

Electric Circuits I – Laboratory 03

Ohms law verification using Pspice

#	Student ID	Student Name	Grade (10)	Instructor signature
1				

Delivery Date	
---------------	--

Procedure

Part 1 : build a simple circuit, and get it analyzed

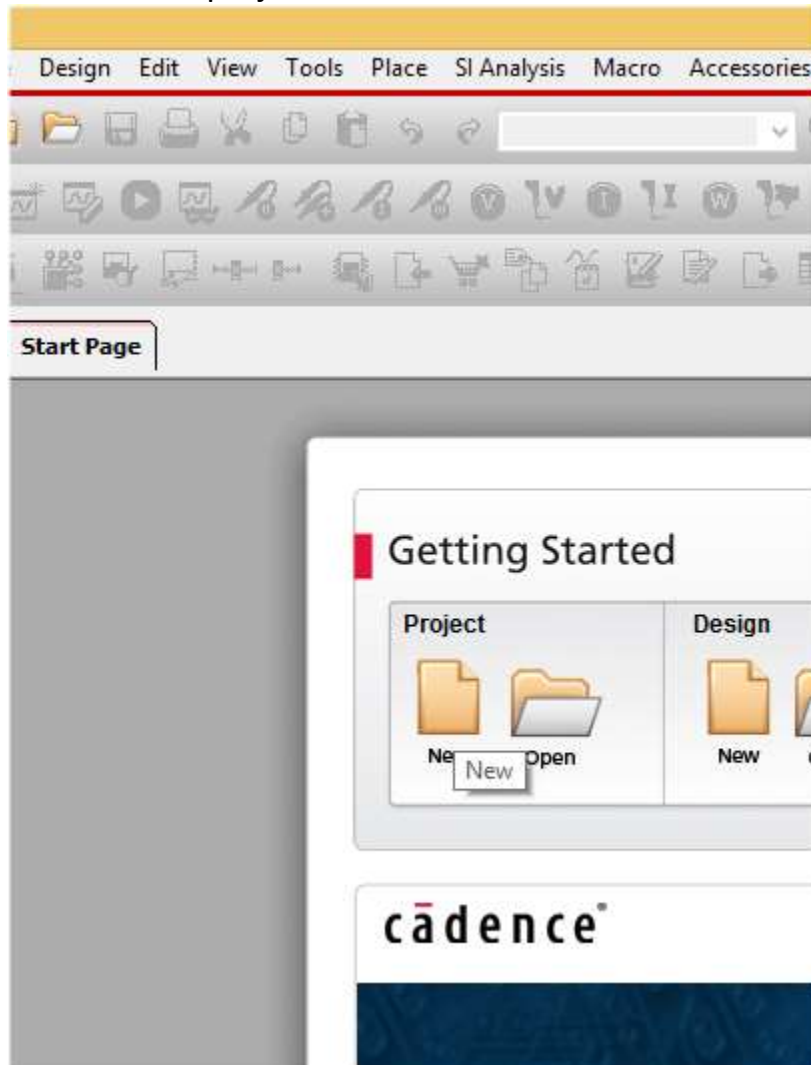
1. Install Cadence orcad lite all products 16.6 software in your PC after downloading it from the following link

<http://www.orcad.com/buy/try-orcad-for-free>

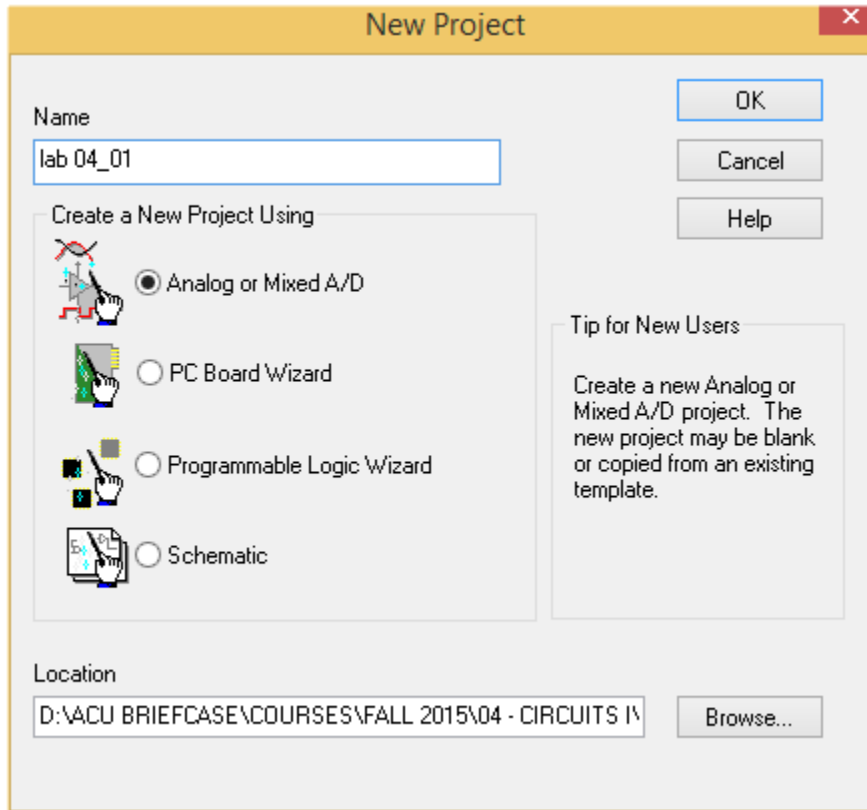
2. Open "OrCAD Capture CIS Lite"



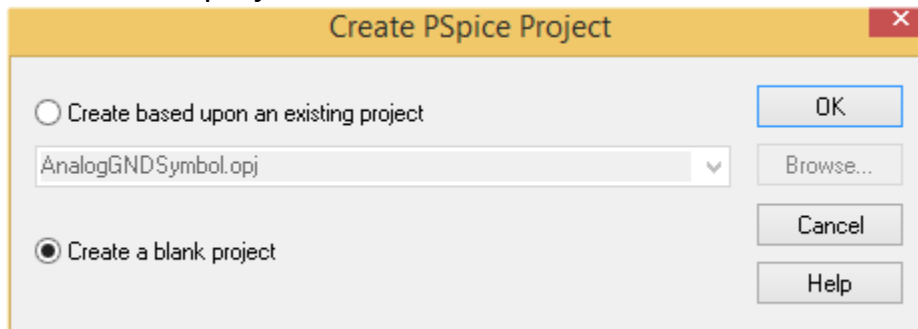
3. Click on new project



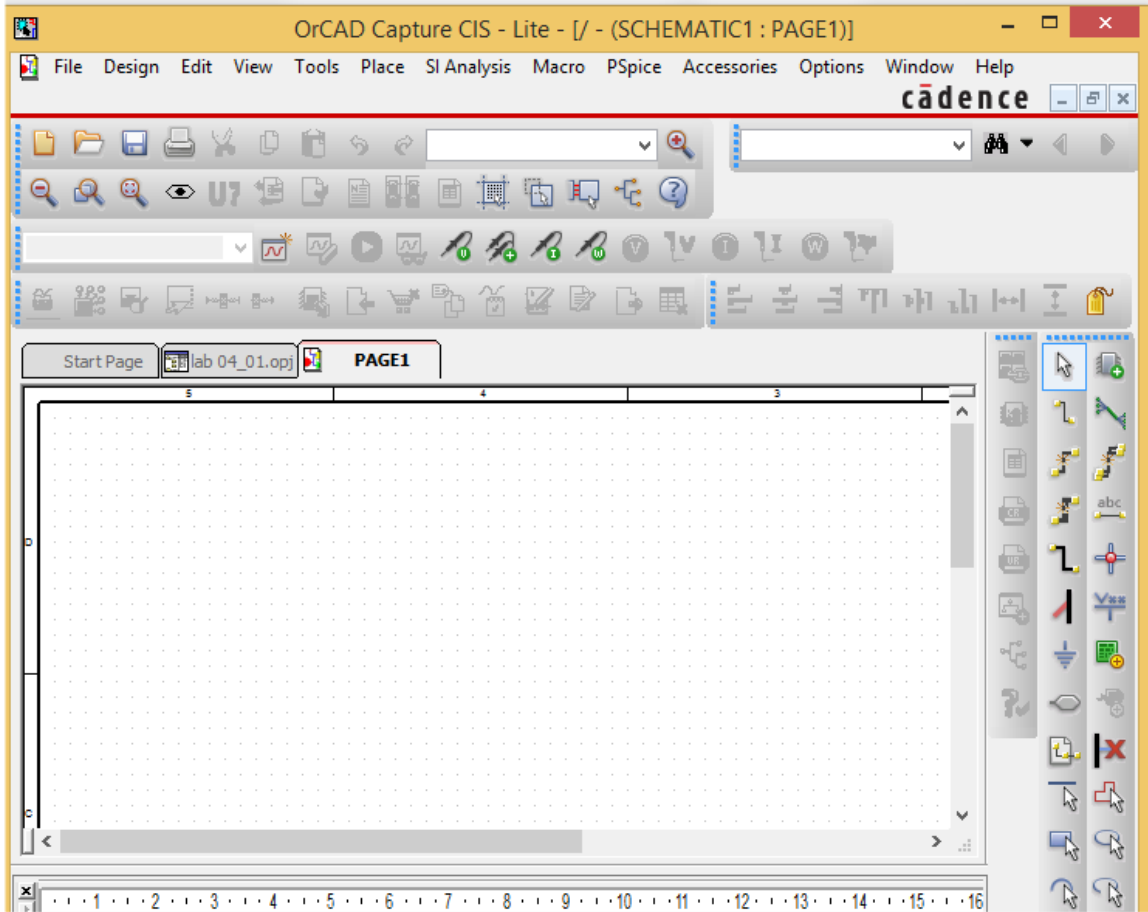
- Choose project type, location, and type project name



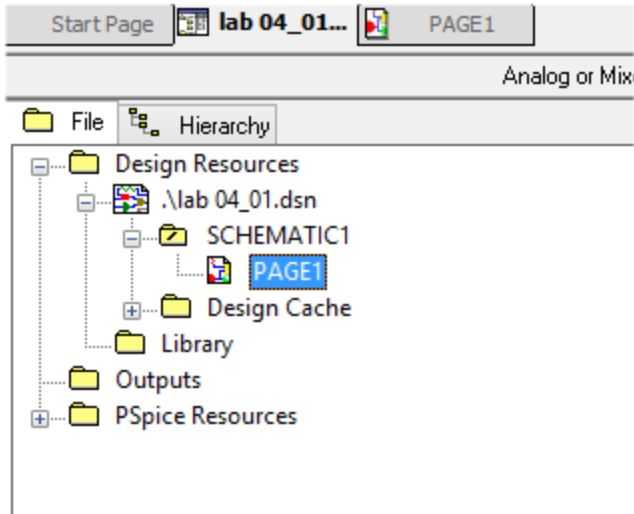
- Choose blank project



6. Now we are ready



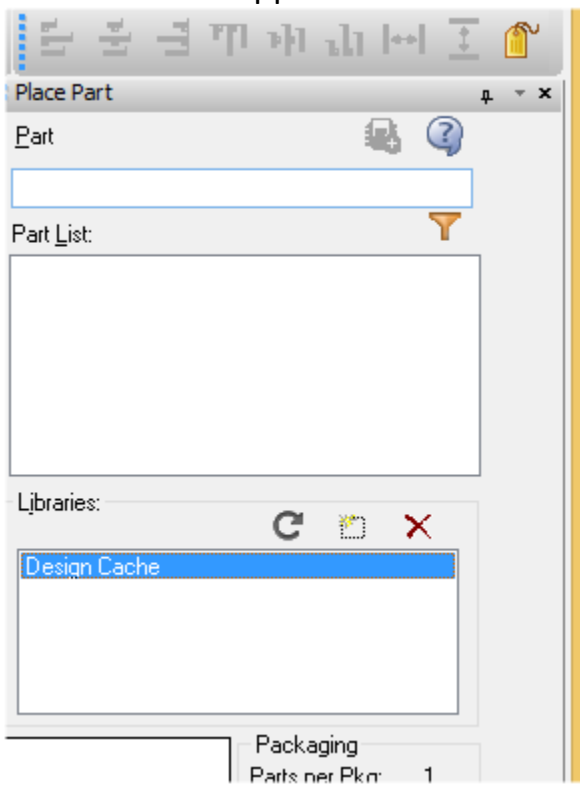
7. Reaching the Schematic file from project tree



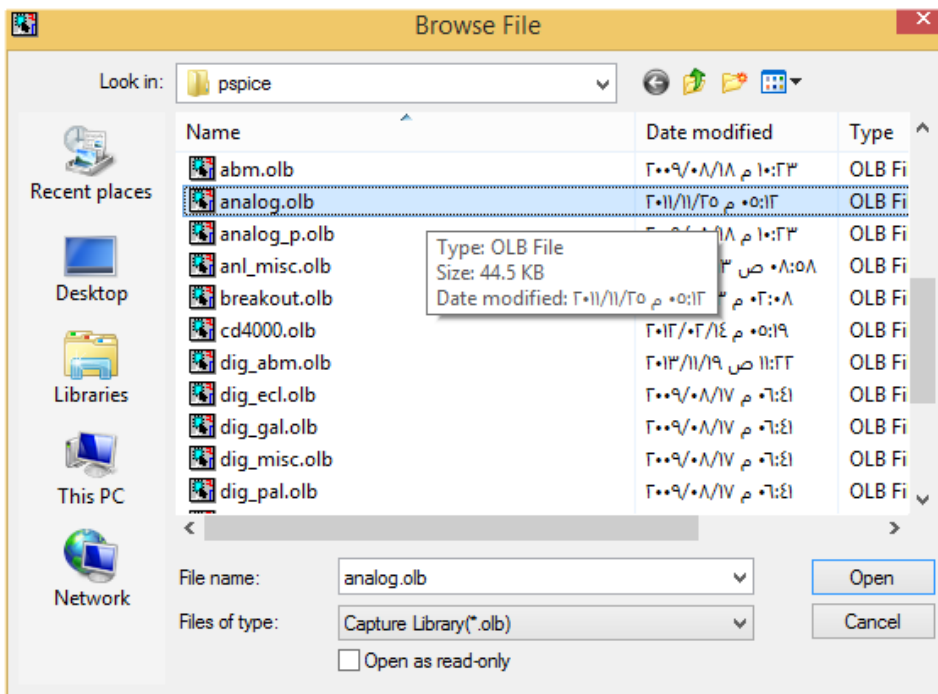
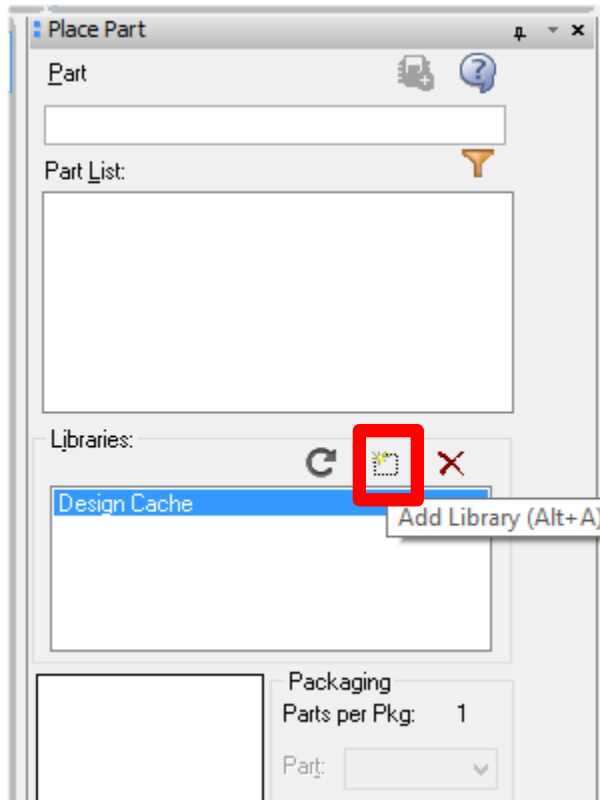
8. Return to schematic and add components

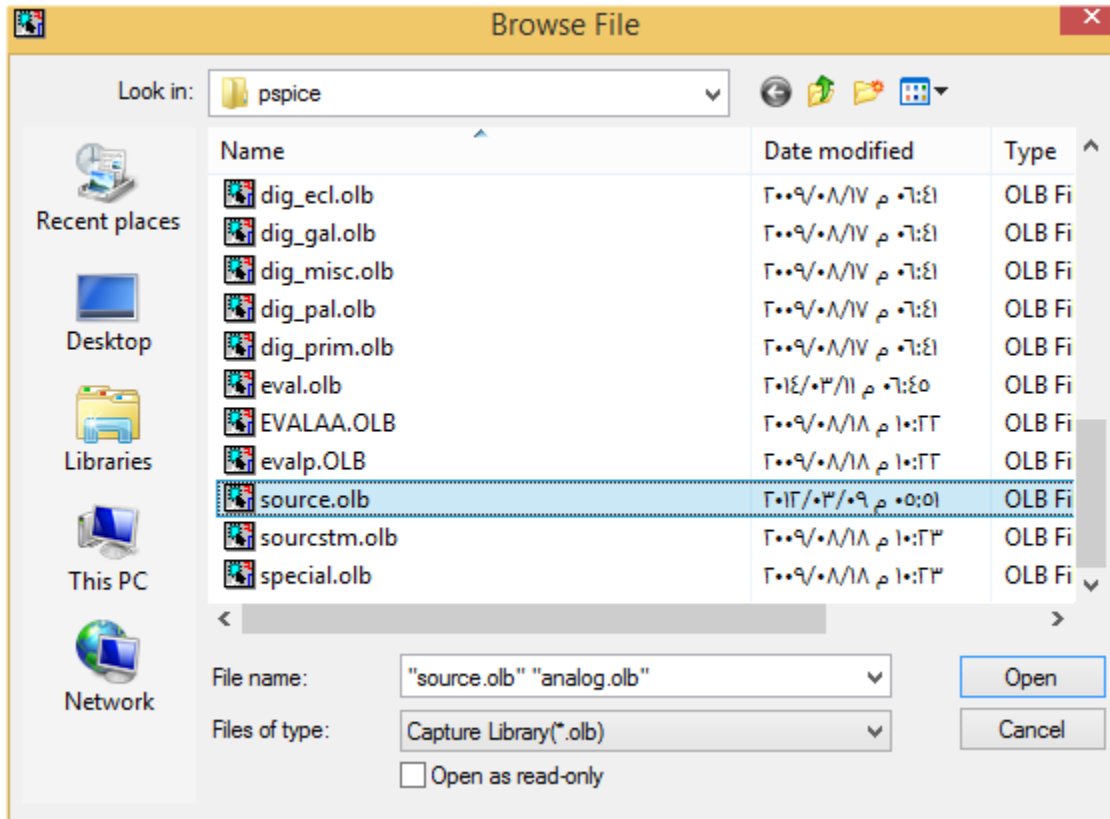


9. A new window appears

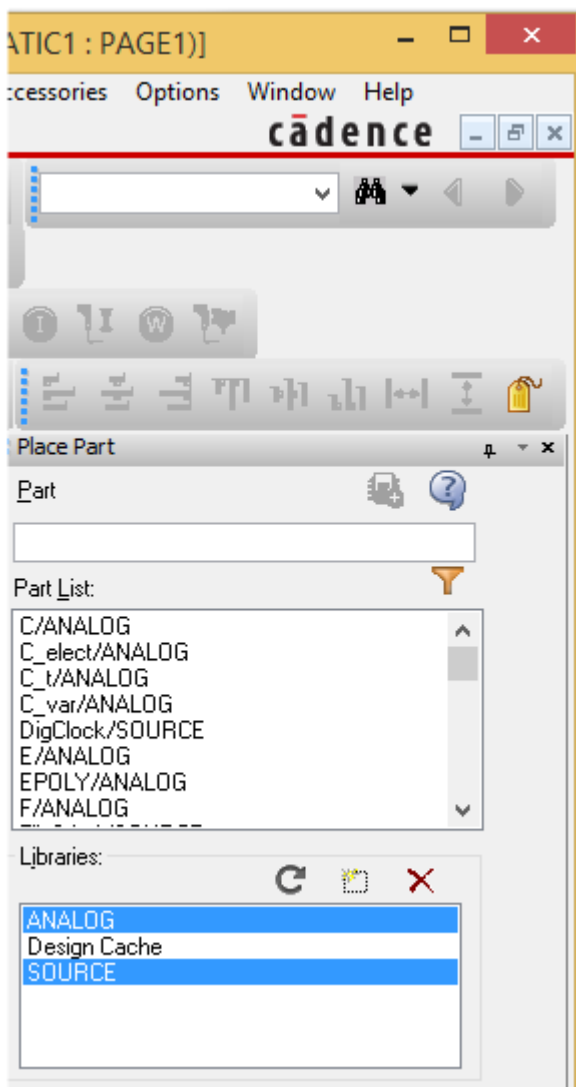


10. Add new library (analog, and sources)

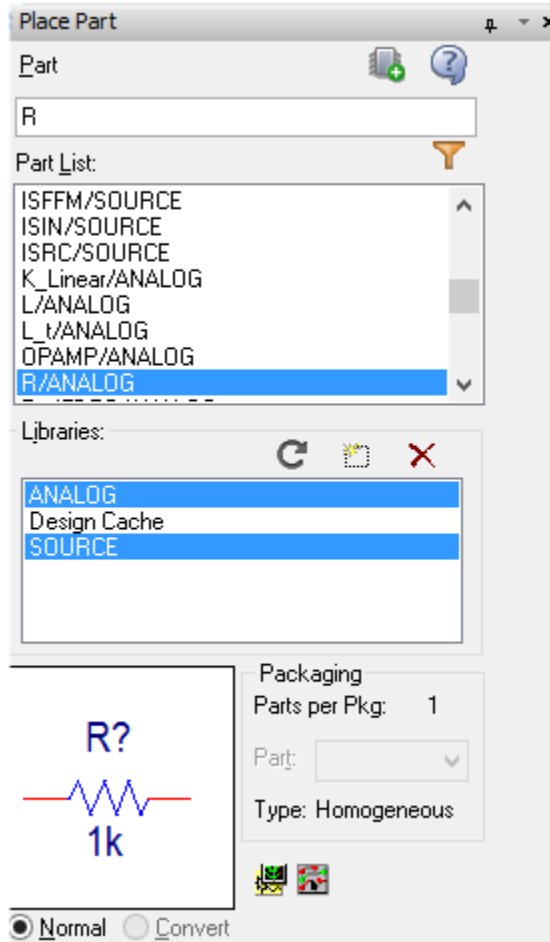




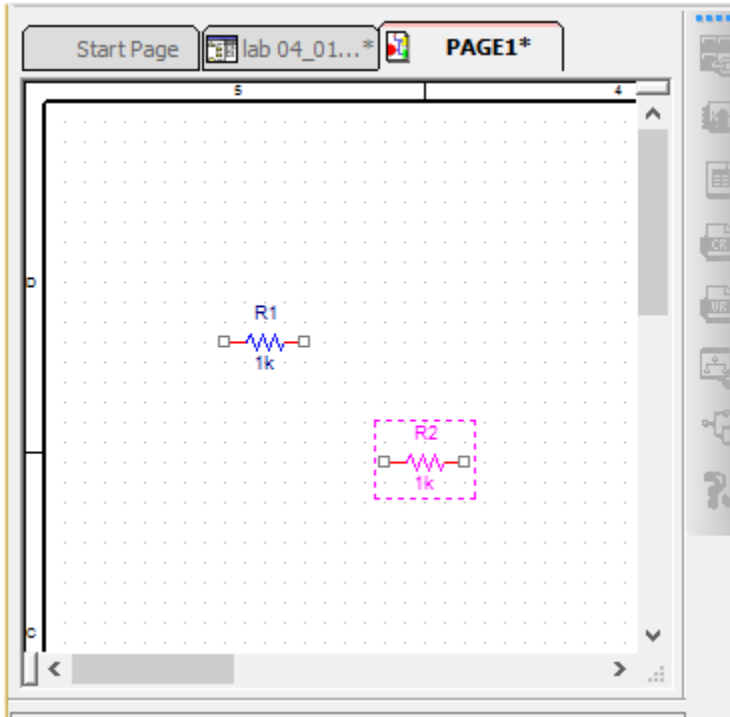
11. A few components will appear



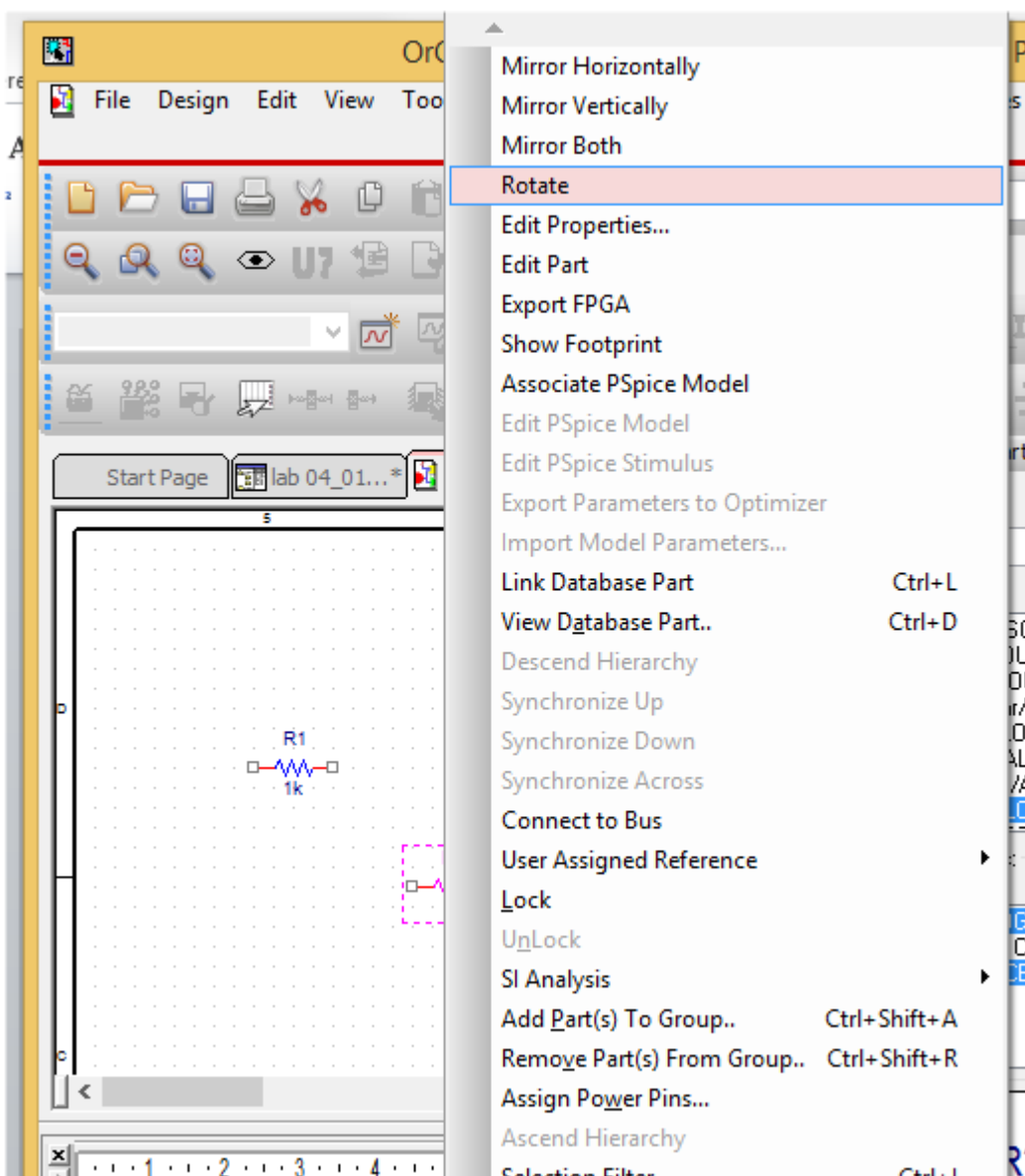
12. Search for Resistor by typing "R" in textbox



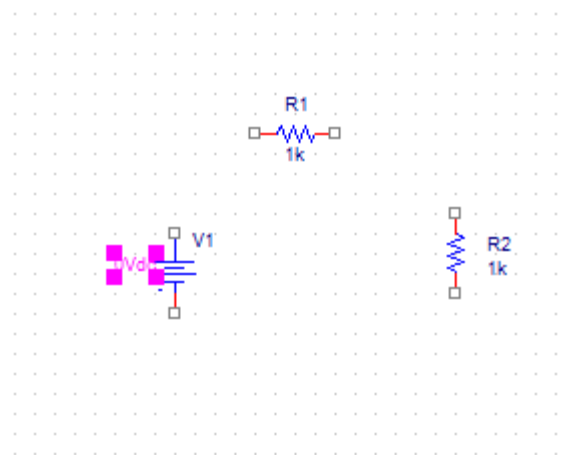
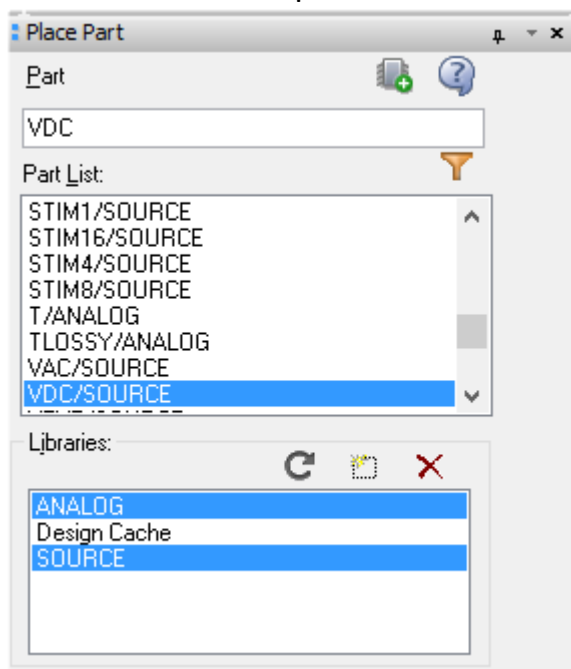
13. Double click on resistor component, mouse cursor will look like a resistor, drag mouse cursor, place it on schematic workspace, click mouse to place the resistor. Click again to add another one, press ESC to end the command



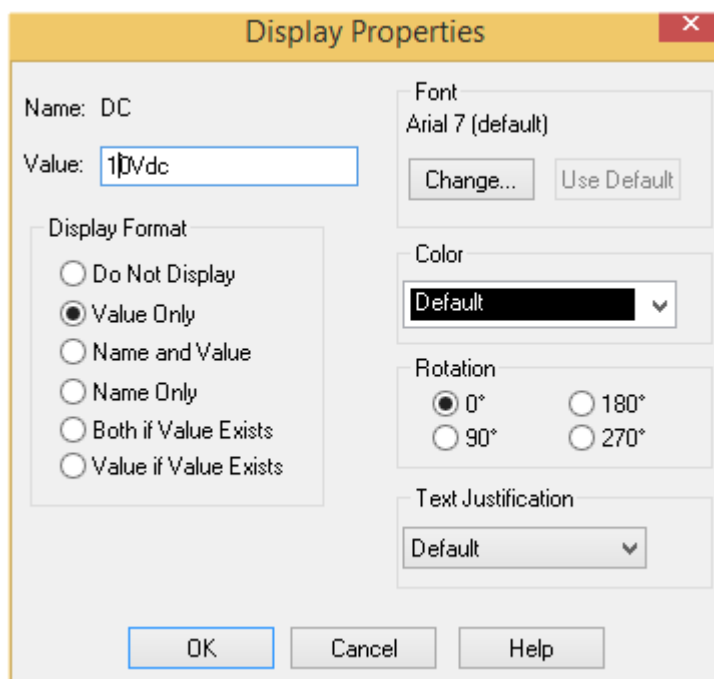
15. Right click on R2 choose rotate



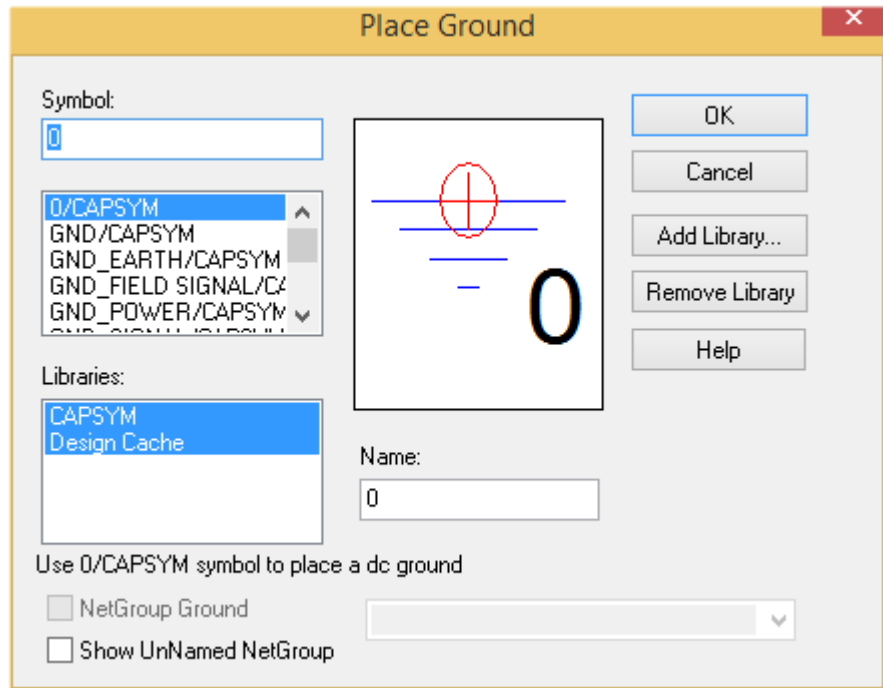
16. Place a VDC component



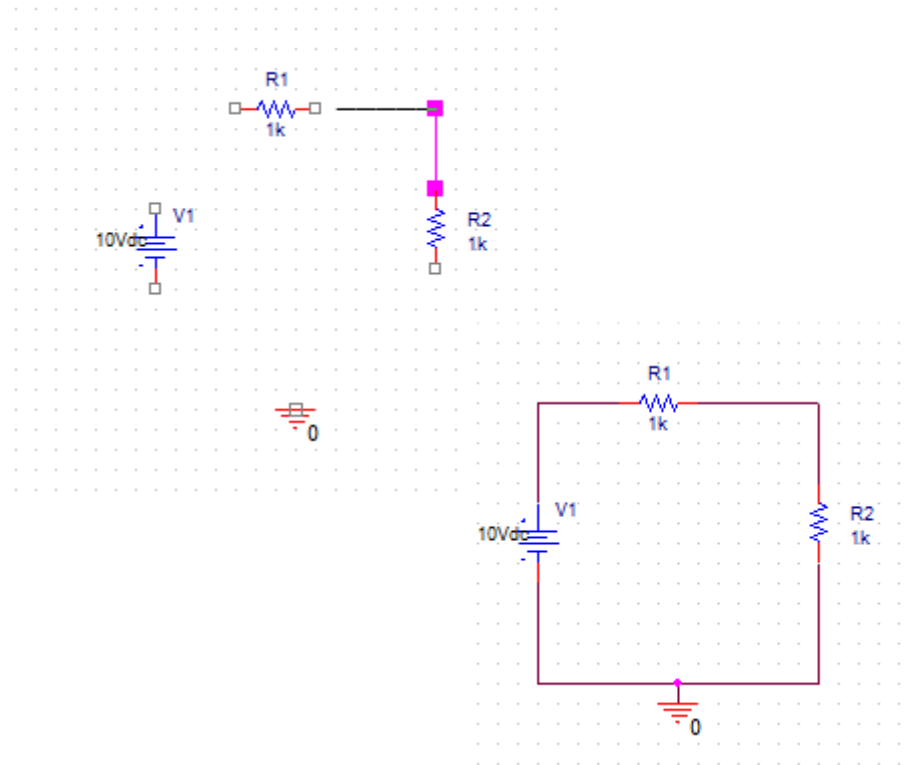
17. Double click on VDC voltage value, to change it to 10 Volt



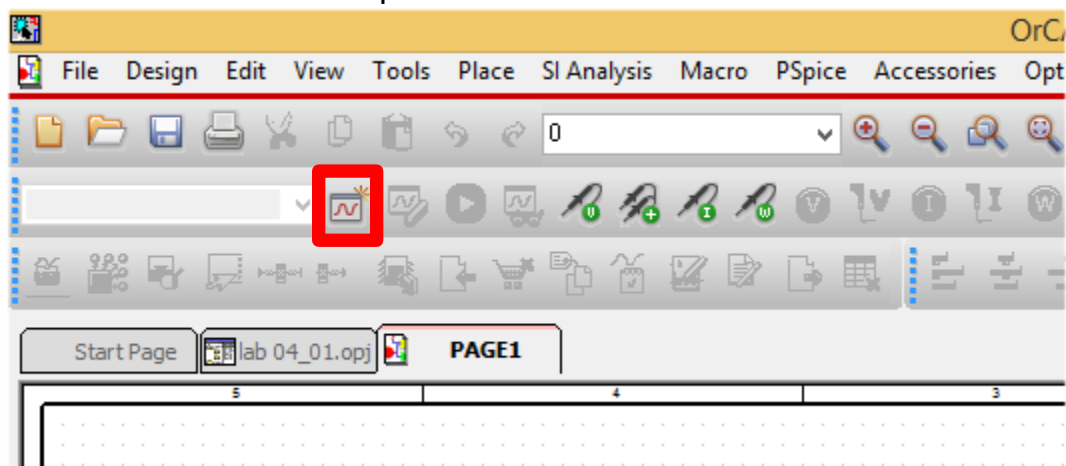
18. Place a ground



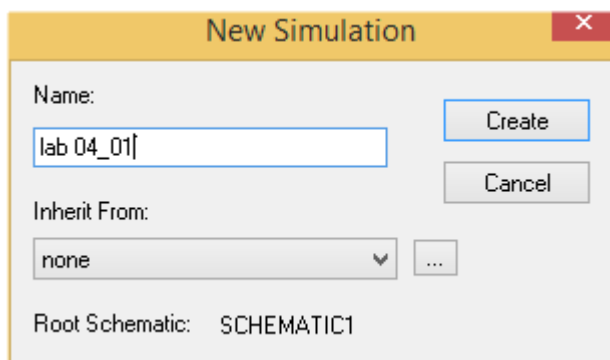
19. It's time to connect wires



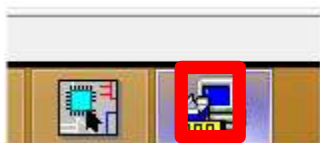
20. Create simulation profile



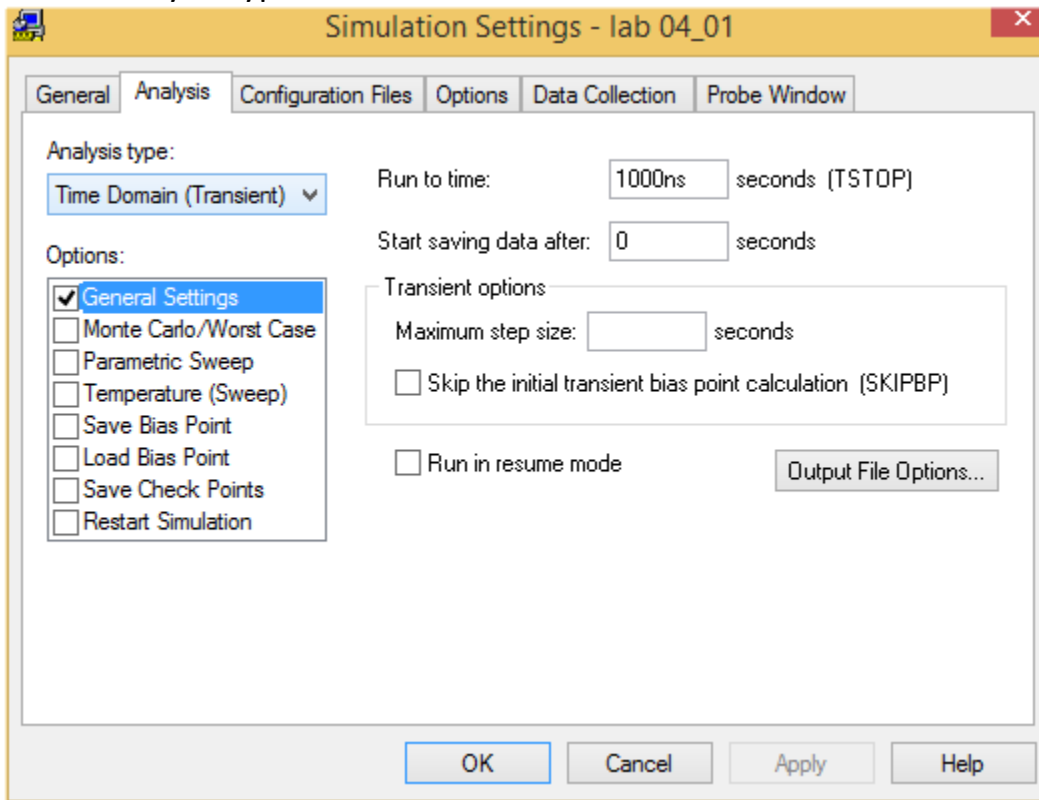
21. Type profile name



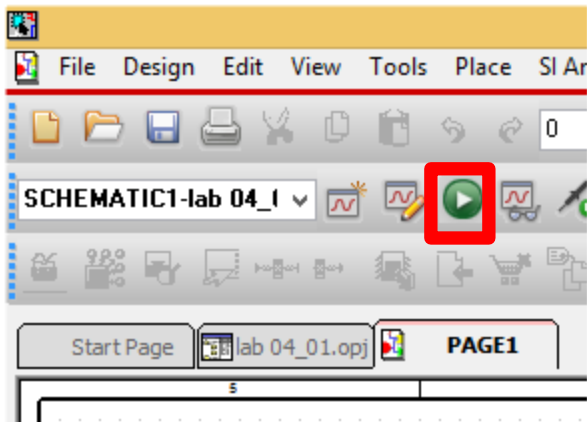
22. A profile editor will appear in task bar, open it



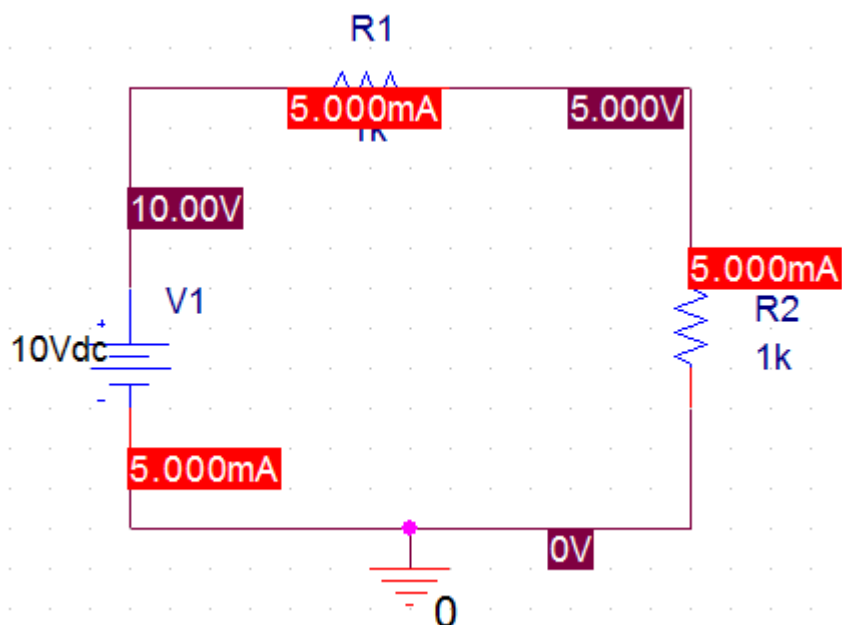
23. Select analysis type "time domain transient"



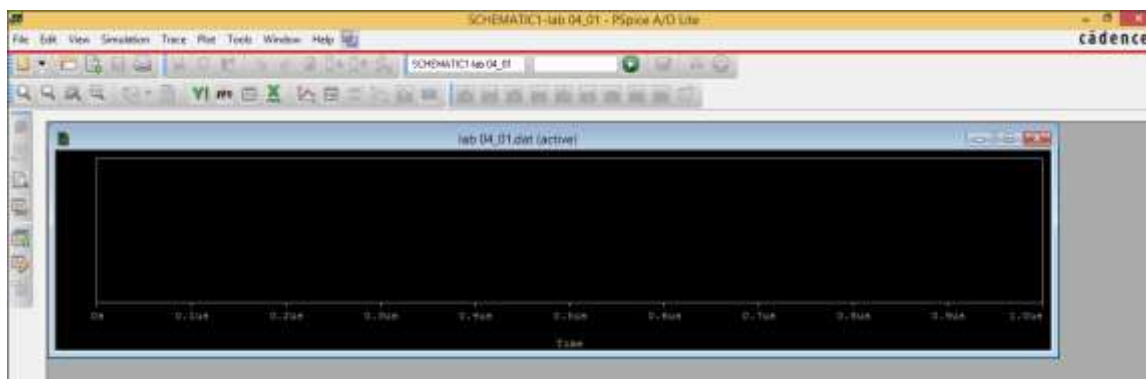
24. Start analysis



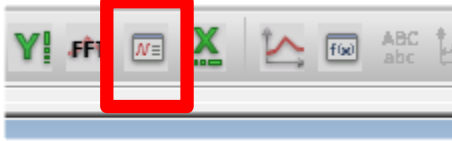
25. Make sure V, I icons are pressed to view branches current and nodes voltage



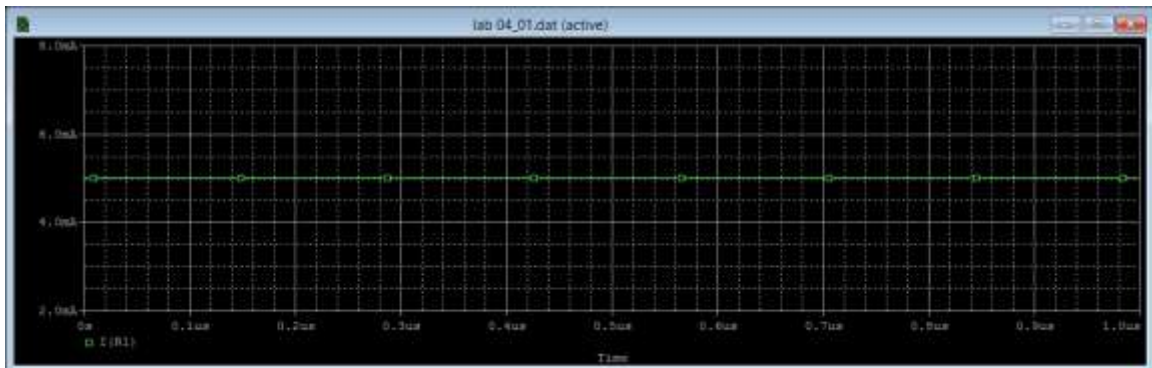
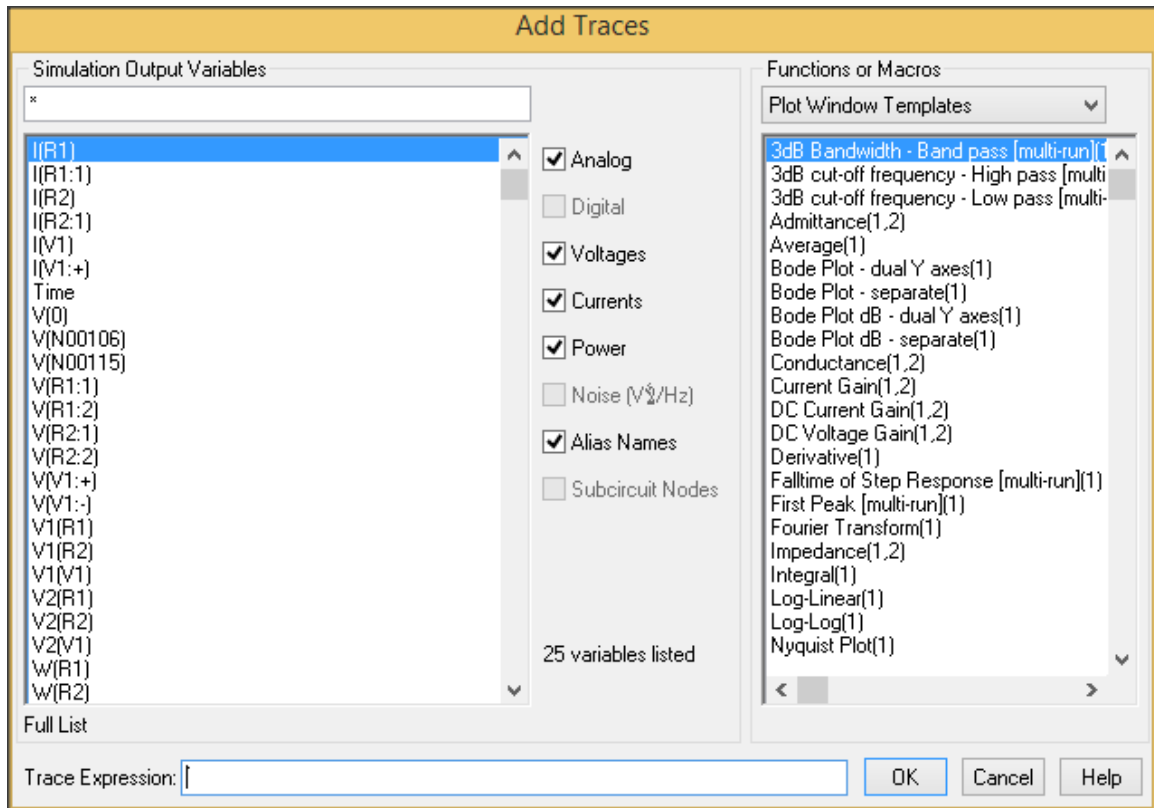
26. Open analyzer window



27. Add a time domain measurement, click on add trace button



28. Select current pass in R1



29. Select voltage difference between resistor R1 terminals

Add Traces

Simulation Output Variables

*

- I(R1)
- I(R1:1)
- I(R2)
- I(R2:1)
- I(V1)
- I(V1: +)
- Time
- V(0)
- V(N00106)
- V(N00115)
- V(R1:1)
- V(R1:2)
- V(R2:1)
- V(R2:2)
- V(V1: +)
- V(V1: -)
- V1(R1)
- V1(R2)
- V1(V1)
- V2(R1)
- V2(R2)
- V2(V1)
- W(R1)
- W(R2)

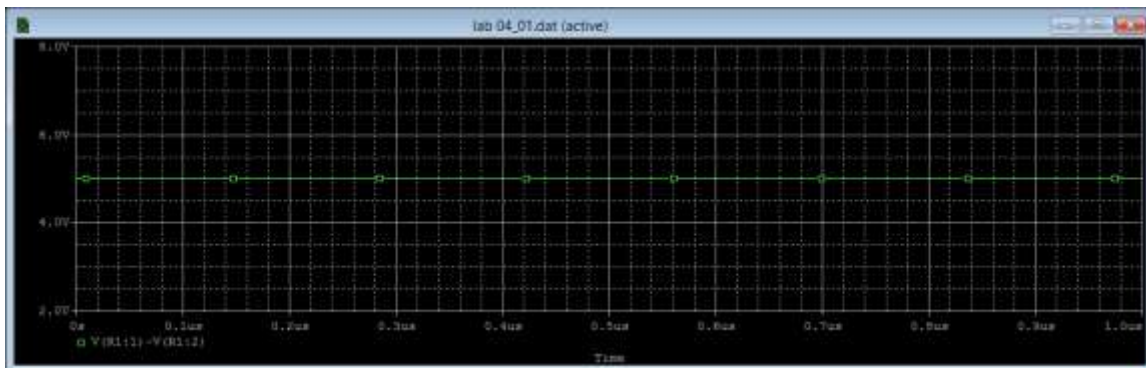
Full List

Functions or Macros

Plot Window Templates

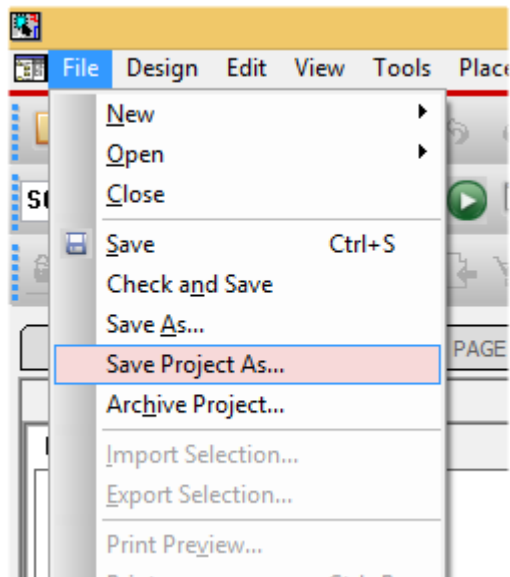
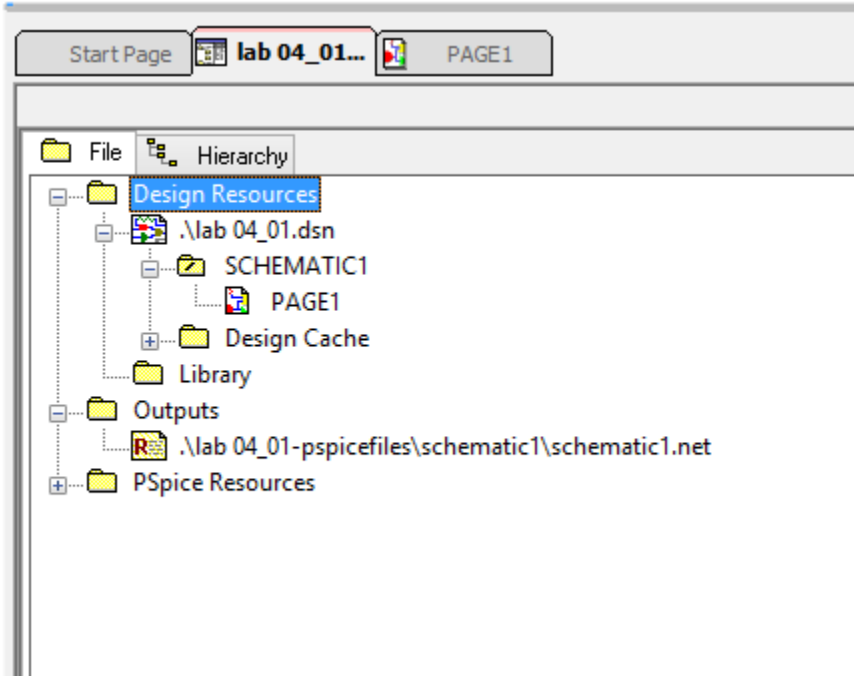
- 3dB Bandwidth - Band pass [multi-run](1)
- 3dB cut-off frequency - High pass [multi-run](1)
- 3dB cut-off frequency - Low pass [multi-run](1)
- Admittance(1,2)
- Average(1)
- Bode Plot - dual Y axes(1)
- Bode Plot - separate(1)
- Bode Plot dB - dual Y axes(1)
- Bode Plot dB - separate(1)
- Conductance(1,2)
- Current Gain(1,2)
- DC Current Gain(1,2)
- DC Voltage Gain(1,2)
- Derivative(1)
- Falltime of Step Response [multi-run](1)
- First Peak [multi-run](1)
- Fourier Transform(1)
- Impedance(1,2)
- Integral(1)
- Log-Linear(1)
- Log-Log(1)
- Nyquist Plot(1)

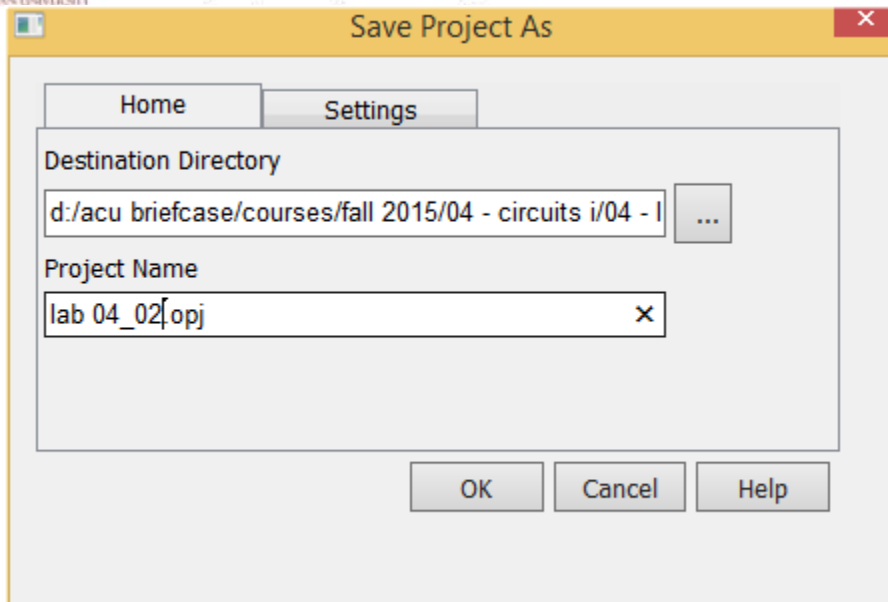
Trace Expression:



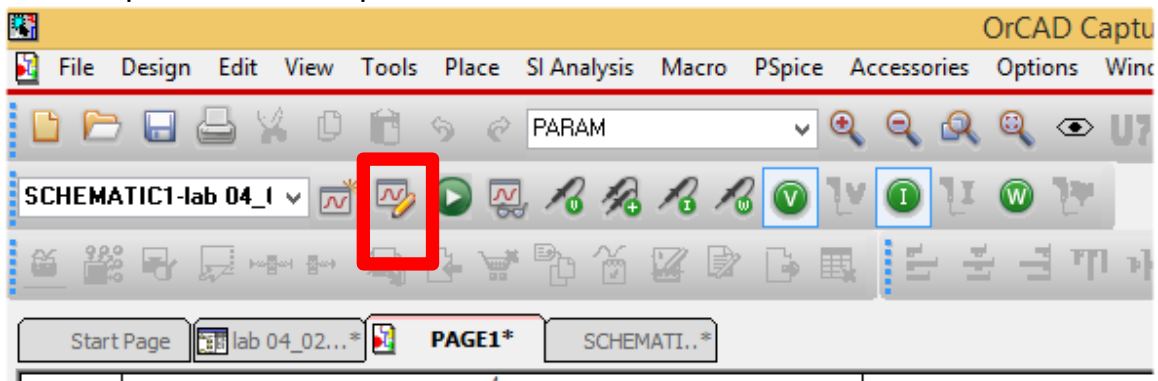
Part 2 : ohm's low verification

1. Save project as ""

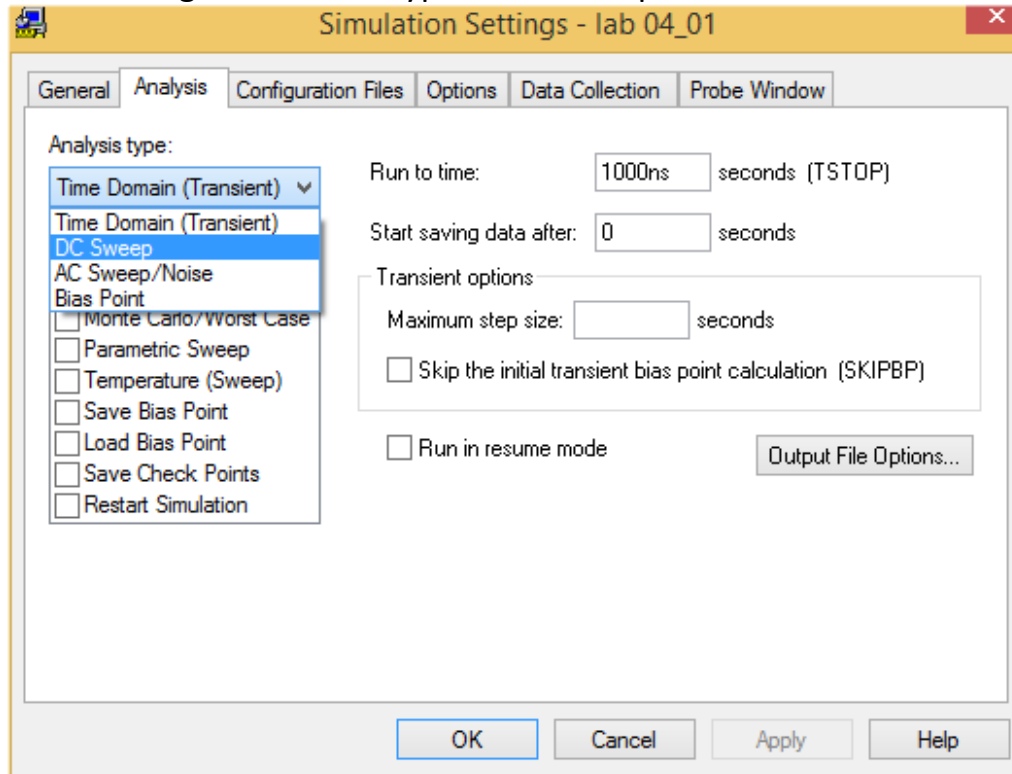




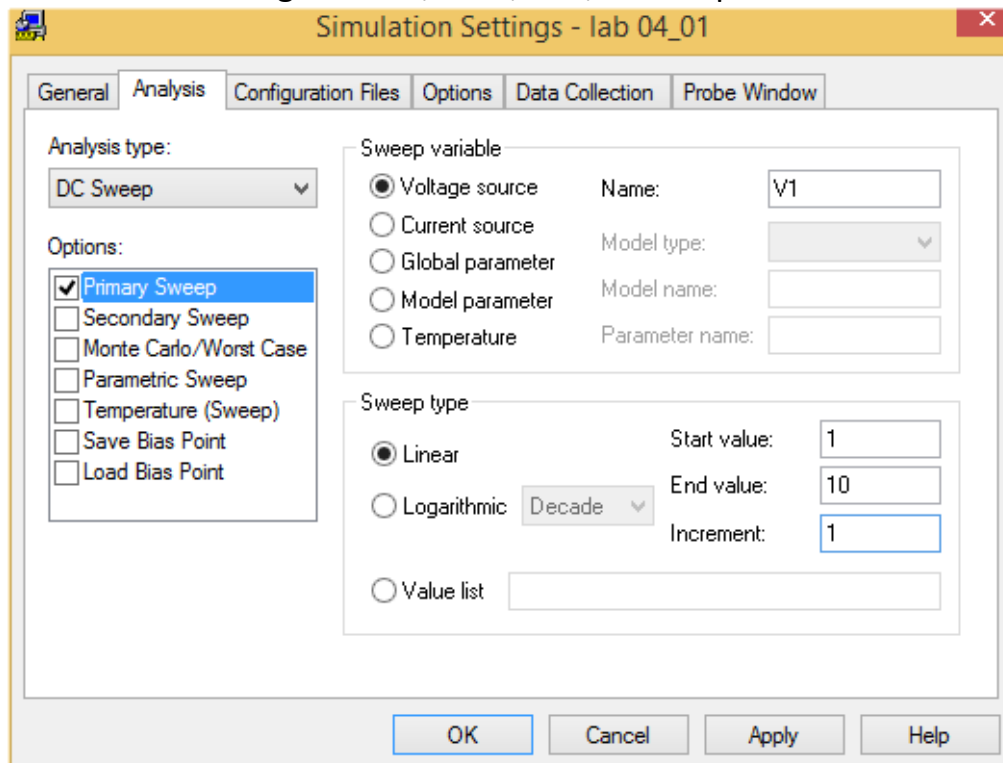
2. Open simulation profile



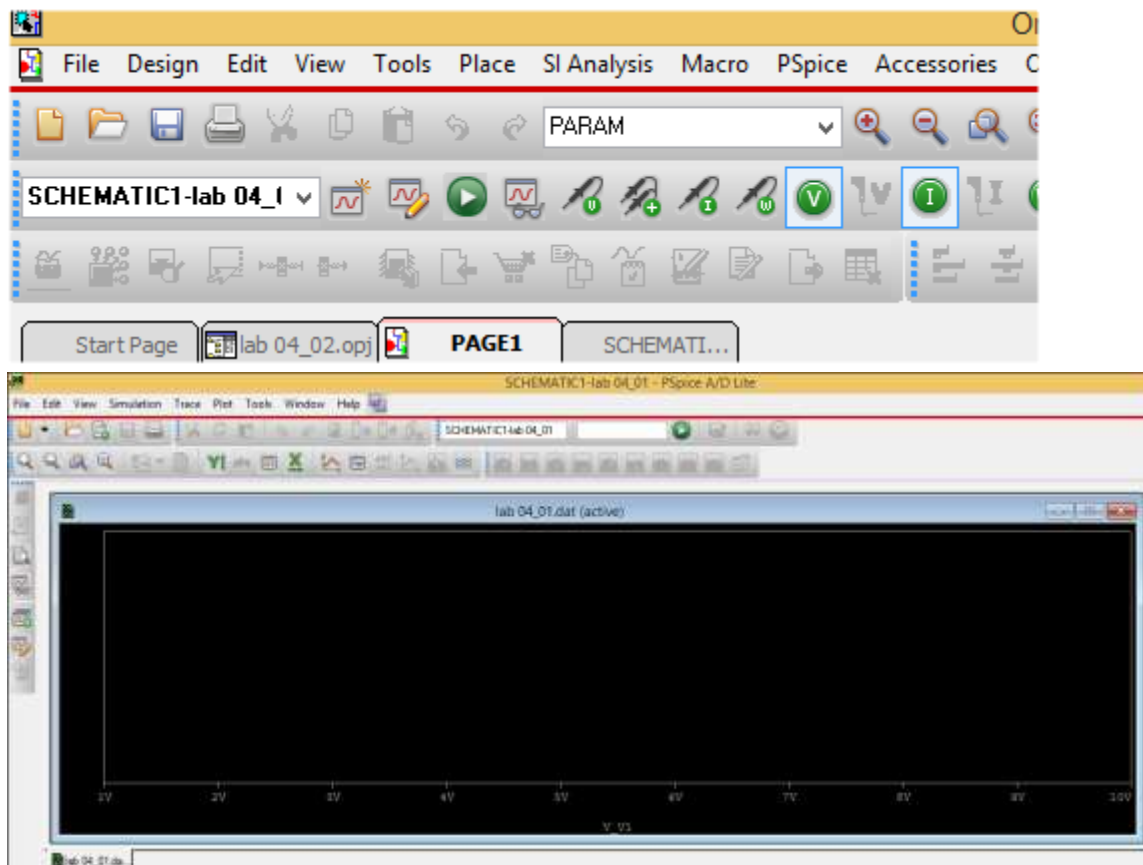
3. Change simulation type to DC sweep



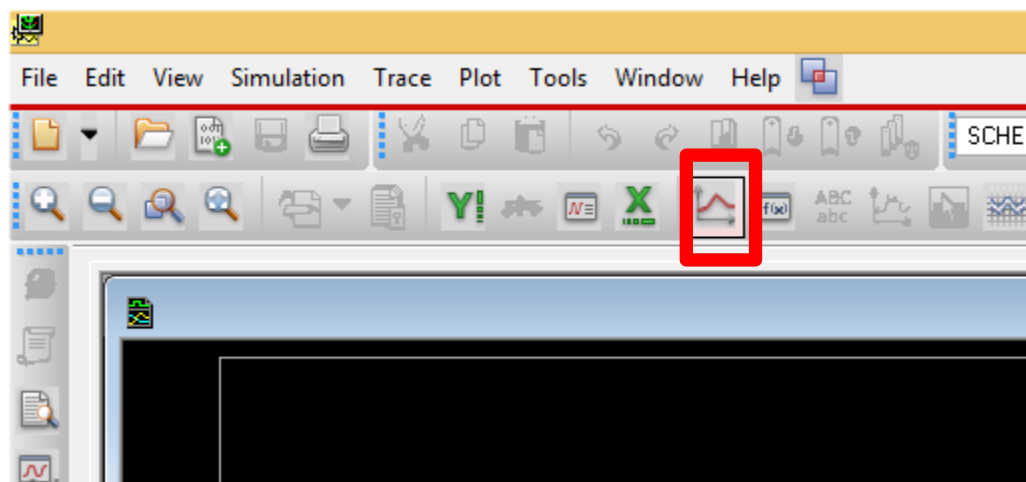
4. Select voltage source, start, end, and step

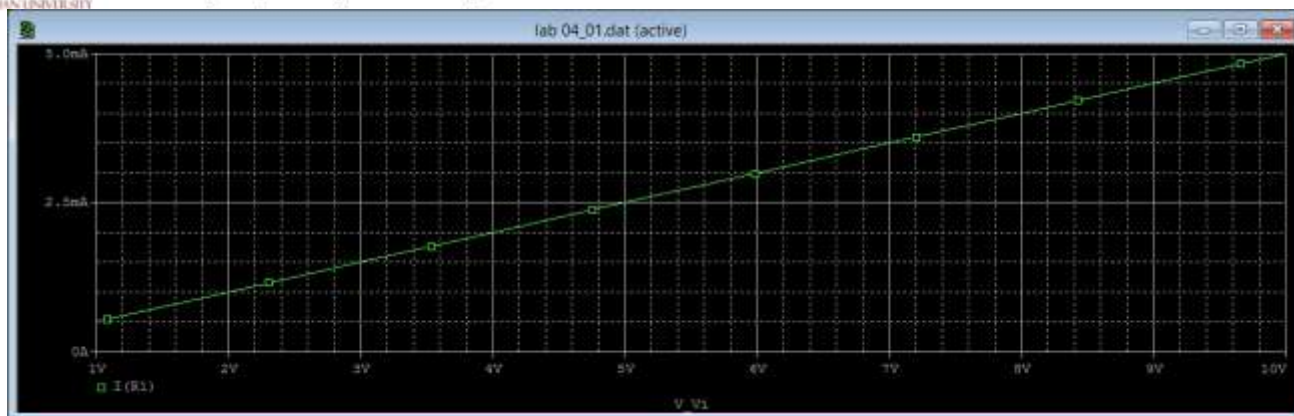


5. Click on simulation



6. Add trace current of R1







Questions and Conclusions

1. Select 2 point on the graph, calculate the Slop?

.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....

2. Dose it meet the expected value? And what is that value, and what is that value presents?

.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....
.....



كلية الهندسة

Faculty of Engineering



جامعة أهرام كندا
AHRAM CANADIAN UNIVERSITY