PSICE Tutorial

by Adam Mills

Step 1) Got to *Start>>Programs>>EE Programs>>PSICE Student>>Capture Student*

Step 2) In the Window that appears, go to
File>>New>>Project

Step3) In the "New Project" window, enter a file name such as "Tutorial."
Also, in the "Location" line at the bottom, click on "Browse" and select a location on your H-Drive. Then Click "OK."

Step 4) The Next window will ask to "Create Based Upon Existing Project" or "Create Blank Project." Choose "Create Blank Project." Click "OK"

Step 5) Now you are in the Main Schematic window. This is where you will draw the schematic of the circuit you wish to build. Soon we will build a simple test circuit with passive components.

**For this Tutorial we are going to build a simple Half-Wave Rectifier

Step 6) - Defining Sources:
Up on the Toolbar go to *Place>>Part*, From here there are several options.

- You may type "V" into the box labeled "Part." This will show you a list of Voltage Sources in the Window labeled "Part List"
- Then scroll through the part list until you find the voltage source you want.
- Select "VSIN"

Step 7) Now you mouse has the VSIN object attached to it. Click on the schematic window where you would like to place the part. Then Right Click and select "End Mode."

Step 8) Go to *Place>>Part* again and select a resistor by typing "R" into the "Part" line. Proceed the same as step 7. Make sure ANALOG is selected in the library section.

Step 9) Repeat step 7 for a DIODE or D element, choose D1n4148. In EVAL Library.

Step 10) Go to *Place>> Ground*. Select the GROUND/0 element.

Step 11) Wire the circuit together using *Place>>Wire.* You may also use the Wire button on the right hand side. It is the third from the top.

Next we must define the values of the source.

On the source are sever values that are undefined.
12) Double Click on the label called VOFF=. Set its value to 0.
13) Do the same for the other two values, setting VAMPL to 5 and FREQ to 100.

Next we must place sensors on to nodes.
14) On the tool bar, to the left of the large capital V, there are three buttons. One has a ball with V in it, the second with I, and the third with two little balls.
Place a V(voltage marker) on both sides to the diode. Place an I(current marker) anywhere on the loop.

Now we want to Run the simulation.
15) go to *PSPICE>>New Simulation Profile.*
16) Give the profile a name

Since we want to see want happens to the diode as the signal changes with time, we will choose time domain analysis.
17) Set the "Run to Time" Field to "15ms" and set the Maximum Step size field to ".01ms"
18) Next, Click on *PSPICE>>Run* or the blue arrow in the toolbar. If you get an error, just click the arrow again.
19) A window named Schematic should pop-up. There you can see the voltage across the source and the load. If you click on the symbol next to the marker node name until it is highlighted, then click on the icon that has a red squiggle with black crosshairs through it, you get a cursor that measures the value of the voltage or current at that node.
20) Add a capacitor to see the change, play with the value.

Next we will do a Frequency Sweep
21) Change the Voltage Source to VAC
22) Edit the simulation Profile for AC analysis: In simulation settings window, change the Analysis type to AC Sweep/Noise. Enter Values for Start, Stop and Points per Decade of .1, 1k and 20 respectively.
23) Run the simulation. Notice how the voltage across the Resistor is a function of source frequency.

Then we will do a DC Sweep
24) Replace VAC with VDC
25) Edit the simulation profile. Change Analysis type to DC Sweep. Where it asks for Source name, enter the name of the DC source. Start at 0Vdc, end at 10V with increment of .1V.

Now try some circuits on your own.