ECE20007: Experiment 3

Nodal Analysis, Spice, and Equivalent Circuits

Ben Manning, Purdue University
Conner Hack, Editor
Last modified: September 11, 2023

---

1   Application

In order to analyze complex circuits, we need to understand basic circuit analysis, where the two main processes are known as nodal and mesh analysis. Node and mesh analysis allows us to take Ohm's Law and Kirchhoff's Laws and apply them to any type of circuit we want.

In electronics we have a lot of tools at our disposal. We have physical tools such as our multimeters and power supplies to monitor and manage our circuits in a controlled manner, and we also have software and simulations that make our lives easier. Since the rules of electronics don't change, we can develop software and programs that are able to analyze and evaluate circuits to much higher precision that we could ever do by hand.

Along with our tools, we also have a lot of techniques that we can use to make circuit analysis easier, the main ones being nodal analysis and developing equivalent circuits. Nodal analysis allows us to analyze a circuit at specific points, and will often allow us to simplify a circuit by being able to temporarily ignore more complex parts as long as we know the voltage and current operating at that location, or find patterns such as voltage dividers to make things easier. We can also simplify the entire circuit down to a single source and a single resistor, no matter the size or complexity. This is extremely useful when it comes to power systems and circuits with multiple power sources.

2   Experiment Purpose

The main purpose of this lab is to explain how even the most basic concepts in electronics can be extremely helpful for engineers. You will find that being able to do quick nodal analysis of simple circuits, such as voltage dividers and filters, will be very useful in the future. Along with learning the math at a more comfortable level, learning to simulate your experiments and projects will also help you make quick work of complex systems. Just like any other piece of software, there are pros and cons, and it takes time to learn the software properly.

3   General Circuit Analysis

There are two main ways that circuits are analyzed: node and mesh analysis. Both methods have their benefits and drawbacks, and it is ultimately up to the individual to decide which method they prefer to use.
This lab will focus on 2-node devices, which includes resistors and power sources. There are other types of devices that have 3 nodes, such as transistors. These require additional analysis, which will be discussed in future classes.

3.1   Nodal Analysis

Nodal circuit analysis relies on the idea of analyzing a circuit from nodes, or unique points in a circuit. A node can technically be put anywhere you want in a circuit, but you want a unique node i.e. a spot on the circuit where something will be changing, to effectively analyze a circuit. Places where nodes are considered unique: addition of a component (such as a a resistor or voltage source) and branches of a circuit (such as changing from series to parallel or vice versa).

Rules with nodes:

1. A circuit component is in series with another component if they share exactly one node.

2. A circuit component is in parallel with another component if they share two nodes.

Kirchhoff's Current Law (KCL), in its simplest form, tells us the current going into a node must equal the current leaving the node. If we have one wire getting split into three separate wires, the sum of the current in the three exiting wires will equal the current in the one entering wire.

3.2   Mesh Analysis

Mesh analysis, otherwise known as loop analysis, allows us to look at parts of a circuit as a series of different loops, or meshes, instead of nodes. Our goal here is to find closed loops, or an area in a circuit that is completely components and wires. Remembering that you can rearrange circuits to be more visually helpful can be very important here.

4   Guided Tutorial on Nodal Analysis and Spice

While previous circuits in the lab were not that complicated to analyze, as circuits grow in complexity, with many components in a wide variety of relationships, it is often best to leave a lot of the general analysis to a computer. SPICE (Simulation Program with Integrated Circuit Emphasis) is an open source circuit simulator that lets us quickly capture and analyze a wide variety of circuits. In ECE20007, we will be using Multisim, a cloud-based Spice simulator made by National Instruments, a major manufacturer of test equipment and instrumentation software. An account can be made and used for free at https://www.multisim.com. Another option is LTspice, a version of SPICE made by Analog Devices, a major semiconductor manufacturer. Searching for LTspice in your preferred search engine will likely give a download link in the first or second option.

Multisim and LTspice both have their pros and cons, and it ultimately depends on the user of which is preferred to be used. With Multisim being cloud based and having a nice user interface, it will work the same regardless of the operating system or internet browser the user is using. That said, Multisim is a proprietary system, and there are additional functions, such as different simulation options and additional circuit components, blocked by a paywall. This will not be an issue for this course, but may come as an issue as one moves forward in future labs and classes. LTspcie is an open-source platform that allows for users to design and implement functions that would work for them, but the user interface leaves a lot to be desired. The version of LTspice that is developed for Apple users is especially difficult to just pickup and use for the general user due to its dependence on hotkeys.

In this tutorial, we will be looking at the circuit in Figure 1. This circuit is made in Multisim, along with all of the images in this section. LTspice has similar options and procedures, but it will take time to get used to that specific user interface, just like any other application.
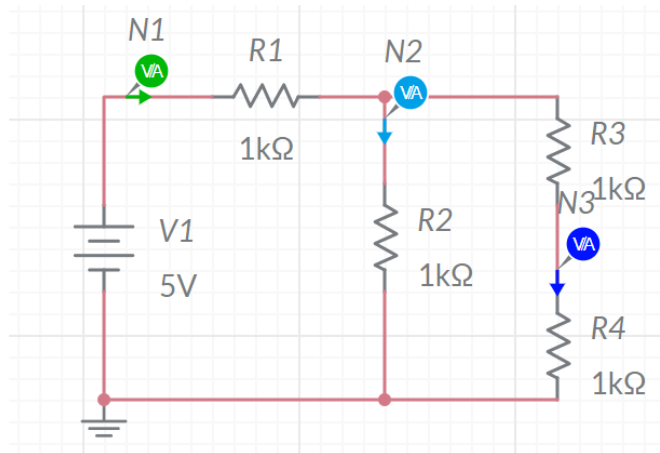

Figure 1: A circuit made in Multisim with nodes labeled. The grounded node will be referenced as "GND"

4.1   Deriving Voltage and Current Equations Using Nodal Analysis

There are four labeled nodes here. Notice that each component must go through a node to reach another component. The negative side of the battery is also grounded. This will be our reference node for the tutorial. Multisim needs

a ground in order to simulate properly. If a ground is not in the circuit, an error will pop up on the error menu asking for a ground to be put in the circuit.

1. Determine series and parallel relationships. Looking at the nodes, we can determine the relationship that each component has with each other.

   (a) N1 is shared exclusively with the voltage source and R1, so R1 is in series with the voltage source.

   (b) N2 splits after R1 to reach R2 and R3. This means that R2 is in parallel with the branch containing R3.

   (c) N3 is shared exclusively with R3 and R4, so R3 is in series with R4.

   (d) GND joins the branch made at N2 back together, where R2 and R4 share GND as a node, so R2 is in parallel with the branch containing R4.

2. Assign Important voltage nodes. This is where Nodal Analysis starts. Assigning voltage nodes is important because it allows us to characterize how the current will flow in the circuit. Current will always flow from high voltage to low voltage.

   (a) N1 = Vs. Vs is directly attached to N1, so there is nowhere for the voltage to drop prior to this node.

   (b) N2 = Vs - Vr1. There is a resistor in the way of the parallel branch at N2, so the voltage will drop some amount before reaching N2.

   (c) GND = 0V. Since we assigned GND as a ground (our reference node), and it is attached to the negative side of the voltage source, we can treat it as 0V in all of these situations.

   (d) N3 = N2 - Vr3. Since there is a resistor between N2 and N3, there will be a voltage drop before the voltage can go from N3 to GND.

3. Assign current directions. This allows us to determine which way current is traveling through each part of the circuit. If we find that current ends up traveling in the same direction once it reaches a node, we add the currents together. If the current is traveling in opposite directions, they get subtracted. The same thing happens with voltages.

   (a) Using conventional current, we will start at the positive side of a source (N1), and work our way to ground (GND), which is also attached to the negative side of our source. Once we reach GND, our reference node, we don't need to go any further.

        i. R1: Current flows from N1 to N2 (N1 → N2)

        ii. R2: Since there is only one source, the current flowing from R1 into N2 has to go somewhere, so it will travel from N2, through R2, to GND. (N2 → GND)

        iii. R3: Same logic as R2. N2 → N3

        iv. R4: Since current is flowing out of N3 from R3, the current will flow from N3 to GND through R4. (N3 → GND)

   (b) From N1, there are two main paths that the current can take:

        i. Vs+ → N1 → R1 → N2 → R2 → GND → Vs-

        ii. Vs+ → N1 → R1 → N2 → R3 → N3 → R4 → GND → Vs-

        iii. It should be noticed the paths are the same up until N2. This also shows that we have 2 parallel branches after N2.

4. Determine current relationships. Using the nodes, we can see that there are three different current values that we should look at.

   (a) I1: The current at the source. One path to go since N1 is a series node.
       Ir1 = Isourse

   (b) I2 and I3: At N2, the circuit splits, and can go either to R2 or R3.
       I2 = Ir2
       I3 = Ir3

   (c) Since R3 is in series with R4, the same current flows through both R3 and R4.
       I3 = Ir3 = Ir4

(d) Knowing Kirchhoff's Current Law, the current entering a node must equal the current leaving the node. Since N2 splits I1 into I2 and I3
    I1 = I2 + I3.

5. Use Kirchhoff's Current Law to determine how the voltage will act at each node. (Yes, we use the current law to help us analyze voltage.) Now that we have the voltages at the nodes and currents at the nodes characterized, let's derive what the voltages will be at each node. Our end goal is to have everything defined by our source voltage and our resistor values.

   (a) N1 = Vs → We know Vs, so that part is done.

   (b) KCL: I1 = I2 + I3 → We don't know any of the currents yet.

   (c) I1 = (Vs - Vn2)/R1 → We don't know the voltage at Node 2 yet.

   (d) I2 = Vn2/R2 → We don't know the voltage at Node 2 yet.
       Note: We can write this as (Vn2-VGND)/R2 to better match step iii, but we know that VGND = 0v, so we can take it out to simplify the equations.

   (e) I3 = V2/(R3 + R4) → We don't know voltage at Node 2 yet.
       This can also be written as I3 = (Vn2-Vn3)/R3.
       We also know that I3 = Vn3/R4.
       Which means that (Vn2-Vn3)/R3 - Vn3/R4 = 0, or (Vn2-Vn3)/R3= Vn3/R4.

   (f) If we treat the parallel branches in this circuit as its own circuit (R2 ∥ (R3+R4)). Let's call this Re. The resistance between the nodes (Re) can be found by:
       1/Re = 1/R2 + 1/(R3 + R4),
       where Re = R2(R3+R4)/(R2+R3+R4).

   (g) That means our circuit can be redrawn as two resistors, R1 and Re in series with a source. The total resistance in the circuit can be written as Rt = R1 + Re, and the total current can be written as Is = Vs/(R1 + Re) = I1.

   (h) With Re known, we can now calculate Vn2. voltage is equal to current multiplied by resistance, so our voltage at Node 2 is
       Vn2 = Vs * Re/(R1 + Re) ← This equation will be coming back.

   (i) With V2 known, we can treat it as the source for I3. Since we have 2 resistors in series for I3, we can follow the same steps starting at vii. This means our voltage at N3 in relation to GND is
       Vn3 = Vn2*R4/(R3 + R4) ← Equation look familiar?

   (j) Taking these values, we can say that the currents in I2 and I3 are
       I2 = Vn2/R2.
       I3 = Vn2/(R3+R4).

All of the node voltages, and currents can now be calculated with numeric values. This may look like a lot of work. That is because this process *is* a lot of work, but it is necessary to understand how general circuit analysis works. Fortunately, when it is just resistors and sources, all of these steps use basic algebra. With practice this process will take less time, and a lot of the steps can be done mentally. More complex circuits can also be directly analyzed using simulations.

4.2   Use derived equations to determine actual values in a circuit

Assign the values to to each component as shown below. These values are all in base units. If needed, LTspice does recognize prefixes to units, such as "K" for kilo and "m" for mili. The values in this example are not realistic, but
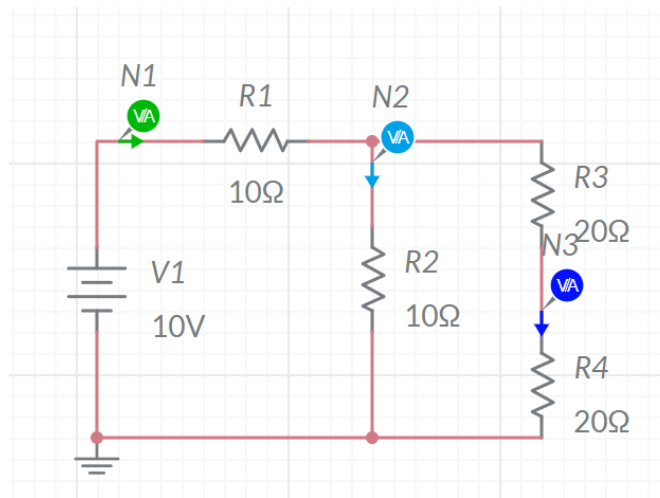
Figure 2: Circuit in Figure 1 with component values.

they will make the calculations more friendly.

Calculate the currents through each resistor, and the voltages at each node.

1. VN1 = Vs = 10V

2. $Re = R2(R3 + R4)/(R2 + R3 + R4) = 8\Omega \rightarrow$ This makes sense. Recall that parallel resistances are always less than the smallest branch resistance in the circuit.

3. $Rt = R1 + Re = 18\Omega$

4. $Is = Vs/(R1 + Re) = IR1 \rightarrow 10V/18\Omega = 0.55A$

5. $Vn2 = Vs * Re/(R1 + Re) \rightarrow 10V * 8\Omega/(18\Omega) = 4.4V$

   This means that 5.6V is dropped over R1 (Vr1 = 10v - 4.4v, Kirchhoff's Voltage Law) . This makes sense since R1 > Re, the larger resistance will take more voltage. This will get explained more in the Voltage Divider section.

   This is also the voltage drop across R2, as N2 is referenced to GND, which is directly attached to R2. This can also be calculated using Ohm's Law by taking the current going through R1 (.55A) and multiplying it by the resistance of R1 (10Ω) to get the voltage drop across R1.

6. $IR2 = Vn2/R2 \rightarrow 4.4V/10\Omega = 0.44A$

7. $IR3 = Vn2/(R3 + R4) \rightarrow 4.4V/(40\Omega) = 0.11A$

   This verifies Is using Kirchhoff's Current law. The current going into N2 (0.55A) equals the current leaving N2, (0.44A + 0.11A)

8. $Vn3 = Vn2 * R4/(R3 + R4) \rightarrow 4.4V * 20\Omega/40\Omega = 2.2V$

   This is also the voltage drop across R4, as N3 is referenced to GND, which is directly attached to R4. It should be noted that the voltage across R4 is half of the total voltage in that branch. This makes sense since R3 and R4 have the same resistance, both should have the same voltage drop across them. (Vr3 = 4.4V - 2.2V = 2.2V, KVL) This can also be looked at as Vn2-Vn3 = VR3.

4.3   Use Spice to Verify Calculated Values

At this point, if it is not done already, make the circuit in Spice.

1. Once you have your circuit made, click on the value of each component to change its value from default to the value you need.

2. Make nodes using the Voltage/current probes.

3. There are a variety of simulations that can be performed under the drop-down menu initially labeled "Interactive." Select the "DC OP" option. This will simulate the circuit using the values you have put in without modifying them at all.

4. Click the "simulate" or "Play" button, and your voltage/current probes should pop up with assorted voltages and currents.

5. To get a list of values instead of having to look around the circuit, select the "Grapher" screen.

DC OP 1

Signal	Value
N1: V(1)	10.000V
N2: V(2)	4.4444V
N3: V(3)	2.2222V
N1: I(R1)	555.56mA
N2: I(R2)	444.44mA
N3: I(R4)	111.11mA

Figure 3: SPICE values for circuit in Figure 2.

Multisim does a great job at labeling your assorted nodes. Notice that voltages are calculated at nodes, where currents are calculated through the resistors. All of these measurements are using the ground as a reference node. If the ground is placed somewhere else, then that will automatically be assumed to be 0V. If you forget to put a ground in your circuit, an error will be displayed in the error menu.

5 Equivalent Circuits

While it is nice to be able to look at individual parts of a circuit using nodal analysis, there are also times we want to be able to look at a circuit as a whole, specifically, how the circuit appears to a load. When attaching motors, sensors, or other devices to a circuit, especially if the resistance of the load changes regularly (like if the load changes resistance significantly due to heat, or there are different components are switching on and off at odd intervals, such as when looking at air conditioning units in a neighborhood, or even across an entire grid), it would be very helpful to know how the primary circuit will react when the device is attached, so we need to create an **equivalent circuit** of the current circuit to help us easily predict what will happen. This would also allow us to save a lot of time so we don't have to do a brand new nodal analysis every time the load is changed.
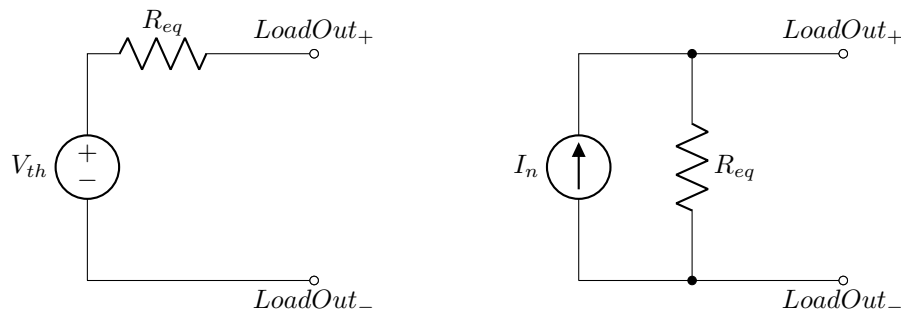
Figure 4: Two Equivalent circuits, Thévenin Circuit (Left) and Norton Circuit (right). Load will attach to the nodes on the right of each circuit.

When creating an equivalent circuit, the goal is to make a circuit that will act the same in reference to the load, but with as few components as possible. There are two circuits that we can make, a single voltage source with a single resistor in series, also known as a Thévenin circuit, or a single current source in parallel with a single resistor, also known as a Norton Circuit. Both circuits will have two nodes going out to attach to the load.

Let's take a standard voltage divider with $R_1 = 3K\Omega$, $R_2 = 7K\Omega$, and $V_{in} = 10V$ where we are attaching a load across R_2 like in Lab 2. The voltage divider schematic in Figure 5 shows the voltage source instead of just having V_{in} on top. This is important with how we approach making an equivalent circuit.

The first step of determining an equivalent circuit is to determining the equivalent resistance. This step starts with removing the load, and "turning off" all supplies. "Turning off" a voltage supply means that there is a 0V drop across the device, so it essentially acts as a wire. "Turning off" a current supply means there is no current flowing through the device, so it acts as an open circuit. In your circuit, replace all voltage sources with a wire, and replace all current sources with an open circuit.
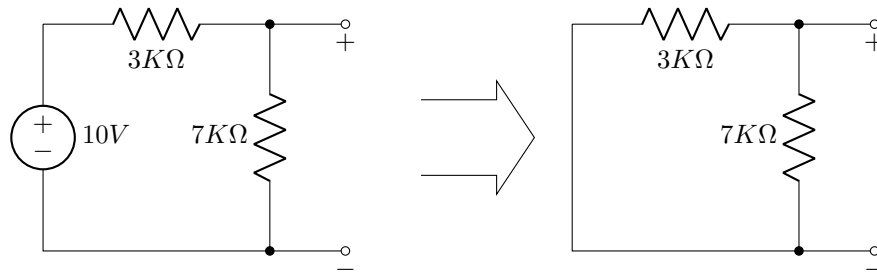

Figure 5: Voltage divider with load (left), and eq. circuit without sources.

While it may look a bit odd, in relation to the outputs, we now just have two resistors in parallel, which means we can easily calculate the resistance in this circuit in relation to the outputs to be 2100Ω. This is the equivalent resistance of the circuit.

Once we have the equivalent resistance, we now need to figure out the output voltage of the circuit. Turn back on the sources (one at a time), and determine the voltage out provided by each source. With this circuit being a voltage divider and only having one source, we can calculate that the output voltage across R_2 is 7V, which is our Thévenin Voltage of our circuit, also known as the "open circuit voltage (V_{oc})" or the voltage that we can expect out of the circuit assuming a large load compared to the eq. resistance.

It would also be helpful to know the effective current through this circuit. To do this, we will use the same equivalent resistance as we found previously, and treat the load as a wire, or a resistor of 0Ω. This allows us to determine a Norton current of $3.33mA$, or the maximum current that we can get through the load if it is very low resistance.
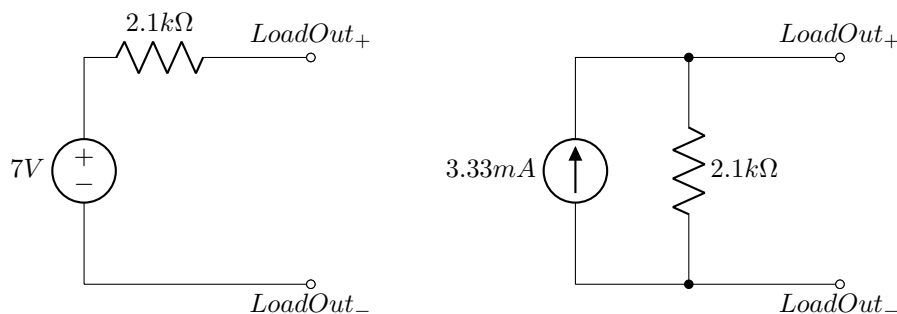

Figure 6: Two Equivalent circuits, Thévenin Circuit (Left) and Norton Circuit (right). Load will attach to the nodes on the right of each circuit.

To verify the Norton current of the voltage divider, think about the operation that was just performed. Looking at the voltage divider in Figure 6, the $7K\Omega$ resistor got shorted with a wire. This means that all of the current is going to flow through the wire instead of the resistor, meaning the $3K\Omega$ resistor is the only resistor in the circuit resisting current. With a 10V source, and a $3K\Omega$ in series, the current in the circuit can be calculated to be $3.33mA$.

This is a fairly simple example, and when the component count is low like this, it is often not completely necessary to do a full equivalent circuit derivation, but as the source count increases, or you end up with multiple branches where current and voltage vary a lot from the main source, creating an equivalent circuit will allow you to make quick work of the circuit for future calculations. Even adding one additional source, current or voltage, can make the circuit much more complicated, so making the equivalent circuit would be very beneficial.

Something to keep in mind is that the load can still impact how the equivalent circuit will act. We are adding a load to a circuit, so loading is a phenomenon we have to keep in mind (Lab 2). Making the circuit as simple as possible, as in having a single resistor and a single source will help keep the math simple and quick. To recap, the steps for determining the equivalent circuits are:

1. Remove the load and "turn off" your sources. Treat voltage supplies as wires and current sources as open circuits.

2. Solve for your equivalent resistance in reference to the load.

3. Determine the Thévenin voltage, or V_{oc} at the load.

4. Determine the Norton Current, or I_{sc} at the load.

Prelab

6 Complete Nodal Analysis

Using the circuit found in Figure 7, do a complete nodal analysis finding the current through each path, and the voltage drop across each component.
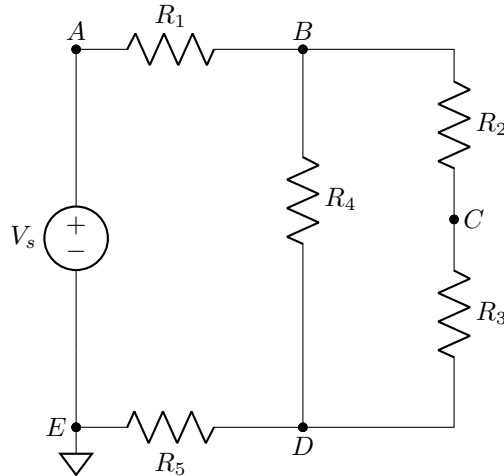

Figure 7: Prelab nodal analysis circuit.

1. Using the node labels in Figure 7, determine if each node is a series or parallel node.

2. Identify and determine the direction of each current path.

3. Derive the voltages at each node and currents of each path in reference to to the node attached to Vs-. Show all work.

4. Calculate the derived voltages and currents using the below values:
   Vs = $5V$, $R_1 = 220\Omega$, $R_2 = 150\Omega$, $R_3 = 330\Omega$, $R_4 = 330\Omega$, $R_5 = 150\Omega$

7 Equivalent Circuits

If R_3 is treated as a load in this circuit, derive the Thévenin and Norton circuits for the circuit in Figure 7. Remember the below steps:

1. Remove the load and "turn off" your sources. Treat voltage supplies as wires and current sources as open circuits.
   When actually verifying this in real life, this does not actually mean turning off the supply. Set the value to 0, either 0V or 0A. Actually turning off the supply will likely create an open circuit, which for a current supply is fine, but for voltage, it will not give you the right results. You can also replace your voltage sources with wires, and break the circuits at current sources.

2. Solve for your equivalent resistance in reference to the load. **Note:** These values were chosen to give you a pretty nice equivalent resistance in comparison to your resistor values in your lab kit. If your numbers don't seem to match up, try again.

3. Determine the Thévenin voltage, or V_{oc} at the load. For this circuit, will R_2 get used in those calculations?

4. Determine the Norton Current, or I_{sc} at the load.

In-lab

8 Complete Nodal Analysis

8.1 Simulate for Verification

1. Create a Spice simulation of the circuit in Figure 7 to verify your calculated values. Provide a screenshot of your Spice circuit, along with the results from the simulation.

2. Do your calculations match your simulation?
   If not, why? Double check your calculations, and make sure you setup the simulation properly.

8.2 Build the circuit and verify

1. Construct the circuit found in Figure 7 in the prelab.

2. Measure the voltages at each node, the current through each resistor, and the current at the source.
   Note: Make sure you are measuring voltage in parallel and current in series!

3. Ideally, you should now have personal calculations, a simulation, and a constructed circuit that all support each other with consistent values. Did this work out? Why or why not? If you need to, go back and adjust your circuit, simulation, and/or your calculations to see what went wrong.

It can probably be stated that this was not the "most fun" thing to do. It takes a fair bit of time to do the calculations, make the simulation, and then ultimately test a circuit, but it is very important. Doing the math and simulations allow us to ensure that the circuit will actually do what it is intended to do, can be done in a safe manner, and can save money and materials. As time goes on, you will be able to do a lot of these calculations fairly quickly, like by finding voltage dividers and other patterns that will make this a less tedious process.

IMPORTANT!! DO NOT take apart your circuit! You will need it for the next task!

9 Equivalent Circuits

9.1 Simulate for Verification

1. Create a Spice simulation of your Thévenin and Norton circuits to verify your calculated values. Provide a screenshot of your Spice circuit, along with the results from the simulation.

2. Add the 330Ω resistor from the original circuit in the load location. Verify that the voltage drop and current through the resistor are the same as in the original circuit.

3. Do your calculations match your simulation?
   If not, why? Double check your calculations, and make sure you setup the simulation properly.

9.2 Build the Circuit and Verify

1. Construct the your Thévenin circuits from the prelab, and verify that it is equivalent to the original circuit from the perspective of the load.

2. Verify that your equivalent circuits provide the same V_{oc} and I_{sc} as your primary circuit with R_3 removed. Calculate your percent error, and explain where that error might be coming from.

3. Attach 100Ω, 1kΩ, and 10KΩ to both the original circuit, and the equivalent circuit to show that V_{Load} and I_{load} are the same in both circuits.

4. Is it worth your time to make an equivalent circuit for larger circuits or not? Explain your reasoning. Note: This is a subjective question, and there is not a right answer, even though it may seem that one is being encouraged. Explain your reasoning either way.