Part 1: The Basics of LTspice

1   What is LTspice?

1.1   Overview

LTspice is a SPICE-based circuit simulator software developed by Linear Technologies, hence the "LT" part in the name. SPICE stands for "Simulation Program with Integrated Circuit Emphasis" and is the main software for circuit simulation.
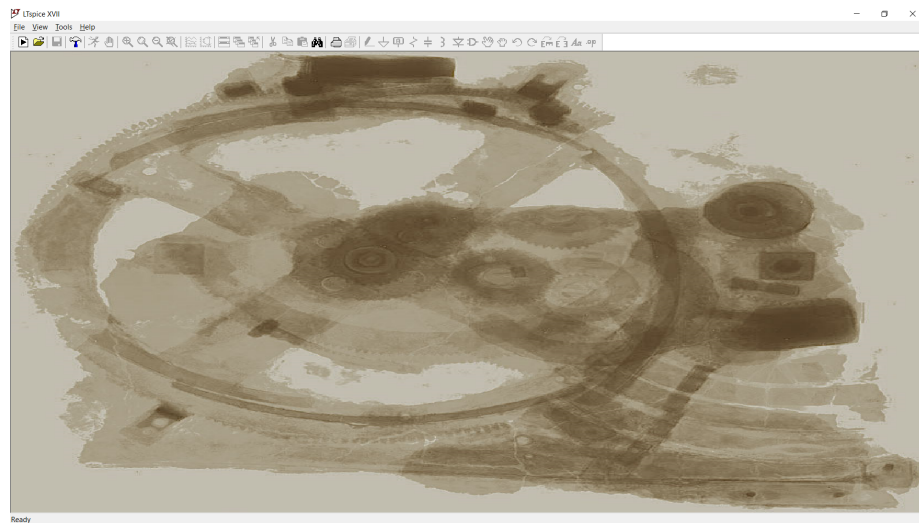

Figure 1: A screenshot of LTspice IV

1.2   What can we do with LTspice?

LTspice is used mainly for simulating analog electronic circuits. These are the circuits that involve resistors, capacitors, inductors, diodes, and transistors. It is also used to show the time graphs of voltages and currents across any component, kind of like an oscilloscope. This program should come in handy when it comes to solving circuit problems as it can provide the answer to double check the obtained answer.

2   Installing LTspice

To install LTspice, go to this link, click on Download LTspice IV, and simply follow the instructions for installation.

3   LTspice Keyboard Shortcuts

Here's a table of all useful keyboard shortcuts in LTspice IV. These shortcuts could make your LTspice experience easier and quicker.

Key	Function
R	Toggle Resistor
G	Toggle Ground
C	Toggle Capacitor
L	Toggle Inductor
D	Toggle Diode
T	Edit Comment on the Schematic
S	Edit SPICE Directive on the Schematic
F1	Help
F2	Select Component Symbol
F3	Draw Wires
F4	Net Name
F5	Delete
F6	Duplicate
F7	Move
F8	Drag
F9	Undo
Del	Delete
Esc	Escape from current function
Shift + F9	Redo
Ctrl + C	Copy
Ctrl + V	Paste
Ctrl + S	Save
Ctrl + O	Open
Ctrl + N	Schematic
Ctrl + R	Rotate Component
Ctrl + E	Mirror Component

4   Simulating your First Circuit

To formally begin the use of LTspice IV, we will try to simulate a circuit that consists of a voltage source, a current source, and some resistors. LTspice IV does a good job in illustrating the circuits by letting the user build the schematic using symbols rather than relying on some programming language to do so. We will now perform circuit analysis in the circuit in Figure 2 by simulating it using LTspice IV.
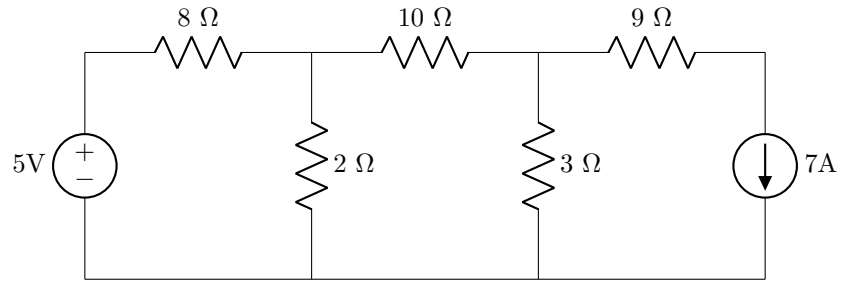
Figure 2: A screenshot of LTspice IV