OrCAD ‘QUICK START’ Guide #3
Running a Simulation with PSpice A/D

Step #1 - Creating a Simulation Profile

Before you configure a simulation type, you need to have edited the circuit’s part values, eg voltages, resistances, etc. If you need to review how to do this, refer:

Step #5 – Editing Part Values in QUICK START Guide #2 - Designing a Circuit with Capture.

Then, before you run a simulation, you must create a simulation profile.

To create a profile, choose the menu path ***PSpice — New Simulation Profile*** and then give it a name in the New Simulation dialogue box as shown in Figure 8.

Note: You can click the 'Open Design' button to open a project with a simulation profile created for you. Choose *PSpice — Edit Simulation Profile* to open the profile and view its settings.

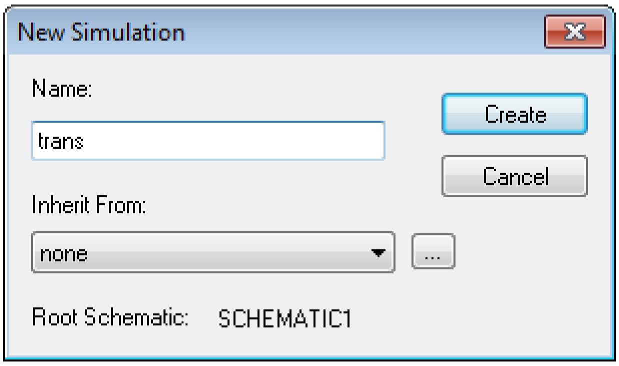


Figure 8. New Simulation Dialogue Box

Click *Create* to open the **Simulation Settings** dialogue box, as shown in Figure 9.

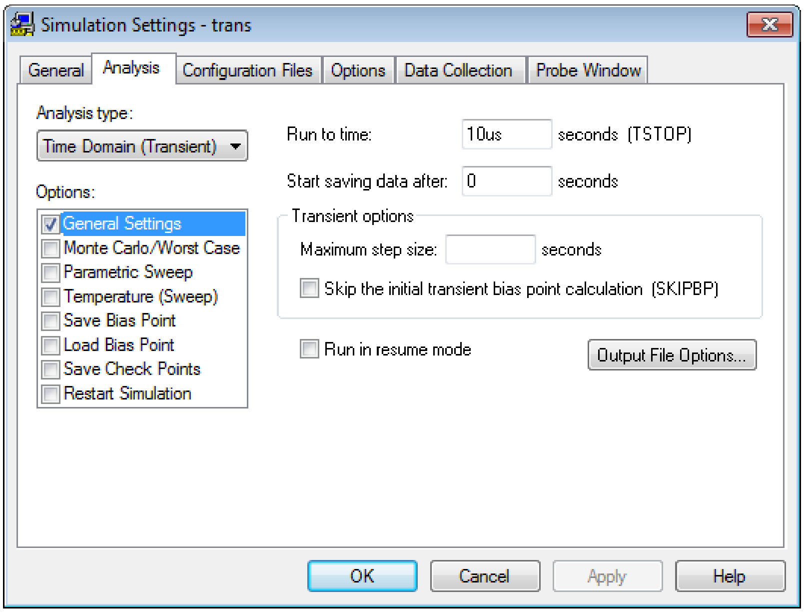


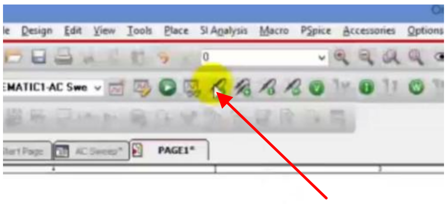
Figure 9. Simulation Settings Dialogue Box

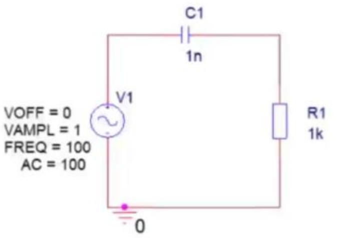
You can specify any of the analysis types, set various parameters for the simulation profile, and select different options for that simulation type.

Once done, select OK, which returns you to the circuit schematic.

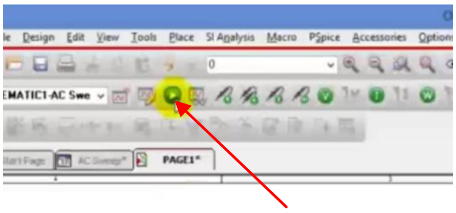
Step #2 – Starting a simulation run

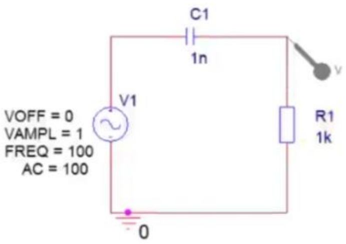
Before starting a simulation run, you need to place a probe (or probes) at the points you wish to measure a characteristic, such as voltage.





Once you've placed the appropriate probe, you can start the run by selecting the RUN icon in the top menu bar.





Alternately, you can use the menu path *PSpice / Run.*

Step #3 - Viewing Results

The Probe window, as shown in Figure 10, opens after a simulation is complete.

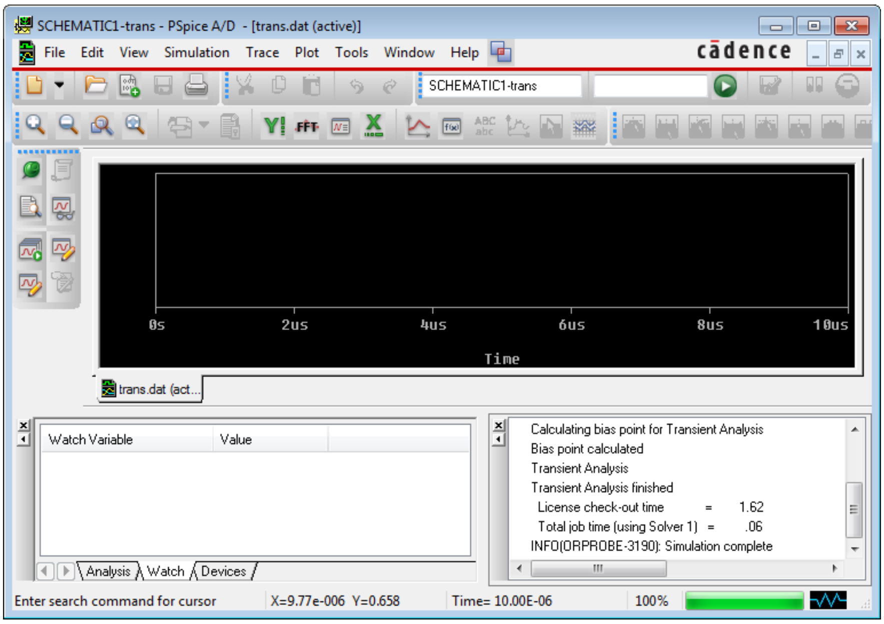


Figure 10. PSpice Probe Window

You will display the waveform for the voltage at *pin 2* of the capacitor, *C1*.

Choose *Trace — Add Trace* to open the Add Traces window. Select *V(C1:2)*, as shown in Figure 11, and click *OK*.

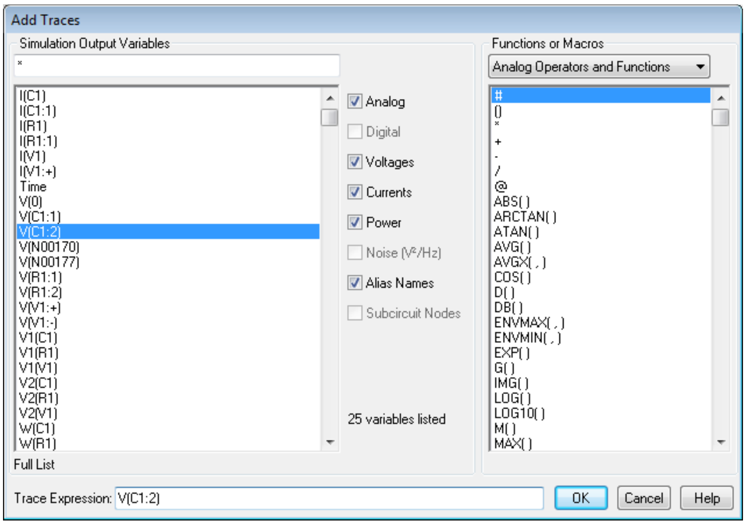


Figure 11. Add Traces Window

The resultant waveform is displayed in the probe window, as shown in Figure 12.

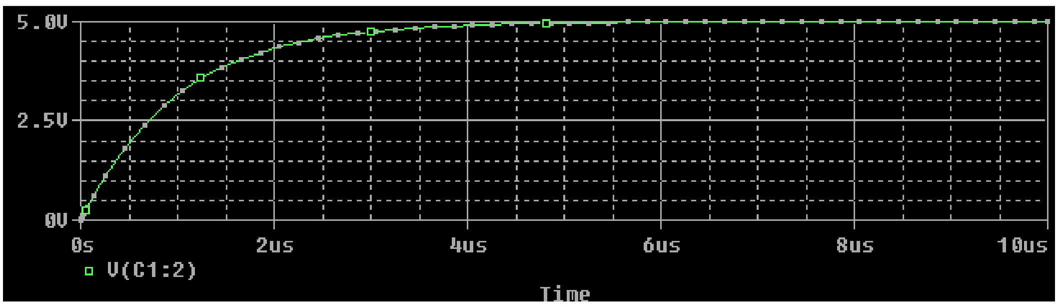


Figure 12. Resultant Waveform

A Selection of PSpice How-To Videos	
Getting Started	https://www.youtube.com/watch?v=mglyHxZ9Qkc&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO&index=6
Simulation Types Tutorial	https://www.youtube.com/watch?v=qhkVgKYoo04&index=3&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO
Monte Carlo Tutorial	https://www.youtube.com/watch?v=tFyMdOSAY_4&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO
Parametric Analysis Tutorial	https://www.youtube.com/watch?v=vaiRDVM9oy8&index=2&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO
PSpice How-To Videos related to the text book 'Analog Design and Simulation using OrCAD Capture and PSpice' *	
PSpice Simple Circuit Exercise Page 13	https://www.youtube.com/watch?v=KxrIxFHzQnM&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO&index=12
PSpice Simple Circuit Exercise Page 57	https://www.youtube.com/watch?v=iCiw_Ha48rQ&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO&index=13
PSpice Simple Circuit Exercise Page 74	https://www.youtube.com/watch?v=8UTCnuAmopw&index=14&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO
PSpice Simple Circuit Exercise Page 131	https://www.youtube.com/watch?v=xy0qkdSvnW8&index=15&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO
PSpice simple circuit Exercise Page 138	https://www.youtube.com/watch?v=B76-szcdNyM&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO&index=16
PSpice simple circuit Exercise Page 151	https://www.youtube.com/watch?v=B2n-HOUTSEk&list=PLCgZ9VoSvNf1KC1wnrpO86Cp3M1IeA9LO&index=17

* Copies can be purchased online at: www.ecadtools.com.au/student-registration.html