LTspice Tutorial Part 4- Intermediate 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)
  - LTspice tutorial part 3 (basic circuts)

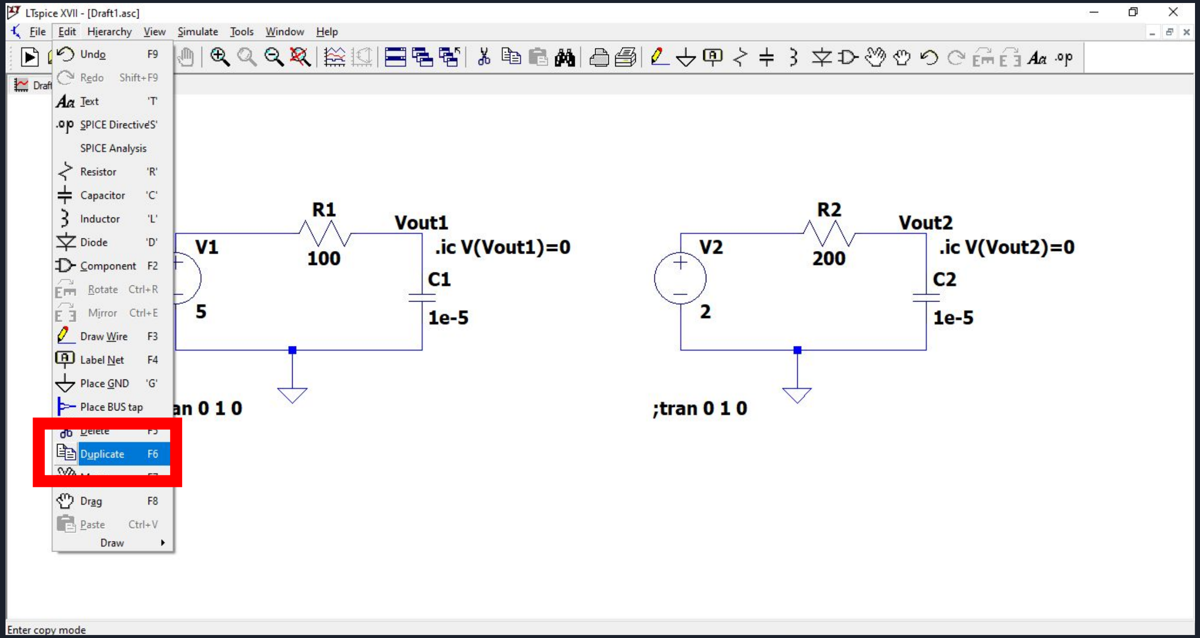
Tutorial 4 Objectives

1. Improve understanding of plotting functions in LT spice
2. Learn how to specify Integrated Circuits (IC’s) for use in LTspice
3. Create a basic Oscillator circuit and observe it’s behavior

Improving plotting

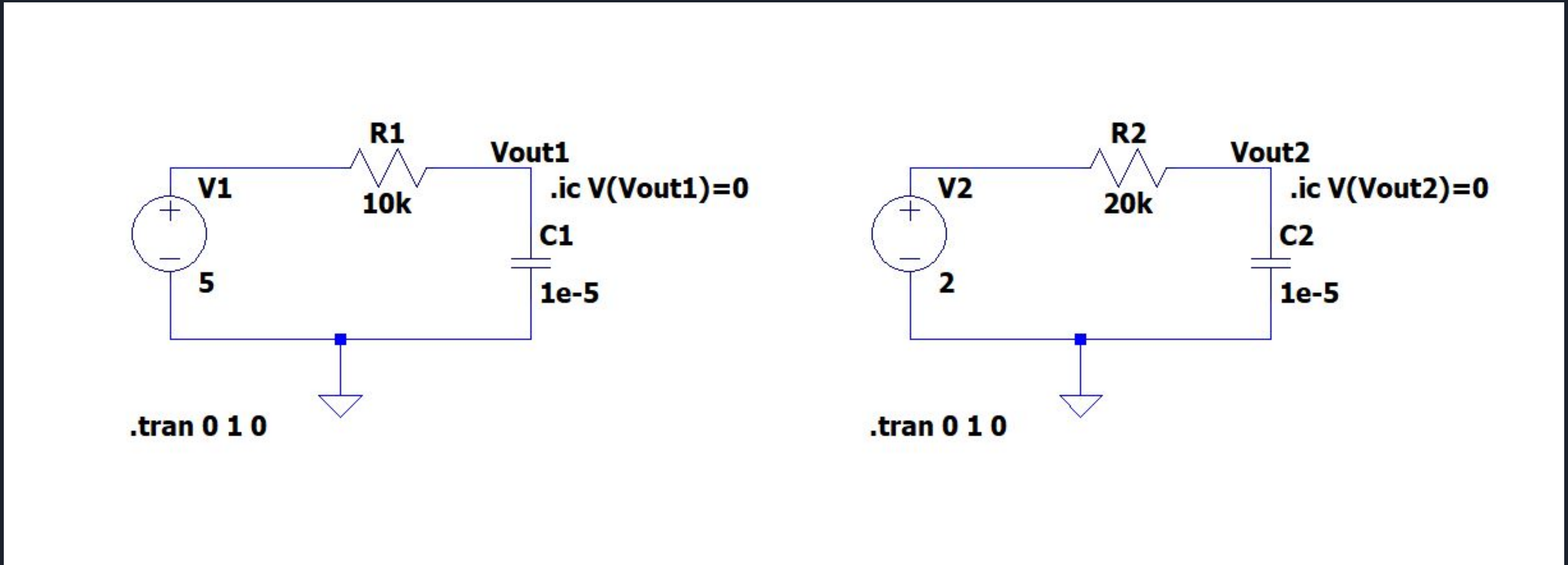
- Begin with the circuit from tutorial 3
- We will plot multiple different circuits in a clearer way

To duplicate the same circuit



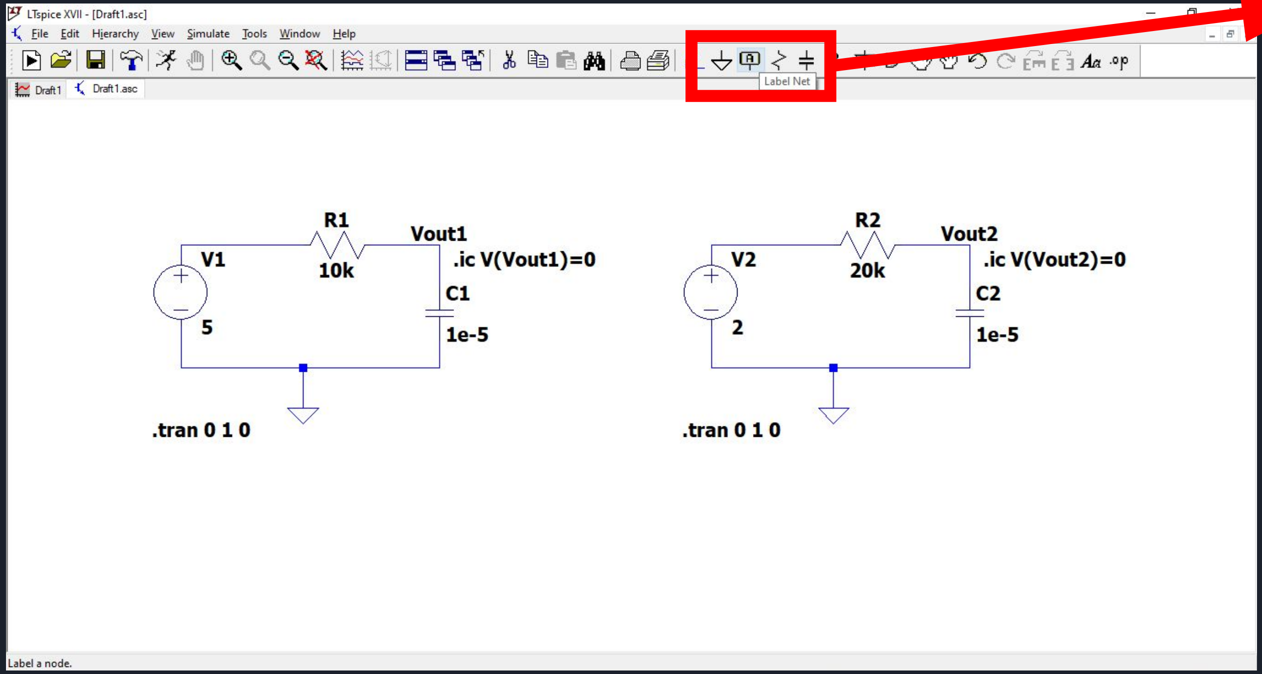
After duplicating

Right Click on each component and change their values as required



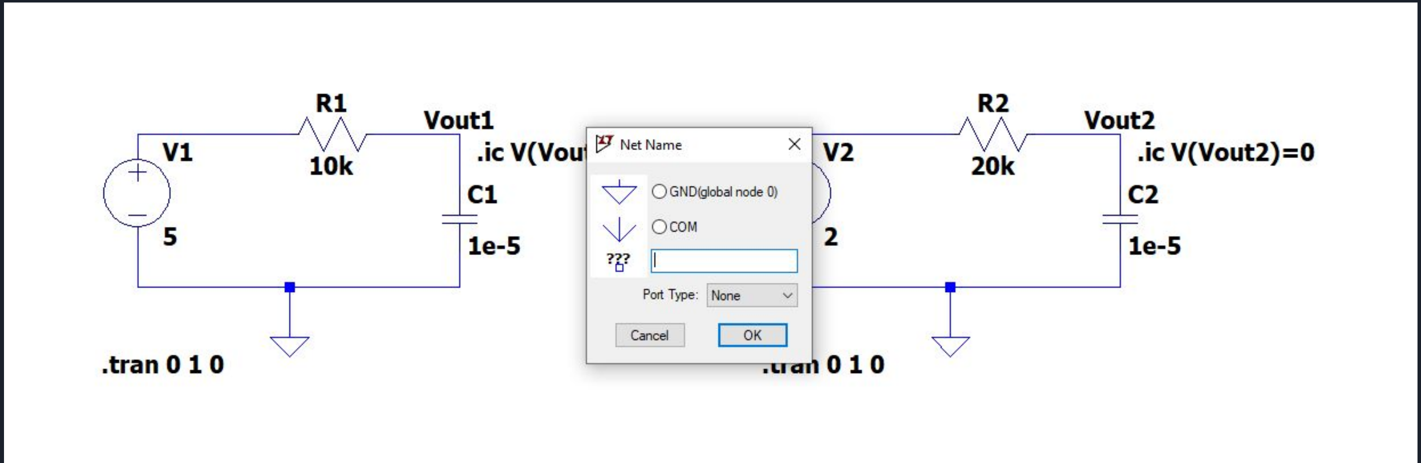
Labeling Nodes

To plot voltage of a certain node: you can label it if required



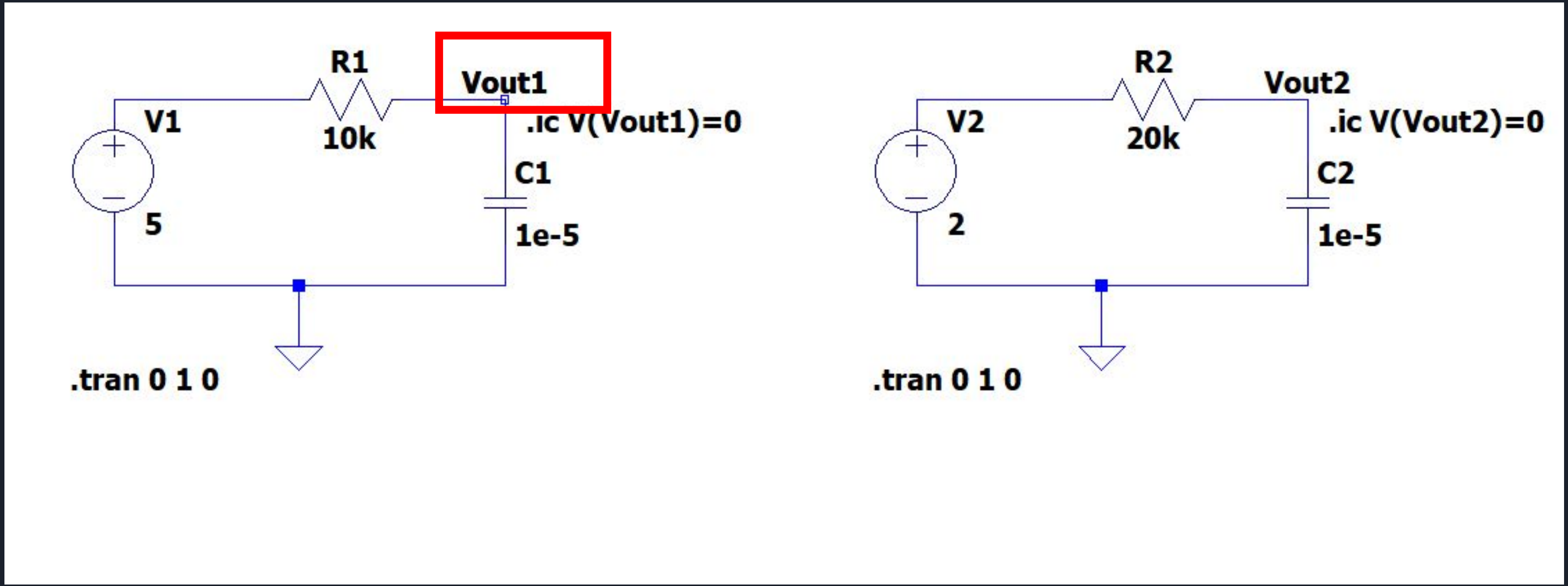
To differentiate two nodes of two circuits

One node can be labeled as Vout1 and the other Vout2. Type the desired name in the “Net Name” window.



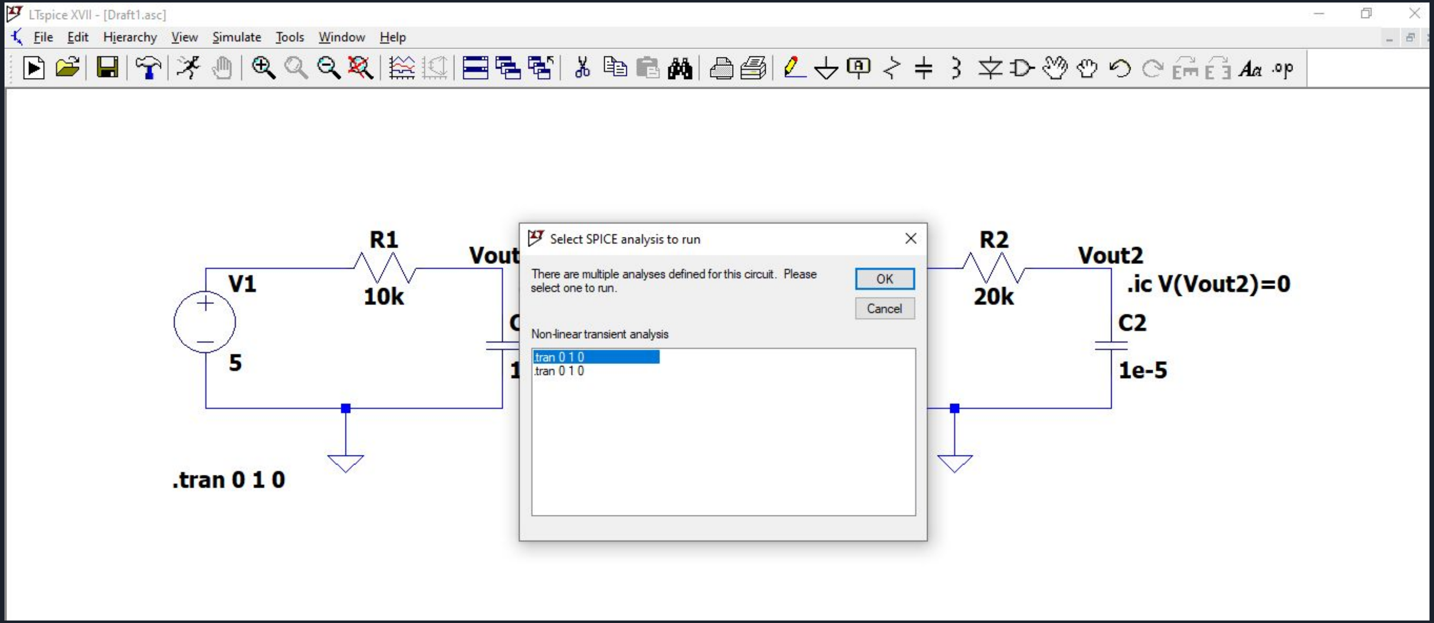
To differentiate two nodes of two circuits

Then click on the desired node that you want to label (shown below)



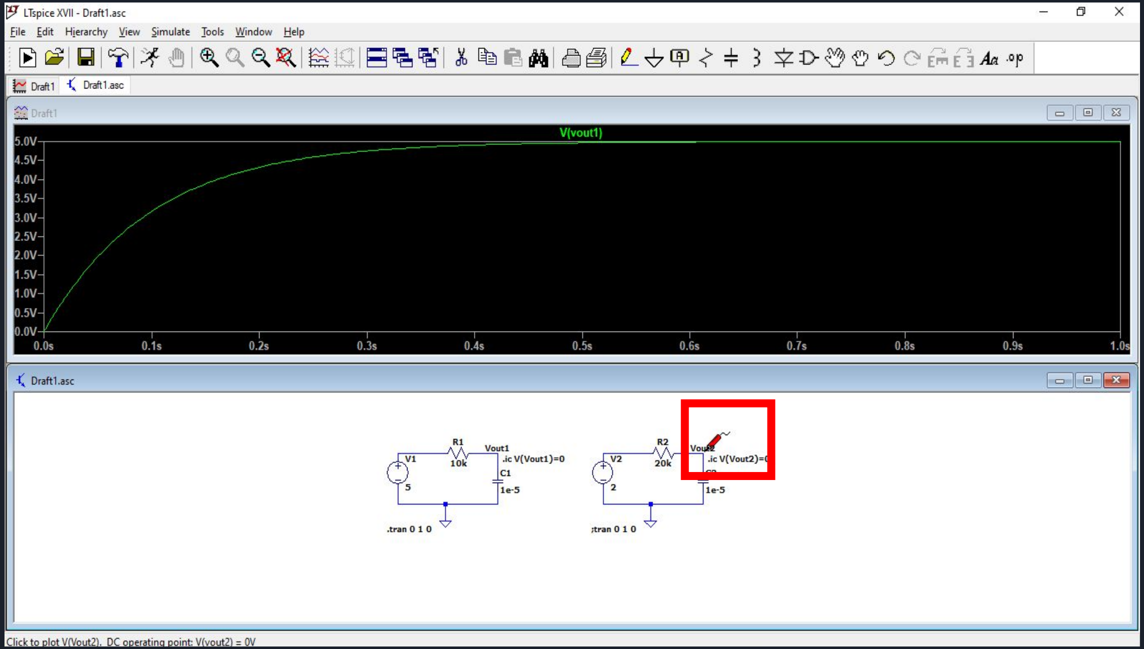
Running Simulation

Then click on any of the transient analysis (both are same since we are running both circuits in the same time frame)



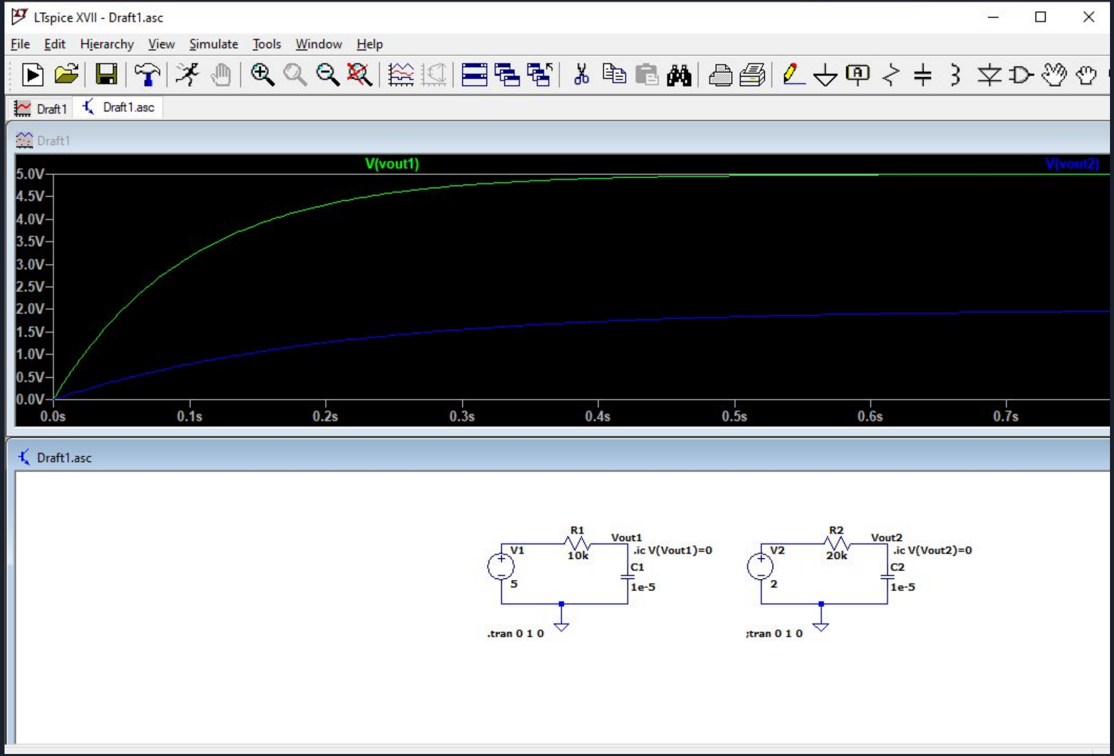
Probe the nodes and plot Them Together

Then click on any of the transient analysis (both are same since we are running both circuits in the same time frame)



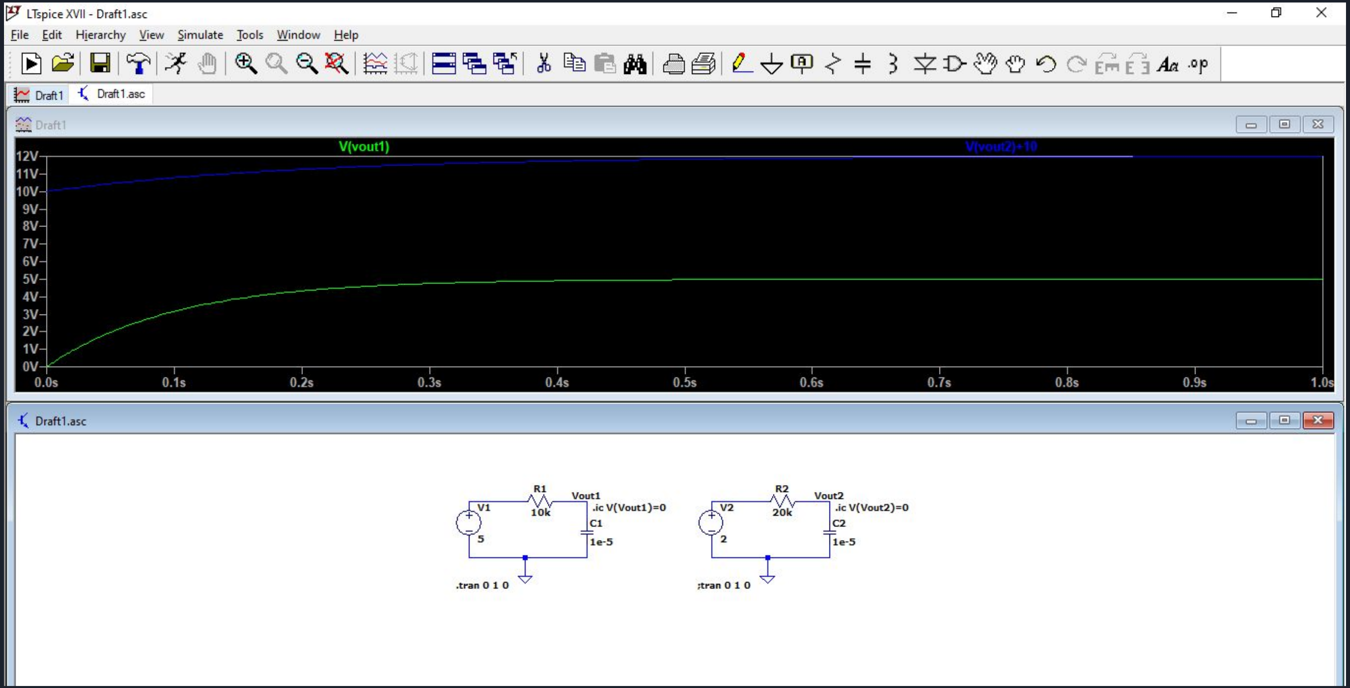
Probe the nodes and plot Them Together

Probing Vout1 and Vout2 will plot them in the same graph as shown below:



Adding Offset

Then by right clicking on the voltage label(from the graph) you can add offset as per the requirements



Advanced plotting

Congratulations! you have completed the advanced plotting section of the tutorial

Specifying IC's

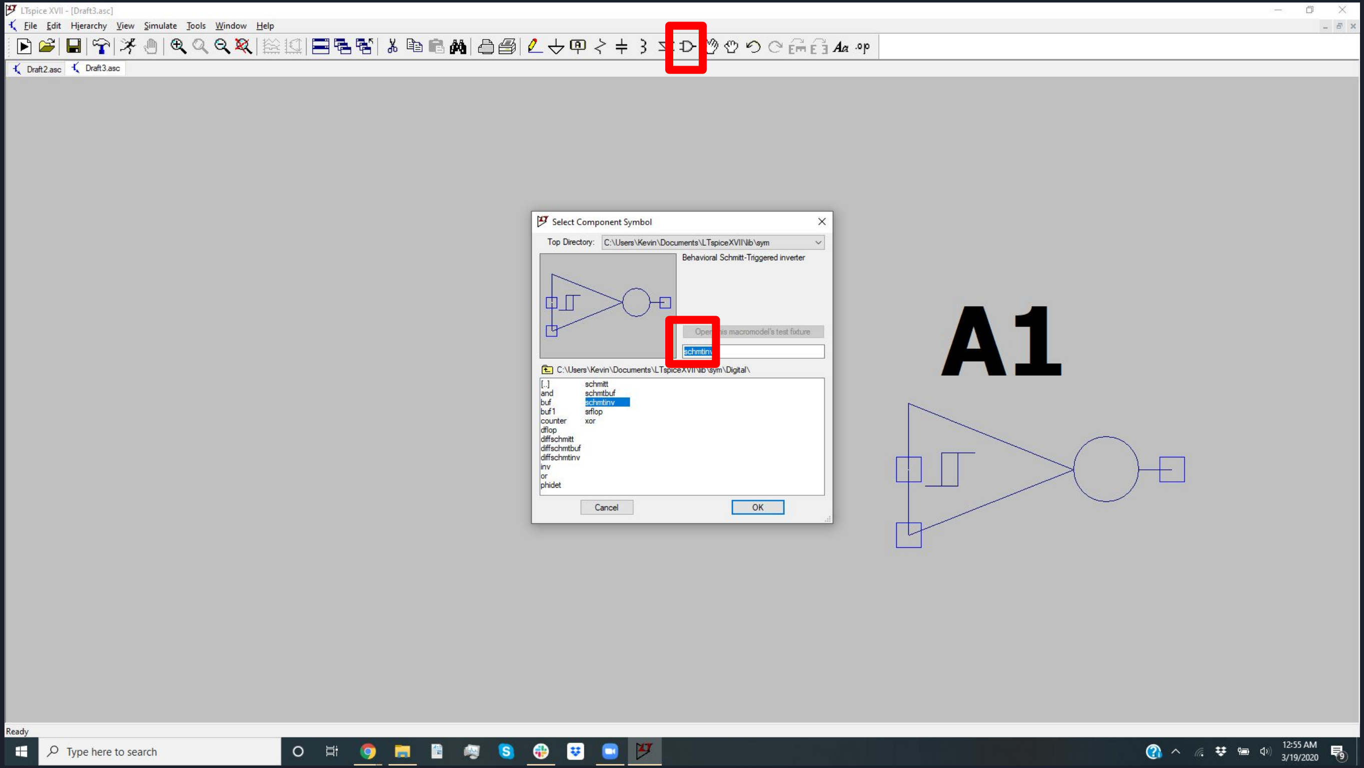
- Often we will want to use a component that is not in a default LTspice library
  - We can make these components ourselves
  - OR we can tweak the specifications of an existing component (much much easier)

Specifying IC’s

- We want to model a schmitt inverter, but “CD40106B” (official IC number in our kit) is not in LTspice
- Solution: update default schmitt trigger model

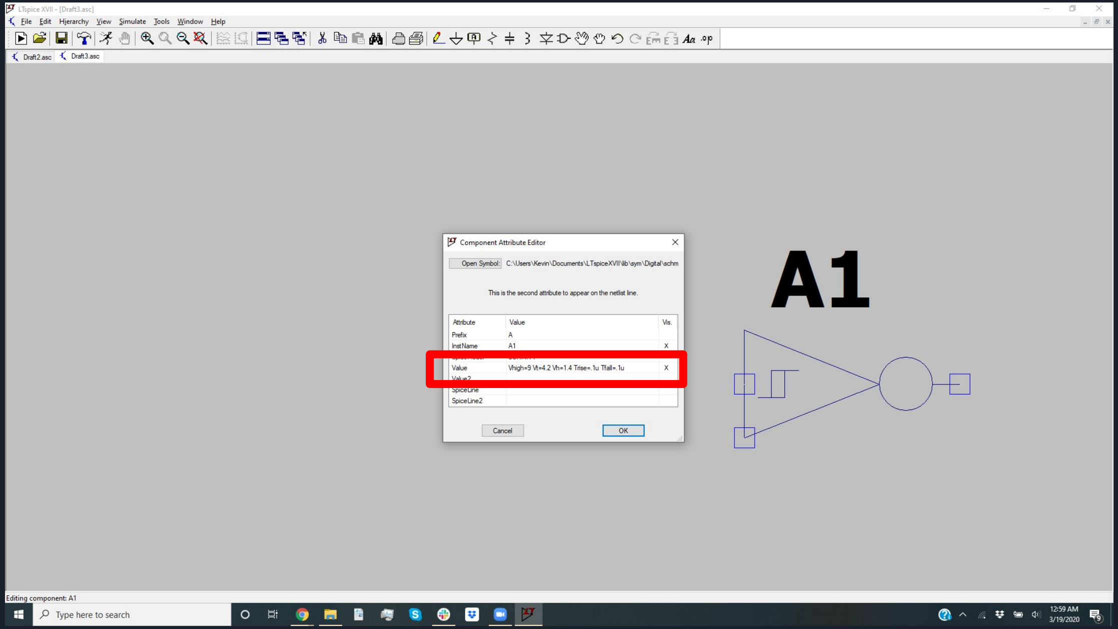
Specifying IC's

- Begin by opening a new model and inserting a component called "schmitinv" into it



Specifying IC’s

- Right click on the model and update the “Value” column with the info you need
  - Vhigh=9 Vt=4.45 Vh=.85 Trise=.1u Tfall=.1u
  - NOTE: this assumes you are giving the IC a 9V power supply- these values will change at different supply voltages or for other IC’s. Please check datasheets to verify!



Specifying IC’s

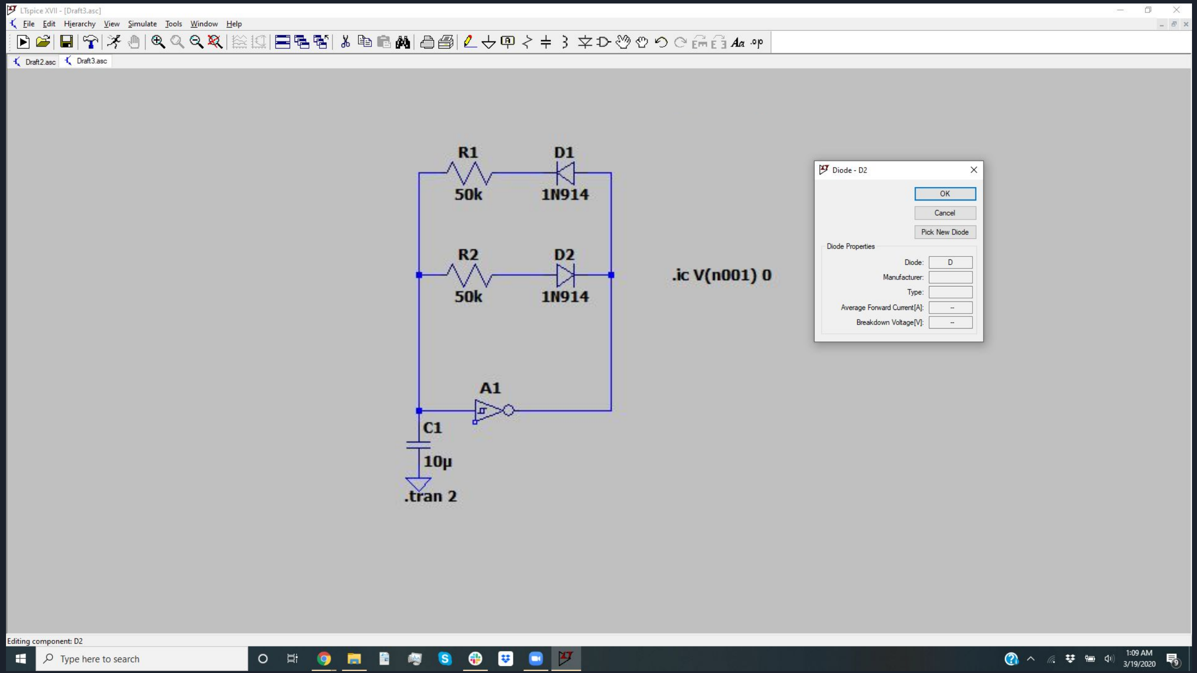
Congratulations! You have now customized a general IC to use for your own circuits

Building an Oscillator Circuit

- Now we will build an Oscillator circuit
- Note the initial condition must be set as well

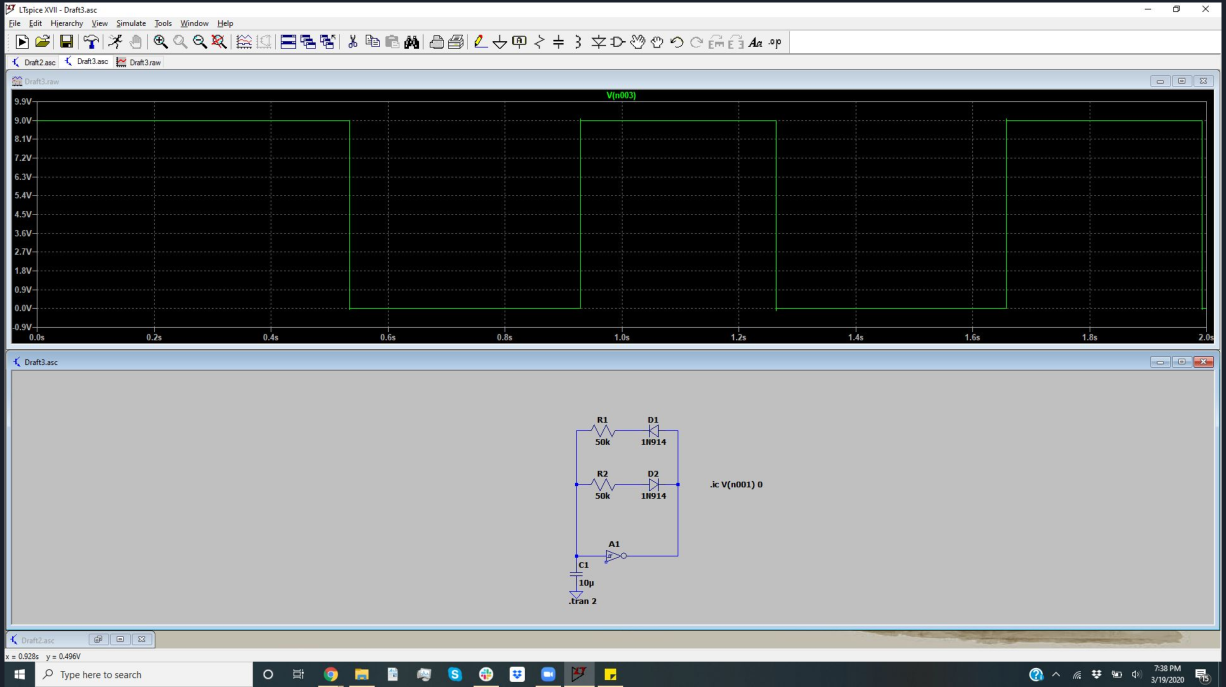
Building an Oscillator Circuit

- You will need to use the previous Schmitt inverter settings
- You will need to select a diode from the premade options
    - For best results select diode 1N914



Building an Oscillator Circuit

- Build the circuit and probe the inverted output of the schmitt inverter
- Run an analysis and get a plot similar to this



Building an Oscillator Circuit

- Congratulations! You have built and analyzed an oscillator in LTspice

Assignment:

Build 2 different oscillators

1. 9V pp, 1Khz, 40% duty cycle
2. 9Vpp , 10 Khz, 20% duty cycle

Hint(try changing the values of the capacitors and resistors and see what happens)

Submit the plots and circuit diagrams to showing your solution