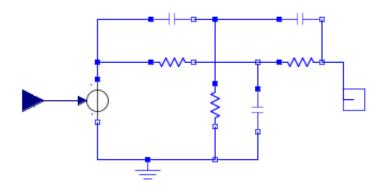# Nodal and Loop Analysis

The process of analyzing circuits can sometimes be a difficult task to do. Examining a circuit with the node or loop methods can reduce the amount of time required to get important information on a circuit. These analyses apply Kirchhoff's laws in a step-by-step approach to solve for unknown voltages and current. Once these values are obtained, further information such as electrical power can be analyzed.

This module introduces Kirchhoff's laws and their relation to circuit analysis. Different methods for analysis are explained, covering both node and mesh analysis. These processes are explained in detail to provide an understanding of how to analyze circuits.
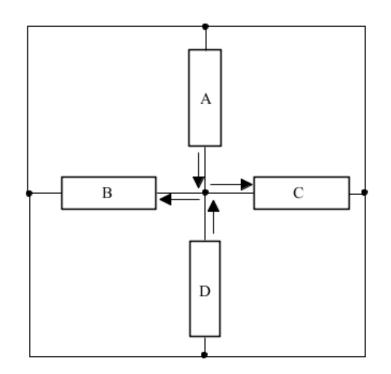
▼ Kirchhoff's Laws

In 1847, Gustav Kirchhoff formulated his voltage law and current law. These laws were derived from the conservation of charge and conservation of energy laws and applied to circuits. These laws are used to develop equations for circuit analysis.

▼ Kirchhoff's Current Law (KCL)

Kirchhoff current law states that the algebraic sum of all currents entering a node of a circuit is always zero. A node in a circuit is the place where circuit elements are connected together. The direction of the current must be considered during the analysis. If the current is entering the node, the current should be subtracted from the algebraic

sum. Likewise, if the current is directed away or out from a node then the current is said to be positive. Another simpler way to analyze the current is to set the sum of the currents directed away from the node equal to the sum of the currents directed towards the node.

The following circuit can be analyzed with KCL.

Figure 1: KCL Analysis of a Circuit

The node in the center of this circuit involves four currents. The sum of the two currents entering the node (the current from elements A and D) are equal to the two currents leaving the node (towards elements B and C). Analyzing the center node of this circuit shows
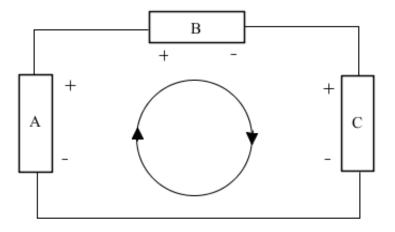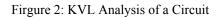
$$i_A + i_D = i_B + i_C$$

▼ Kirchhoff's Voltage Law (KVL)

Kirchhoff's voltage law states that the algebraic sum of the voltages around any loops in

a circuit is always zero. A loop in a circuit is any closed path along a circuit that does not encounter the same node more than once. The polarity of a voltage across an element changes the sign of the voltage in the sum of a loop. If analyzing the loop in a clockwise manner means that the $-$ side of the polarity of an element is encountered before the $+$, then the voltage should be subtracted. Similarily, if the voltage drops from a $+$ to a $-$ across an element, the voltage should be added.

The circuit in figure 2 consists of multiple elements. We can analyze it with KVL.

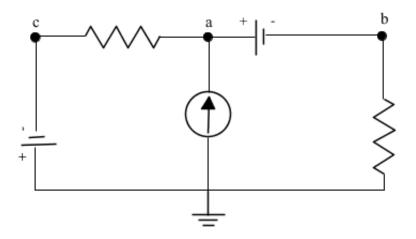Firgure 2: KVL Analysis of a Circuit

The loop analysis of this circuit element is in a clockwise direction. The loop encounters the negative polarity of A, the positive polarity of B and the positive polarity of C if analyzed in a clockwise direction. KVL on this circuit reveals the following equation.
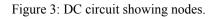
$$-v_A + v_B + v_D = 0$$

Methods of Analysis

Nodal Analysis

We use nodal analysis on circuits to obtain multiple KCL equations which are used to solve for voltage and current in a circuit. The number of KCL equations required is one less than the number of nodes that a circuit has. The extra node may be referred to as a

reference node. Usually, if a circuit contains a ground, whichever node the ground is connected to is selected as the reference node. This is used to find the voltage differences at each other node in the circuit with respect to the reference.

Figure 3: DC circuit showing nodes.

Ideally we set the voltage to 0 V at the reference node to simplify calculations, however it can be set to any value as long as the other nodes account for the different reference voltage. Solving the node equations can provide us with the node voltages.

The node equations are obtained by completing two things:

1. Express the current through an element in terms of the node voltages.

2. With the exception of the reference node, apply KCL to each other node in the circuit.

Figure 4 below shows an example of a DC circuit with current and voltage sources. It contains 3 nodes **a**, **b** and **c**, as well as the reference node at the grounded connection.
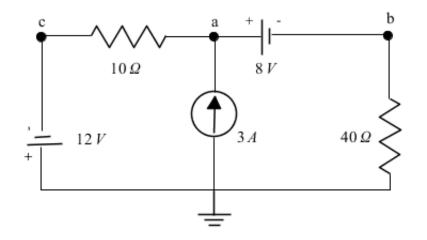
Figure 4: DC Circuit with Voltage and Current Sources

Here, node **c** is an example of a **supernode** which is a connection between two nodes via an independant or dependant voltage source. Because supernodes are connected to a voltage source we can find their voltage immediately. In this case, with the ground at $0\ V$, the voltage across the source will be $12 = 0 - v_c$, therefore $v_c = -12\ V$. Similarily, node **a** is related to node **b** as a supernode, $v_a = v_b + 8\ V$. We can substitute v_a and v_c into our KCL equations to solve for v_b.

Calculate the KCL equations at node **a** in figure 3. The current source is directing current into node **a**, and we will assume the current flows away from **a** towards node **b** and **c**. The KCL equation for node **a** is

$$3\ [\![A]\!] = i_b + i_c$$

The current across the 40 Ω resistor represented by the node voltages is found through Ohm's law as the potential across the element divided by the resistance. Note the assumed direction of the current to ensure the correct polarity of the difference in potential.

$$i_b = \frac{v_b - 0\ V}{40\ \Omega} = \frac{v_b}{40}$$

The current through node **c** will be equivalent to the potential across the 10 Ω resistor divided by the resistance.

$$i_c = \frac{v_a - v_c}{10\ \Omega}$$

Substituting in the currents i_b and i_c and using the equations for v_a and v_c, the KCL equation at node A is found to be

$$3 = \frac{v_b}{40} + \frac{\left(v_b + 8\right) - \left(-12\right)}{10}$$

Solving for v_b shows the voltage is 8 volts. Finally, the voltage at **a** is $v_a = v_b + 8$, so $v_a = 16$ volts.

▼ Loop/Mesh Analysis

Loop analysis is a special application of KVL on a circuit. We use a special kind of loop called a 'mesh' which is a loop that does not have any other loops inside of it. A mesh starts at a node and traces a path around a circuit, returning to the original node without hitting any nodes more than once. We can only apply mesh analysis to planar circuits, that is circuits without crossover connections. If a circuit cannot be redrawn without the intersecting disconnected lines then we cannot use mesh analysis.

Similar to nodal analysis, we want to obtain the mesh equations to be able to interpret the circuit. The mesh equations are obtained by

> 1. Applying Kirchhoff's voltage law (KVL) to each mesh in the circuit.
>
> 2. Express the voltages of elements in terms of the mesh currents.

The current along a mesh may not be uniform but this is accounted for by considering the current imposed by other meshes. Observe the circuit shown below.
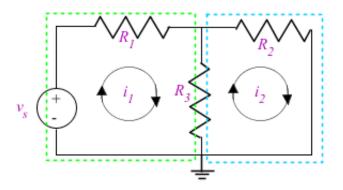
Figure 5: Mesh circuit

This circuit has 2 meshes in it, outlined in green and blue dotted lines. The currents i_1 and i_2 are the currents around a mesh. In places where the currents affect each other, such as R_3 in figure 5, take the sum of the currents but consider the directions. Here, if we examine the green mesh, the current through R_3 will be $i_1 - i_2$ because i_2 is going in the opposite direction as i_1.

The method behind mesh analysis is to examine the mesh in terms of the voltages of each element and express that with the mesh currents. Using Ohm's law $V = I \cdot R$, we can express the voltage across R_1 as $R_1 \cdot i_1$. Use this approach across a circuit applying KVL to find the mesh equations, then you can use these equations to solve for unknown current.

In the case where a dependent source is present in a circuit, the controlled current or voltage of that source must also be expressed by the mesh currents. Examine the circuit and find which part of the circuit controls the dependent source and express that voltage or current with the mesh currents that affect it.
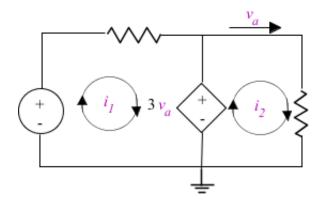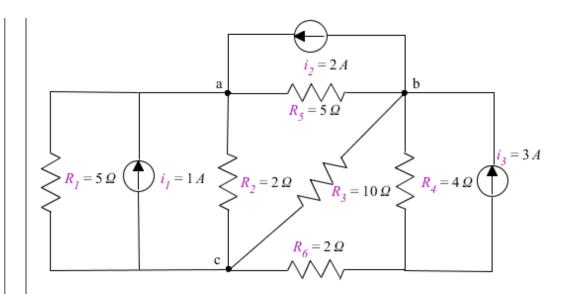
Figure 6: Circuit with dependent source

In this circuit, a dependent source generates voltage depending on the current at v_a. We want to express this in terms of the mesh currents. Examining the circuit shows that the current i_a is equal to the mesh current i_2. Therefore the dependent source will force $3 \cdot v_2$ volts into the system, and now mesh analysis can be performed to solve for the remaining unknowns.
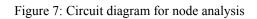
Both analyses are appropriate in most cases. Mesh analysis should not be used in instances where the circuit has a crossover. In this case, the nodal method should be used. A reliable way to determine which method to use comes from the number of equations that each will generate. It would be a better choice to use mesh analysis if the circuit contained more nodes than meshes, and if the opposite is true then use nodal analysis.

Examples with MapleSim

Example 1: Nodal Analysis of a Circuit

Problem Statement: Determine the node voltages for the circuit in the following figure when $i_1 = 1\,A$, $i_2 = 2\,A$, $i_3 = 3\,A$, $R_1 = 5\,\Omega$, $R_2 = 2\,\Omega$, $R_3 = 10\,\Omega$, $R_4 = 4\,\Omega$, $R_5 = 5\,\Omega$, and $R_6 = 2\,\Omega$.

Figure 7: Circuit diagram for node analysis

Analytical Solution

restart;

Data:

$i_1 := 1[\![A]\!]$:

$i_2 := 2[\![A]\!]$:

$i_3 := 3[\![A]\!]$:

$R_1 := 5[\![\Omega]\!]$:

$R_2 := 2[\![\Omega]\!]$:

$R_3 := 10[\![\Omega]\!]$:

$R_4 := 4[\![\Omega]\!]$:

$R_5 := 5[\![\Omega]\!]$:

$R_6 := 2[\![\Omega]\!]$:

Solution:

Let the voltages

v_a, v_b and v_c represent the voltages at nodes **a**, **b** and **c** respectively.

We will begin with the KCL equation for node **a**. The currents i_1 and i_2 are entering the node, and assume the currents through the resistors leading to the ground are leaving the node.

$$eq1 := -\left(\frac{v_a - v_c}{R_1}\right) + i_1 - \left(\frac{v_a - v_c}{R_2}\right) + i_2 - \left(\frac{v_a - v_b}{R_5}\right) = 0 :$$

Rearranging this by extracting the node voltages leaves

$$eq1 := \left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_5}\right) \cdot v_a - \left(\frac{1}{R_5}\right) \cdot v_b - \left(\frac{1}{R_1} + \frac{1}{R_2}\right) \cdot v_c = i_1 + i_2 :$$

Use the same method to solve for the node equation at **b**.

$$eq2 := -i_2 + \left(\frac{v_a - v_b}{R_5}\right) - \left(\frac{v_b - v_c}{R_3}\right) - \left(\frac{v_b}{R_4}\right) + i_3 = 0 :$$

Solving for the node voltages,

$$eq2 := -\left(\frac{1}{R_5}\right) \cdot v_a + \left(\frac{1}{R_3} + \frac{1}{R_4} + \frac{1}{R_5}\right) \cdot v_b - \left(\frac{1}{R_3}\right) \cdot v_c = i_3 - i_2 :$$

For node **c**, the KCL equation will be

$$eq3 := \left(\frac{v_a - v_c}{R_1}\right) + \left(\frac{v_a - v_c}{R_2}\right) + \left(\frac{v_b - v_c}{R_3}\right) - \left(\frac{v_c - 0}{R_6}\right) = -i_1 :$$

Again, rearranging this to isolate the coefficients of v_a, v_b and v_c will complete our matix.

$$eq3 := -\left(\frac{1}{R_1} + \frac{1}{R_2}\right) \cdot v_a - \left(\frac{1}{R_3}\right) \cdot v_b + \left(\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_6}\right) \cdot v_c = -i_1 :$$

This can be solved easily as a matrix with Maple using the *solve* command. Otherwise, use substitution and elimination with the KCL equations to solve for the values of the node voltages.

$$solve\left(\{eq1, eq2, eq3\}, \{v_a, v_b, v_c\}\right) = \left\{v_a = \frac{136}{19} \; [\![V]\!], v_b = \frac{96}{19} \; [\![V]\!], v_c = \frac{66}{19} \; [\![V]\!]\right\}$$

Thus the potential of the node voltages are v_a = 7.16 V, v_b = 5.05 V and v_c = 3.47 V.
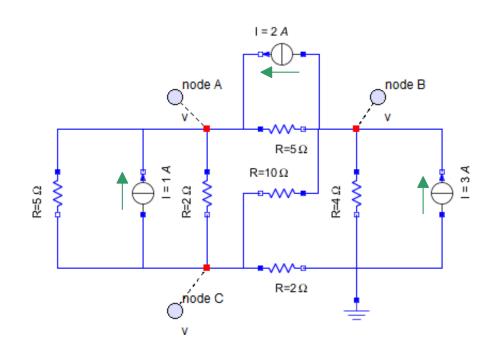
MapleSim Solution

Step 1: Insert Components

Drag the following components into a new workspace.

Component	Location
  Ground	Electrical > Analog > Common
  Resistor (6 required)	Electrical > Analog > Common
  Constant Current (3 required)	Electrical > Analog > Sources > Current

Step 2: Connect the components.

Connect the components as shown in the diagram below.

Figure 8: MapleSim model diagram

Note the direction of the current sources. Green arrows have been added to the image to emphasize this.

Step 3: Set up the resistors.

For each resistor,
1. Select the **Resistor** block. On the 'Inspector' tab, set the R parameter to the following values: $R_1 = 5\ \Omega$, $R_2 = 2\ \Omega$, $R_3 = 10\ \Omega$, $R_4 = 4\ \Omega$, $R_5 = 5\ \Omega$, and $R_6 = 2\ \Omega$.

Step 4: Set up the current sources.

For all three of the current sources,
1. Select the **Constant Current** block and on the 'Inspector' tab, set the I parameter to the following values: $i_1 = 1\ A$, $i_2 = 2\ A$, $i_3 = 3\ A$.

2. Connect probes to the proper locations and ensure the 'v' option is checked on the

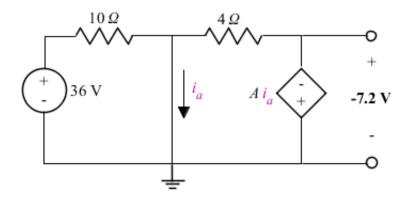'Inspector' tab to ensure the voltage at each spot will be measured.
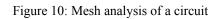
Step 4: Run Simulation

Run the simulation. Measure the value of the plots that the probes generate to obtain the values of the node voltages.

▶ **Results**

Example 2: Mesh Analysis of a Circuit

Problem Statement: Consider the following circuit. A voltmeter reads -7.2 volts across the dependent source. Find the gain *A* of the current-controlled voltage source.

Figure 10: Mesh analysis of a circuit

▼ **Analytical Solution**

restart;

Data:

$v_S := 36 [\![V]\!]$:

$R_1 := 10 [\![\Omega]\!]$:

$R_2 := 4 [\![\Omega]\!]$:

Solution:

First draw the meshes to be examined in the circuit. They are labelled in green with current i_1 and red with i_2 below.
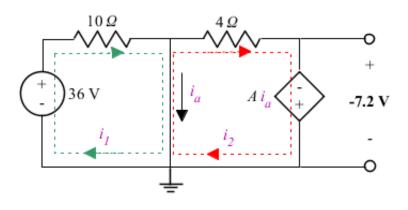

Figure 11: Label meshes

Consider the dependent source. It is supplying a voltage in the opposite direction to the reading from the voltmeter. From this, we can infer the voltage of the source element.

$A \cdot i_a = - (-7.2)$:

$eq1 := A \cdot i_a = 7.2 \, [\![\, V \,]\!]$:

The dependent current i_a is between both meshes. We can represent this current in terms of the mesh currents.

$i_a := i_1 - i_2$:

Now that we can represent the dependent source in terms of the mesh currents, apply KVL to obtain the equations. Mesh analysis on the green loop shows,

$eq2 := -v_s + R_1 \cdot i_1 = 0$:

Which simplifies to show the current i_1.

$i_1 = 3.6 \llbracket A \rrbracket$:

On the second mesh with the dependent source, the mesh equation will be,

$eq3 := R_2 \cdot i_2 - A \cdot i_a = 0$:

Of course we know the voltage of the dependent source so we can substitute this into the second mesh equation.

$R_2 \cdot i_2 = 7.2 \llbracket V \rrbracket$:
$i_2 = 1.8 \llbracket A \rrbracket$:

Using the values for i_1 and i_2 from the mesh equations, the dependent source gain can finally be found.

$A = \dfrac{7.2 \llbracket V \rrbracket}{i_a} = \dfrac{7.2}{i_1 - i_2} = 4$:

Therefore the voltage of the dependent source is 4 times greater than the current through i_a.

Alternatively we could use Maple's *solve* function to determine the currents and gain in one step.

$solve\left(\{eq1, eq2, eq3\}, \{i_1, i_2, A\} \right)$

$$\{A = 4.0 \, \llbracket \Omega \rrbracket, i_1 = 3.6 \, \llbracket A \rrbracket, i_2 = 1.8 \, \llbracket A \rrbracket \} \qquad \textbf{(3.2.1.1)}$$

MapleSim Solution

Step 1: Insert Components

Drag the following components into a new workspace.

Component	Location
Ground	Electrical > Analog > Common
Resistor (2 required)	Electrical > Analog > Common
Constant Voltage (2 required)	Electrical > Analog > Sources > Voltage

Step 2: Connect the components.

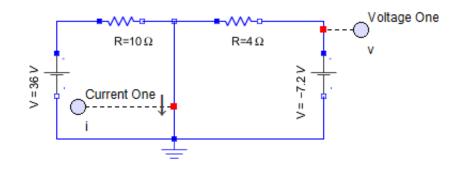Connect the components as shown in the diagram below.

Figure 12: MapleSim model

The current source on the right is replacing the dependent voltage because the question supplied the voltage across the element. It will be easier to use the known value.

Note that the direction of the probe arrow is dependent on the direction you moved to connect the nodes. For example, here the top node was connected to the bottom node instead of the bottom connecting to the top, so the arrow points downwards. It is an assumption made by the program.

Step 3: Set Up the Circuit

1. Set the resistance to **10 Ω** and **4 Ω** for R_1 and R_2 respectively. Select the **Resistor** block and change the parameter on the 'Inspector' tab.
2. Set the left voltage source at **36 V** on the 'Inspector' tab. Also, set the right voltage source to **-7.2 volts** and confirm the positive terminal is at the top of the circuit. This emulates the controlled voltage source.
3. Connect a **probe** along the middle connection between the resistors. On the 'Inspector' tab, deselect the voltage ' v ' and select the current ' i '. An arrow will show up at the probe designating which direction the program assumes the current will be travelling. If the current is negative, then it is travelling in the opposite direction of the arrow.

Step 3: Run Simulation

Run the simulation. The probe will return the value of i_a. Divide the voltage across the dependent source by the current i_a to get the magnitude of the gain.

Step 4: Check the Solution

To check your solution,
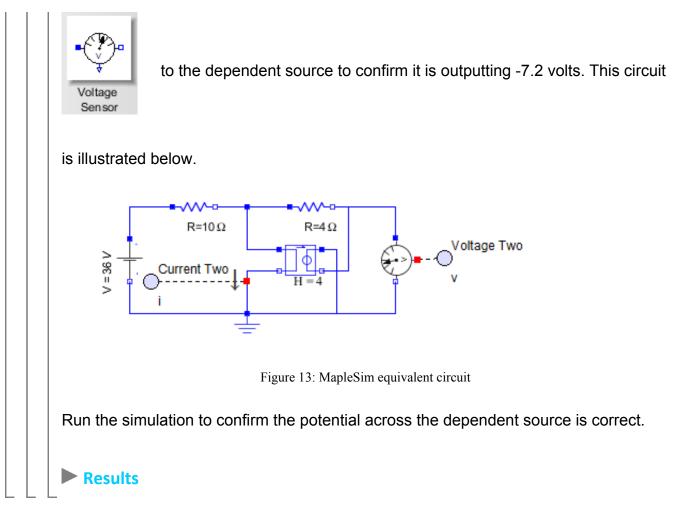1. Connect a current-controlled voltage source (Electrical > Analog > Sources >

Controlled > CCV) in place of the right voltage source. On the

'Inspector' tab, set the gain to 4.
2. Connect a voltmeter (Electrical > Analog > Sensors > Voltage Sensor)

Voltage Sensor

to the dependent source to confirm it is outputting -7.2 volts. This circuit

is illustrated below.

Figure 13: MapleSim equivalent circuit

Run the simulation to confirm the potential across the dependent source is correct.

▶ **Results**

References:

R. Dorf, J. Svoboda. "Introduction to Electric Circuits", 8th Edition. RRD Jefferson City, 2010, John Wiley and Sons, Inc.