

Circuit Emulation Software

A shareware Berkeley Spice interface is available at:

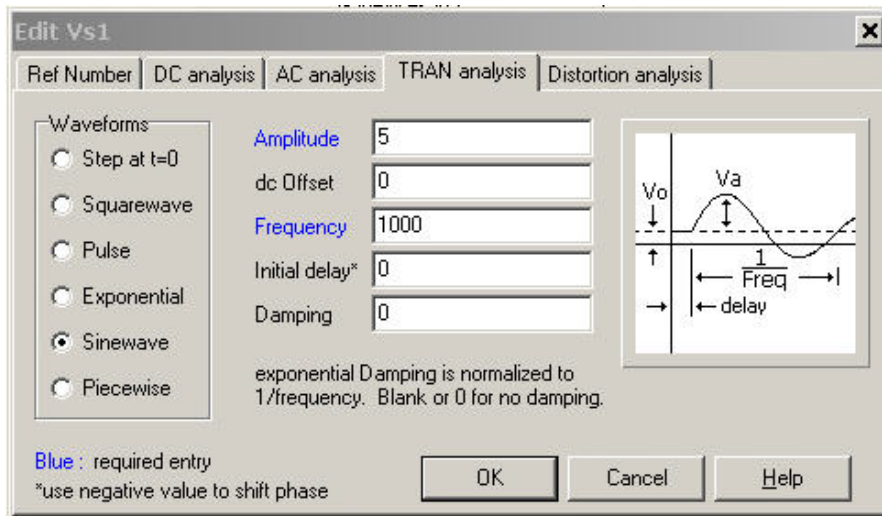
<http://www.5spice.com>

Load it onto your windows machine. When asked your company name put student.

Using the Software - Drawing a Circuit

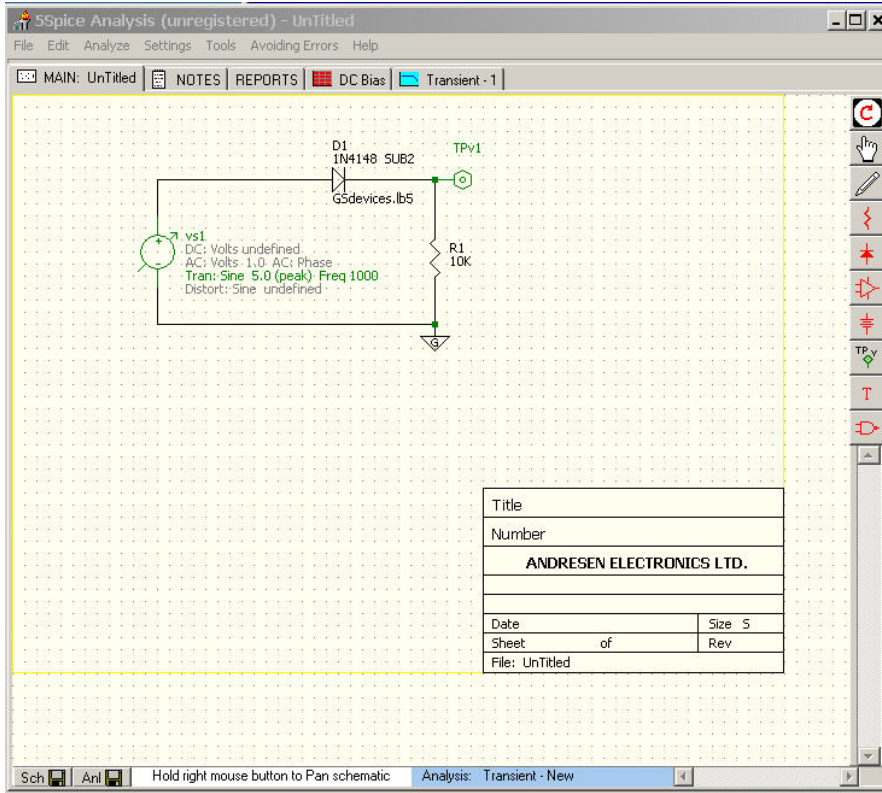
File -> New Schematic

- 1) On the screen drag over a resistor. Click on the resistor to rename it.
- 2) Drag over a diode. Left click to rotate it. Right click to rename and choose a model. Click on 1N4148 as model.
- 3) Drag over a power supply. Rename it. Click on TRAN tab (transient) and set up the power supply as in the screen below:



- 4) Drag over a ground.
- 5) Drag over a voltage test point. Click on it and rename it.
- 6) Connect the circuit with wires.

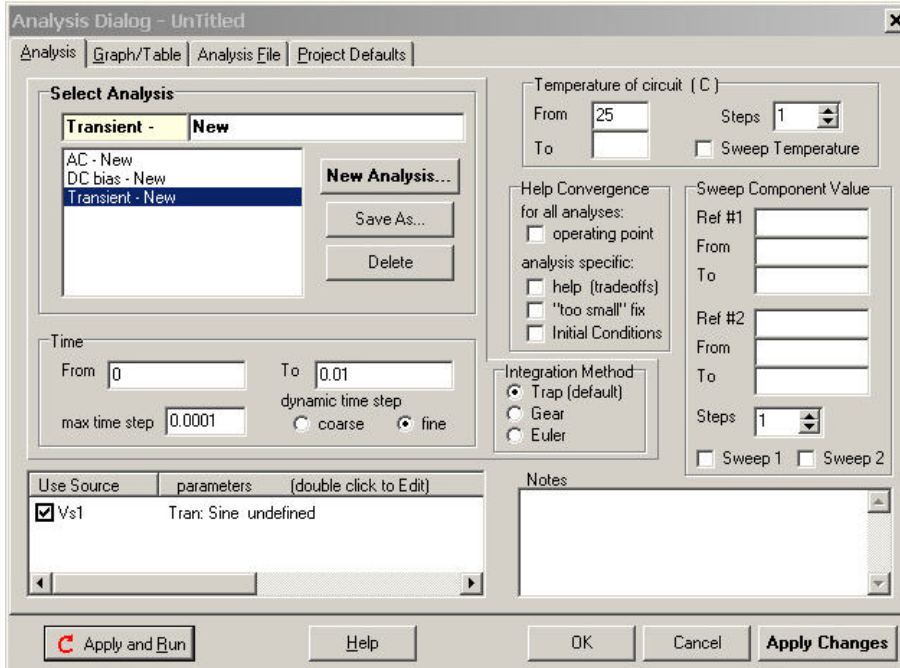
7) Your completed circuit should look like:



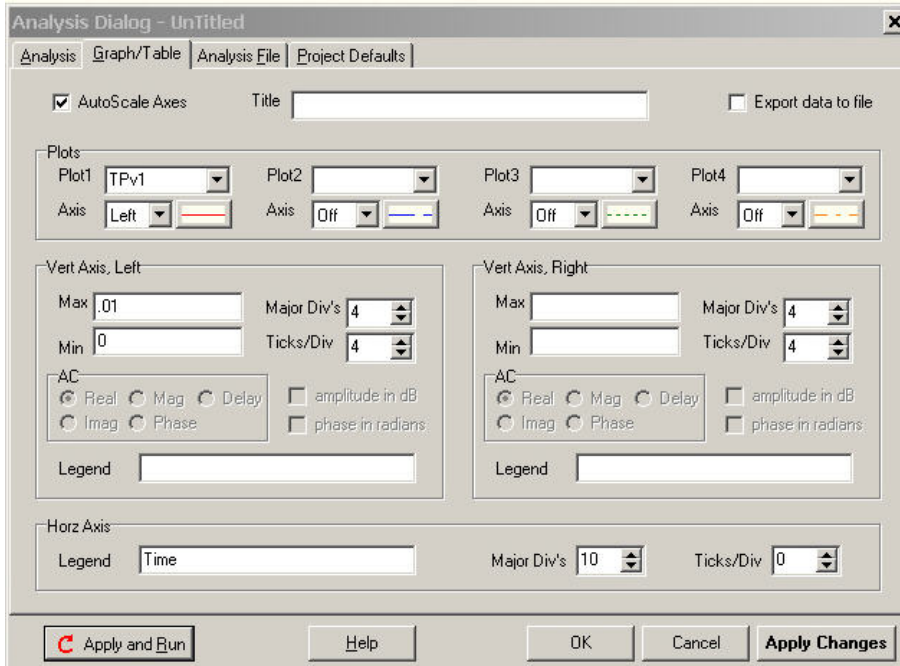
Using the Software – Analyzing the Circuit

Analyze -> (Select/Edit) Analysis Dialog

1) Select transient. Click on Power Supply. Set time and step. See below



2) Click on Graph/Table tab. Fill in table as below.



3) Click on Apply Changes. Click on Apply and Run. You should see your simulated output graphed as below.

