LTspice Basics Workshop

Presented by AECES TnD

Juan Glicerio C. Manlapaz

Ateneo de Manila University

27 November 2020

Objectives

Objectives of the LTspice Workshop

The main goal of this workshop is to be familiarized with LTspice for the future ENGG classes (mostly for circuits and electronics). Specifically, it aims to:

1. Simulate simple electrical circuits and measuring quantities such as voltages and currents.
2. Familiarize with the capabilities of LTspice by learning some examples of electrical circuits.
3. Incorporate your learnings from ENGG classes without the need of physical components.

Introduction to LTspice

What is LTspice?

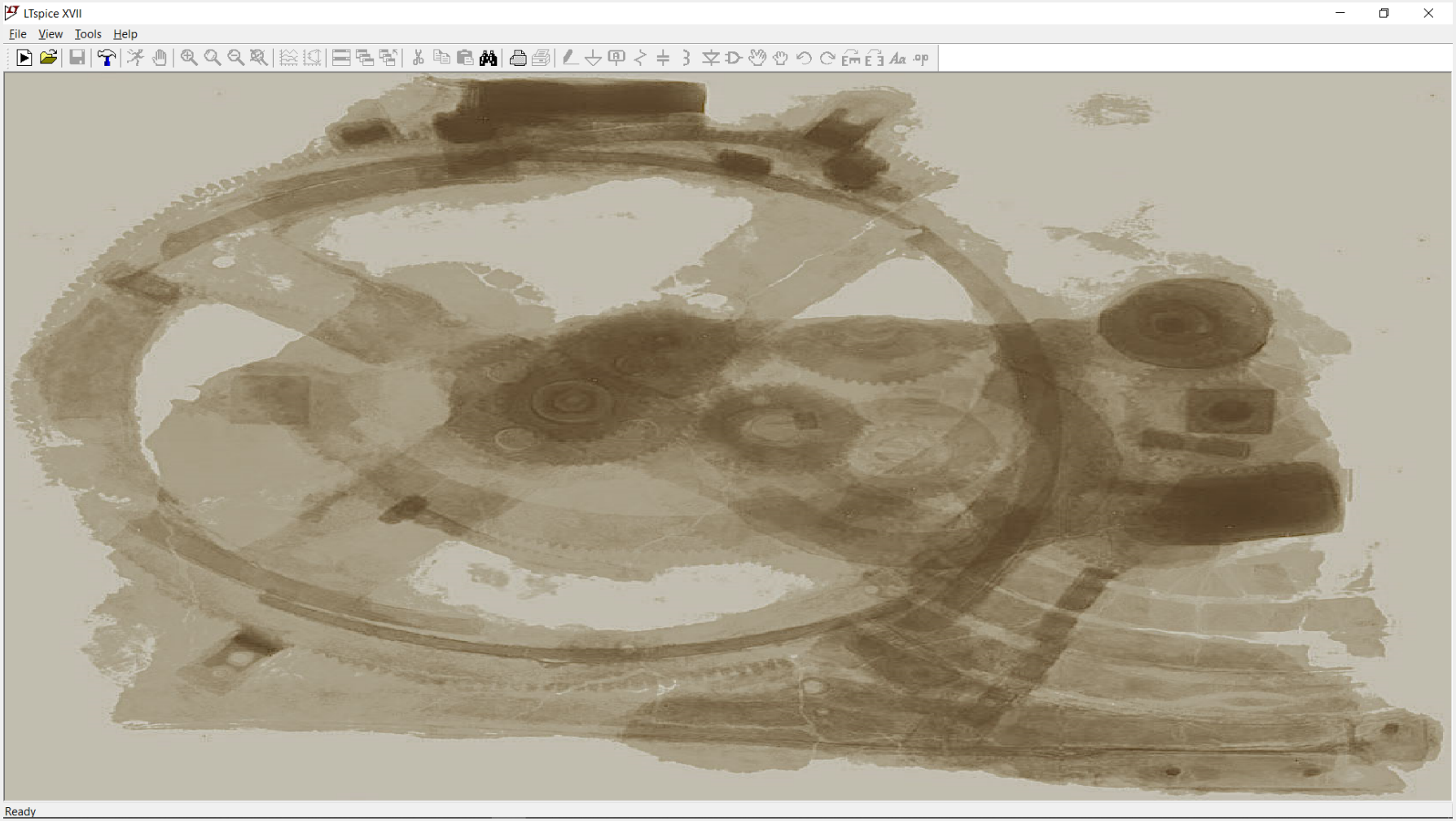
LTspice is a combination of two words:

- **LT** which stands for **L**inear **T**echnology[1], the company who developed the software (now known as Analog Devices).
- **SPICE** which stands for **S**imulation **P**rogram with **I**ntegrated **C**ircuit **E**mphasis[2], an open source circuit simulator program.

So basically, LTspice can simulate electrical circuits! Though there are some things that make the program stand out; such as it being **open source** (which means its free), and it **emphasizes on electrical diagrams for simulation** rather than code.

[1] https://en.wikipedia.org/wiki/LTspice
[2] https://en.wikipedia.org/wiki/SPICE

LTSpice window



Keyboard Shortcuts for LTspice

Keyboard Shortcuts: Alphabetical Characters

Keyboard Shortcut	Function
R	Draw Resistor
C	Draw Capacitor
L	Draw Inductor
V	Draw Voltage Source
G	Draw Ground
D	Draw Diode
T	Edit Comment on Schematic
S	Edit SPICE Directive on Schematic

Keyboard Shortcut	Function

Keyboard Shortcuts: Function Keys

Keyboard Shortcut	Function
F1	Open Help
F2	Select Component Symbol
F3	Draw Wire
F4	Net Name
F5	Cut
F6	Duplicate
F7	Move
F8	Drag
F9	Undo
Shift + F9	Redo
Keyboard Shortcut	**Function**

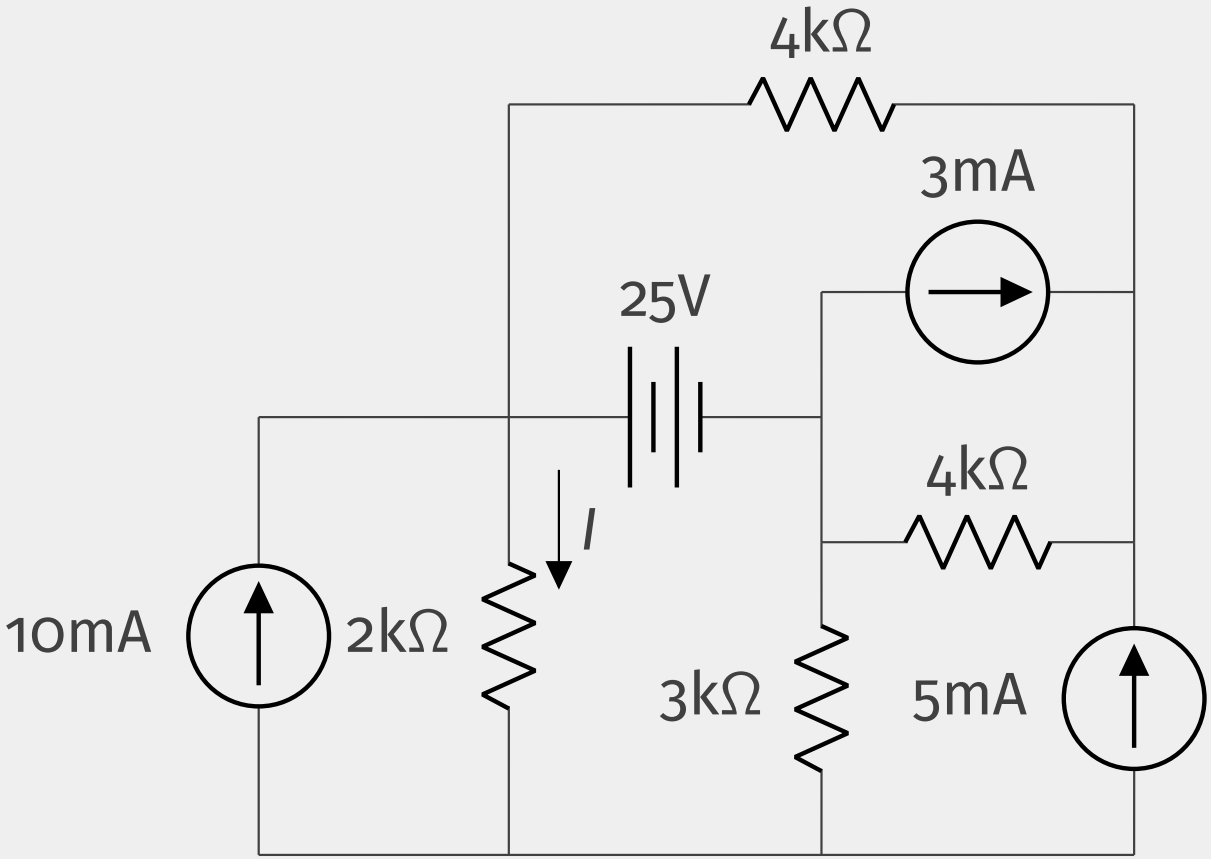
Keyboard Shortcuts: Combinations

Keyboard Shortcut	Function
Ctrl + N	New Schematic
Ctrl + O	Open
Ctrl + S	Save
Ctrl + P	Print
Ctrl + R	Rotate
Ctrl + E	Mirror
Ctrl + V	Paste
Keyboard Shortcut	**Function**

Simulating your First Circuit

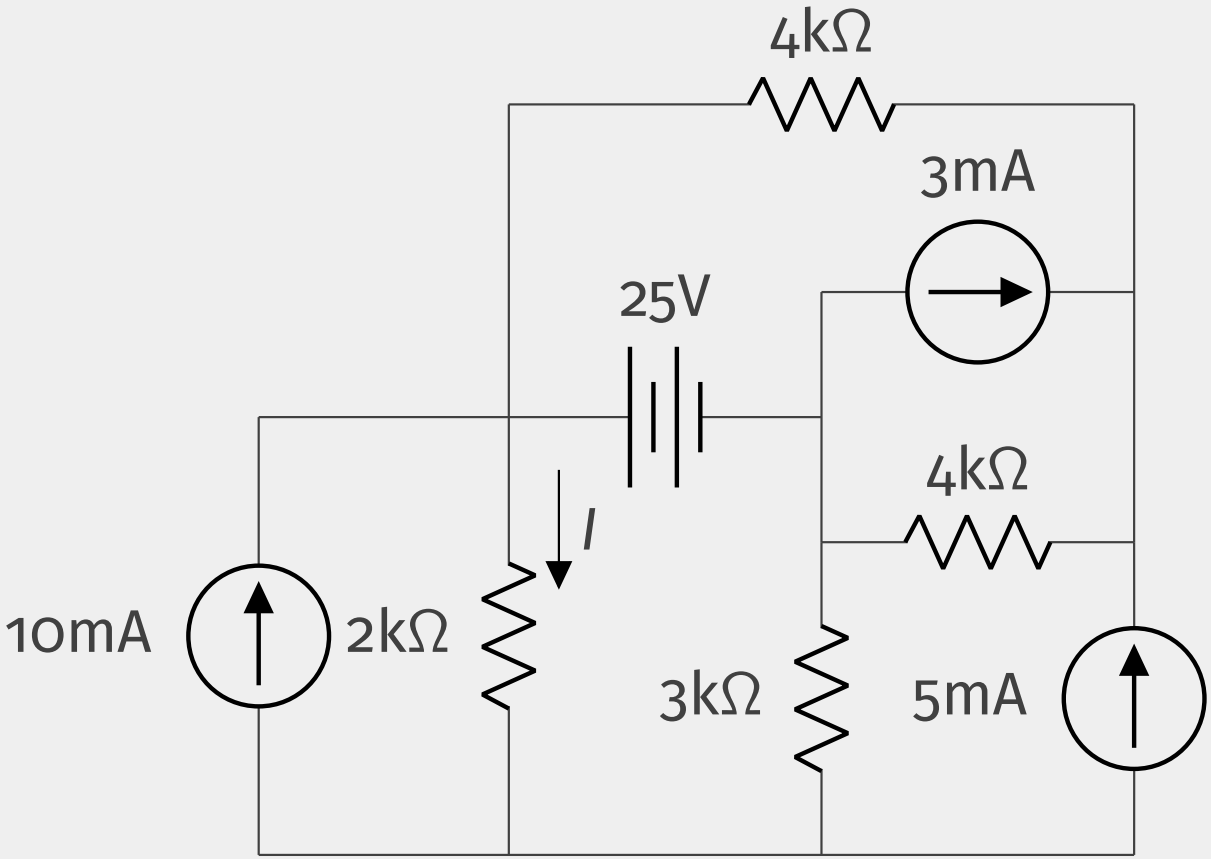
Simulating your First Circuit

Suppose you want to solve the following circuit:



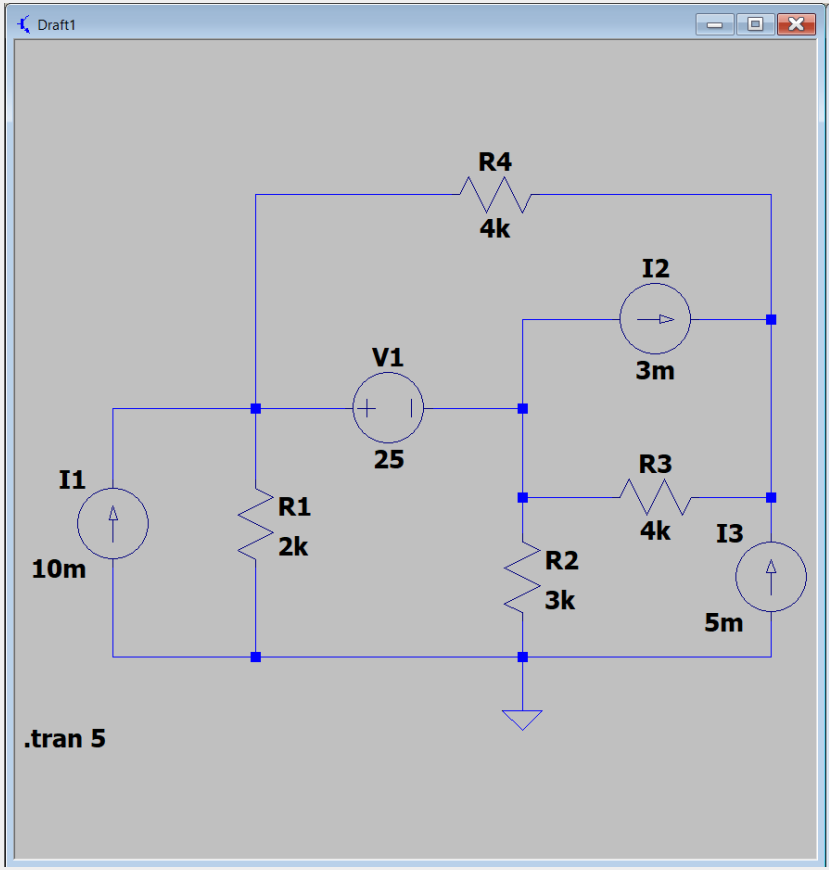
Simulating your First Circuit

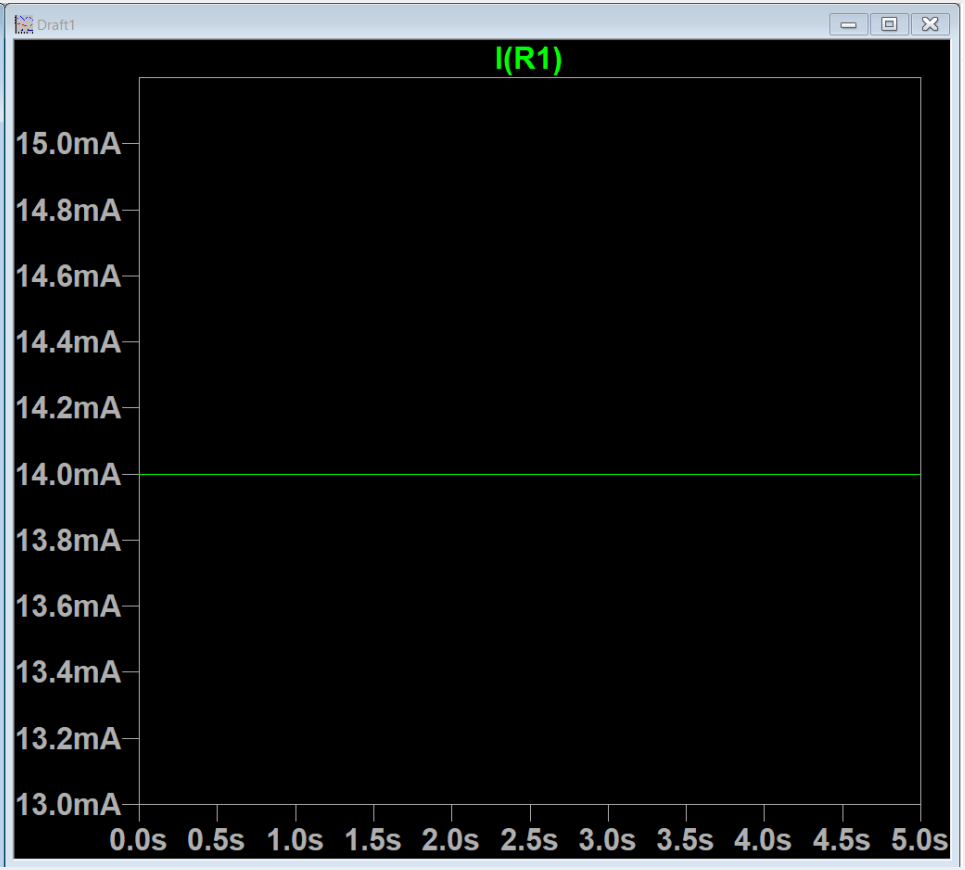
Let's solve for *I* using LTspice:



Time to open LTSpice!

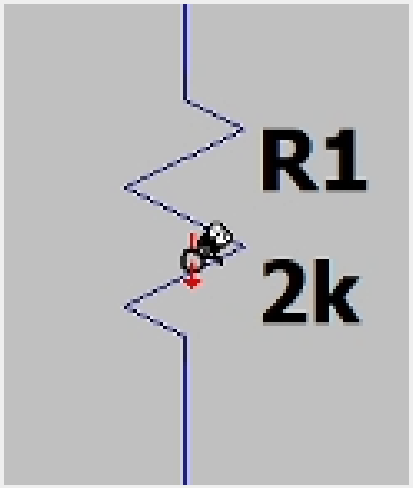
Simulating a circuit using LTspice

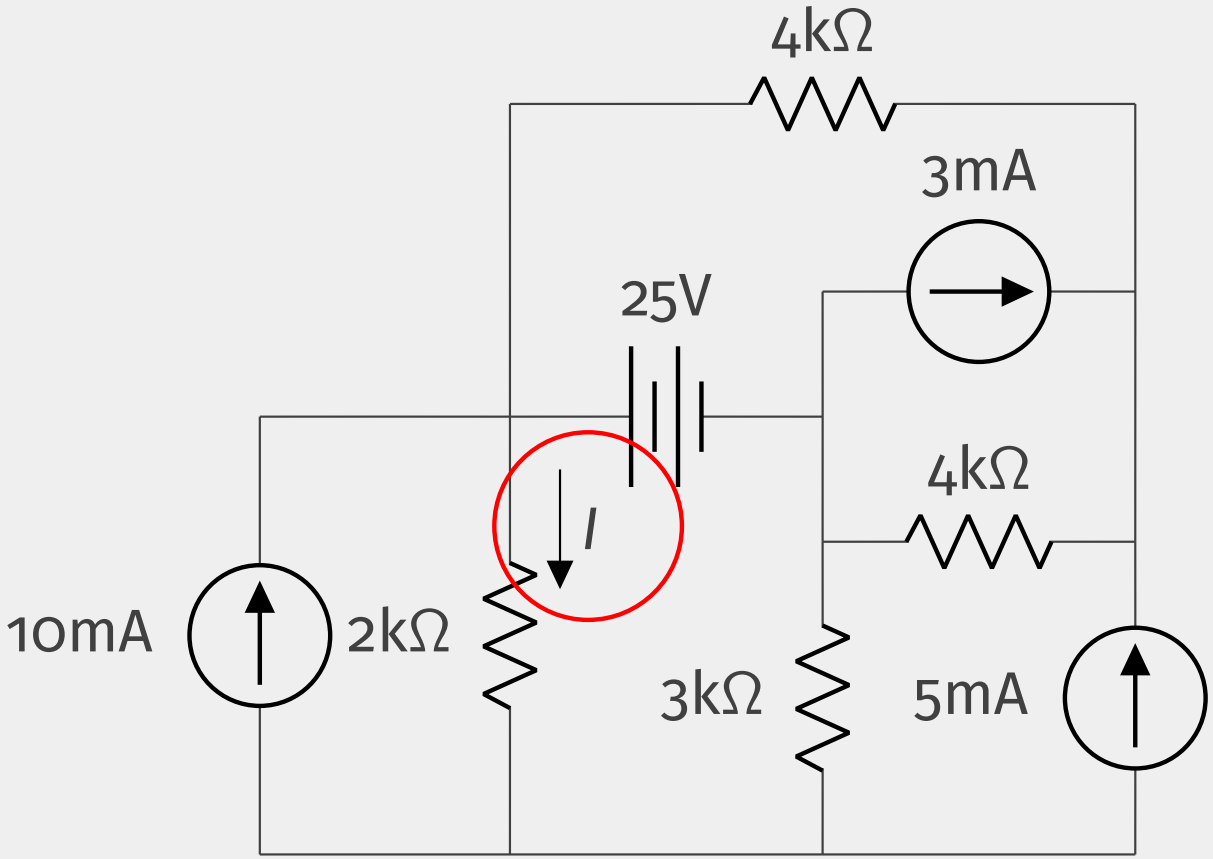




Simulating a circuit using LTspice

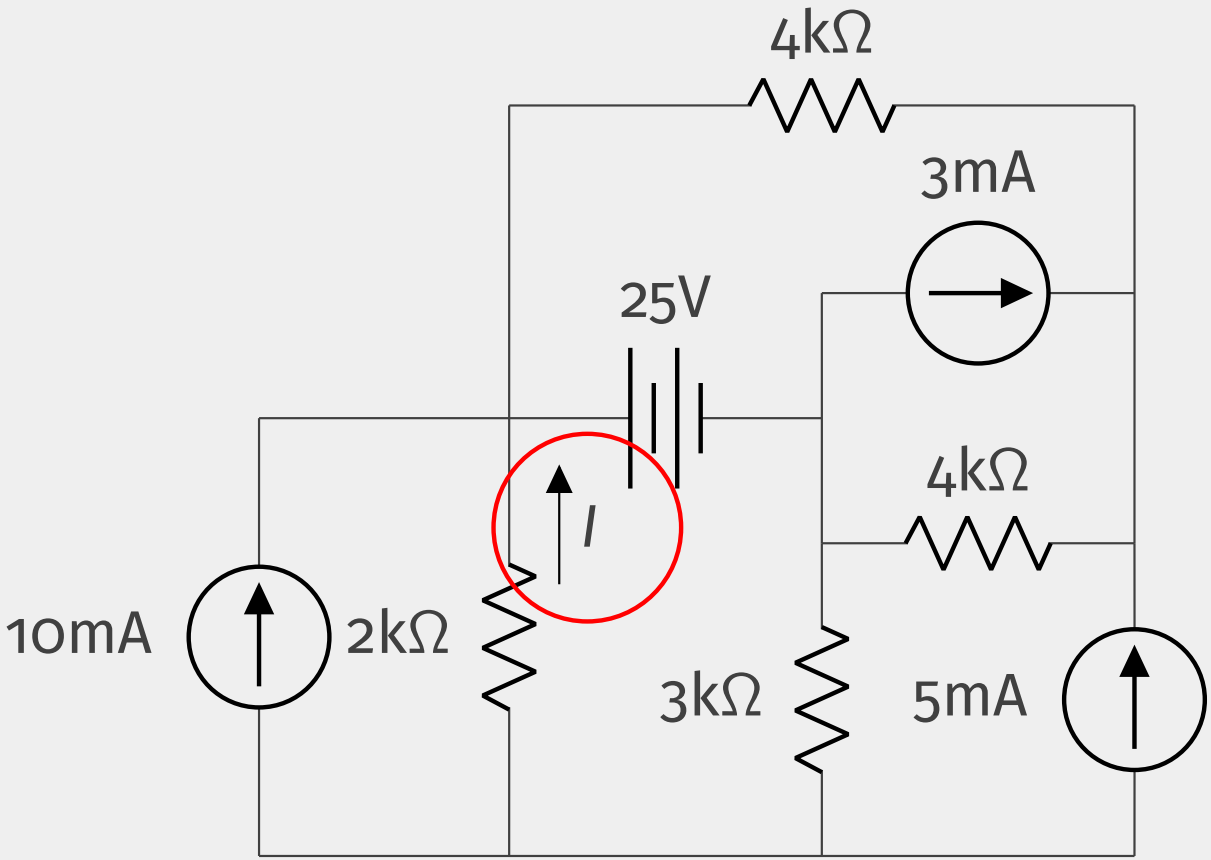
Note that the direction of the current being measured (that appears when you hover a component) is the same as the direction of the arrow in the diagram.





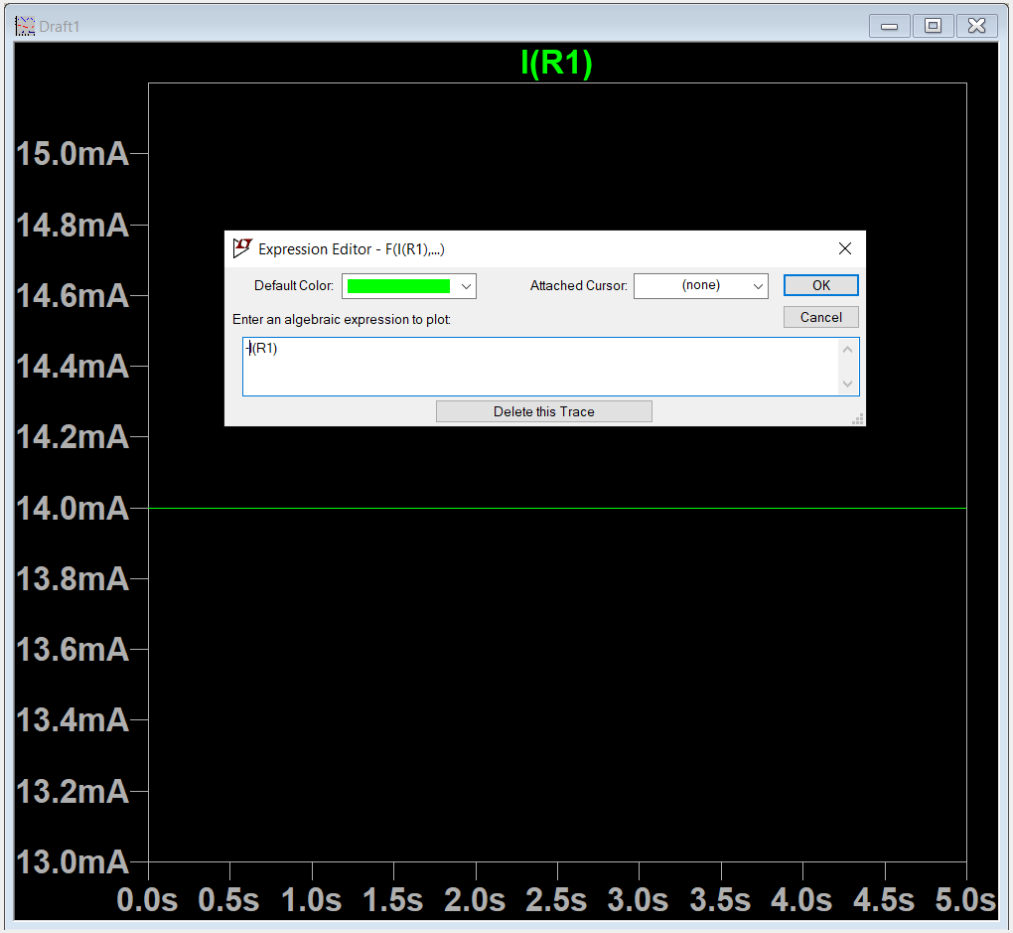
Simulating a circuit using LTspice

But what if the problem asks for the current in reverse?



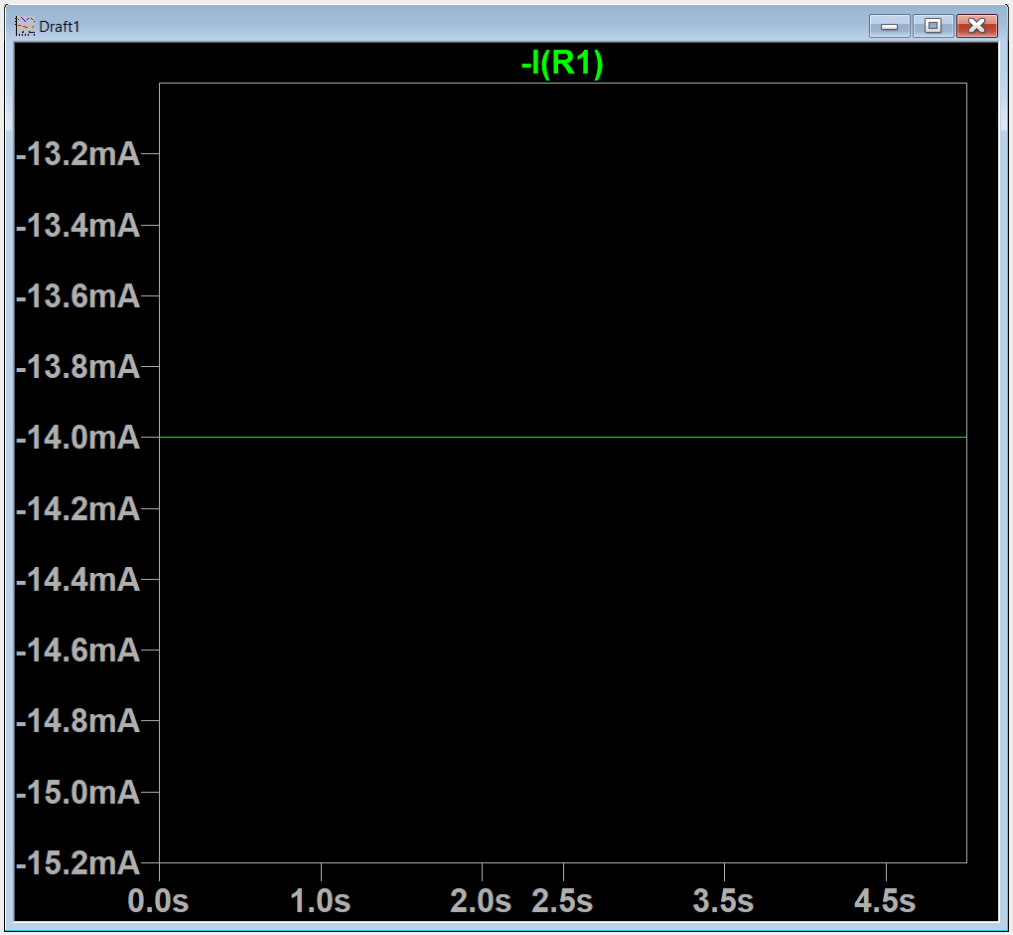
Simulating a circuit using LTspice

Solution: Right-click "I(R1)" and a window will pop-up called Expression Editor. A textbox below shows the text "I(R1)". Simply append a "-" at the beginning of "I(R1)".



Simulating a circuit using LTspice

Recall that a negative sign in a current variable means a reverse in current direction.

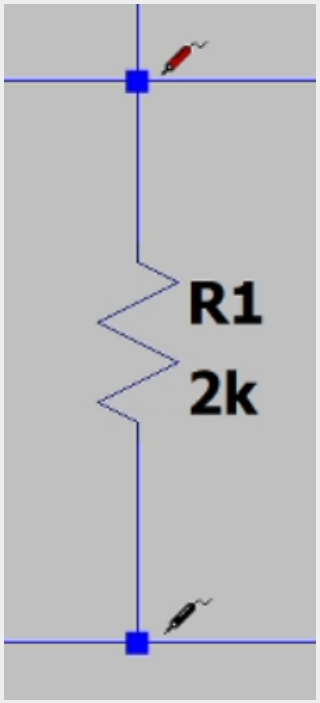


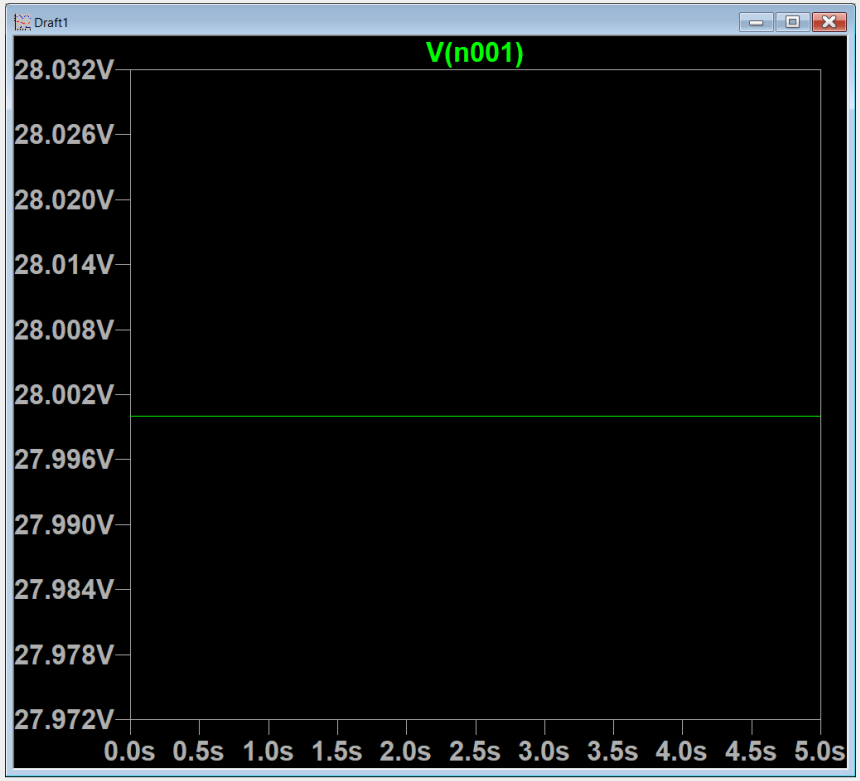
Simulating a circuit using LTspice

Using the same circuit, solve for the voltages across all the resistors (assume that the positive polarity is at the upper/left terminal while the negative terminal is at the lower/right terminal).

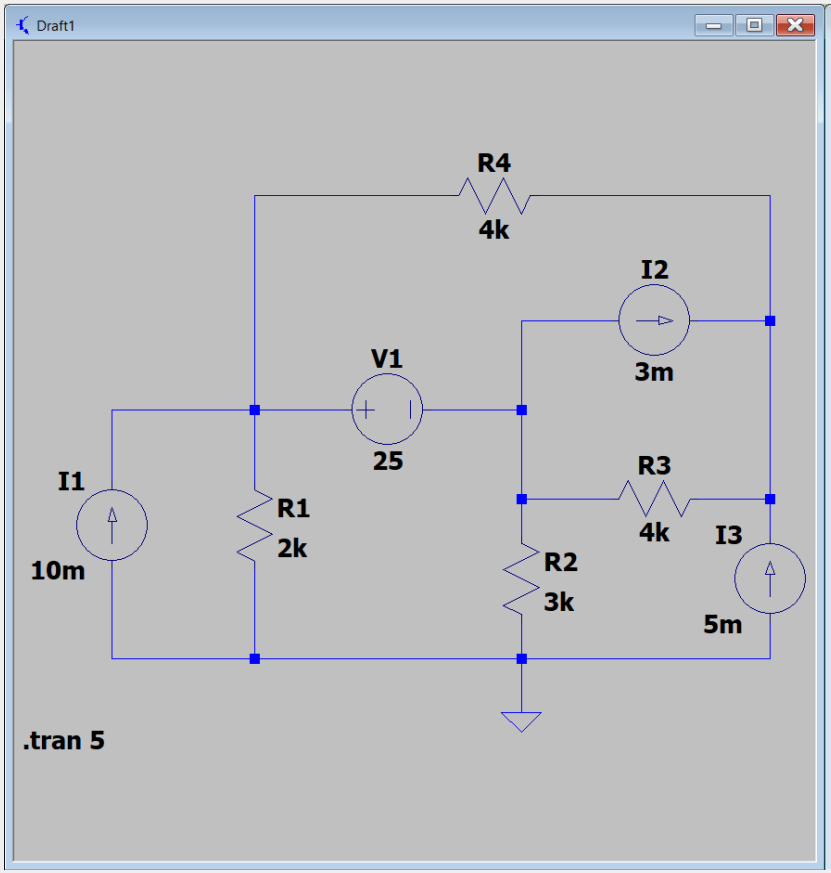
Simulating a circuit using LTspice

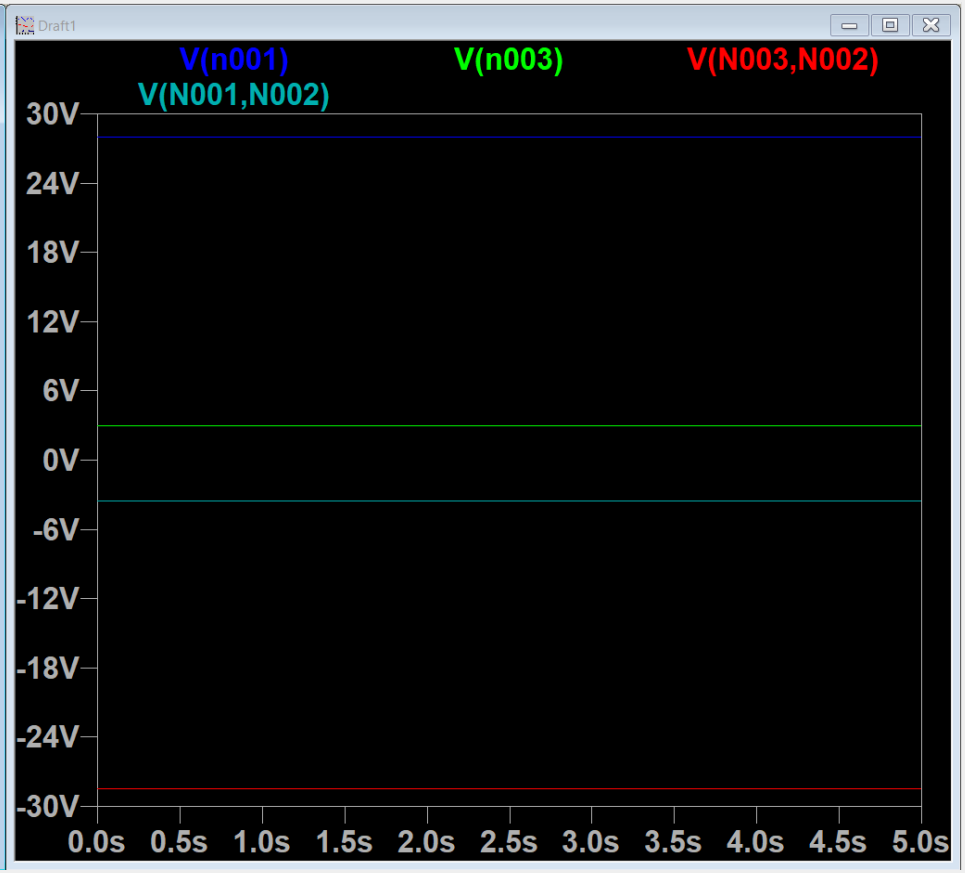
Solution: Hover over any node in the circuit. A cursor resembling a red multimeter probe will appear. Click and hold until a black multimeter probe appears. Holding the mouse, hover over the next node of choice then release.





Simulating a circuit using LTspice





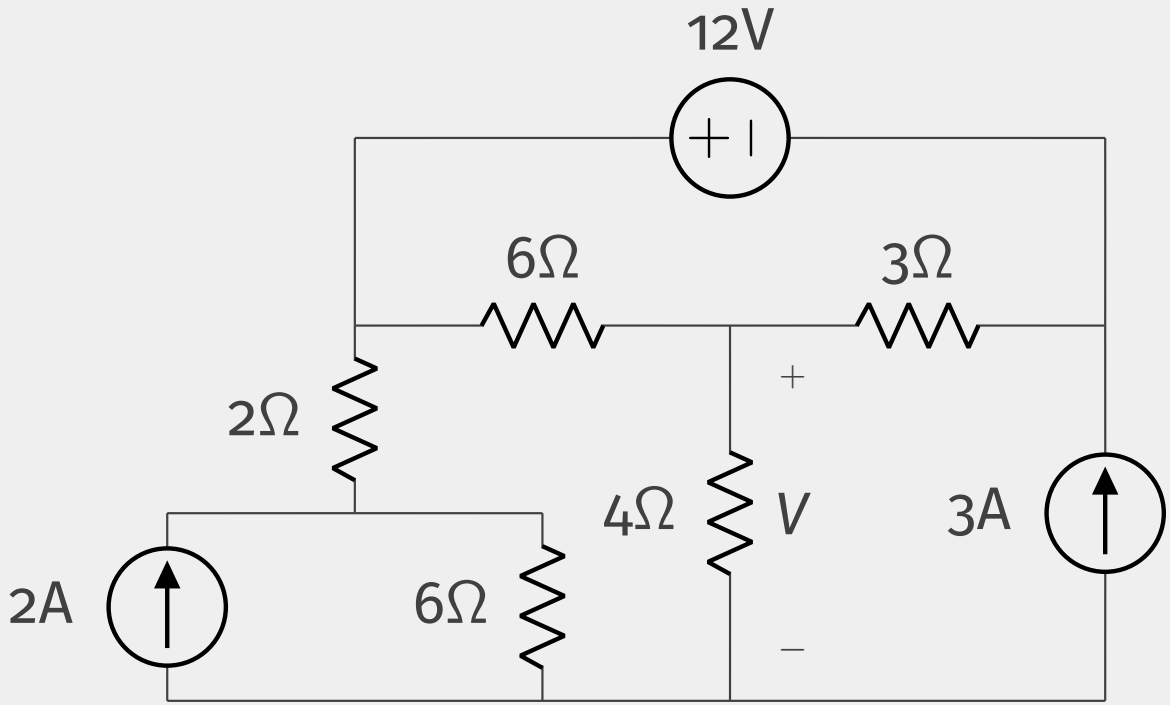
Some Key Reminders

When simulating a circuit using LTspice:

1. Always indicate a ground node. In most cases where the figure does not indicate a ground node, use the lowermost node as the ground node.
    - LTspice needs to have a reference point for all electrical measurements.
2. Always make sure that all components of the circuit have values.
    - To change the value, simply right-click the component.
3. Always input the Stop Time before simulating the circuit.
    - LTspice actually outputs the measurements as **time-based waveforms** where you can actually see the values at a specific time.

Your Turn!

Let's solve for *V* using LTspice.



More things you can do in LTspice

Once you know the basics of LTspice, you can start creating more things!

More things you can do in LTspice

Some of the things you can do in LTspice include:

1. Ideal Op-Amp Circuits
2. Circuits with Dependent Sources
3. First and Second-Order Circuits with Initial Conditions (Basically circuits with Capacitors and Inductors)
4. AC Circuits using Method of Phasors
5. Diode Circuits, Transistor Circuits
6. Transformers
7. and more!

Further Learning

Additional Sources for learning LTspice

1. **LTwiki**: The official wiki for LTspice
    - http://ltwiki.org/
2. **Analog Devices**: The official website of the company that created LTspice
    - https://www.analog.com/
3. **Some Practice Problems**: A GitHub repository with more materials and simulations. Download as ZIP folder.
    - https://github.com/jgmanlapaz/ltspice-workshop

References

Ltspice.
***Wikipedia*, Nov 2020.**

David E Johnson, Johnny R. Johnson, and John L. Hilburn.
***Electric Circuit Analysis*.**
Prentice Hall, 1992.