



LTspice tutorial Part 3- Basic circuits



Prerequisites

- Please make sure you have completed the following:
 - LTspice tutorial part 1 (download and installation)
 - LTspice tutorial part 2 (components and basic interface)

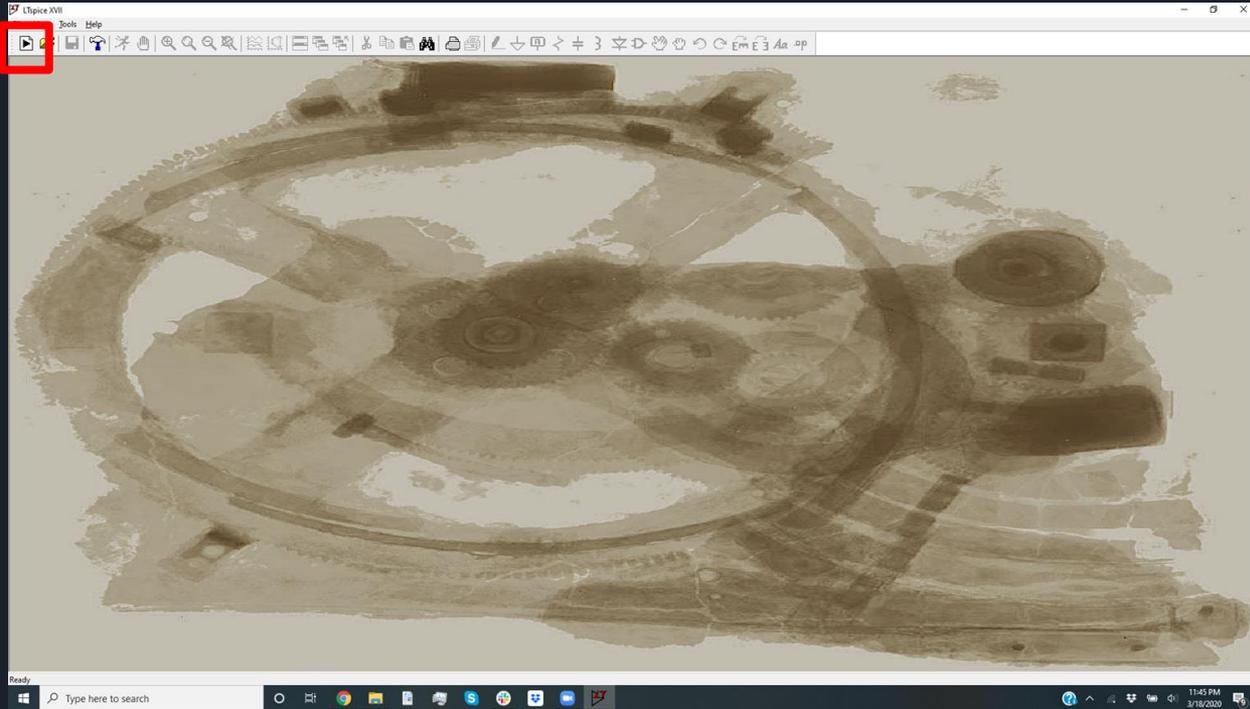


Tutorial 3 Objectives

1. Learn how to make a basic circuit in LTspice
2. Learn how to use LTspice's circuit analysis tools
3. Determine the time constant of an RC circuit

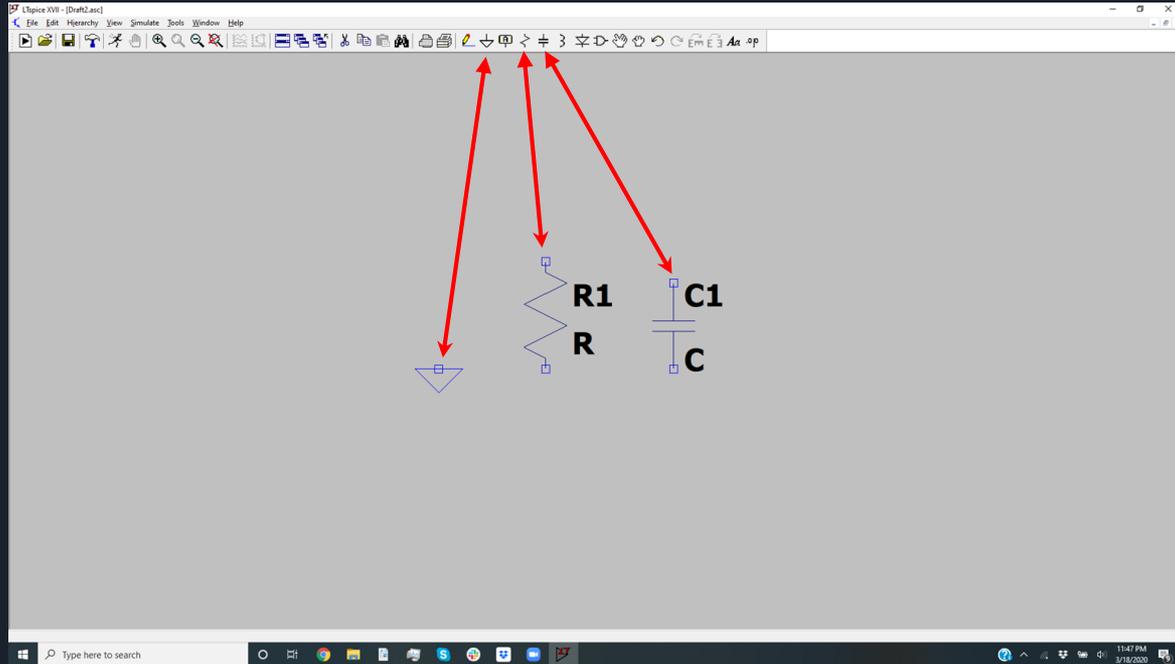
Basic circuit creation

- Begin by opening LTspice and creating a new schematic



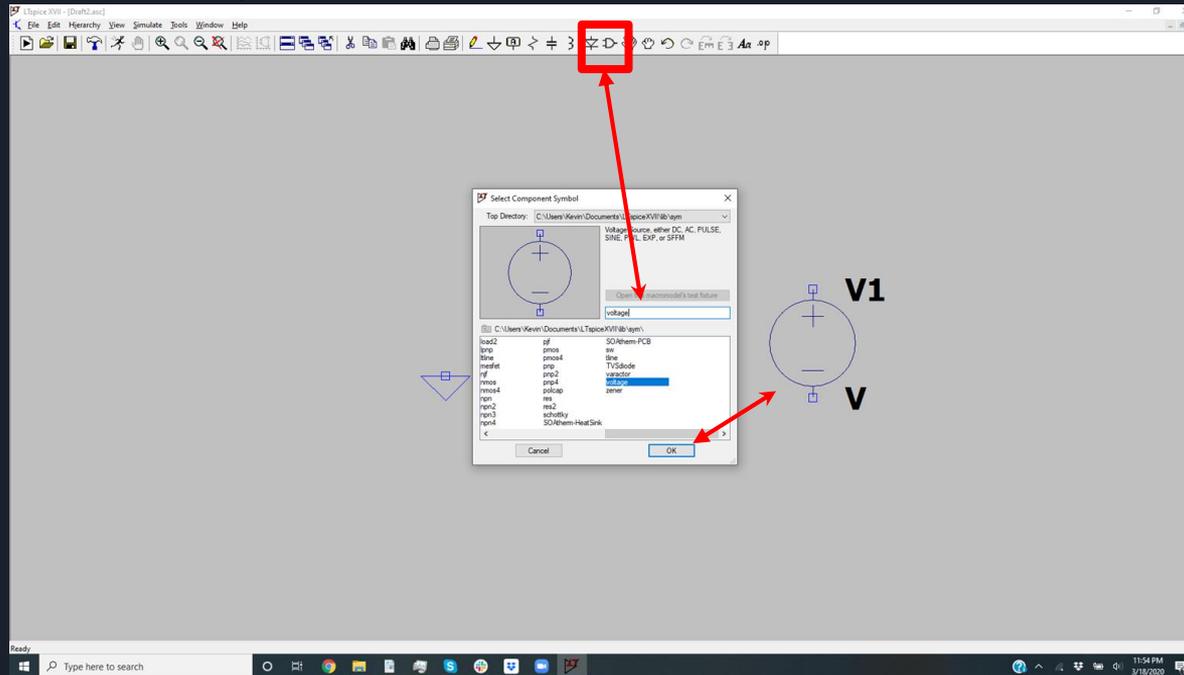
Basic circuit creation

- Now we will import our circuit components
 - a. Ground
 - b. Resistor
 - c. Capacitor



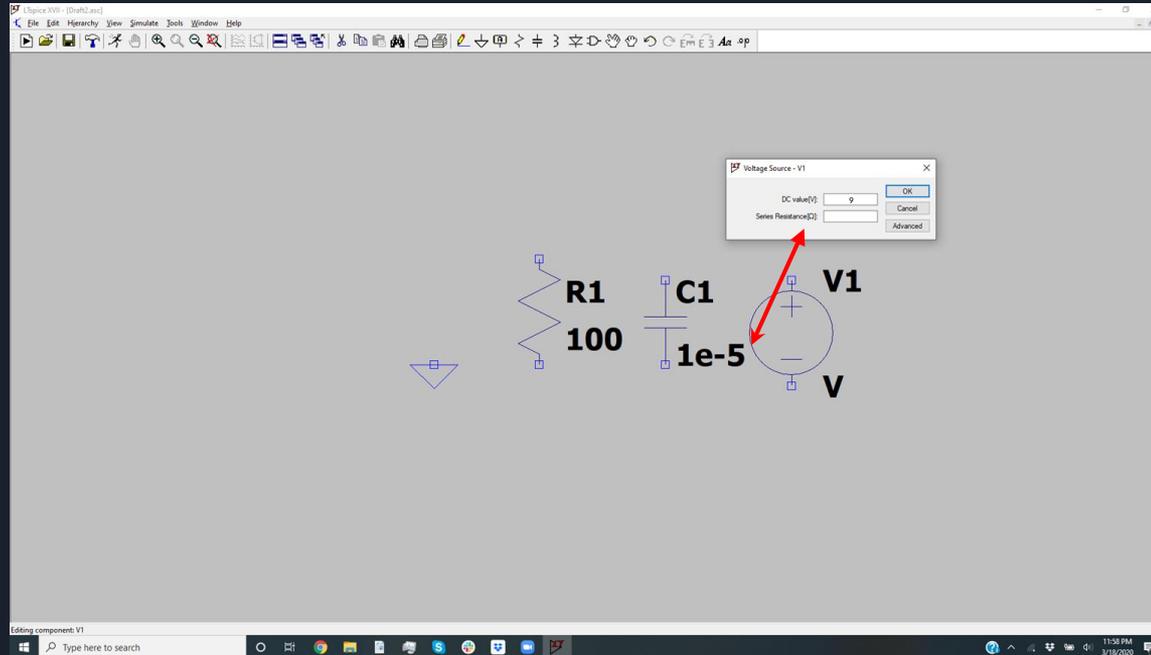
Basic circuit creation

- Now we will import the voltage supply
 - a. Select “components”
 - b. Type “voltage” in the pop up box
 - c. Hit okay



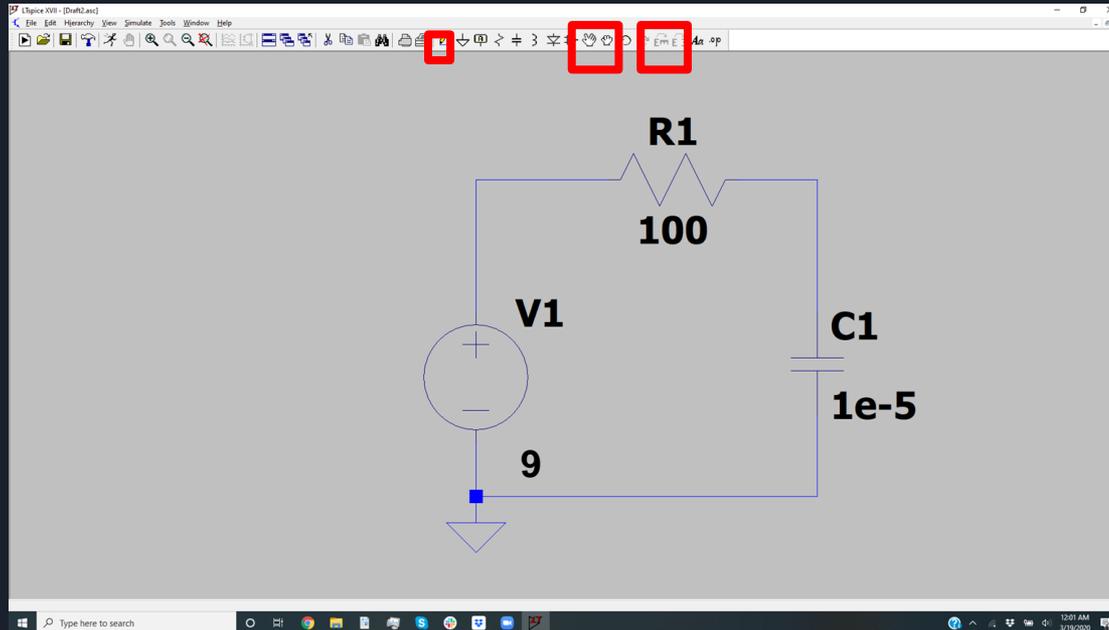
Basic circuit creation

- Now specify the values of your components
 - Right click on them and fill in the appropriate value



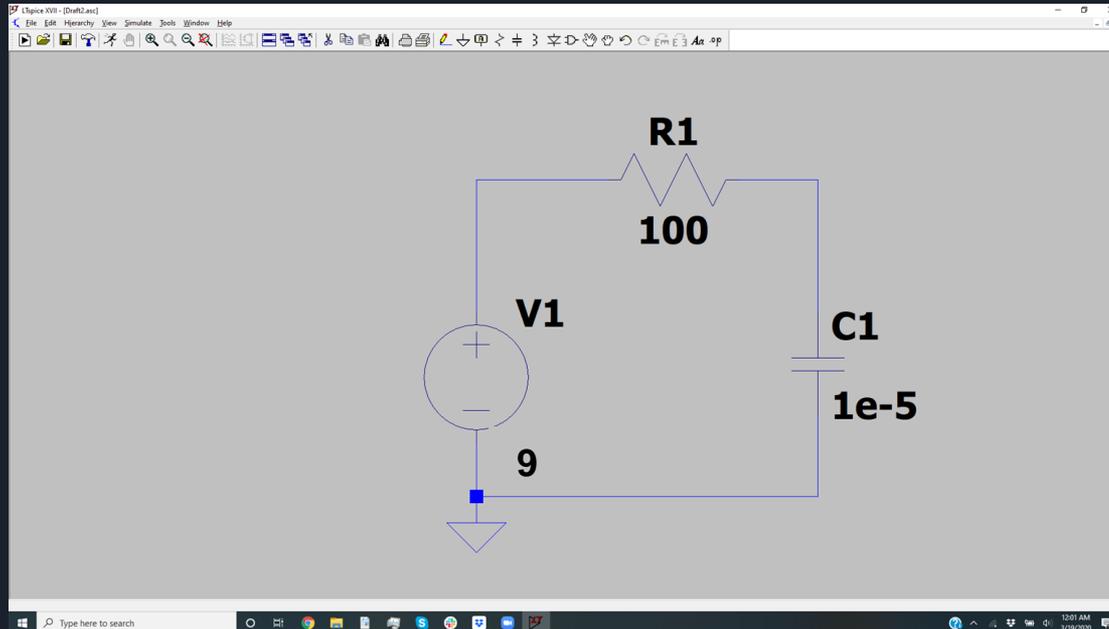
Basic circuit creation

- Connect your circuit components using the wire tool
- Use the move and drag tool to move components
- Use the rotate and flip tools as needed (ctrl+R) is an easy shortcut to rotate



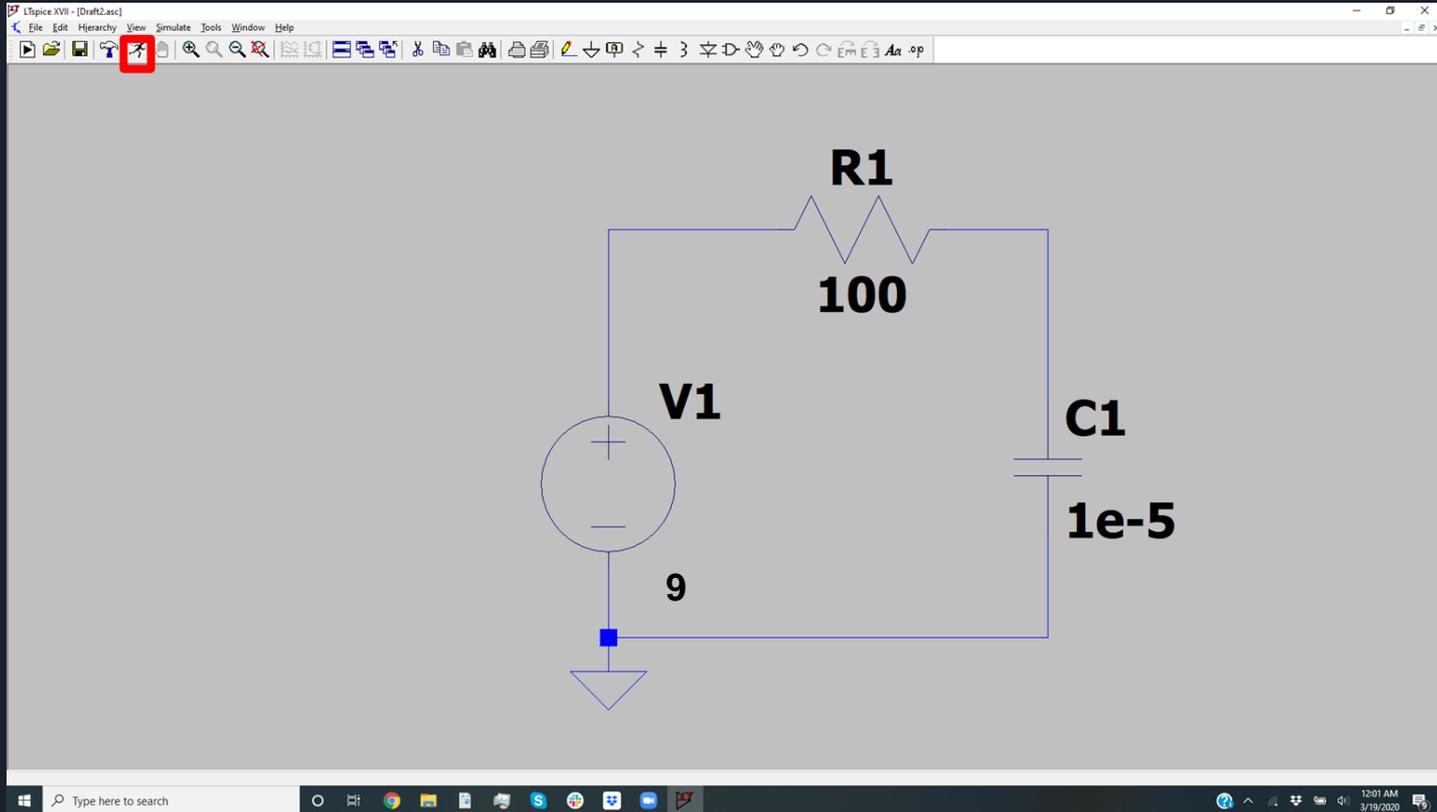
Basic circuit creation

Congratulations! Your first circuit is complete and should look like this:



Basic Circuit Analysis

- Begin by creating a new analysis



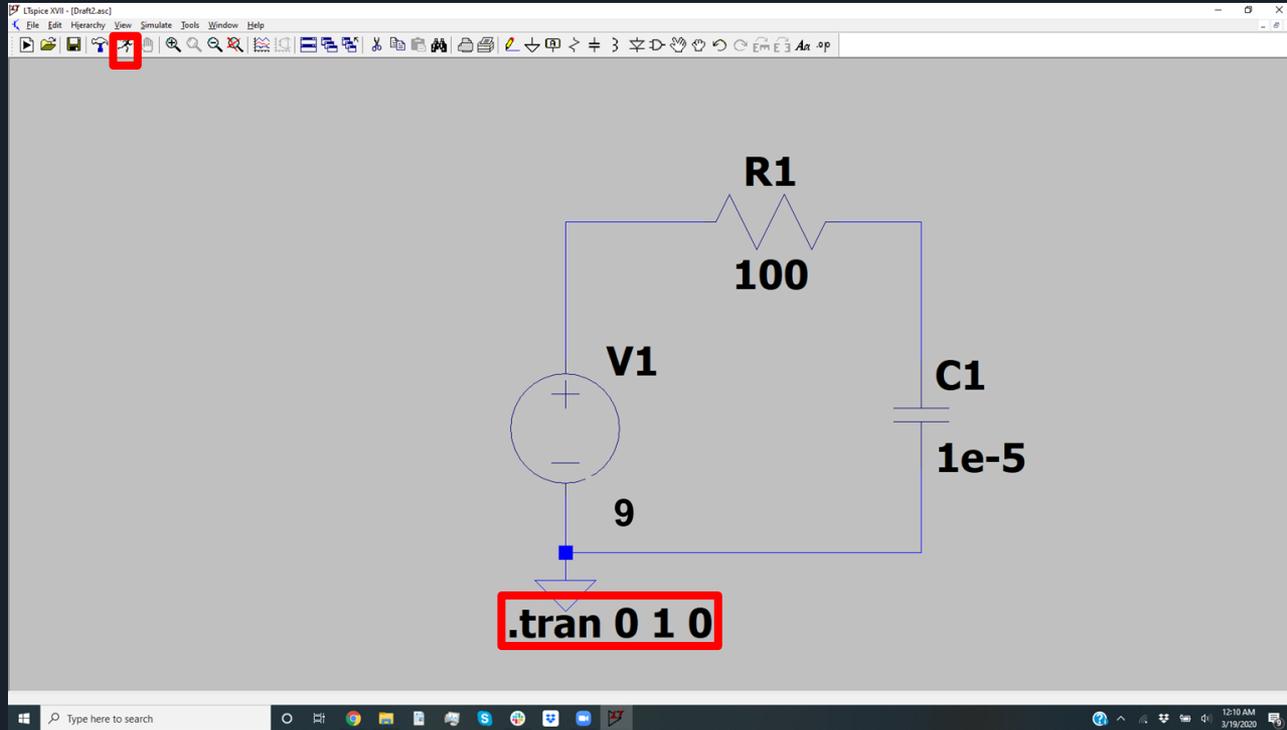
Basic Circuit Analysis

- We will need to set the parameters of our analysis
 - Recommended settings: Transient sweep start time 0 end time 1
 - The start and end times are up to you and can vary a lot from problem to problem

The screenshot displays the LTSpice software interface. On the right, a circuit diagram is shown with a 9V DC voltage source (V1), a 100Ω resistor (R1), and a 1e-5F capacitor (C1) connected in a loop. On the left, the 'Edit Simulation Command' dialog box is open, with the 'Transient' tab selected. The 'Stop time' is set to 1 and the 'Time to start saving data' is set to 0. The dialog box also includes options for starting external DC supply voltages, stopping simulation on steady state detection, and skipping initial operating point solutions. The syntax field at the bottom contains the command: `tran 0 1 0`.

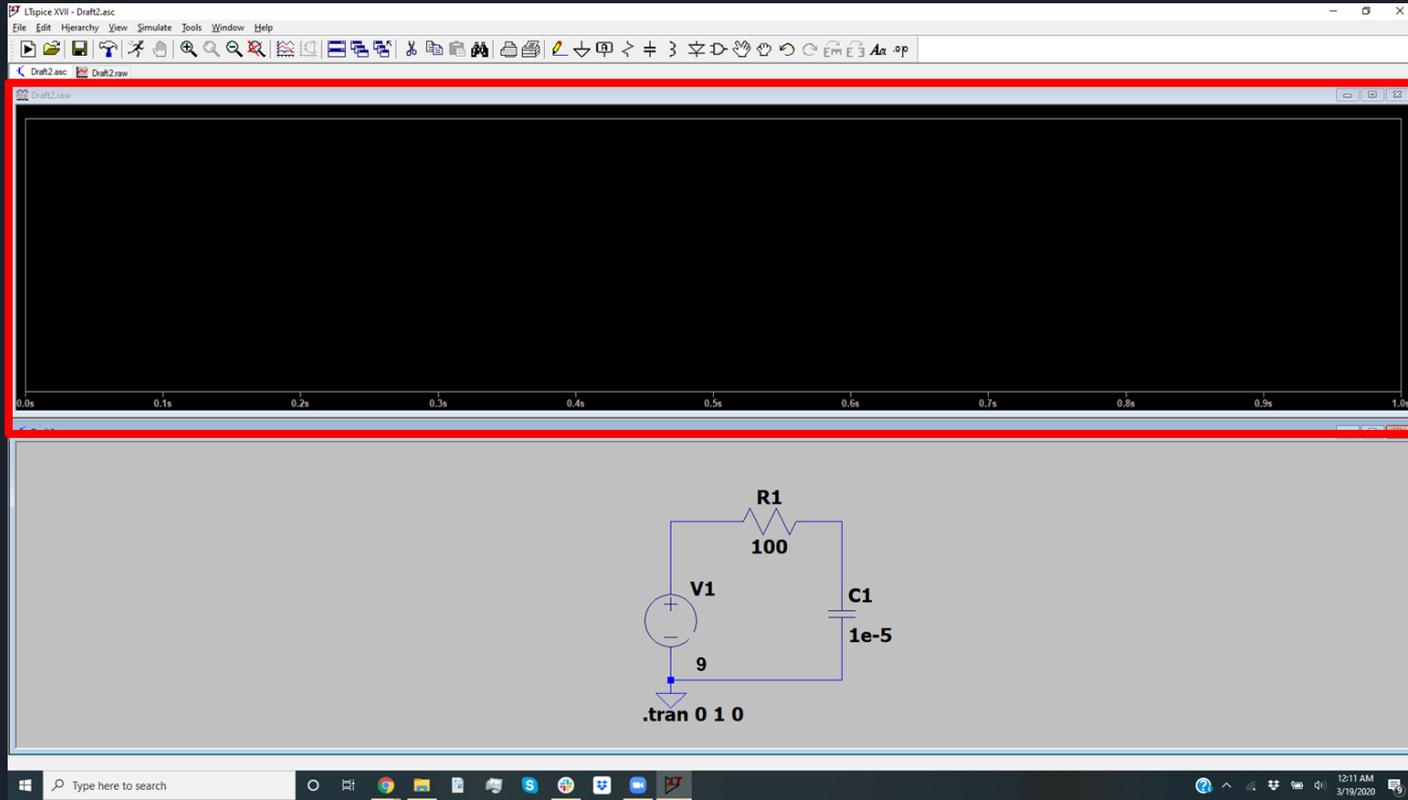
Basic Circuit Analysis

- Now we will need to run our analysis, be patient this will take some time
 - This is done by hitting the “run” button again
 - If simulation parameters need to be changed, right click the analysis object on the model



Basic Circuit Analysis

- An empty plot will appear once the analysis is complete



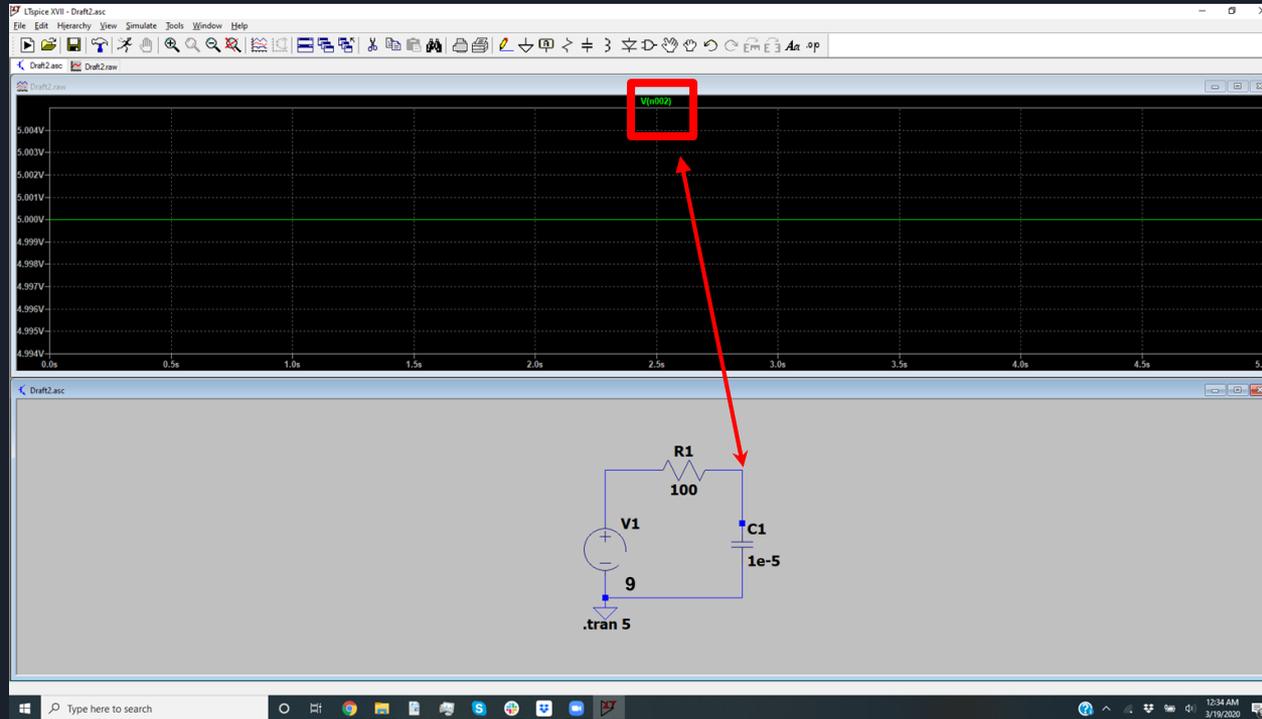
The screenshot displays a circuit simulation software window titled "LTSpice XVII - Draft2.asc". The interface includes a menu bar (File, Edit, Hierarchy, View, Simulate, Tools, Window, Help) and a toolbar with various simulation and editing tools. The main workspace is divided into two sections:

- Plot Area:** A large black rectangular area at the top, outlined in red, which is currently empty. The x-axis at the bottom of this area is labeled with time values from 0.0s to 1.0s in increments of 0.1s.
- Circuit Diagram:** A schematic diagram of a series circuit below the plot area. It consists of a DC voltage source labeled "V1" with a value of "9", a resistor labeled "R1" with a value of "100", and a capacitor labeled "C1" with a value of "1e-5". The circuit is connected to ground. Below the diagram, the simulation command `.tran 0 1 0` is visible.

The Windows taskbar at the bottom shows the system clock as 12:11 AM on 3/19/2020.

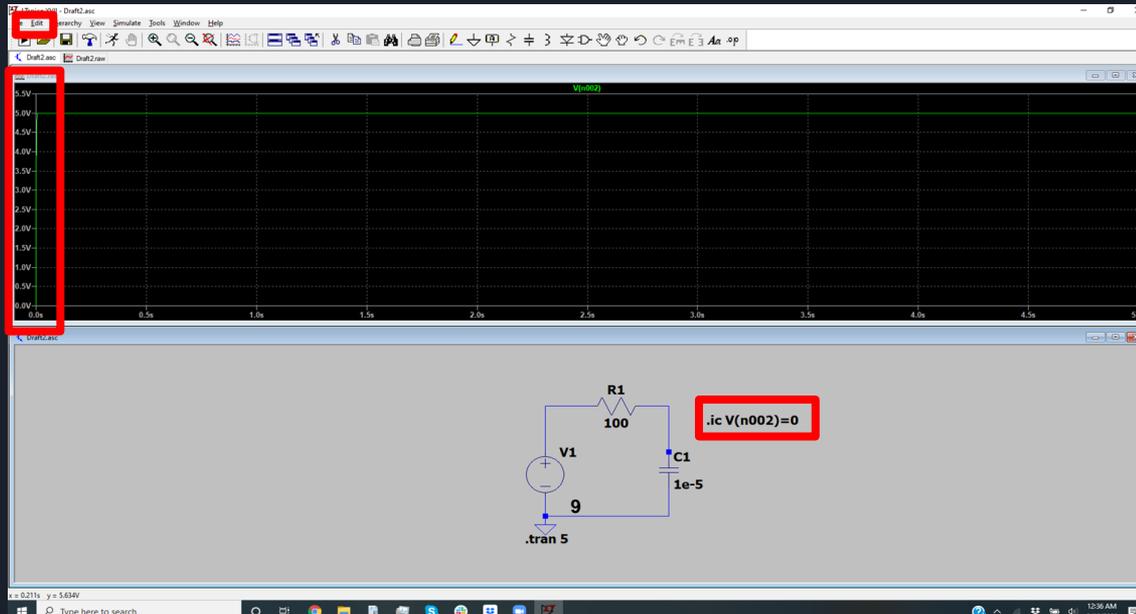
Basic Circuit Analysis

- Select the node on the circuit you want to probe and its waveform (called a trace) will appear on the plot
 - Multiple nodes can be selected, resulting in multiple traces being plotted



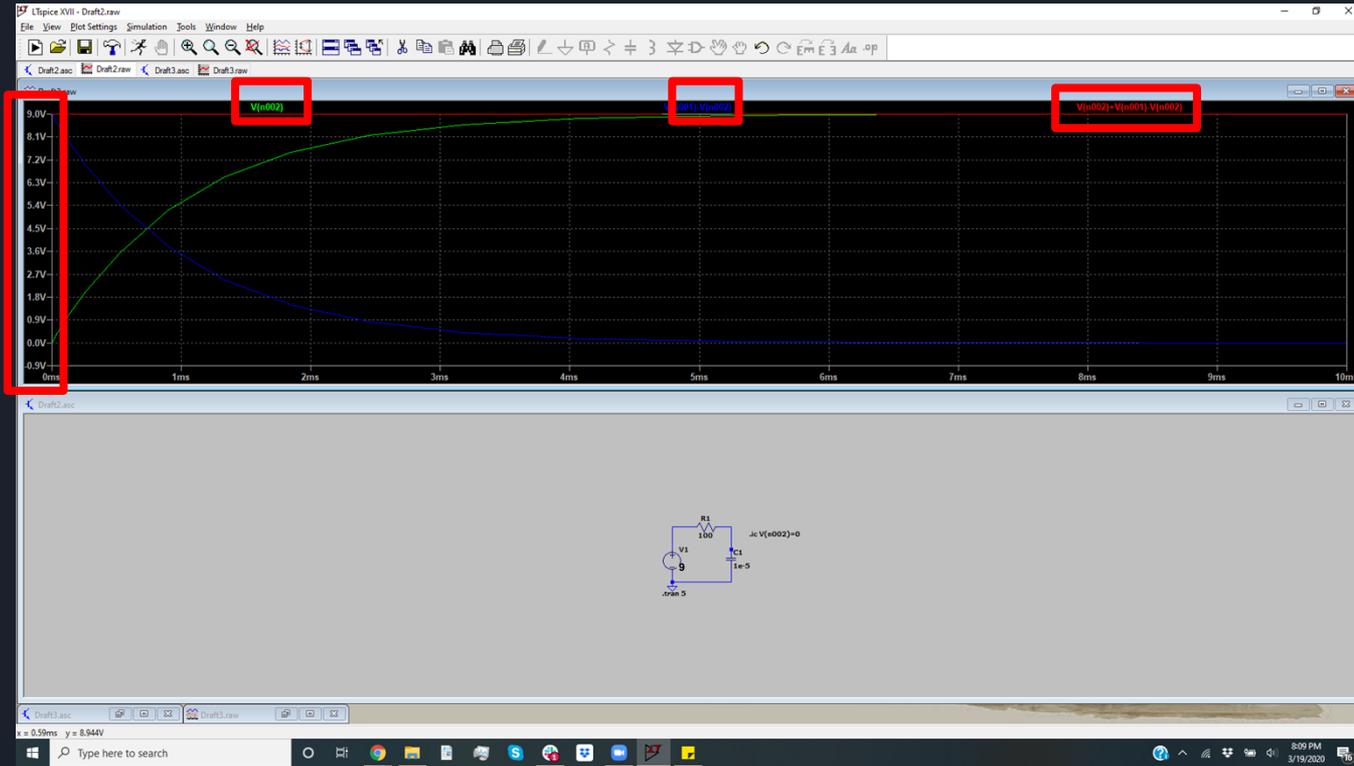
Basic Circuit Analysis

- That's not right
 - We expect an exponential, but we don't
- Need to set initial condition
 - edit-> SpiceDirectives-> type ".ic V(n002)=0" -> hit ok
 - .ic V(n002)=0 -> initial condition of Voltage of node 002 is 0
 - We do this because LT spice doesn't assume nodes are at 0 when the simulation starts



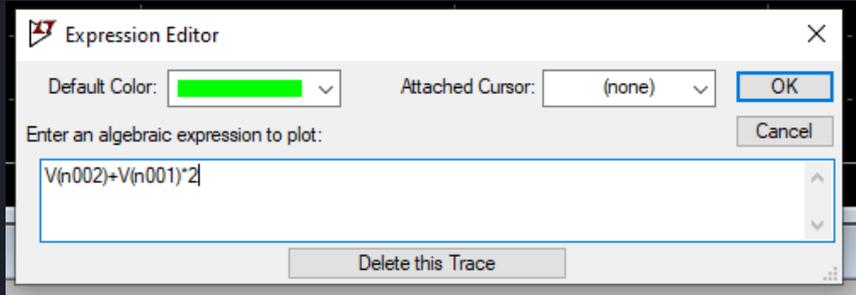
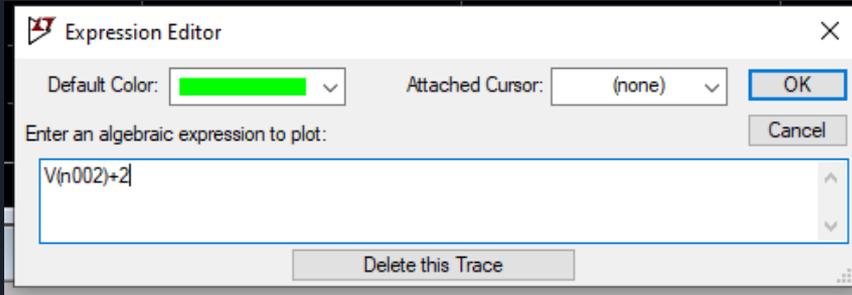
Basic Circuit Analysis

- Once a trace is plotted, algebraic manipulations can be performed with the waveform by right clicking it
 - Example: We can subtract voltages to do KVL



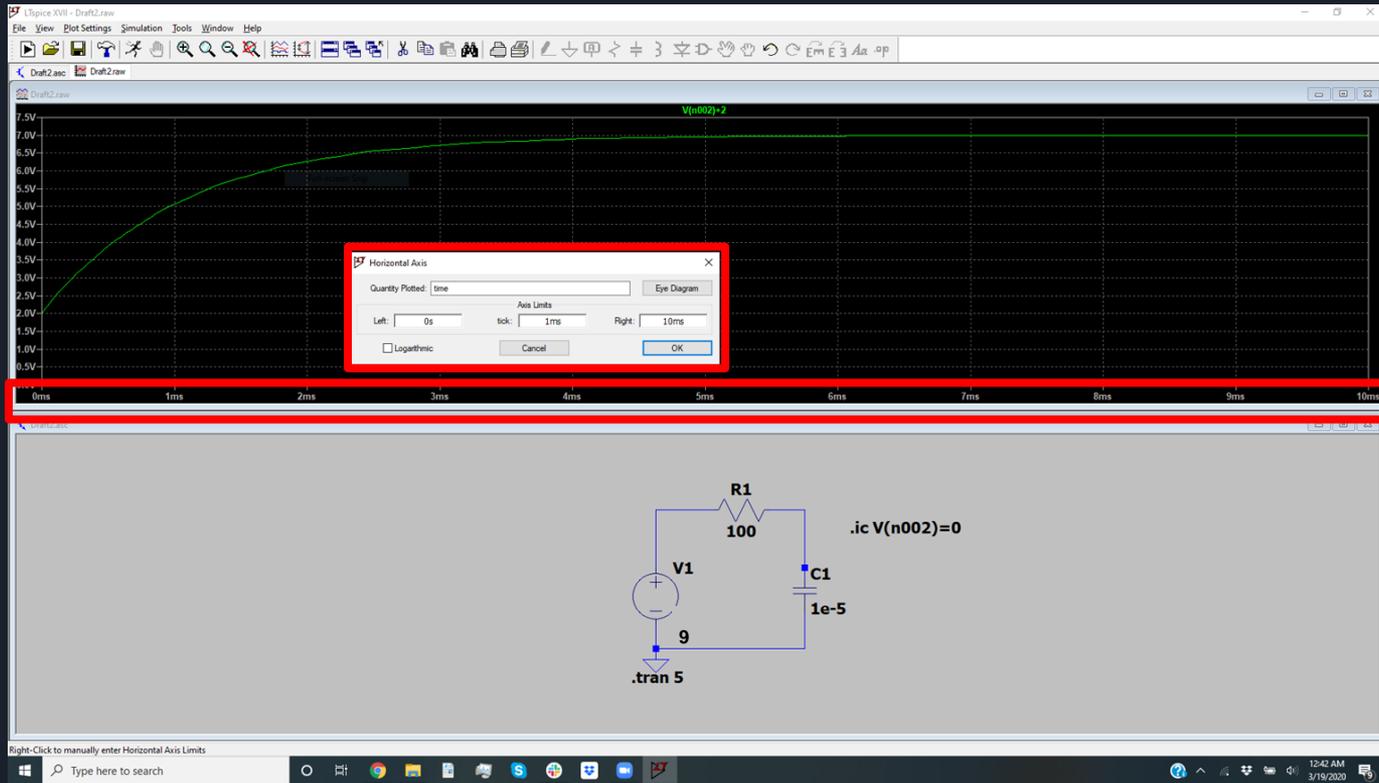
Basic Circuit Analysis

- We do this by right clicking on the trace label on the top of the plot then entering the expression we want to plot
- We can do basic algebraic operations using both different traces as well as constants
- We can rename nodes to make this process easier as well (this is covered in more detail in tutorial 4)



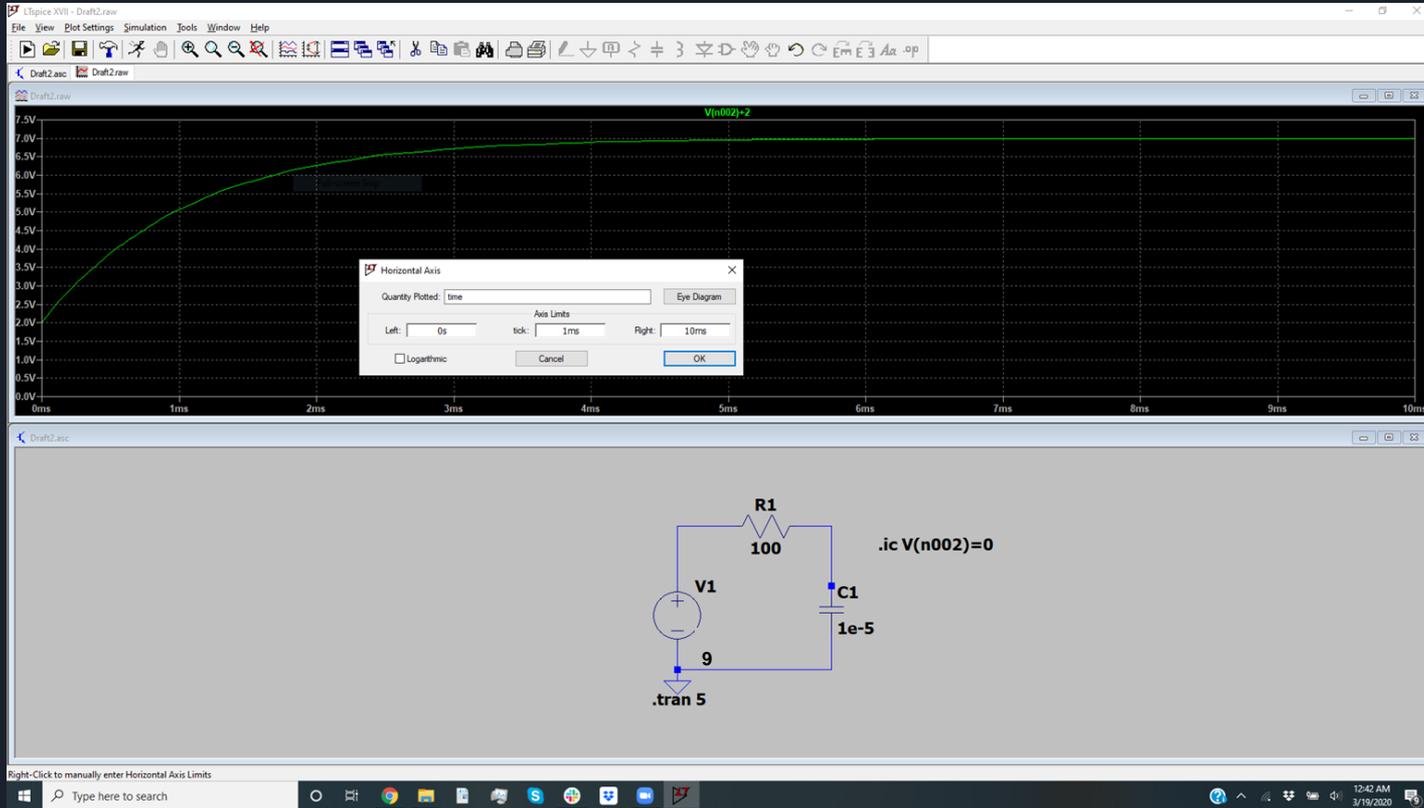
Basic Circuit Analysis

- Just like an oscilloscope we may need to change the scaling to see good data. This is done by right clicking the axis and changing the limits



Basic Circuit Analysis

- Now we can see the waveform we want





Basic Circuit Analysis

Congratulations! You have completed your first analysis of a circuit in LTspice!



Performing basic RC circuit analysis

Assignment Submit plots and circuit schematics of the following 3 waveforms:

Create 3 RC circuits, and display their waveforms (hint change resistor and capacitor values to see what happens)

Waveform 1: Time constant 500 ms, 5V amplitude, 0V offset

Waveform 2: Time constant 200 ms, 2V amplitude, 10V offset

Waveform 3: next page

Performing basic RC circuit analysis

Assignment:

Create 3 RC circuits, and display their waveforms (hint change resistor and capacitor values to see what happens)

Waveform 3: right click on the voltage source, go to advanced settings and configure the source as below, and plot the voltage across the capacitor

