Electronic Circuit Design Using Resistors

Introduction

• To get started with electronic circuits I start with simple resistor only circuits
• Hopefully you will be able to recall some knowledge from previous high school or university course work
  – Think back to a physics course you may have taken
• What is a resistor anyway?
  – It is a circuit element that resists the flow of current
  – It usually made of a carbon composition material, e.g., a carbon film on an insulating substrate


• Ohm’s law is the starting point
Ohms Law

• The unit for resistance is ohm, where one ohm is the resistance the allows one amp of current to flow and produce a voltage (potential) drop of one volt

• As a mathematical expression Ohm’s law states that

\[ V = I \cdot R \]  (4.1)

• In circuit schematic form

\[ \text{Note: By convention current flow is always from plus to minus even though electrons are flowing in the opposite direction} \]

\[ V = I \cdot R \]

\[ R = \frac{V}{I} \]

\[ I = \frac{V}{R} \]

• Notes: I am using conventional current flow, which is from plus to minus across the device

• Electrons are actually flowing in the opposite direction

• Ohm’s law actually can be written in three equivalent forms, giving rise to a different solution needs

• \[ V = I \cdot R \]: The voltage across a resistor is the product of the current passing through the resistor times the resistance

• \[ R = \frac{V}{I} \]: The resistance is equal to the voltage across the resistor divided by the current passing through it; note in this course we always have \( R \geq 0 \)

• \[ I = \frac{V}{R} \]: The current passing through the resistor is the voltage across the resistor divided by the resistance
• In words, Ohm’s law says that the voltage drop across the resistor is always the resistance $R$ times the current $I$ passing through the resistor.

**Example 4.1:** Build an LTspice model to verify Ohm’s law

• Build the exact model described in the definition.

• Run the simulation:

![LTspice model](image)

Example 4.2: Unknown resistor

• An unknown resistance is in circuit with 1.25 V across it and 3 mA of current passing through it.

• The unknown resistance must be

$$ R = \frac{V}{I} = \frac{1.25 \text{ V}}{0.003 \text{ A}} = 416.7 \Omega \quad (4.2) $$

where the symbol for ohm is capital omega, $\Omega$.

Example 4.3: Unknown current

• Consider a 10 k$\Omega$ resistor has a voltage drop of 3.4 V across
it

• Find the current passing through the resistor

\[ I = \frac{V}{R} = \frac{3.4 \text{ v}}{10 \text{ k}\Omega} = \frac{3.4}{10000} = 340\mu\text{a} \]  (4.3)

---

**Example 4.4:** An unknown voltage

• Suppose a 1500 ohm resistor is known to have 1.2 ma of current passing through it

• Find the voltage drop across the resistor

\[ V = I \cdot R = 1.2 \times 10^{-3} \cdot 1500 = 1.8 \text{ v} \]  (4.4)

---

**Extending Ohms Law**

• In this section I will extend Ohm’s law to some useful results
  – Series connection of resistors
  – Parallel connection of resistors
  – Voltage divider

• Along the way Kirchoff’s current and voltage laws will be introduced
Kirchoff’s Laws

Current Law (KCL)

Entering or leaving sum to zero

The current entering any junction is equal to the current leaving that junction. \( i_2 + i_3 = i_1 + i_4 \)

Voltage Law (KVL)

The sum of all the voltages around a loop is equal to zero.

\[ v_1 + v_2 + v_3 - v_4 = 0 \]

• Consider a series connection of three resistors:

• Suppose the current flowing through the resistors is \( I \)
• The voltage drop across all three resistors is

---

\[ V = V_1 + V_2 + V_3 \]
\[ = (IR_1 + IR_2 + IR_3) = I(R_1 + R_2 + R_3) \]  \hspace{1cm} (4.5)

so

\[ R_{\text{eq}} = \frac{V}{I} = \frac{I(R_1 + R_2 + R_3)}{I} = R_1 + R_2 + R_3 \] \hspace{1cm} (4.6)

**Summary:** Resistors add in series

- Consider a parallel connection of three resistors:

\[ \begin{align*}
I &= I_1 + I_2 + I_3 \\
V &= \frac{V}{R_1} + \frac{V}{R_2} + \frac{V}{R_3} = V \left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}\right) \end{align*} \] \hspace{1cm} (4.7)

so
Extending Ohms Law

\[ R_{eq} = \frac{V}{I} = \frac{V}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}} \]  

(4.8)

**Summary**: The equivalent resistance for resistors in parallel is as given by (4.8)

– Common notation is say \( R_{eq} = R_1 \| R_2 \| R_3 \), etc.

**Special Case**: For just two resistors in parallel you have

\[ R_{eq} = \frac{1}{\frac{1}{R_1} + \frac{1}{R_2}} = \frac{R_1R_2}{R_1 + R_2} \]  

(4.9)

**Voltage Divider**

– Using Ohm’s law it is easy to establish some simple circuit formulas

– One such result is the voltage divider:

![Voltage Divider Diagram]

• The voltage drop across the two resistors in series is the total voltage \( V \) (as shown here 5v)
• The current flowing through the resistors is

\[ I = \frac{V}{R_1 + R_2} \]  

(4.10)

• The voltage across \( R_2 \) is just \( I \times R_2 \), so

\[ V_2 = \frac{V}{R_1 + R_2} \cdot R_2 = V \cdot \frac{R_2}{R_1 + R_2} \]  

(4.11)

• For the numbers given in the figure

\[ V_2 = 10\,\text{V} \cdot \frac{5\,\text{k}\Omega}{(10 + 5)\,\text{k}\Omega} = 3.333\,\text{V} \]  

(4.12)

**Example 4.5: A Three Resistor Voltage Divider**

• This circuit was simulated in LTspice (below) and then analyzed theoretically

![LTspice simulation](image)

\[ R_{\text{in}} = \frac{V_1}{I} = \frac{5}{0.00022723} = 22000\,\Omega \]

• The theoretical resistance looking into \( R_1 \) is a series/parallel combination. i.e.,
Extending Ohms Law

\[ R_{\text{in}} = R_1 + (R_2 \parallel R_3) = R_1 + \frac{R_2 R_3}{R_2 + R_3} \]

\[ = 10k + \frac{20k \cdot 30k}{20k + 30k} = 22k\Omega \quad (4.13) \]

• The results are summarized below

| Simulated R_input: | 22004.14 ohms |
| Theory R_input:    | 22000.00 ohms |
| Simulated Voltage Divider Output: | |
| V(Out)             | 2.73 volts   |
| Theory Voltage Divider Output: | |
| V(Out)             | 2.73 volts   |

• When the operating point dialog is dismissed you can hover over nodes and branches to display values, e.g.:
Thevenin’s Theorem

• There are times when it is convenient to simplify a circuit into the Thevenin Equivalent circuit
  – One case in point is in designing the bias network for a transistor amplifier
  – Another case is shown by example below

• Consider the following:

\[
V_T = V \cdot \frac{R_2}{R_1 + R_2}, \quad R_T = \frac{R_1 R_2}{R_1 + R_2}
\]

• The voltage source and resistor combination (can be far more complicated than shown here) is replaced by a single voltage source \(V_T\) and series resistor \(R_T\)
  – The voltage \(V_T\) is the *open circuit voltage* at the load terminals without the load connected
  – The series resistance is the equivalent resistance seen looking to the left of the load terminals with the voltage source(s) short circuited (current sources if present are open circuited)

• In this example the open circuit voltage is the voltage divider
result

\[ V_T = V \cdot \frac{R_2}{R_1 + R_2} \]  \hspace{1cm} (4.14)

- The equivalent resistance with \( V \) short circuited is just \( R_1 || R_2 \) or

\[ R_T = \frac{R_1 R_2}{R_1 + R_2} \]  \hspace{1cm} (4.15)

**Example 4.6: Rework of 3 Resistor Divider**

- In the above the original \( V_1, R_1, \) and \( R_2 \) constitute the circuit to be replaced by a Thevenin equivalent
- When the load of \( R_3 \) is attached \( V_{out} \) drops and current now flows through both \( R_2 \) and \( R_3 \)
- The equivalent circuit properly reflects this by choosing \( V_T \) according to (4.5) and \( R_T \) according to (4.6)
Running the LTspice operating point analysis yields:

The Potentiometer (variable resistor)

A popular form of variable resistor is the potentiometer (pot)

As seen above the pot has three terminals

A mechanical wiper divides $R$ into two resistors $R_a$ and $R_b$
• Turning the shaft moves the division point from one end to the other across $R$

**Working with Actual Resistors**

• To work with real resistors you need to be aware of:
  – The resistor color code
  – The resistor value tolerance
  – The resistor power rating
Resistor Color Code

- In this class the 4-band code will be used

- **Note**: The chart above shows you how to convert to the newer 5-band code

Resistor Tolerance

- The last band of the color code indicates the value tolerance
- This is a statistical measure of variability associated with the labeled value
  - A 10% tolerance means that the true resistance value lies

somewhere on the interval
\[ R - 0.1R \leq R_{\text{true}} \leq R + 0.1R \]
\[ 0.9R \leq R_{\text{true}} \leq 1.1R \] (4.16)

• A shorthand notation for tolerance is \( R_{\text{true}} = R \pm 10\% \)

**Resistor Power Rating**

• The DC power dissipated by a resistor is given by

\[ P = V \times I \text{ watts} \]

\[ = V \times \frac{V}{R} = \frac{V^2}{R} \text{ watts} \] (4.17)

\[ = (I \cdot R) \times I = I^2 R \text{ watts} \]

• The carbon film resistors used in this class are typically rates at 0.25 watts

• If this value is exceeded the resistor will become very warm (hot) and may be damaged or destroyed!
  – Damaged means the value may change as a result of the heating

• *Power resistors* are available in many forms

• Consider the 50w unit below:

Handles 50w when heat sinked
Waveforms in Resistor Circuits

- Up to this point I have only considered a DC voltage source connected to a resistor network.
- In practice both DC and time varying signals are present.
- Recall from Chapter 3 the sinusoidal signal and its spectrum.
- In LTspice full waveform circuit simulation is possible using the directive `.tran` for transient or time domain simulation and converting the source from DC to `SINE()`
  - Operating point, `.op`, (DC) simulation was easy, `.tran` is much more involved.

To bring up this dialog click `Edit Simulation Command` under the `Simulate top level` menu.

Various spice simulation directives, i.e., the `.commands`

To simply run a 4ms simulation:

```
.tran 4m
```

Directive seen on the schematic.
Moving on, now I modify the voltage source to produce a sinusoid signal:

The schematic for the 3 resistor voltage divider now appears as:

- When you run this simulation a plot window opens and in the schematic you can probe the circuit to visualize waveforms (voltage and current) at the corresponding circuit test point.
- Circuit node labels may be added for convenience.
• Probing the input and output points of the circuit:

- The waveform cursor at the output is adjusted to show the peak signal value of approximately 544.6 mv (peak)
- Does this makes sense?
- The voltage divider action takes the 1v peak input and scales it by

\[
\frac{20k || 30k}{10k + 20k || 30k} = 0.54545 \quad (4.18)
\]

- So, the expected peak sinewave output from the simulator is

\[
v_{\text{out}}(t) = 0.5455 \sin[2\pi(1000)t] \quad (4.19)
\]

- The simulated results compare favorably
Hardware simulation using the Analog Discovery is next:

Using the Channel 1 waveform generator, create a 1 kHz sine-wave having amplitude of 1v peak.

Input (orange)  Output (blue)

Both readings a little, but ratio is good 0.5420

Spectrum a spike at 1 kHz
References


Appendix: LTspice

• Provide a few LTspice usage tips
• Describe common simulation types for this class