can be addressed directly, such as the path length, area and flux. Don't forget to convert the centimeter values into meters.

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1E−4	1E−4			0.12	

Now we determine the flux density to complete the left side of the table.

$$B = \frac{\Phi}{A}$$

$$B = \frac{1E-4Wb}{1E-4m^2}$$

$$B = 1T$$

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1E−4	1E−4	1		0.12	

We use the BH curve of Figure 10.3.6 to find H from this flux density. From the blue curve (A) the value is about 190 amp-turns per meter.

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1E−4	1E−4	1	190	0.12	

At this point we find the Hl "drop" with a simple multiplication.

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1E−4	1E−4	1	190	0.12	22.8

This "drop" is 22.8 amp-turns and the coil has 200 turns. Therefore the required current is:

$$I = \frac{Hl}{N}$$

$$I = \frac{22.8At}{200t}$$

$$I = 114mA$$

Next, let's consider a core with two sections.

Example 10.3.4

Given the magnetic system shown in Figure 10.3.10 , assume section A is made of sheet steel and section B is made of cast steel. Each part has a cross section of 2 cm by 2 cm. The path length of A is 12 cm and the path length of B is 4 cm. If the coil has 50 turns, determine the coil current required to achieve a flux of 2E−4 webers.

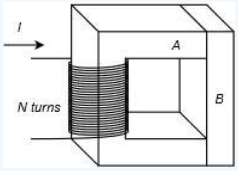


Figure 10.3.10 : Magnetic system for Example 10.3.4 .

The analogous circuit consists of a single source and two reluctances. This is shown in Figure 10.3.11 .

Therefore:

$$NI = H_{sheet}l_{sheet} + H_{cast}l_{cast}$$

For our table there will be two rows, one for sheet steel and the second for cast steel. The first row will require using curve A (sheet steel) from Figure 10.3.6 while the second row will require using curve B (cast steel).

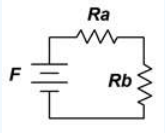


Figure 10.3.11 : Analogous electrical circuit for the system of Figure 10.3.10 .

The flux, like the current in a series loop, will be the same in both sections. From here we can fill out a series of other values in the table to arrive at:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	2E−4	4E−4	0.5		0.12	
Cast Steel	2E−4	4E−4	0.5		0.04	

The BH curves are used to transition to the right side, and we arrive at:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	2E−4	4E−4	0.5	70	0.12	
Cast Steel	2E−4	4E−4	0.5	290	0.04	

And now we fill in the Hl "drops":

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	2E−4	4E−4	0.5	70	0.12	8.4
Cast Steel	2E−4	4E−4	0.5	290	0.04	11.6

Using our KVL analogy, the total "drop" is 8.4 At + 11.6 At, or 20 ampturns. The coil was specified as having 50 turns. Therefore:

$$I = \frac{Hl}{N}$$

$$I = \frac{20At}{50t}$$

$$I = 400mA$$

Notice that even though the cast steel section is shorter than the sheet steel section, it produces a larger "drop"; just like a larger resistor in an electrical circuit. This requires a larger current in the coil than if the entire core had been made of sheet steel.

Finally, it should be apparent that the current demand can be reduced if the coil has more turns. There is a practical limit here because all of the turns have to fit within the opening of the core. Once that space is full, the only way to increase the number of turns is to use finer gauge wire but doing so increases wire resistance and power loss, and also lowers maximum current capacity.

Dealing With Air Gaps

Some systems include an air gap in the core. If there is only a single material for the remainder, this will create a system like that depicted in Figure 10.3.12 . The main observation here is that the permeability of air is far below that of ferromagnetic materials and thus the gap will exhibit a relatively large reluctance compared to its path length. The obvious question then is, why would we use a gap? One possibility is an electro-magnetic relay, the internals of which are illustrated in Figure 10.3.13 .

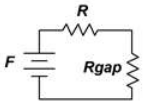


Figure 10.3.12 : Analogous electrical circuit for a system with an air gap.

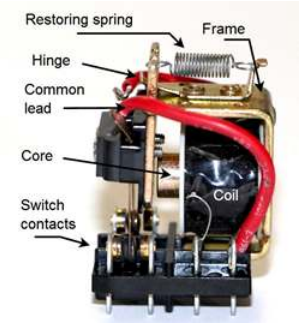


Figure 10.3.13 : Internals of an electrical relay.

To create a relay, we place one portion of the core on a hinge which can be held open with a small spring. This creates the air gap (to the immediate left of the core in the Figure). If we apply a sufficiently large current, the resulting magnetic flux will be enough to overcome the spring tension and close the second piece onto the first (i.e., magnetic attraction). Once this happens, the gap disappears, reducing the reluctance around the loop. The system will stay closed until the coil current is turned off and the restoring spring pulls apart the two sections again. We add insulated metal contacts to the moving section and what we end up with is a heavy duty switch that is "thrown" by a controlling current rather than a mechanical lever moved by a human.

It is worth repeating that for air gaps, instead of a BH curve we can use Definition 14.3.8 which shows that for air, $H \approx 7.958E5B$ or alternately, $B \approx 1.257E-6H$.

Example 10.3.5

Given the magnetic system shown in Figure 10.3.14 , assume the core is made of sheet steel. The cross section throughout is 3E−4 m^2. The path length of the main core is 8 cm and the path length of the gap is 1 mm. How many turns will the coil need in order for a coil current of 400 mA to achieve a flux of 1.2E−4 webers?

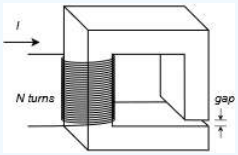


Figure 10.3.14 : Magnetic system for Example 10.3.5 .

The analogous circuit consists of a single source and two reluctances, one for the sheet steel core and a second for the air gap. This is shown in Figure 10.3.12 . Further, the analogous KVL relation is:

$$NI = H_{sheet}l_{sheet} + H_{gap}l_{gap}$$

For this table there will be two rows, one for the sheet steel core and one for the air gap. As we have seen, the flux will be the same in both sections. From there we can fill out some of the other values in the table resulting in:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1.2E−4	3E−4	0.4	63	8E−2	
Gap	1.2E−4	3E−4	0.4	3.183E5	1E−3	

The BH curve for sheet steel and Definition 14.3.8 for the air gap are used to transition to the right side, yielding:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1.2E−4	3E−4	0.4	62	8E−2	5.0
Gap	1.2E−4	3E−4	0.4	3.183E5	1E−3	318.3

The Hl "drops" are filled in:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	1.2E−4	3E−4	0.4		8E−2	
Gap	1.2E−4	3E−4	0.4		1E−3	

Using our KVL analogy, the total "drop" is 5.0 At + 318.3 At, or 523.3 ampturns. The coil current was specified as being 400 mA. Therefore:

$$N = \frac{Hl}{I}$$

$$N = \frac{523.3At}{400mA}$$

$$N \approx 1308 \text{ turns}$$

And finally, an example using two coils.

Example 10.3.6

Given the magnetic system shown in Figure 10.3.15 , assume the core is made of sheet steel. The cross section throughout is 3E−4 m^2 and the length of the core is 20 cm. Coil one consists of 2000 turns and coil two consists of 500 turns. If a current of 120 mA into coil one achieves a flux of 2.4E−4 webers, determine the current of coil 2.

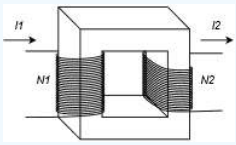


Figure 10.3.15 : Magnetic system for Example 10.3.6 .

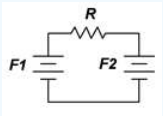


Figure 10.3.16 : Analogous electrical circuit for the system of Figure 10.3.15 .

The analogous circuit consists of two voltage sources and a single reluctance for the sheet steel core. This is illustrated in Figure 10.3.16 .

The analogous KVL relation is:

$$N_1 I_1 - N_2 I_2 = H_{sheet} l_{sheet}$$

As we are seeking I_2, we can rearrange this equation into a more useful form:

$$N_2 I_2 = N_1 I_1 - H_{sheet} l_{sheet}$$

https://eng.libretexts.org/@go/page/53026

Only one row will be required for this table. We fill out the obvious values in the table resulting in:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	2.4E−4	3E−4	0.8		0.2	

The BH curve for sheet steel is used to transition to the right side:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	2.4E−4	3E−4	0.8	100	0.2	

The Hl "drop" is computed:

Section	Flux Φ (Wb)	Area A (m^2)	Flux Density B (T)	Magnetizing Force H (At/m)	Length l (m)	"Drop" Hl (At)
Sheet Steel	2.4E−4	3E−4	0.8	100	0.2	20

Using our KVL analogy, we find:

$$N_2 I_2 = N_1 I_1 - H_{sheet} l_{sheet}$$

$$N_2 I_2 = 2000 \text{ turns } \times 120mA - 20At$$

$$N_2 I_2 = 240At - 20At$$

$$N_2 I_2 = 220At$$

Coil two was specified as having having 500 turns, therefore its current is:

$$I_2 = \frac{220At}{500turns}$$

$$I_2 = 440mA$$

An item of key importance is that the preceding example must make use of an AC current in order to function as described. A DC current will not produce the predicted results. The reason for this goes back to Definition 10.2.1 and Equation 10.2.2: Unless the flux is changing relative to the conducting coil, no voltage will be induced in a coil. So, while a DC input current will create a flux in the core, that flux will be static and unchanging. Consequently, the second coil will not produce an output. In contrast, an AC current smoothly varies in amplitude and polarity. This creates a smoothly varying magnetic flux which in turn allows a voltage to be induced in the second coil.

Assuming for a moment that an AC current was used in the preceding example, notice that the current is nearly four times higher in coil two than in coil one. If we had been able to reduce the core "drop" to zero, perhaps by reducing the core's reluctance to a negligible value, then the current would have been increased precisely by a factor four (to 480 mA versus 440 mA). This is the same as the ratio of the number of turns of coil one to the number of turns of coil two, and is known as the turns ratio. It is a key parameter describing transformers, which just happens to be the subject of the next section.

References

[1]Curves based on https://en.Wikipedia.org/wiki/Saturation_(magnetic); Steinmetz (1917), Theory and Calculation of Electric Circuits; and Boylestad (2010), Introductory Circuit Analysis.

This page titled 14.3: Magnetic Circuits is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.3: Magnetic Circuits** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

14.4: Transformers

Transformers have been in wide use in the electrical industry for over a century. Beginning with Michael Faraday and working through a sequence of scientists and inventors, the fundamentals of modern transformer design had been established by the late 1800s.

Transformer have three basic uses:

- Voltage scaling.
- Isolation.
- Impedance matching.

Transformers range from very small units used in communications systems up to large power distribution units many times the size of an adult human. A couple of transformers typical of the type used in consumer electronics devices is shown in Figure 10.4.1 . These devices are designed to handle standard residential AC input voltages (120 volts in North America, 220 to 240 volts in much of the remainder of the planet) and feeding circuits that demand perhaps 10 to 30 watts of power. The unit on the left is designed to be mounted directly onto a printed circuit board while the unit on the right is designed for chassis mount.



Figure 10.4.1 : Transformers suitable for consumer electronics devices. The unit on the left features four separate coils for enhanced flexibility.

In contrast, Figure 10.4.2 shows pole mounted transformers used to reduce typical North American residential distribution lines (in the vicinity of 15 kV) down to home voltage (120 volts). These transformers are capable of delivering 1000 or more times as much power as the transformers of Figure 10.4.1 . The transformers in these figures by no means present the extremes. Transformers are available that are considerably smaller or larger than the ones pictured here.

Transformers exploit magnetic circuit theory for their operation, as illustrated previously in Example 10.3.6. In its most basic form, the device has two coils or windings; one for the input side (called the primary winding) and one for the output side (the secondary winding). It is possible to have multiple input and/or output windings. Such is the case for the transformer on the left side of Figure 10.4.1 ; it has two input primary coils and two secondary coils. This allows for a variety of interconnection options, as we shall see. A typical schematic symbol is shown in Figure 10.4.3 . This symbol is appropriate for the magnetic system of Figure 10.3.15 which featured two coils. The vertical lines in the symbol represent the magnetic core.



Figure 10.4.2 : A cluster of three residential power transformers.

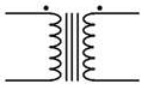


Figure 10.4.3 : Schematic symbol for a simple transformer.

Voltage Scaling and Isolation

A particularly important transformer parameter is the ratio of the number of turns on the primary coil (N_P) to the number of turns on the secondary coil (N_S). This is denoted N, and referred to as the turns ratio.

$$N = \frac{N_P}{N_S} \tag{14.4.1}$$

As the two windings are on the same core and ideally see the same flux, the voltage ratio of the primary to secondary will be equal to N for an ideal lossless transformer. Thus, if $N = 10$ and the primary voltage (V_P) is 60 volts, the secondary voltage (V_S) will be 6 volts. Thus:

$$V_S = \frac{V_P}{N} \tag{14.4.2}$$

The primary and secondary appear as "NI" potentials in the magnetic circuit, therefore, the current will change inversely with N. For example, if $N = 10$ and the primary current is 1 amp, the secondary current will be 10 amps. Thus:

$$I_S = NI_P \tag{14.4.3}$$

It is important to note that the product of primary current and voltage must equal the product of secondary current and voltage for an ideal transformer. Such a device does not dissipate power but rather transforms it (hence the name).

$$V_S I_S = V_P I_P \tag{14.4.4}$$

Real world devices will exhibit some loss due to finite winding wire resistance, nonideal behavior of the magnetic circuit and the like. Instead of a power dissipation rating, power transformers are given a VA (volt-amps) rating, typically thought of as secondary voltage times maximum secondary current. Transformers like the ones pictured in Figure 10.4.1 have ratings in the range of 10 to 30 VA. Generally, the higher the VA rating, the larger and heavier the transformer. On the other hand, applications using signal transformers in communications systems tend to focus on the turns ratio and not on the VA rating as the associated voltages and currents tend to be small. Their goal generally is to increase the amplitude of a signal voltage.

Also, because there is no direct electrical connection between the primary and secondary, the transformer may be used solely for isolation. This is a safety feature. If $N = 1$, then there is no change in the voltage being presented to the device, ideally.

Example 10.4.1

A microphone feeds a small transformer which is used as the first stage of a preamplifier. The transformer has a primary winding of 50 turns and a secondary winding of 250 turns. Assuming an ideal transformer, determine the output voltage of the transformer if the signal from the microphone is 35 microvolts.

As this is an ideal transformer, the turns ratio indicates how much the input voltage is scaled. First we use Equation 14.4.1 to determine the turns ratio.

$$N = \frac{N_P}{N_S}$$

$$N = \frac{50}{250}$$

$$N = 0.2$$

Now we use Equation 14.4.2 to determine the secondary voltage.

$$V_S = \frac{V_P}{N}$$

$$V_S = \frac{35\mu V}{0.2}$$

$$V_S = 175\mu V$$

Power applications will make use of N to scale the voltage in order to create a more appropriate or efficient level. If the voltage is brought down $(N > 1)$, it is referred to as a step-down transformer. If $N < 1$, it is referred to as a step-up transformer. For example, a consumer electronics product may require a modest DC voltage for its operation, say 15 or 20 volts. To achieve this, the standard North American residential "wall" voltage of 120 volts may be reduced using a step-down transformer with $N = 5$. This would result in a secondary voltage of 24 volts that could then be rectified, filtered, and regulated to the desired DC level. Conversely, a step-up transformer could be used to create a much higher secondary potential (for example, in long distance power transmission to reduce I^2R losses). For some applications, split secondaries or multiple secondaries may be used. A center tap is relatively common and is used to divide the secondary into two equal voltage sections (e.g., a 24 volt center-tapped secondary would behave as two 12 volt secondaries in series). The schematic symbol for a transformer with a centertapped secondary is shown in Figure 10.4.4 .

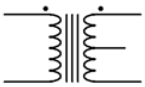


Figure 10.4.4 : Schematic symbol for a transformer with a centertapped secondary.

Multiple secondaries are also common and are more flexible than a center-tapped secondary. Multiple secondaries can be combined in series to increase the secondary voltage or combined in parallel to increase secondary current capacity (i.e, load current).

Example 10.4.2

The power transformer shown in Figure 10.4.5 has turns ratio of 10. If the primary is connected to a 120 volt source and the secondary is connected in series with a 32 Ω load resistor, determine the current through the load resistor. Assume this is an ideal transformer.

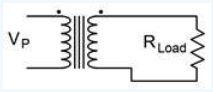


Figure 10.4.5 : Circuit for Example 10.4.2 .

The turns ratio indicates secondary voltage via Equation ???.

$$V_S = \frac{V_P}{N}$$

$$V_S = \frac{120V}{10}$$

$$V_S = 12V$$

Ohm's law can be used to find the secondary current.

$$I_S = \frac{V_S}{R_L}$$

$$I_S = \frac{12V}{32\Omega}$$

$$I_S = 375mA$$

As mentioned earlier, transformers work with AC voltages and currents. Unlike the consistent polarity of DC sources, AC (literally, Alternating Current) flips back and forth in voltage polarity and current direction. In order to determine the phase relationship between the primary and secondary, transformers use "dot notation". The dot indicates the positive polarity of voltage. This is illustrated in the circuit of Figure 10.4.6 .

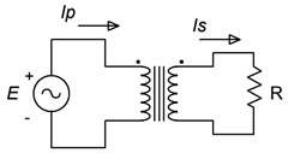


Figure 10.4.6 : Transformer dot convention.

On the primary side, the current flows into the dot and establishes the positive reference. In this case, the primary is seen as the load for the source, E. On the secondary side, current flows out of the dot and also notates the positive reference because the secondary is seen as the source for the load, R.

Impedance Matching

The third use of the transformer is impedance matching. Impedance is an AC phenomenon that consists of both resistance and reactance (an ohmic value associated with capacitors and inductors). The symbol for impedance is Z. Although impedance is measured in ohms, it differs from ordinary resistance in that it is a vector quantity and has a phase angle. For our present purposes, we can ignore this distinction.[1]

Because both the voltage and current are being scaled, the source's “view” of the load changes. The impedance seen by the source can be defined via Ohm's law as the voltage it produces divided by its output current.

$$Z_P = \frac{V_P}{I_P} \tag{14.4.5}$$

From Equation 14.4.2, we know that $V_P = V_S \cdot N$. Substituting this into Equation 14.4.5 yields:

$$Z_P = \frac{V_S N}{I_P} \tag{14.4.6}$$

From Equation 14.4.3, we know that $I_P = I_S/N$. Substituting this into Equation 14.4.6 yields:

$$Z_P = \frac{V_S N^2}{I_S} \tag{14.4.7}$$

By definition,

$$Z_S = \frac{V_S}{I_S} \tag{14.4.8}$$

Combining Equations 14.4.7 and 14.4.8 leaves us with:

$$Z_P = N^2 Z_S \tag{14.4.9}$$

Thus, the impedance seen by the source is the secondary impedance times the square of the turns ratio. This is called the reflected impedance.

Example 10.4.3

Assume that the transformer shown in Figure 10.4.6 has turns ratio of 5. If the primary is connected to an 20 volt source and the secondary is connected to an 8 Ω load resistor, determine the primary current and the reflected impedance. Assume this is an ideal transformer.

First we'll use Equation 14.4.9 to find the reflected impedance seen in the primary.

$$Z_P = N^2 Z_S$$

$$Z_P = 5^2 \times 8\Omega$$

$$Z_P = 200\Omega$$

Ohm's law can be used to find the primary current.

$$I_P = \frac{V_P}{R_P}$$

$$I_P = \frac{20V}{200\Omega}$$

$$I_P = 100mA$$

Here is where impedance matching comes in. In the preceding example, if the source, E, had a large internal resistance compared to the load resistance, R, and there was no transformer, there would be considerable loss in the system. For example, if the source impedance was 10 Ω and it was directly connected to the 8 Ω load without the transformer, the resulting voltage divider would cause a great deal of internal power dissipation in the source leading to inefficiency. On the other hand, in the given circuit with transformer, the source sees a reflected impedance of 200 Ω. The resulting voltage divider between 200 Ω and the 10 Ω internal resistance of the source will cause very little loss, yielding a much more efficient transfer.

References

[1]The concept of impedance is covered in great detail in the companion text, AC Electrical Circuit Analysis, another free OER title.

This page titled 14.4: Transformers is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.4: Transformers** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

14.5: Summary

This chapter has extended the material presented in Chapter 9 on inductance into the realm of magnetic circuits. We began with the concept of electromagnetic induction, which in simple terms states that if a conductor experiences changing magnetic lines of force, then a voltage will be induced in that conductor. This concept is exploited by an array of different devices including electric motors, generators, relays, loudspeakers, microphones, and other transducers and sensors. For example, dynamic loudspeakers and microphones both make use of a coil of wire suspended in a fixed magnetic field. In the case of the loudspeaker, the signal fed into the coil creates a changing magnetic field that interacts with the permanent magnet's field and causes an attached diaphragm to vibrate against the air, which in turn creates sound. A dynamic microphone works in an opposite manner. Sound waves cause a diaphragm to vibrate back and forth. This diaphragm is attached to a coil, and the motion of the coil within the permanent magnet's field causes a voltage to be induced in the coil. This voltage can then be amplified by electronic circuitry.

Magnetic circuits are constructed by creating one or more coils of wire wrapped around a core that is made of a ferromagnetic material such as steel. A key part of analyzing magnetic circuits is Hopkinson's law (also called Rowland's law), which is the magnetic circuit version of Ohm's law. In this analogy, magnetic flux is likened to current flow, magnetic reluctance stands in for resistance, and magnetomotive force is analogous to electromotive force (i.e., voltage). Further, the magnetomotive force is the product of the number of turns in a coil and the current through said coil, or NI. When current is passed through the coil, a magnetizing force, H, and an associated magnetic flux are created. Given this, the core material will have a corresponding flux density, B. The linkage between B and H is found through a BH curve for that particular material. BH curves are non-linear for ferromagnetic materials. They also exhibit hysteresis, meaning that the recent operational history of the material plays a role in its current state. Every element of the core has a certain reluctance and thus exhibits an analogous “voltage drop”. These “drops” must add up to the NI “rises” of the coil(s), in a KVL analogy.

Transformers use one or more primary coils to transform an input voltage into one more output voltages appearing on secondary coils. The ratio between the number of turns of the primary coil to the number of turns of the secondary coil is called the turns ratio, N. The secondary voltage is equal to the primary voltage divided by N while the secondary current is equal to the primary current times N. If N is greater than one, we have a step-down transformer (reducing the voltage), and if N is less than one, we have a step-up transformer (increasing the voltage). Ideally, transformers do not dissipate power. Thus, they do not have a power rating, but instead have a VA (volt-amps) rating to indicate their capacity. Transformers can also be used for impedance matching to increase the efficiency of a system. The impedance seen by the source on the primary is equal to the secondary impedance times the square of the turns ratio. This is called the reflected impedance.

Review Questions

1. Describe the concept of electromagnetic induction.
2. Describe Ohm's law for magnetic circuits (Hopkinson's or Rowland's law).
3. Describe hysteresis.
4. Define residual magnetism and coercivity.
5. Outline the operation of a relay.
6. What are some of the uses of transformers?
7. What is turns ratio? How does it affect voltage and current in the primary and secondary?
8. What is the difference between a step-up and a step-down transformer?
9. What is reflected impedance?

This page titled 14.5: Summary is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.5: Summary** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

14.6: Exercises

Analysis

1. In the magnetic circuit shown in Figure 10.6.1 , assume the cross section is 1 cm by 1 cm with a path length of 8 cm. The entire core is made of sheet steel and there are 100 turns on the winding. Determine the current to establish a flux of 8E−5 webers.

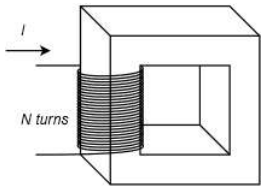


Figure 10.6.1

2. Repeat problem 1 using cast steel for the core.

3. Given the core shown in Figure 10.6.1 , assume the cross section is 2 cm by 2 cm with a path length of 10 cm. The entire core is made of cast steel and there are 200 turns on the winding. Determine the current to establish a flux of 4E−4 webers.

4. Repeat problem 3 using sheet steel for the core.

5. In the magnetic circuit shown in Figure 10.6.1 , assume the cross section is 1 cm by 1 cm with a path length of 8 cm. The entire core is made of sheet steel. Determine the number of turns required to establish a flux of 8E−5 webers given a current of 50 mA.

6. Given the core shown in Figure 10.6.1 , assume the cross section is 2 cm by 2 cm with a path length of 10 cm. The entire core is made of sheet steel Determine the number of turns required to establish a flux of 4E−4 webers given a current of 200 mA.

7. In the magnetic circuit shown in Figure 10.6.2 , assume the cross section is 1 cm by 1 cm. Section A is sheet steel with a path length of 6 cm. Section B is cast steel with a length of 2 cm. There are 500 turns on the winding. Determine the current to establish a flux of 5E−5 webers.

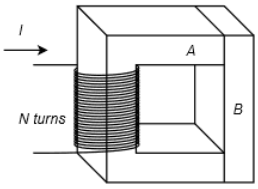


Figure 10.6.2

8. In the magnetic circuit shown in Figure 10.6.2 , assume the cross section is 1 cm by 1 cm. Section A is sheet steel with a path length of 6 cm. Section B is cast steel with a length of 2 cm. Determine the number of turns on the coil to establish a flux of 6E−5 webers with a current of 50 mA.

9. A transformer is shown in Figure 10.6.3 , assume the cross section is 5 cm by 5 cm. The core is sheet steel with a path length of 20 cm. $N1$ is 500 turns and $N2$ is 200 turns. Determine the secondary current $(I2)$ if a primary current $(I1)$ of 1 amp establishes a flux of 1.5E−3 webers.

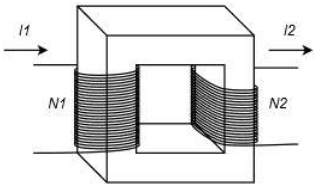


Figure 10.6.3

10. Given the same conditions of problem 9, alter the secondary turns $(N2)$ so that the secondary current $(I2)$ is 3 amps given the original primary current of 1 amp.

11. In general, how would the performance noted in problem 9 change if cast steel was substituted for sheet steel?

12. Given the results of problems 9 through 11, what does the ratio of $N1$ to $N2$ represent in terms of idealized performance, and what steps should be taken to make the transformer operate as close to ideal as possible?

13. Given the magnetic circuit shown in Figure 10.6.4 , assume the cross section is 1 cm by 2 cm with a path length of 6 cm. The entire core is made of sheet steel with the exception of a 1 mm air gap. Determine the current required to establish a flux of 4E−4 webers if $N = 1000$ turns.

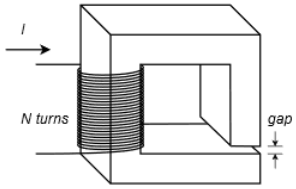


Figure 10.6.4

14. Using the data given in problem 13, determine the number of turns required to establish the same flux when the input current is 200 mA.

15. An ideal transformer has a 6:1 voltage step-down ratio. If the primary is driven by 24 VAC and the load is 100 Ω, determine the load voltage and current, and the primary side current.

16. A 120 VAC transformer is specified as having a 36 volt center-tapped secondary. If each side of the secondary is connected to its own 50 Ω load, determine the load currents and the primary side current.

17. An ideal transformer has a 12:1 voltage step-down ratio. If the secondary is connected to a 10 Ω load, what impedance is seen from the primary side?

18. An ideal transformer has a 1:5 voltage step-up ratio. If the secondary is connected to a 2 Ω load, what impedance is seen from the primary side?

Design

19. A step-down transformer with $N = 8$ has a 15 volt RMS secondary which is connected to a load with an effective value of 5 Ω. Determine the minimum acceptable VA rating of the transformer.

20. A step-up transformer with $N = .5$ is driven from a 120 VAC source. The secondary is connected to a load with an effective value of 150 Ω. Determine the minimum acceptable VA rating of the transformer.

Challenge

21. A transformer specified as having a 120 VAC primary with an 18 volt secondary is accidentally connected backwards, with its secondary connected to the source and its primary connected to a 16 Ω load. Determine the load current in both the normal and reversed connections. Also determine the required transformer VA rating for both connections.

22. Consider the distributed public address system for an airport as shown in Figure 10.6.5 . It consists of an audio power amplifier with a nominal 70 volt RMS output that is connected to four remote loudspeakers, each separate from the others and some 150 meters away from the amplifier. Each loudspeaker assembly includes a 10:1 voltage step-down transformer that feeds the loudspeaker impedance of 8 Ω (resistive) off its secondary. These four lines are fed in parallel by the amplifier. Determine the power delivered to each loudspeaker and the total current delivered by the power amplifier. Assume the transformers are ideal and ignore any cable resistance.

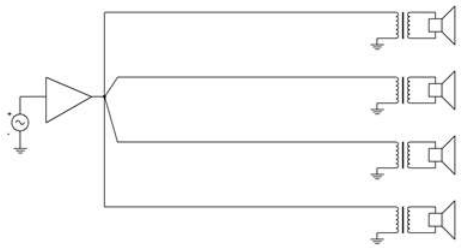


Figure 10.6.5

23. Continuing with the preceding problem, assume that the wiring connecting each transformer back to the amplifier is AWG 22. Determine the power lost in each of the 150 meter long sections of dual cable. Further, suppose the system is reconfigured without the transformers and the output of the amplifier is lowered to 7 volts RMS to compensate. Determine the power lost in each of the cable feeds under the new configuration.

Simulation

24. Use a transient analysis to verify the results of problem 15.

25. Use a transient analysis to verify the results of problem 16.

This page titled 14.6: Exercises is shared under a CC BY-NC-SA license and was authored, remixed, and/or curated by James M. Fiore.

- **10.6: Exercises** by James M. Fiore is licensed CC BY-NC-SA 4.0. Original source: http://www.dissidents.com/resources/DCElectricalCircuitAnalysis.pdf.

Index

B

Bode plot
7.3: Bode Plots

K

Kirchhoff's Current Law
2.3.4: Kirchhoff's Current Law

N

negative feedback
7.9.3: Basic Concepts

Glossary

Sample Word 1 | Sample Definition 1

Detailed Licensing

Overview

Title: Circuits and Devices

Webpages: 174

Applicable Restrictions: Noncommercial

All licenses found:

- CC BY-NC-SA 4.0: 91.4% (159 pages)
- Undeclared: 8.6% (15 pages)

By Page

- Circuits and Devices - *Undeclared*
  - Front Matter - *Undeclared*
    - TitlePage - *Undeclared*
    - InfoPage - *Undeclared*
    - Table of Contents - *Undeclared*
    - Licensing - *Undeclared*
  - 1: Basic Concepts and Quantitites - *CC BY-NC-SA 4.0*
    - 1.1: Introduction - *CC BY-NC-SA 4.0*
    - 1.2: An Atomic Model - *CC BY-NC-SA 4.0*
    - 1.3: Charge and Current - *CC BY-NC-SA 4.0*
    - 1.4: Energy and Voltage - *CC BY-NC-SA 4.0*
    - 1.5: Power and Efficiency - *CC BY-NC-SA 4.0*
    - 1.6: Energy Cost and Battery Life - *CC BY-NC-SA 4.0*
    - 1.7: Resistance and Conductance - *CC BY-NC-SA 4.0*
    - 1.8: Instrumentation and Laboratory - *CC BY-NC-SA 4.0*
    - 1.9: Summary - *CC BY-NC-SA 4.0*
    - 1.10:Exercises - *CC BY-NC-SA 4.0*
  - 2: Fundamental Laws - *Undeclared*
    - 2.1: Ohms Law and Kirchhof's Voltage Law - *CC BY-NC-SA 4.0*
      - 2.1.1: Introduction - *CC BY-NC-SA 4.0*
      - 2.1.2: Conventional Current Flow and Electron Flow - *CC BY-NC-SA 4.0*
      - 2.1.3: The Series Connection - *CC BY-NC-SA 4.0*
      - 2.1.4: Combining Series Components - *CC BY-NC-SA 4.0*
      - 2.1.5: Ohm's Law - *CC BY-NC-SA 4.0*
      - 2.1.6: Kirchhoff's Voltage Law - *CC BY-NC-SA 4.0*
      - 2.1.7: Series Analysis - *CC BY-NC-SA 4.0*
      - 2.1.8: Potentiometers and Rheostats - *CC BY-NC-SA 4.0*
      - 2.1.9: Summary - *CC BY-NC-SA 4.0*
      - 2.1.10: Exercises - *CC BY-NC-SA 4.0*
    - 2.2: Network Topology- Nodes, Branches and Loops - *Undeclared*
    - 2.3: Kirchhof's Current Law - *CC BY-NC-SA 4.0*
      - 2.3.1: Introduction - *CC BY-NC-SA 4.0*
      - 2.3.2: The Parallel Connection - *CC BY-NC-SA 4.0*
      - 2.3.3: Combining Parallel Components - *CC BY-NC-SA 4.0*
      - 2.3.4: Kirchhoff's Current Law - *CC BY-NC-SA 4.0*
      - 2.3.5: Parallel Analysis - *CC BY-NC-SA 4.0*
      - 2.3.6: Current Limiting - Fuses and Circuit Breakers - *CC BY-NC-SA 4.0*
      - 2.3.7: Summary - *CC BY-NC-SA 4.0*
      - 2.3.8: Exercises - *CC BY-NC-SA 4.0*
    - 2.4: Series-Parallel Resistive Circuits - *CC BY-NC-SA 4.0*
      - 2.4.1: Introduction - *CC BY-NC-SA 4.0*
      - 2.4.2: Series-Parallel Connections - *CC BY-NC-SA 4.0*
      - 2.4.3: Simplifying Series-Parallel Components - *CC BY-NC-SA 4.0*
      - 2.4.4: Series-Parallel Analysis - *CC BY-NC-SA 4.0*
      - 2.4.5: Summary - *CC BY-NC-SA 4.0*
      - 2.4.6: Exercises - *CC BY-NC-SA 4.0*
  - 3: Circuit Simulations Using Computers - *CC BY-NC-SA 4.0*
  - 4: Analysis Theorems and Techniques - *CC BY-NC-SA 4.0*
    - 4.1: Introduction - *CC BY-NC-SA 4.0*
    - 4.2: Source Conversions - *CC BY-NC-SA 4.0*
    - 4.3: Superposition Theorem - *CC BY-NC-SA 4.0*
    - 4.4: Thévenin's Theorem - *CC BY-NC-SA 4.0*
    - 4.5: Norton's Theorem - *CC BY-NC-SA 4.0*

- 4.6: Maximum Power Transfer Theorem - *CC BY-NC-SA 4.0*
- 4.7: Advanced Topic- Delta-Y Conversions - *CC BY-NC-SA 4.0*
- 4.8: Summary - *CC BY-NC-SA 4.0*
- 4.9: Exercises - *CC BY-NC-SA 4.0*

- 5: Nodal and Mesh Analysis, Dependent Sources - *CC BY-NC-SA 4.0*
  - 5.1: Introduction - *CC BY-NC-SA 4.0*
  - 5.2: Nodal Analysis - *CC BY-NC-SA 4.0*
  - 5.3: Mesh Analysis - *CC BY-NC-SA 4.0*
  - 5.4: Dependent Sources - *CC BY-NC-SA 4.0*
  - 5.5: Summary - *CC BY-NC-SA 4.0*
  - 5.6: Exercises - *CC BY-NC-SA 4.0*
- 6: Capacitors and Inductors - *Undeclared*
  - 6.1: Capacitors - *CC BY-NC-SA 4.0*
    - 6.1.1: Introduction - *CC BY-NC-SA 4.0*
    - 6.1.2: Capacitance and Capacitors - *CC BY-NC-SA 4.0*
    - 6.1.3: Summary - *CC BY-NC-SA 4.0*
    - 6.1.4: Exercises - *CC BY-NC-SA 4.0*
  - 6.2: Inductors - *CC BY-NC-SA 4.0*
    - 6.2.1: Introduction - *CC BY-NC-SA 4.0*
    - 6.2.2: Inductance and Inductors - *CC BY-NC-SA 4.0*
    - 6.2.3: Summary - *CC BY-NC-SA 4.0*
    - 6.2.4: Exercises - *CC BY-NC-SA 4.0*
- 7: Advanced Topic- Operational Amplifiers - *CC BY-NC-SA 4.0*
  - 7.1: Introduction - *CC BY-NC-SA 4.0*
  - 7.2: The Decibel - *CC BY-NC-SA 4.0*
  - 7.3: Bode Plots - *CC BY-NC-SA 4.0*
  - 7.4: Combining the Elements - Multi-Stage Effects - *CC BY-NC-SA 4.0*
  - 7.5: The Differential Amplifier - *CC BY-NC-SA 4.0*
  - 7.6: Summary - *CC BY-NC-SA 4.0*
  - 7.7: Problems - *CC BY-NC-SA 4.0*
  - 7.8: Operational Amplifier Internals - *CC BY-NC-SA 4.0*
    - 7.8.1: Introduction to Operational Amplifiers - *CC BY-NC-SA 4.0*
    - 7.8.2: What Is An Op Amp? - *CC BY-NC-SA 4.0*
    - 7.8.3: Simple Op Amp Comparator - *CC BY-NC-SA 4.0*
    - 7.8.4: Op Amp Manufacture - *CC BY-NC-SA 4.0*
    - 7.8.5: Summary - *CC BY-NC-SA 4.0*
    - 7.8.6: Problems - *CC BY-NC-SA 4.0*
  - 7.9: Negative Feedback - *CC BY-NC-SA 4.0*
    - 7.9.1: Introduction to Negative Feedback - *CC BY-NC-SA 4.0*
    - 7.9.2: What Negative Feedback Is and Why We Use It - *CC BY-NC-SA 4.0*
    - 7.9.3: Basic Concepts - *CC BY-NC-SA 4.0*
    - 7.9.4: The Four Variants of Negative Feedback - *CC BY-NC-SA 4.0*
    - 7.9.5: Limitations On The Use of Negative Feedback - *CC BY-NC-SA 4.0*
    - 7.9.6: Summary - *CC BY-NC-SA 4.0*
    - 7.9.7: Problems - *CC BY-NC-SA 4.0*
  - 7.10: Basic Op Amp Circuits - *CC BY-NC-SA 4.0*
    - 7.10.1: Introduction - *CC BY-NC-SA 4.0*
    - 7.10.2: Inverting and Noninverting Amplifiers - *CC BY-NC-SA 4.0*
    - 7.10.3: Single Supply Biasing - *CC BY-NC-SA 4.0*
    - 7.10.4: Current Boosting - *CC BY-NC-SA 4.0*
    - 7.10.5: Summary - *CC BY-NC-SA 4.0*
    - 7.10.6: Problems - *CC BY-NC-SA 4.0*
  - 7.11: Integrators and Differentiators - *CC BY-NC-SA 4.0*
    - 7.11.1: Introduction - *CC BY-NC-SA 4.0*
    - 7.11.2: Integrators - *CC BY-NC-SA 4.0*
    - 7.11.3: Differentiators - *CC BY-NC-SA 4.0*
    - 7.11.4: Analog Computer - *CC BY-NC-SA 4.0*
    - 7.11.5: Alternatives to Integrators and Differentiators - *CC BY-NC-SA 4.0*
    - 7.11.6: Extended Topic- Other Integrator and Differentiator Circuits - *CC BY-NC-SA 4.0*
    - 7.11.7: Summary - *CC BY-NC-SA 4.0*
    - 7.11.8: Problems - *CC BY-NC-SA 4.0*
- 8: Transient Behavior- First and Second Order Circuits - *Undeclared*
  - 8.1: Introduction - *Undeclared*
  - 8.2: Initial and Steady-State Analysis of RC Circuits - *CC BY-NC-SA 4.0*
  - 8.3: Transient Response of RC Circuits - *CC BY-NC-SA 4.0*
  - 8.4: Initial and Steady-State Analysis of RL Circuits - *CC BY-NC-SA 4.0*
  - 8.5: Transient Response of RL Circuits - *CC BY-NC-SA 4.0*
  - 8.6: Initial and Steady-State Analysis of RLC Circuits - *CC BY-NC-SA 4.0*
  - 8.7: Summary - *CC BY-NC-SA 4.0*
  - 8.8: Exercises - *CC BY-NC-SA 4.0*
- 9: AC Signal Fundamentals - *CC BY-NC-SA 4.0*
  - 9.1: Introduction - *CC BY-NC-SA 4.0*
  - 9.2: Sinusoidal Waveforms - *CC BY-NC-SA 4.0*

- 9.3: Basic Fourier Analysis - *CC BY-NC-SA 4.0*
- 9.4: Complex Numbers - *CC BY-NC-SA 4.0*
- 9.5: Reactance and Impedance - *CC BY-NC-SA 4.0*
- 9.6: Phasor Diagrams - *CC BY-NC-SA 4.0*
- 9.7: Summary - *CC BY-NC-SA 4.0*
- 9.8: Exercises - *CC BY-NC-SA 4.0*

10: Series RLC Circuits - *CC BY-NC-SA 4.0*

- 10.1: Introduction - *CC BY-NC-SA 4.0*
- 10.2: The Series Connection - *CC BY-NC-SA 4.0*
- 10.3: Series Impedance - *CC BY-NC-SA 4.0*
- 10.4: Series Circuit Analysis - *CC BY-NC-SA 4.0*
- 10.5: Concerning Practical Inductors - *CC BY-NC-SA 4.0*
- 10.6: Summary - *CC BY-NC-SA 4.0*
- 10.7: Exercises - *CC BY-NC-SA 4.0*

11: Parallel RLC Circuits - *CC BY-NC-SA 4.0*

- 11.1: Introduction - *CC BY-NC-SA 4.0*
- 11.2: The Parallel Connection - *CC BY-NC-SA 4.0*
- 11.3: Parallel Impedance - *CC BY-NC-SA 4.0*
- 11.4: Parallel Circuit Analysis - *CC BY-NC-SA 4.0*
- 11.5: Summary - *CC BY-NC-SA 4.0*
- 11.6: Exercises - *CC BY-NC-SA 4.0*

12: AC Circuit Analysis Theorems and Techniques - *CC BY-NC-SA 4.0*

- 12.1: Introduction - *CC BY-NC-SA 4.0*
- 12.2: Source Conversions - *CC BY-NC-SA 4.0*
- 12.3: Superposition Theorem - *CC BY-NC-SA 4.0*
- 12.4: Thévenin's and Norton's Theorems - *CC BY-NC-SA 4.0*
- 12.5: Maximum Power Transfer Theorem - *CC BY-NC-SA 4.0*
- 12.6: Advanced Topic- Delta-Y (Pi-T) Conversions - *CC BY-NC-SA 4.0*
- 12.7: Summary - *CC BY-NC-SA 4.0*
- 12.8: Exercises - *CC BY-NC-SA 4.0*

13: AC Power - *CC BY-NC-SA 4.0*

- 13.1: Introduction - *CC BY-NC-SA 4.0*
- 13.2: Power Waveforms - *CC BY-NC-SA 4.0*
- 13.3: Power Triangle - *CC BY-NC-SA 4.0*
- 13.4: Power Systems - *CC BY-NC-SA 4.0*
- 13.5: Summary - *CC BY-NC-SA 4.0*
- 13.6: Exercises - *CC BY-NC-SA 4.0*

14: Advanced Topic- Magnetic Circuits and Transformers - *CC BY-NC-SA 4.0*

- 14.1: Introduction - *CC BY-NC-SA 4.0*
- 14.2: Electromagnetic Induction - *CC BY-NC-SA 4.0*
- 14.3: Magnetic Circuits - *CC BY-NC-SA 4.0*
- 14.4: Transformers - *CC BY-NC-SA 4.0*
- 14.5: Summary - *CC BY-NC-SA 4.0*
- 14.6: Exercises - *CC BY-NC-SA 4.0*

Back Matter - *Undeclared*

- Index - *Undeclared*
- Glossary - *Undeclared*
- Detailed Licensing - *Undeclared*