
EECE 200 – Introduction to Electrical and Computer Engineering

Lecture 4 – Introduction to SPICE
October 15, 2009



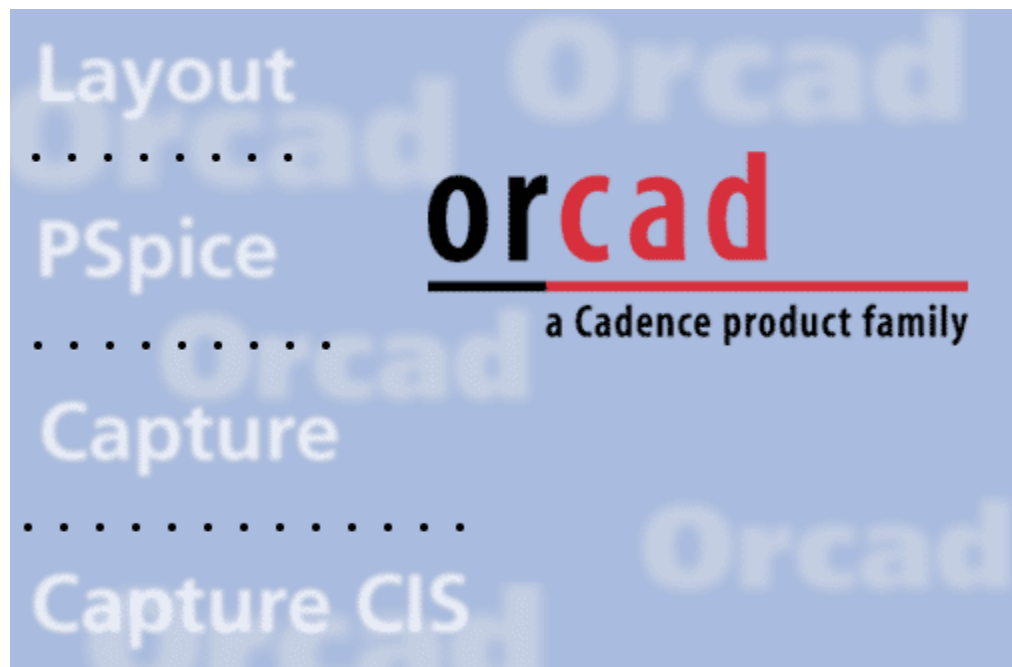
SPICE

•Simulation



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

The image shows a screenshot of Adobe Acrobat Professional displaying the Orcad Release 9.2 online manuals and quick reference cards. The main window is titled "Adobe Acrobat Professional - [rel92pdf.pdf]" and shows a list of manuals and quick reference cards. A smaller window titled "Learning Capture Lite Edition - Introduction" is overlaid on the main window, displaying a circuit diagram and text about the tutorial.

Adobe Acrobat Professional - [rel92pdf.pdf]

File Edit View Document Comments Tools Advanced Window Help

Search Create PDF Comment & Markup Send for Review Secure Sign Forms

103%

Note Tool Text Edits Show

orcad Release 9.2
a Cadence product family

Online Manuals and Quick Reference Cards

If an error occurs and you can't view a document, it may not have been installed with your selected products. These online documents are also available in the DOCUMENT directory on the Cadence PCB Series Software CD.

Click on a document title to view the document with Adobe® Acrobat® Reader

Manuals	Manuals and Quick Reference Cards
Orcad Capture User's Guide	Orcad Layout SmartRoute User's Guide
Orcad Component Information System User's Guide	Orcad Layout Visual CADD Tutorial
PSpice User's Guide	
PSpice Reference Guide	
PSpice Library List	
PSpice Optimizer User's Guide	
Orcad Layout Getting Started	
Orcad Layout User's Guide	
Orcad Layout Autoplacement User's Guide	

Learning Capture Lite Edition - Introduction

Define a netlist:
(cellType generic)
(view NetlistView)
(viewType netlist)
(interface)
(contents)
(instance &C4
(viewPRef NetlistView
(cellPRef &CAP
(library OrcAD_LIB)))
(property PartValue (string "01UF1"))
(property ModuleValue (string "CK05"))

What are user-defined properties?

Annotate — Update part references

Welcome to Learning Capture. Orcad Capture(R) is part of the Orcad family.

This tutorial is structured so you learn about a tool or process, then you can practice what you learned by doing exercises in Capture. Step-by-step instructions are provided for the exercises.

Click the Lesson Menu button to display the lesson menu.

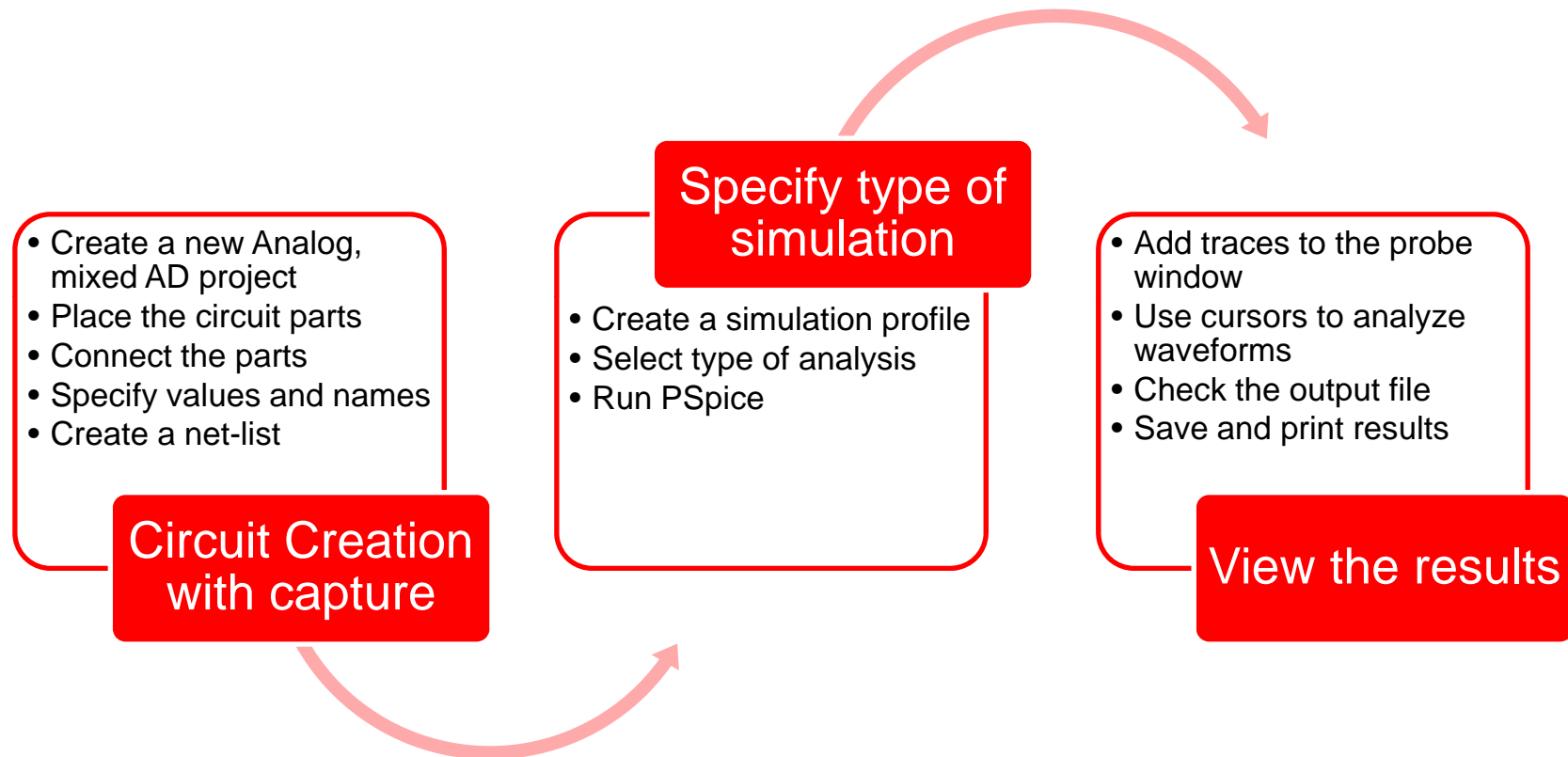
Press F1 for help

orcad
a Cadence product family

Lesson Menu **Quit**



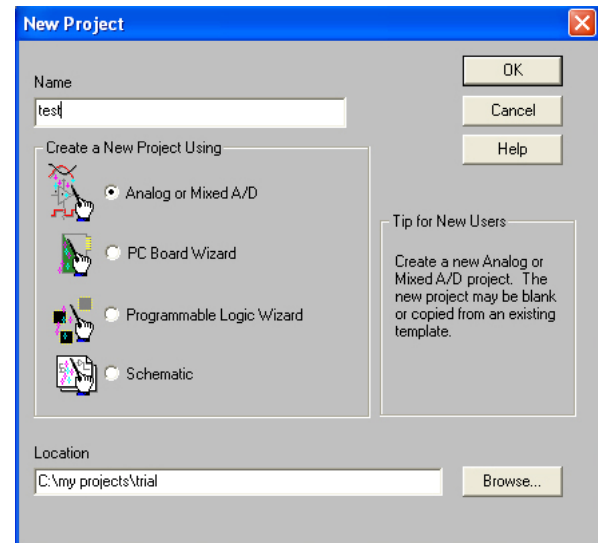
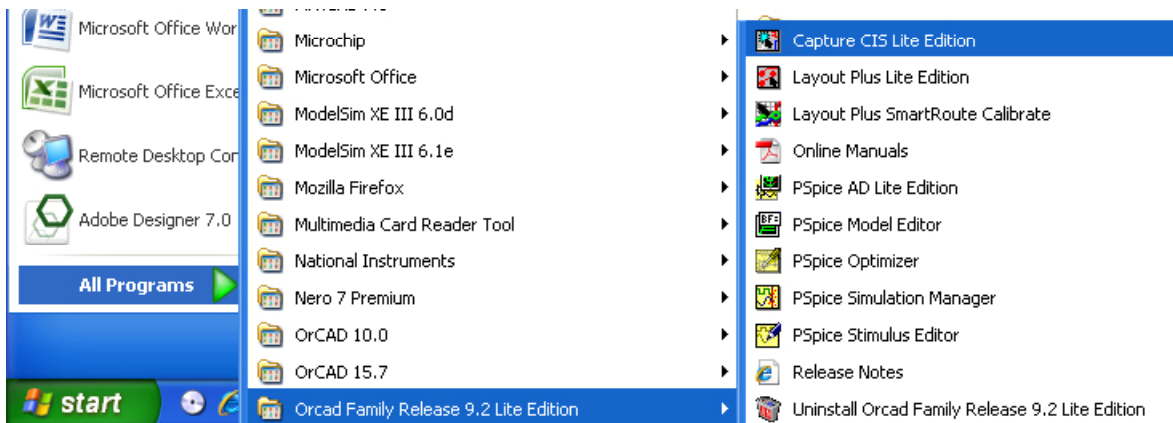
Steps involved



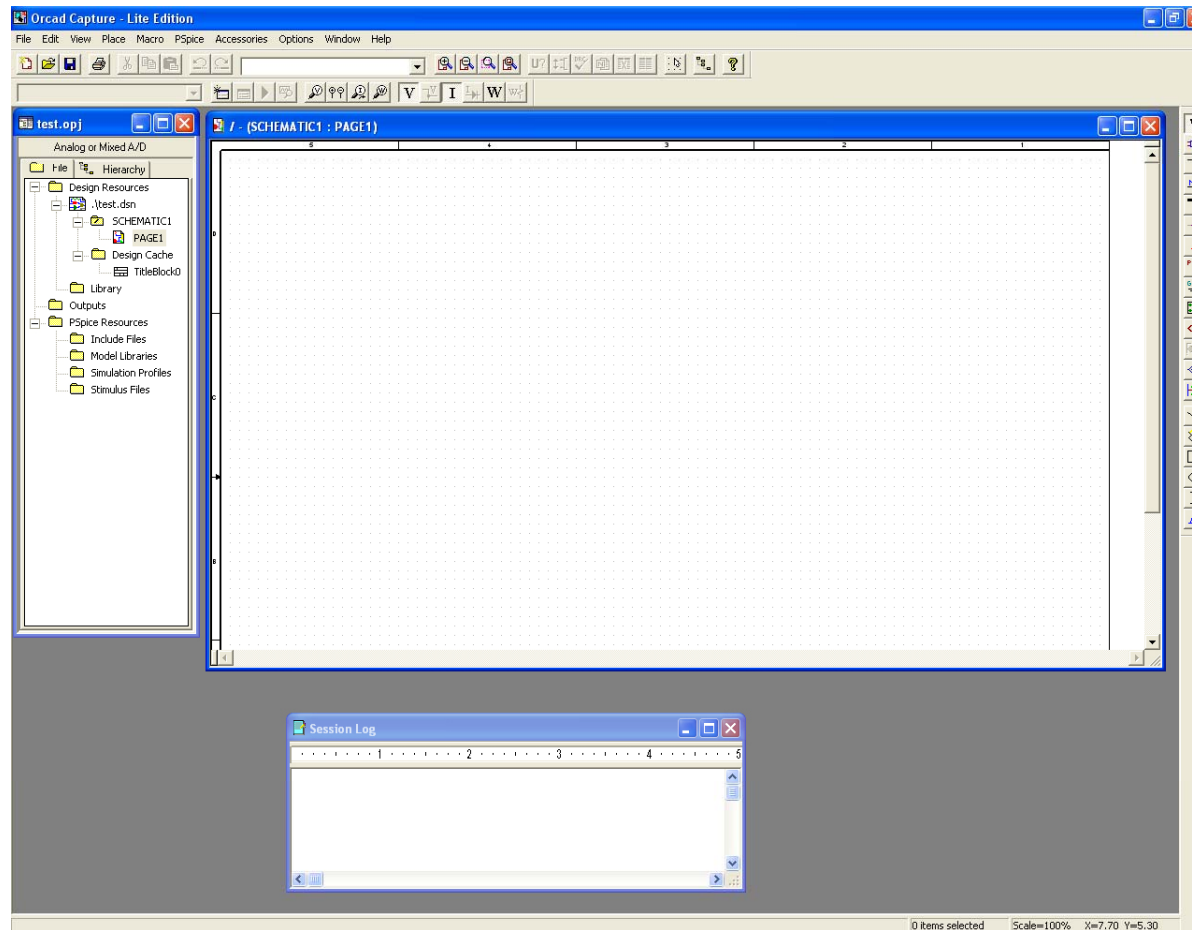
Circuit Creation with Capture

- 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

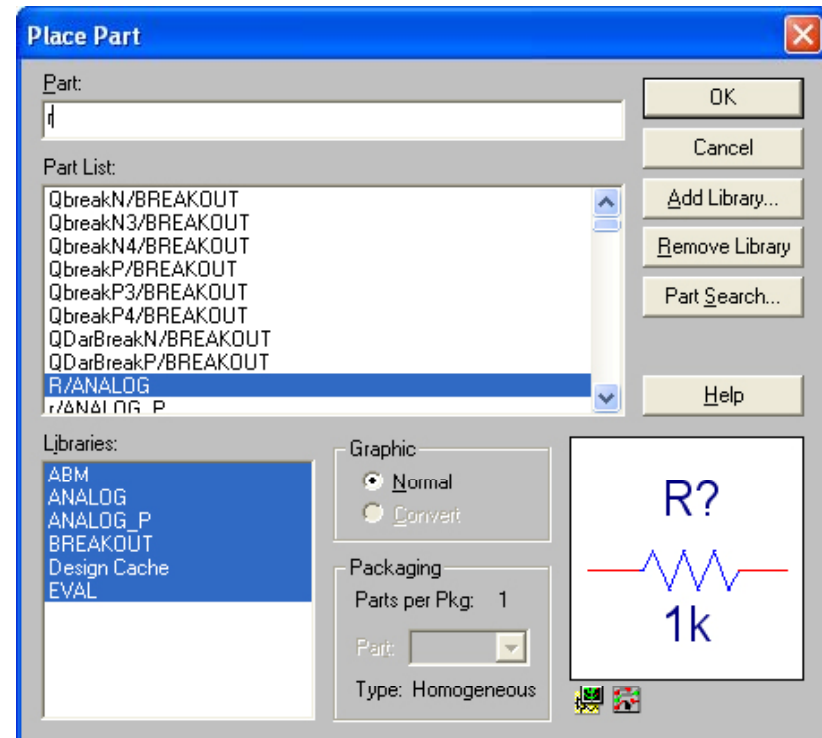


The Capture window



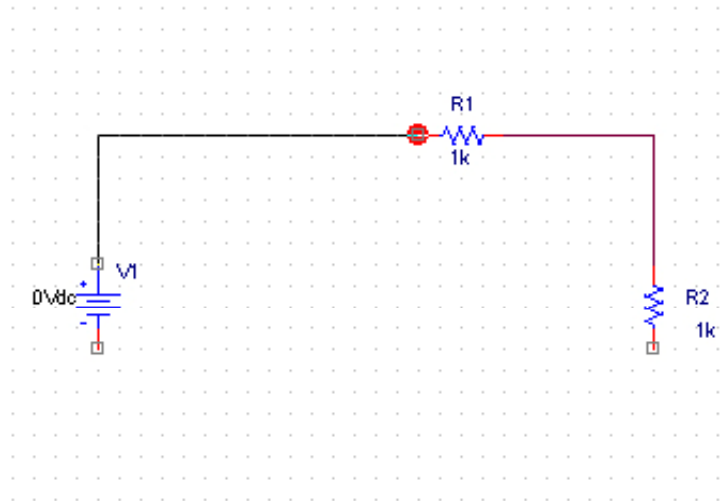
Placing the circuit parts

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



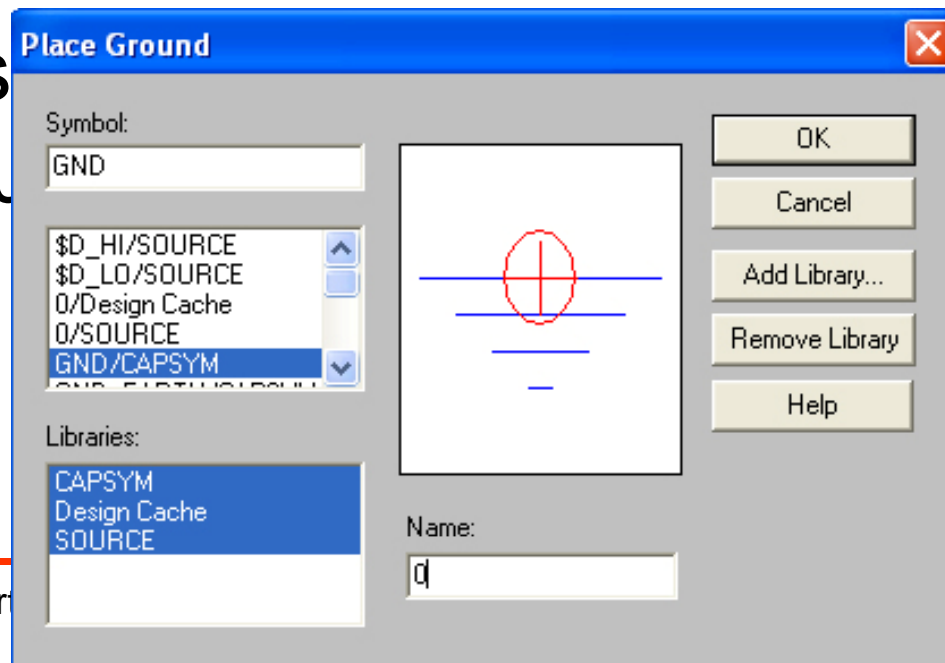
Connecting the circuit

- 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 insert a ground symbol → ground



place



AUB Depart

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)



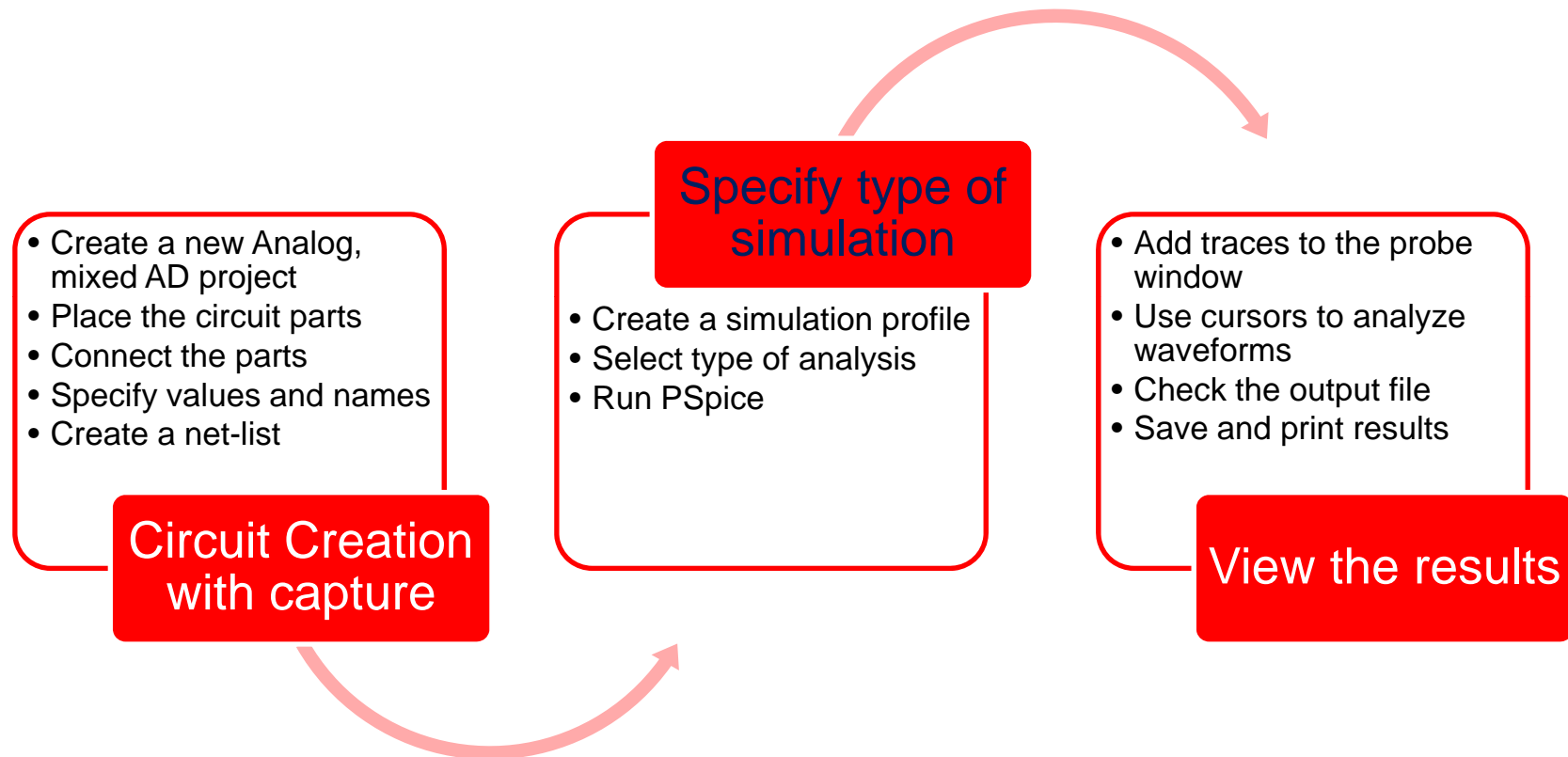
Creating a netlist

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

The screenshot displays the PSpice software interface. On the left, the 'File Hierarchy' pane shows the project structure, including 'Design Resources', 'Design Cache', 'Library', 'Outputs', and 'PSpice Resources'. The 'Outputs' folder is expanded, showing the file '.test-schematic1.net'. The main workspace shows a schematic diagram of a circuit with a 0Vdc voltage source (V1), a 1k resistor (R1), and another 1k resistor (R2) connected in a loop. A small window titled 'test-schematic1.net' is open, displaying the netlist content:

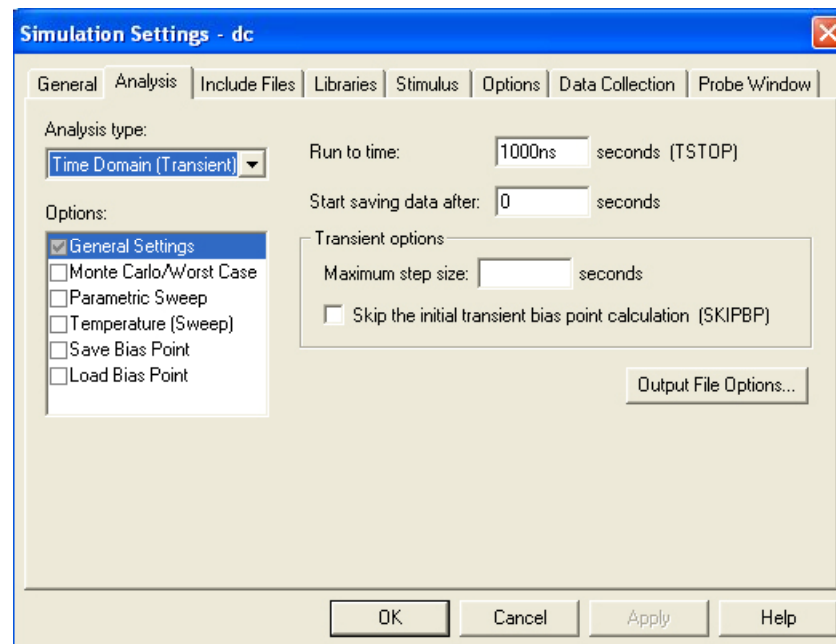
```
1: * source TEST
2: R_R1      NO0373 NO0113 1k
3: R_R2      0 NO0113 1k
4: V_V1      NO0373 0 0Vdc
5:
```

Steps involved



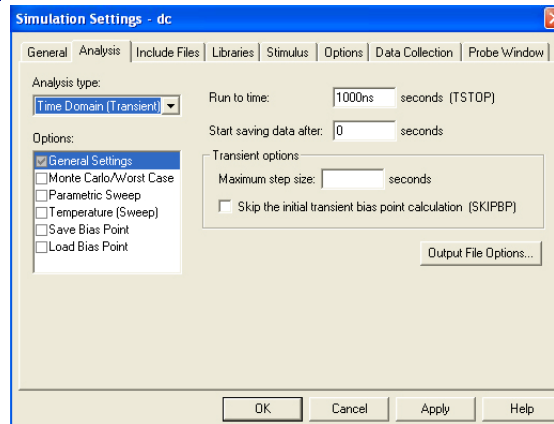
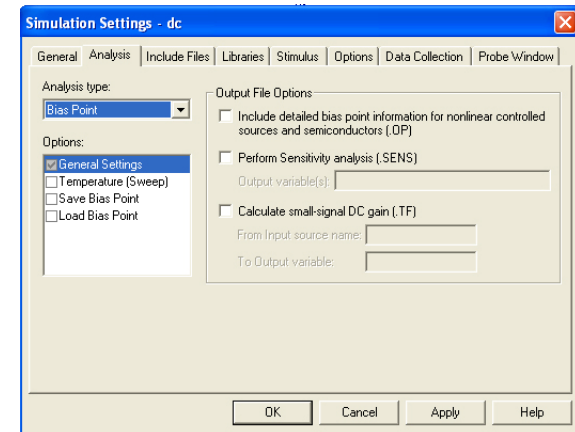
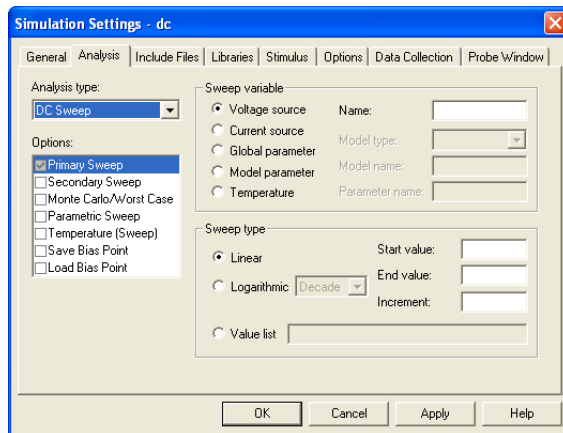
Specifying the type of simulation

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



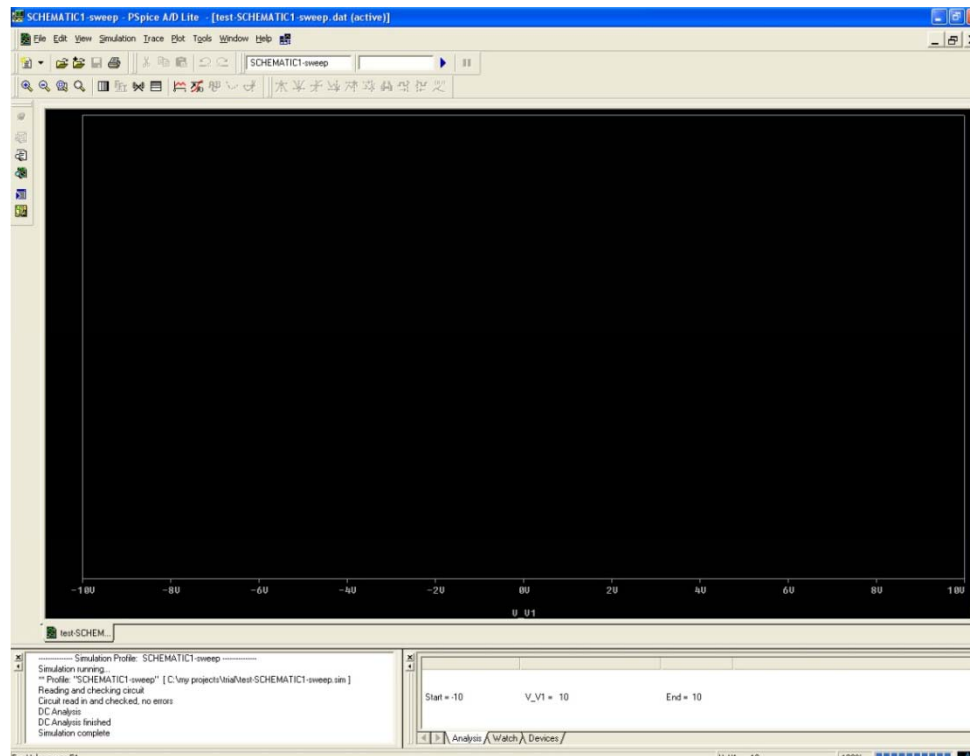
Specifying the type of simulation

- Selecting the type of Analysis

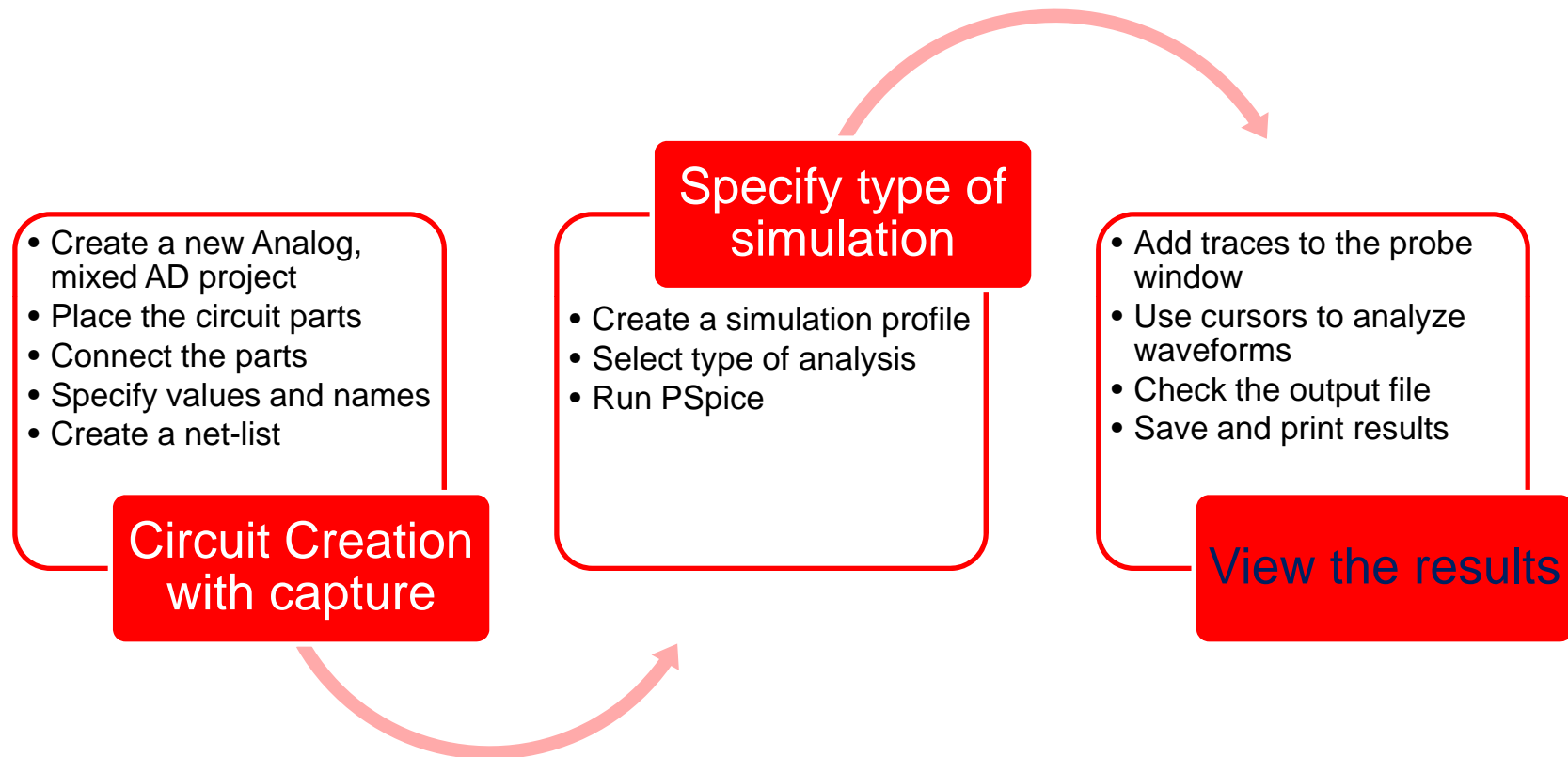


Specifying the type of simulation

- Running the simulation
 - Select PSpice → Run

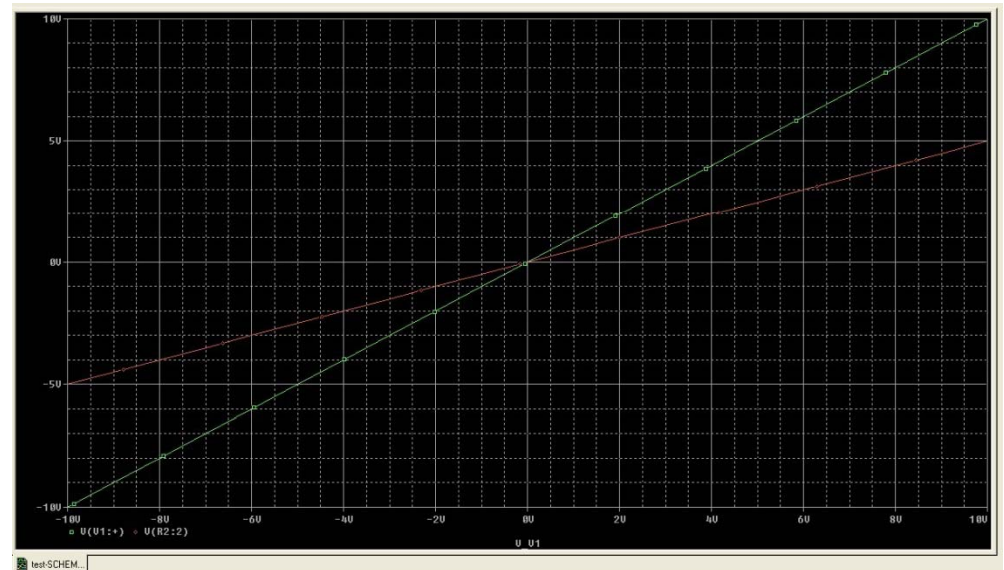
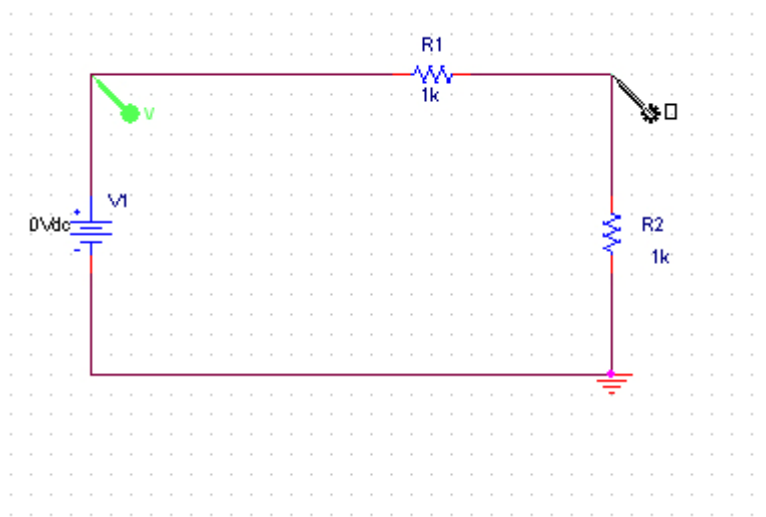


Steps involved



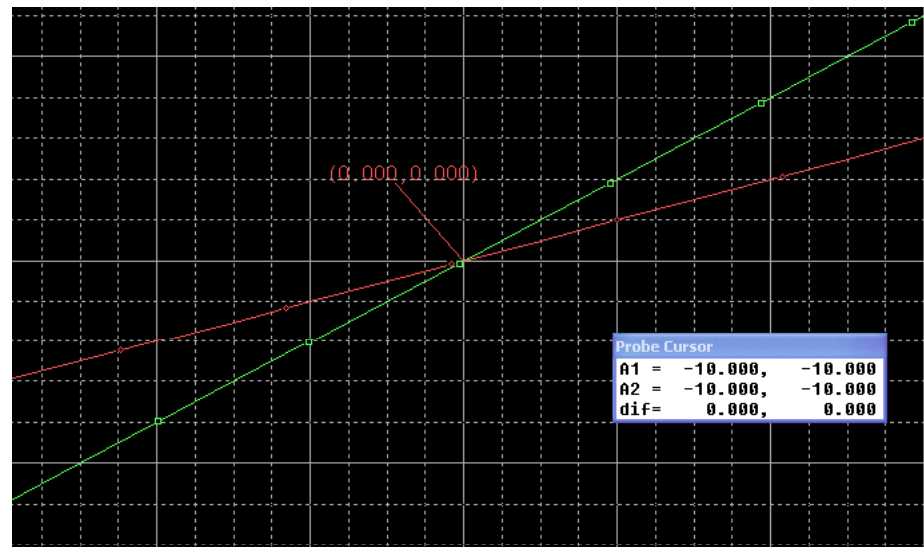
Viewing the results

- 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

