Cadence Capture and PSpice Tutorial

This tutorial is intended to give you needed elements for using Cadence Capture and PSpice to design and simulate the digital logic circuit in Homework 3, Problem 1. The tutorial is intended to be followed on a **computer in any ITaP laboratory**. While this tutorial is intended to be clear and unambiguous, it is recommended that you see the TA during his office hours (in **ENAD 420**) if you have any issues. For your convenience, they are listed here:

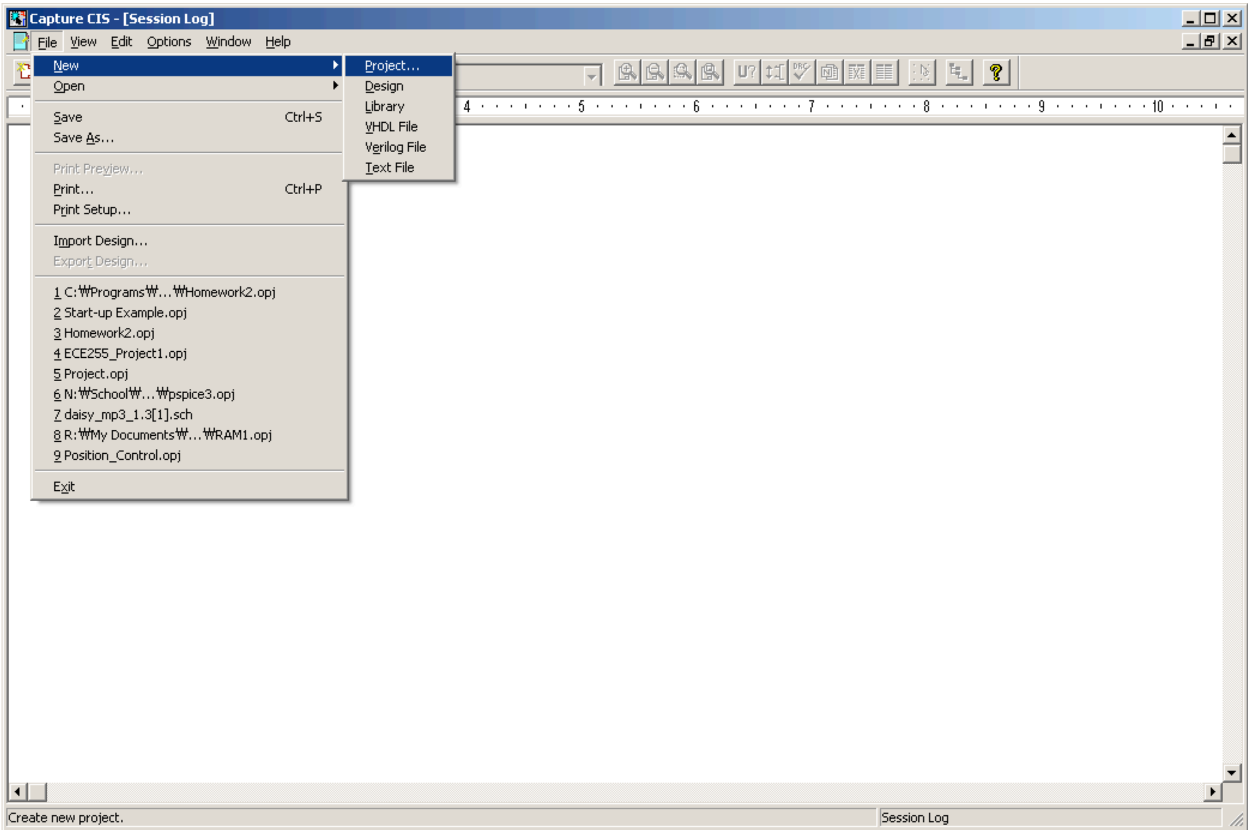
Monday: 4:30 – 6:00 pm
Tuesday: 3:00 – 4:30 pm
Wednesday: 4:30 – 6:00 pm
Thursday: 3:00 – 4:30 pm
Friday: 4:30 – 6:00 pm

In this tutorial, we will design and simulate the circuit representing the logic formula **Y = A1A0 + B1B0**. Seat yourself behind a computer in the ITaP lab and here we go!

TUTORIAL:

1. After logging into an ITaP computer, you must install the Cadence tools. To do this, access the Cadence install by going to:
Start Menu->All Programs->Course Software->engineering->ece ->OrCAD PSD 15.1
2. Once the installation is complete, go back to the Start Menu and go to: Start Menu->All Programs->Cadence PSD 15.1->Capture
3. A box comes up with a drop-down menu with the option "PCB Design Studio with Capture CIS". Select "OK".

4. Once Capture is up, start a new project going to File->New->Project...

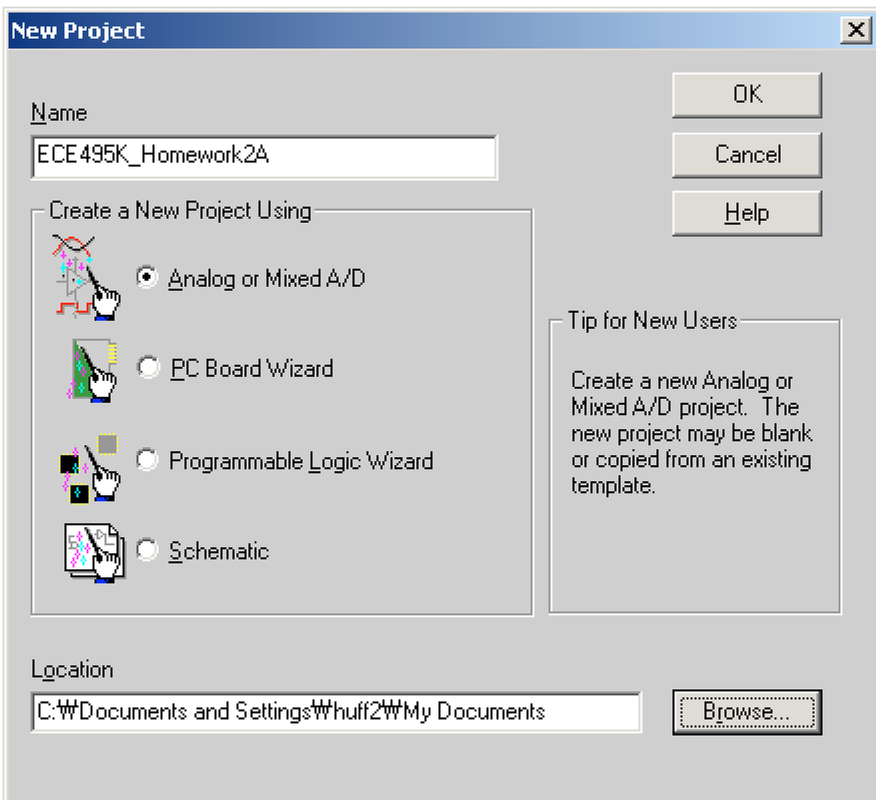


5. The project dialogue box should appear as below. **Select "Analog or Mixed A/D" for your project type.**

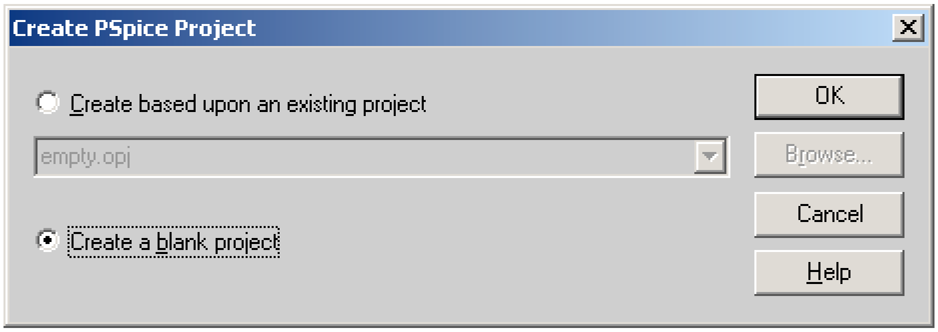
   <u>TITLE YOUR HOMEWORK:</u> ECE495K_Homework2A.

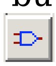
   <u>FOR THIS TUTORIAL, YOU CAN USE ANY NAME.</u>

   Change the LOCATION of the saved project to your "My Documents" folder.

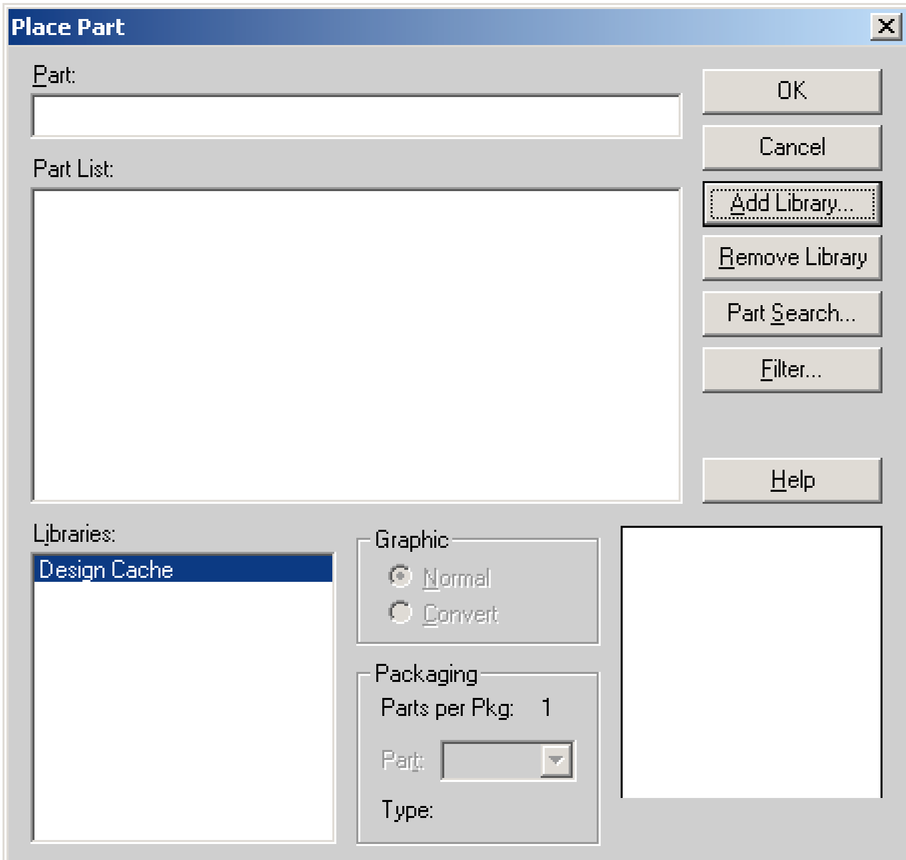


6. After selecting “OK” to the above dialog box the box below comes up. Make sure you select “Create a blank project” before clicking OK.



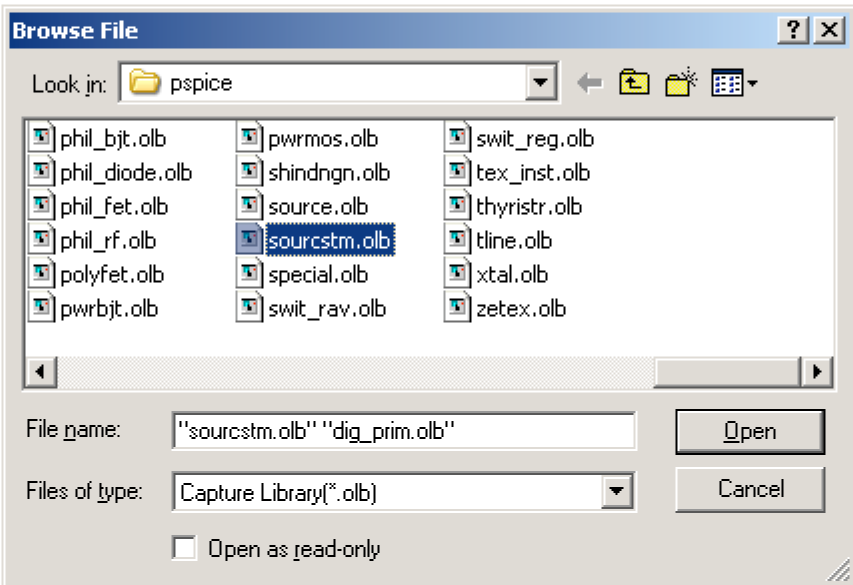
7. You should be seeing a blank grid. On the right hand side of the grid, there are a lot of buttons. To place a certain part, select the button shaped like this: 

8. When you select the "place tool" button as above, you should get the following dialogue box as shown below. Select the "Add Library…" button.

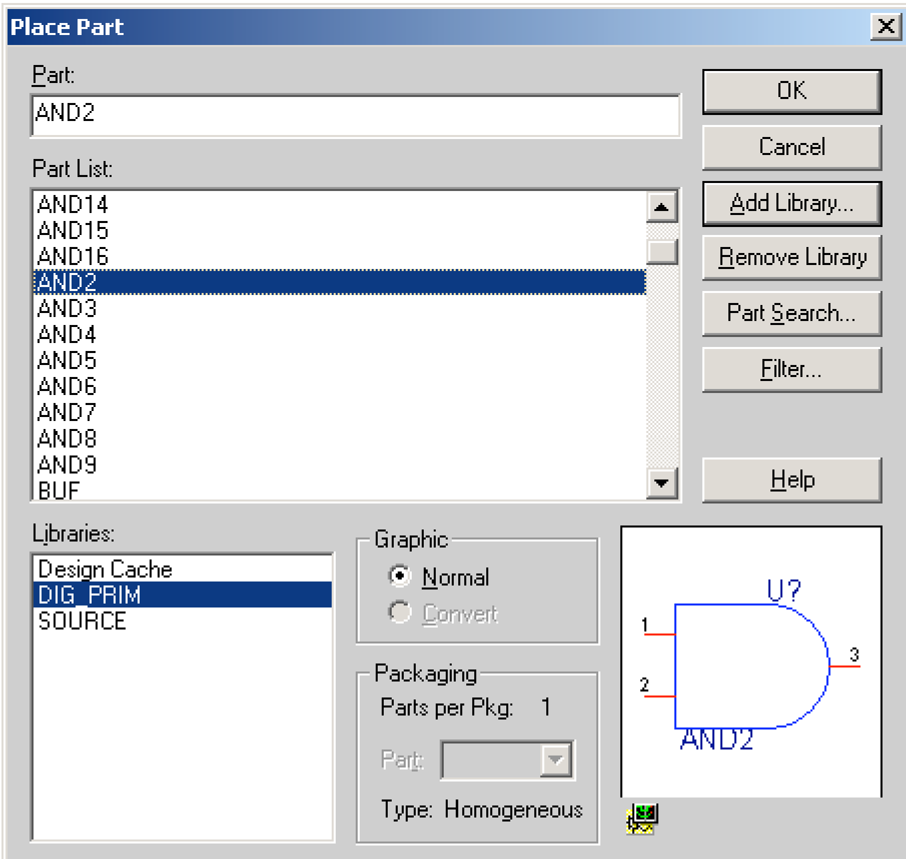


9. After selecting the “Add Library…” button, you should get a selection list that looks like the one below. Add the following libraries:

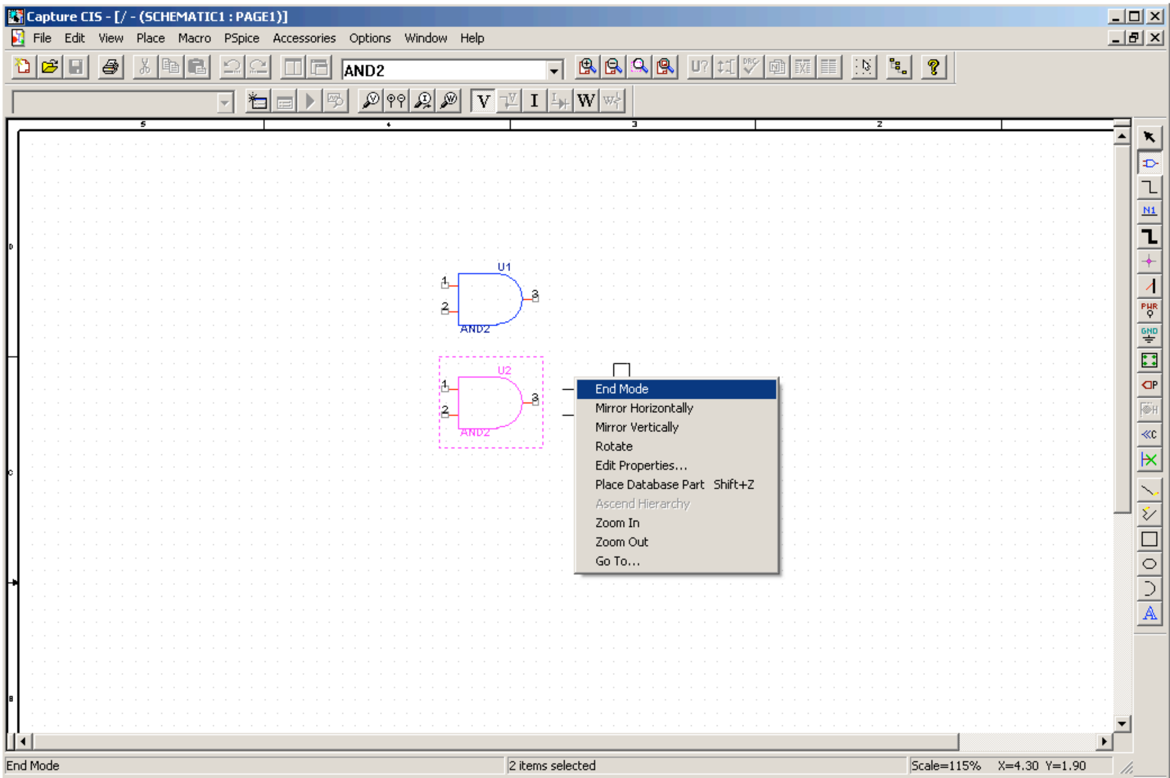
    sourcstm.olb
    dig_prim.olb



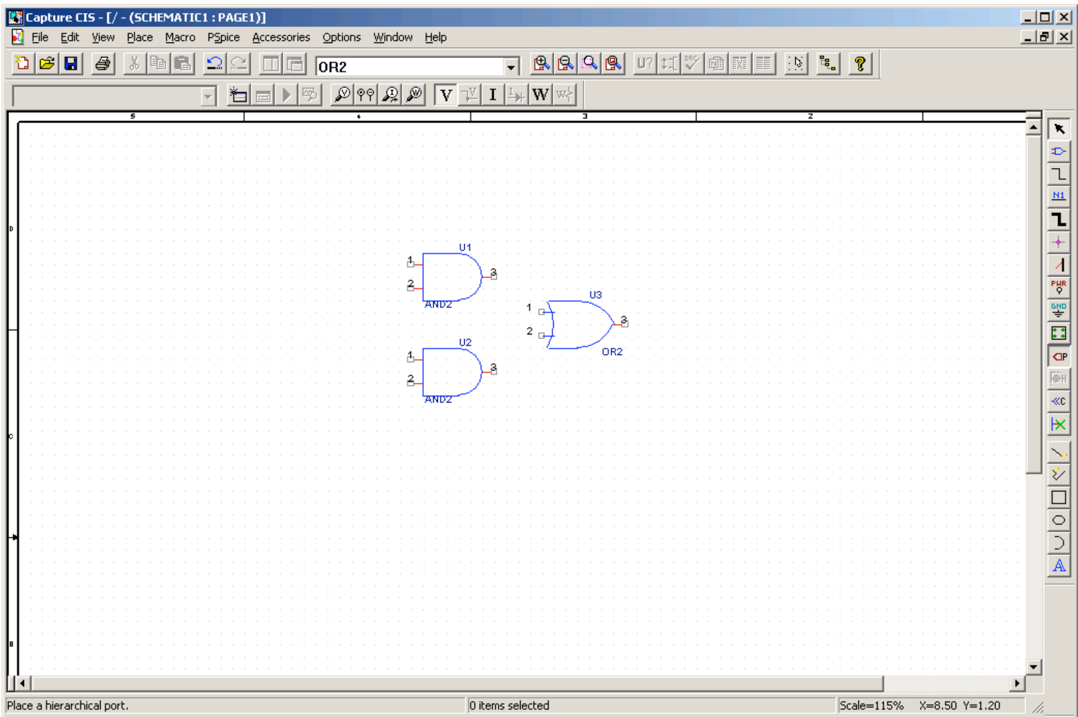
10. If you notice, you now are in the dialog box in step 8, but there are two more libraries beside "Design Cache". Select the DIG_PRIM library. This will have all of the logic gates you need. For this tutorial, select the "AND2" part as shown below (AND2 means an AND gate with 2 inputs):



11. After selecting “OK” in the above dialog box, you return to the schematic grid. When you left click your mouse, you now place the AND2 part. Place two “AND2” parts as shown below. To return to a normal cursor right-click the mouse and select “End mode”.

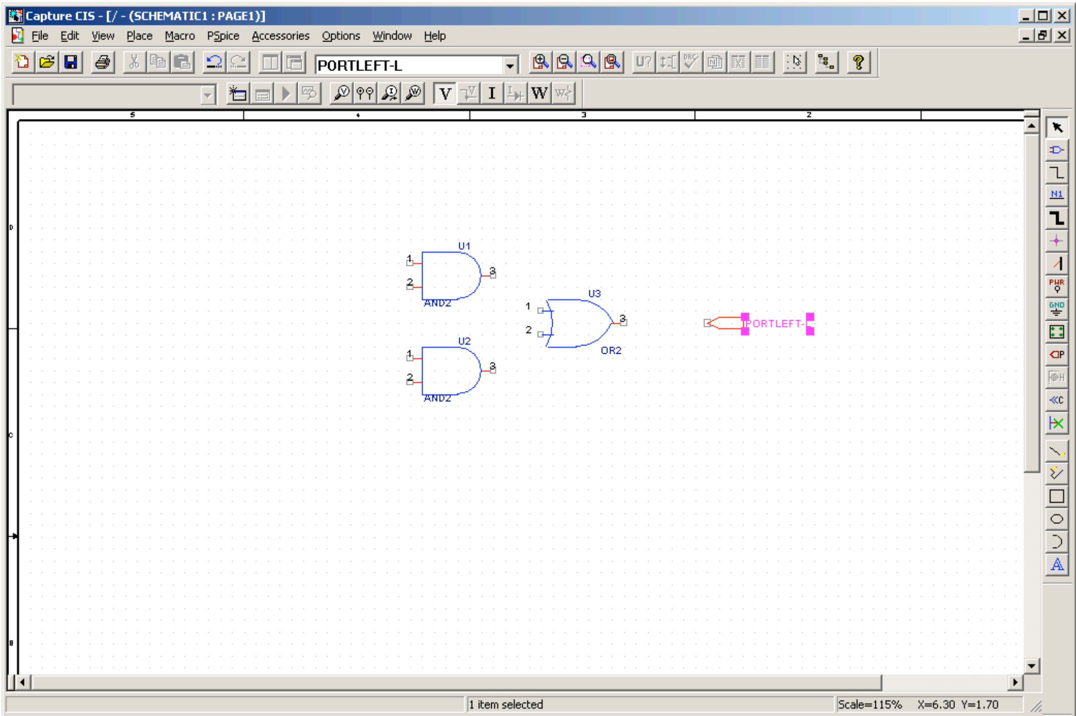


12.Just as you added the AND2 part, add an OR2 part and place as shown below.

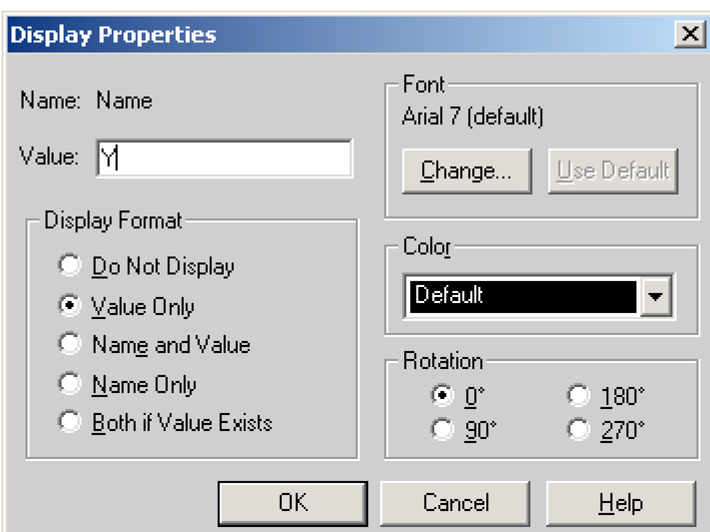


13.Now we must select and label a port for the output of the logic formula Y=A1A0 + B1B0. As indicated in our example, we will call this port Y. To add this port select the button on the right that looks like this .

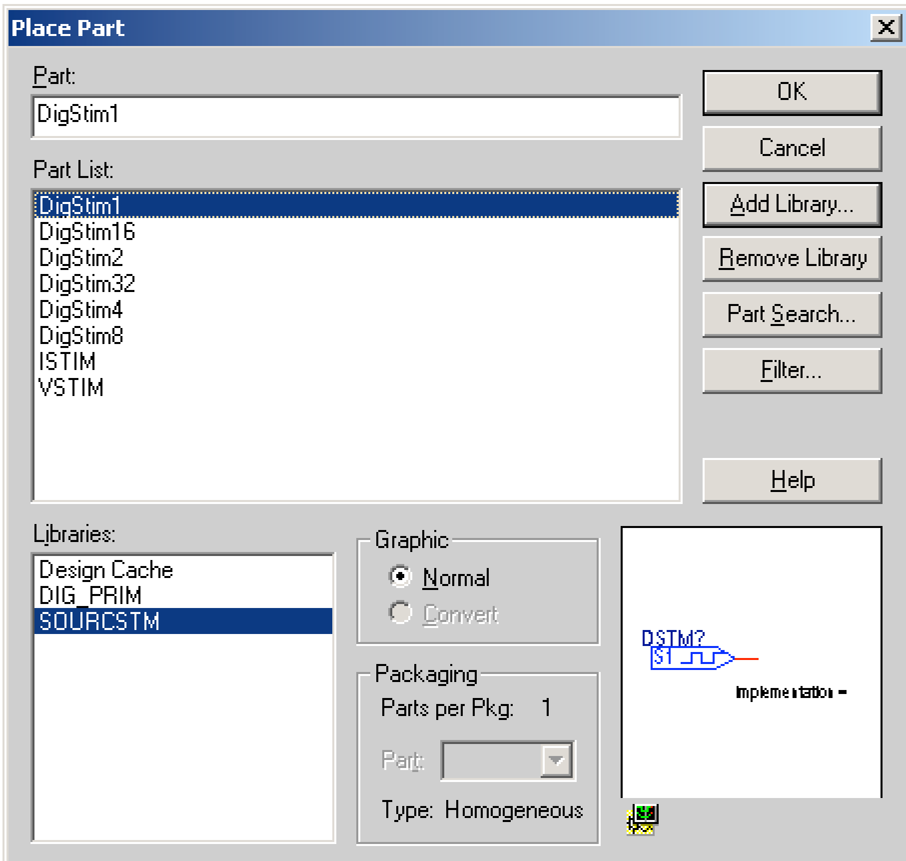
14.Select the port called “PORTLEFT-L” and place as shown below:



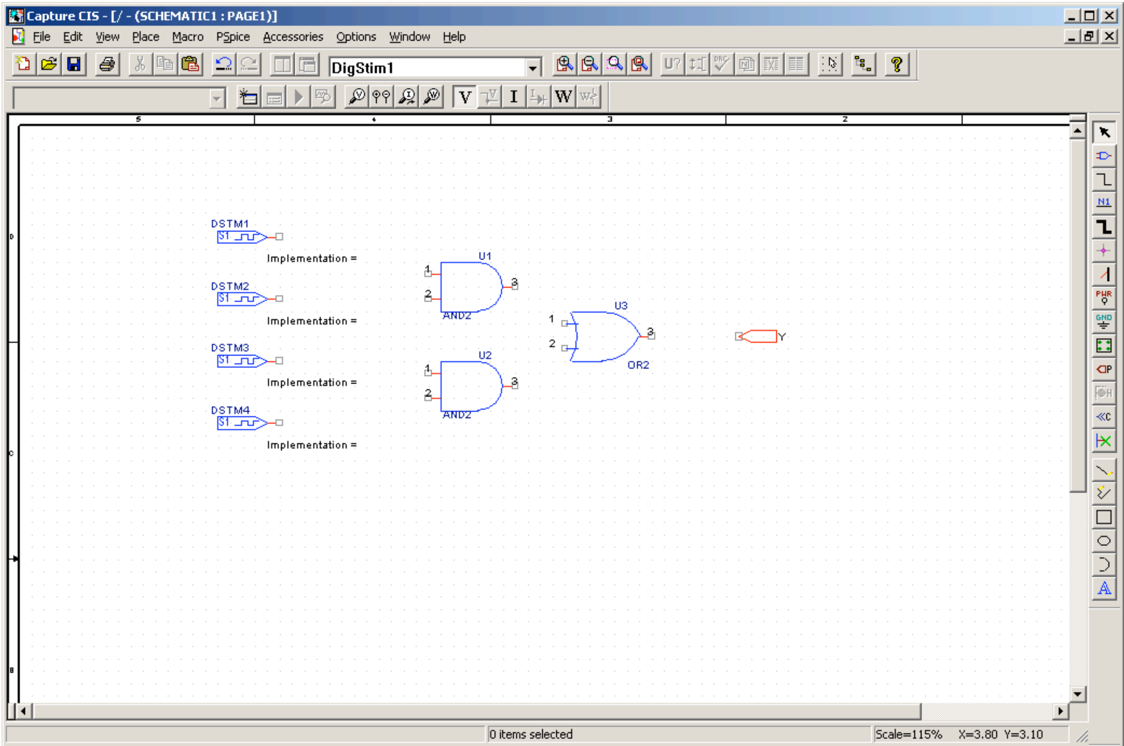
15.To rename the port, double-click the name “PORTLEFT-L” to get a dialog box as shown below. Change the name to “Y”.



16.To add inputs, select the “place part” button on the right hand side as you did when you added the AND2 and OR2 part. When you get the place part dialog box, make sure the SOURCESTM library is selected. Select the “DigStim1” part.

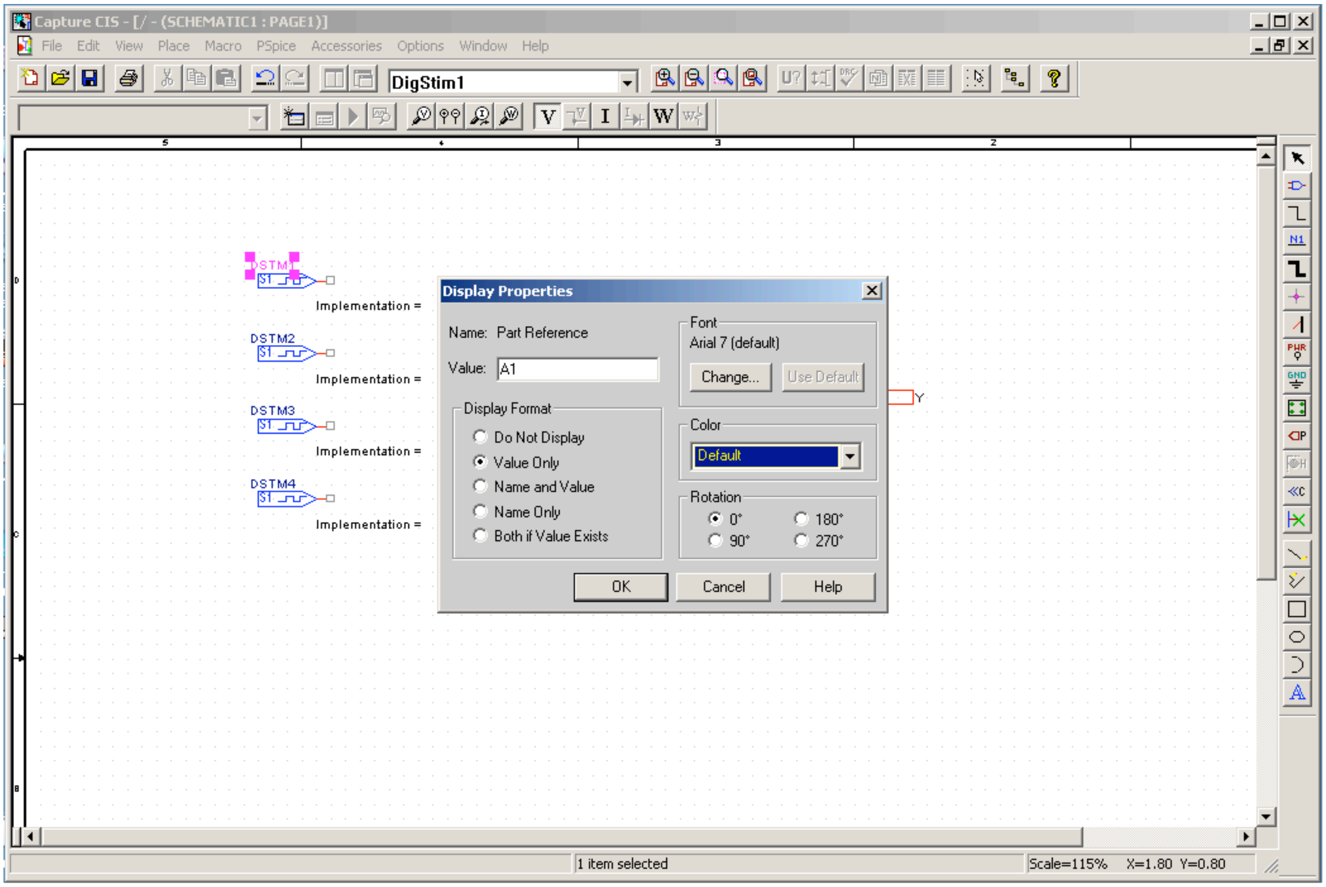


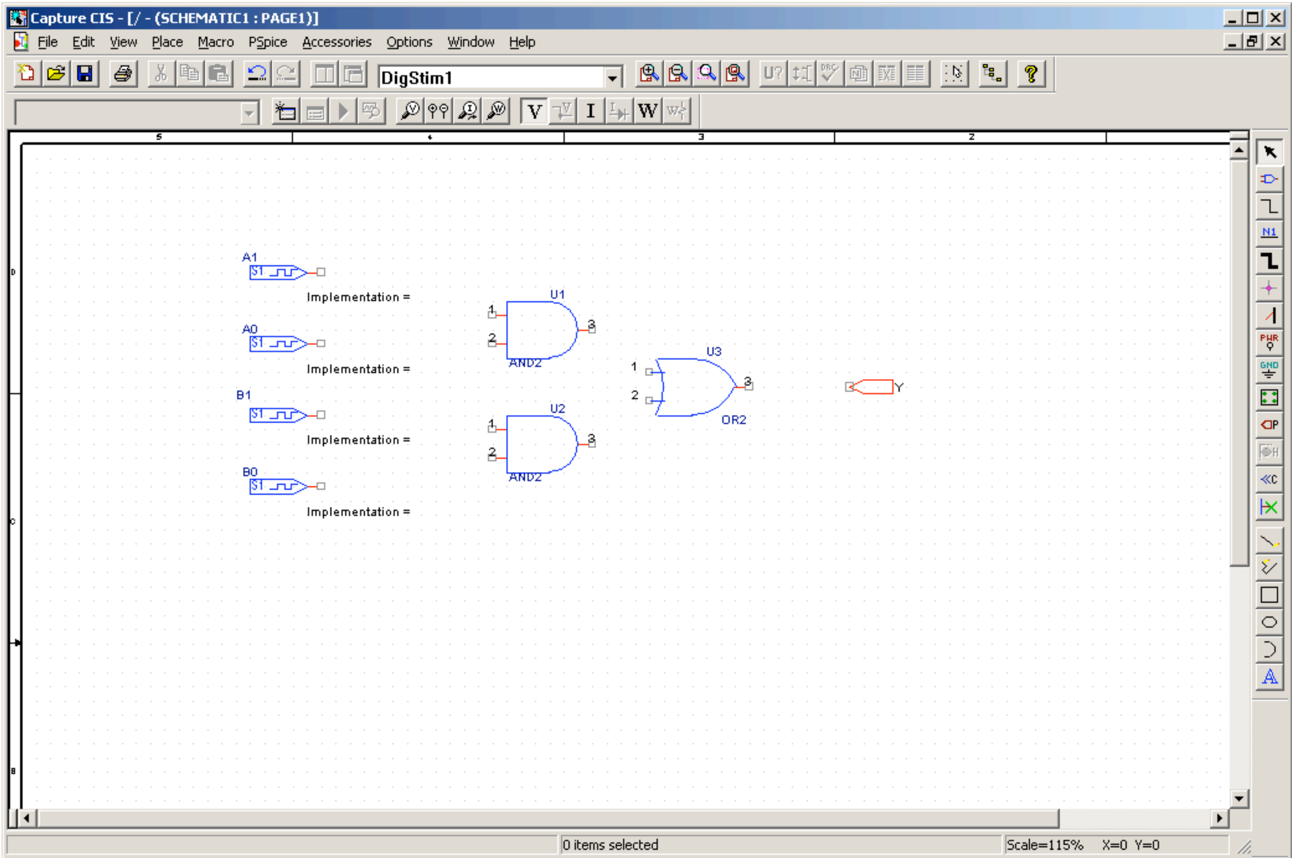
17. This part will serve as your input. Therefore, place four of them as shown below:



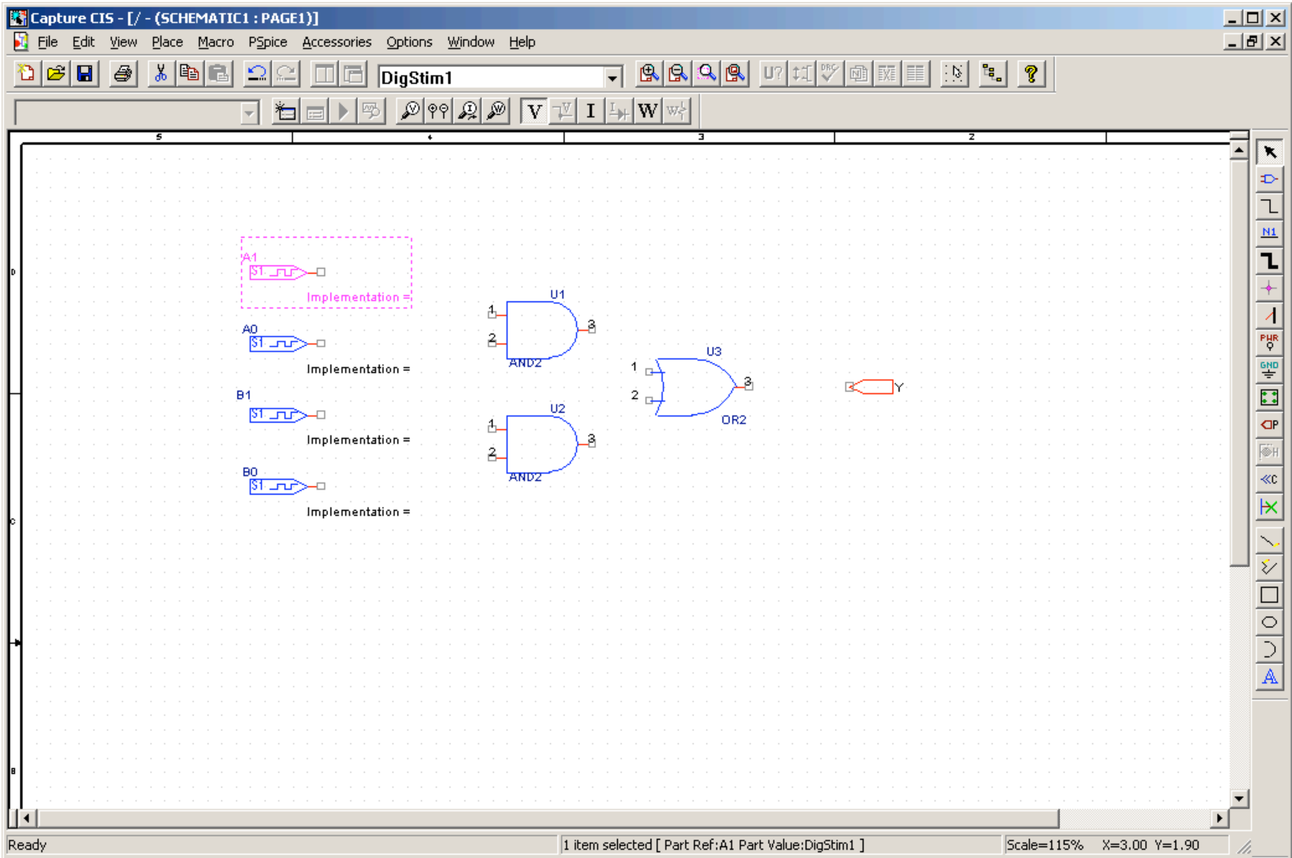
18.Rename the inputs to be A1, A0, B1, and B0 as the following two figures illustrate (Rename the sources as you did in step 15).

IN YOUR HOMEWORK, YOU MUST NAME THE INPUTS A3, A2, A1, A0 and B3, B2, B1, B0 with A3 and B3 being the most significant digit!!! IF YOU FAIL TO DO THIS YOU WILL RECEIVE NO CREDIT FOR THE PROBLEM!!!

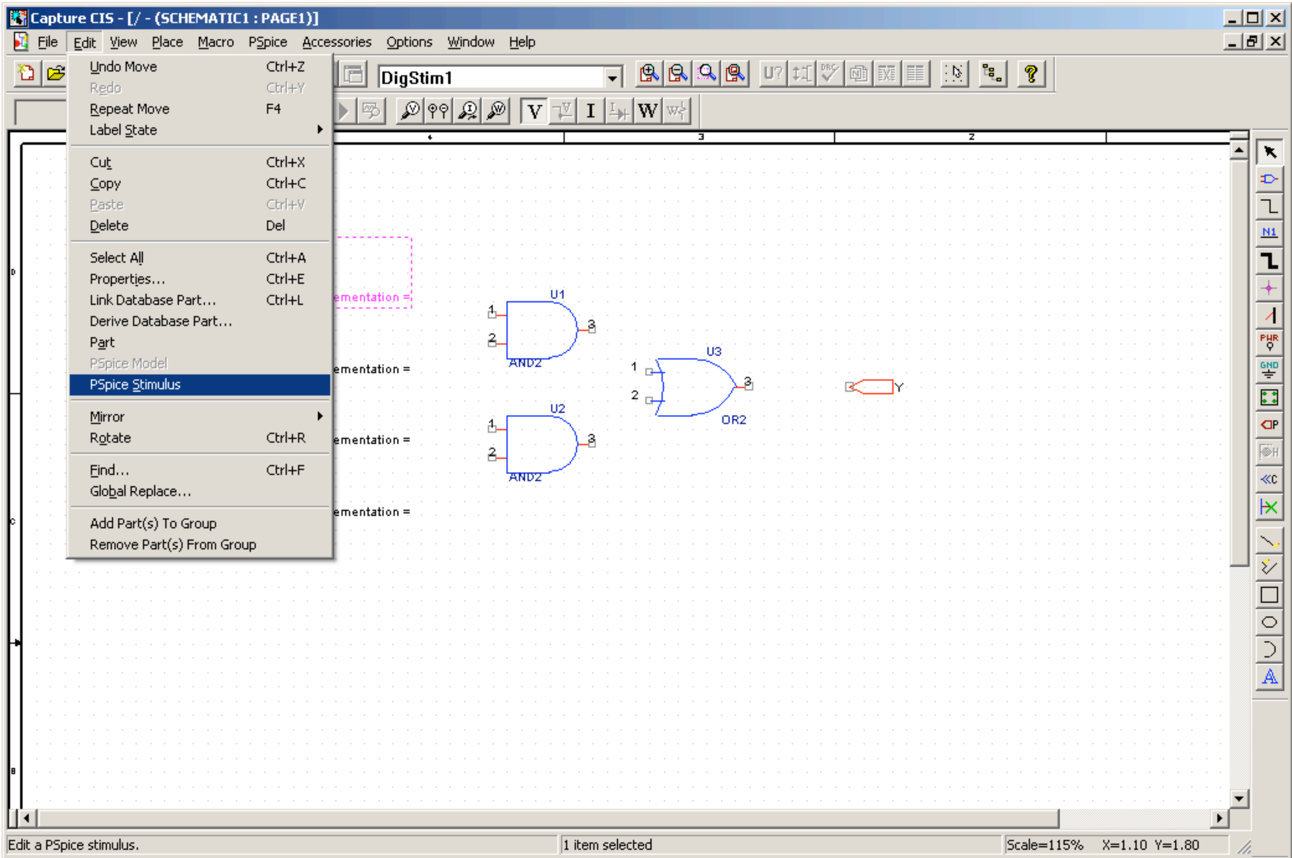




19. Now we want to make sure that our sources will cover all possible input values. Since there are four inputs, that means we have 2^4 = 16 input combinations (in your homework you will have 2^8 = 256 input combinations). We can ensure that we cover all combinations of inputs as follows. First select the part A1 as shown below.

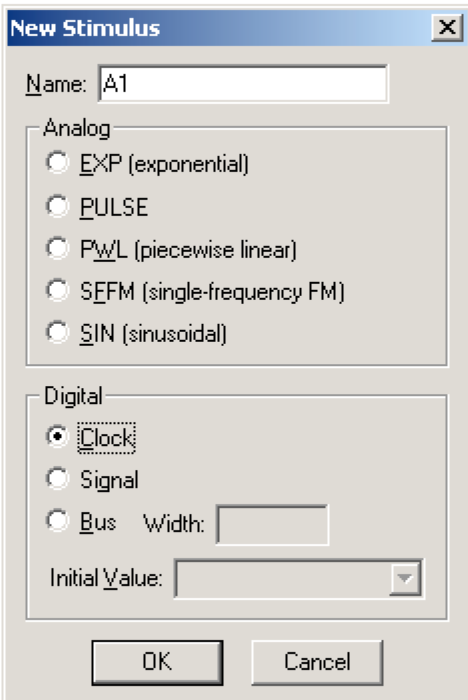


20.While the part is still selected go to Edit->PSpice Stimulus


Capture CIS - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
DigStim1
Undo Move Ctrl+Z
Redo Ctrl+Y
Repeat Move F4
Label State
Cut Ctrl+X
Copy Ctrl+C
Paste Ctrl+V
Delete Del
Select All Ctrl+A
Properties... Ctrl+E
Link Database Part... Ctrl+L
Derive Database Part...
Part
PSpice Model
PSpice Stimulus
Mirror
Rotate Ctrl+R
Find... Ctrl+F
Global Replace...
Add Part(s) To Group
Remove Part(s) From Group
U1
AND2
U2
AND2
U3
OR2
Y
Edit a PSpice stimulus.
1 item selected
Scale=115% X=1.10 Y=1.80

21. You should get a box as seen below. Make sure you name the stimulus whatever the part is named (in this case A1). **YOU MUST PAY ATTENTION TO THIS STEP FOR THE HOMEWORK OR YOU WILL RECEIVE NO CREDIT FOR THE PROBLEM!!!**

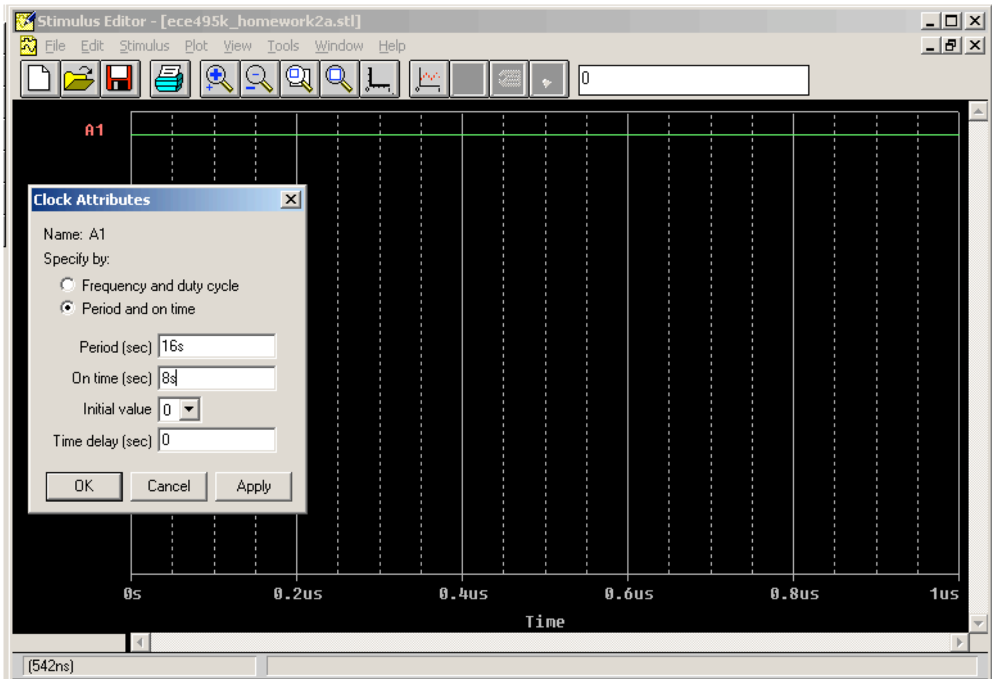
Make the input a “Clock” and click “OK”.



22. Upon clicking OK, you get a dialog box as shown below. Select “Specify by Period and on time”. For this input, you should make the period 2^4 = 16 s and the on time half of the period (8 s) as shown below. Click “OK”.

    Close out of the “Stimulus Editor”.

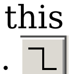
    Save the changes and click “Yes” when asked if you want to update the schematic



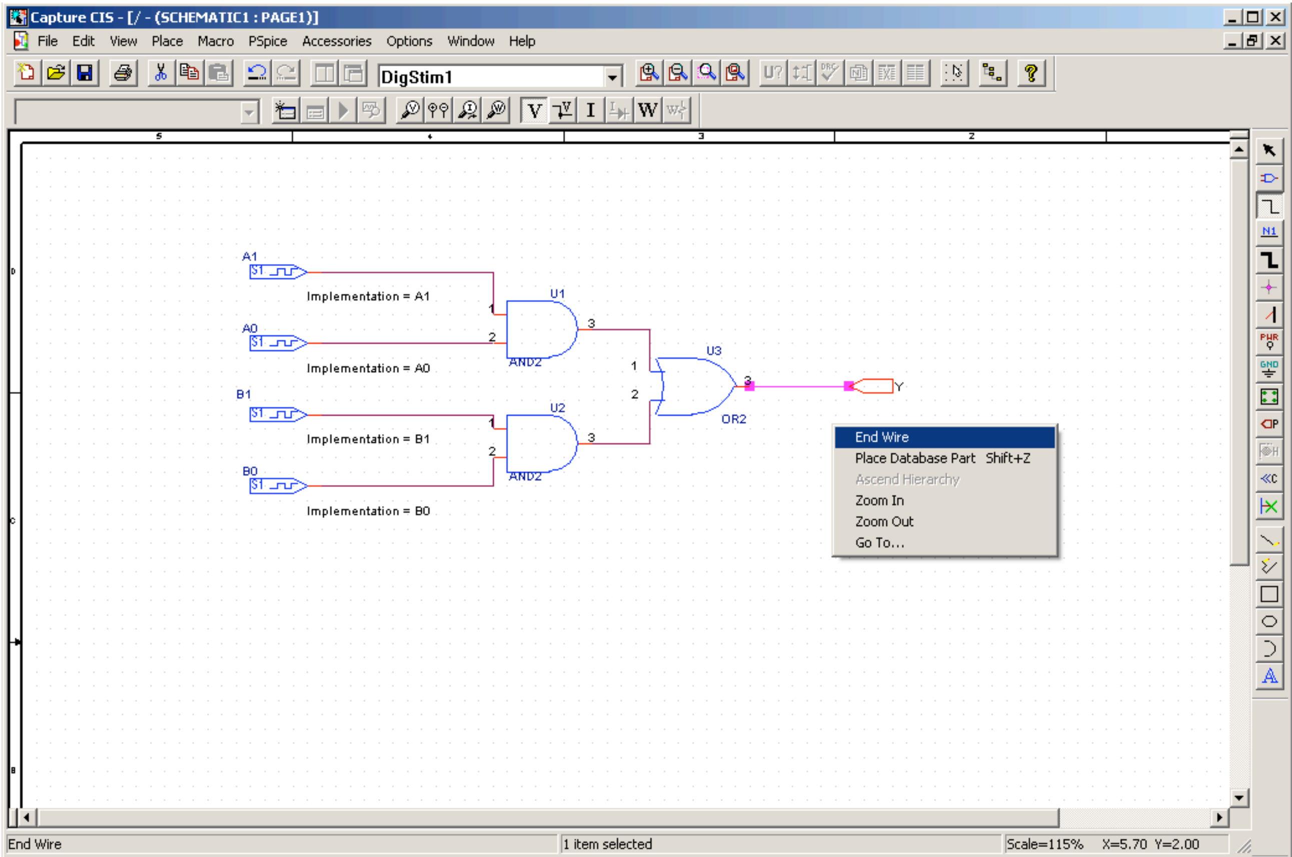
23. Configure the inputs in the order AX…A0, BX…B0 (where X is the largest number). In this tutorial, X is 1, for your homework, X is 3.

    IN YOUR HOMEWORK, YOU SHOULD CONFIGURE THE STIMULUS INPUTS IN THE ORDER A3, A2, A1, A0, B3, B2, B1, B0. THIS IS IMPORTANT FOR REASONS SEEN IN THE FOLLOWING STEPS.

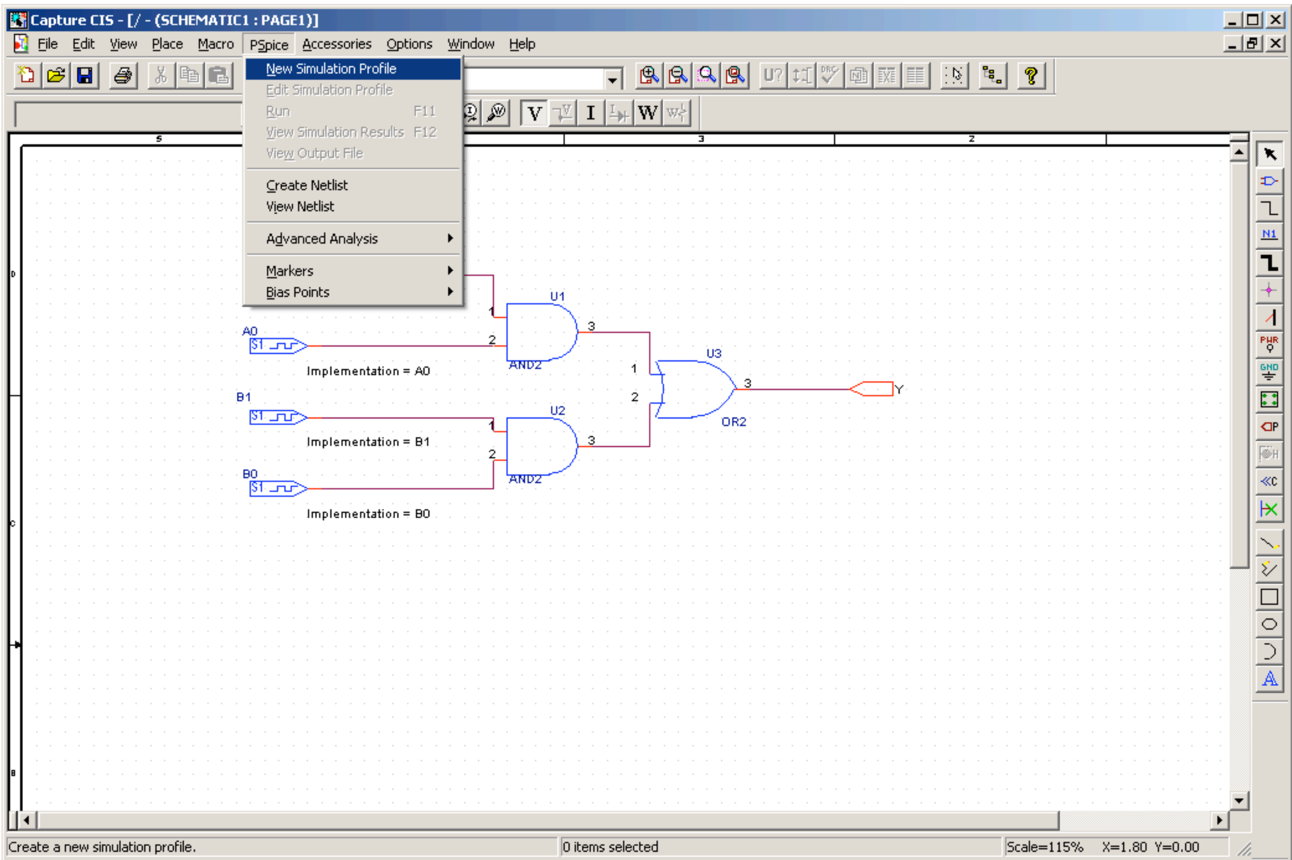
24. For the subsequent inputs configure exactly as in step 22, but make the period half of the previous one, so in this tutorial A0 should have a period of 8s and on time of 4s, B1 – a period of 4s and on time of 2s, and B0 – a period of 2s and on time of 1s.

25. Now we need to wire the circuit together. To do this select the “wire” button on the right hand side that looks like this: 

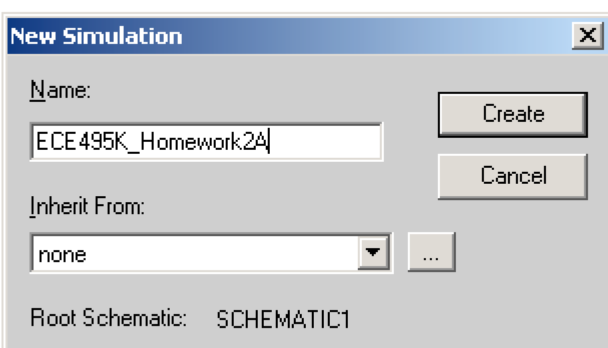
26. When you left click, your mouse lays down wire. Notice that when you connect two parts, both ends of the wire are a bright red dot. Wire the circuit as shown below. Right Click and select "End Wire" to return your mouse to a normal cursor.



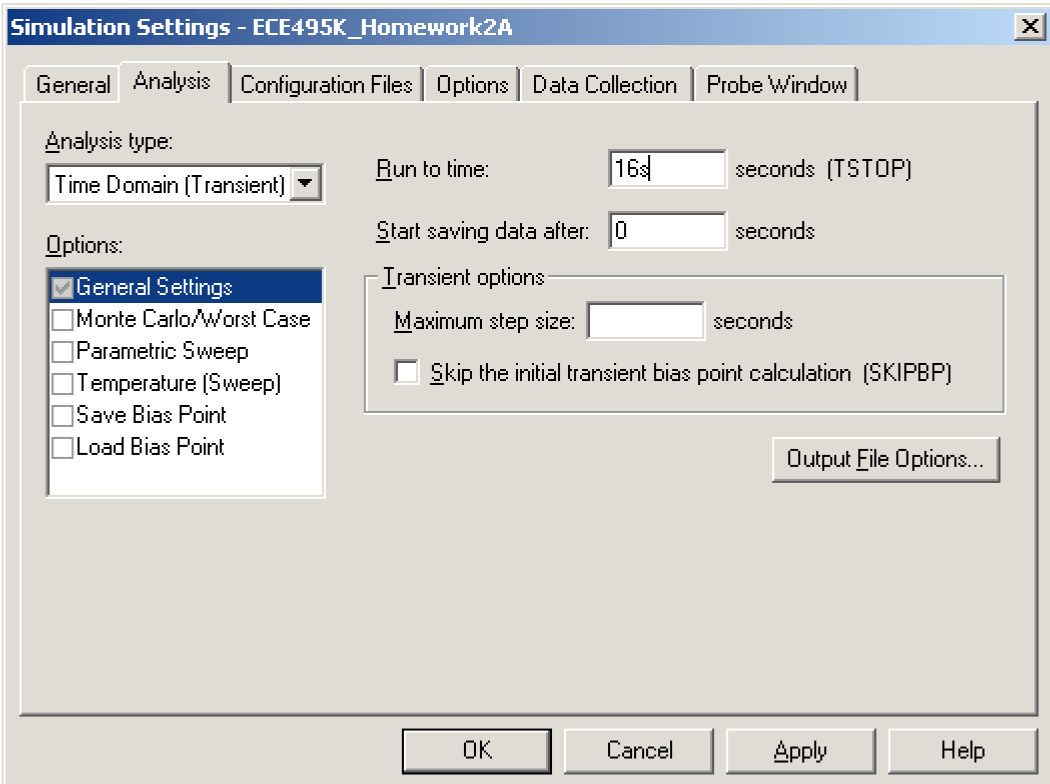
27.You are now ready to simulate the circuit. To do this select PSpice -> New Simulation Profile



28.You will get a dialog box as seen below. Give your simulation profile a name, inherit from "None" and click "Create"

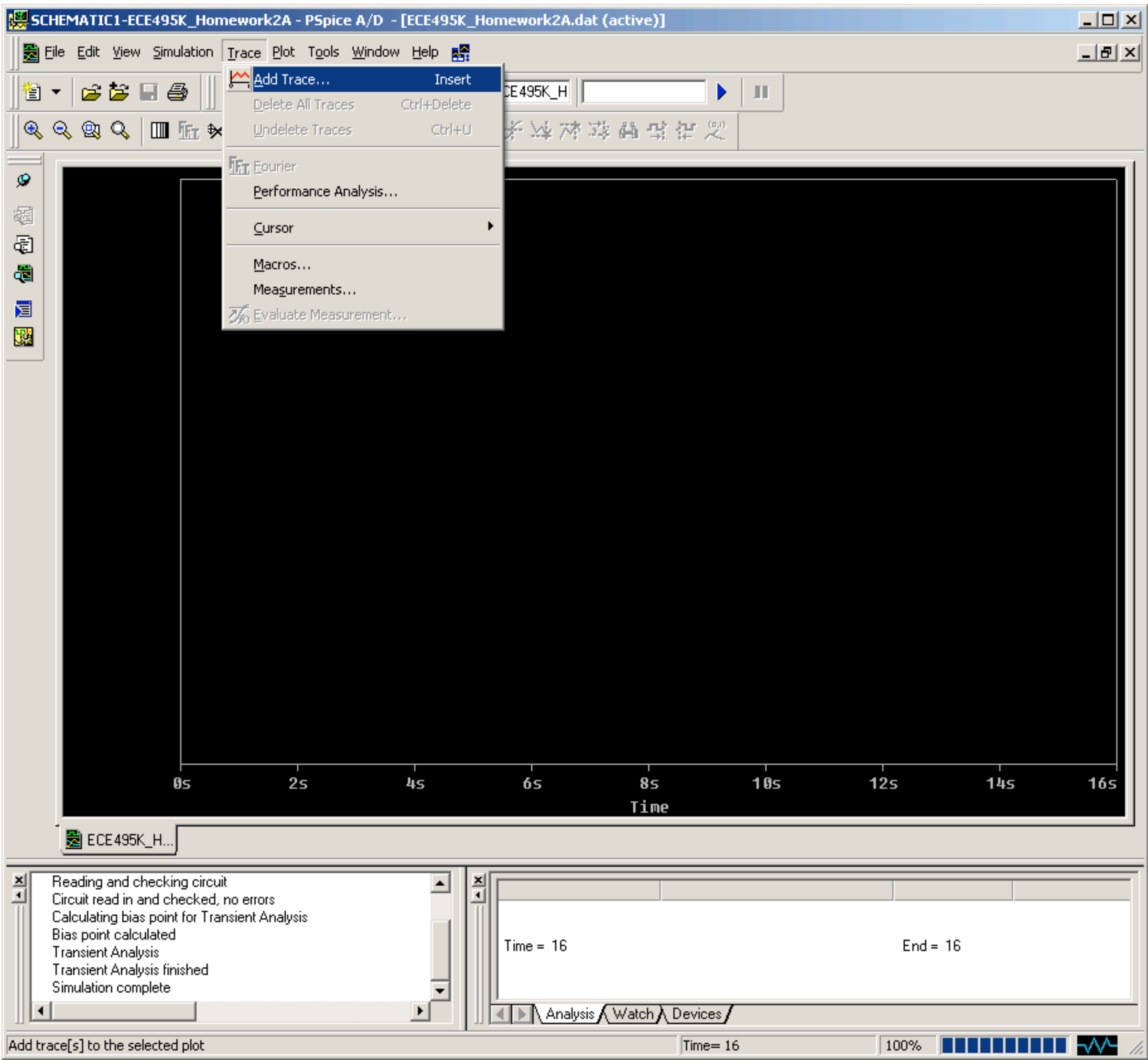


29. You should then get a dialog box as seen below. If you have set up your inputs correctly above, the only thing you need to do is change the “Run to time:” of the simulation to the largest period you set up (in this example – 16s, in your homework, it should be 256s).

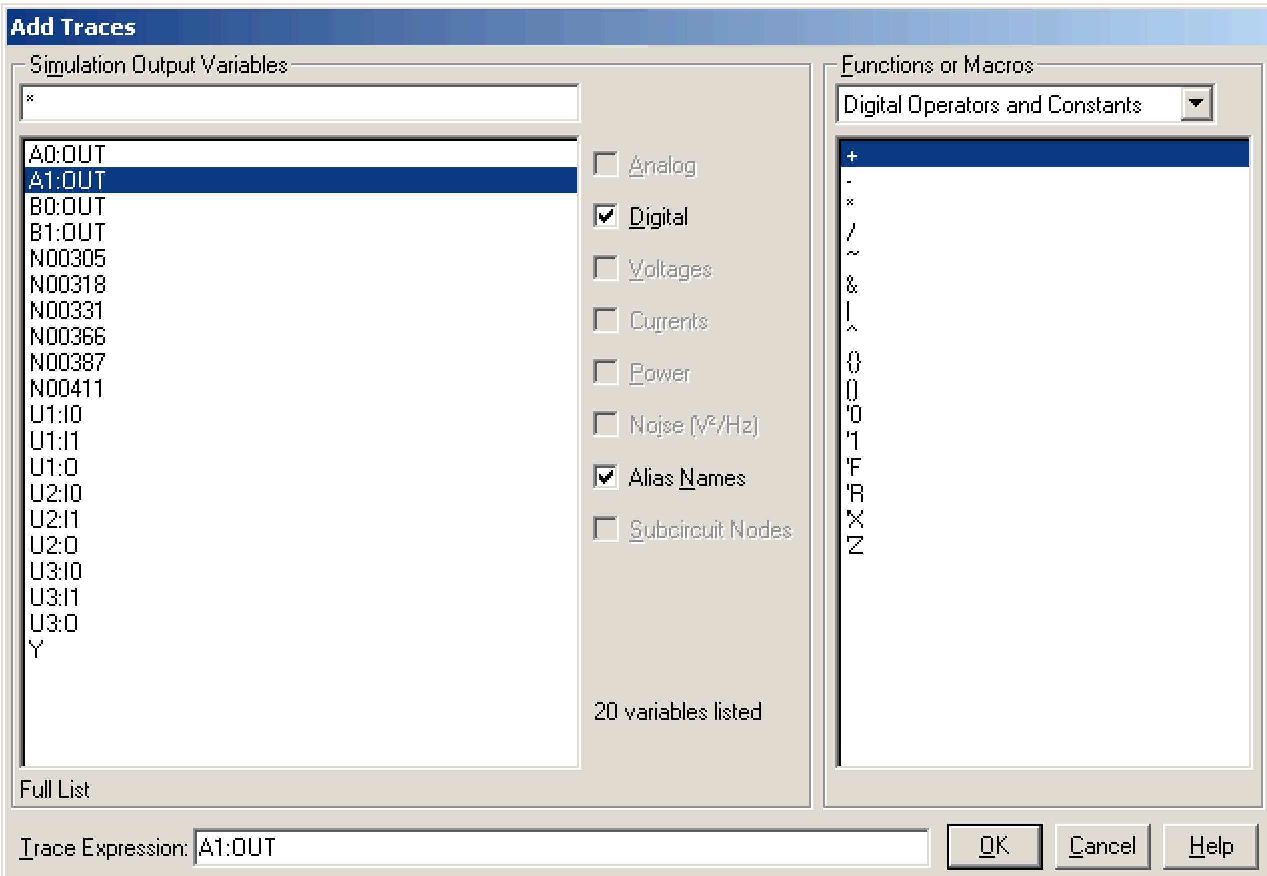


30. Click OK on the above dialog box. You should now be ready to run your simulation. Do so by clicking the  button.

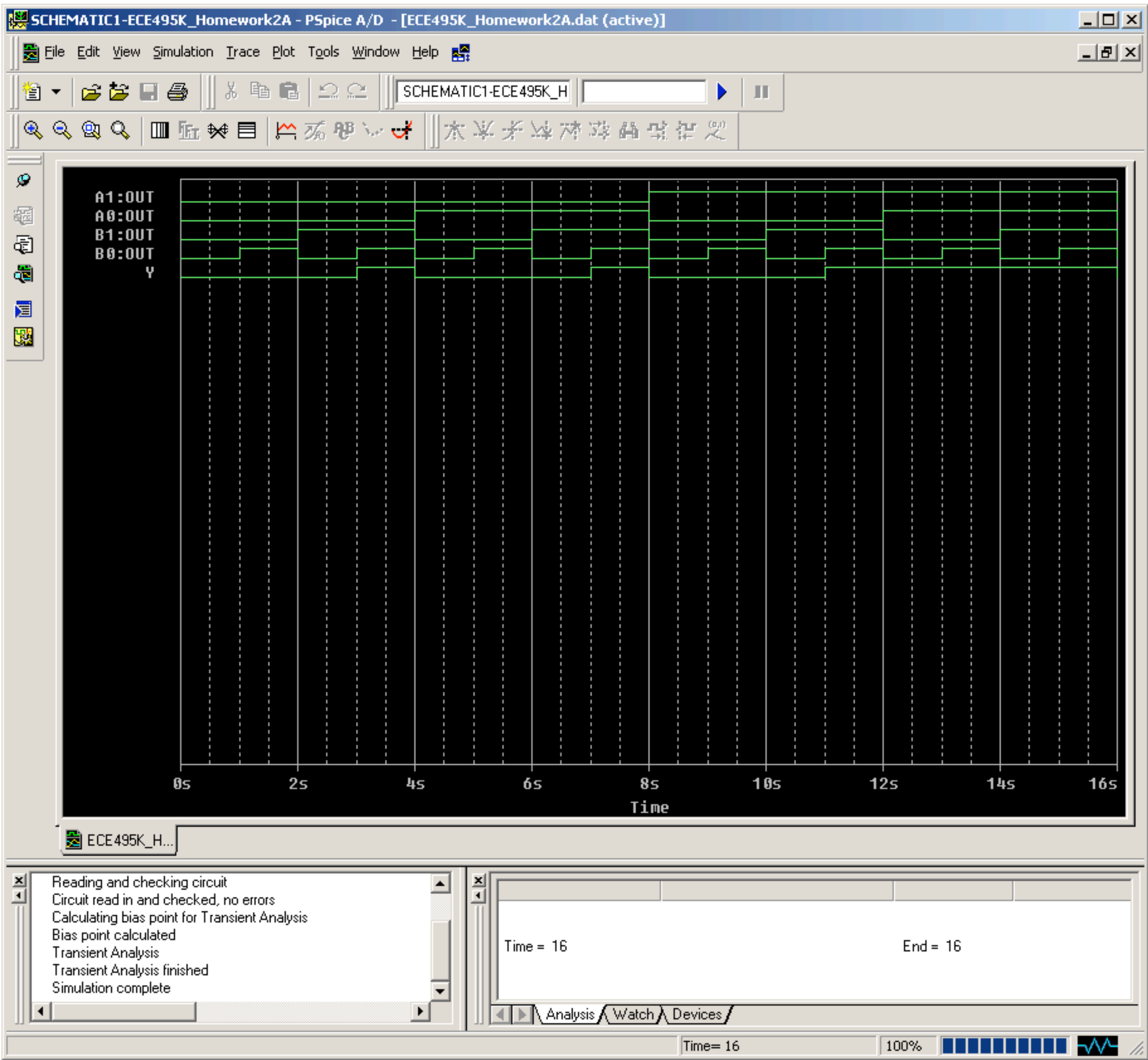
31. After the simulation has completed you should get an empty black box as shown below. We want to look at our inputs compared to our output. To do so, we select "Trace->Add Trace…"



32. The "Add Trace..." dialog box is seen below. We can only add one trace at a time. For now, select "A1:OUT" and click "OK".



33. Repeat steps 30 and 31 in the following order: A0:OUT, B1:OUT, B0:OUT, and Y. After doing so, your graph should look like below. If it does, you have completed the tutorial successfully! BUT keep reading, you are not quite done yet…



34. **For the actual homework (not this tutorial):** Put your .opj and .DSN file in a folder and zip it. Submit the zipped archive using Blackboard.

35. Further questions, comments, please address to the TA.