EECE 200 – Introduction to Electrical and Computer Engineering

Lecture 4 – Introduction to SPICE

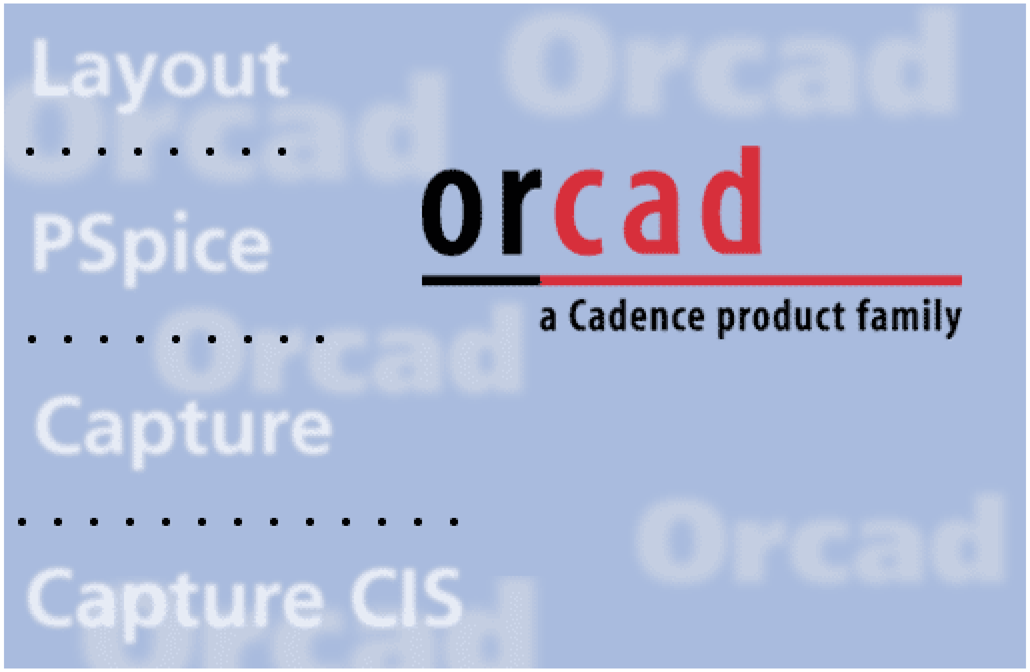
October 15, 2009

•Simulation

SPICE

PSpice

- PC version of Spice
  - Student version (9.2) is limited but is for free
  - Also available in the labs is the full version (15.7)



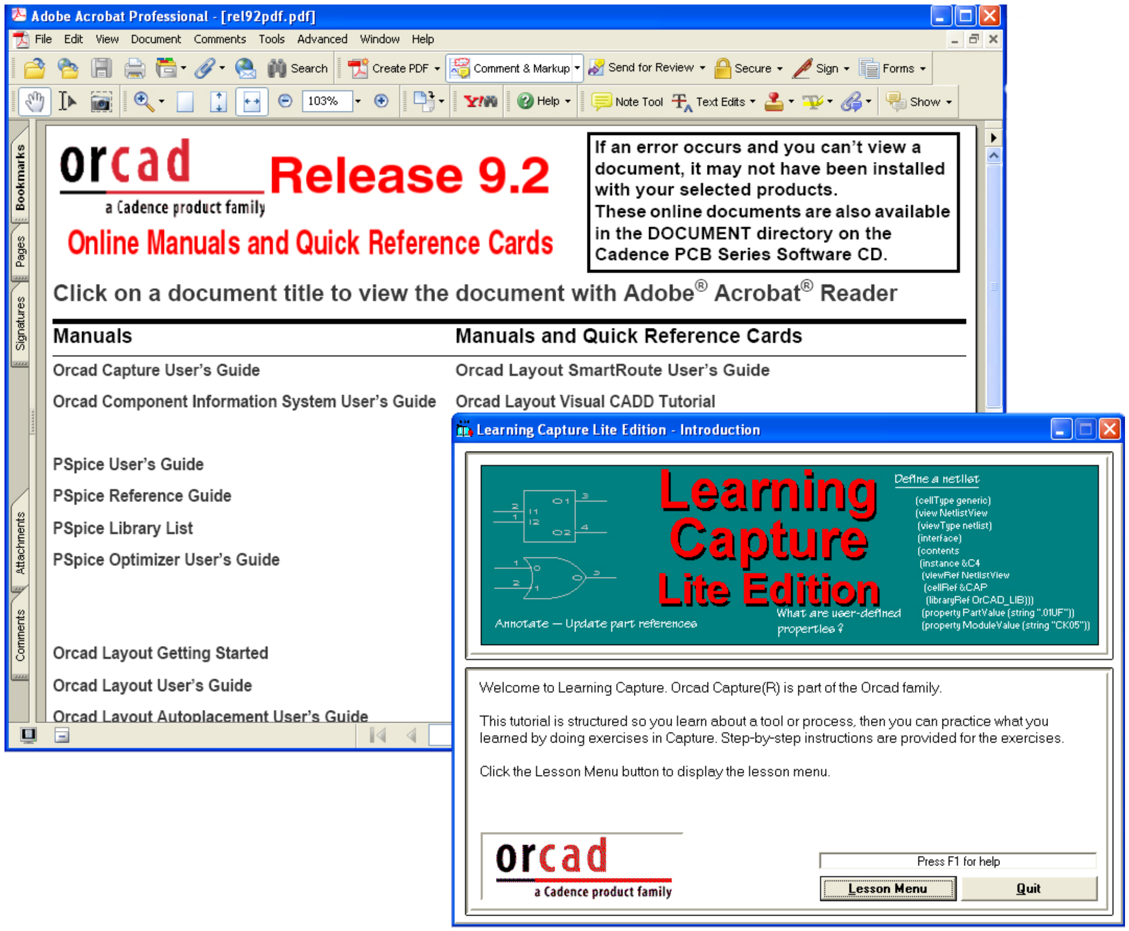
Types of circuit analysis

- Non-linear DC analysis
- Non-linear transient and Fourier analysis
- Linear AC analysis
- Parametric analysis
- Monte Carlo analysis

Component Libraries

- Independent and dependent voltage and current sources
- Resistors
- Capacitors
- Inductors
- Mutual inductors
- Transmission lines
- Operational amplifiers
- Switches
- Diodes
- Bipolar transistors
- MOS transistors
- JFET
- MESFET
- Digital gates

Learning Your self



Steps involved

Circuit Creation with capture

- Create a new Analog, mixed AD project
- Place the circuit parts
- Connect the parts
- Specify values and names
- Create a net-list

Specify type of simulation

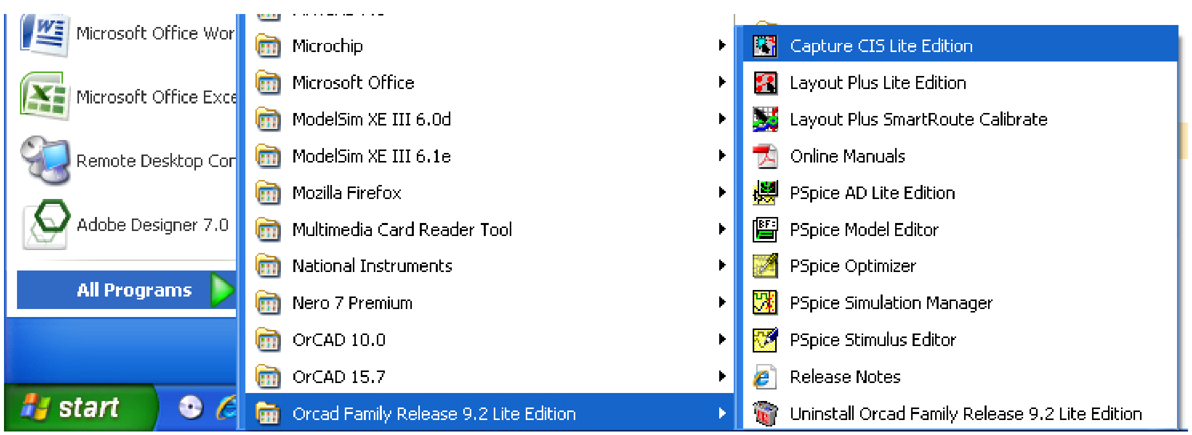
- Create a simulation profile
- Select type of analysis
- Run PSpice

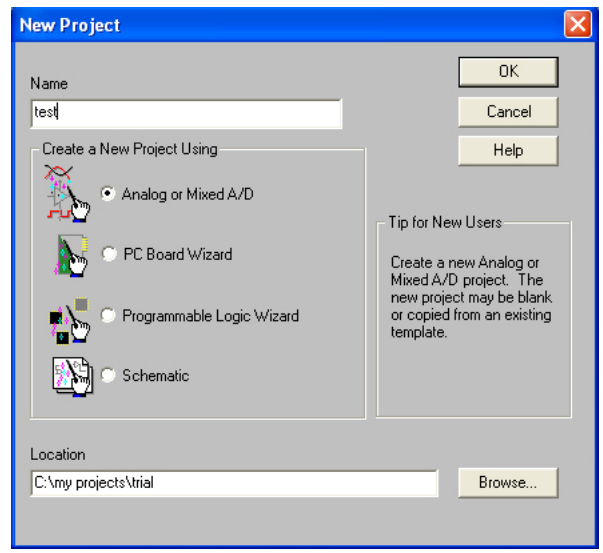
View the results

- Add traces to the probe window
- Use cursors to analyze waveforms
- Check the output file
- Save and print results

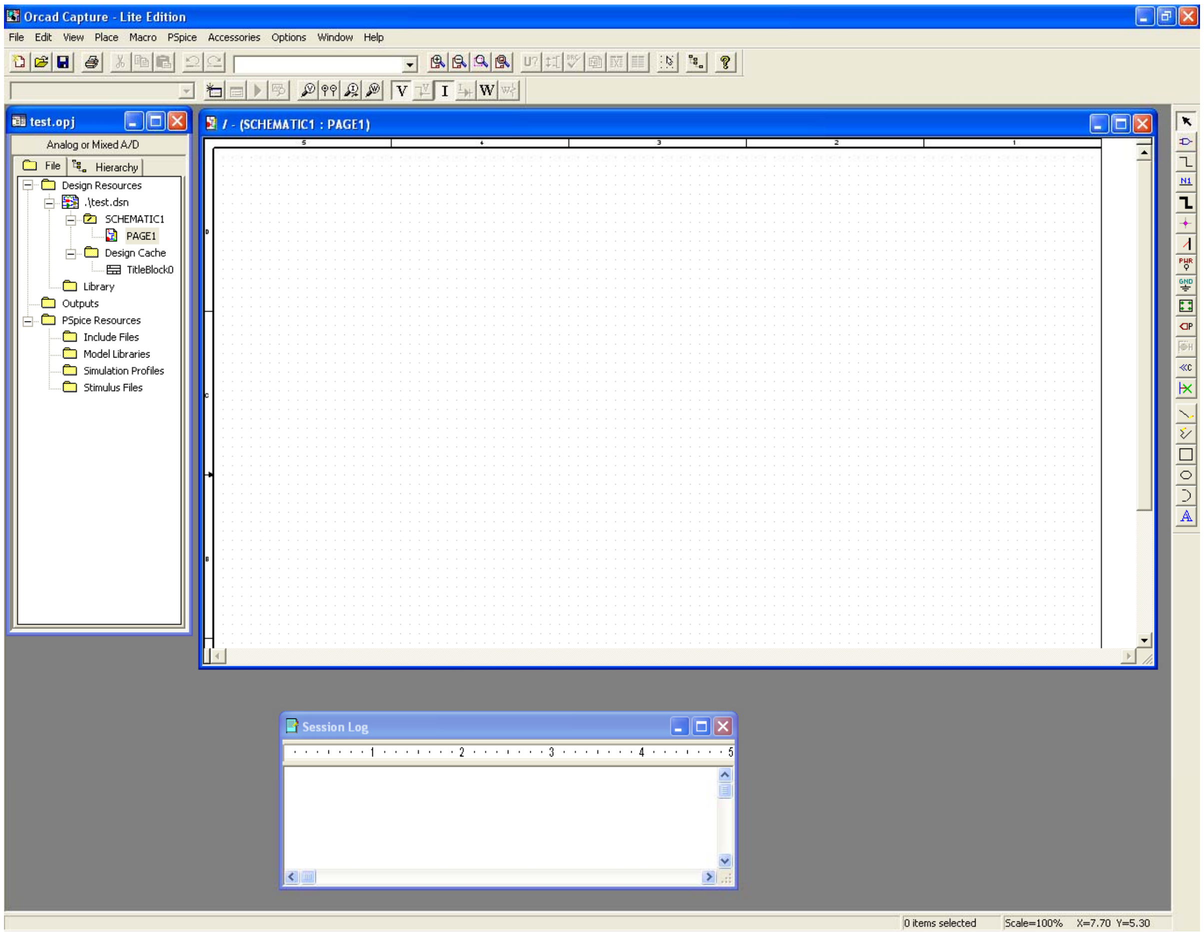
Circuit Creation with Capture

- Creating a new Analog, mixed AD project
  - Go the start menu and select start → All Programs → Orcad Family Release 9.2 Lite Edition → Capture CIS lite edition
  - Once Capture CIS is open you need to set up your circuit project. Make sure you select Analog or Mixed A/D



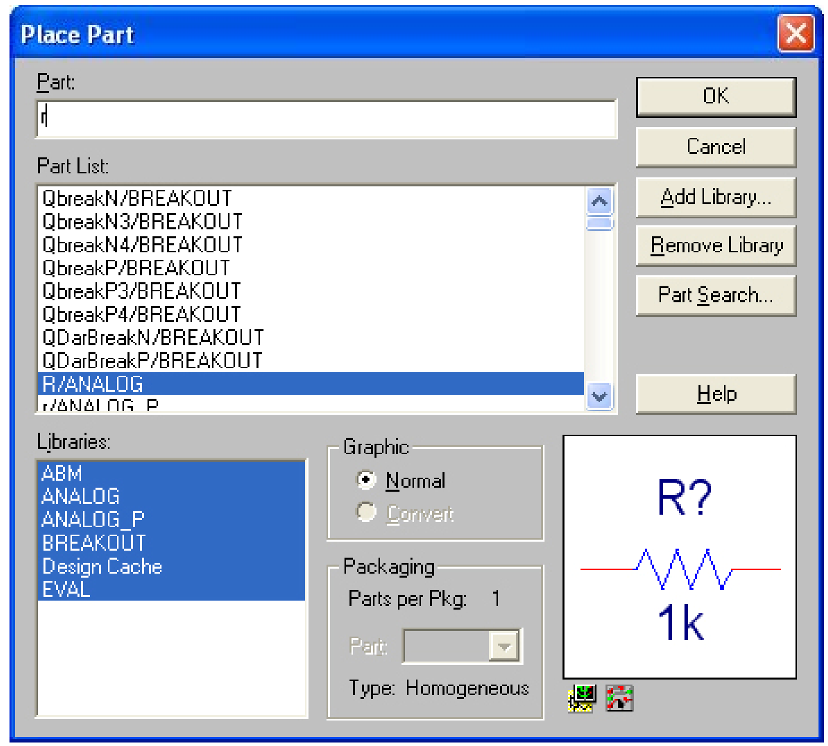


The Capture window



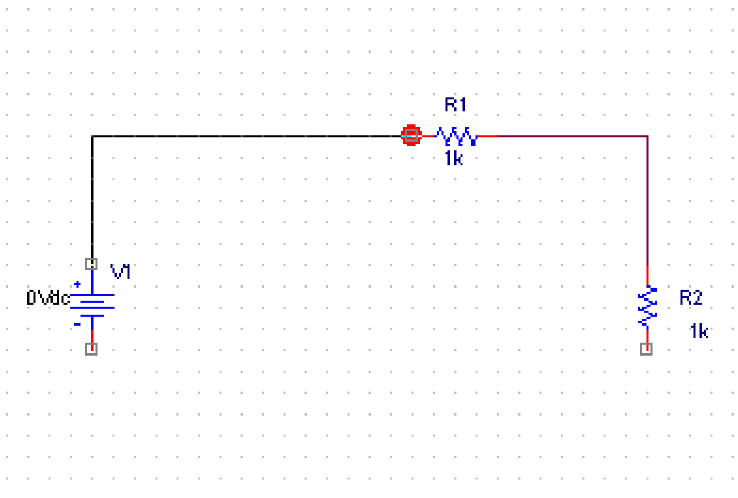
Placing the circuit parts

- Go to Place → Part
  - Analog: Passive components R, L, C
  - Source: Voltage and current VDC, IDC
  - Eval: Diodes op-amps transistors



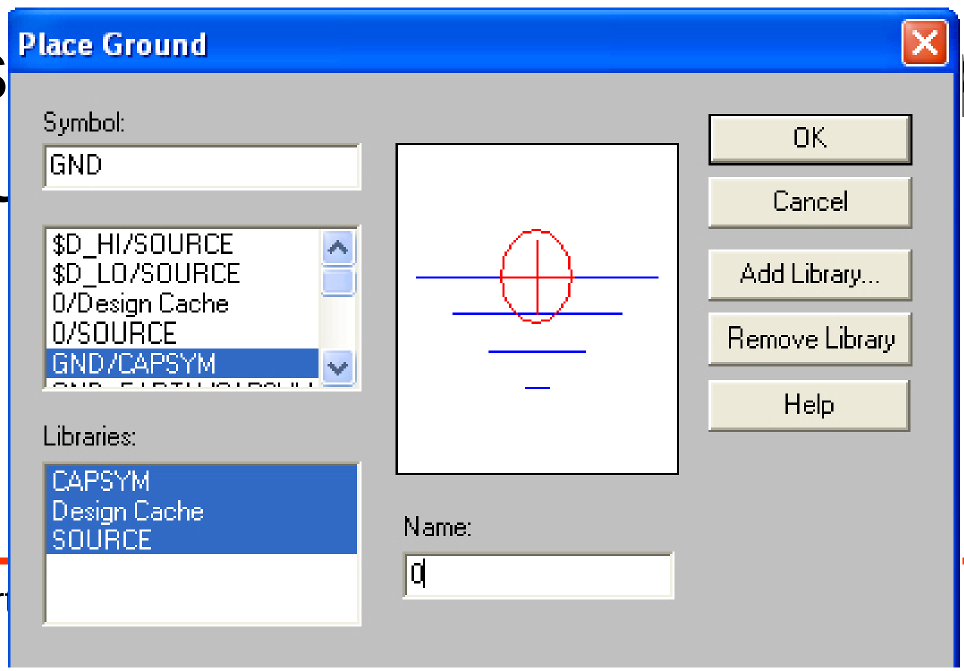
Connecting the circuit

- All you need to do is use the wiring tool
- Be careful of unwanted nodes
- Two crossing wires are connected if and only if they have a DOT on the intersection.



Circuit Ground (0)

- Each PSpice circuit requires a ground point
- The ground point should have “0” as a name
- To ins place →grou

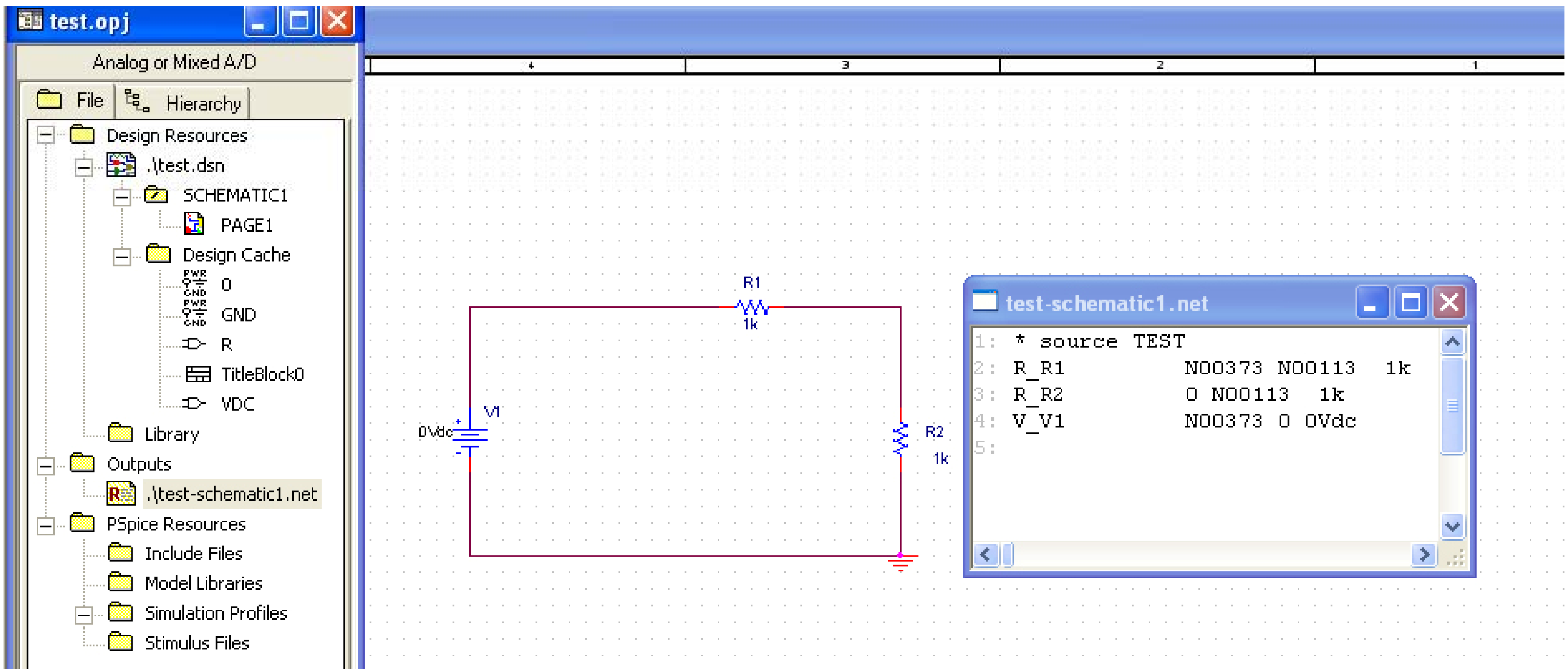


Specifying the values

- Double click on the value of each element and replace it with the desired one.
- Never change the parts identifier number
- Scale factors
  - T for Tera (= 1E12);
  - G for Giga (= E9);
  - MEG for Mega (= E6);
  - K for Kilo (= E3);
  - M for Milli (= E-3);
  - U for Micro (= E-6);
  - N for Nano (= E-9);
  - P for Pico (= E-12)
  - F for Femto (= E-15)

Creating a netlist

- Select PSpice → Create netlist
- In the outputs folder a file .net should appear



Steps involved

Circuit Creation with capture

- Create a new Analog, mixed AD project
- Place the circuit parts
- Connect the parts
- Specify values and names
- Create a net-list

Specify type of simulation

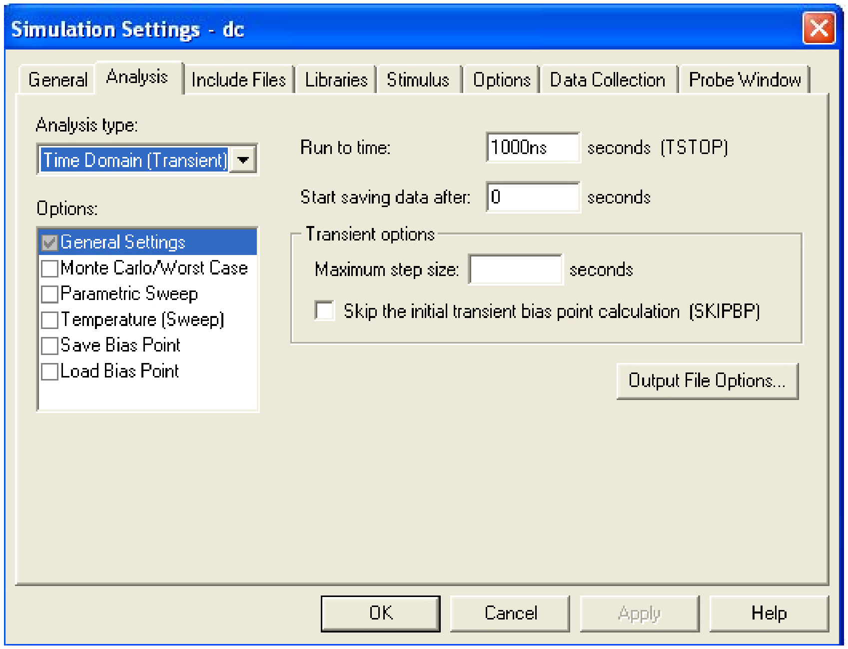
- Create a simulation profile
- Select type of analysis
- Run PSpice

View the results

- Add traces to the probe window
- Use cursors to analyze waveforms
- Check the output file
- Save and print results

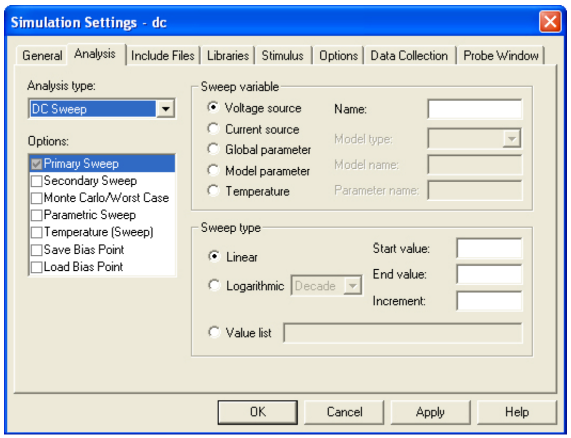
Specifying the type of simulation

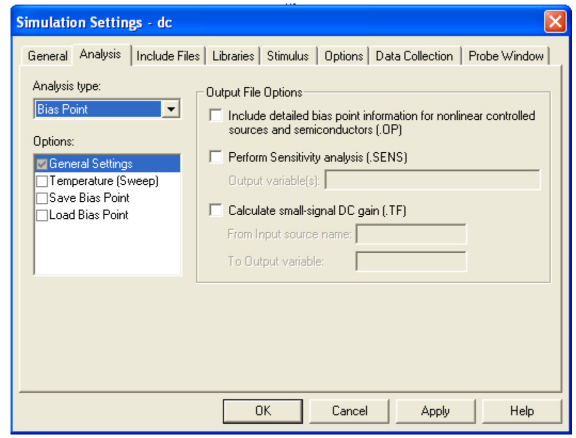
- Create a simulation profile
  - Select PSpice →New simulation profile
  - Give the simulation a name

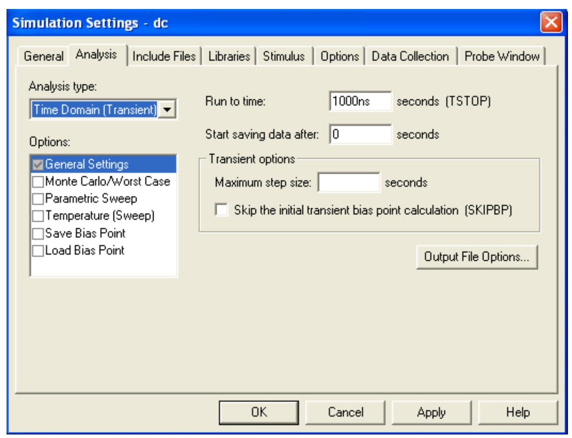


Specifying the type of simulation

- Selecting the type of Analysis

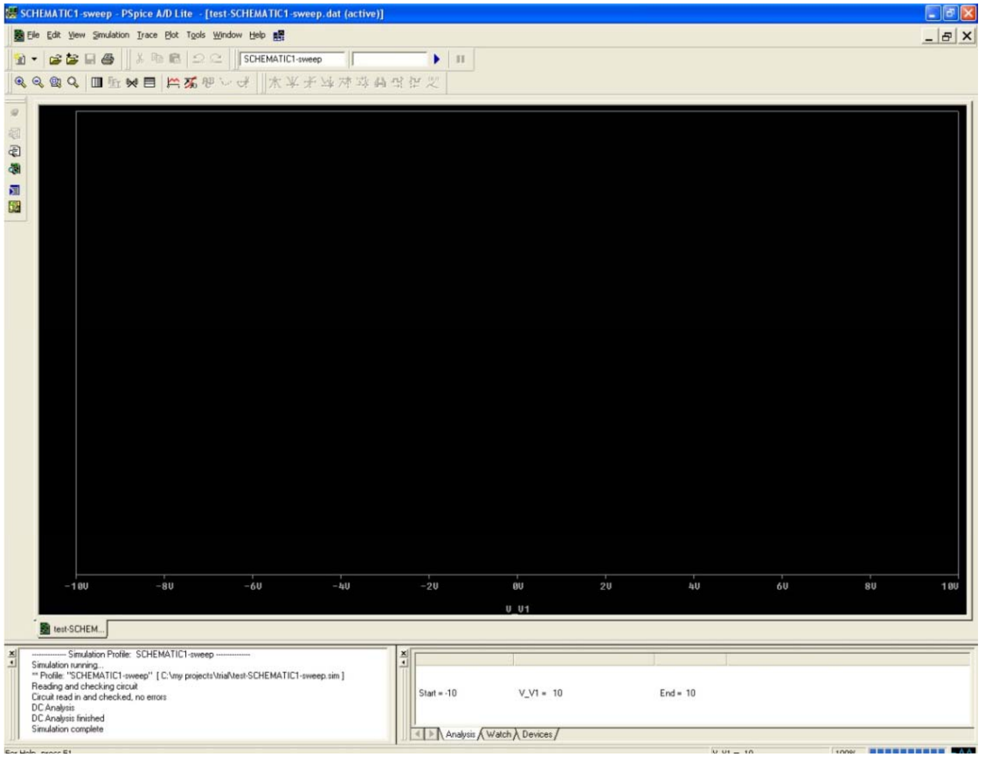






Specifying the type of simulation

- Running the simulation
  - Select PSpice →Run



Steps involved

Circuit Creation with capture

- Create a new Analog, mixed AD project
- Place the circuit parts
- Connect the parts
- Specify values and names
- Create a net-list

Specify type of simulation

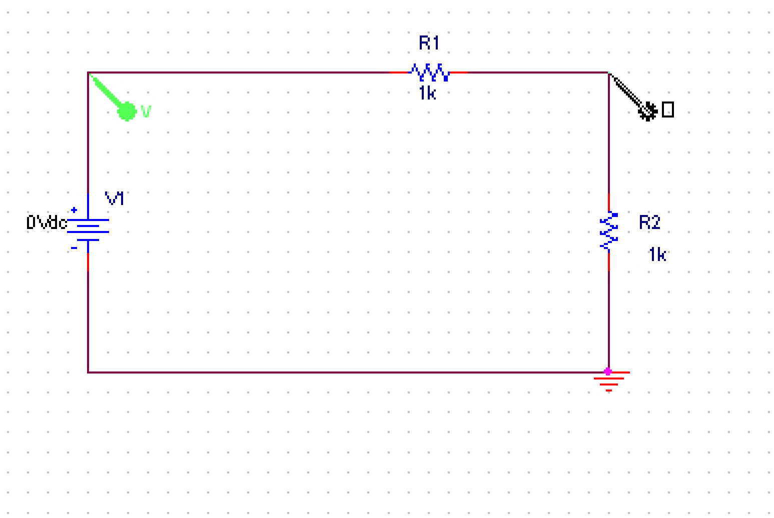
- Create a simulation profile
- Select type of analysis
- Run PSpice

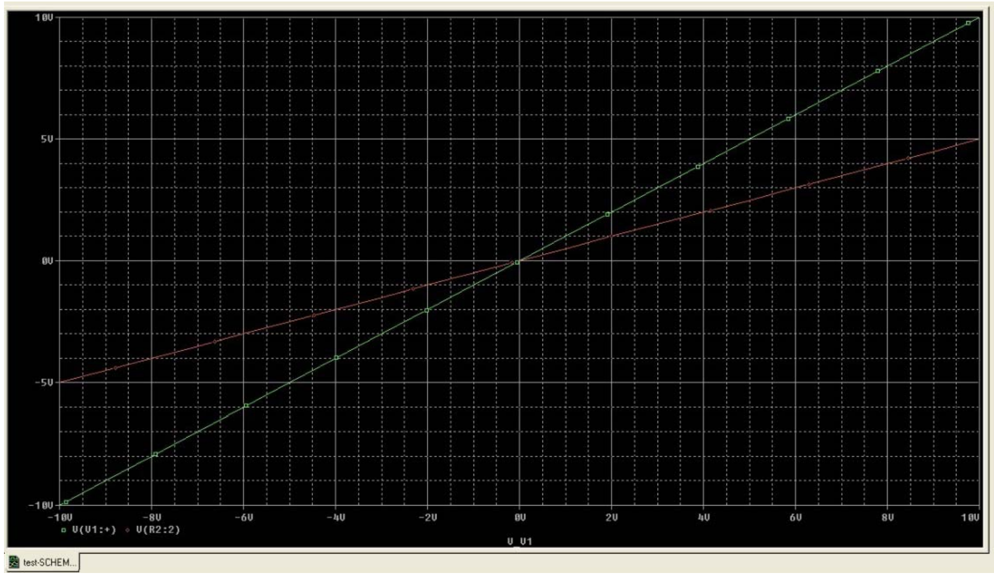
View the results

- Add traces to the probe window
- Use cursors to analyze waveforms
- Check the output file
- Save and print results

Viewing the results

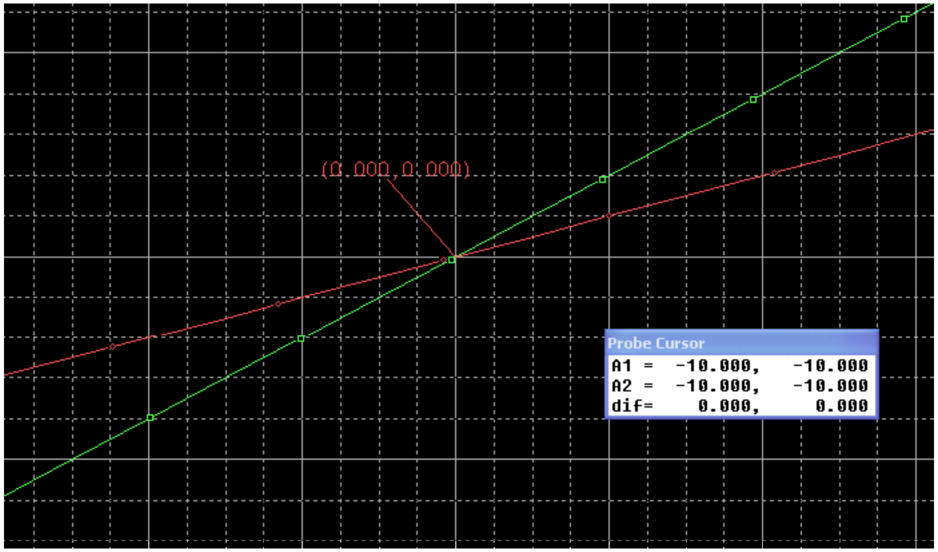
- Adding a trace to the simulation window
  – Select a probe and place it on the desired node





Viewing the results

- Using the cursor to analyze waveforms
  – Click on Toggle cursor icon
  – Move the cursor to the desired point
  – Click on mark label



DEMO