Introduction to PSpice

Dr. Wael Salah

Introduction to PSpice

Outline

- What is PSpice?
- Types of Analysis
- Main Menus, Tools and Modules
- Steps of Performing Analysis
- Get Stated Step By Step
- A Step by Step Tutorial Examples
- Common errors

What is PSpice?

- PC version of the older SPICE software (Simulation Program with Integrated Circuit Emphasis)
- SPICE has been the standard method of device-level IC circuit simulation since the early 1970s
- Performs **non**linear DC, nonlinear **transient**, and linear ac circuit solutions. Also **temperature** analysis and Introduction analysis.
- Other circuit simulation software include: MultiSim, MATLAB, and others …..

Introduction

- SPICE (***Simulation Program for Integrated Circuits Emphasis)*** is a general purpose analog circuit simulator that is used to **verify** circuit designs and to **predict** the circuit behavior.
- PSpice is a PC version of SPICE and HSpice is a version that runs on workstations and larger computers**.**
- PSpice has **analog** and **digital** libraries of standard components (such as NAND, NOR, flip-flops, and other digital gates, op amps, etc) which makes it a useful tool for a wide range of analog and digital applications.

Types of Analysis

- Non-linear DC analysis
- Non-linear transient analysis
- Linear AC Analysis
- Noise analysis
- Sensitivity analysis
- Distortion analysis
- Fourier analysis
- Monte Carlo Analysis

Types of Analysis Available

Basic Single-Run Analyses:

- **DC Bias Point** Details
- **DC Sweep**:
  e.g for Transfer Function
- **AC Sweep** and Noise:
  e.g. for Frequency Response
- **Transient and Fourier**: Time Domain Analysis

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
- ……………… and others

File structure PSPICE (Capture)

- Input file for PSpice that has the file type "**.CIR**"
- When a circuit is simulated "**.OUT**" is generated
- "*.**DAT**" file is by default a binary file to save the data
- "*.**LIB**" files where the details of complex parts are saved
- Additional files
  - *.**SCH** (the schematic data, itself)
  - *.**ALS** (alias files)
  - *.**NET** (network connection files)
  - ………….. and others ….

Note: Save your design in a new Folder

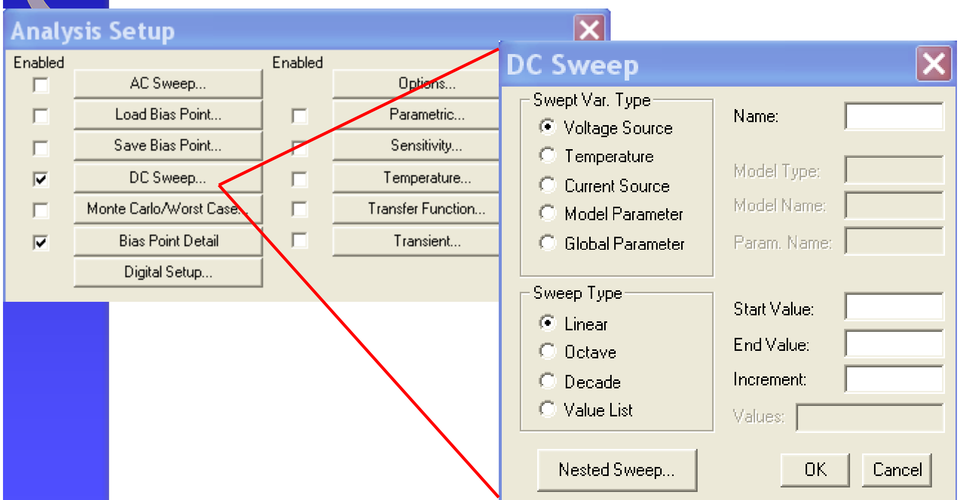
Libraries

- These are files that contain reusable part data.
- They can be custom or default ones
- Some of the libraries available in the Evaluation version of PSPICE are:
    - *abm.slb for special functions like square root and multipliers*
    - *analog.slb for analog components like resistors and capacitors*
    - *breakout.slb for pots*
    - *connect.slb, .plb for connectors*
    - *eval.slb, .plb for semiconductor, digital devices, switches*
    - *port.slb for grounds, high/low digital ports*
    - *special.slb for ammeters, viewpoints*
    - *source.slb for various analog and digital sources*
    - *……….. and others*

Parts

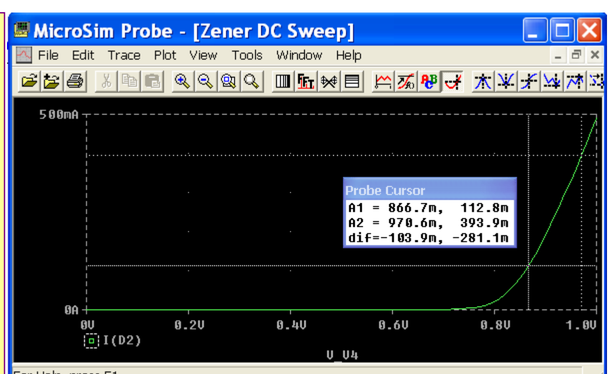
- **Part** is a basic building block which may represent:
    - One or more physical element
    - Function
    - Simulation model
    - When a part is placed for first time; its entry is added into "**Design Cache**"

Analysis Setup (ver. , v9.1)

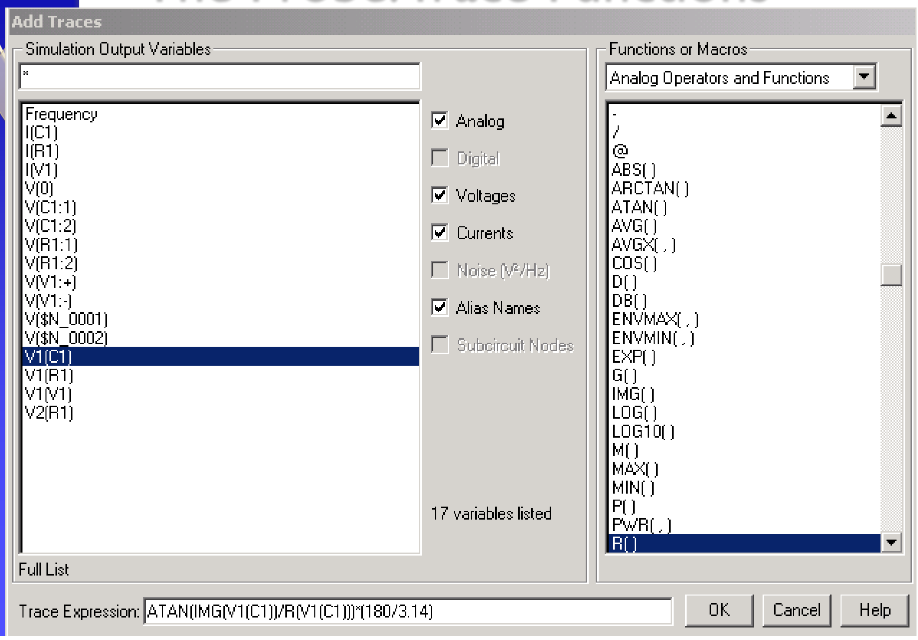


The Probe (v 9.1)

- Multi-trace plots versus time, frequency, voltage, temperature, etc.
- Two independent Cursors
- Trace data can be exported to Excel
- Trace function can be user-defined



The Probe: Trace Functions



Steps of Performing Analysis

1. **Prepare the schematic:**
   - Get and place parts
   - Set values and attributes (symbols: R1, Z2, …)
   - Wire-up the circuit (Remember grounding)
   - Save into a new **Folder** (Save **full** project)
2. **Simulate and Debug**
3. **Correct error (if any)**
4. **Analyze, capture waveforms, save results**

Things to remember before starting

- PSpice is not case sensitive
- All element names must be unique [Annotate]
- 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 **save** again and **Run** simulation again to get updated results.

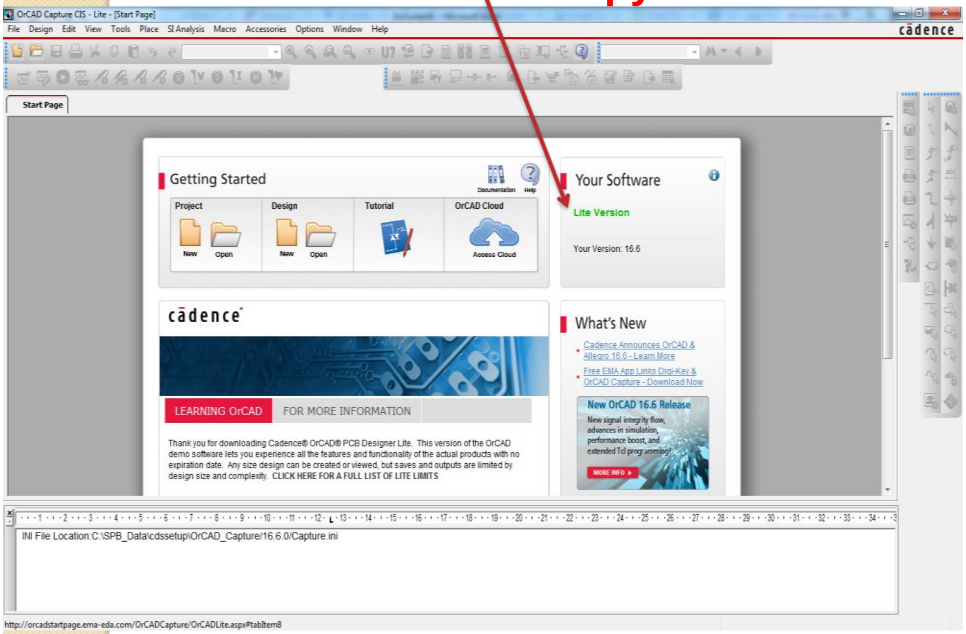
Step By Step
Get Stated

PSpice Step by Step v16.6 Lite

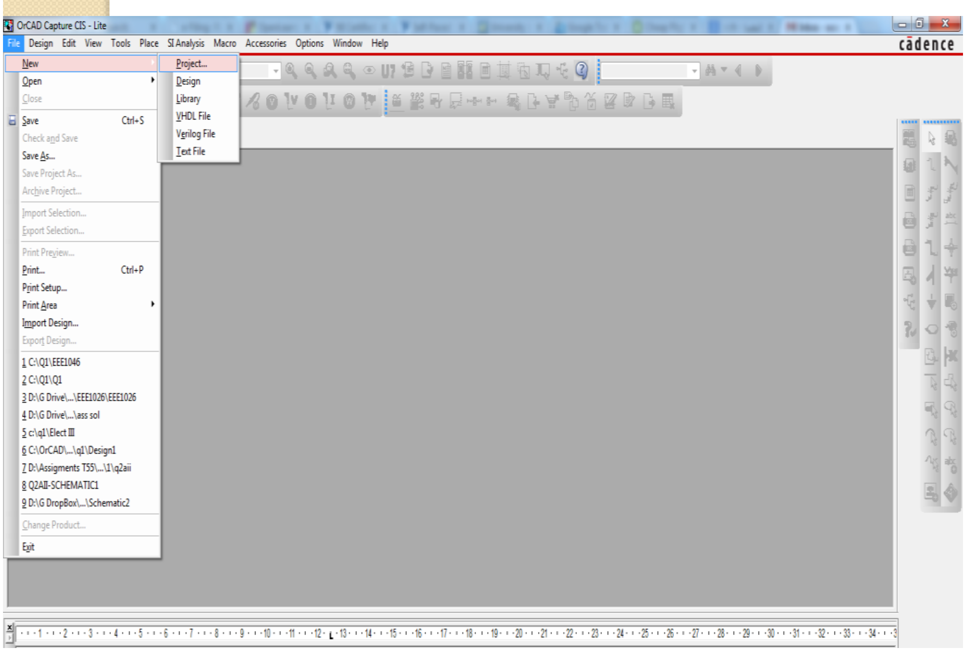
Create a New Project

Create a New Project

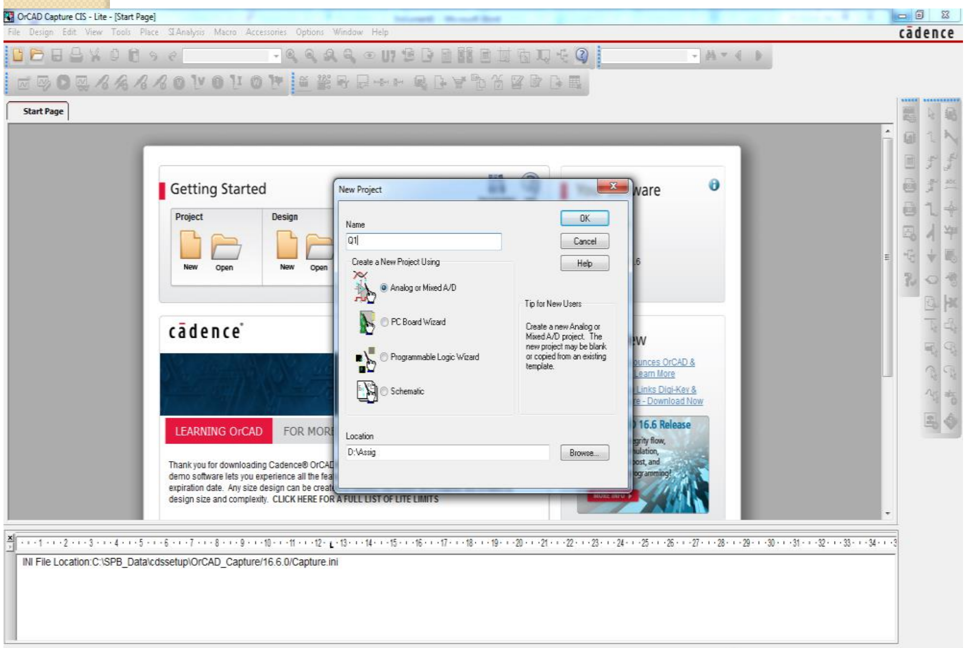
Free copy



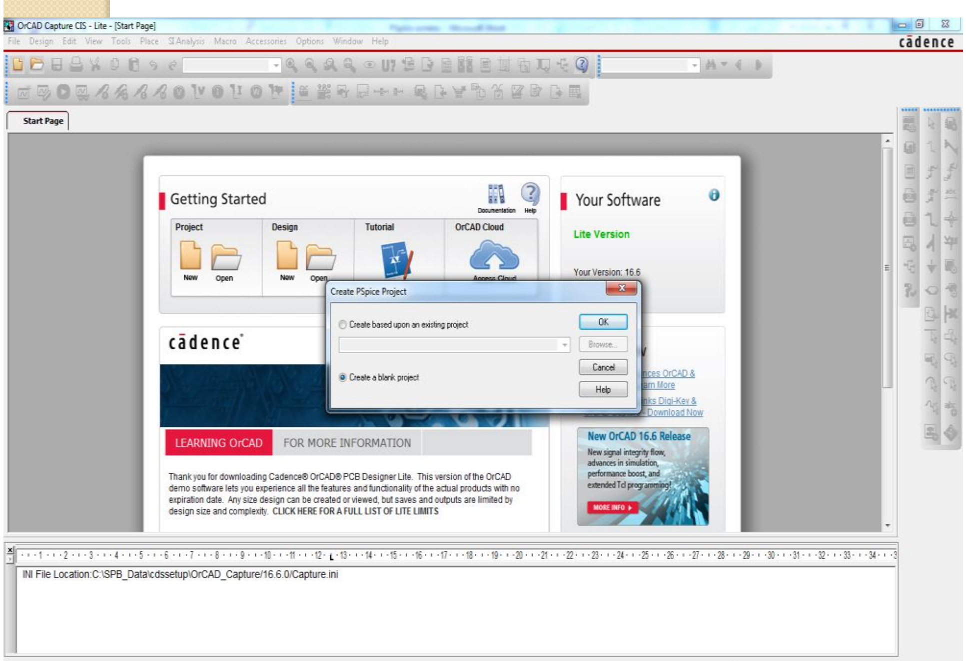
Create a New Project



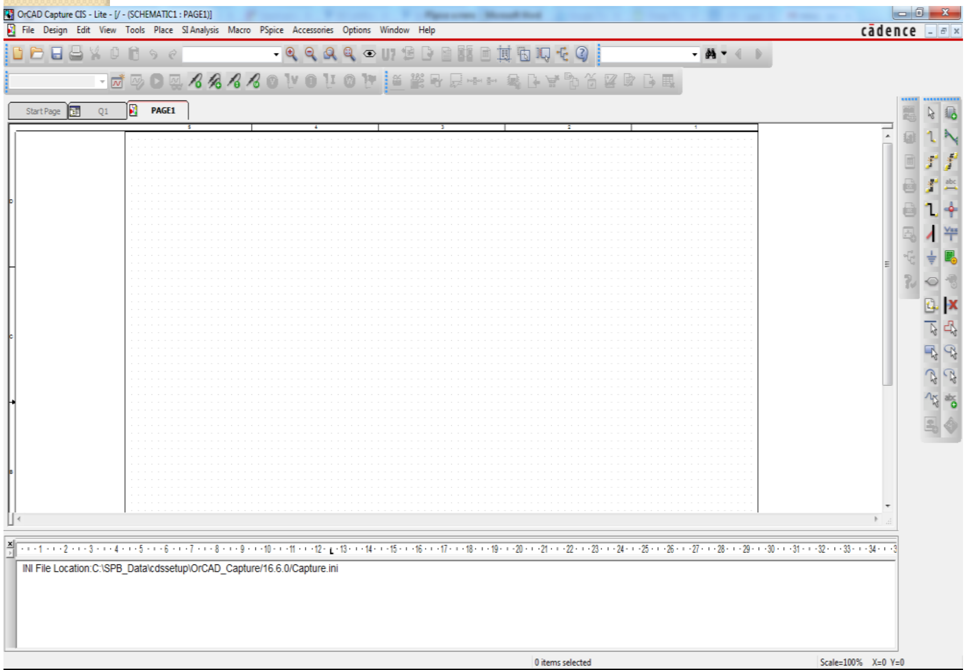
Create a New Project



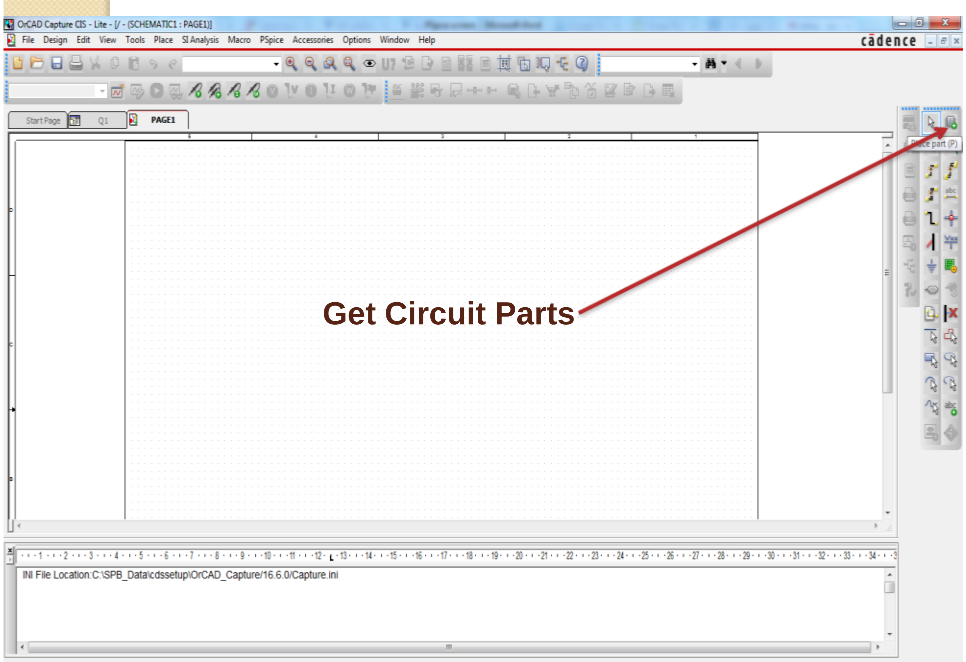
Create a New Project

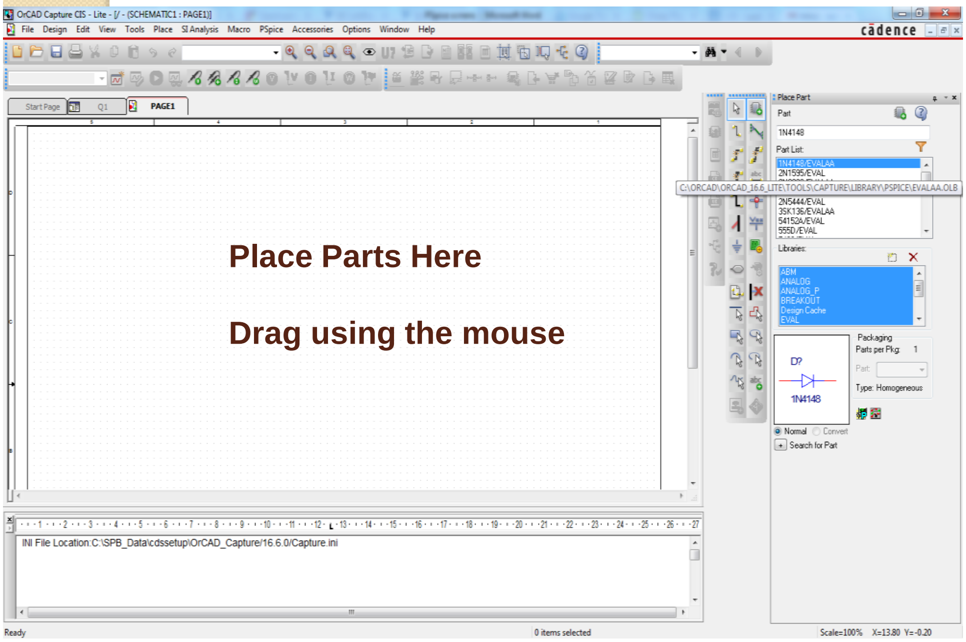


Create a New Page

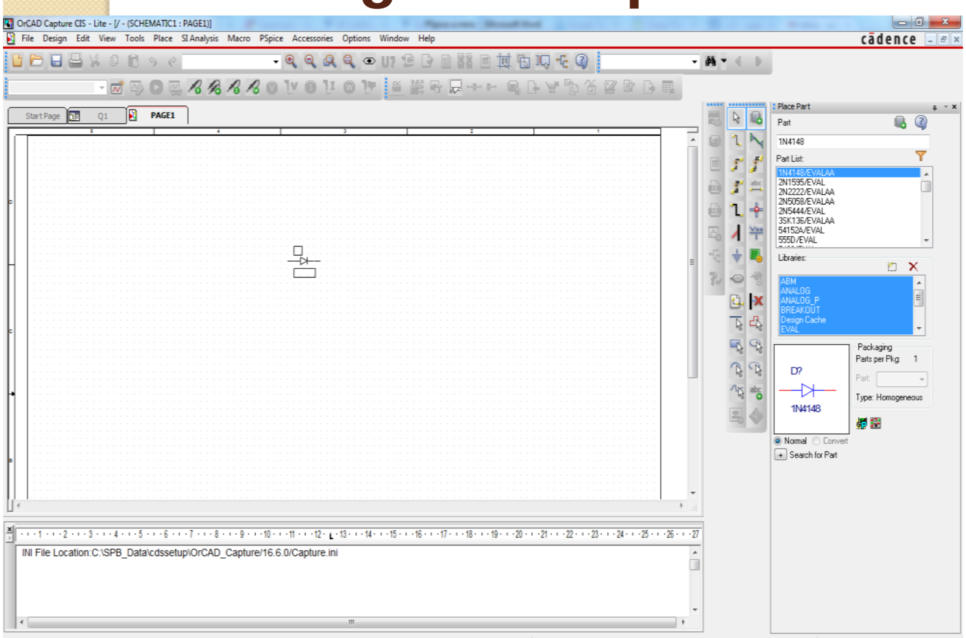


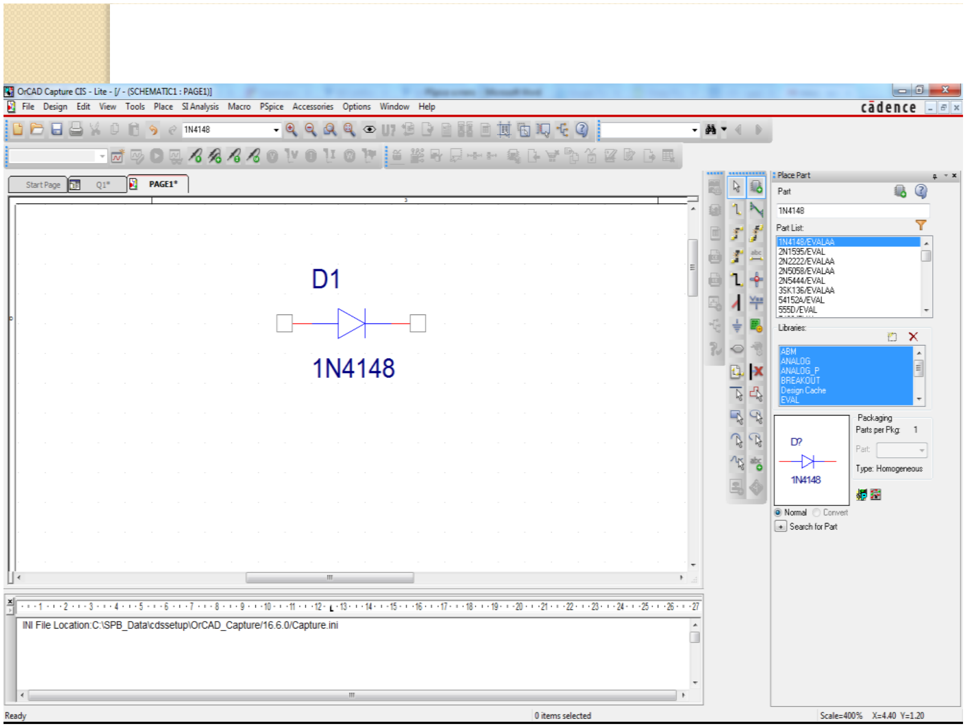
Create a New Project



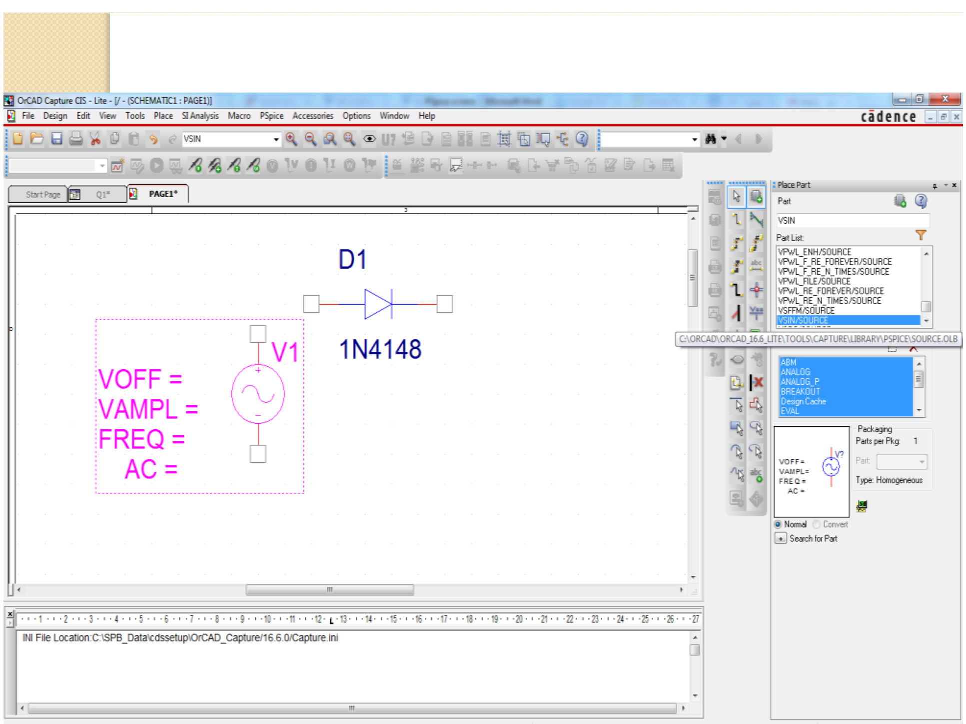


Design Example 1



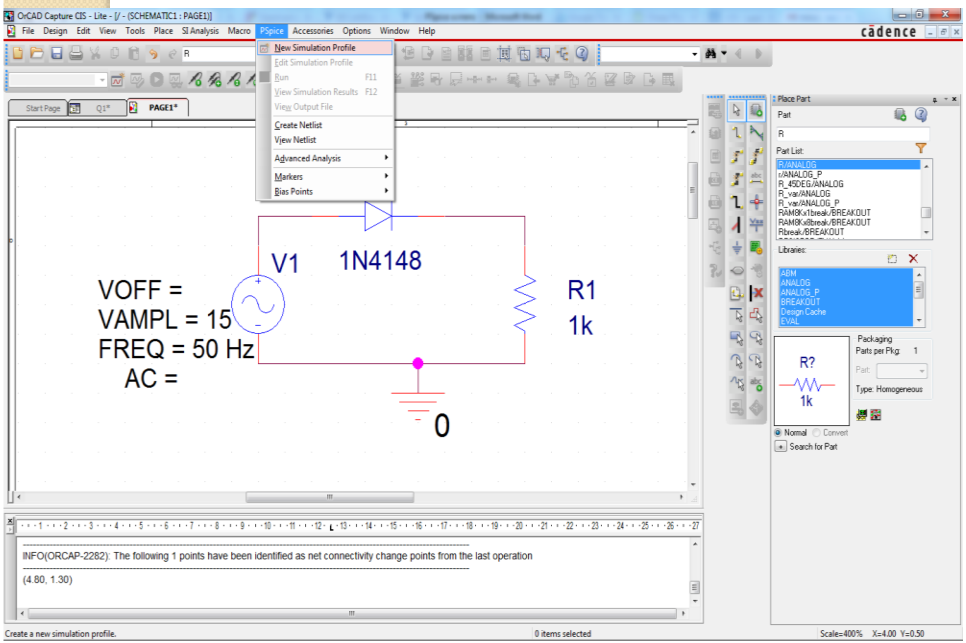


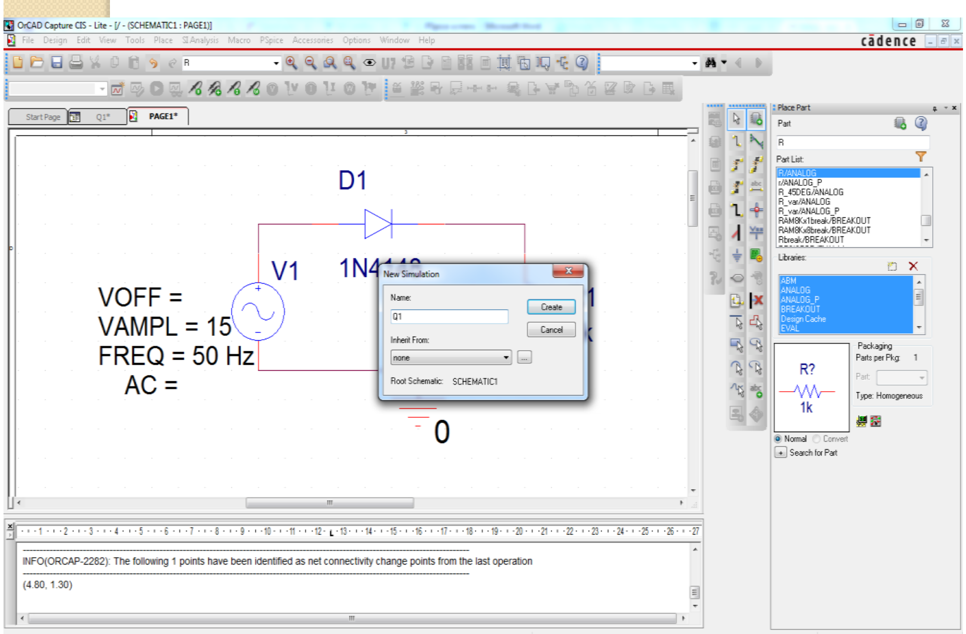
D1
1N4148
INI File Location:C:\SPB_Data\cdssetup\OrCAD_Capture/16.6.0/Capture.ini



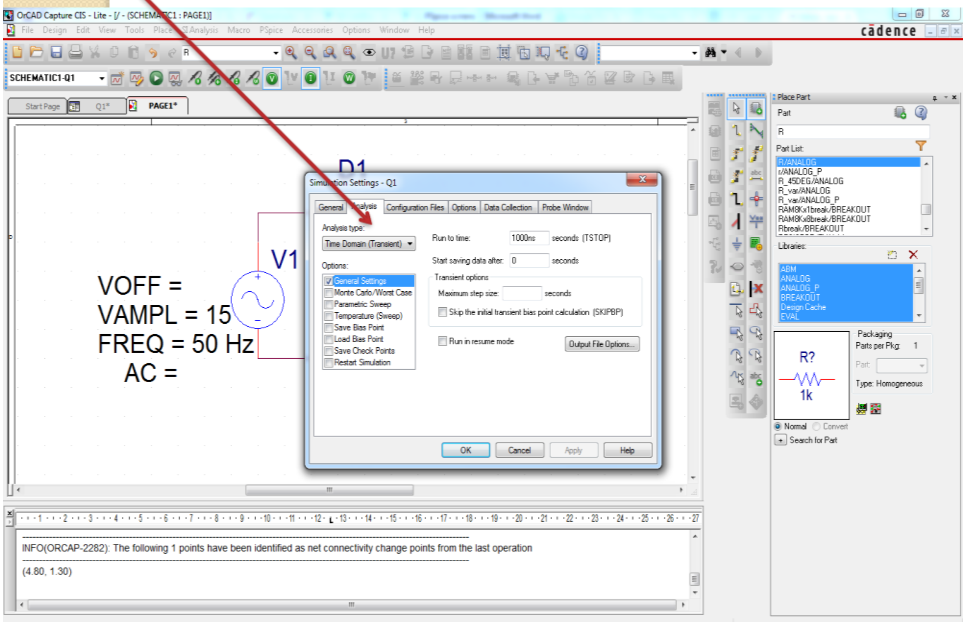
D1
1N4148
V1
VOFF =
VAMPL =
FREQ =
AC =
INI File Location:C:\SPB_Data\cdssetup\OrCAD_Capture/16.6.0/Capture.ini

Start Simulation

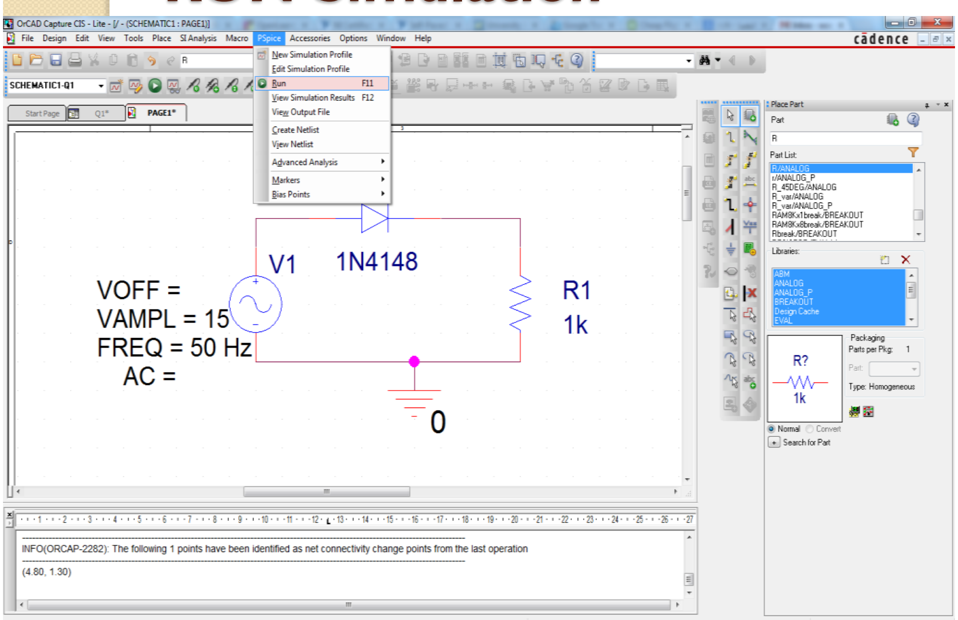




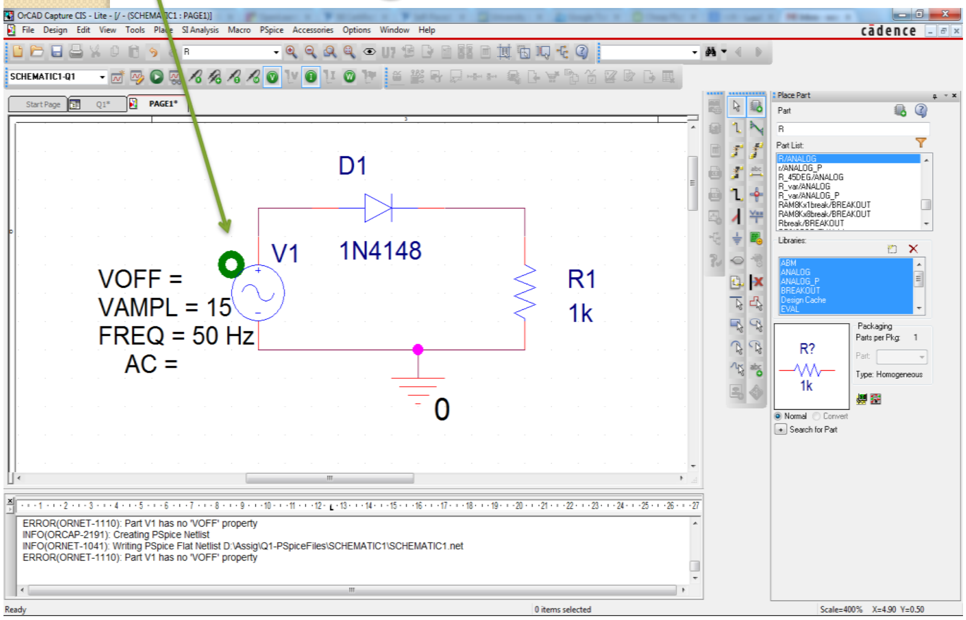
Select Analysis Type



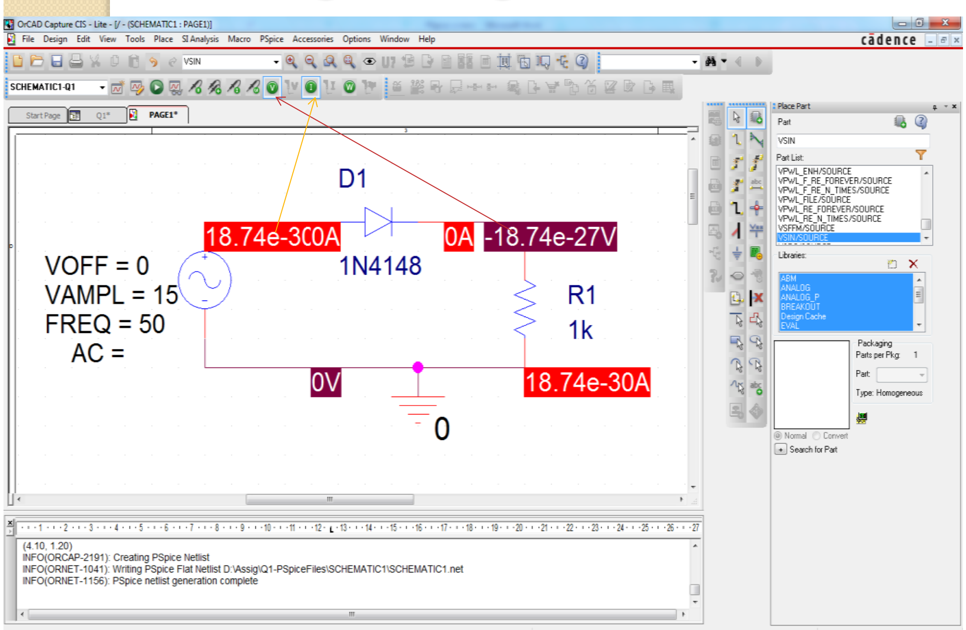
RUN Simulation

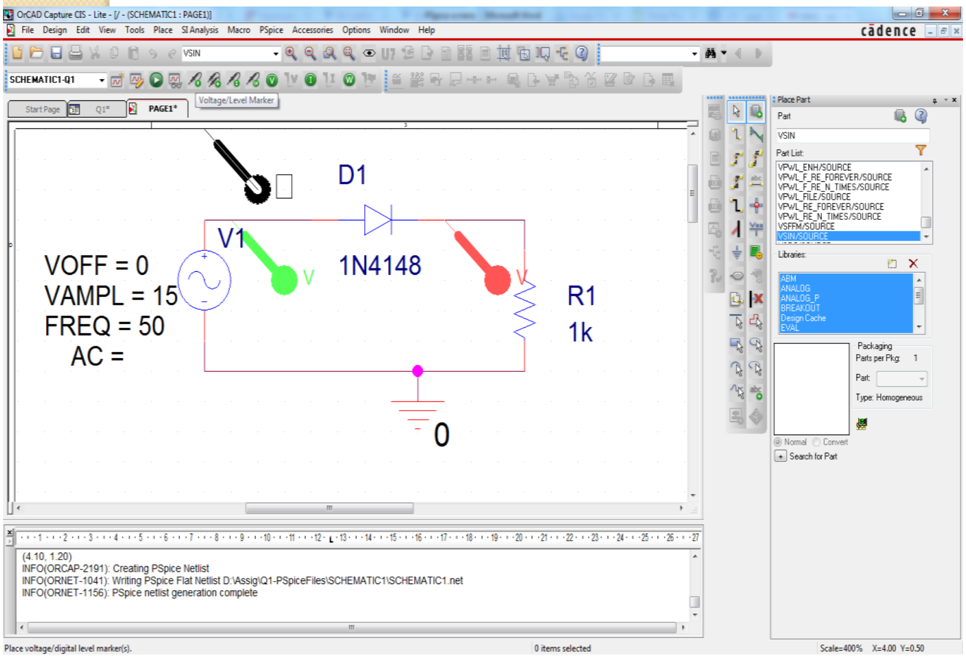


Error Sign

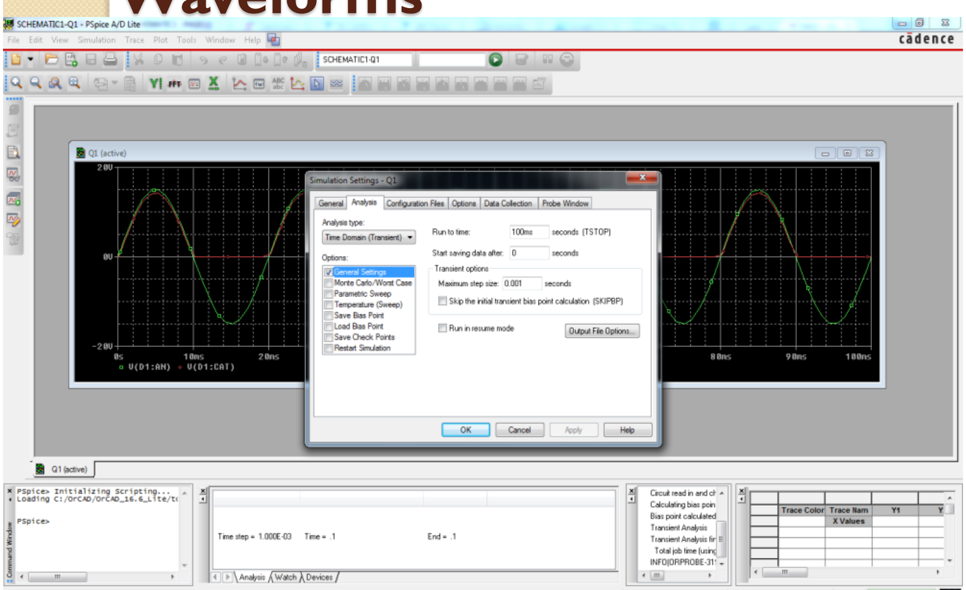


Showing Voltages and Currents





Showing Voltages and Currents Waveforms



Points to remember

- When creating a new project , **use Analog or Mixed A/D** set-up option.
- Watch the file location / directory structure.

AC-DC Converters (Rectifiers)

AC-DC converters (Rectifiers) can be classified as:

- Half wave rectifier
- Full wave rectifier

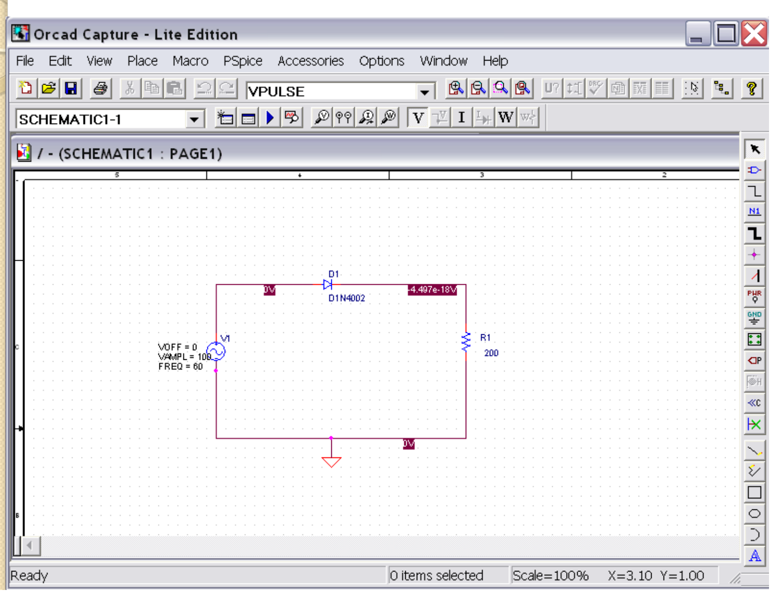
Half Wave Uncontrolled Rectifier

a) For Resistive Load

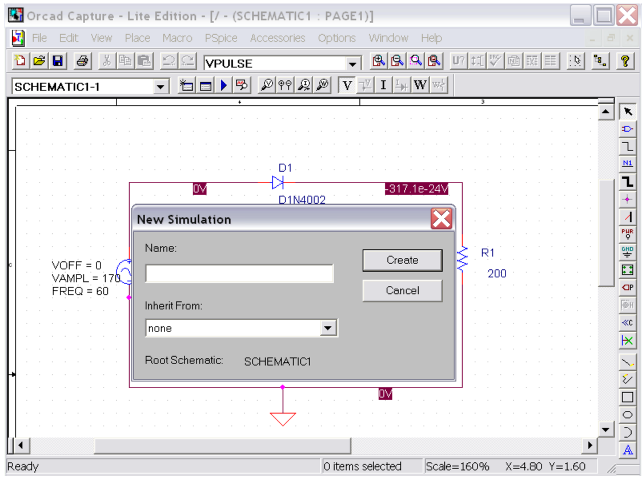
1) **Make the circuit in PSpice using steps mentioned earlier.**

Components used are:

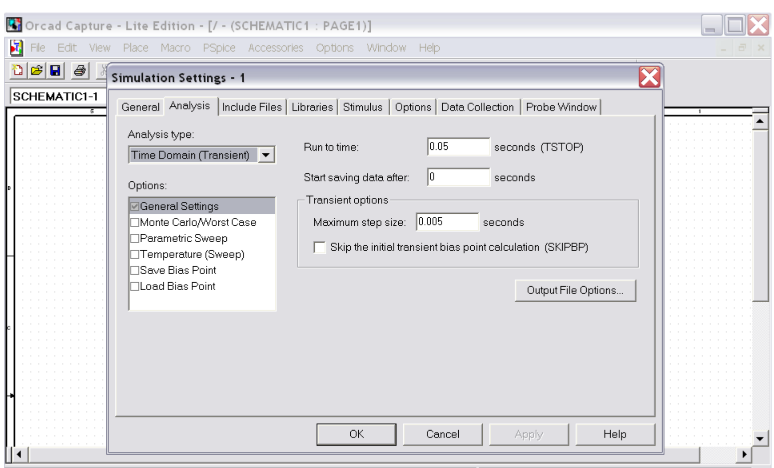
- VSin (voltage source)
- DIN4002 (diode)
- R (Resistance)
- GND_SIGNAL/CAPSYM.



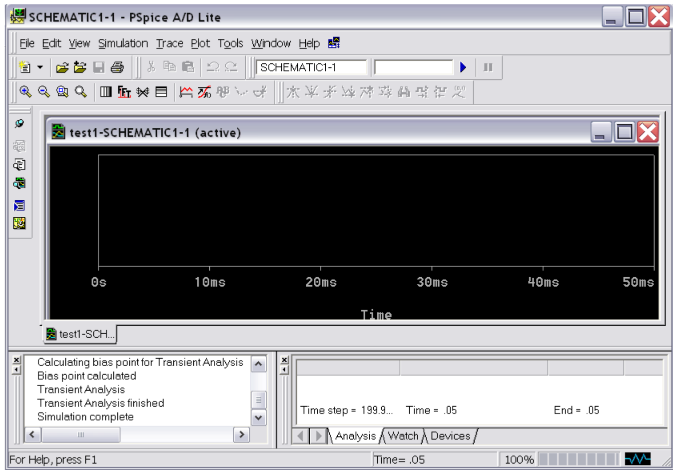
2) Make the **simulation profile** using “New Simulation Profile” command from “PSpice” menu.



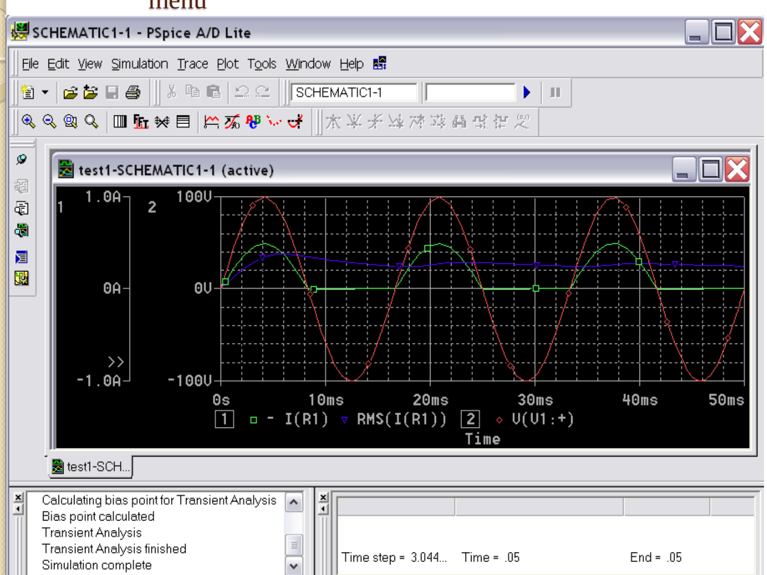
3) Go to “**Edit** Simulation Profile” in “PSpice” menu, simulation settings window will open. Go to “Analysis” and set the simulation parameters as shown below.



4) Run the simulation by using command “RUN” from “PSpice” menu.



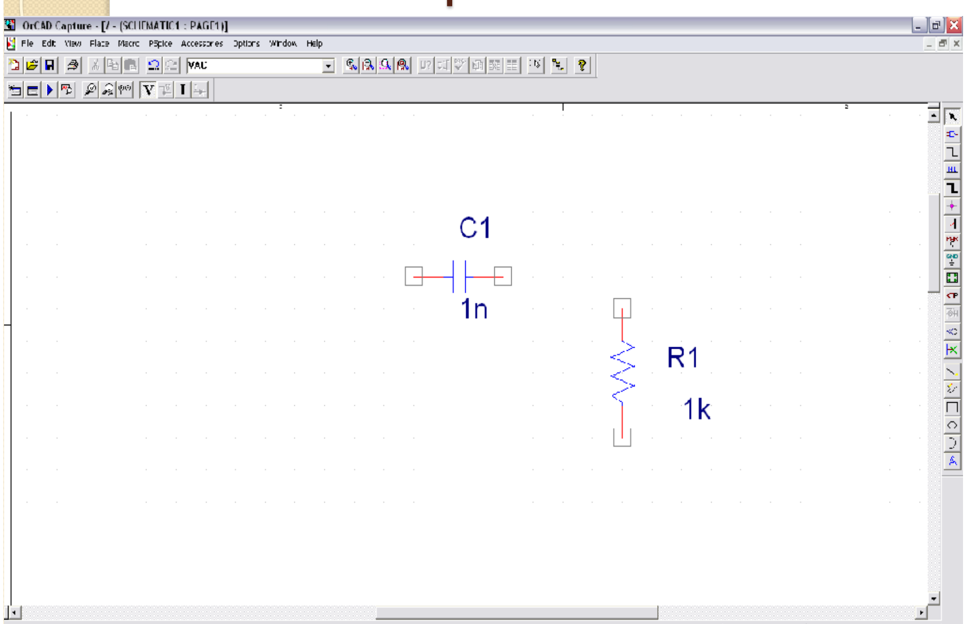
5) Get **plots** for IR1(resistance current), RMS value of IR1 and V1(input voltage) using “Add Trace” command from “Trace” menu



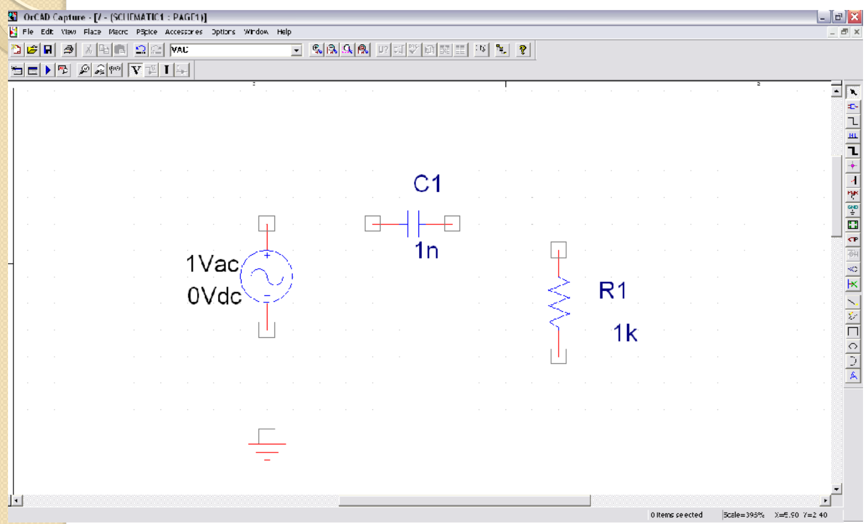
Design Example 2

- Take example of a high pass filter (RC)
- First we will follow initial steps of creating a new project
- Now placing the components.

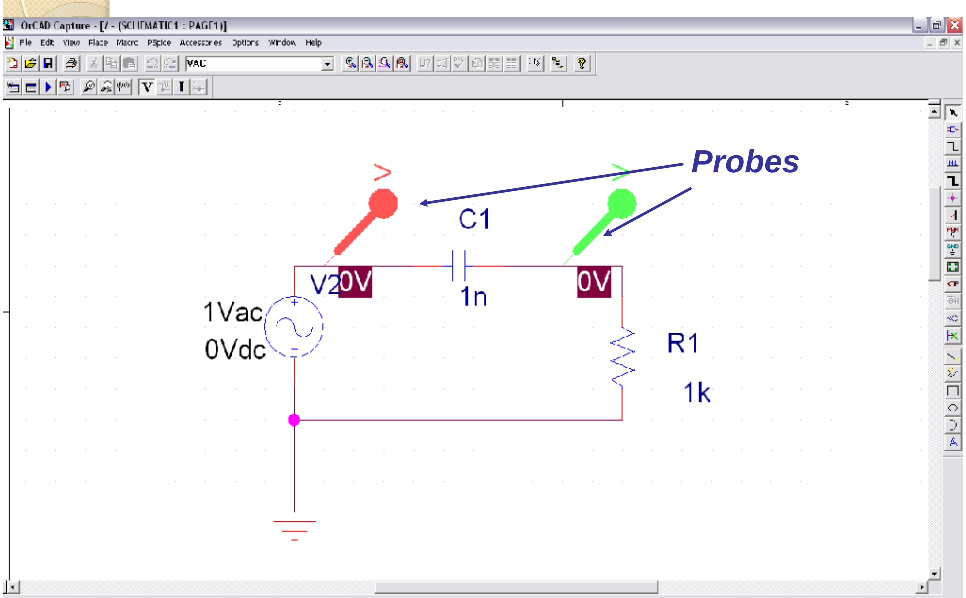
Place the components



Place the source and ground



Add Probe

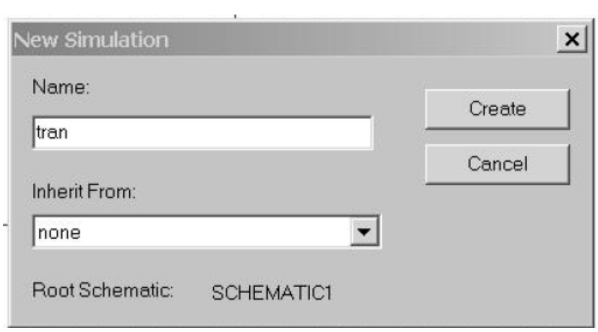


Points to remember

- In order to simulate the circuit signal reference must be named **0** (the number **zero**)

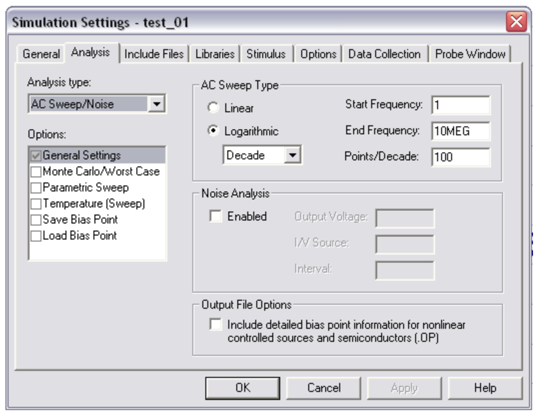
Creating new simulation profile

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



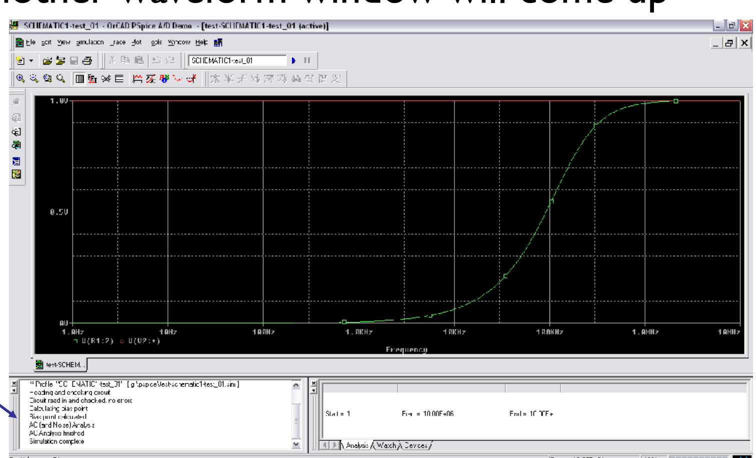
AC Analysis

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



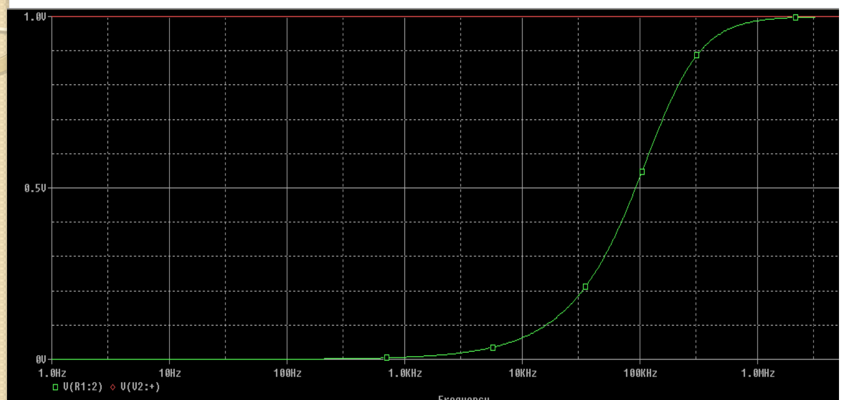
Run

- Go to PSPICE → Run
- Another waveform window will come up



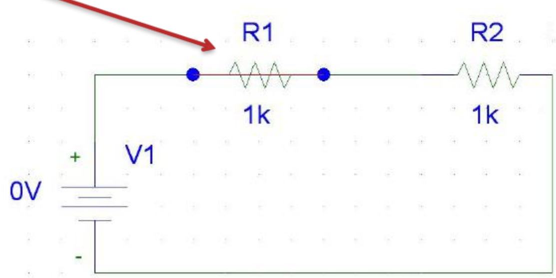
25

Result



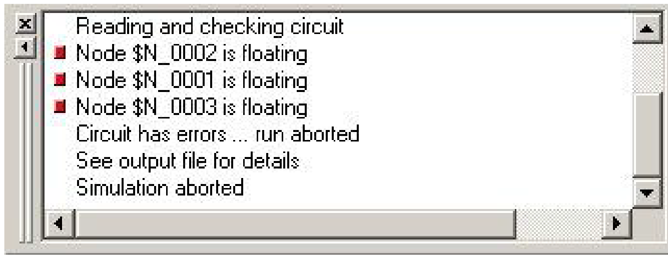
Common error

•If your circuit looks like this: (GND)



You have shorted out resistance R1. That is why you see the big dots on either side of the resistance and the wire running through it. (shown here in red).
•Rectify it by deleting the wire running through the resistance R1.

Common error



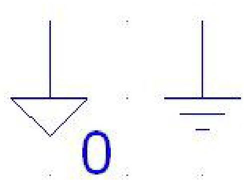
You probably got an error as shown below:
•*If you see the "Node floating" error, you have* ***not*** *placed a* ***ground*** *in the circuit*
–Every circuit in PSpice MUST have a ground

All analog schematics must have a ground symbol otherwise a floating error will be displayed.

Grounds

- •There are two kinds of grounds in PSpice - Analog ground (agnd or gnd_analog in PSpice) and digital ground (egnd or gnd_earth in PSpice)

- •Analog GND



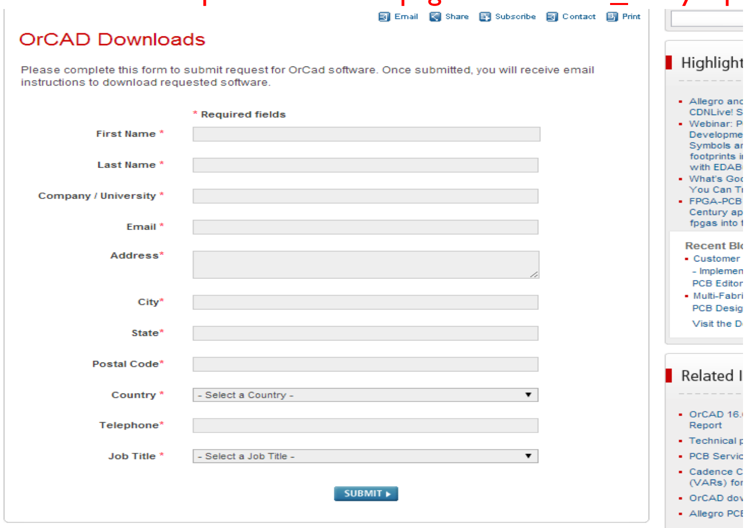
-
- •Digital Earth GND

27

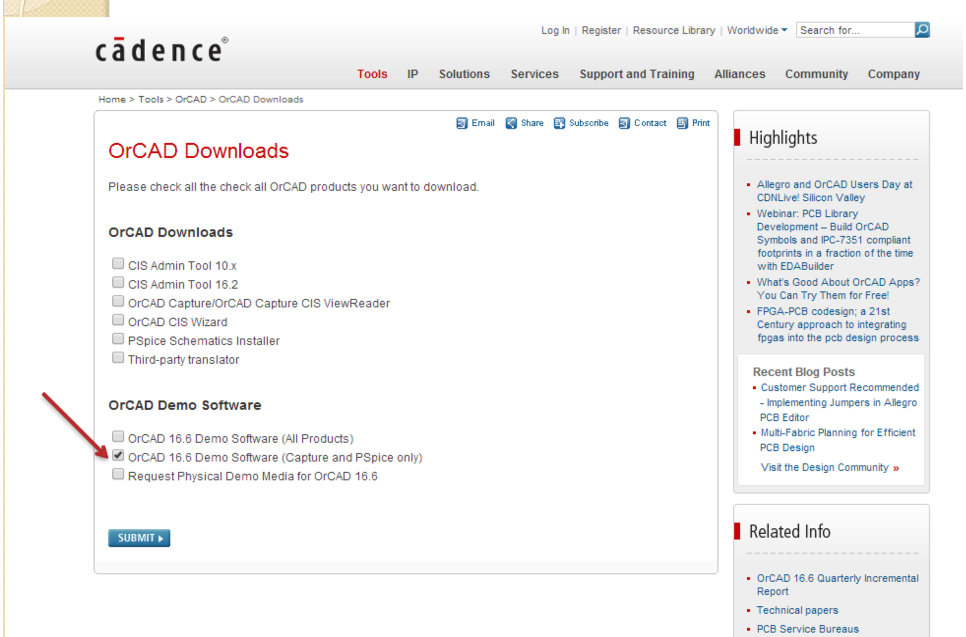
Get Free copy of Spice v.16.6

- Register you details at:

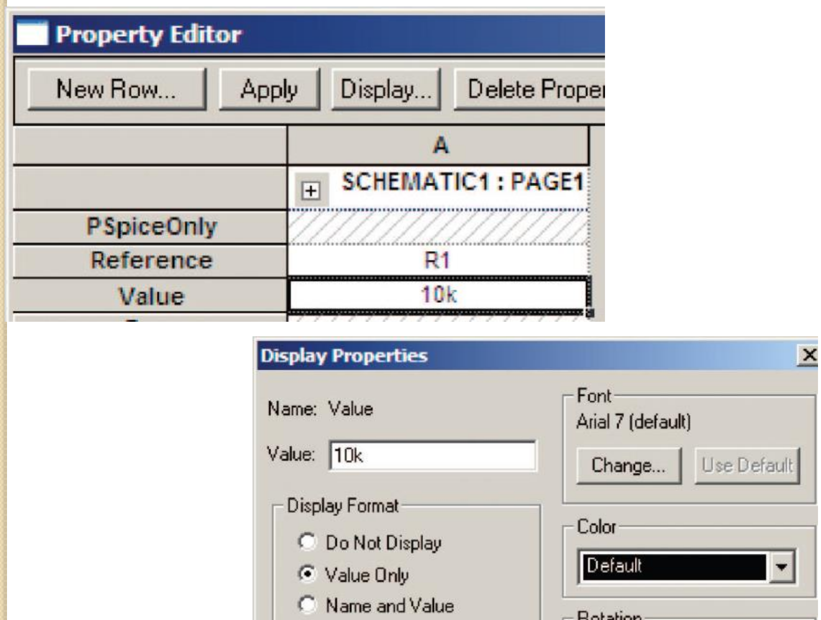
http://www.cadence.com/products/orcad/pages/downloads_verify.aspx



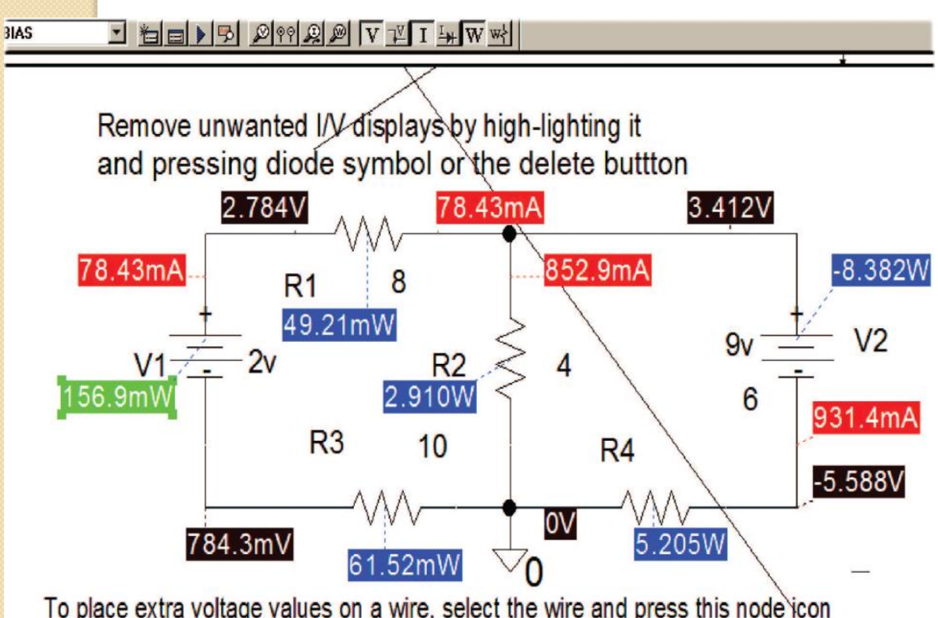
You will get the link to download the lite free version by email



Changing the part value



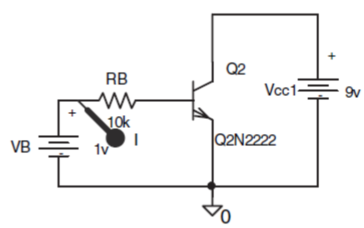
Displaying DC conditions

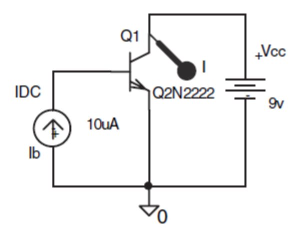


29

The Input andOutput BJT Characteristics

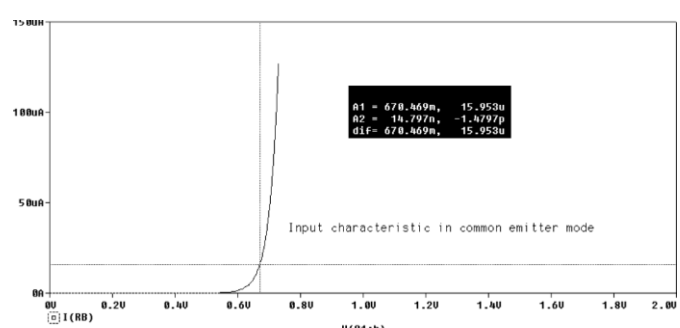
- A transistor may be connected in three modes: common-emitter, common-base, and commoncollector.



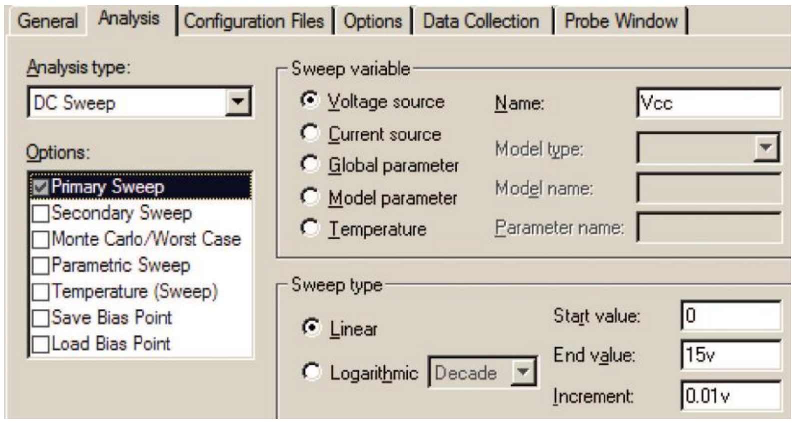


The input characteristic is a plot of base-emitter voltage versus base current

- **Analysis Setup** menu and ticking
- **DC Sweep**.

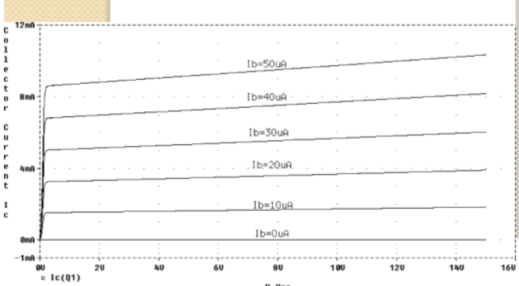


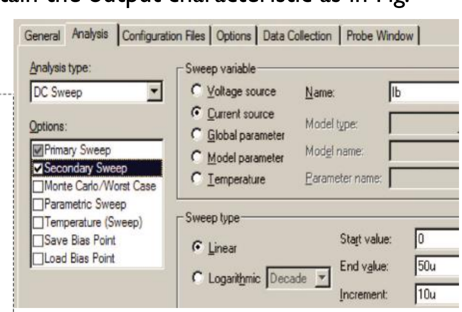
30



The Output Characteristic

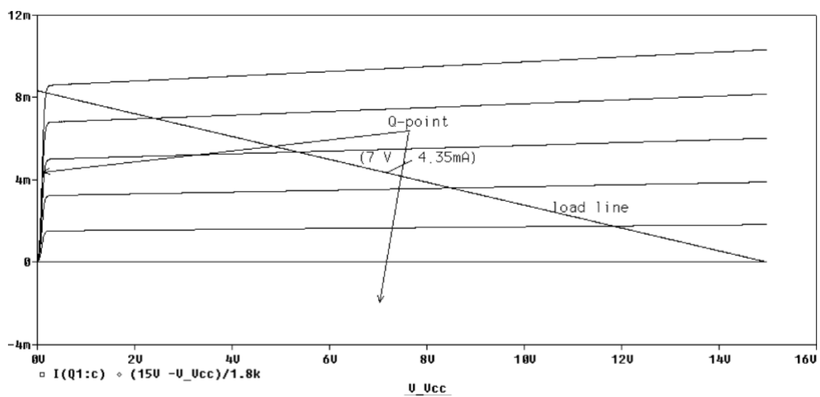
- From the **Analysis**menu, select **DCSweep** shown in Fig. and
- set the **Vcc** parameters in the **Primary Sweep** menu.
- Select the **Secondary Sweep** and enter the values shown in Fig.
- Press the **F11** button to obtain the output characteristic as in Fig.





DC Load Lines

- Obtain the output characteristic
- as before but increase the collector voltage, Vcc, to 15 V. To superimpose a load line on the
- output characteristic as shown in Fig. 7.23, click the **ProbeTrace** menu and select **Add/Trace**



Thank You