# 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<br>a. Single lamp<br>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 |

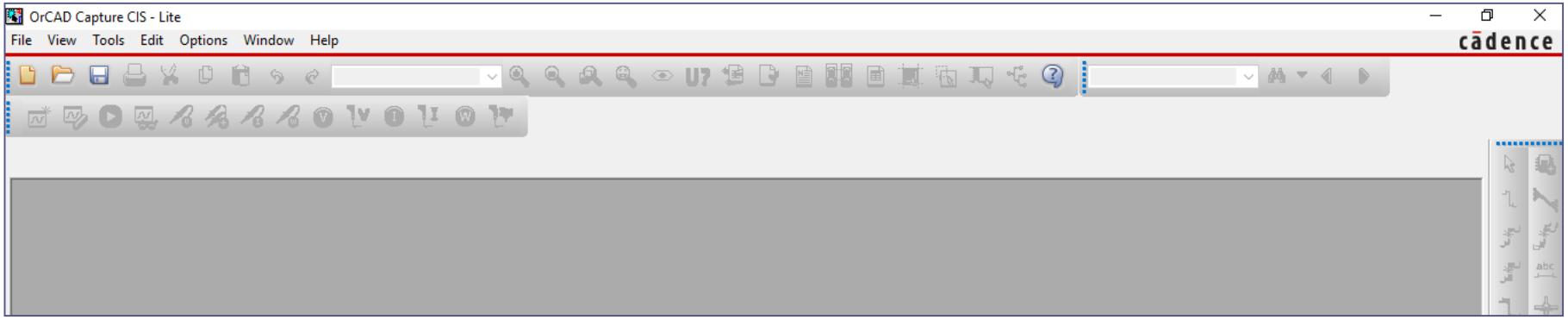
# *Introduction*

# Getting Started with OrCAD Pspice Software

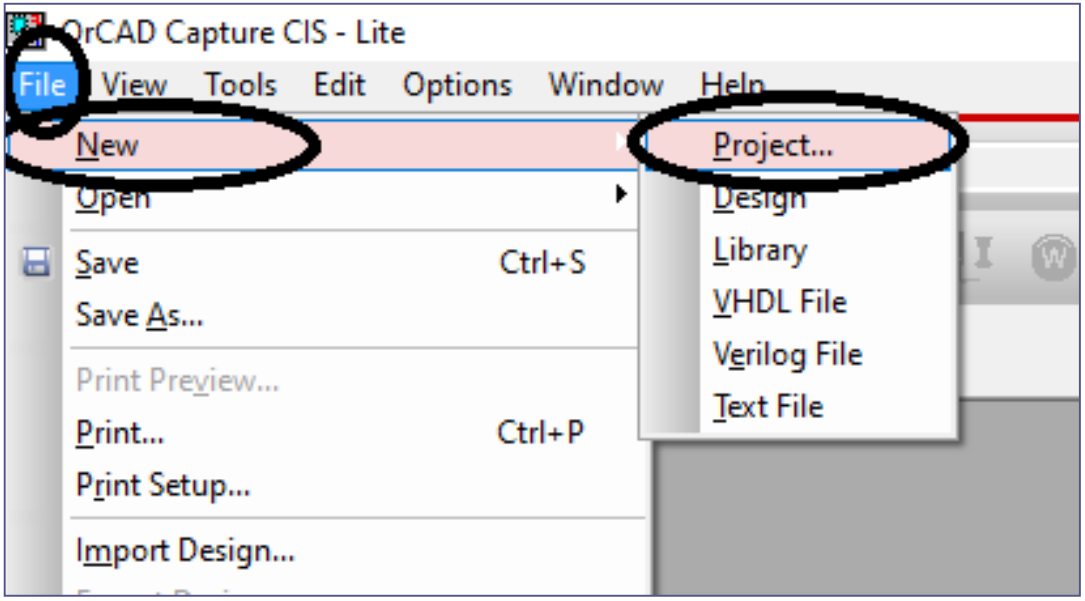
# *Circuit diagram for introduction to ORCAD*

# Getting started with ORCAD Pspice software

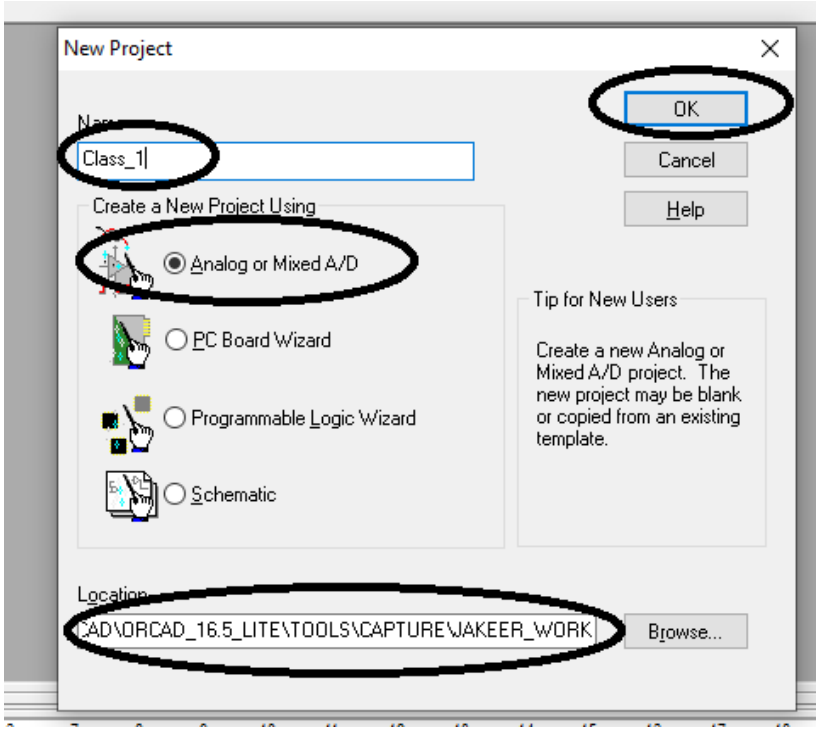
## 1. Start the application:



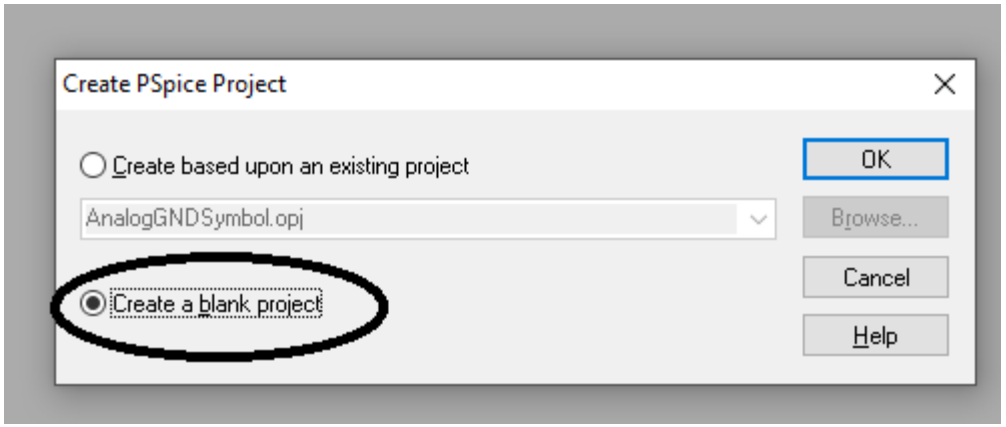

## 2. Start New Project:



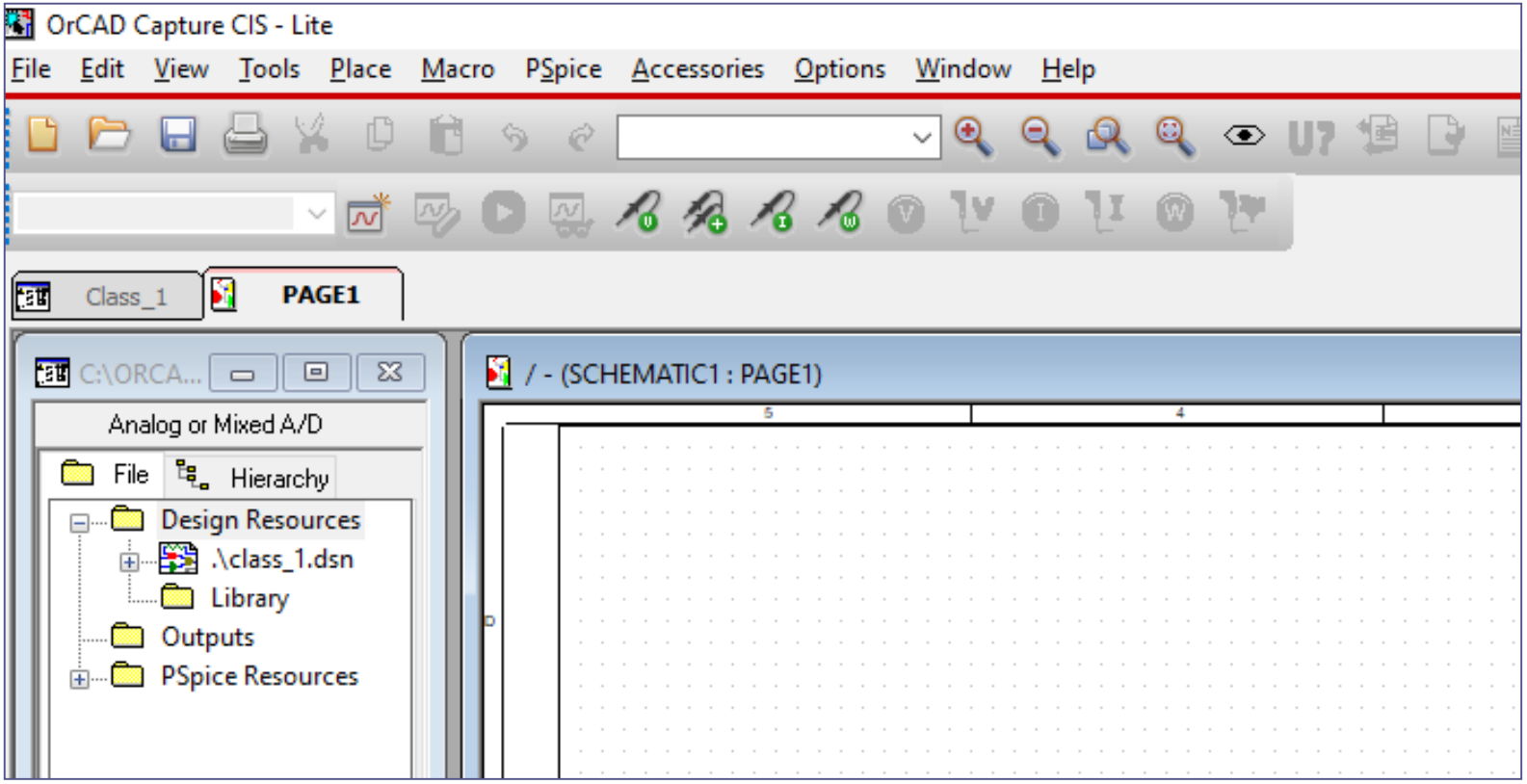
# 3. Give name to the project and create new project using Analog or mixed A/D




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

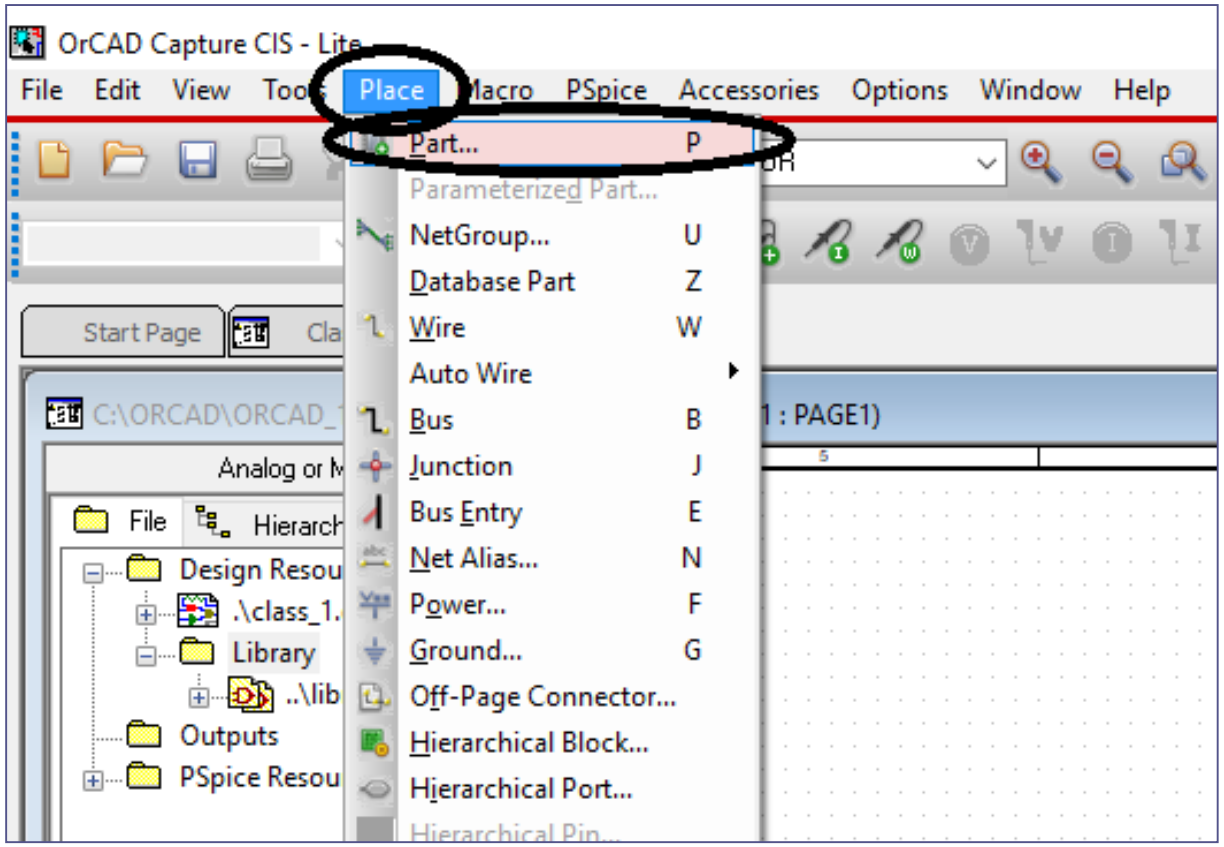


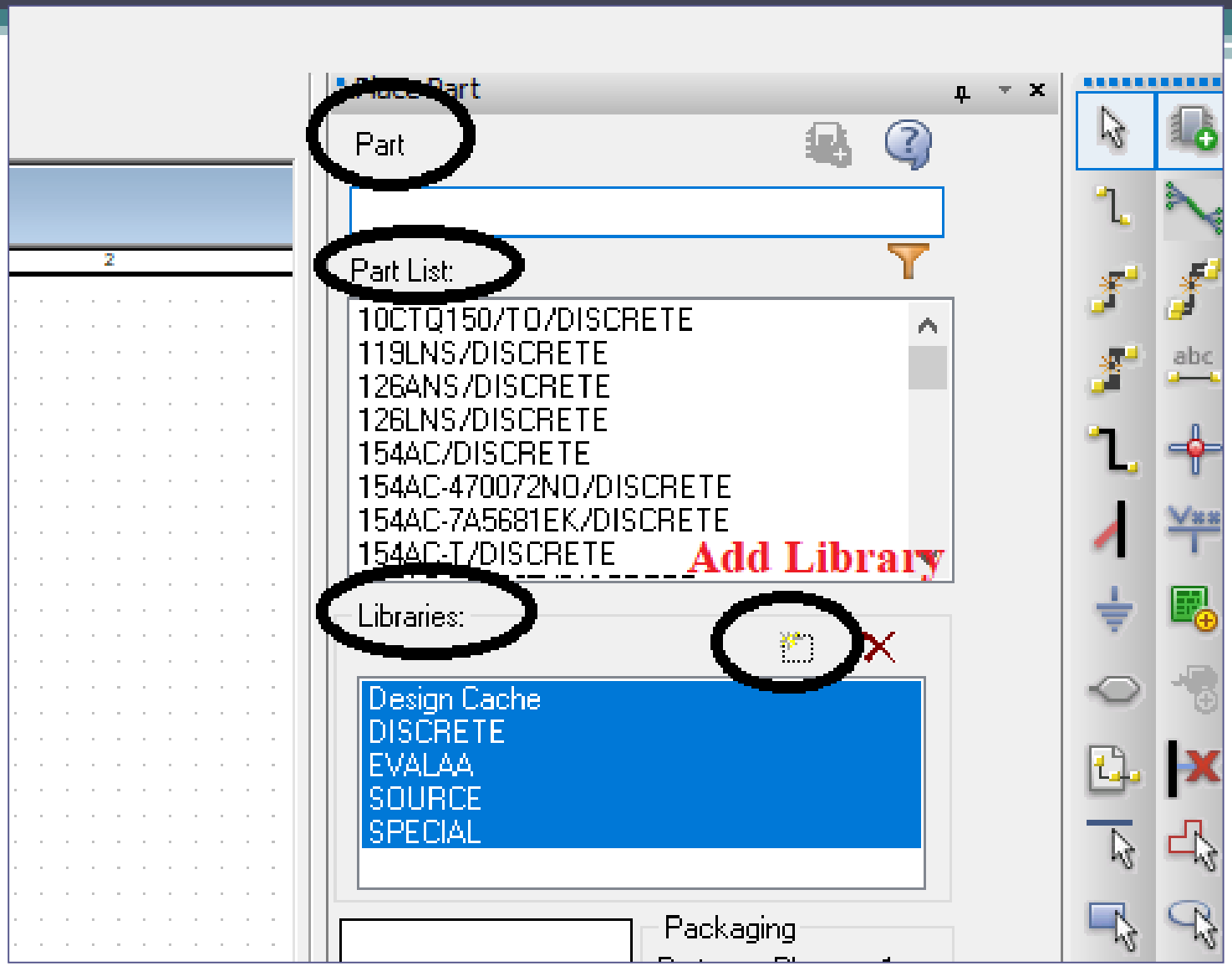
# 4. The editor window where the circuit diagram can be drawn:



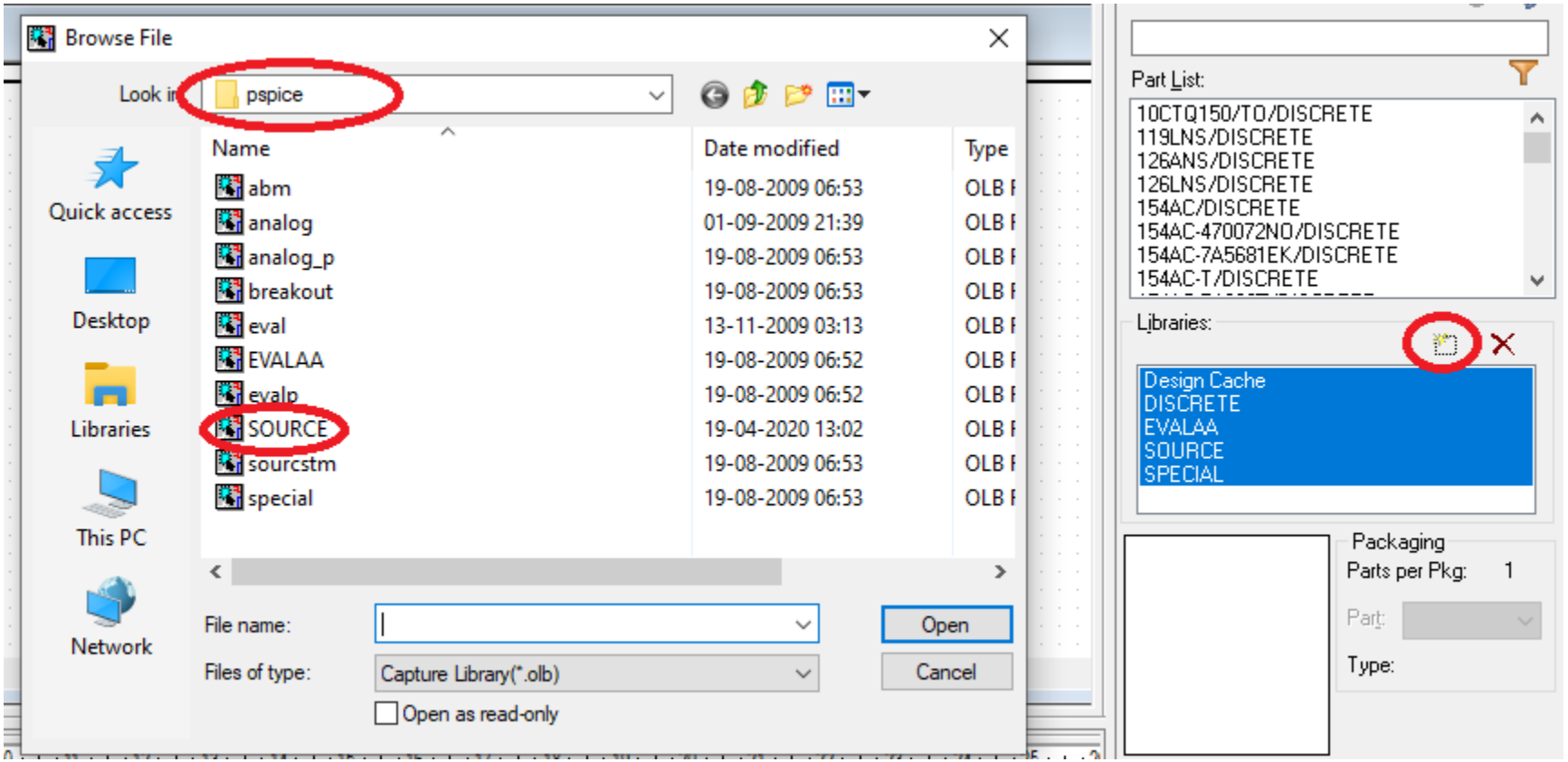
# 5. Start drawing the schematic (circuit diagram)

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




Part
Part List:
10CTQ150/TO/DISCRETE
119LNS/DISCRETE
126ANS/DISCRETE
126LNS/DISCRETE
154AC/DISCRETE
154AC-470072NO/DISCRETE
154AC-7A5681EK/DISCRETE
154AC-T/DISCRETE
Add Library
Libraries:
Design Cache
DISCRETE
EVALAA
SOURCE
SPECIAL
Packaging

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

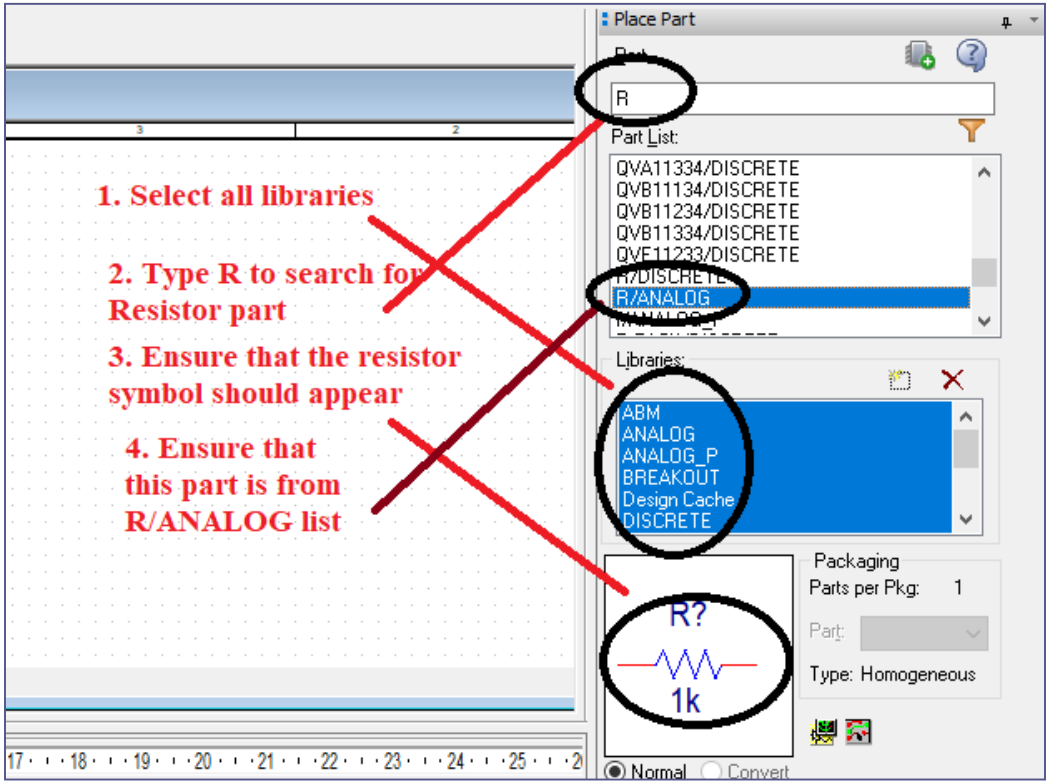



Ensure that the adding libraries are under the pspice folder

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)

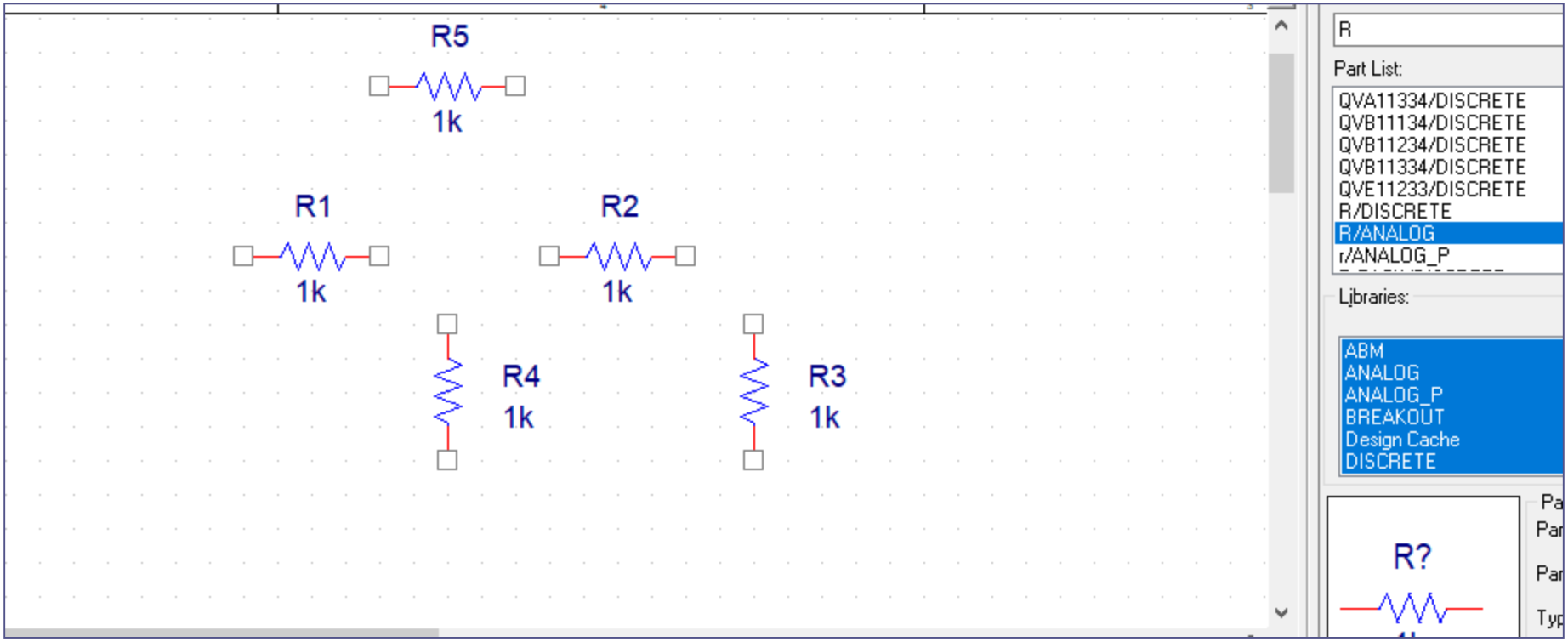



Double click on R/ANALOG and place the resistor on the editor page at preferred place by moving the cursor. You can place multiple Resistor parts on the schematic page.

After placing required no of resistor parts at appropriate places,

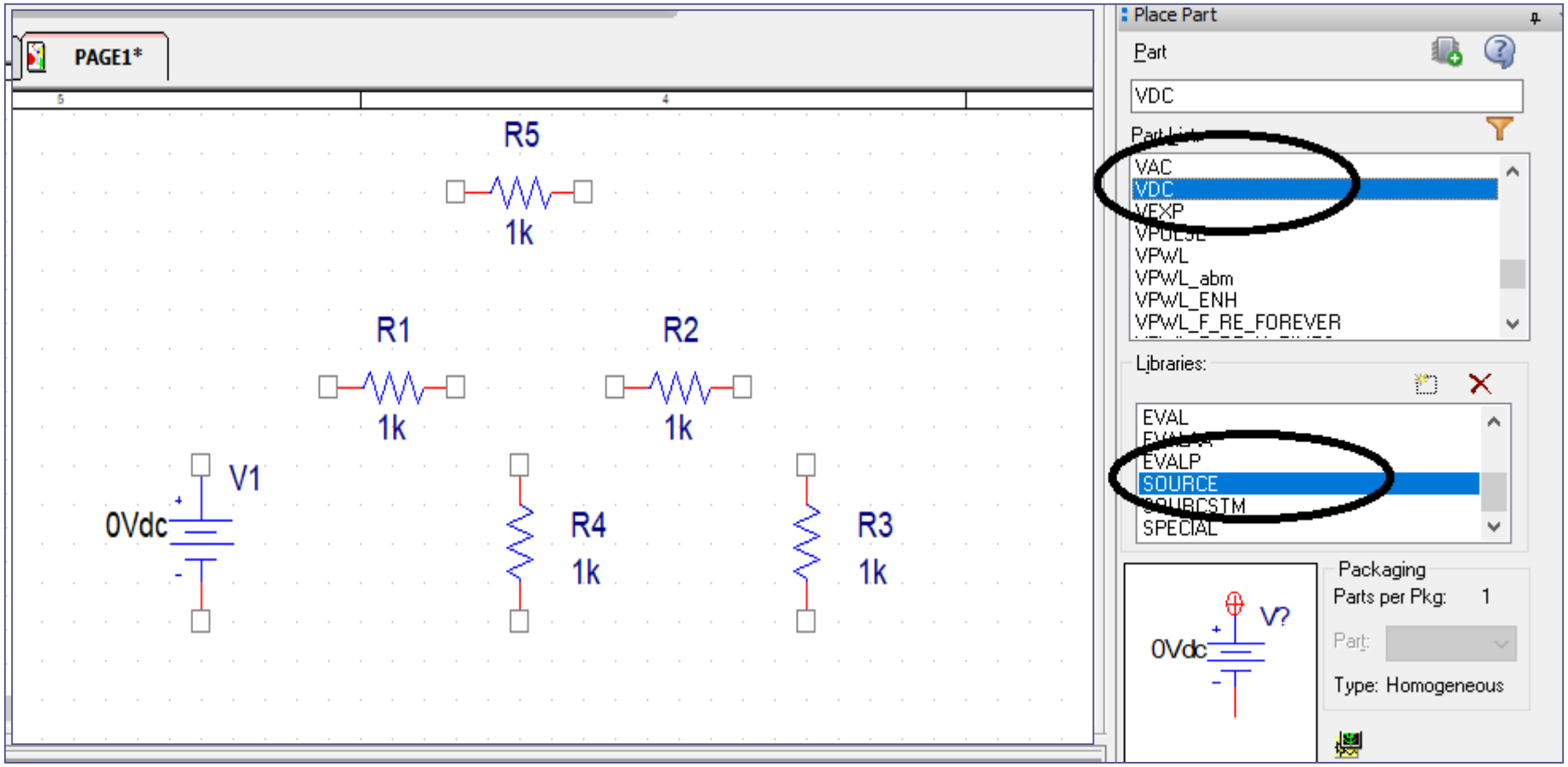
click the mouse (observe the change in Resistor's color)

press escape (or Right click mouse and select "End mode")

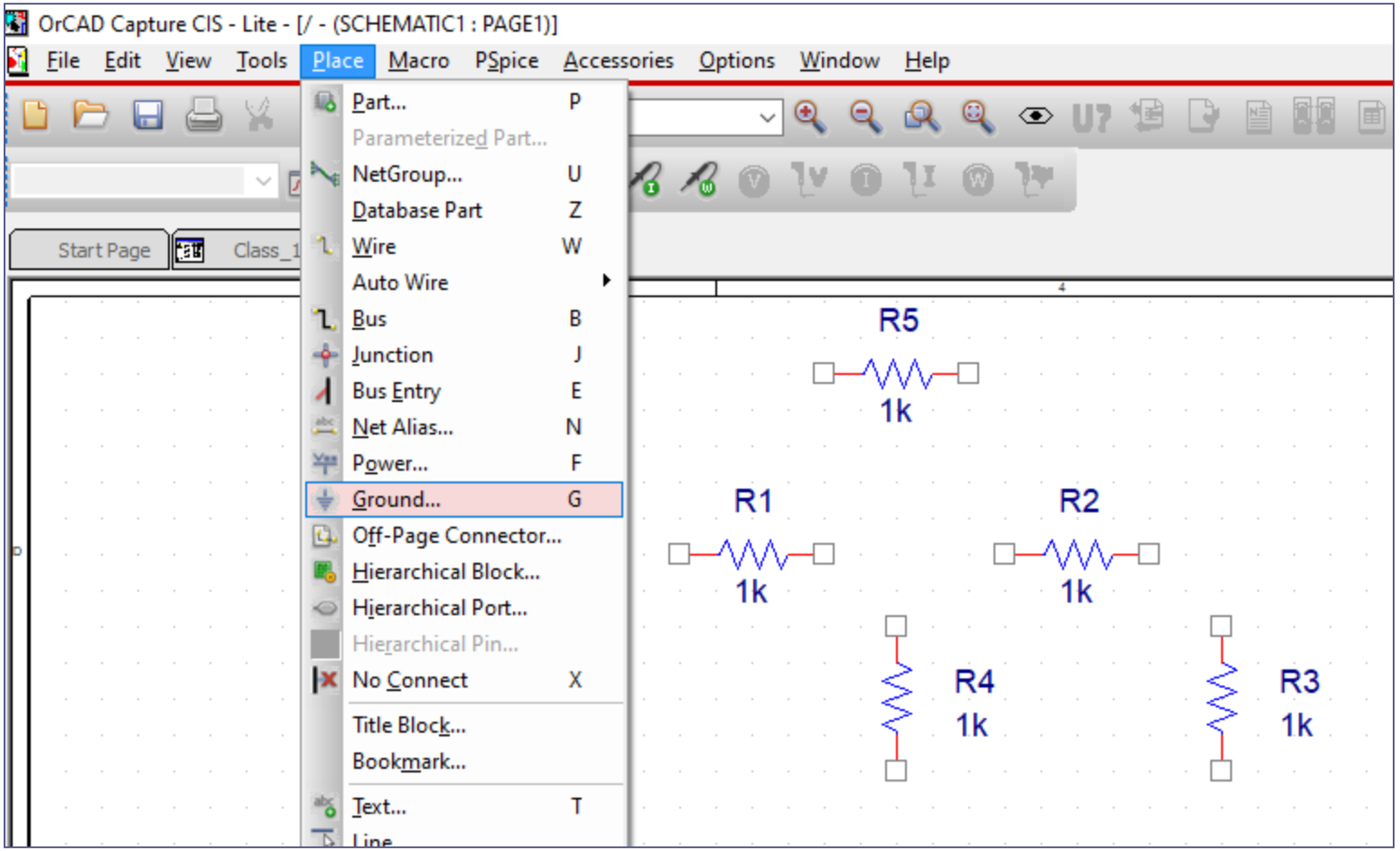


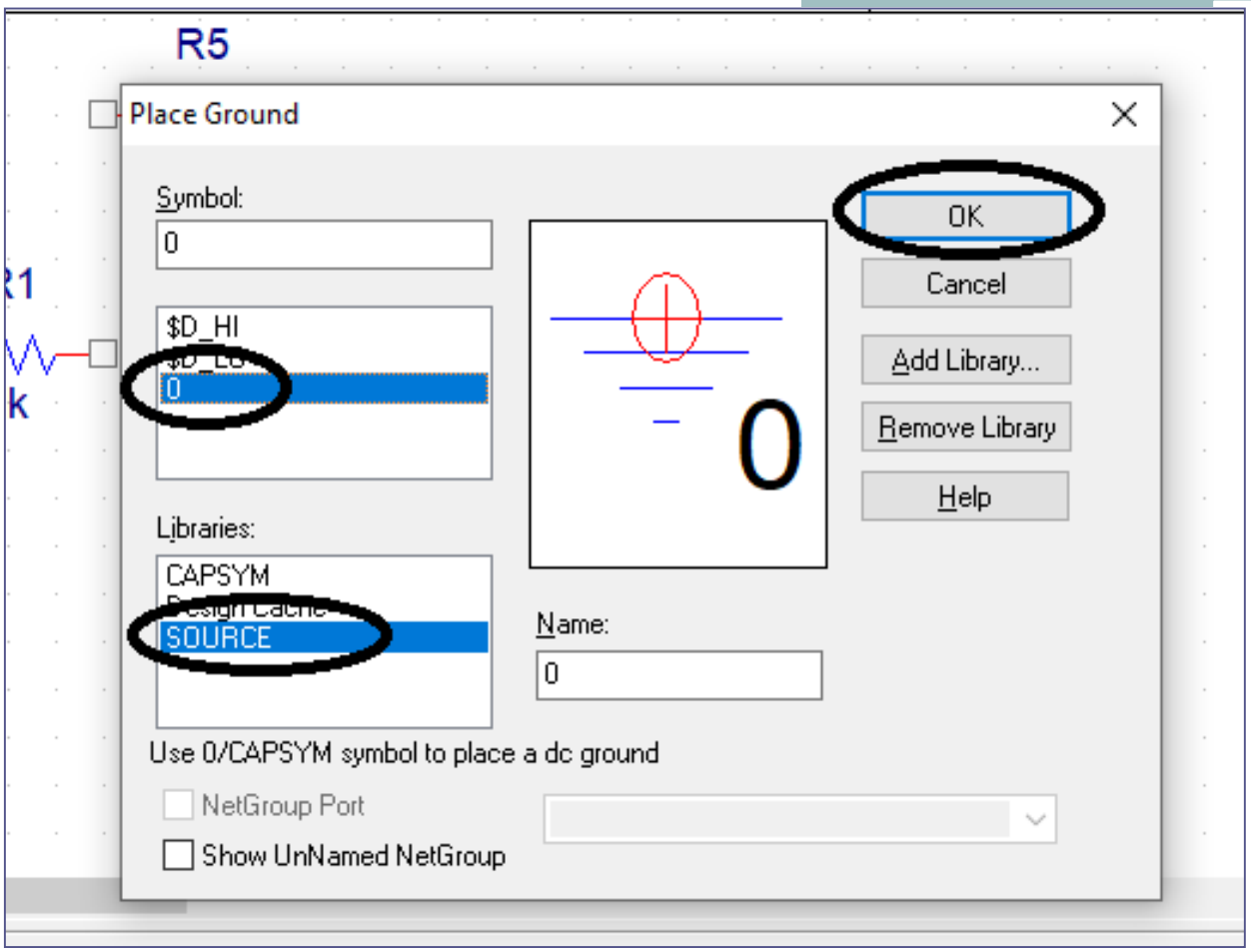
# 5.e . Place a voltage source part onto the schematic editor page

## Go to SOURCES >> VDC



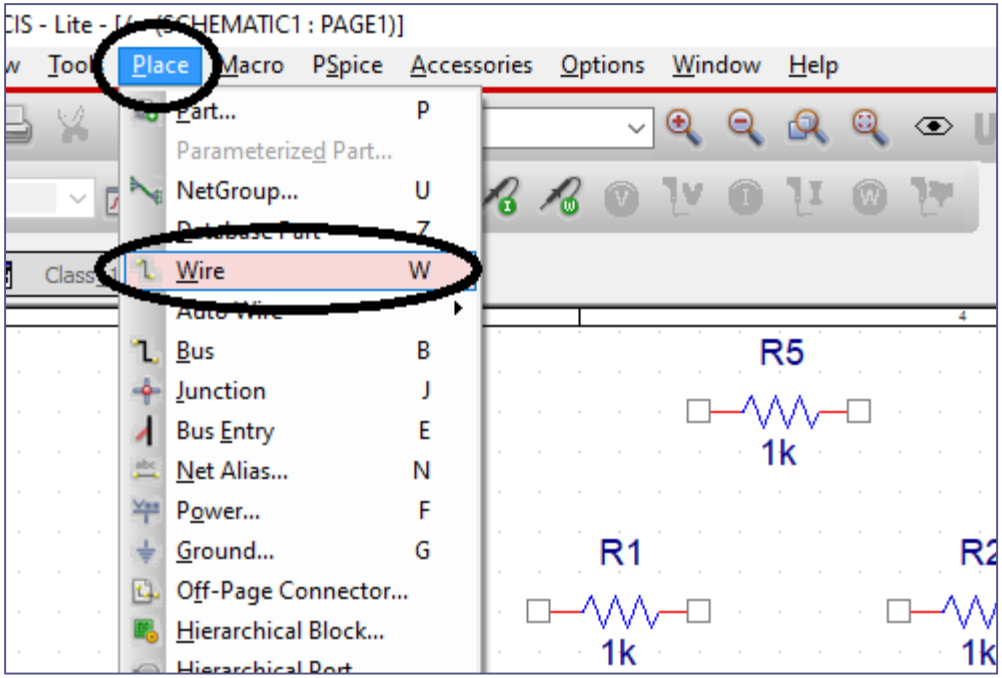
# 5.f . Place a ground part to the circuit schematic editor page






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.



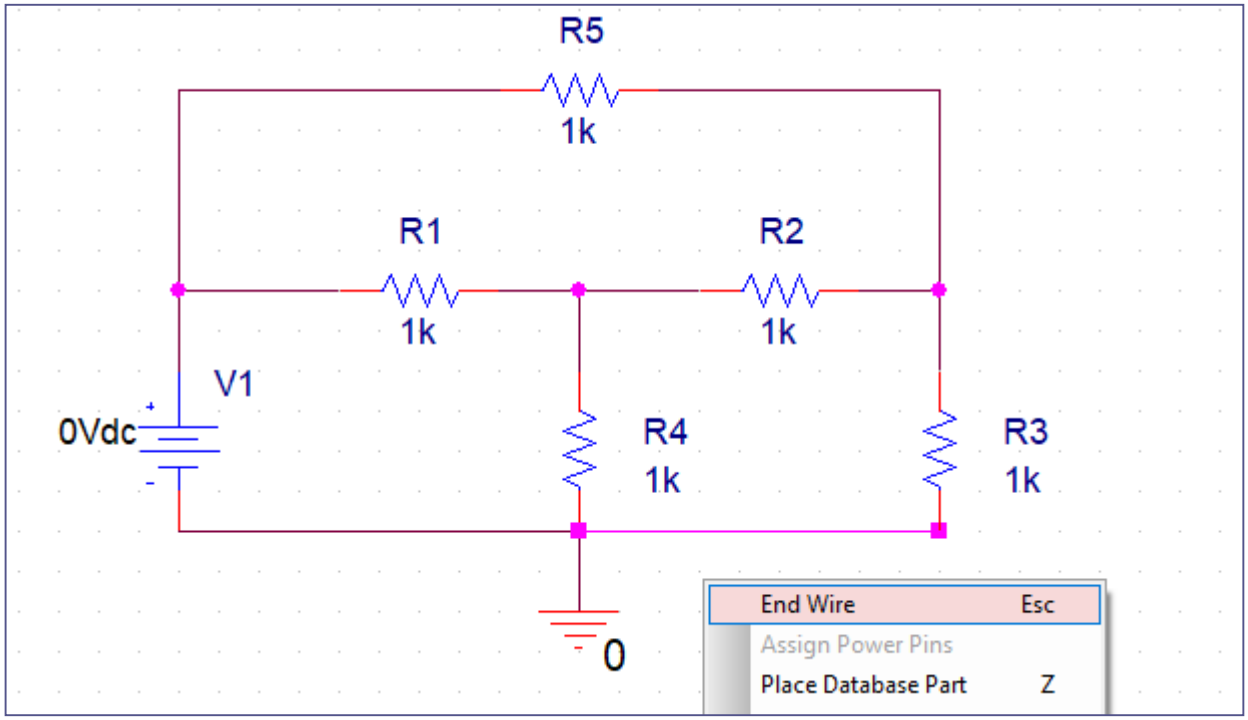

Connect all the circuit elements as per the circuit.

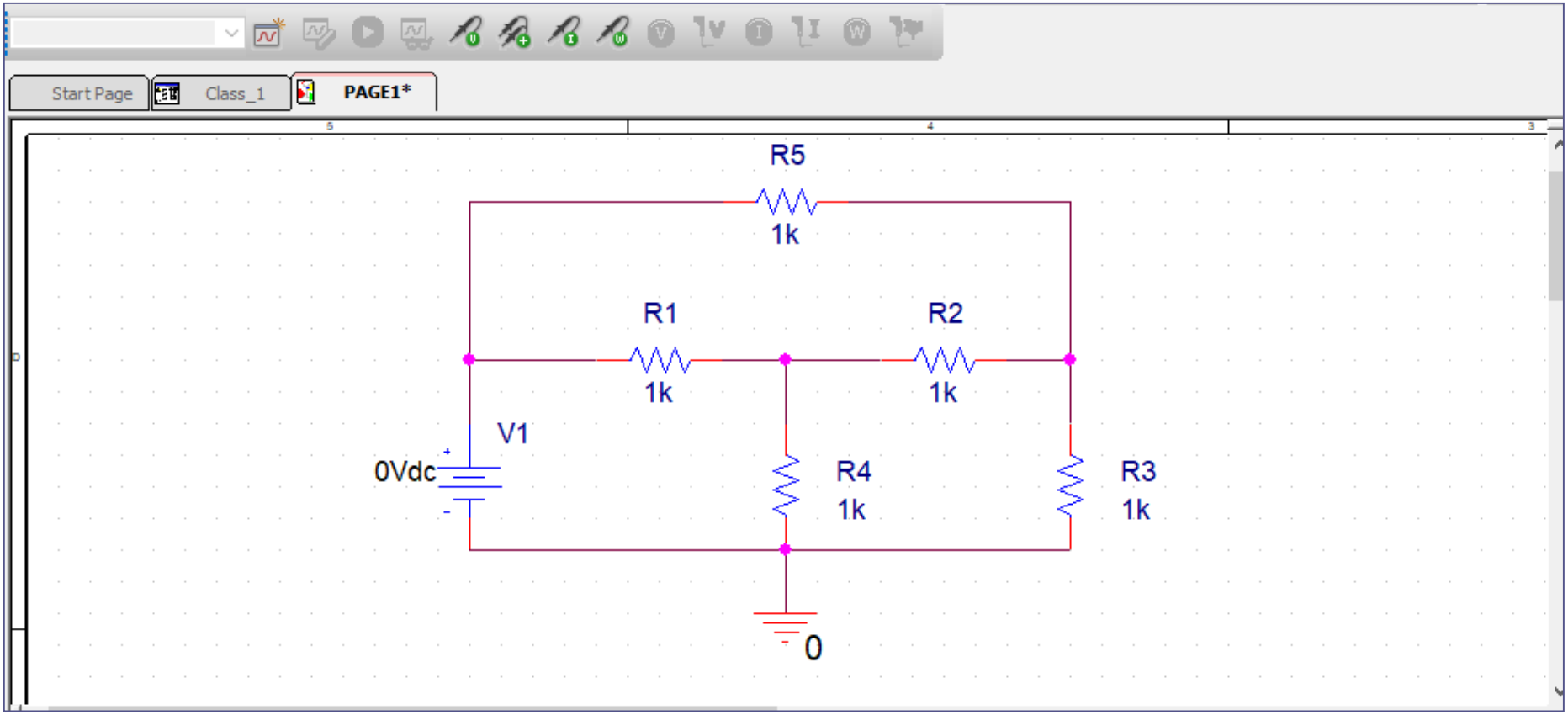
Select the wire part

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.

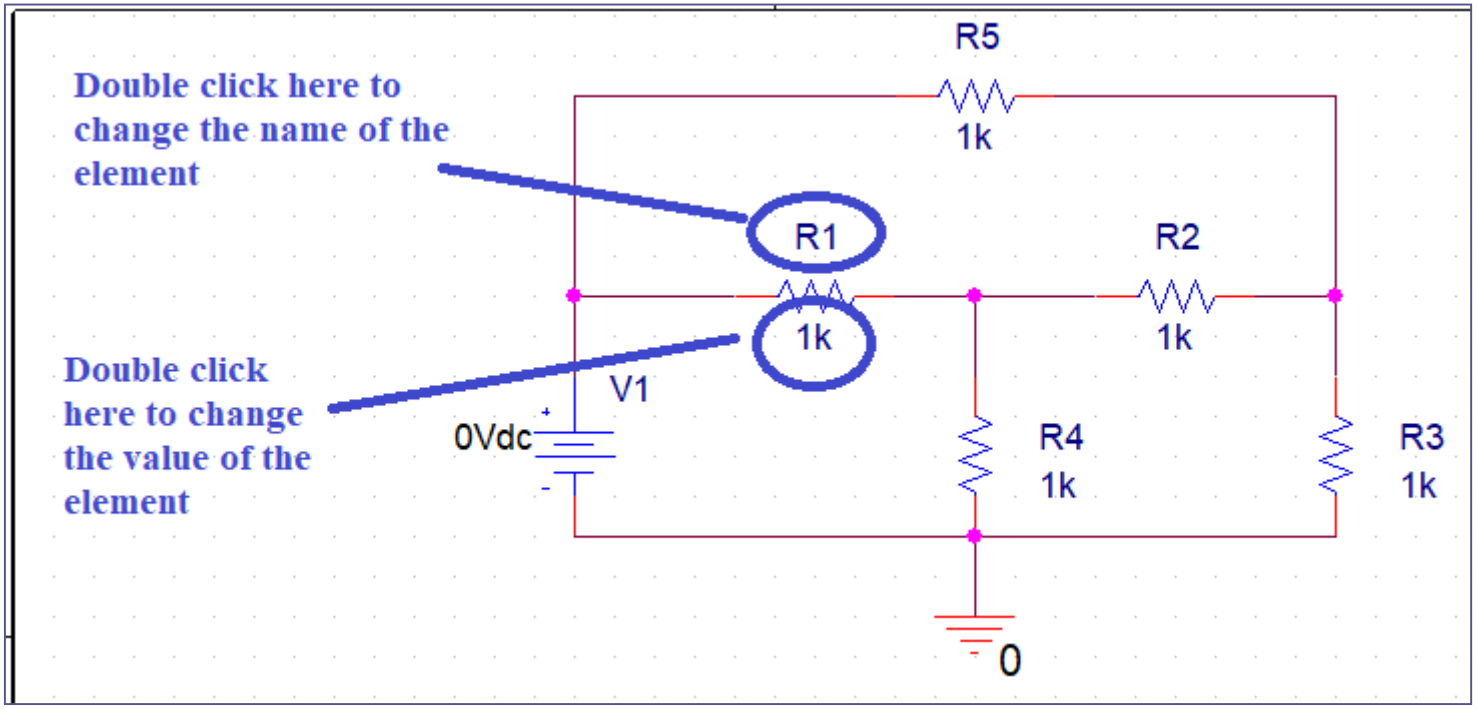
After completion of all the connections, then end the connection mode by clicking at “End wire”.



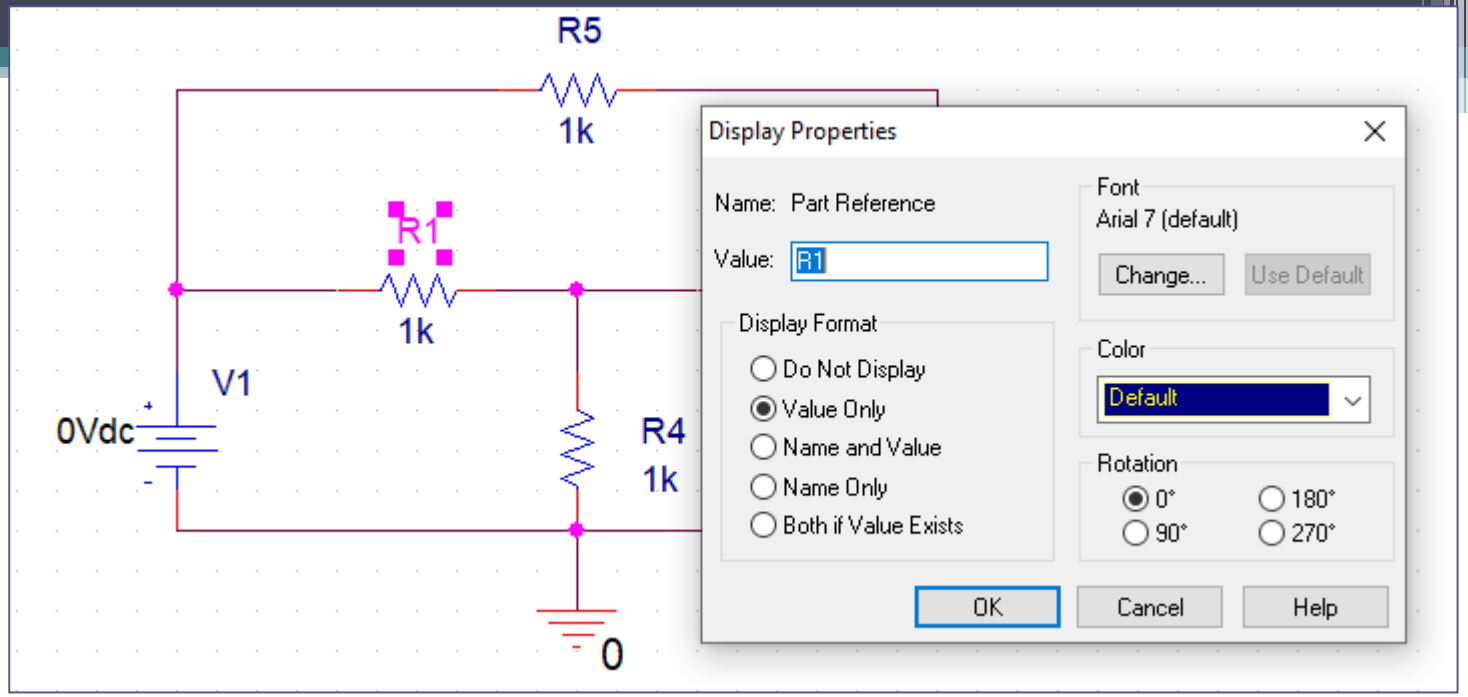

Start Page
Class_1
PAGE1*
R5
1k
R1
1k
R2
1k
V1
0Vdc
R4
1k
R3
1k
0

# 5.g . Edit the values of the components

To edit the Resistor element:

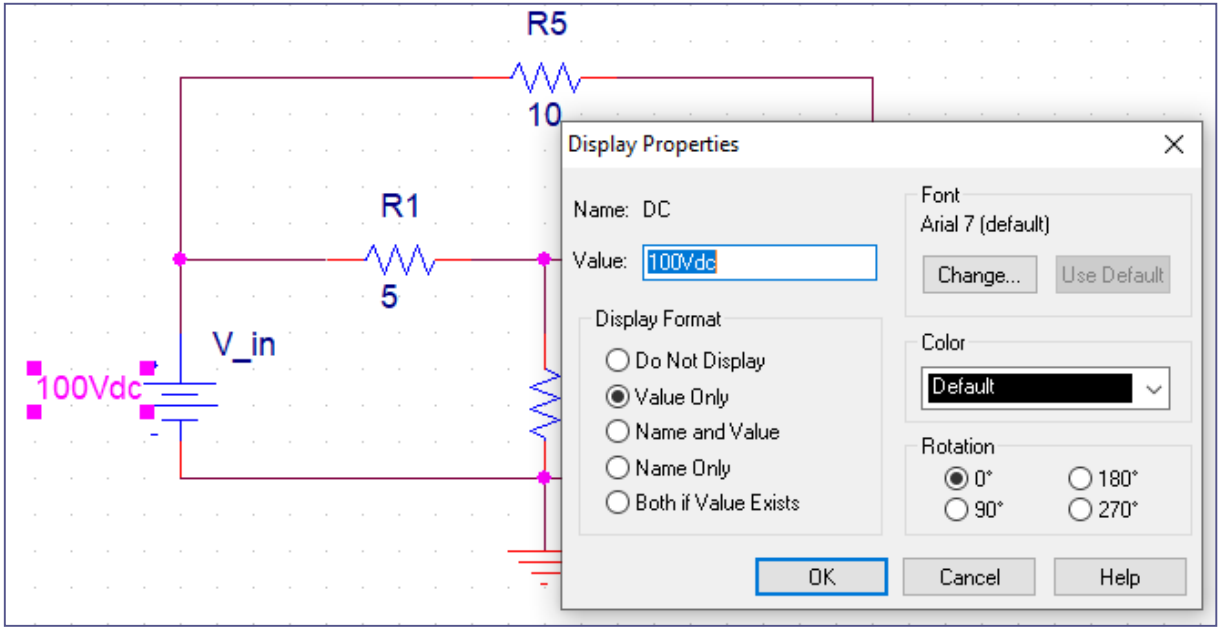


**To edit the name:**

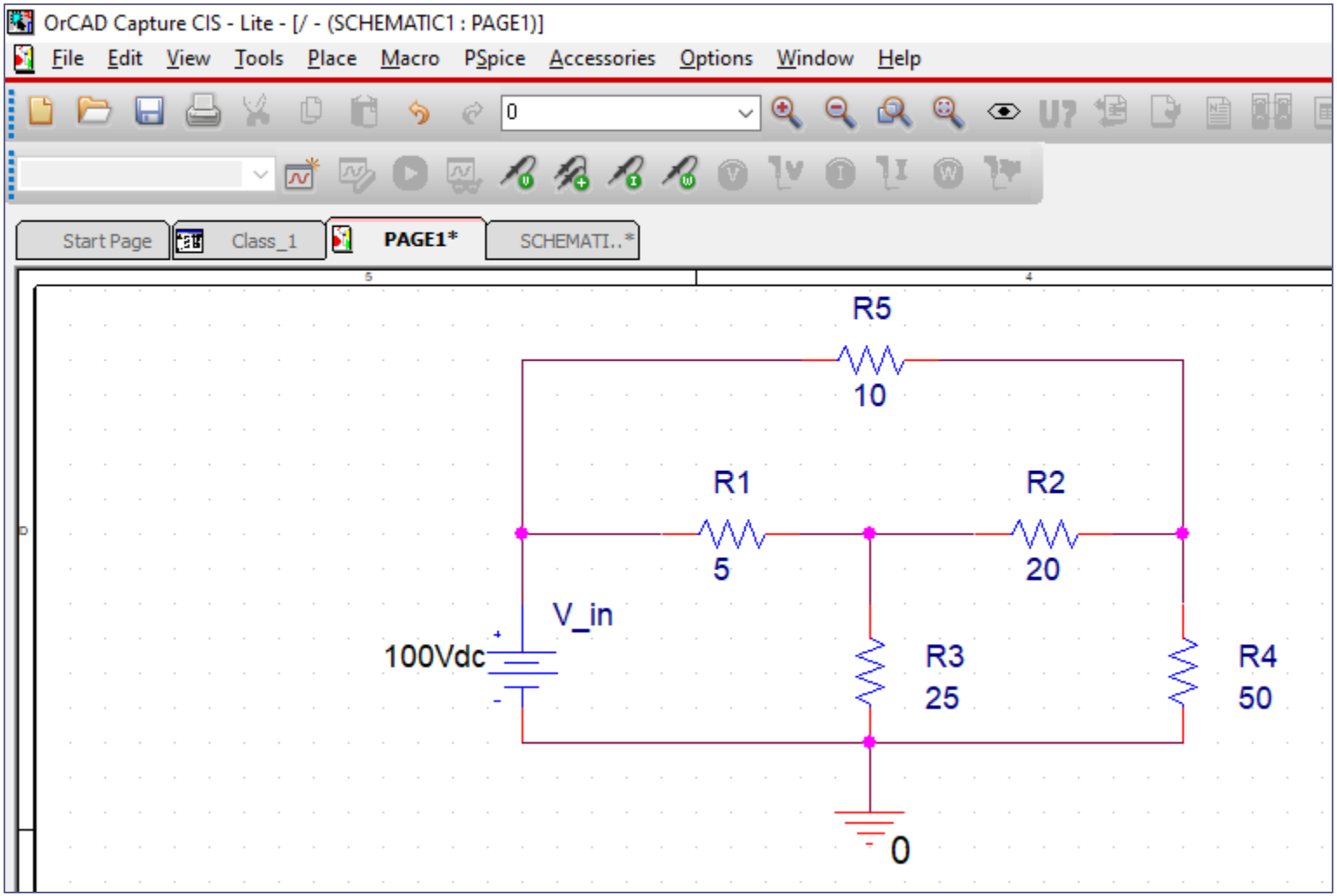



**To change the value:**

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

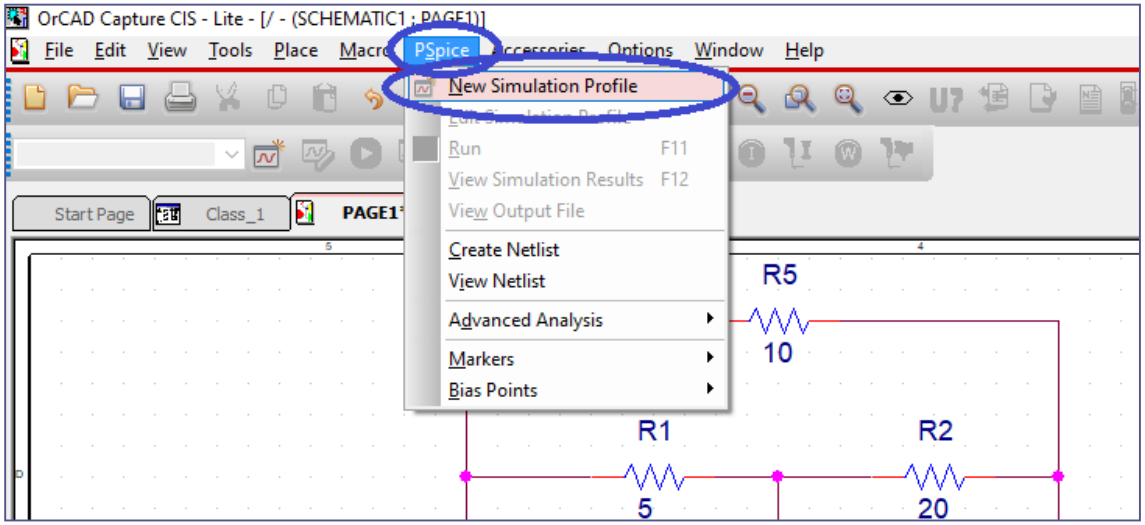


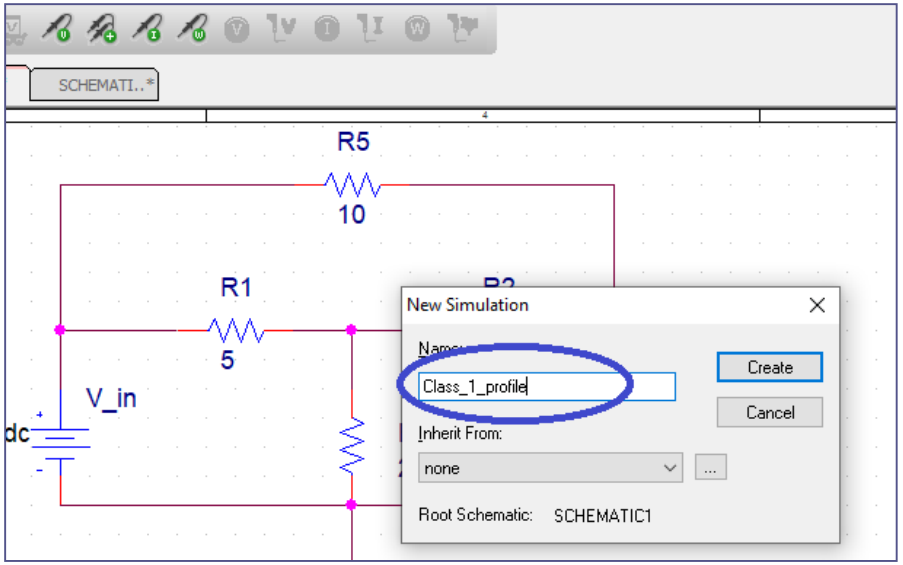
# 5. End of the schematic diagram drawing step



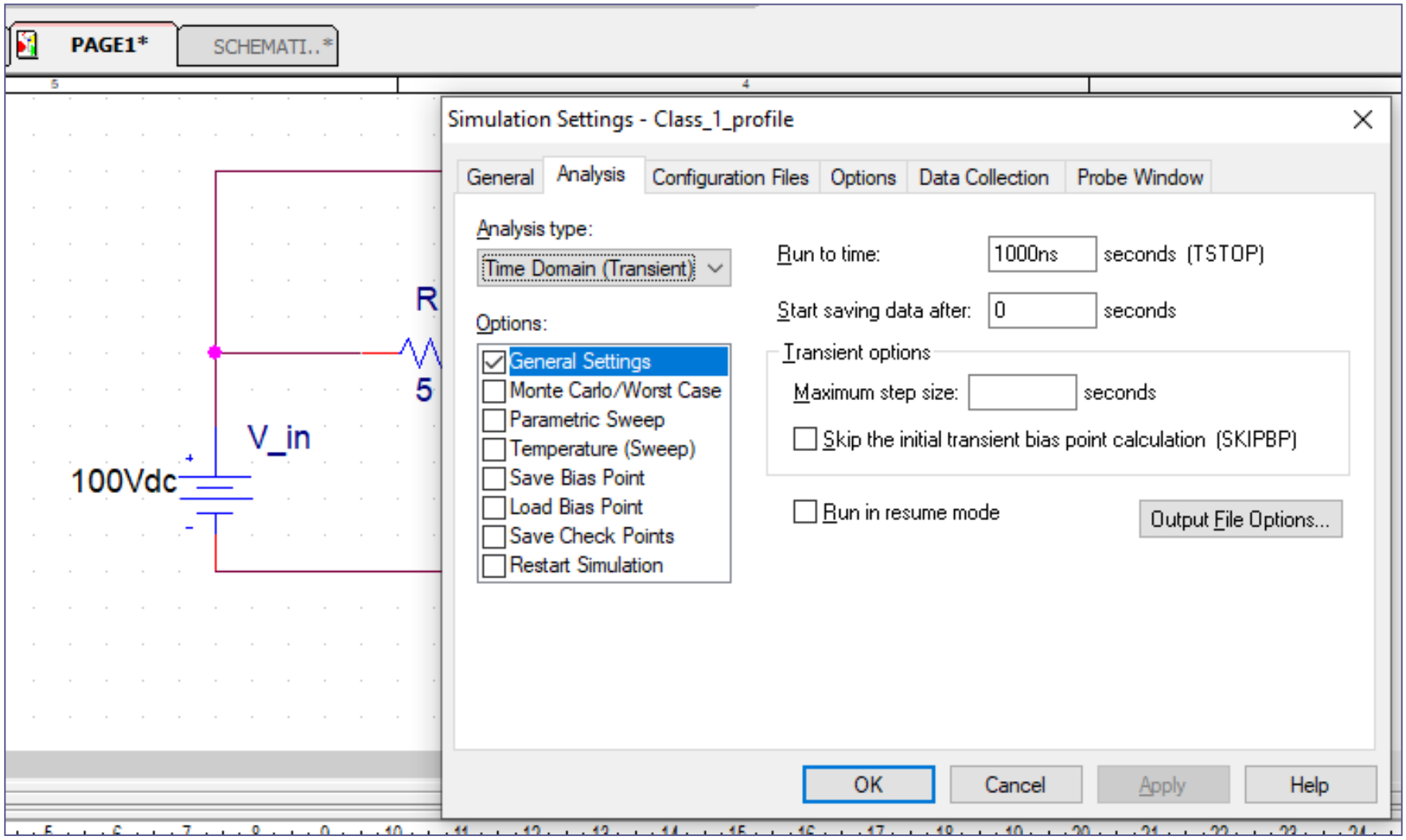
# 6. Simulate the circuit

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

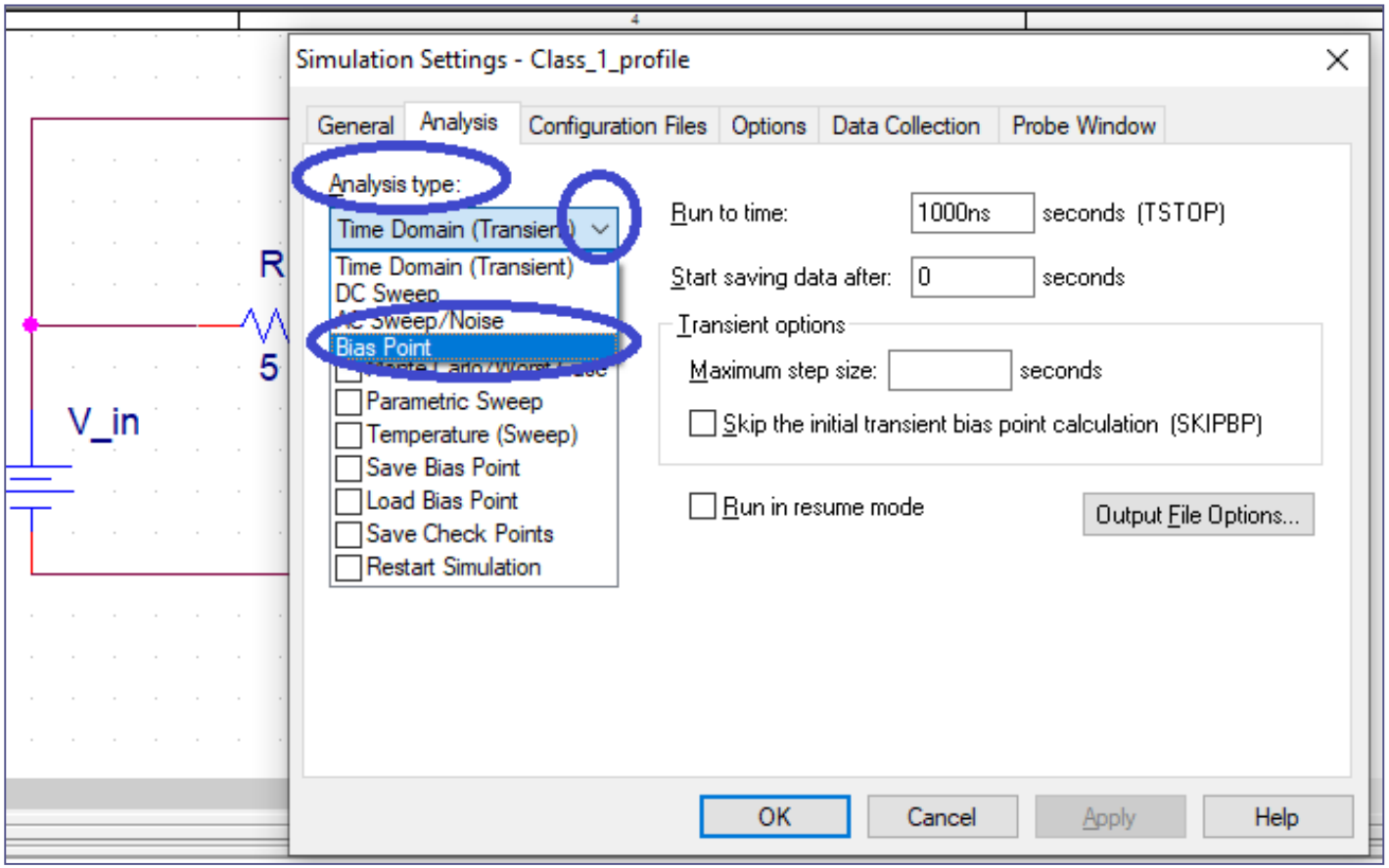


# 6.a. Maximize the “Simulation settings-Class_1_profile”

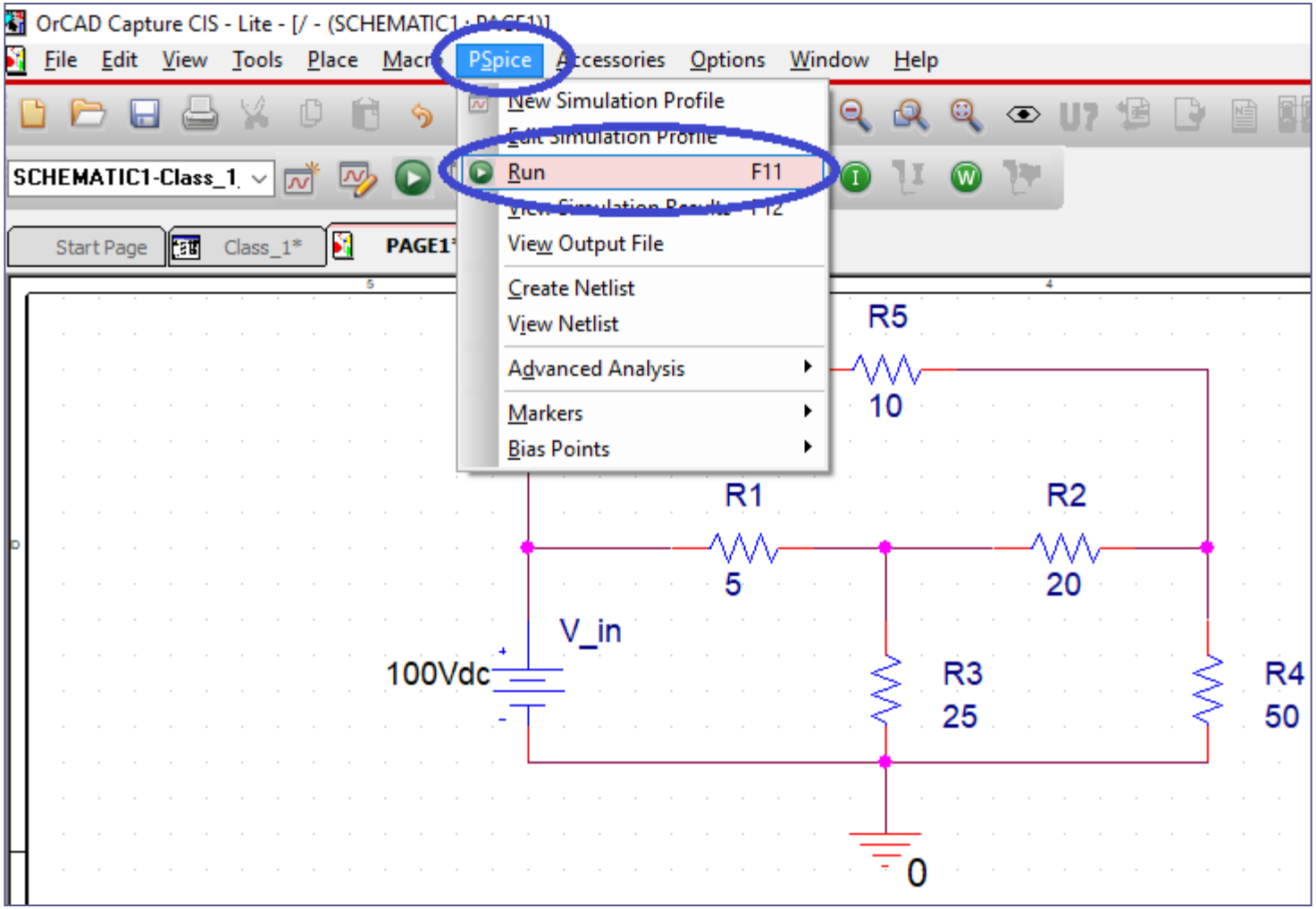


## 6.b. Select the type of the analysis you need to perform:

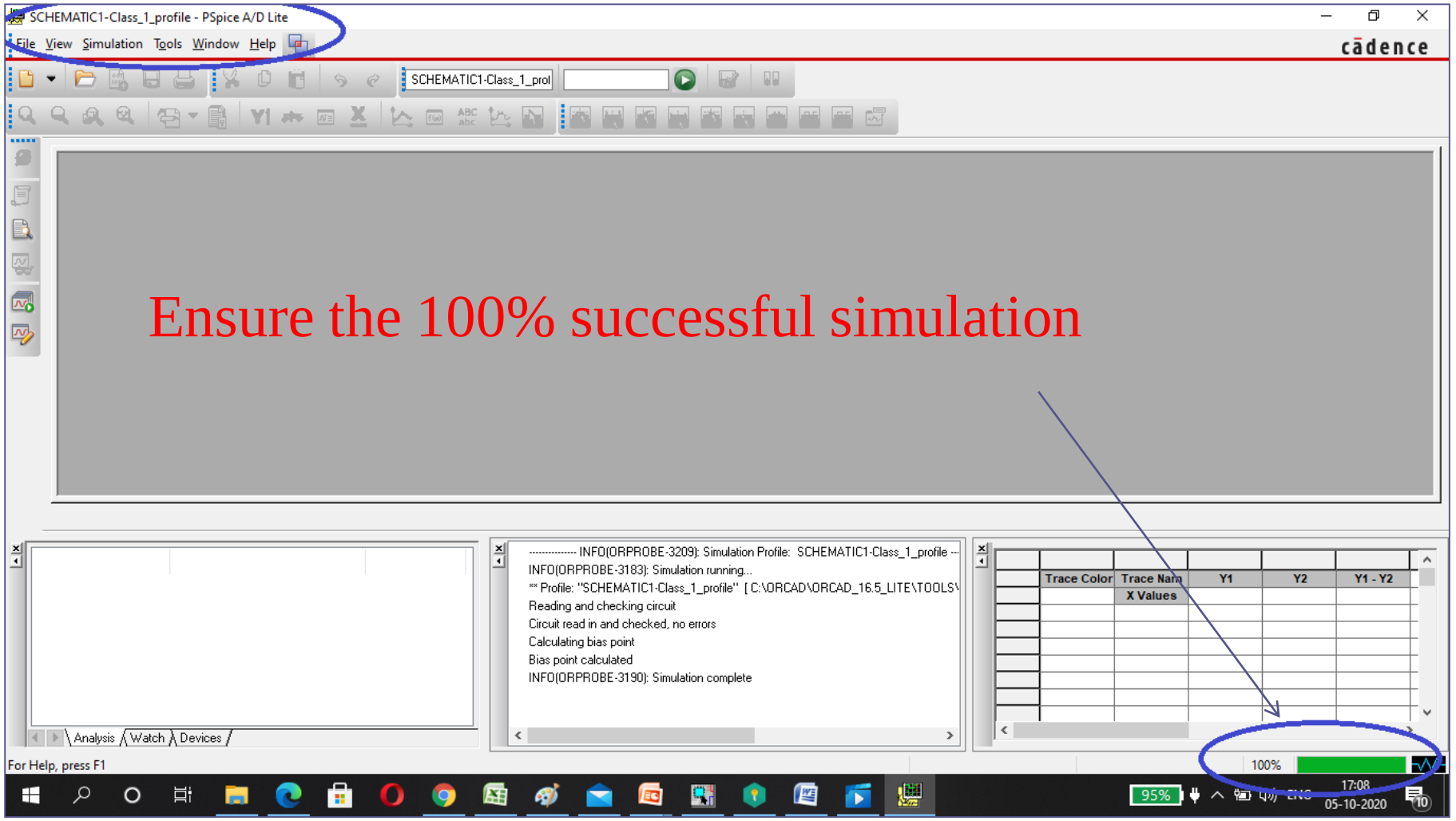



**For DC circuit analysis, Bias Point analysis is needed. This analysis will provide the steady state values of voltages and currents at various parts of the circuit.**

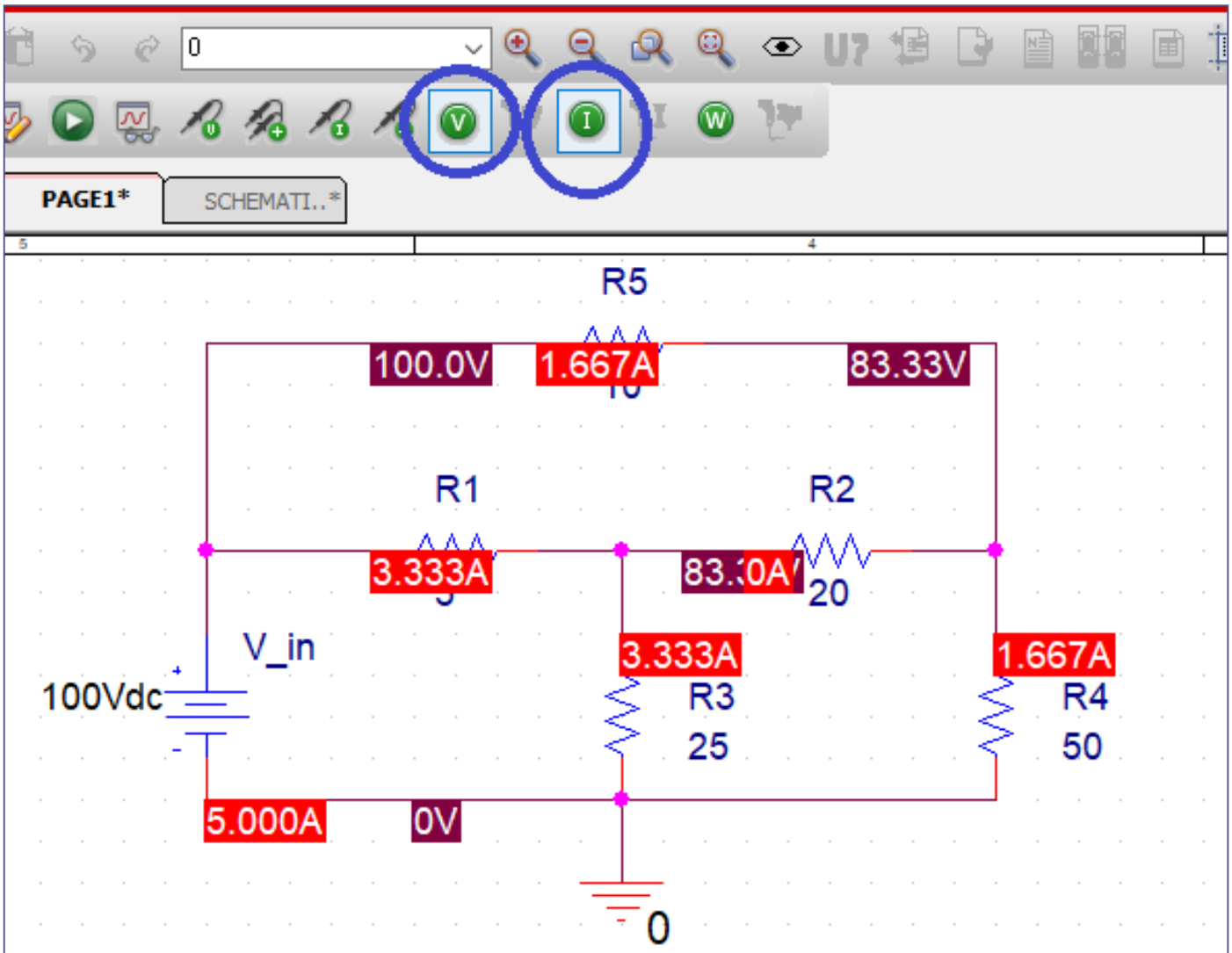
# 6.c. Run the circuit simulation



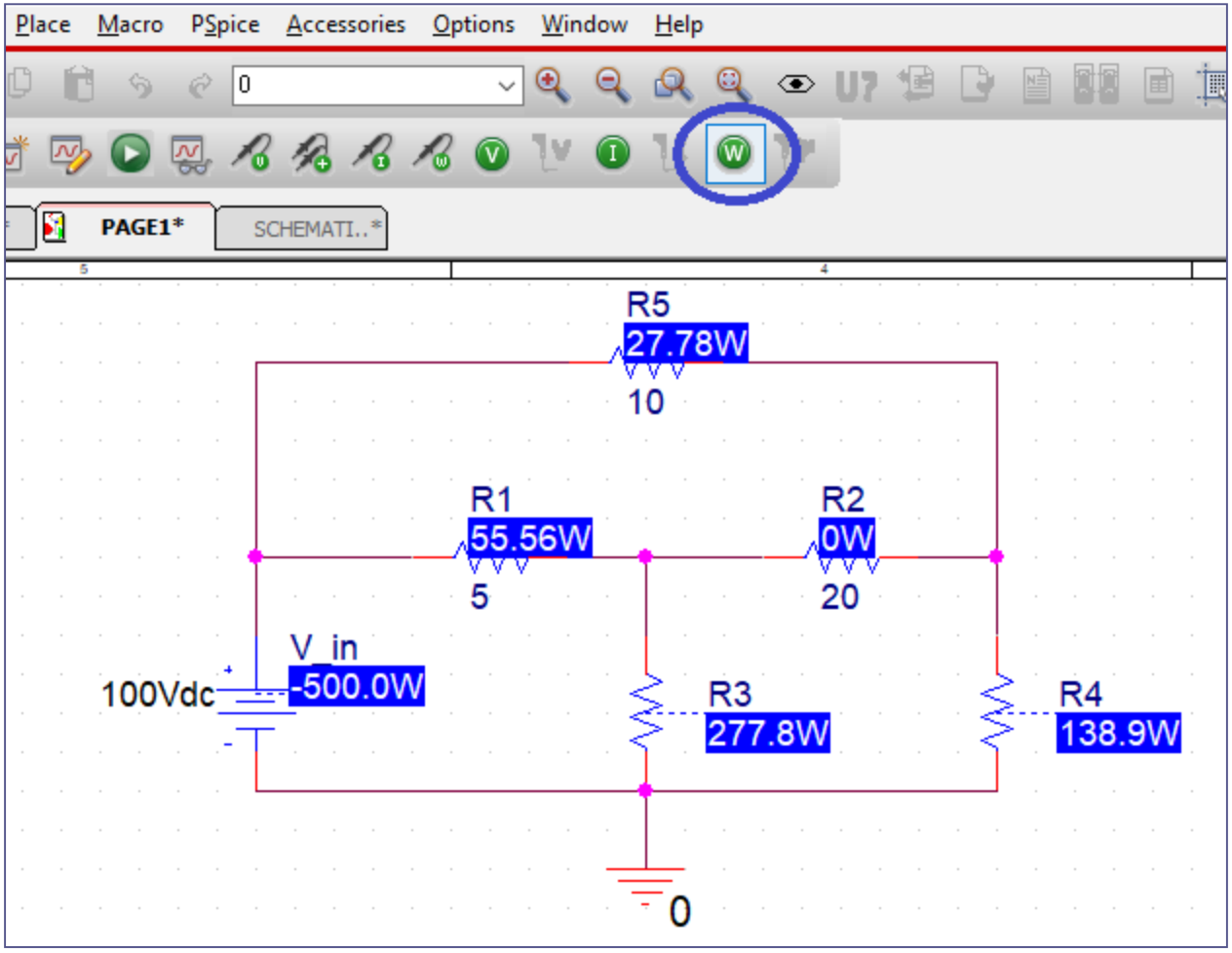
# Maximize the SCHEMATIC1_Class-1_profile-Pspice A/D Lite window



## 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