

Basic Electrical Engineering Lab (BEEE102P)

Getting started with ORCAD

Dr. Sonam Shrivastava/ Assistant Professor (Sr.) /SELECT

BEEE102P
Basic Electrical and Electronics Engineering Lab
List of Experiments

Cycle-1

S.No	Experiment Title
1	Verification of Mesh current analysis using ORCAD/Capture CIS
2	Verification of Nodal voltage analysis using ORCAD/Capture CIS
3	Verification of Thevenin's theorem using ORCAD/Capture CIS
4	Verification of Maximum power transfer theorem using ORCAD/Capture CIS
5	Verification of Resonance phenomenon in a series RLC circuit using ORCAD/Capture CIS

Cycle-II

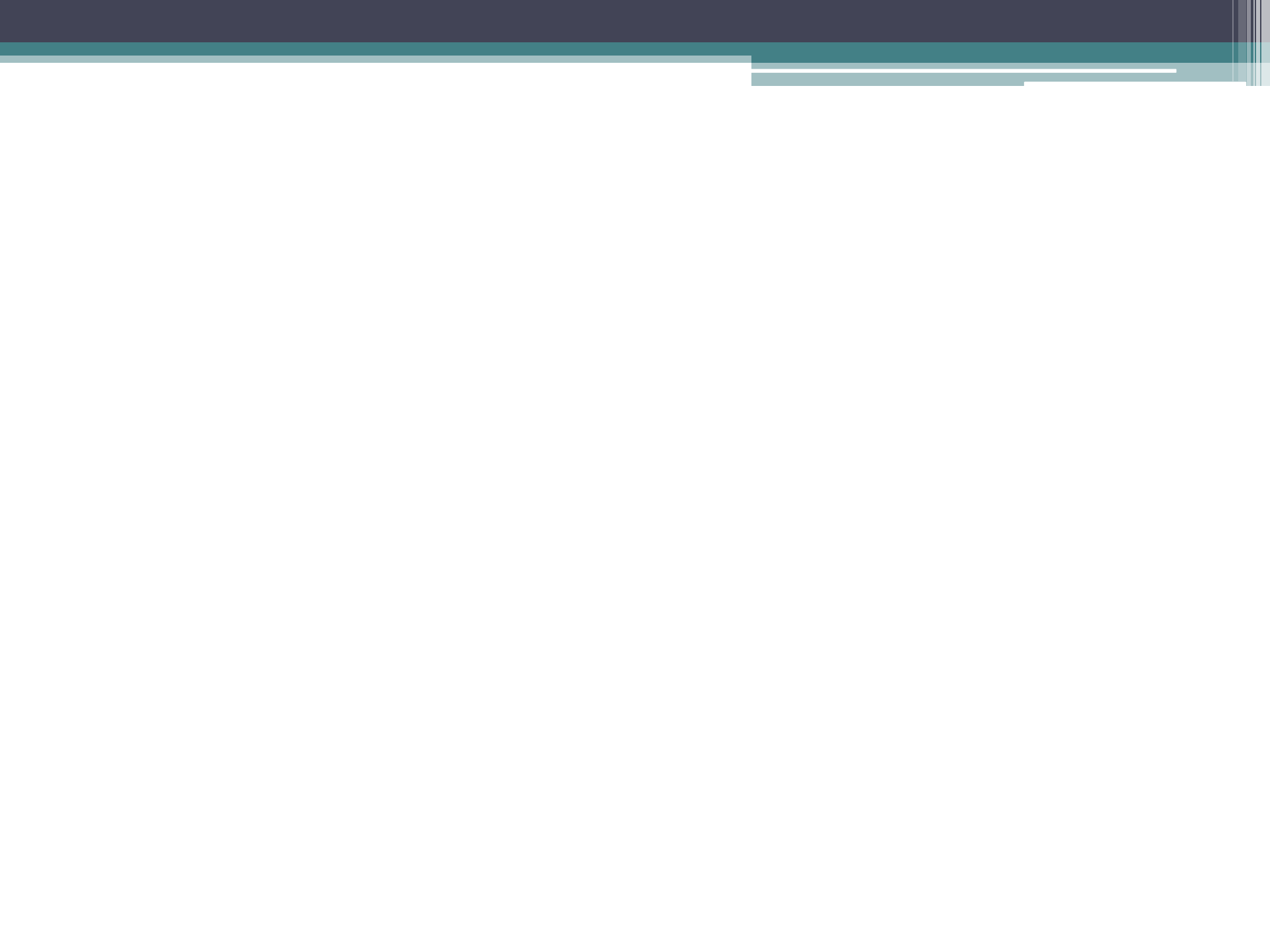
S.No	Experiment Title
1	Analysis of step-up and step-down transformer using ORCAD/Capture CIS
2	Wiring circuit for a. Single lamp b. Fan with regulator
3	Wiring circuit for godown with two-way switch
4	Measurement of power and energy consumed by a given 3-phase AC load
5	Study of earthing and measurement of earth pit resistance



Introduction

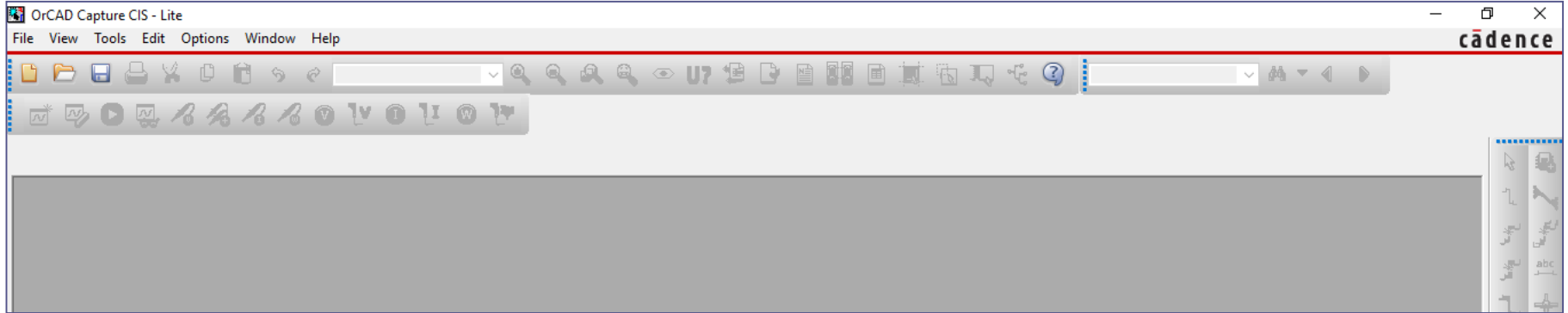
Getting Started with OrCAD Pspice Software

Circuit diagram for introduction to ORCAD

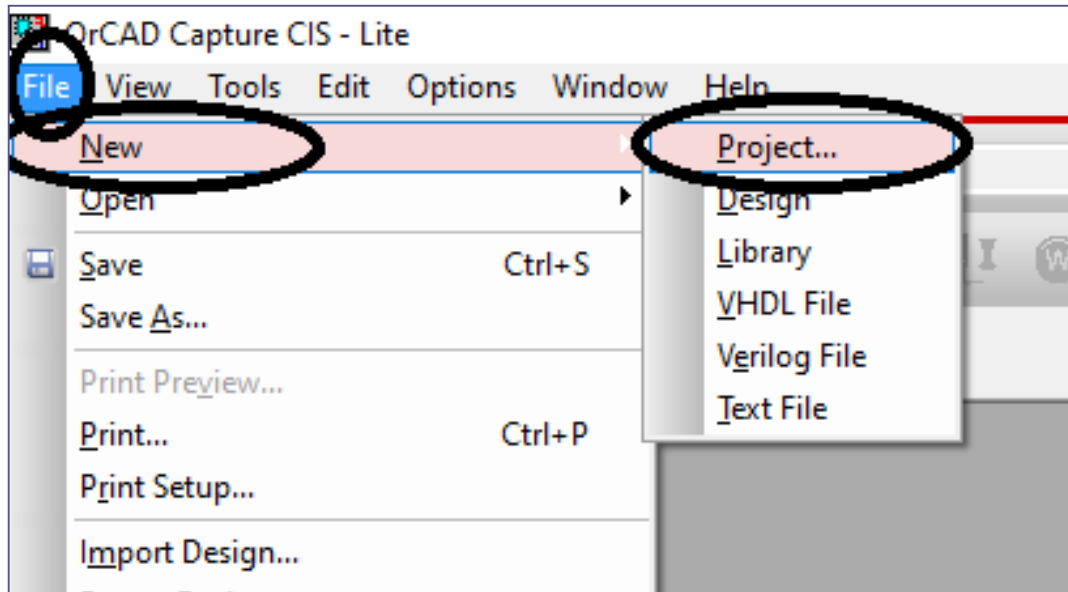


Getting started with ORCAD Pspice software

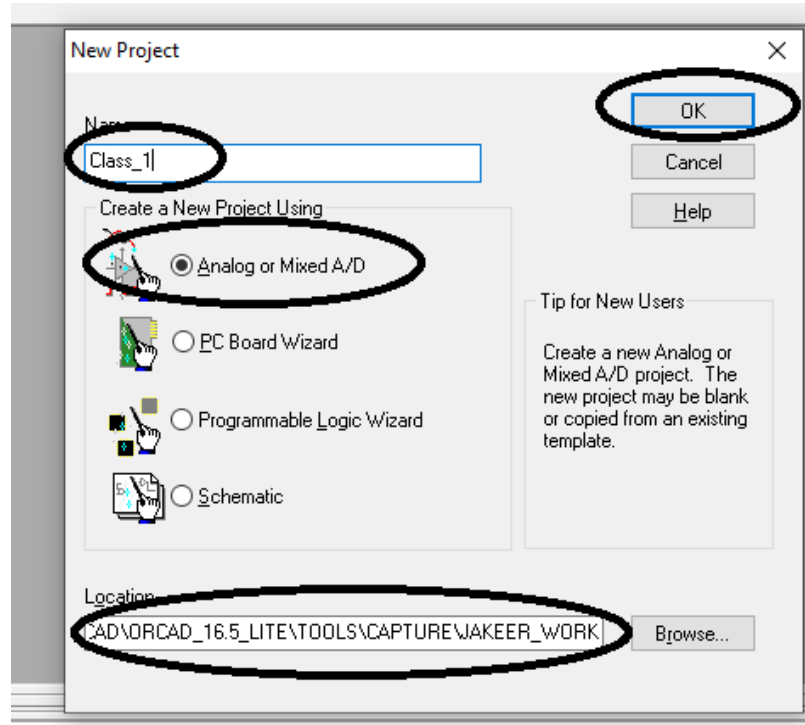
1. Start the application:



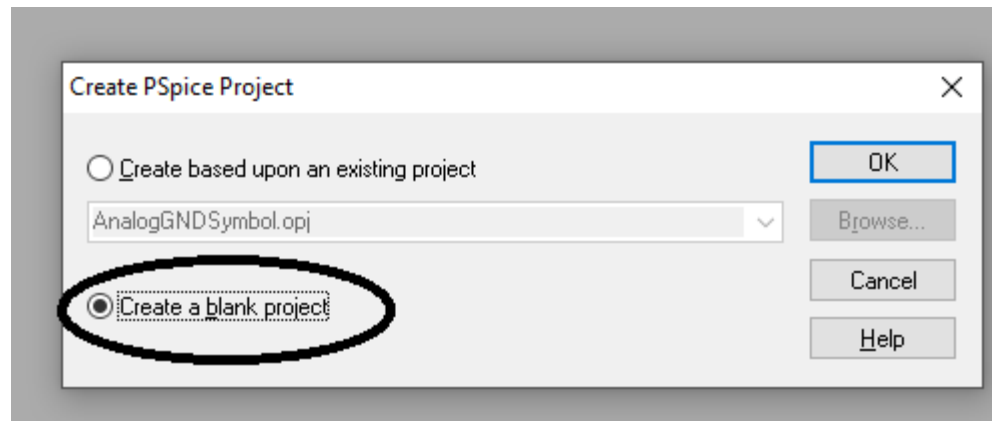
2. Start New Project:



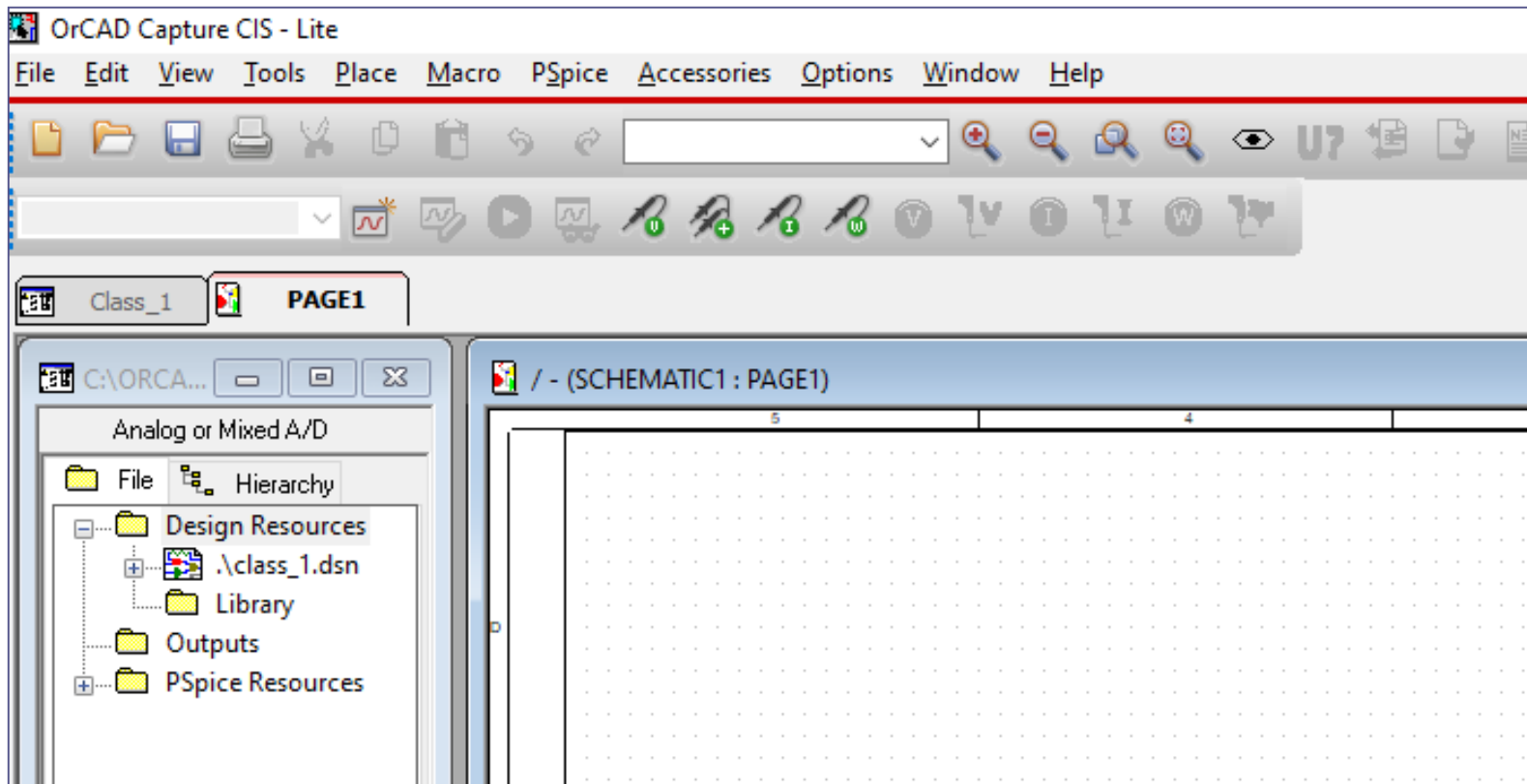
3. Give name to the project and create new project using Analog or mixed A/D



Location: You should create a new directory for your project since PSpice will generate a bunch of project files in this folder.

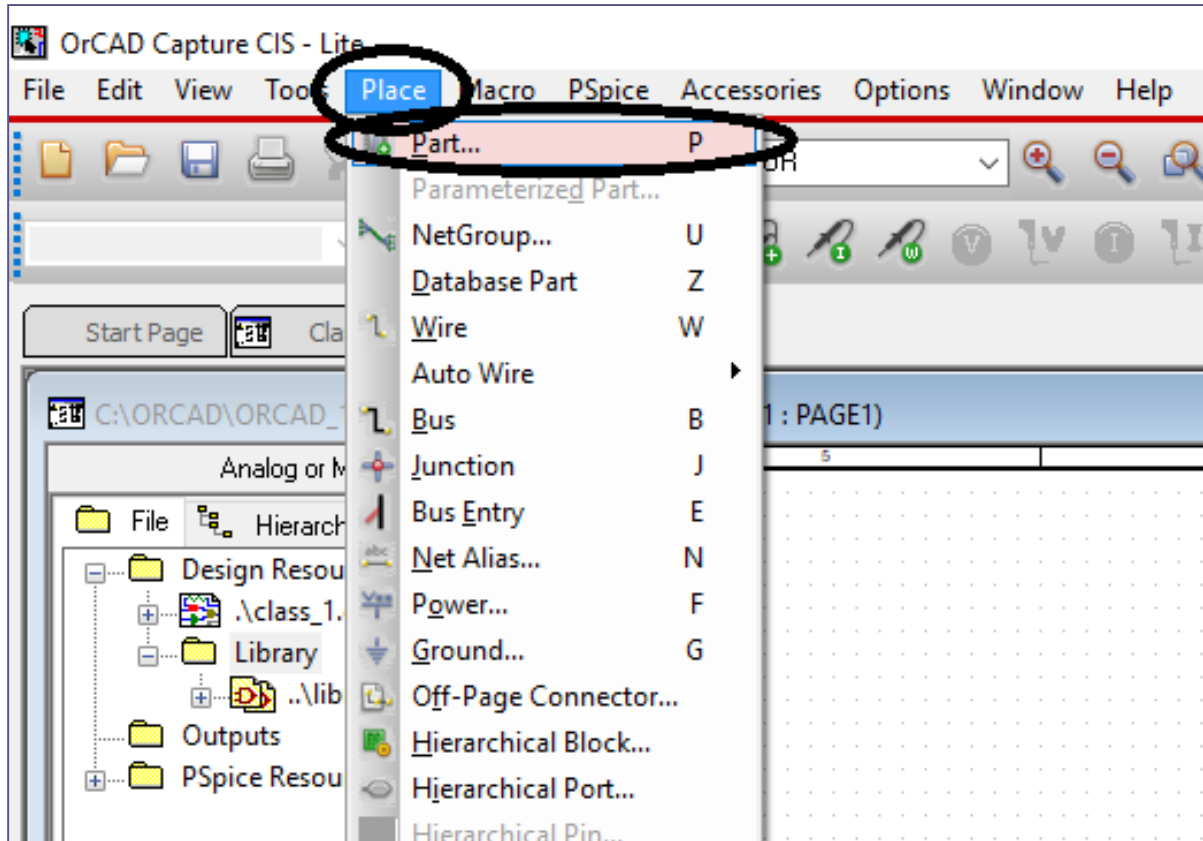


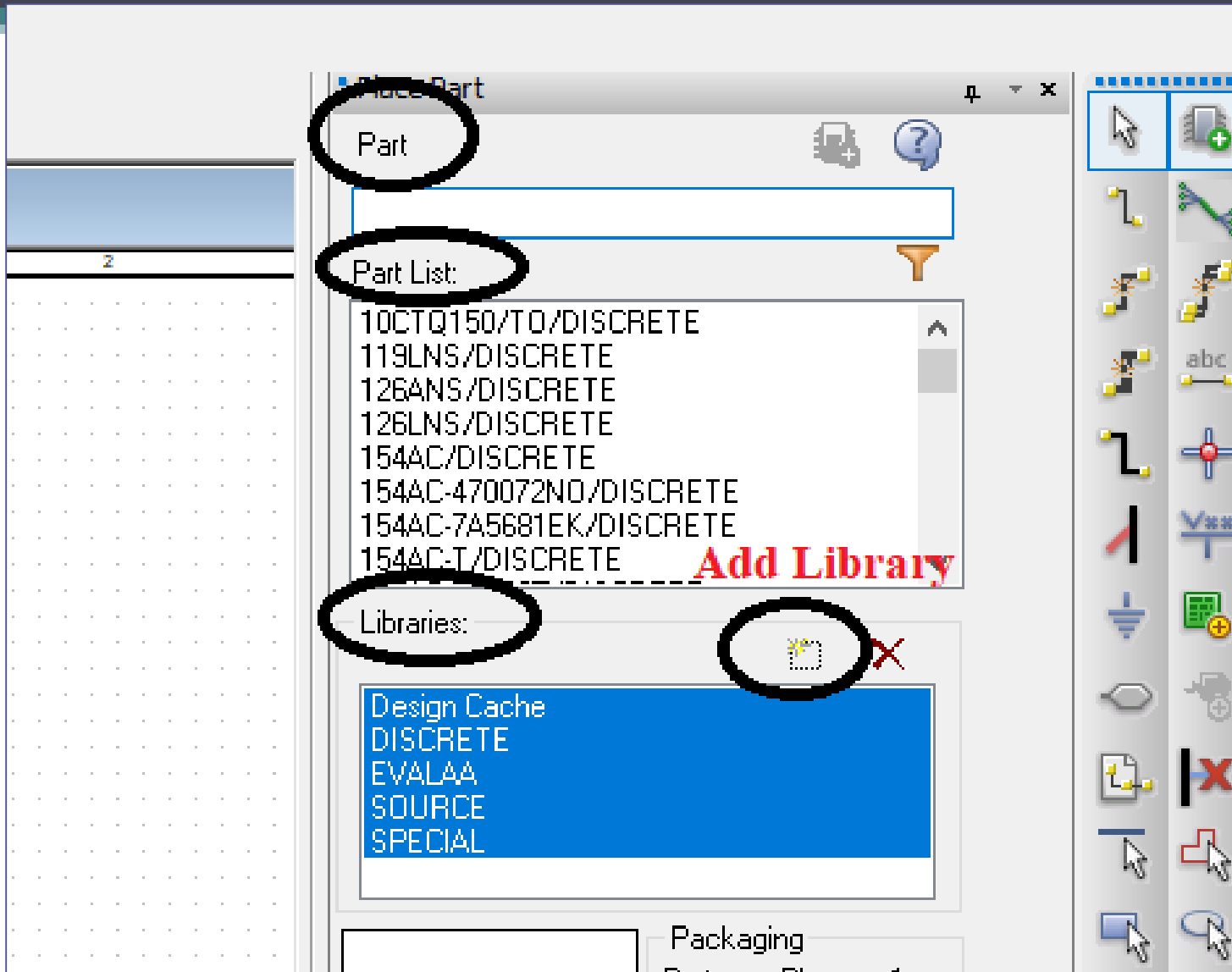
4. The editor window where the circuit diagram can be drawn:



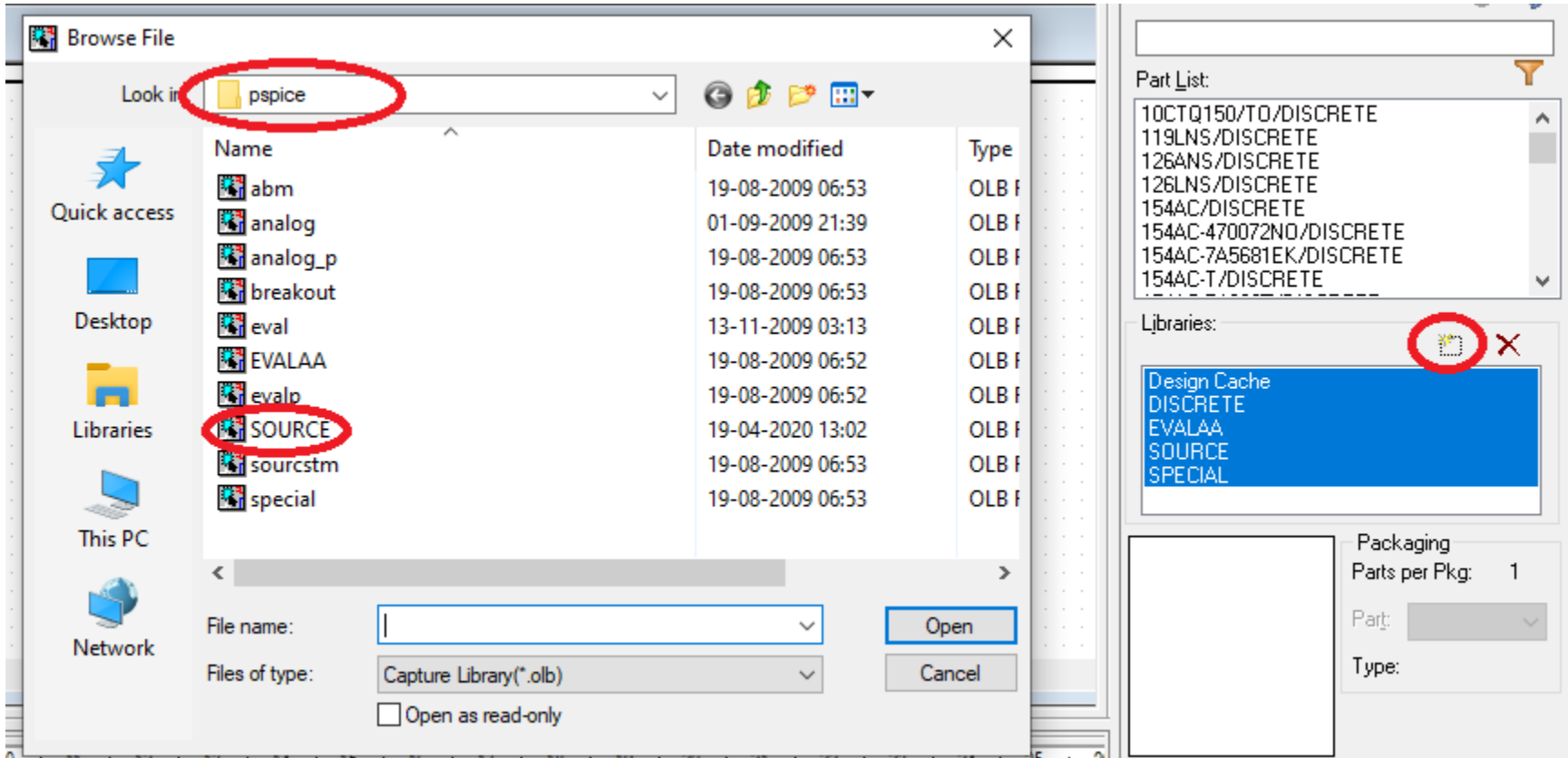
5. Start drawing the schematic (circuit diagram)

5.a . We need to copy the parts into present working directory.





5.b . Add libraries to the project: Click on “Add library” symbol



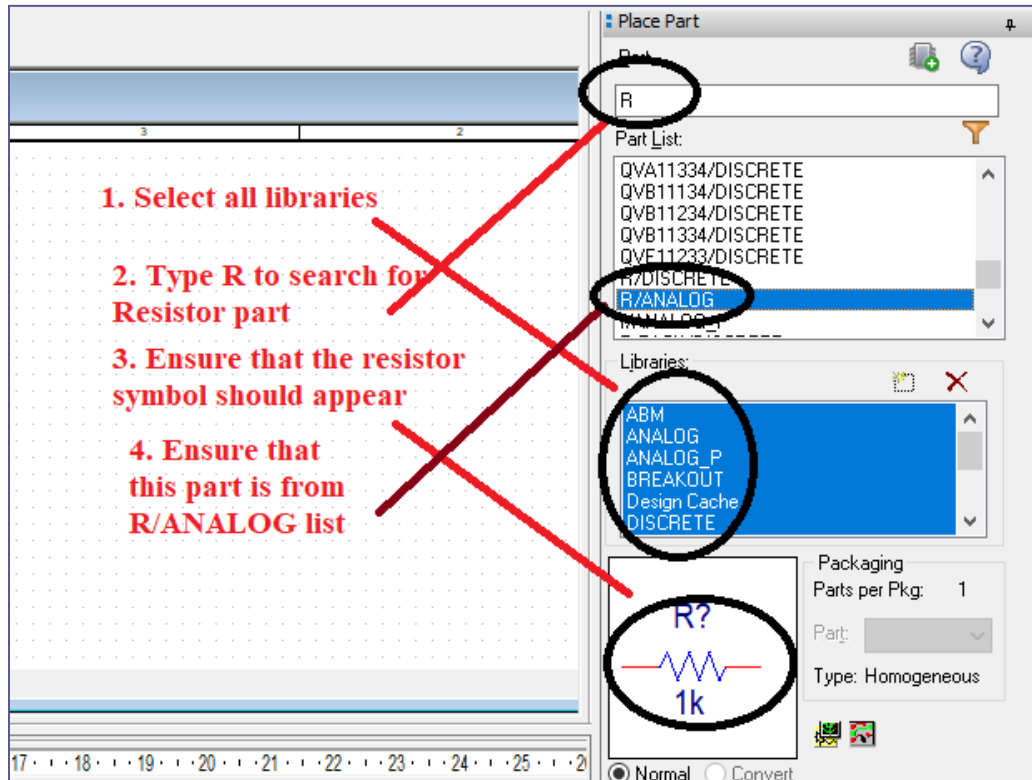
Ensure that the adding libraries are under the pspice folder

Select all the library parts (select any one library and then **ctrl+a**)

under the Pspice and click on “open”

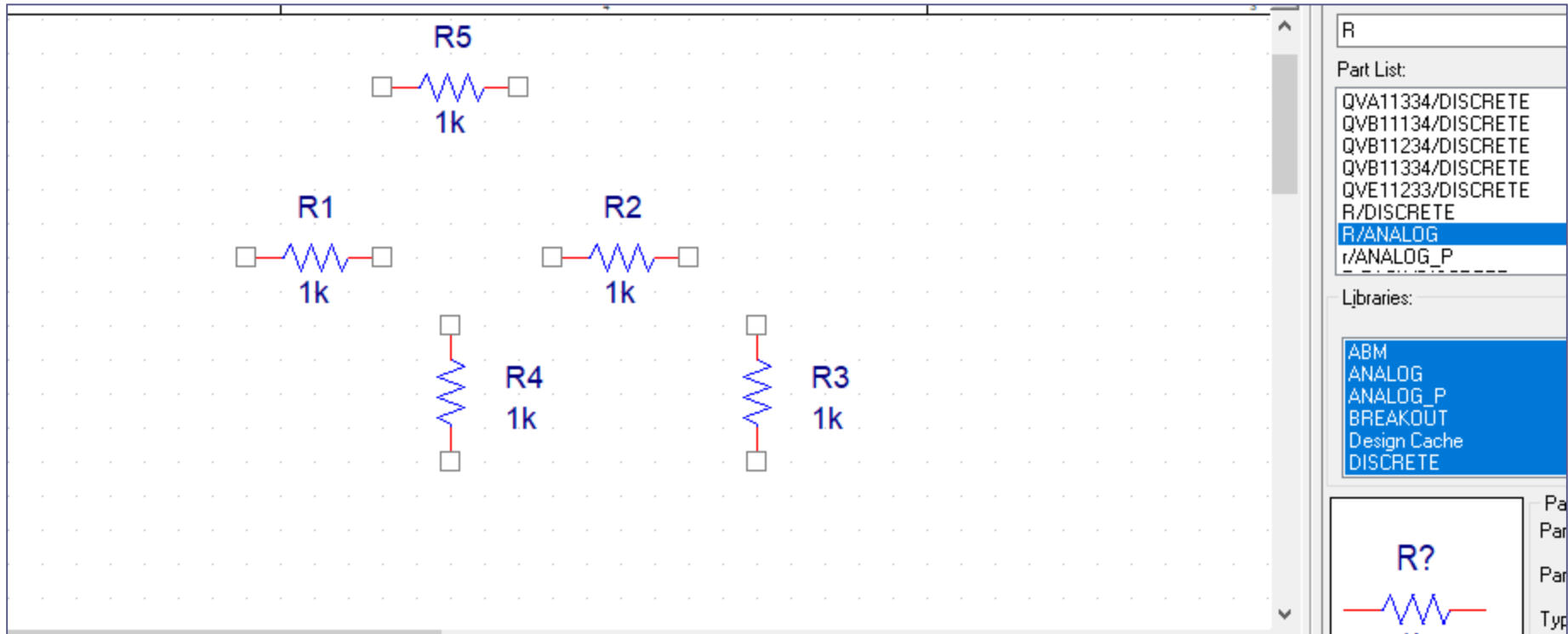
5.d . Place a resistor part onto the schematic editor

Locate the resistor part (which is available in R/ANALOG)



Double click on R/ANALOG and place the resistor on the editor page at preferred place by moving the cursor. You can place multiple Resistor parts on the schematic page.

After placing required no of resistor parts at appropriate places,
click the mouse (observe the change in Resistor's color)
press escape (or Right click mouse and select "End mode")

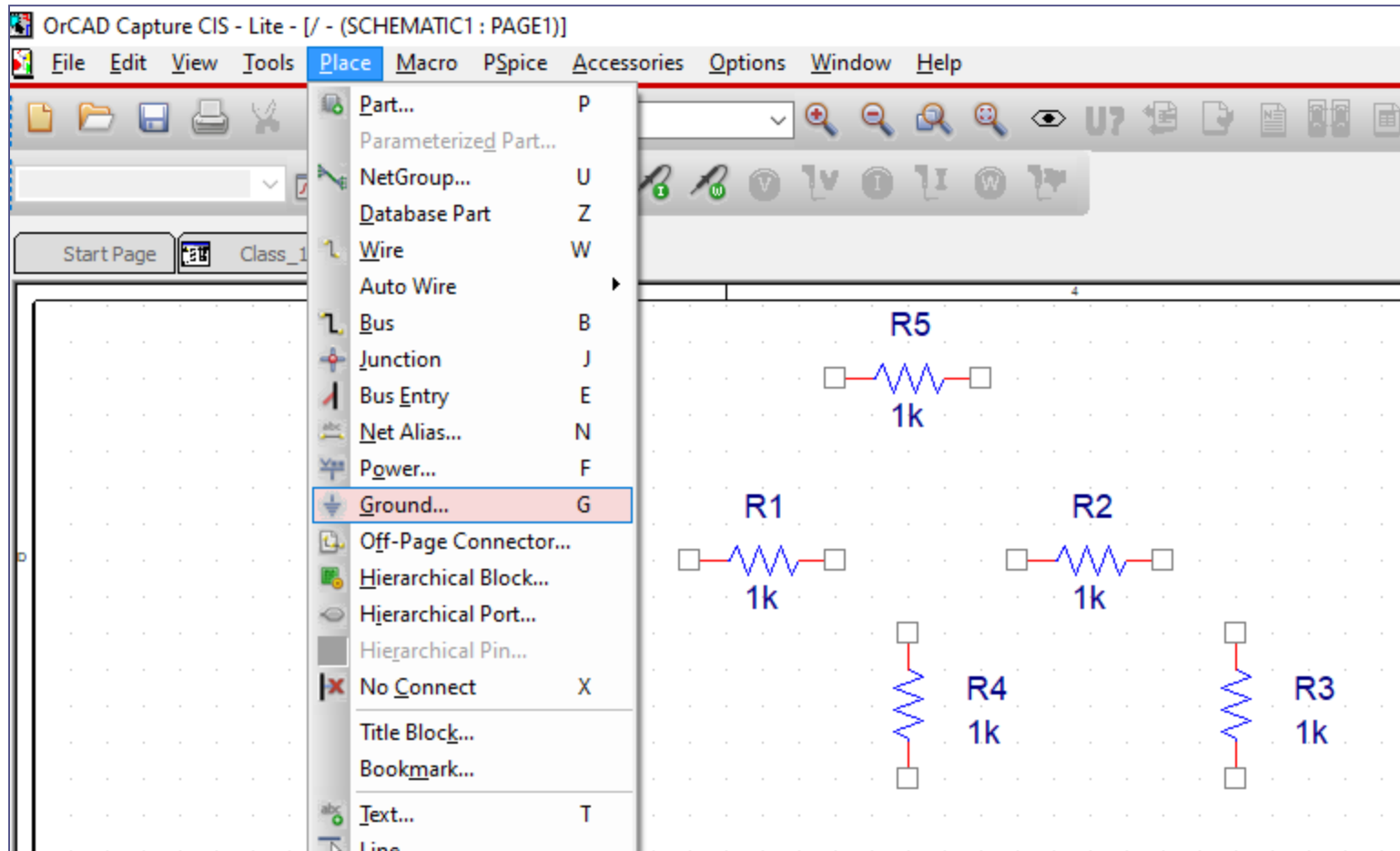


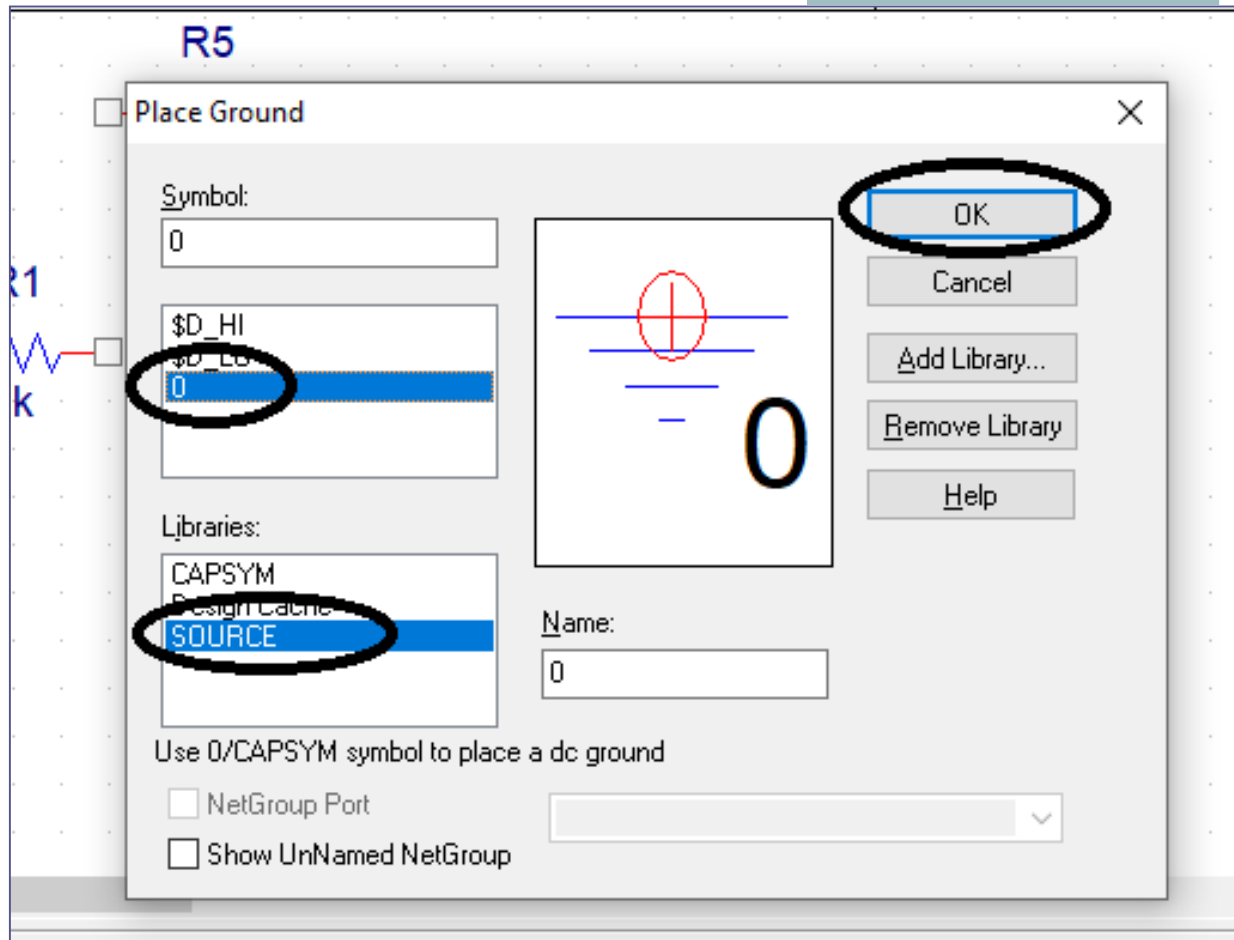
5.e . Place a voltage source part onto the schematic editor page

Go to **SOURCES >> VDC**

The image shows a screenshot of a schematic editor interface. The main workspace displays a circuit diagram with a voltage source labeled V1 (0Vdc) and five resistors labeled R1, R2, R3, R4, and R5, all with a value of 1k. The voltage source is connected to the top of R1, R2, and R3. R4 and R5 are connected to ground. The 'Place Part' dialog box is open on the right side. The 'Part' field contains 'VDC'. The 'Part List' shows a list of parts: VAC, VDC, VEXP, VPULSE, VPWL, VPWL_abm, VPWL_ENH, and VPWL_F_RE_FOREVER. The 'VDC' part is highlighted with a blue bar and a black circle. The 'Libraries' section shows a list of libraries: EVAL, EVALA, EVALP, SOURCE, SOURCESTM, and SPECIAL. The 'SOURCE' library is highlighted with a blue bar and a black circle. The 'Packaging' section shows 'Parts per Pkg: 1', 'Part:' (empty), and 'Type: Homogeneous'. A preview of the VDC part is shown at the bottom left of the dialog box.

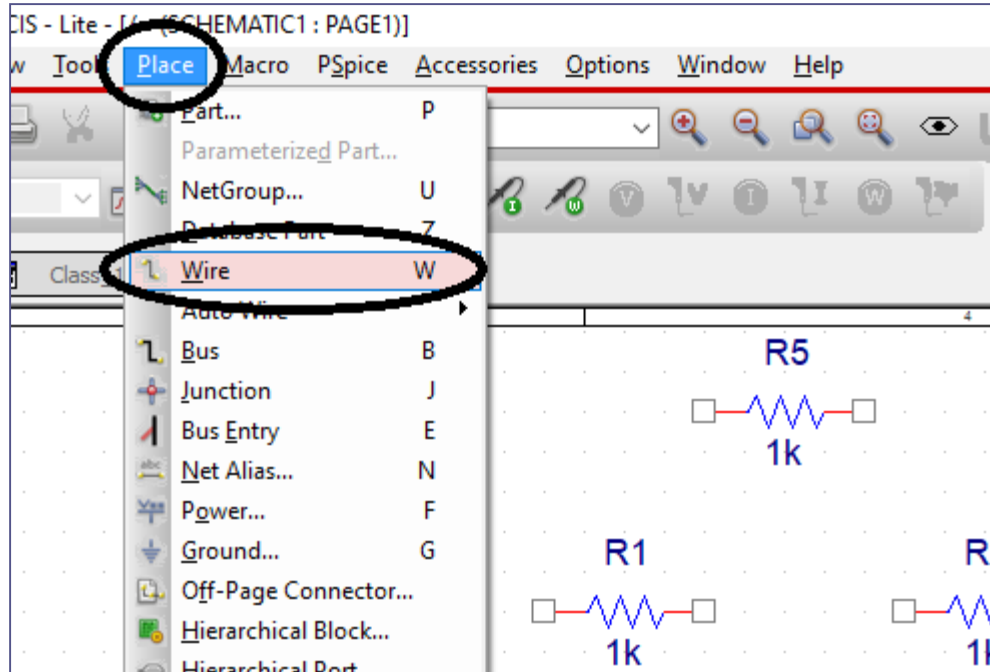
5.f . Place a ground part to the circuit schematic editor page





If SOURCE library doesn't appear, then use "Add Library" to add from "Pspice>>source" folder.

5.f . Connect the components as per the circuit given using wire part.



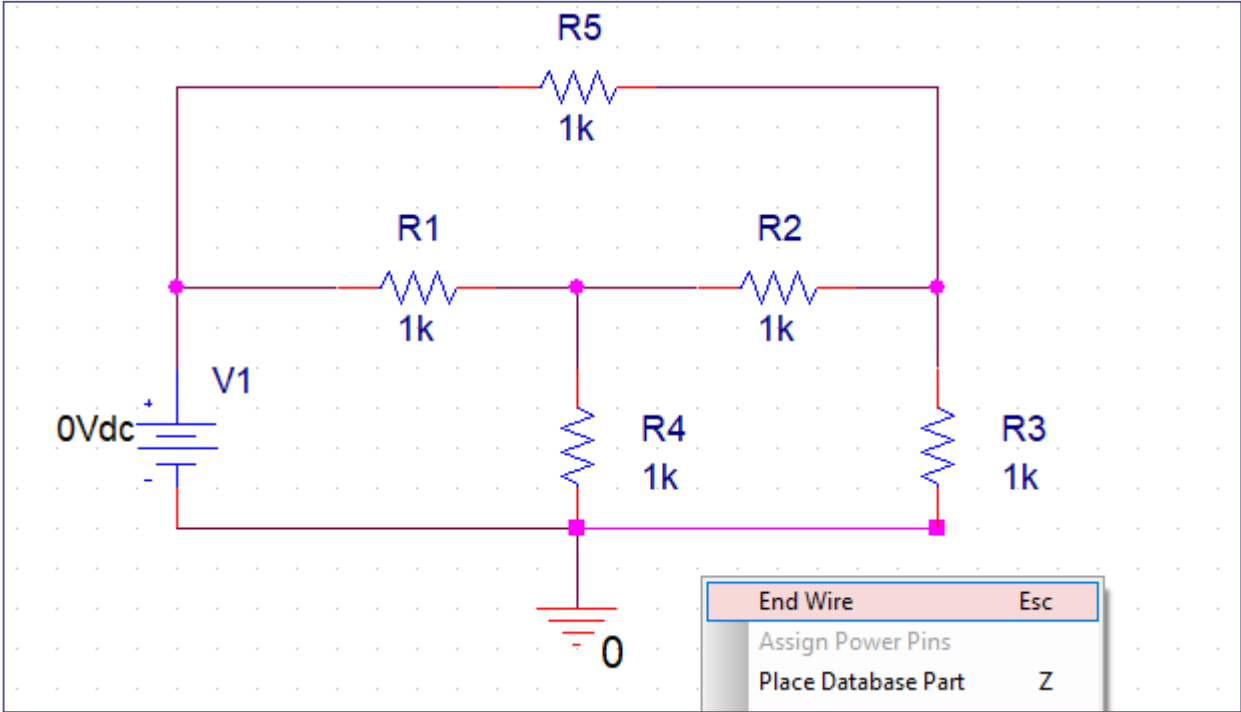
Connect all the circuit elements as per the circuit.

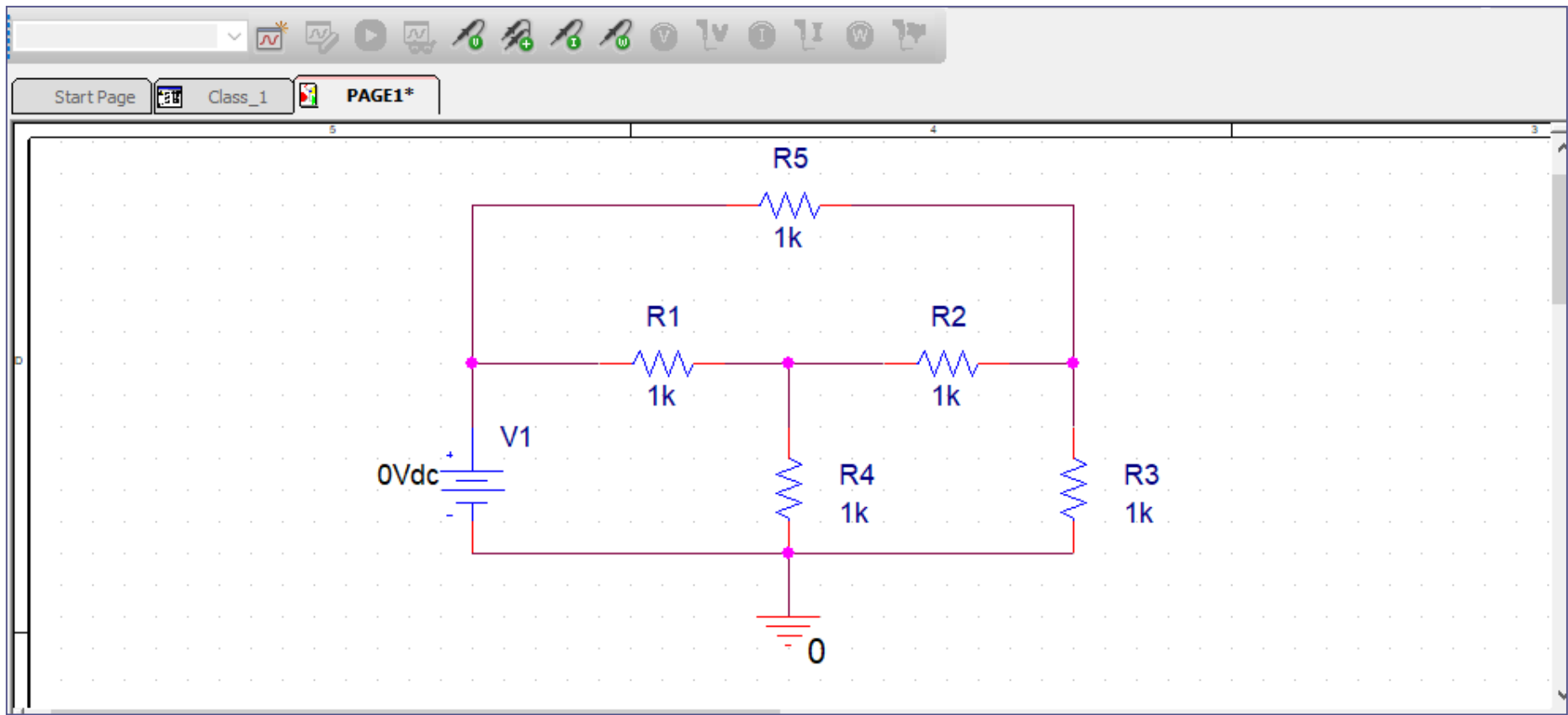
Select the wire part

Click (and release) the mouse at the starting end of the connection

Take the cursor to the other end of the connection and click (and release) the mouse there to complete the connection.

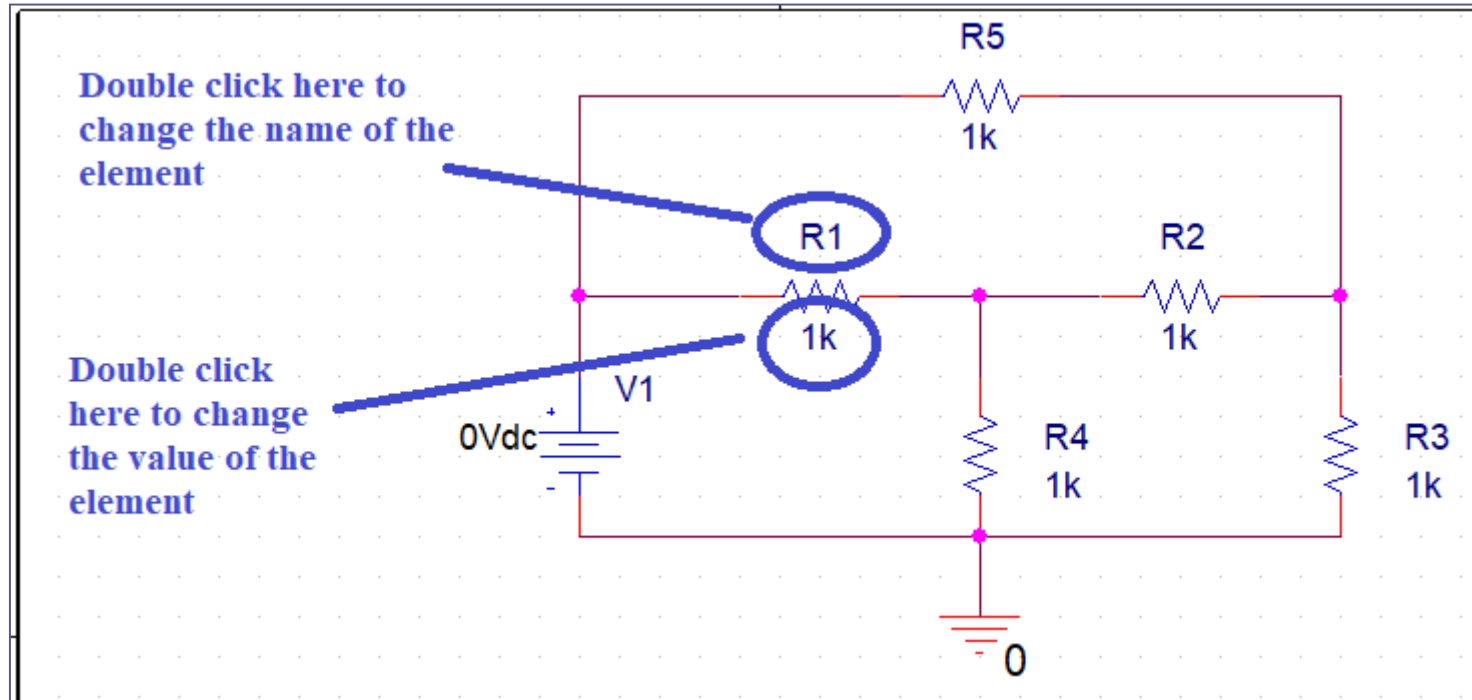
After completion of all the connections, then end the connection mode by clicking at “End wire”.



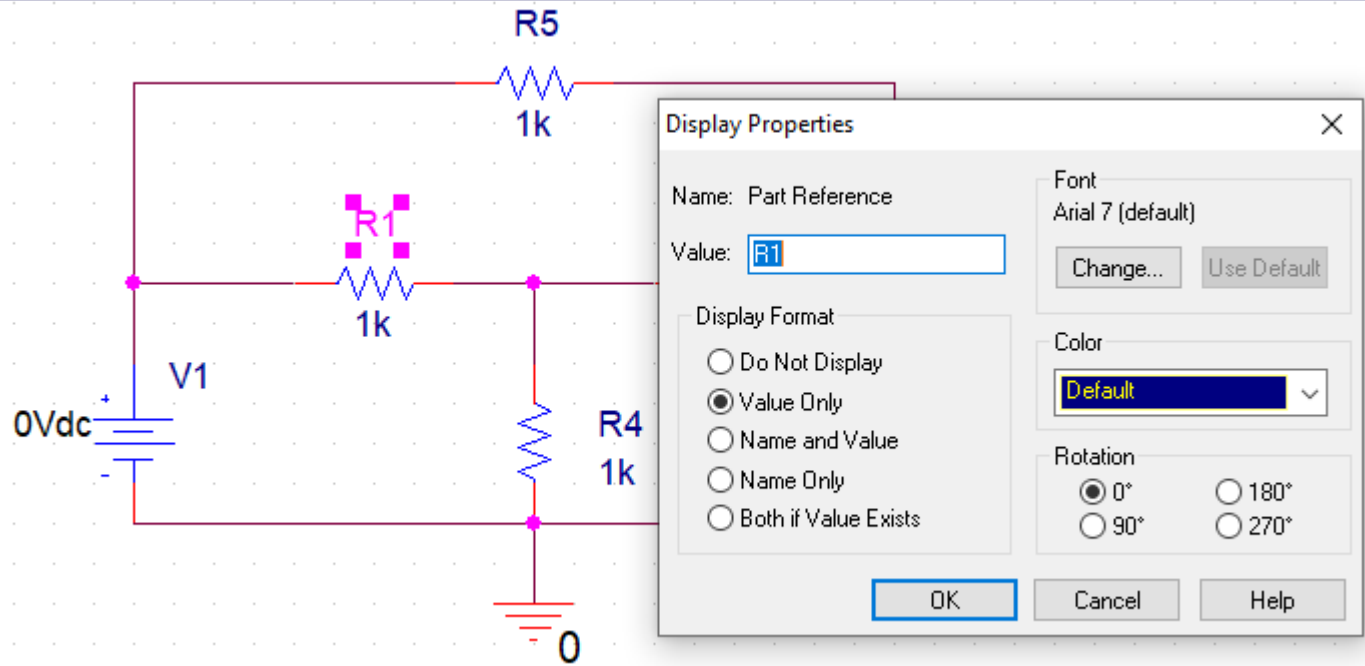


5.g . Edit the values of the components

To edit the Resistor element:

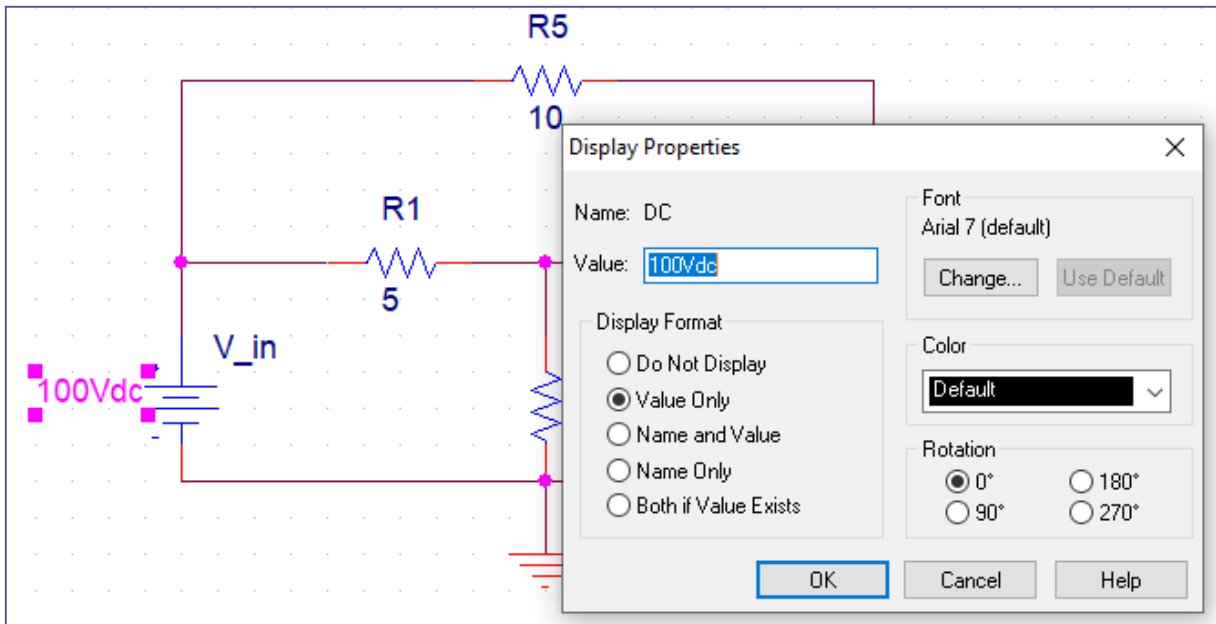


**To edit
the name:**

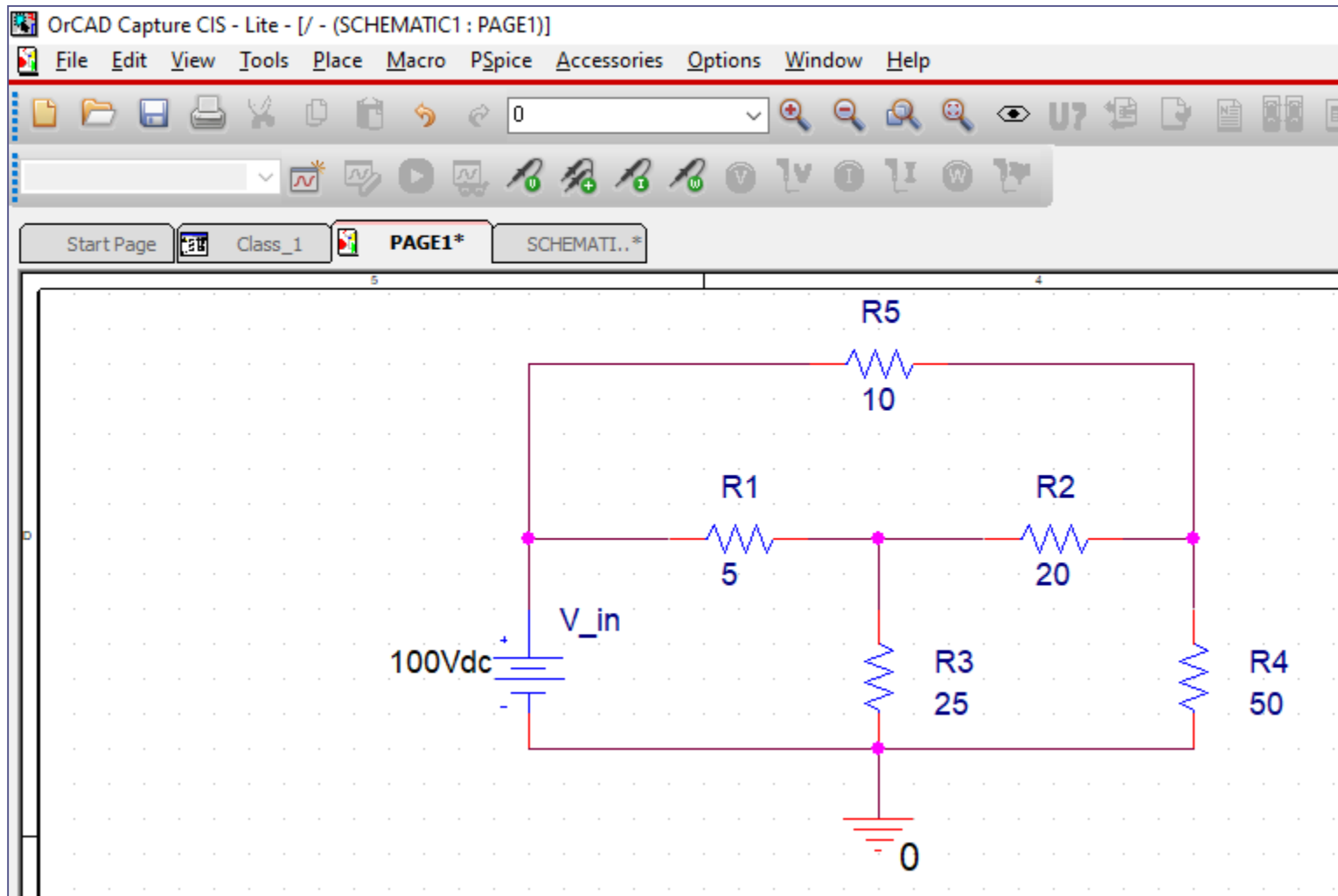


**To change
the value:**

**Don't give
space between
100, V, and dc**

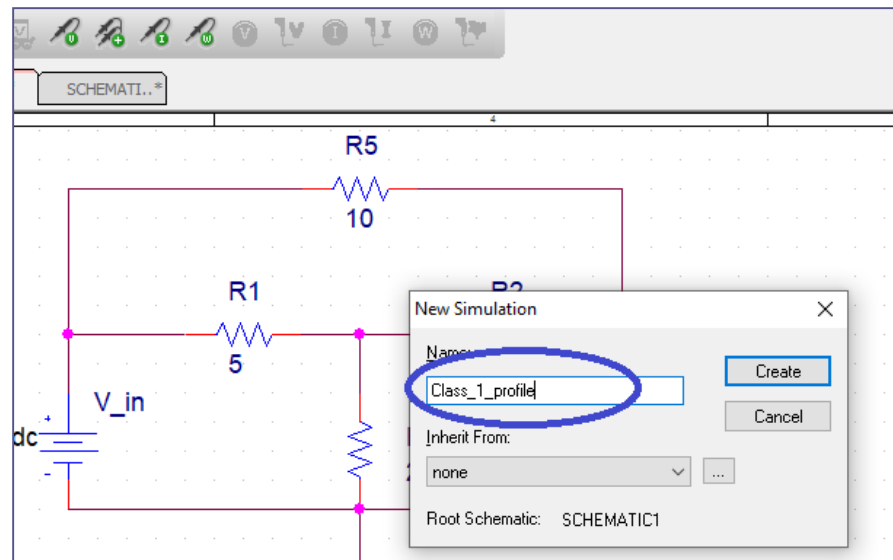
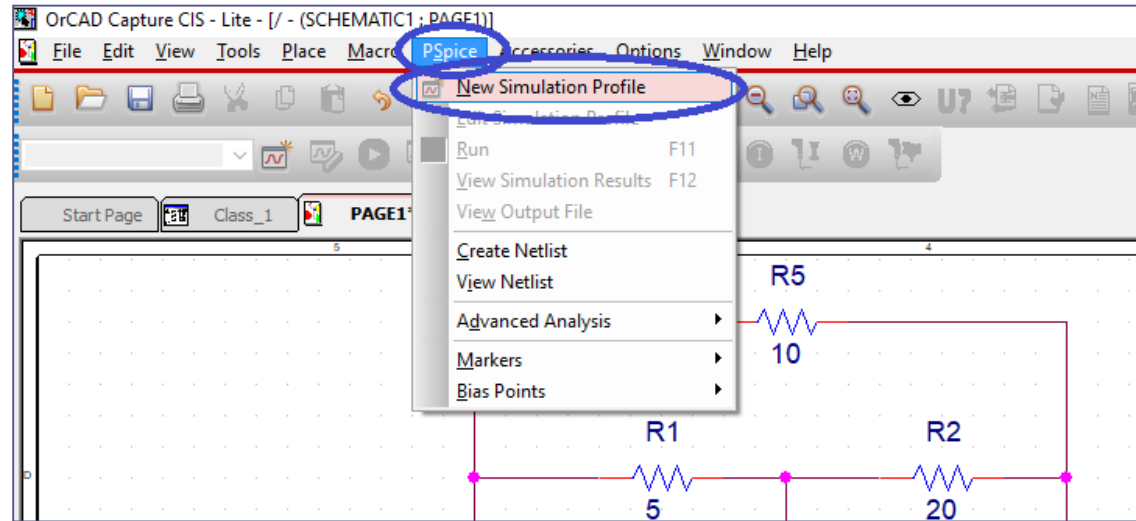


5. End of the schematic diagram drawing step



6. Simulate the circuit

We should create a new simulation profile for each circuit to run it.



6.a. Maximize the “Simulation settings-Class_1_profile”

The image shows a circuit simulation software interface. On the left, a circuit diagram is displayed on a grid. It consists of a 100Vdc DC voltage source connected in series with a resistor labeled 'R' with a value of 5. The node between the source and the resistor is labeled 'V_in'. The circuit is connected to ground.

Overlaid on the right side of the circuit is a dialog box titled "Simulation Settings - Class_1_profile". The dialog has several tabs: "General", "Analysis", "Configuration Files", "Options", "Data Collection", and "Probe Window". The "Analysis" tab is currently selected.

Under the "Analysis" tab, the "Analysis type:" dropdown menu is set to "Time Domain (Transient)".

The "Options:" section contains a list of checkboxes:

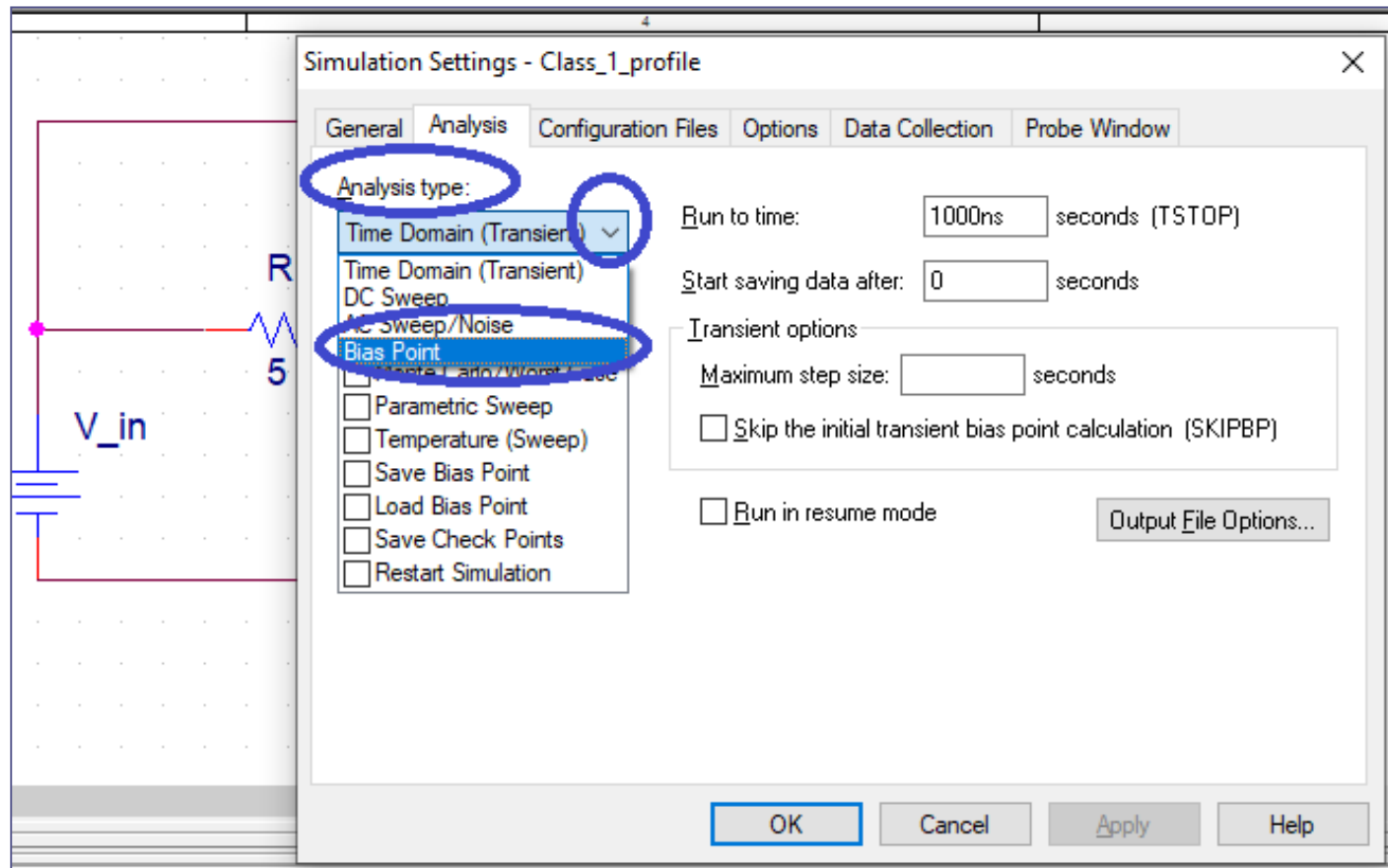
- General Settings
- Monte Carlo/Worst Case
- Parametric Sweep
- Temperature (Sweep)
- Save Bias Point
- Load Bias Point
- Save Check Points
- Restart Simulation

Other settings in the dialog include:

- Run to time:** 1000ns seconds (TSTOP)
- Start saving data after:** 0 seconds
- Transient options:**
 - Maximum step size:** [] seconds
 - Skip the initial transient bias point calculation (SKIPBP)
 - Run in resume mode
- Output File Options...** button

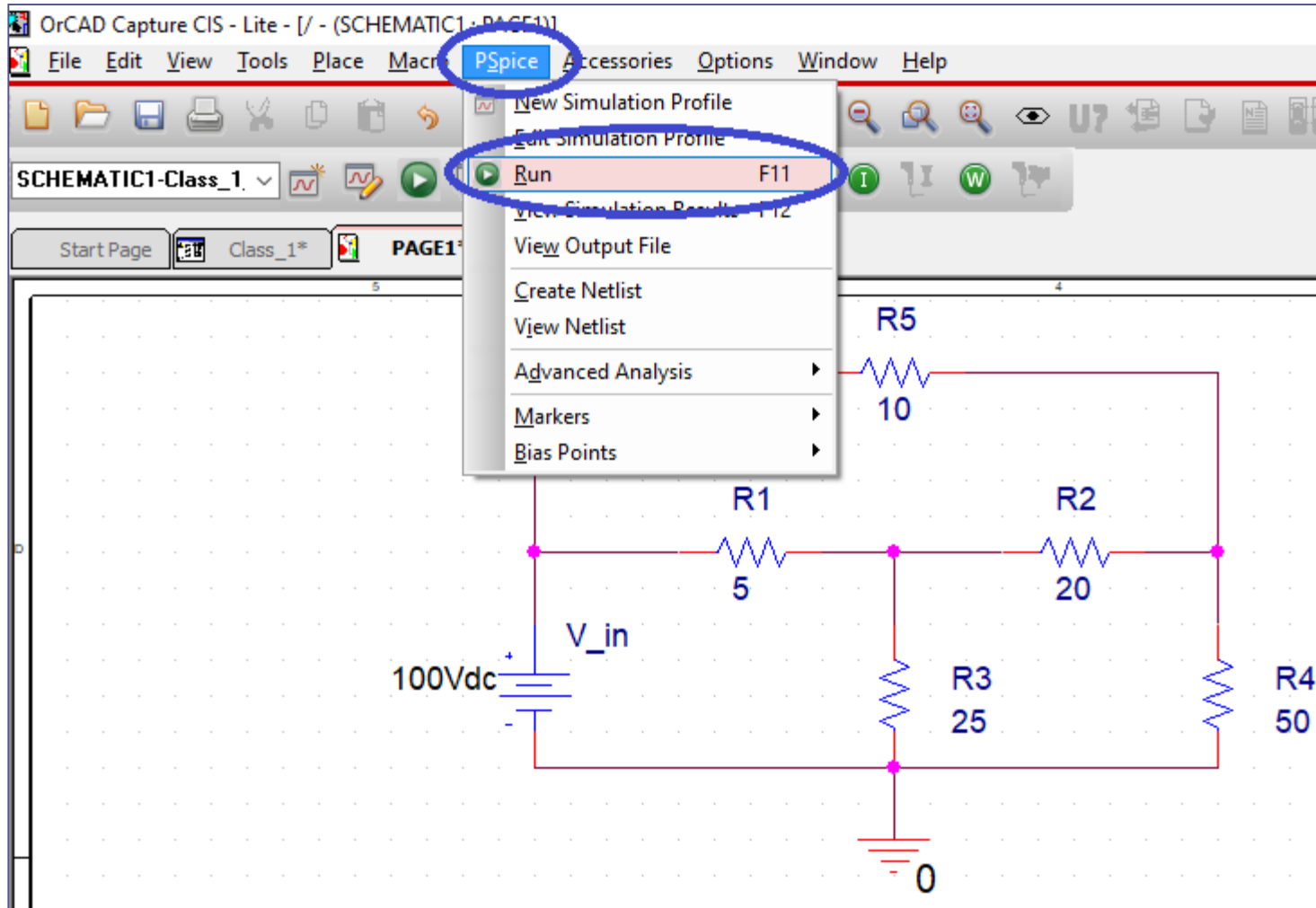
At the bottom of the dialog, there are four buttons: "OK", "Cancel", "Apply", and "Help".

6.b. Select the type of the analysis you need to perform:



For DC circuit analysis, Bias Point analysis is needed. This analysis will provide the steady state values of voltages and currents at various parts of the circuit.

6.c. Run the circuit simulation



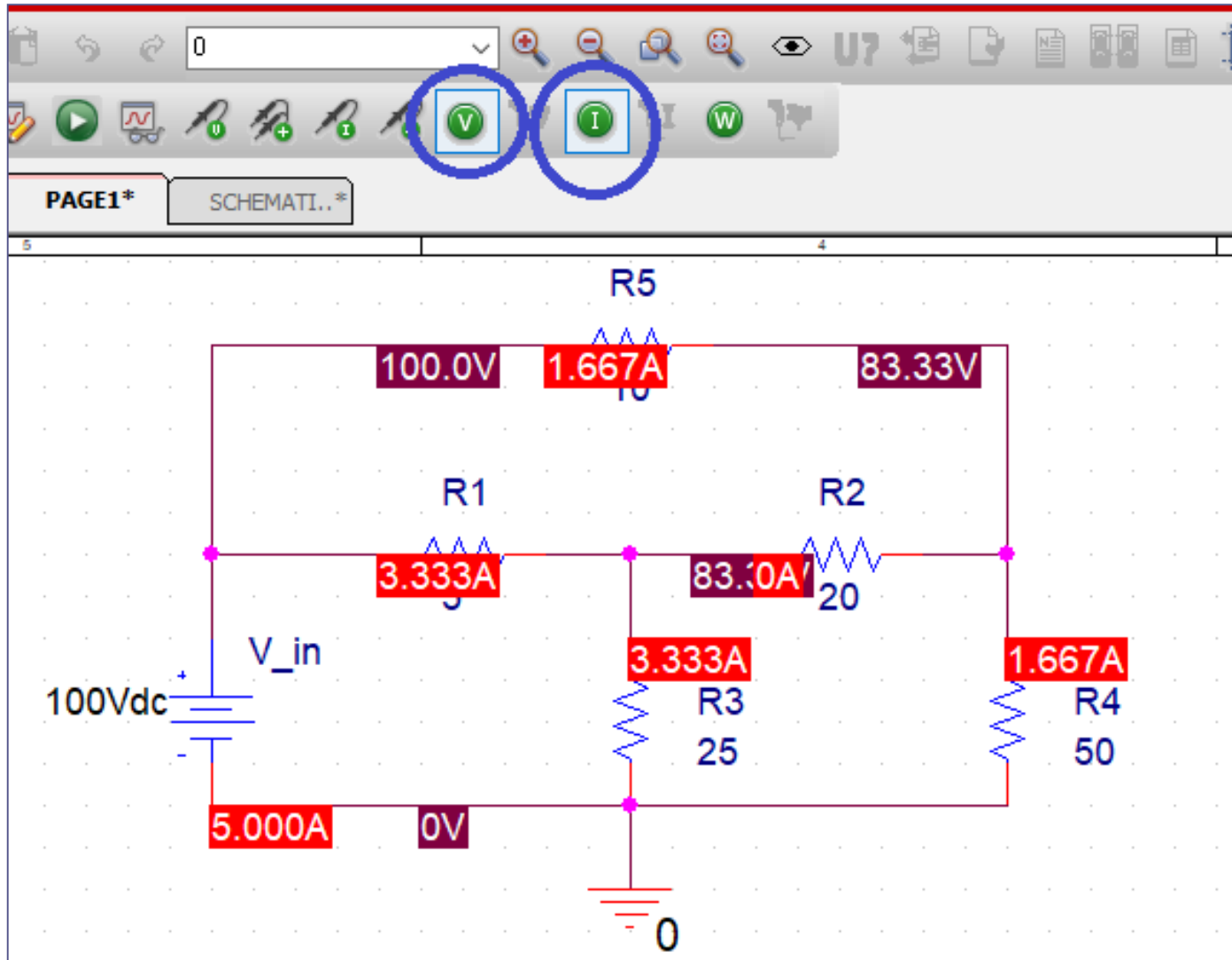
Maximize the SCHEMATIC1_Class-1_profile-Pspice A/D Lite window

The screenshot displays the PSpice A/D Lite software interface. The title bar reads "SCHEMATIC1_Class_1_profile - PSpice A/D Lite". The menu bar includes "File", "View", "Simulation", "Tools", "Window", and "Help". The toolbar contains various icons for file operations and simulation. The main workspace is a large gray area with the text "Ensure the 100% successful simulation" in red. Below the workspace, there are three panels: a left panel for "Analysis / Watch / Devices", a central console window showing simulation logs, and a right panel for "Trace Color" and "Trace Name". The console window displays the following text:

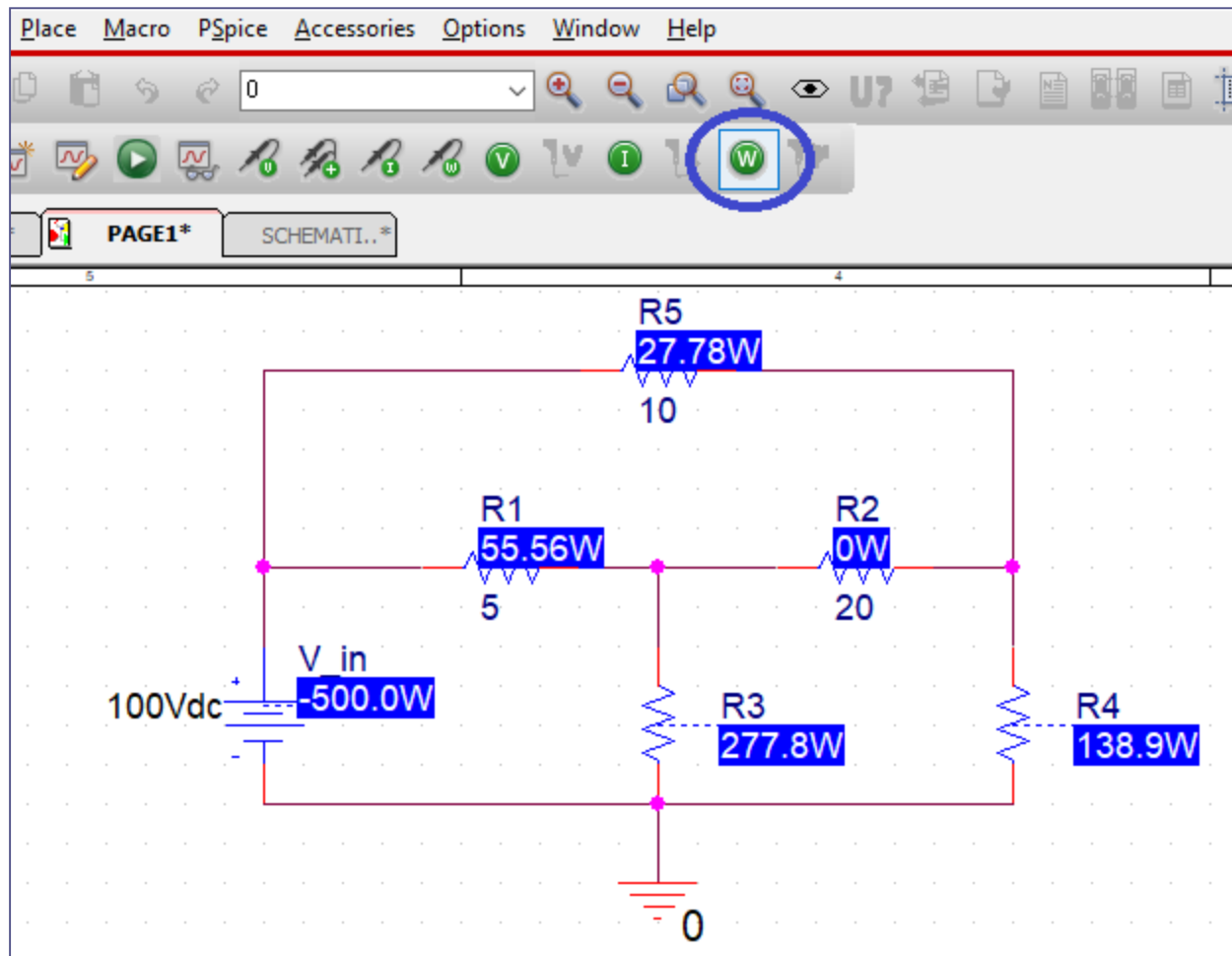
```
..... INFO(DRPROBE-3209): Simulation Profile: SCHEMATIC1_Class_1_profile ...
INFO(DRPROBE-3183): Simulation running...
*** Profile: "SCHEMATIC1_Class_1_profile" [ C:\ORCAD\ORCAD_16.5_LITE\TOOLS\
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point
Bias point calculated
INFO(DRPROBE-3190): Simulation complete
```

The right panel shows a table with columns "Trace Color", "Trace Name", "Y1", "Y2", and "Y1 - Y2". The "Trace Name" column contains "X Values". At the bottom right, a progress bar is shown at 100% completion, with a green bar and a small waveform icon. A blue arrow points from the main workspace text to the progress bar. The Windows taskbar at the bottom shows the system tray with a 95% battery level, the date 05-10-2020, and the time 17:08.

7. Observe the voltages and currents at different points of the network:



7.1. Observe the power generated by the source and power dissipated across each element





THANK YOU