LTspice tutorial
Part 3- Basic circuits

Prerequisites

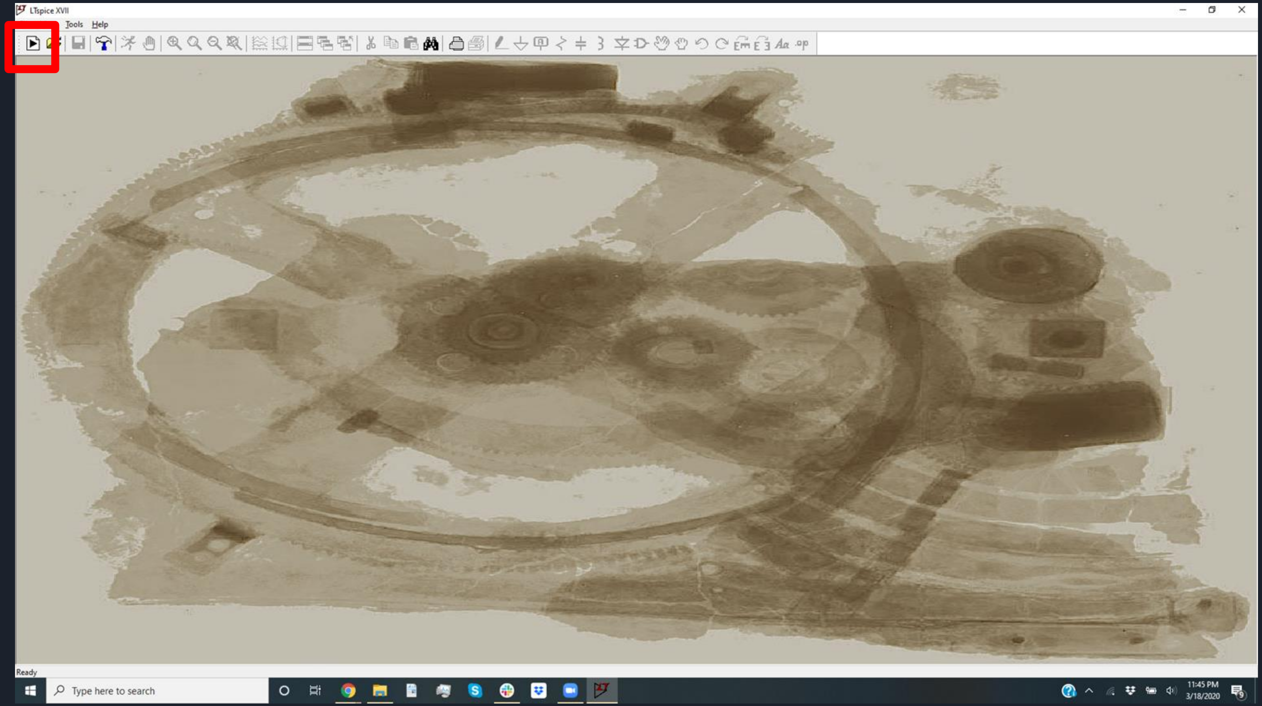
- Please make sure you have completed the following:
  - LTspice tutorial part 1 (download and installation)
  - LTspice tutorial part 2 (components and basic interface)

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

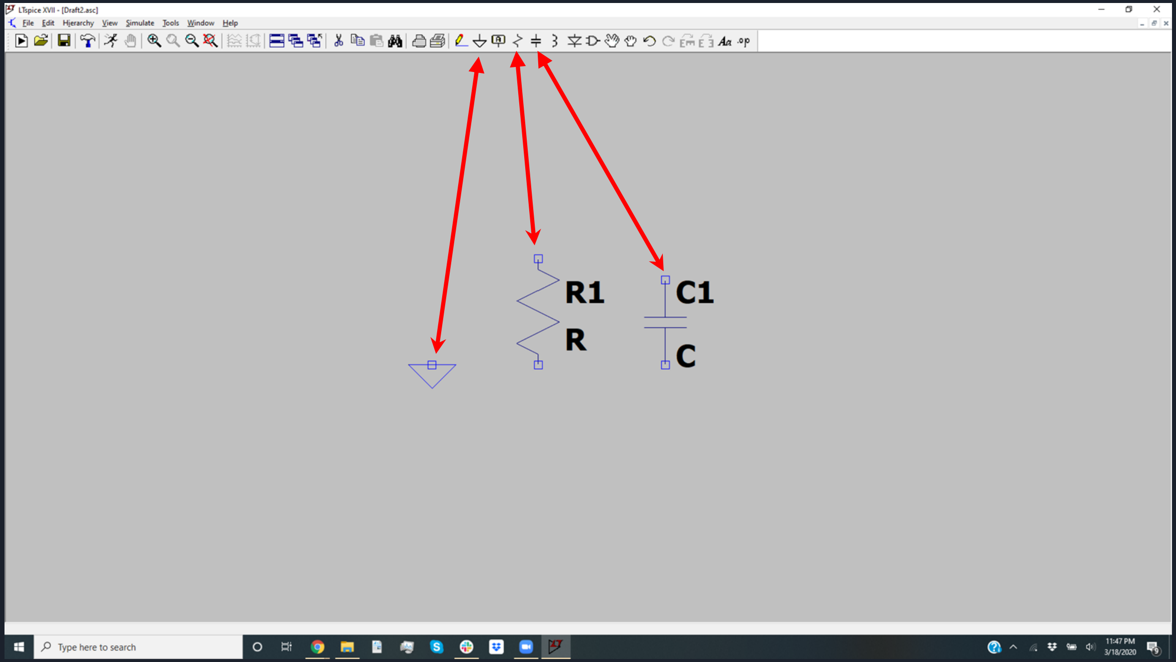
Basic circuit creation

- Begin by opening LTspice and creating a new schematic



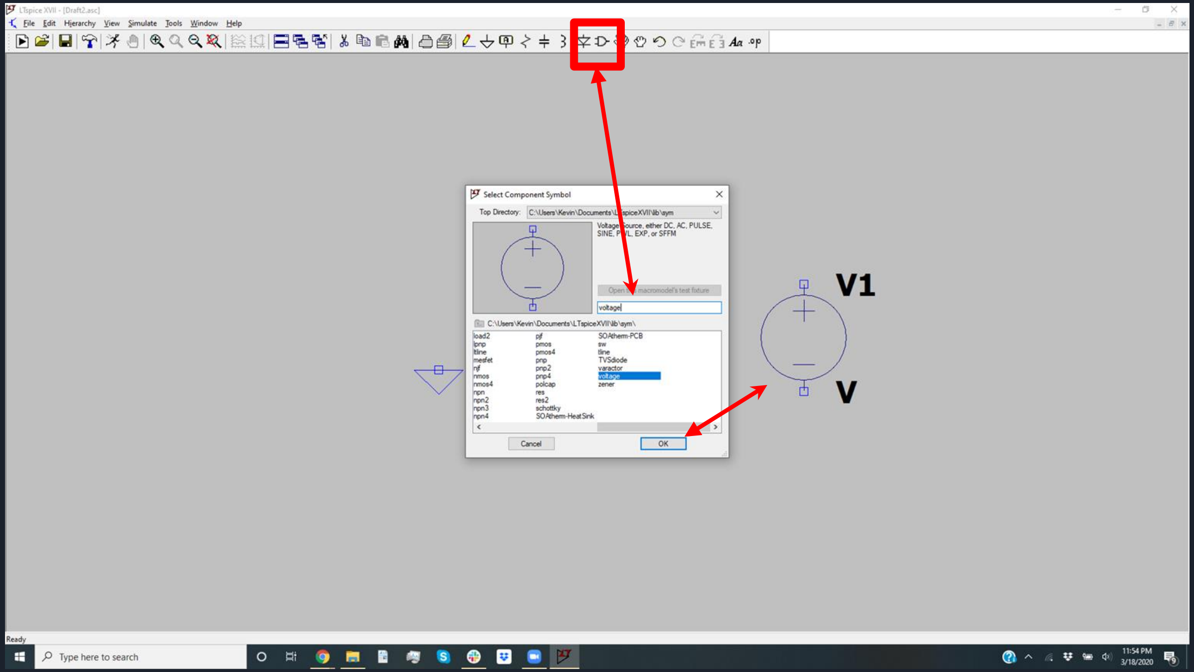
Basic circuit creation

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



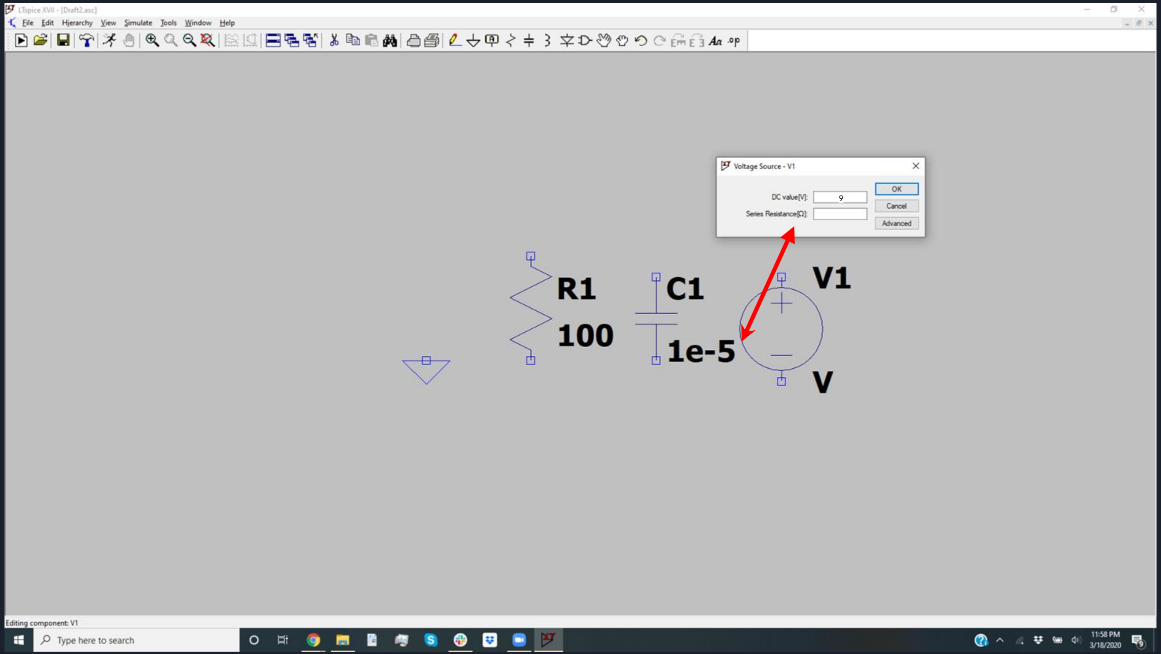
Basic circuit creation

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



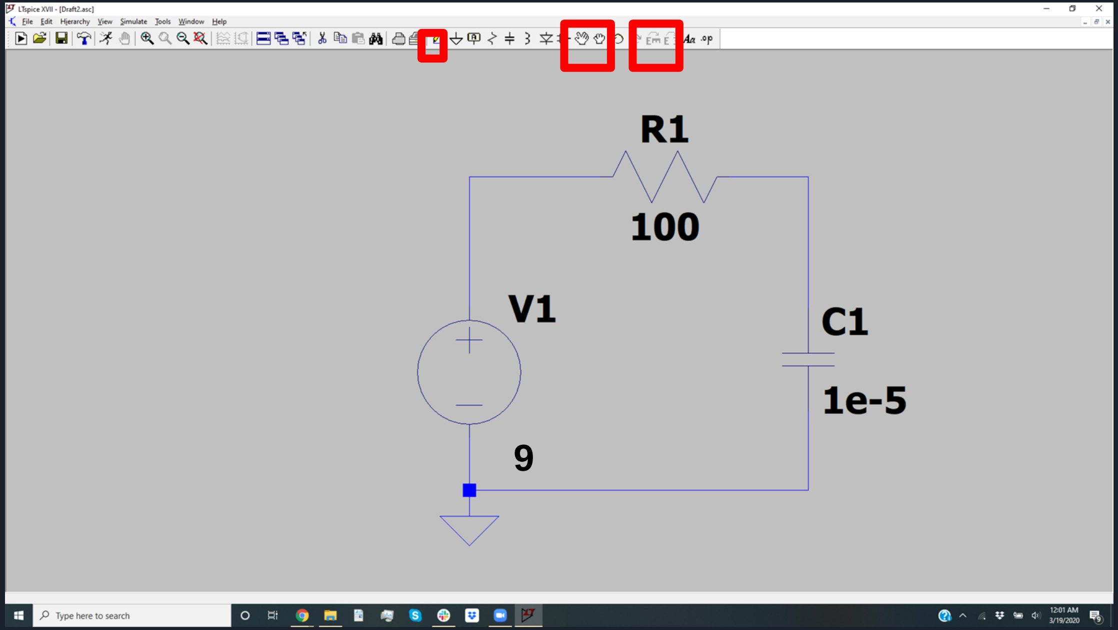
Basic circuit creation

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



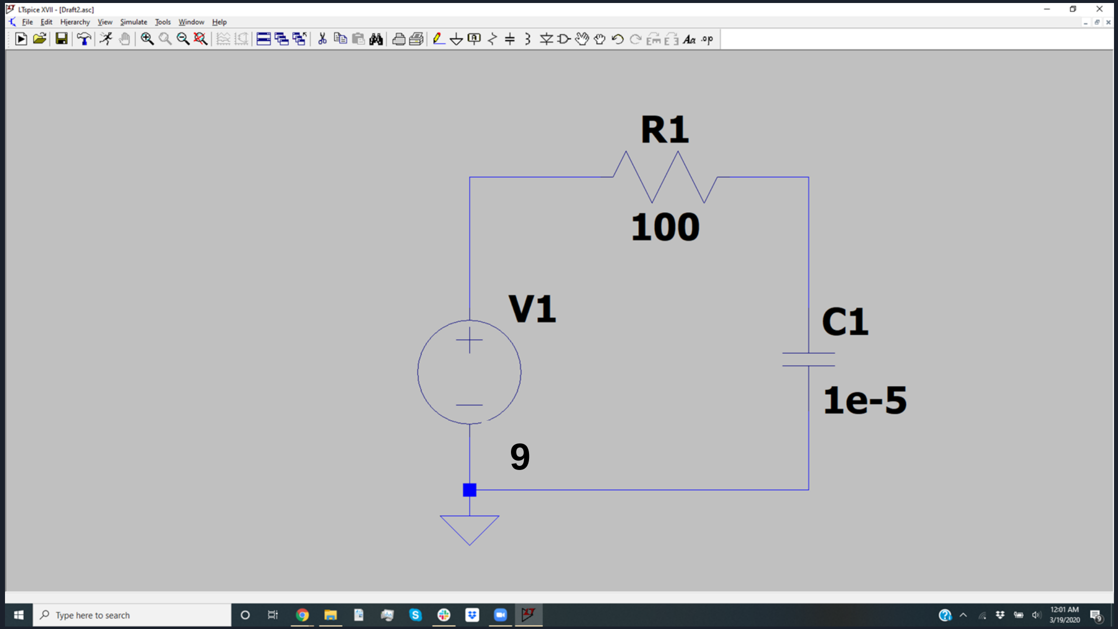
Basic circuit creation

- 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



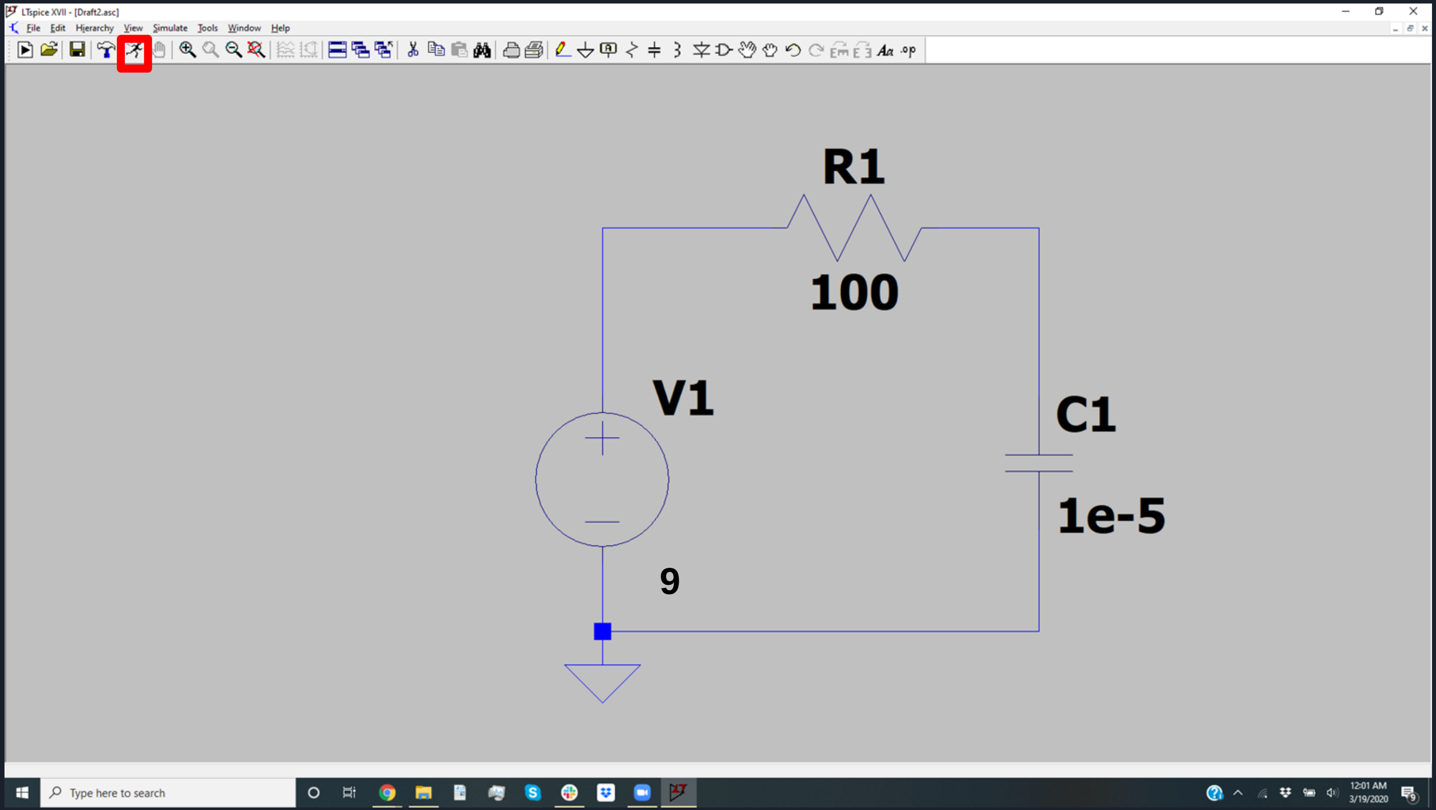
Basic circuit creation

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



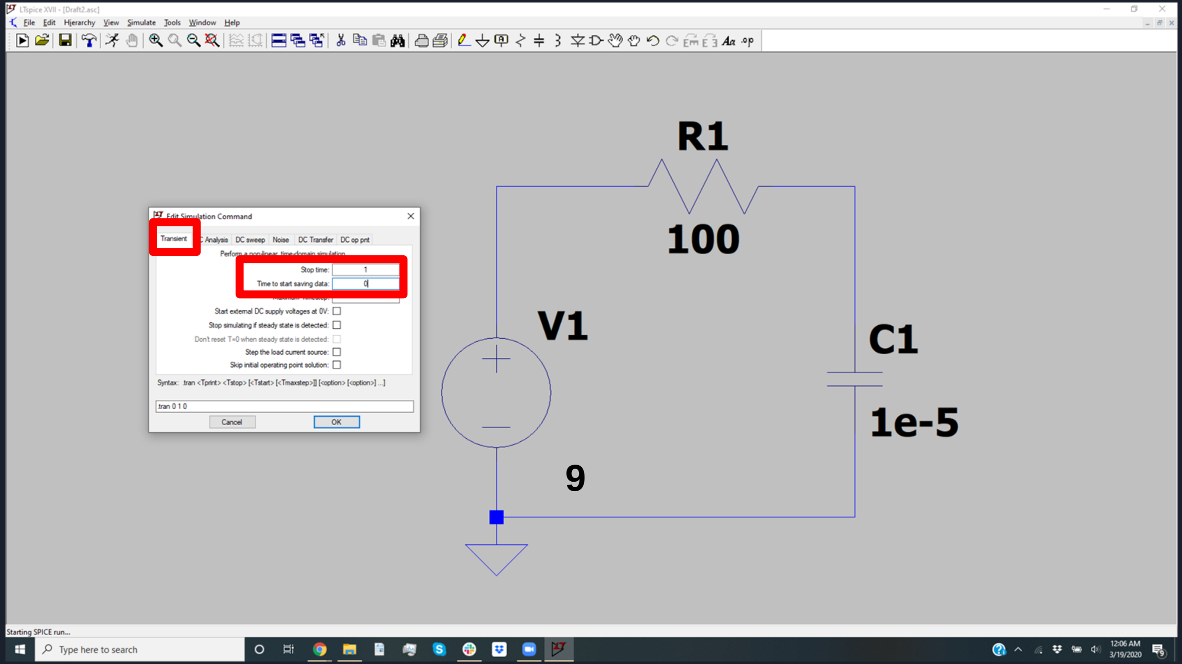
Basic Circuit Analysis

- Begin by creating a new analysis



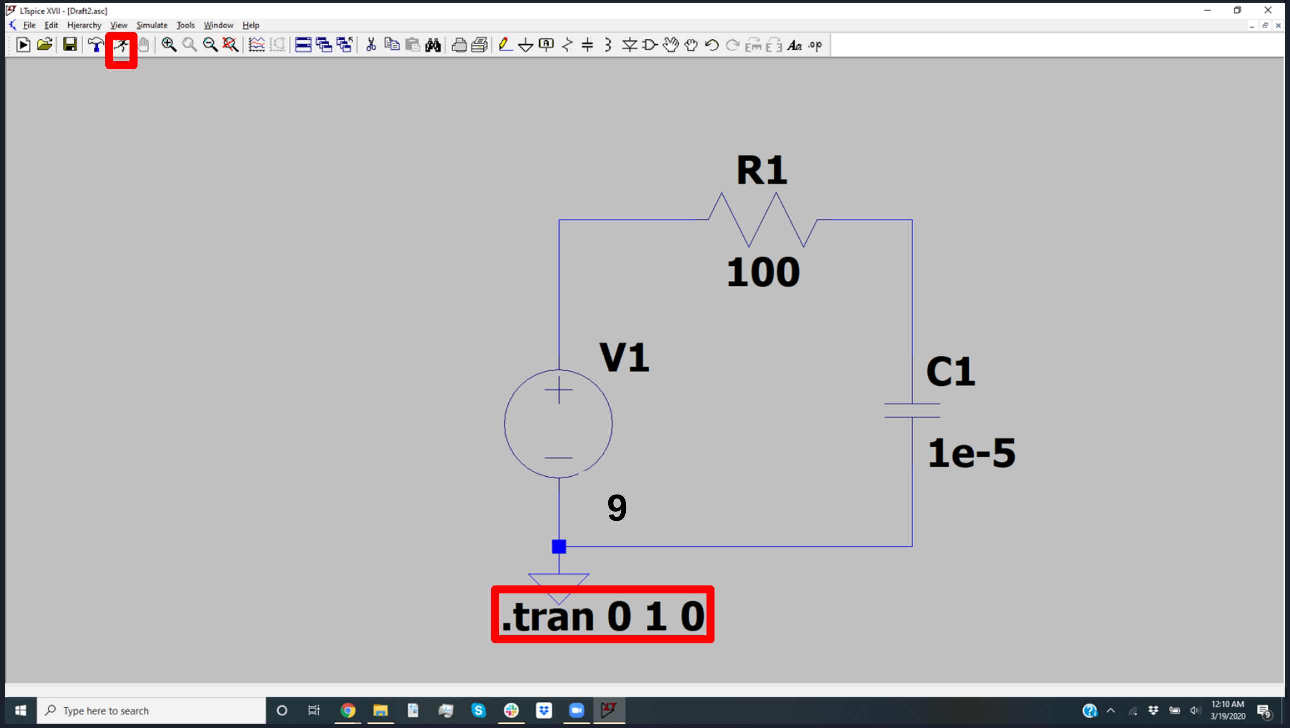
Basic Circuit Analysis

- 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



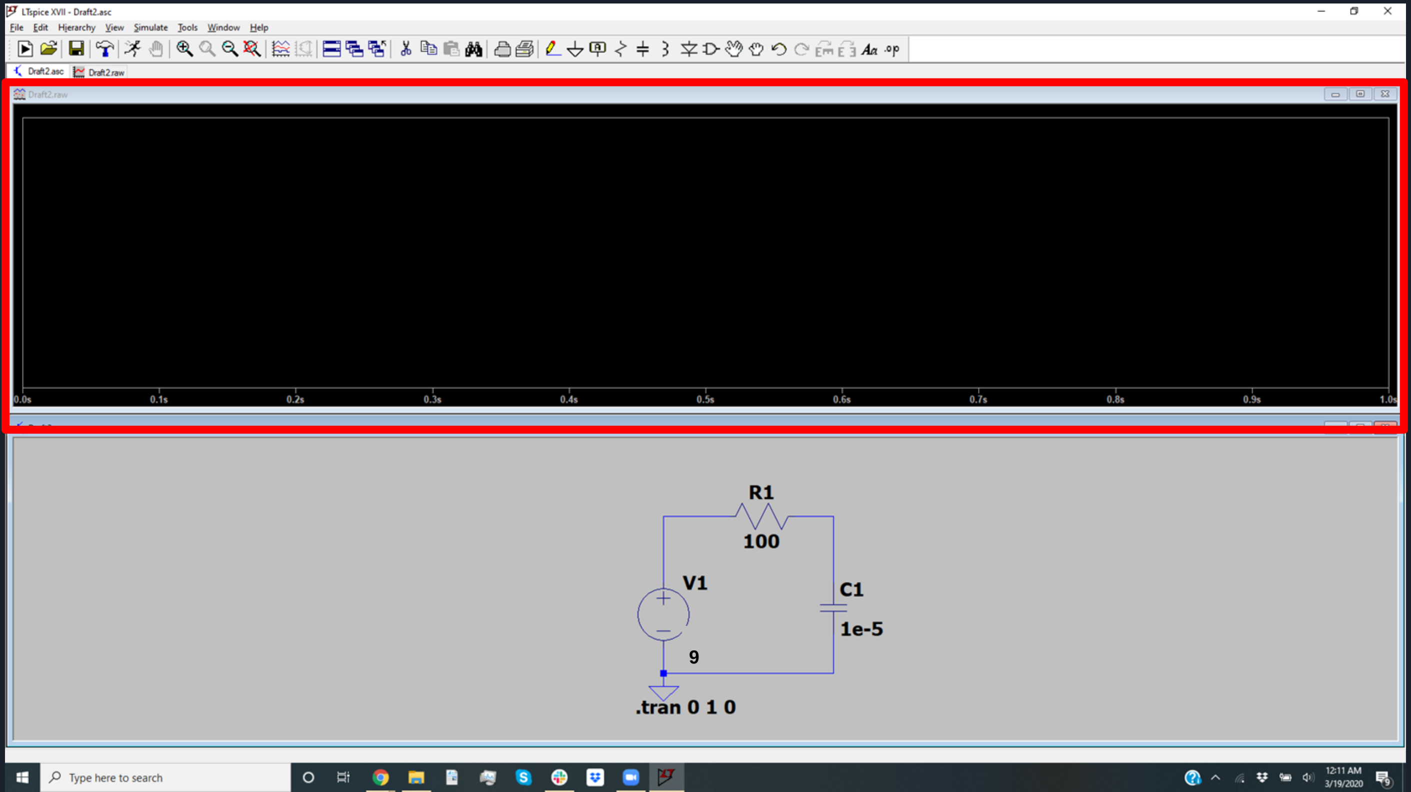
Basic Circuit Analysis

- 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



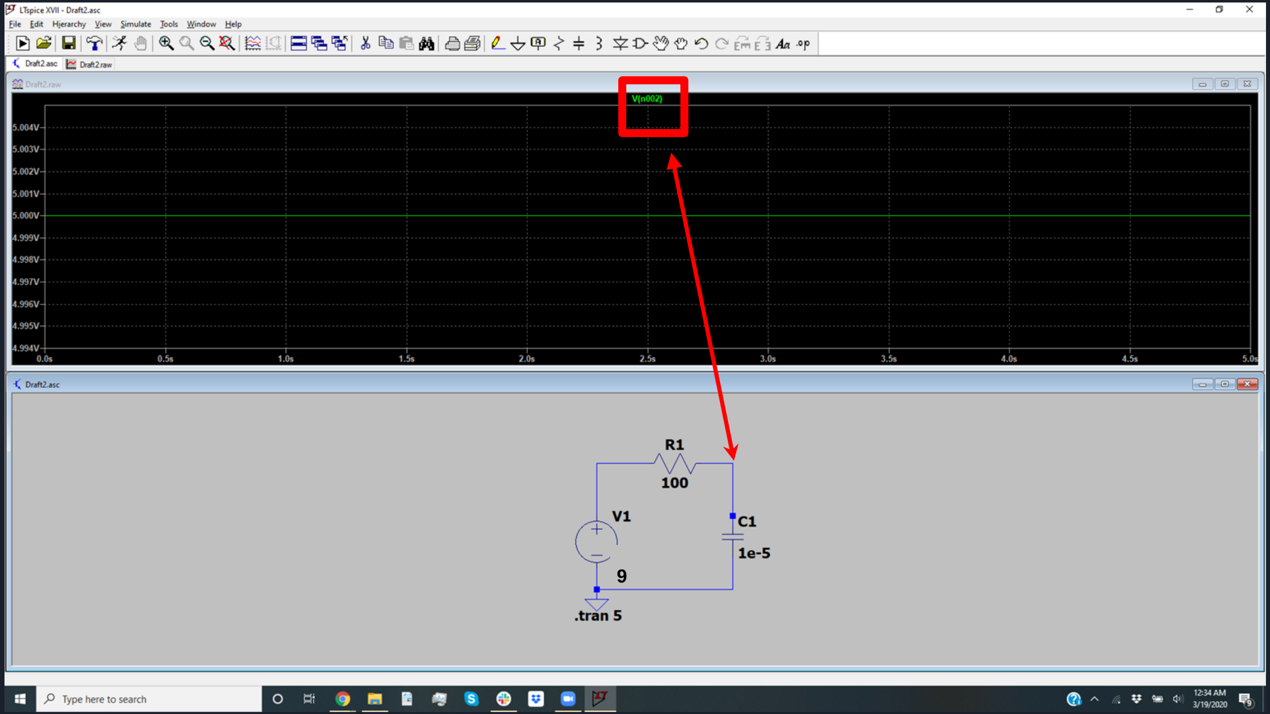
Basic Circuit Analysis

- An empty plot will appear once the analysis is complete



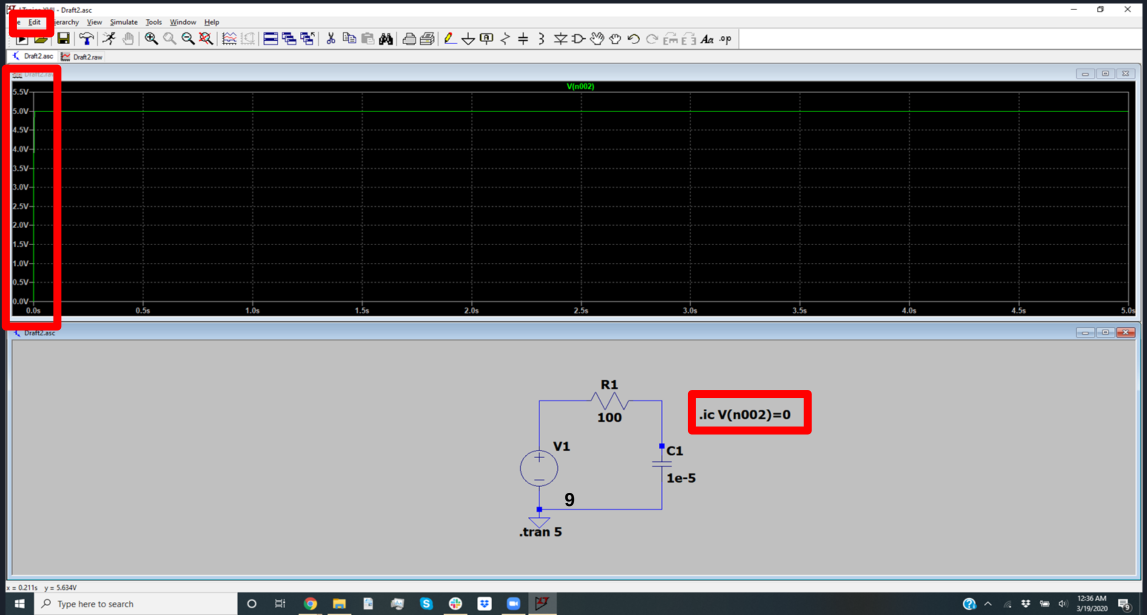
Basic Circuit Analysis

- 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



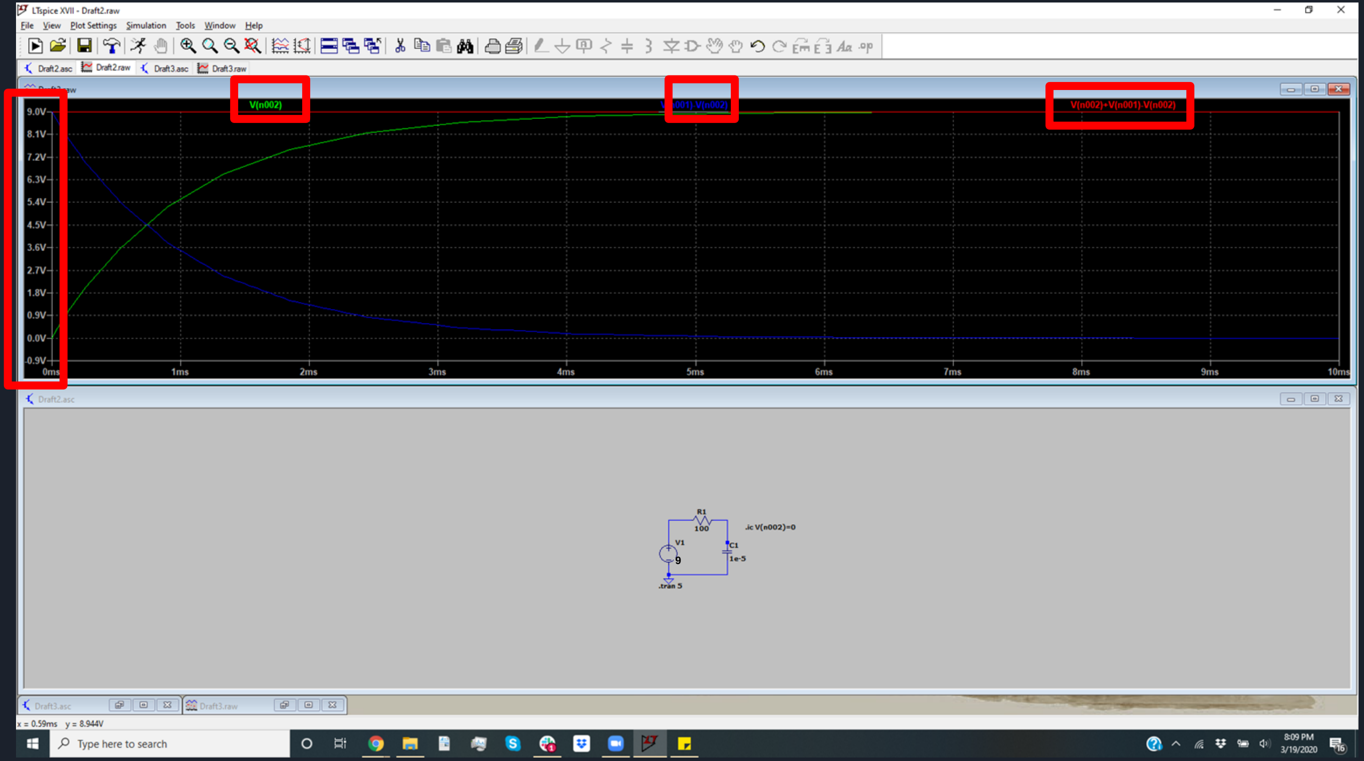
Basic Circuit Analysis

- 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 -> <u>i</u>nitial <u>c</u>ondition of <u>V</u>oltage of <u>n</u>ode <u>002</u> is <u>0</u>
  - We do this because LT spice doesn't assume nodes are at 0 when the simulation starts



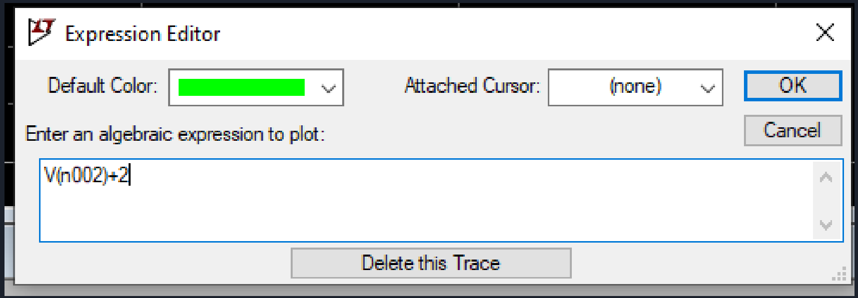
Basic Circuit Analysis

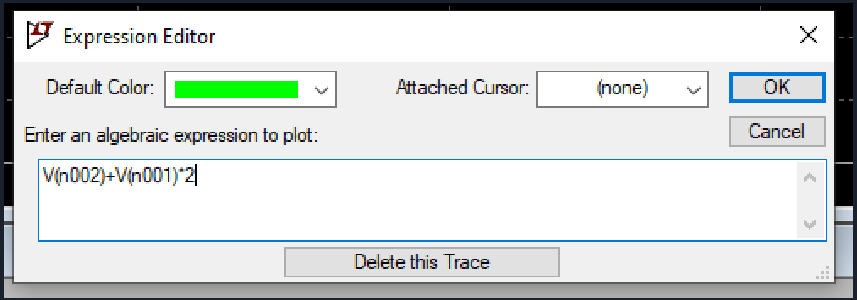
- Once a trace is plotted, algebraic manipulations can be performed with the waveform by right clicking it
  - Example: We can subtract voltages to do KVL



Basic Circuit Analysis

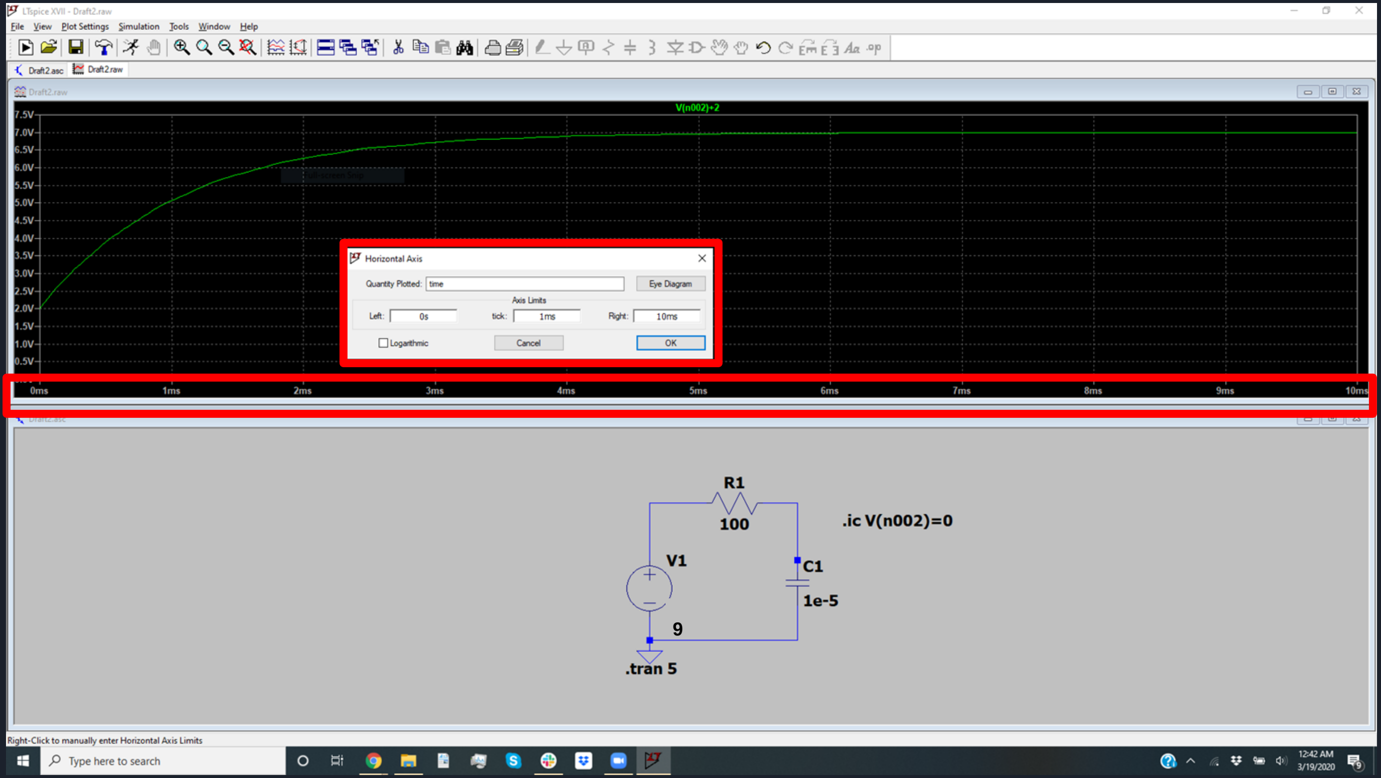
- We do this by right clicking on the trace label on the top of the plot then entering the expression we want to plot
- We can do basic algebraic operations using both different traces as well as constants
- We can rename nodes to make this process easier as well (this is covered in more detail in tutorial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





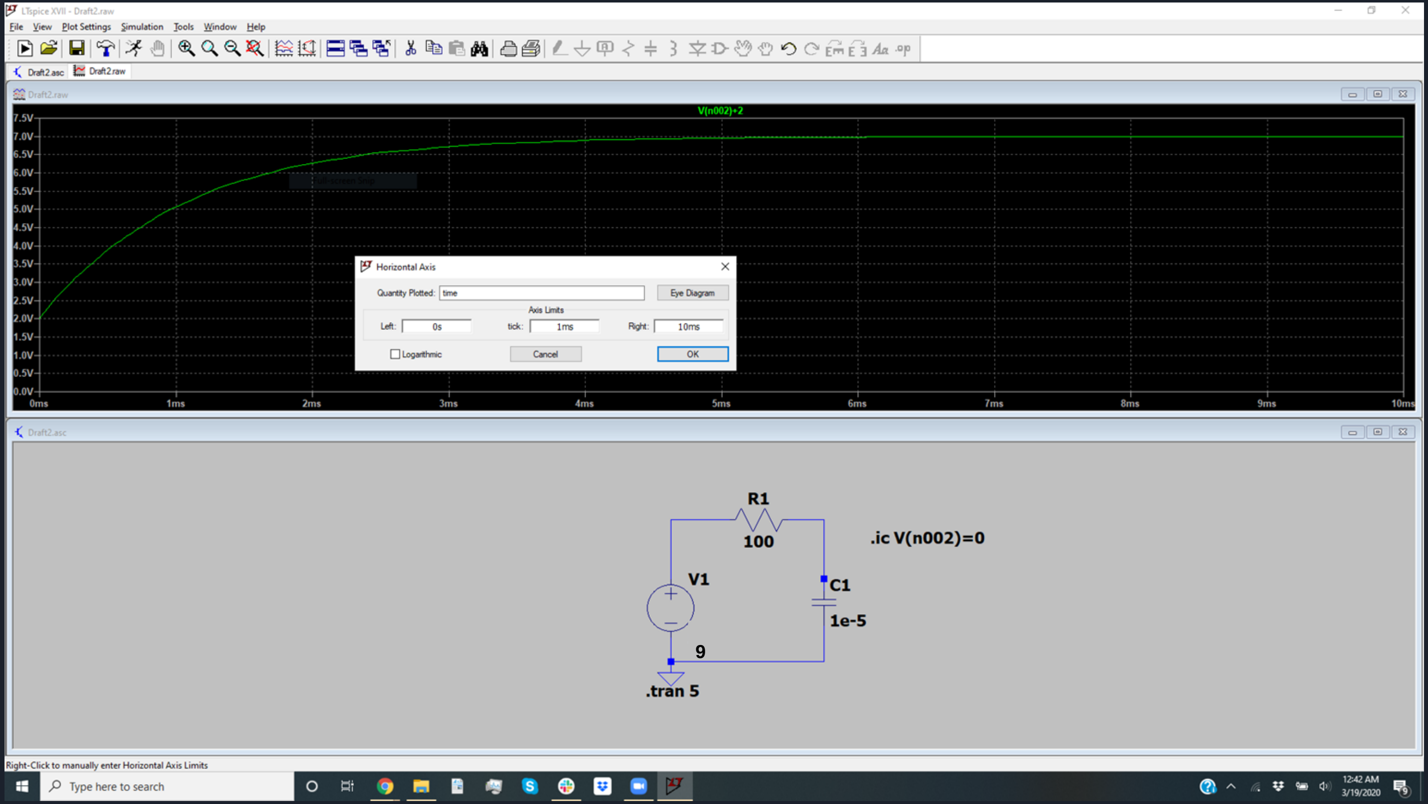
Basic Circuit Analysis

- 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