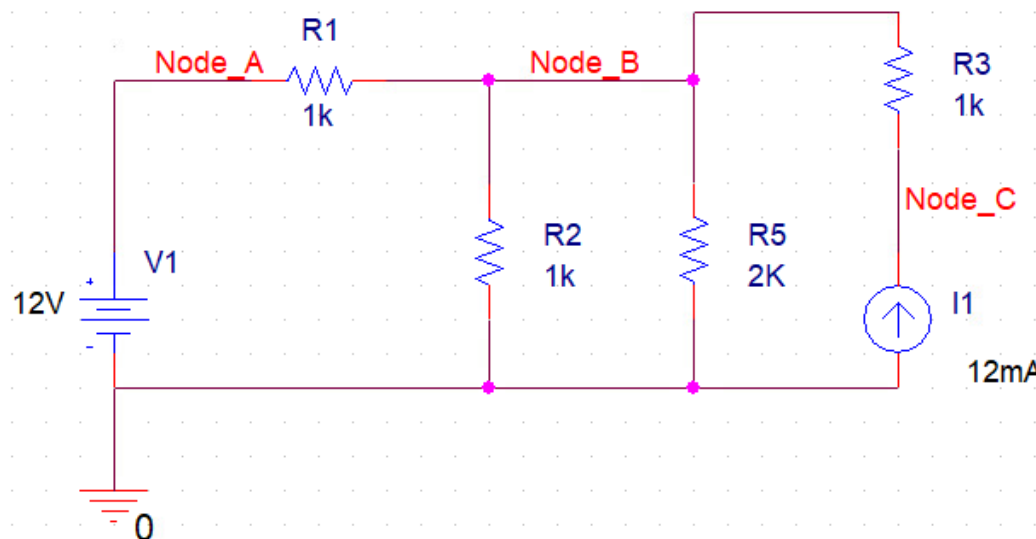


The objective of this homework is to test your understanding of the content of Module 5 and Lab-3 lecture based on PSpice. Due date of the homework is: **Friday, October 23rd, 2020**. You have to upload a scanned version of your solutions on Blackboard. If you don't have a scanner around you, you can use Cam Scanner—a mobile app that scans images in a neat way, as if they're scanned through a copier. Also you can upload the PDF version converted from a word file of your report.

For questions related to the homework, please email the two teaching assistants: (a) Wasim (email: wasim.dipon@my.utsa.edu) and (b) Yuvaraj (email: yuvaraj.munian@my.utsa.edu).

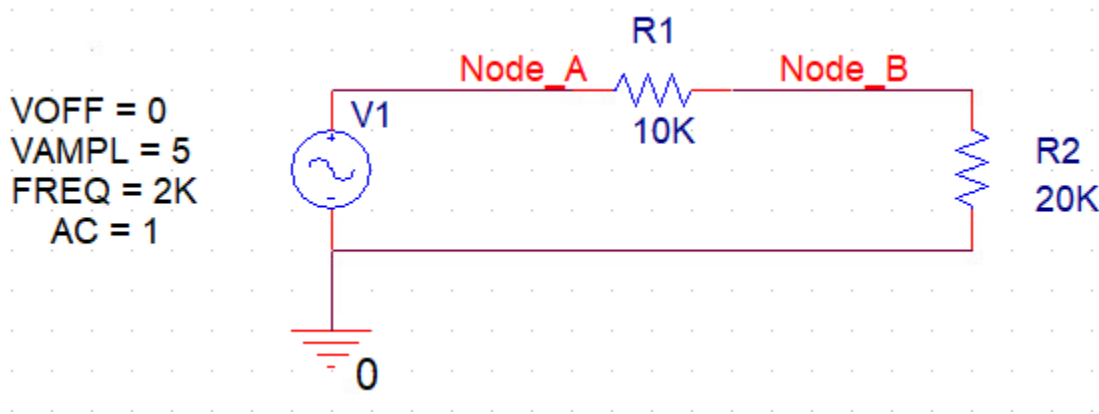
You can send your question to both of them on the same email. Feel free to CC me as well, but that is not necessary.

1. Enter the circuit below. The voltage source is part VDC, the current source is part IDC, the resistors are part R, and the zero-ground is part 0. Chose to set up a new simulation profile. Select Bias point simulation type. Go back to the schematic when the simulation ends and use the Enable Bias Voltage Display, Enable Bias Current Display, and the Enable Bias Power Display buttons to answer the following questions. Snip the schematic with all the values on and put in the report. [60 points]



- a. What is value of the node voltage at Node_B?
- b. Calculate the node voltage at Node_C using the value of the voltage at Node_B. Show calculation. Verify your calculation by recording the node voltage at Node_C.
- c. What is the power dissipation through resistor R5?
- d. What is the power of the current source I1 (Include the sign)? Why is the power negative?

- e. Make the value of I1 to be zero (0). Edit the simulation profile and run Transient type simulation. Use differential voltage marker to measure the voltage across R3. What is the value of the voltage across R3? You are supposed to get 0V across R3. Explain why.
2. Enter the circuit below. The voltage source is part VSIN. Name the input node Vin and the output node Vout. To do so, chose the Place net alias icon and name the illustrated nodes Vin and Vout. [40 points]



Set up a Transient simulation type. Set the Run to time = 3ms. Set up the Maximum step size = 3us. Apply the settings and then run the simulation. In the resulting simulation results window, use Add Trace menu item to display the input Vin and the output Vout. Snip the resulting plot and include it in the report. Also, answer the following questions from the plot.

- a. Move your assigned cursor on Vin signal to a time *approximately 2.25ms*. Read off the voltage value of Vin at this time and mention it in the report.
- b. Move your assigned cursor on Vout signal to a time *approximately 2.75ms*, read off the voltage value of Vout at this time and mention it in the report.