PSPICE Tutorial

Group Members

Piyush Dhamdhere – 16BCE0102

Himadri Biswas – 16BCE0682

Om Ashish Mishra – 16BCE0789

Aditya Dhall – 16BCE2234

# Circuit Components available

- 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

Things to remember before starting

▶ PSpice is not case sensitive

▶ All element names must be unique

▶ There must be a node designated "0" (Zero). This is the reference node against which all voltages are calculated.

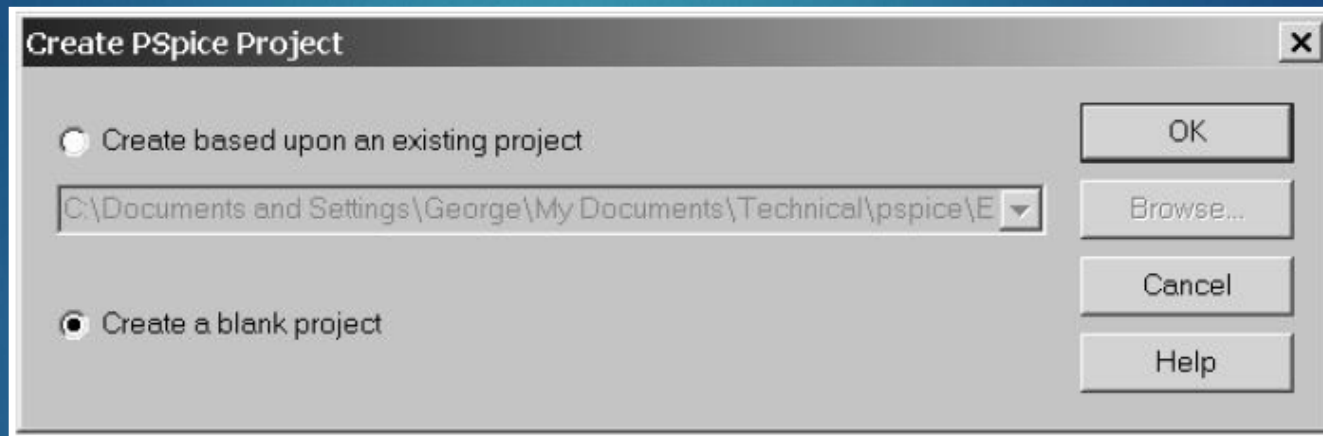▶ If any change is made in circuit make sure you create netlist again before simulating it.

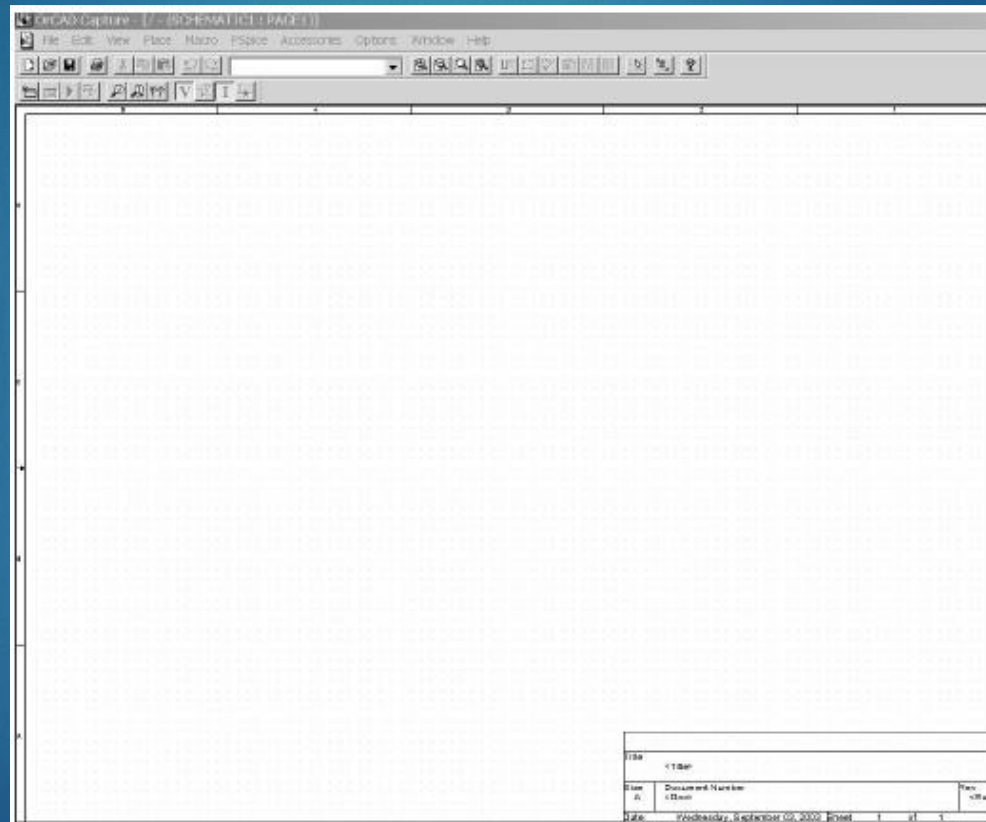# Getting Started

- File → New → Project → Analog or Mixed A/D
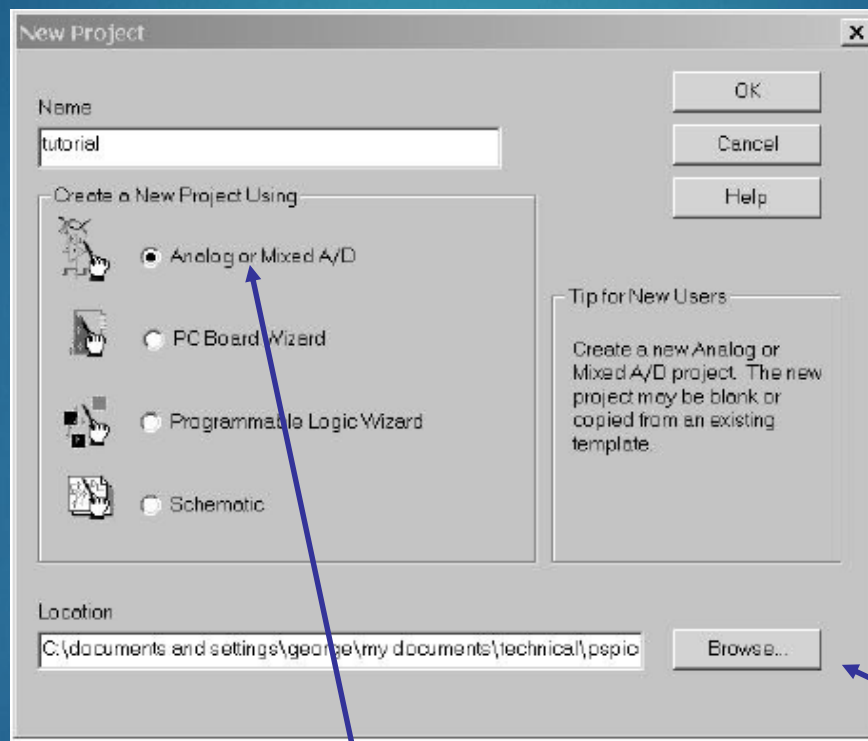
Make sure to choose Analog or Mixed A/D and proper location for project
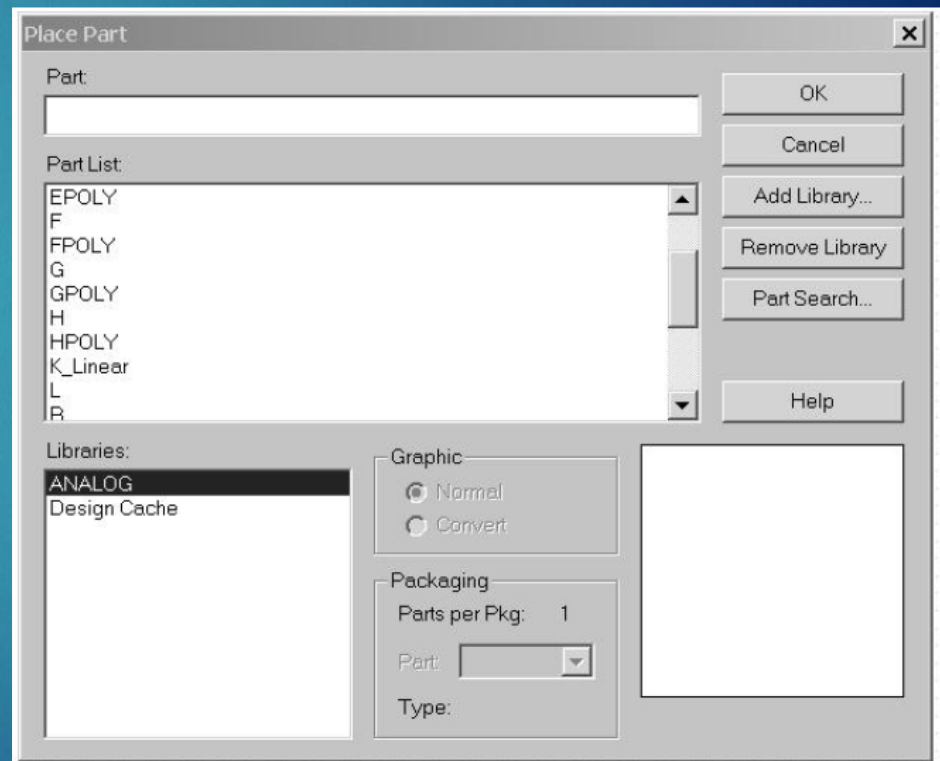
► Next comes following window

Make sure you select "Create a blank project"

This is how a typical "Schematic" Window will look like

► Now start placing components and connecting them (this part is quite similar to what you have done in last semester's lab)

Add a library as per the requirement for eg. here analog library has been added
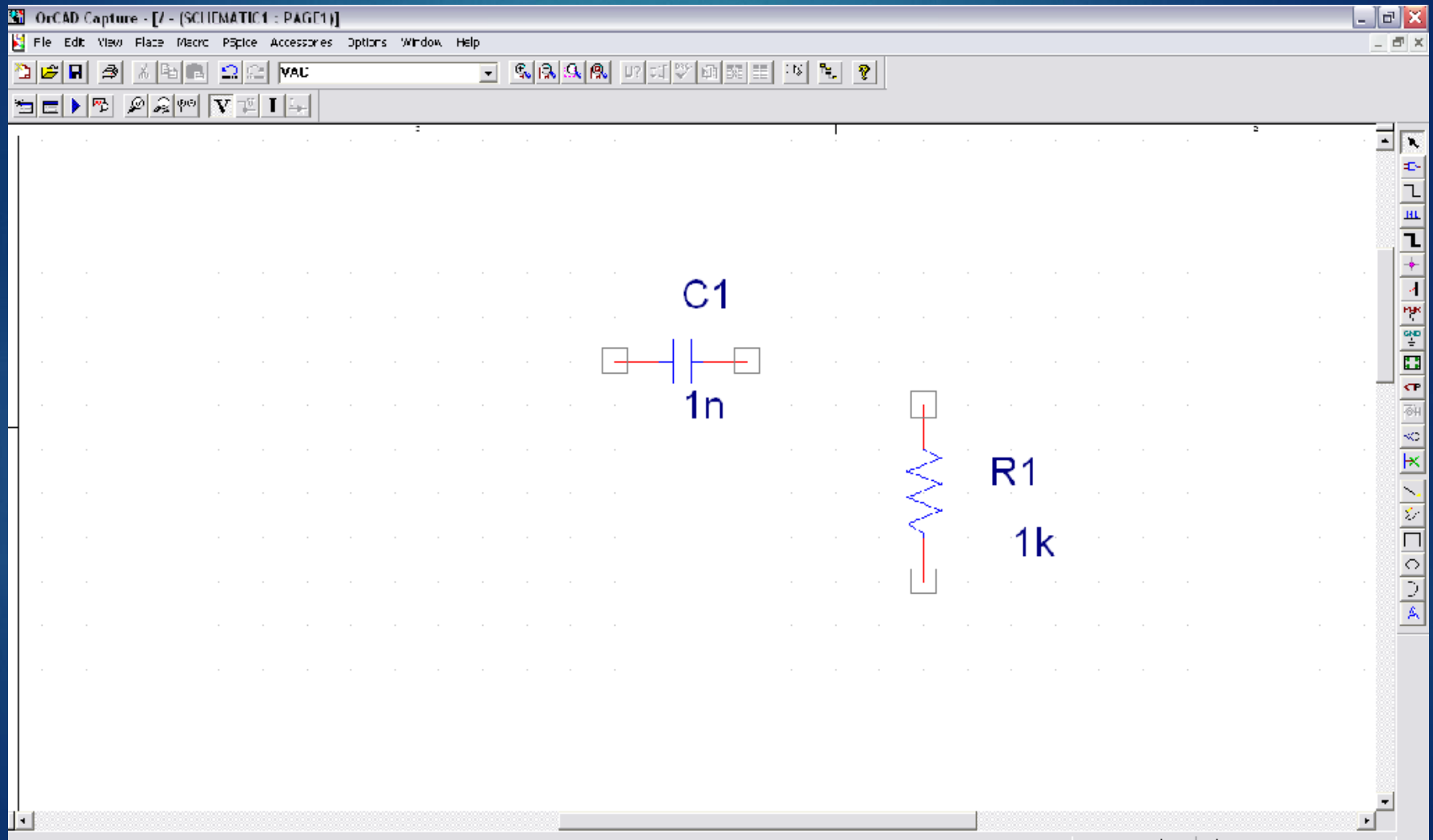
- ▶ After completing the circuit save it and run it.
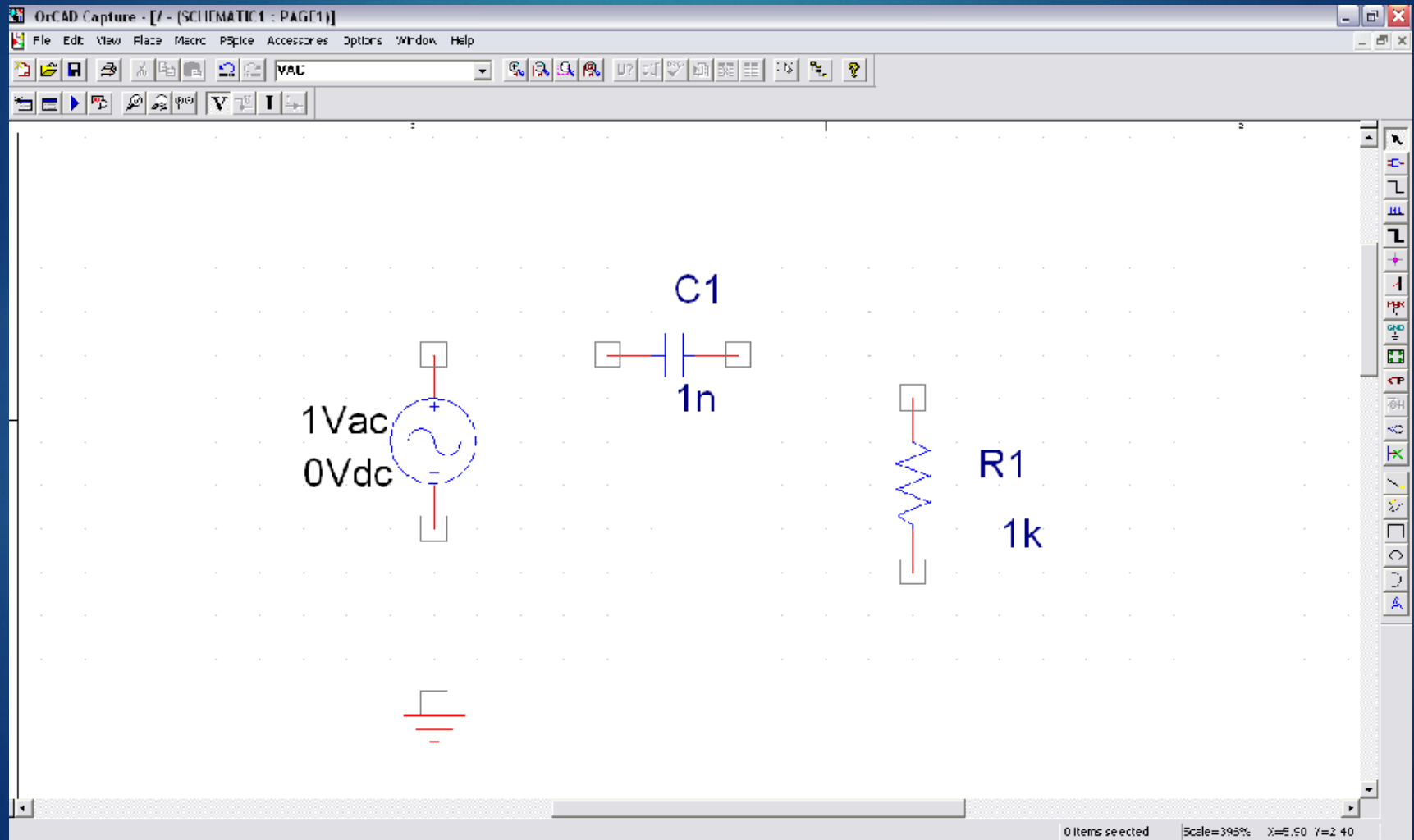- ▶ Plot appropriate waveform by probing on circuit.

Example

- Taking example of a circuit
- First we will follow initial steps of creating a new project
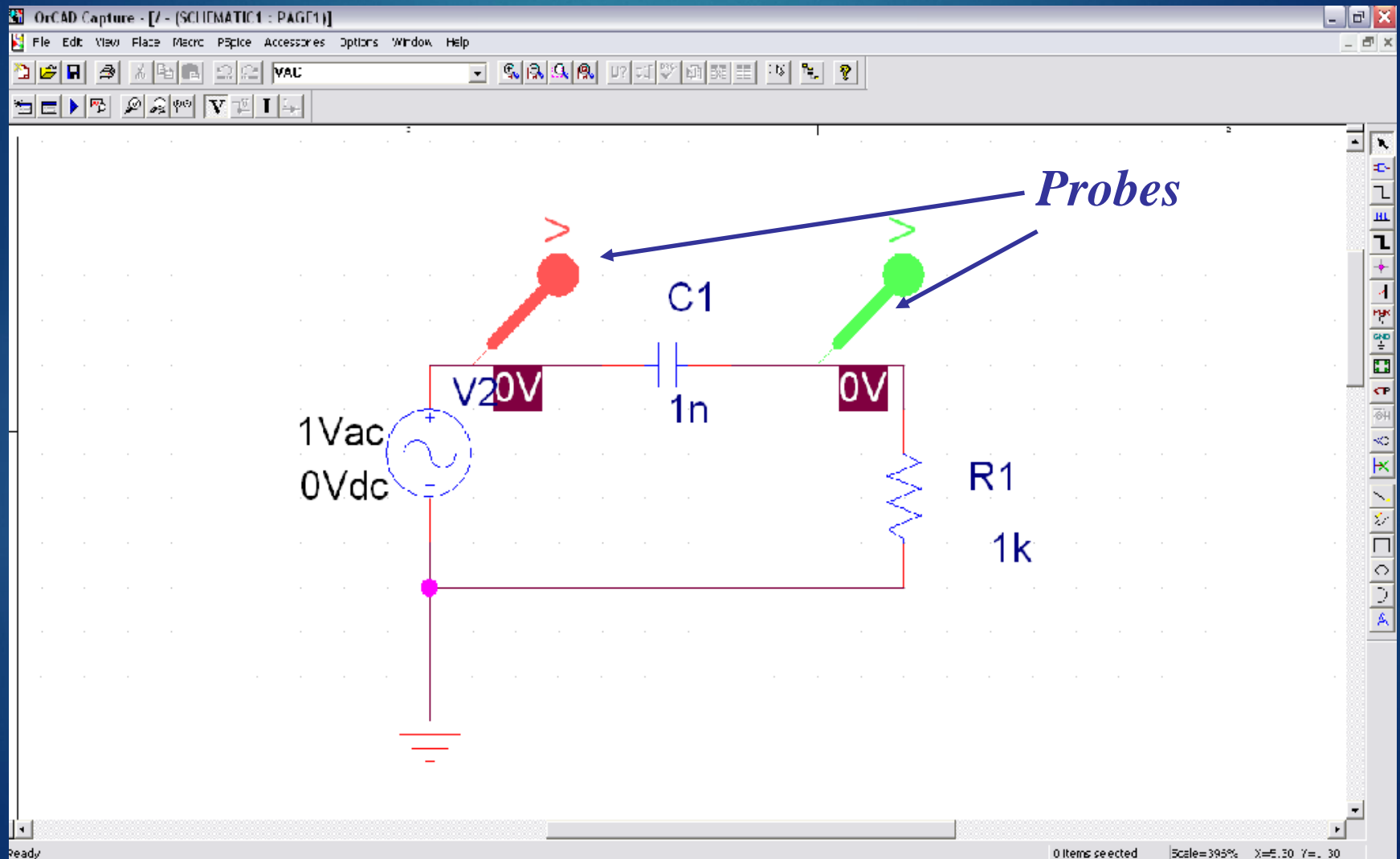- Now placing the components.

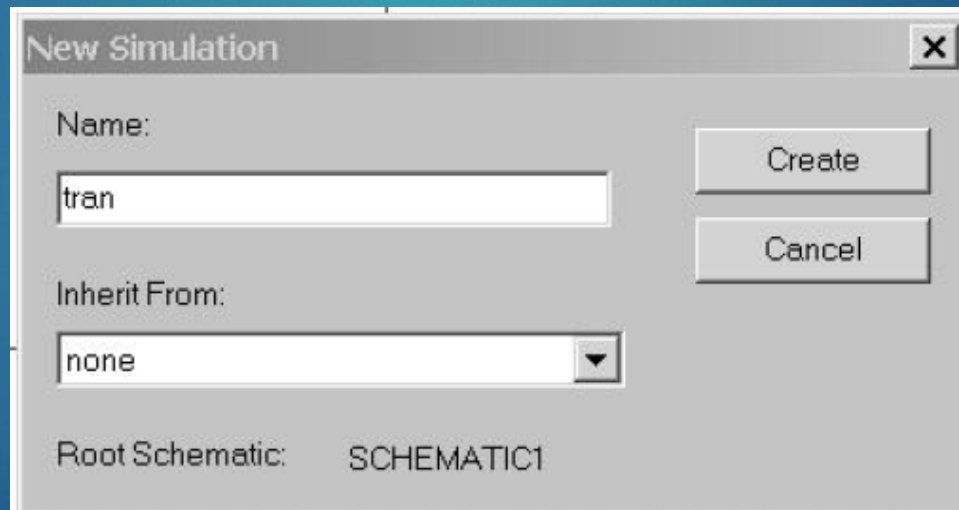# Place the components
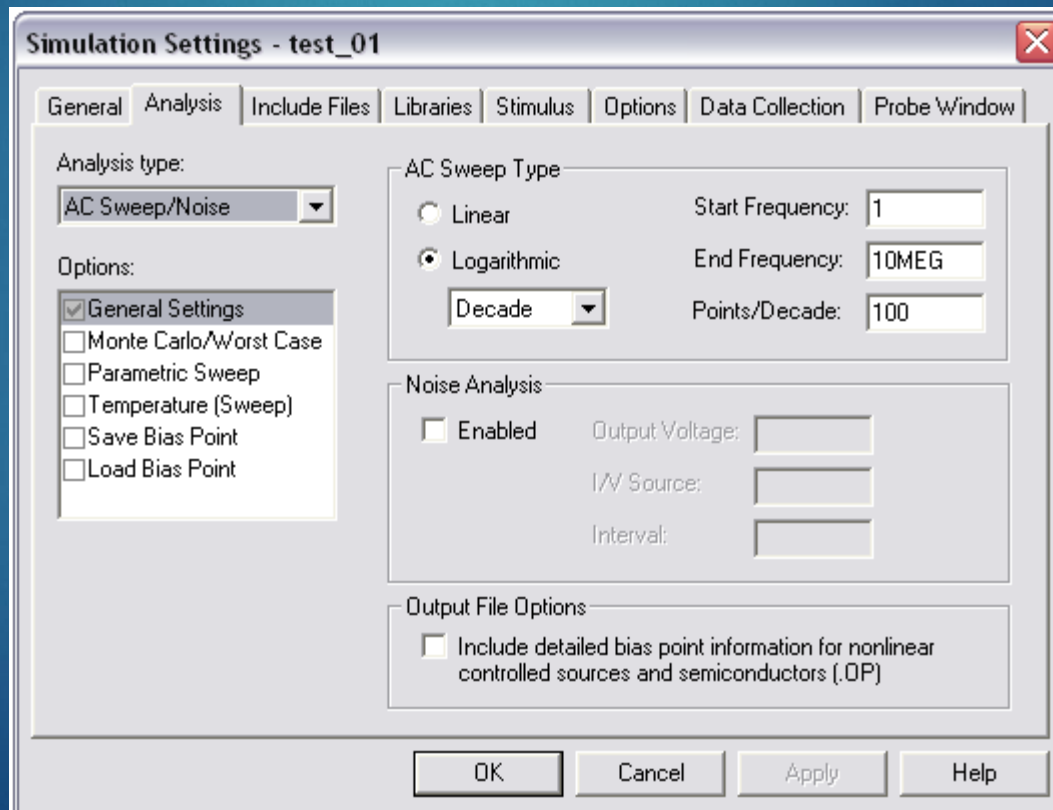
Place the source and ground

Add Probe

Creating new simulation profile

▶ Go to PSPICE → New Simulation Profile
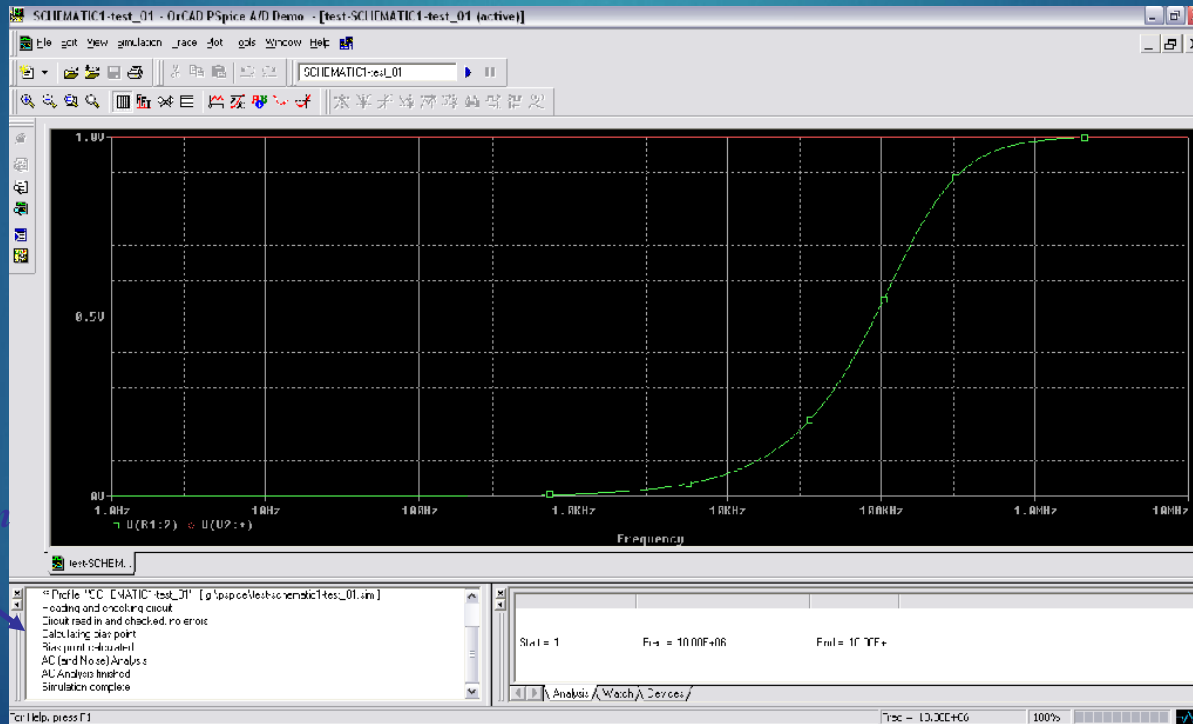
▶ Give a name to profile and create

AC Analysis

- ▶ We can choose any of the analysis and options
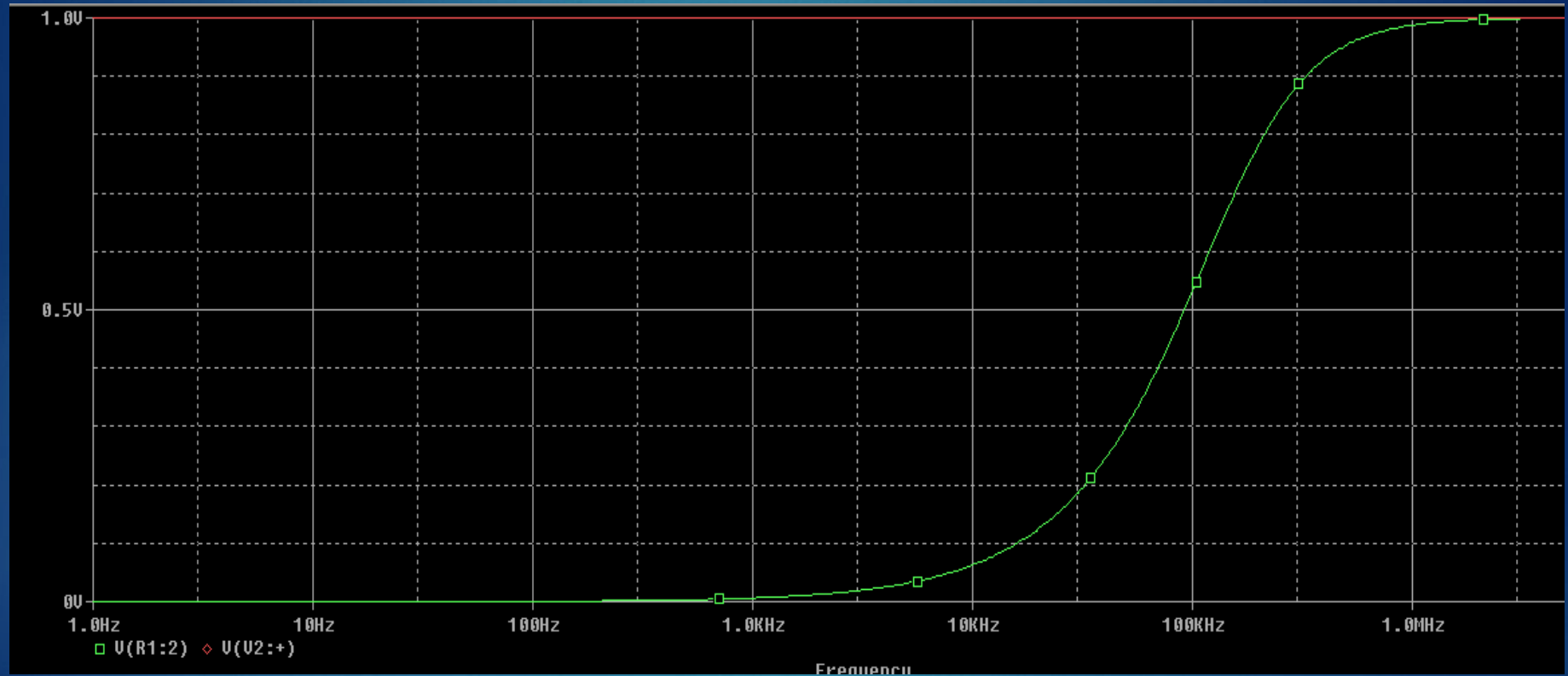- ▶ Here AC analysis is done to see operation of filter

Run

▶ Go to PSPICE → Run

▶ Another waveform window will come up

See this window to keep track of current simulation

Result

THANK YOU