

Tutorial to Pspice9.1:

1. Go to the Orcad folder and start the Capture CIS(demo).
2. On the File menu of Orcad capture window, select New→ “Project”
3. Use an appropriate name for the project and select “Analog or Mixed Signal Wizard” and use “c:\temp” as the path
4. Add the following libraries to the list of libraries to be used: analog.olb, source.olb, sourcstm.olb, special.olb and click on finish.
5. Click on the Place menu, select Part. On the pop-up window, select “ANALOG” for libraries and select “C” for part, and click ok to place a capacitor on the capture window. Place the part on the window, right click and select “End Mode”. Double click on the value(1n) and change the value to 100n in the pop-up window.
6. Repeat the step 5 but select R to place to resistor in the capture window. Double click on the value(1K) and change the value to 100. Don’t forget to right click and select “End Mode”.
7. Click on the “GND” icon on the right-side of the window, select “0/SOURCE”(or select “SOURCE” for libraries and “0” for symbol) click ok. Place the ground symbol on the window.
8. Repeat step 5 but select “SOURCE” libraries and select VAC.
9. Click on Place menu and select wire. Connect all the above parts as shown in the following figure.

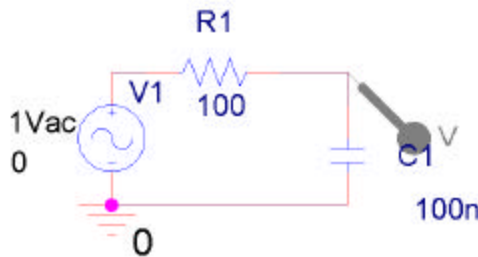


Figure 1: A simple RC circuit

10. Click on Pspice menu, select “Markers”, select “Voltage level”. Place the marker on the node between R1 and C1. The markers automatically plot the voltage at the corresponding node once the simulation is completed.
11. Click on Pspice menu, select “New Simulation Profile”. Give an appropriate name and click ok. Select “AC Sweep” for Analysis type. Select “Logarithmic” for “AC sweep type”. Use 100 for the start frequency, 1Meg for “End Frequency” and 100 for “Points Per decade and click ok.
12. Click on Pspice menu, select “run”. A Pspice A/D window showing you the plot should open up. Also you can plot any node by selecting “Add trace” on the Trace menu.
13. Delete the VAC source. Using the procedure in step 5 select “SOURCE” for libraries and “VPULSE” for part. Substitute VPULSE in place of the VAC source. Modify the values in VPULSE to the following

V1=0, V2=5, TD=1u, TR=1n, TF=1n, PER=1m

14. Click on Pspice menu, select "Edit Simulation profile". Change the Analysis Type to "Time Domain(Transient)". Use 100u for Run time, 0 for Start saving data after and 0.1u for Maximum step size.
15. Click ok. Click on Pspice menu in the Capture window and select "Run". The plot window should open up.

]