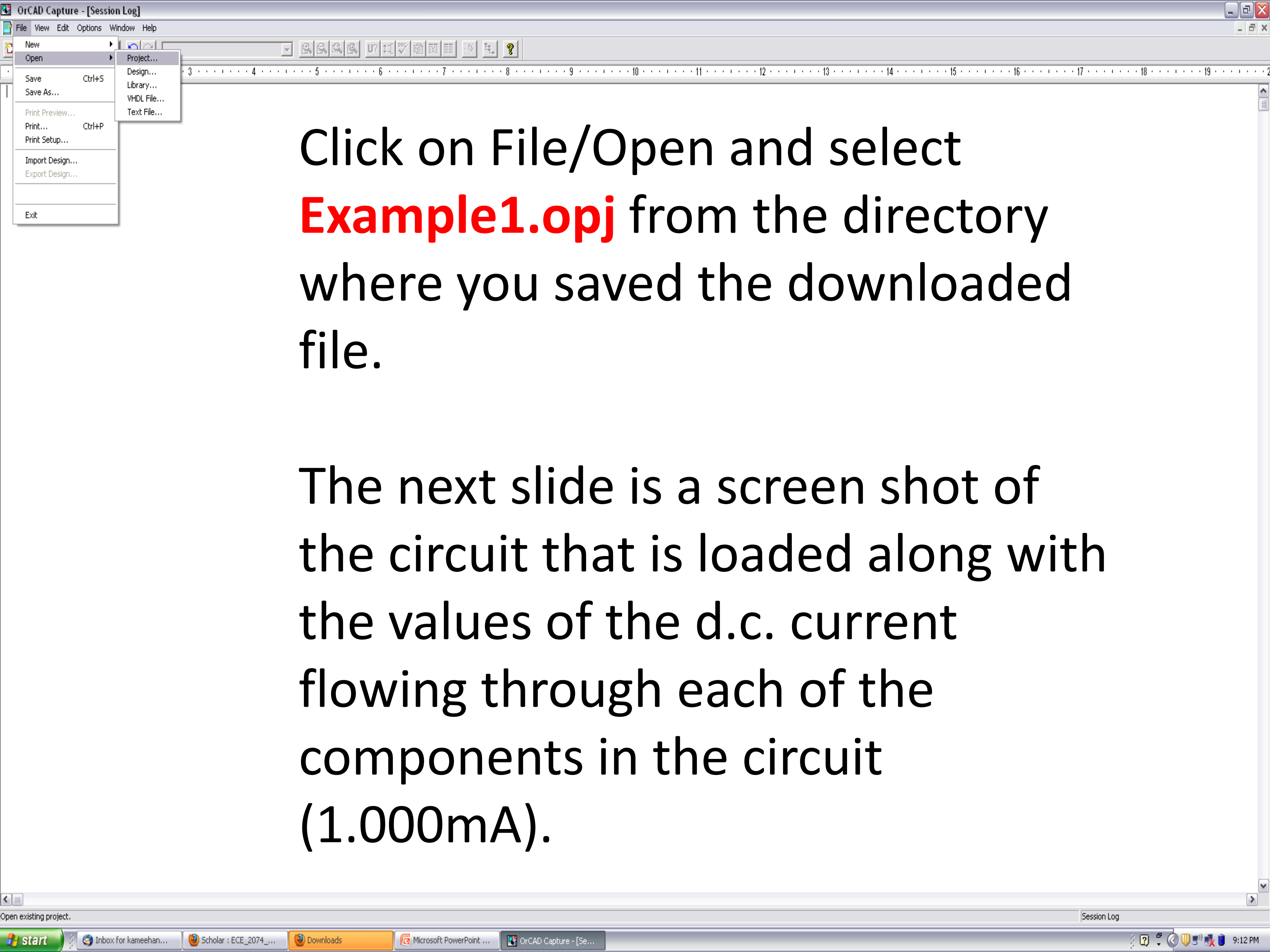
Verification of Proper Installation of 9.1 Capture

File for Simulation Test

- Download Example1.opj from the ECE 2984 LiaB Scholar site under Resources in the folder called:

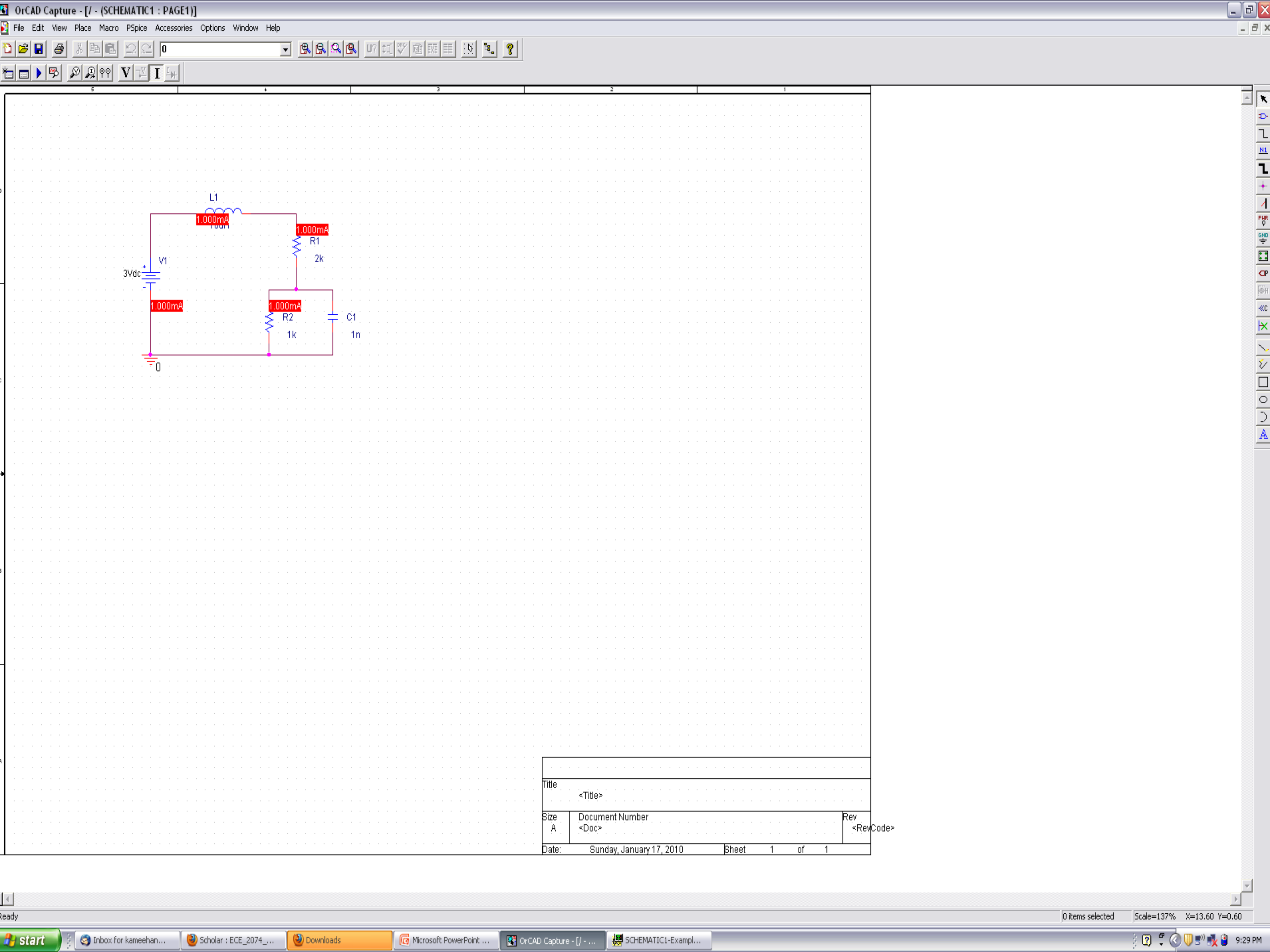
  Pspice/Capture

- Then, run PSpice Student/Capture Student.
  - One application should launch
    - **OrCAD Capture – [Session Log]**



Click on File/Open and select **Example1.opj** from the directory where you saved the downloaded file.

The next slide is a screen shot of the circuit that is loaded along with the values of the d.c. current flowing through each of the components in the circuit (1.000mA).

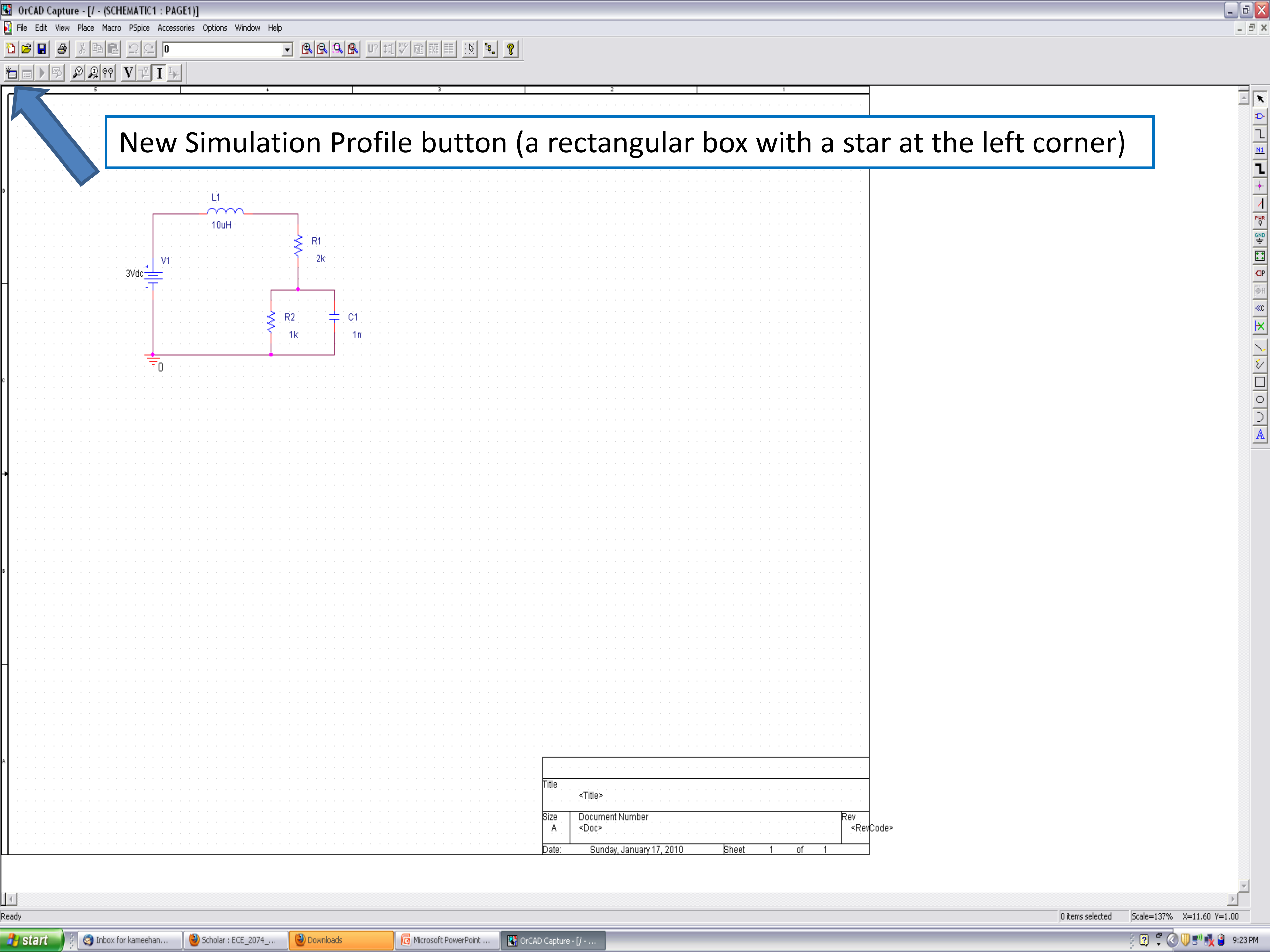


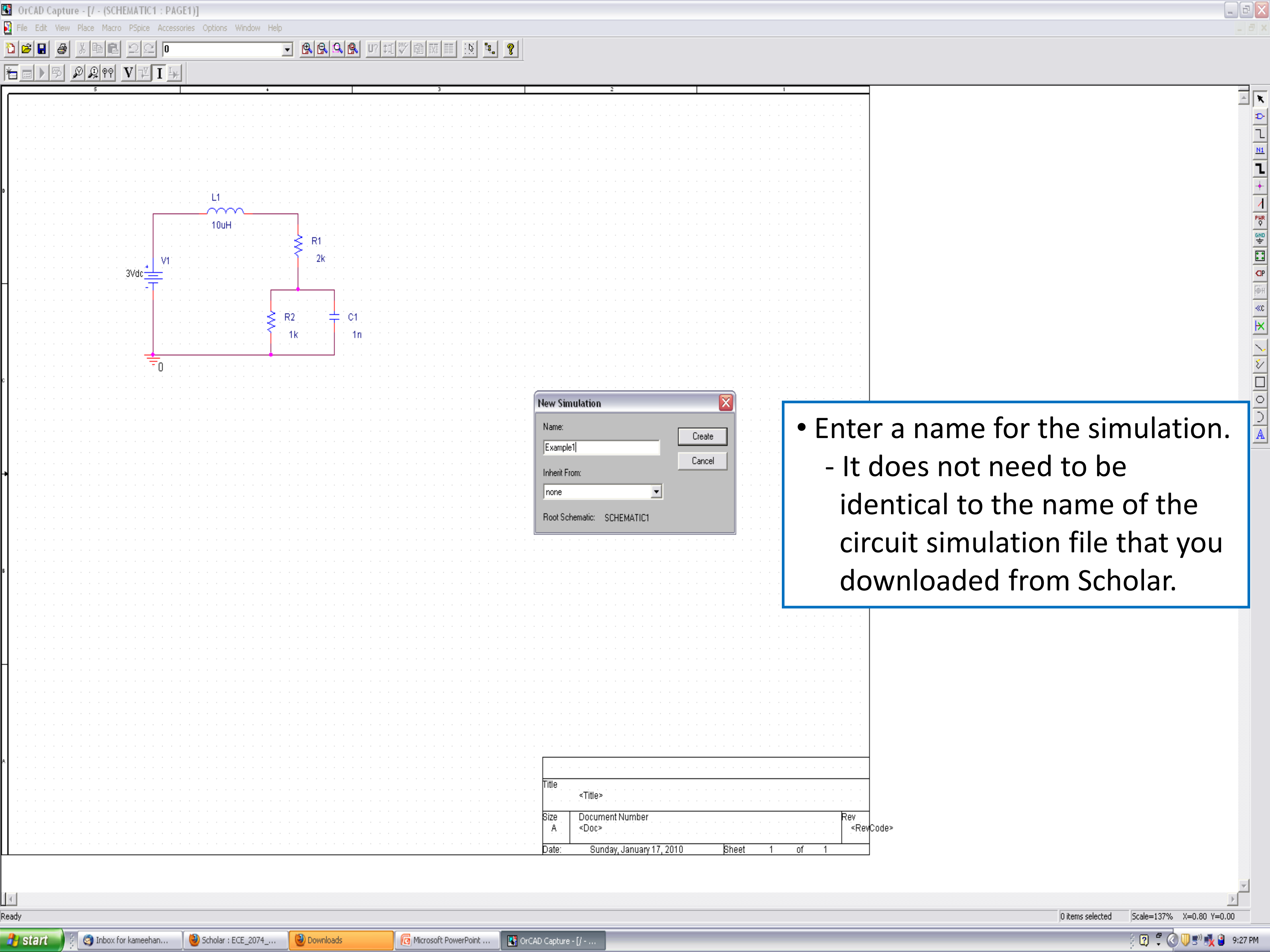
OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
L1
1.000mA
1.000mA
R1
2k
V1
3Vdc
1.000mA
1.000mA
R2
1k
C1
1n
0
Title
<Title>
Size
A
Document Number
<Doc>
Rev
<RevCode>
Date: Sunday, January 17, 2010
Sheet 1 of 1
Ready
0 items selected
Scale=137%
X=13.60 Y=0.60
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
Microsoft PowerPoint ...
OrCAD Capture - [/ - ...
SCHEMATIC1-Exampl...
9:29 PM

If the Currents aren't Shown

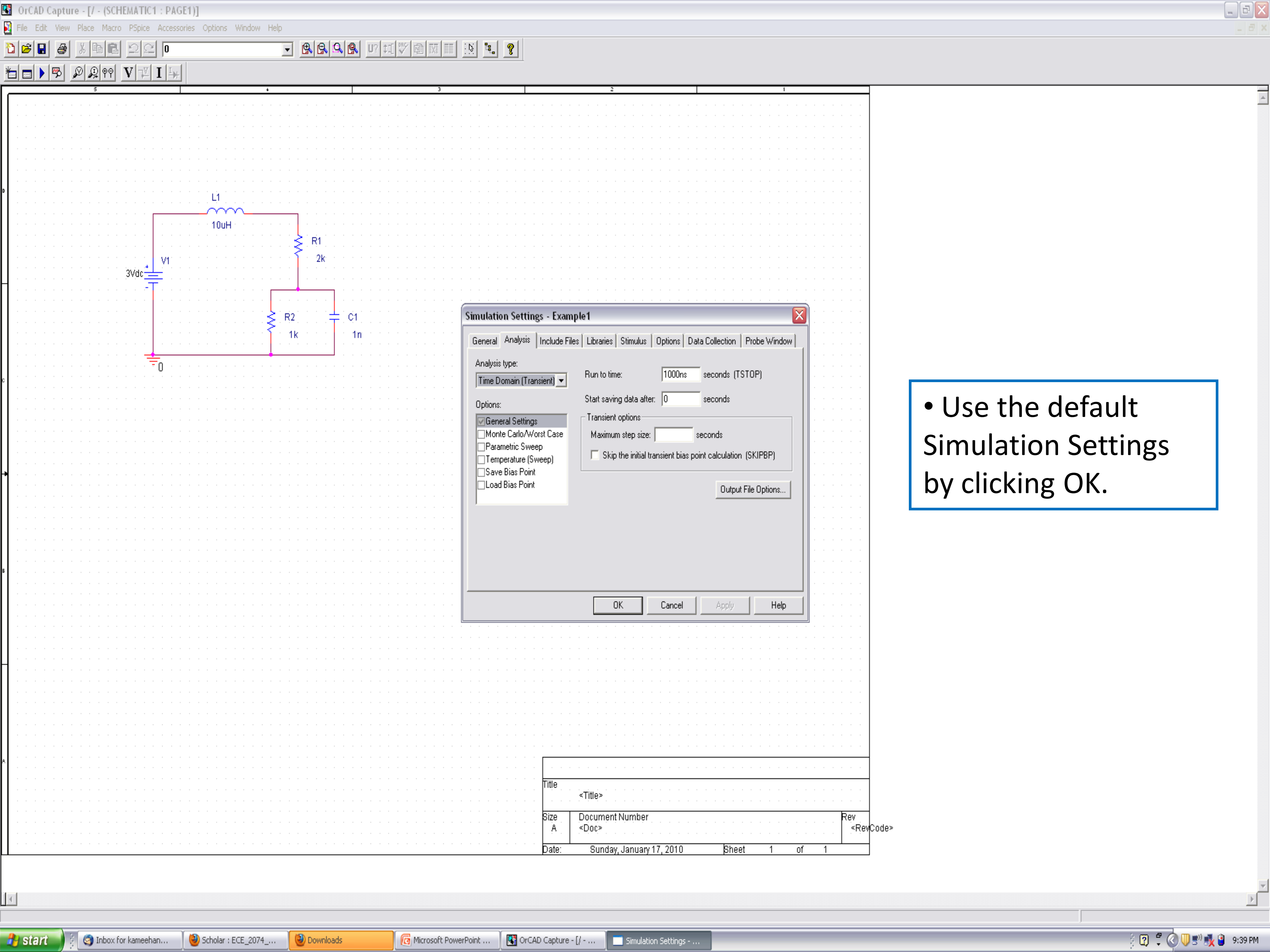
- Click on the New Simulation Profile button to select the type of calculation to be run.
  - The values for the currents through the components should appear on the circuit after **Simulation complete** appears in the pop-up box called Example1 – ORCAD A/D DEMO.

New Simulation Profile button (a rectangular box with a star at the left corner)



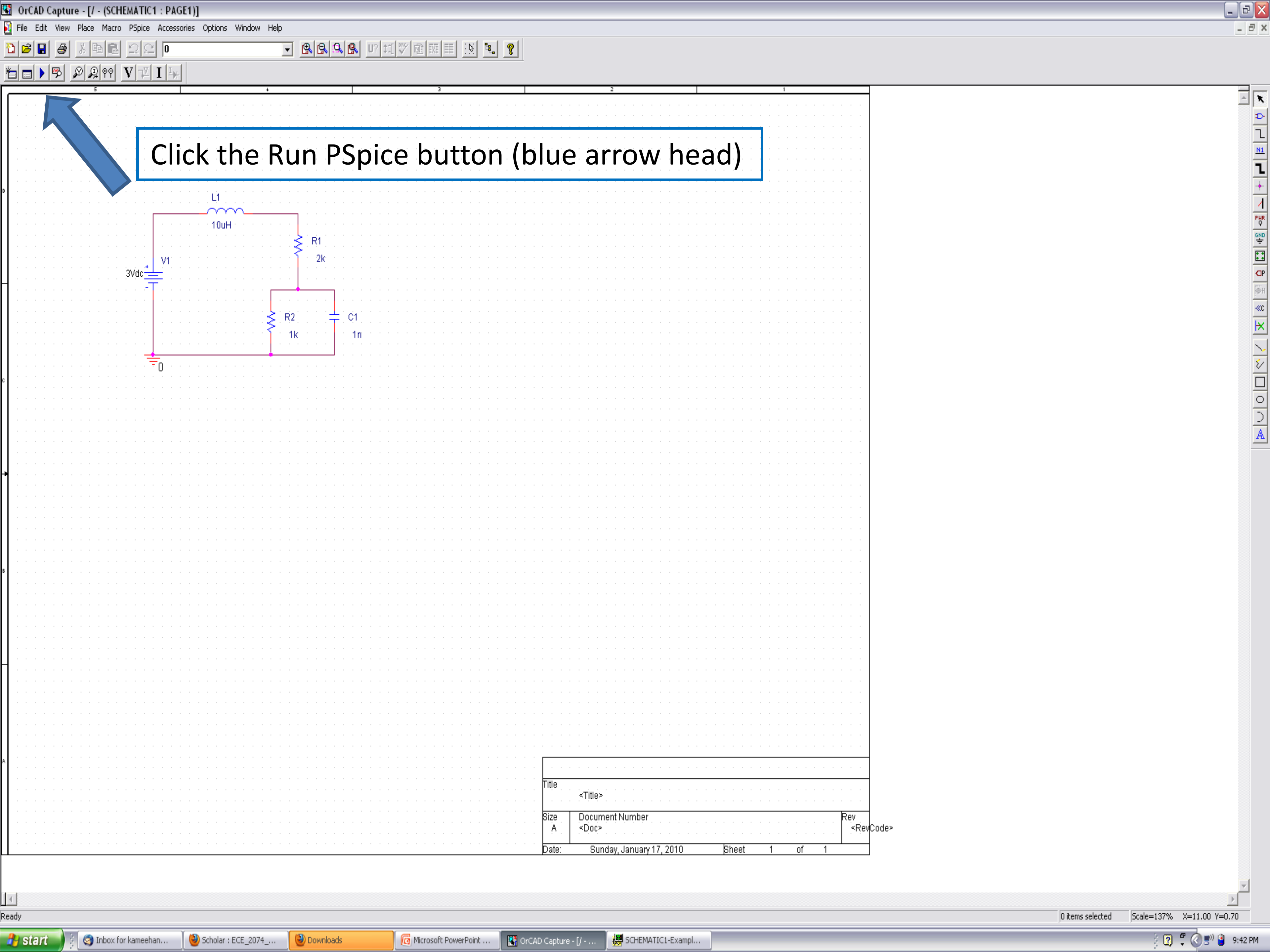


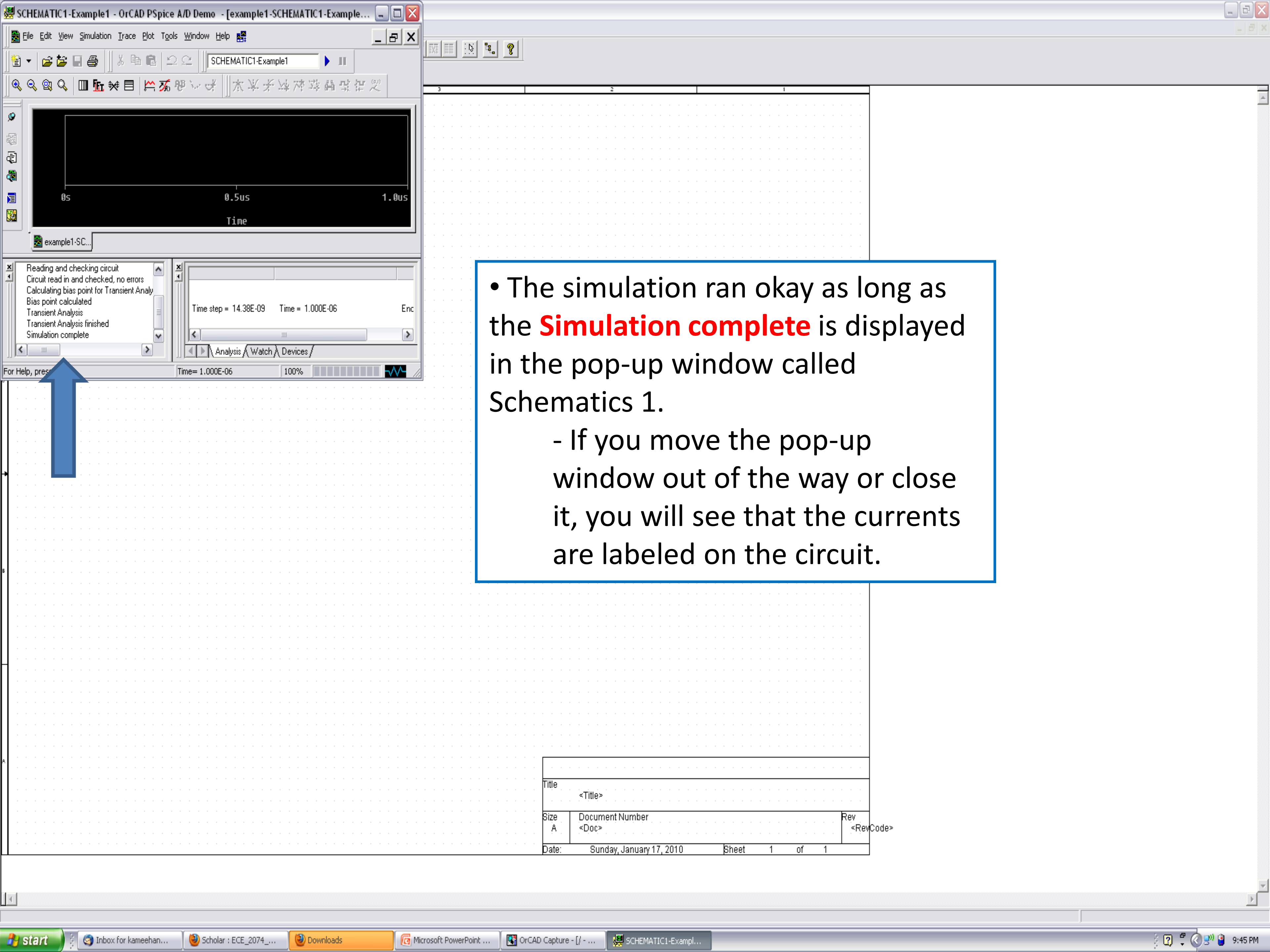
- Enter a name for the simulation.
  - It does not need to be identical to the name of the circuit simulation file that you downloaded from Scholar.



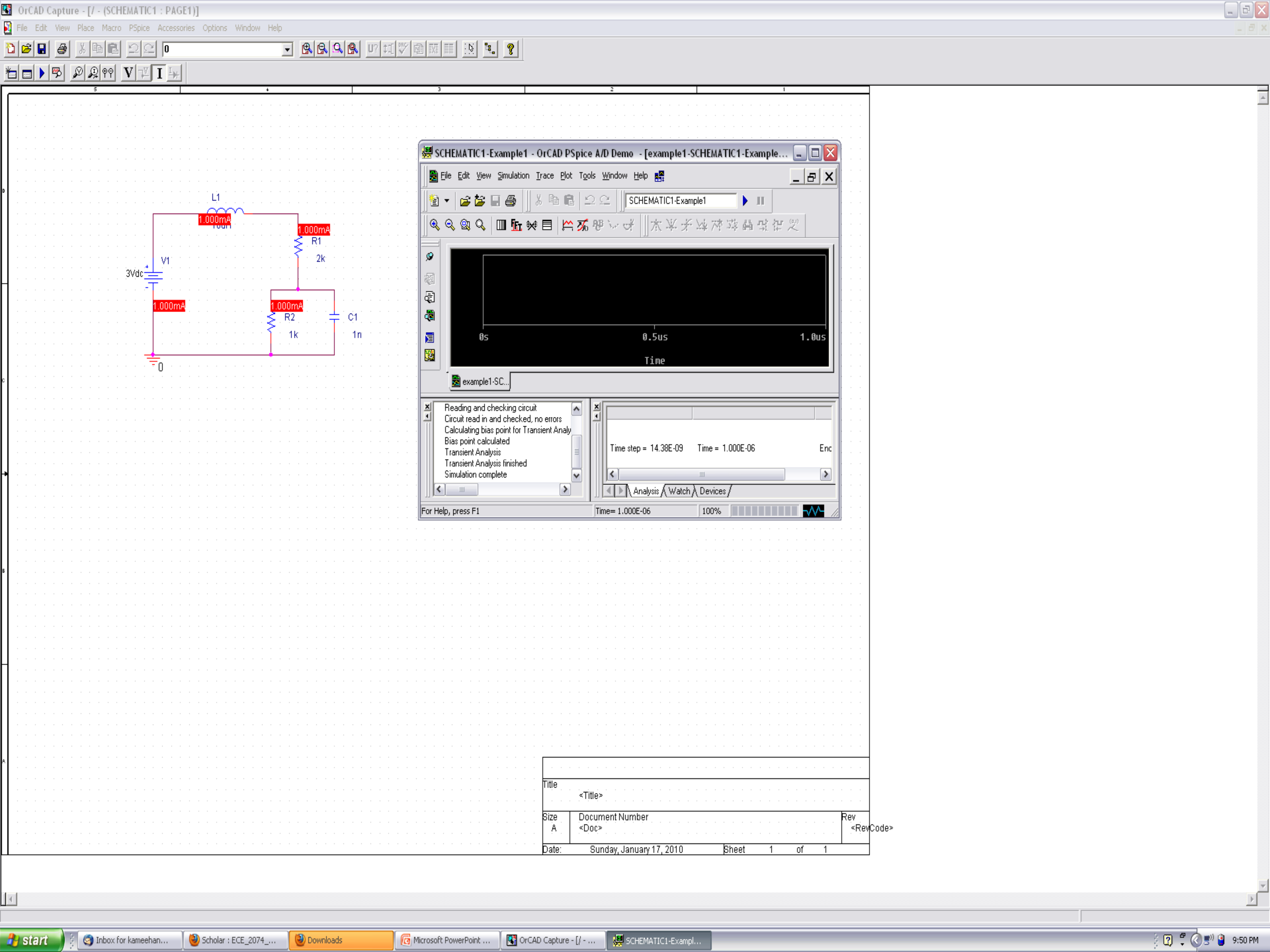
- Use the default Simulation Settings by clicking OK.

Click the Run PSpice button (blue arrow head)





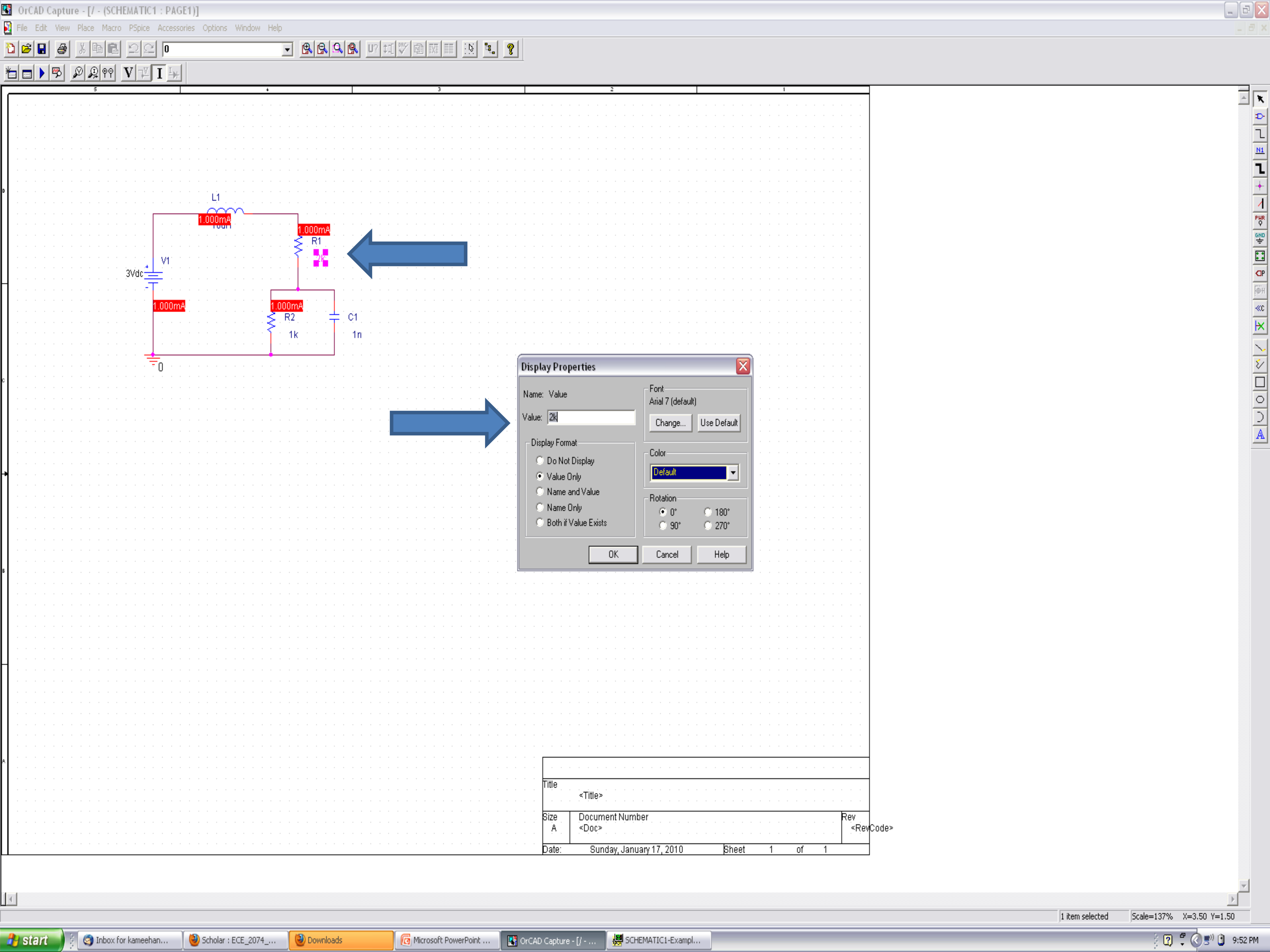
- The simulation ran okay as long as the **Simulation complete** is displayed in the pop-up window called Schematics 1.
    - If you move the pop-up window out of the way or close it, you will see that the currents are labeled on the circuit.

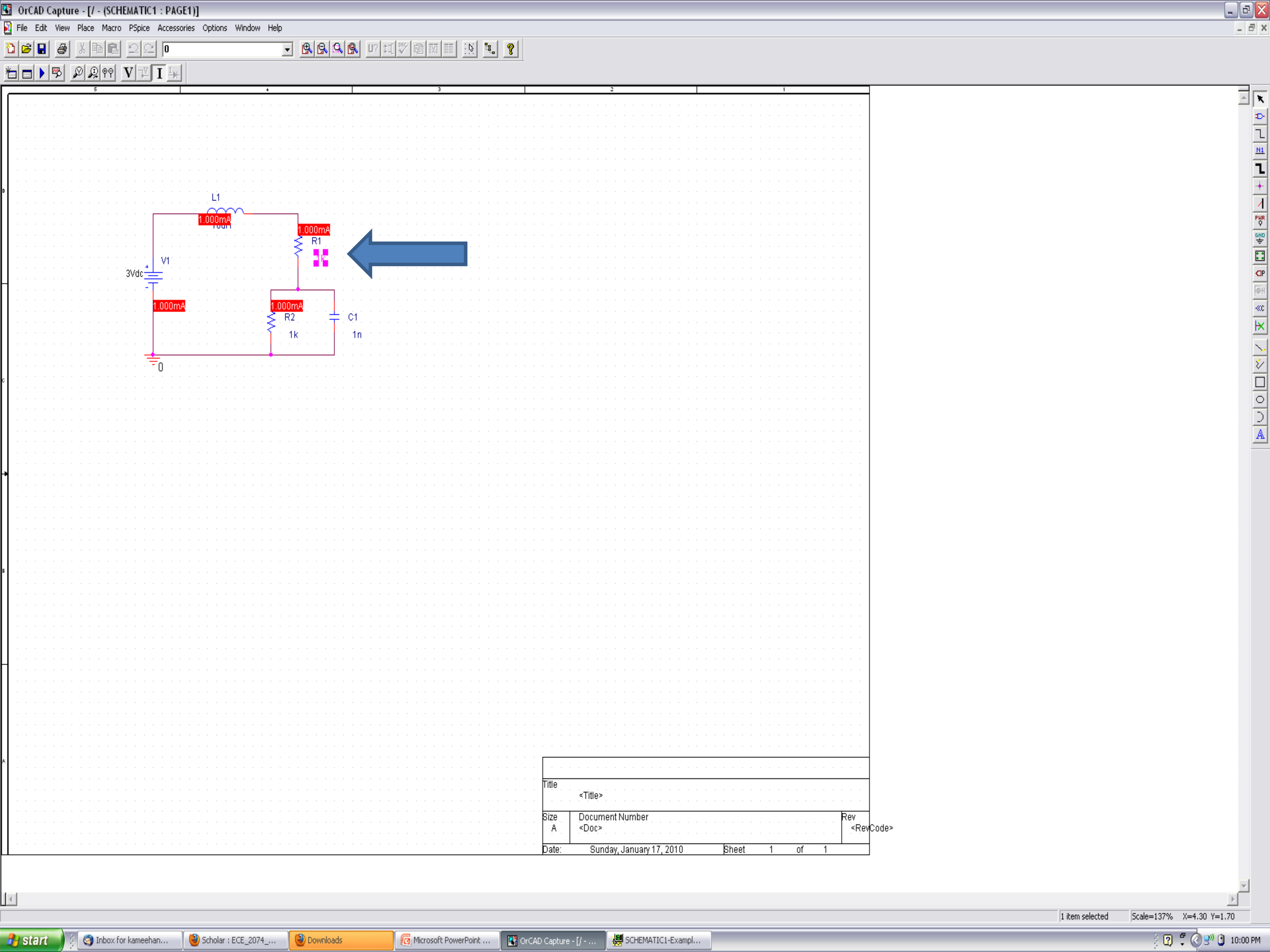


OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
L1
1.000mA
1.000mA
R1
2k
V1
3Vdc
1.000mA
1.000mA
R2
1k
C1
1n
0
SCHEMATIC1-Example1 - OrCAD PSpice A/D Demo - [example1-SCHEMATIC1-Example...
File Edit View Simulation Trace Plot Tools Window Help
SCHEMATIC1-Example1
0s
0.5us
1.0us
Time
example1-SC...
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point for Transient Analy
Bias point calculated
Transient Analysis
Transient Analysis finished
Simulation complete
Time step = 14.38E-09
Time = 1.000E-06
Analysis
Watch
Devices
For Help, press F1
Time= 1.000E-06
100%
Title
<Title>
Size
A
Document Number
<Doc>
Rev
<RevCode>
Date:
Sunday, January 17, 2010
Sheet
1
of
1
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
Microsoft PowerPoint ...
OrCAD Capture - [/ - ...
SCHEMATIC1-Exampl...
9:50 PM

Change the Value of Resistor R1

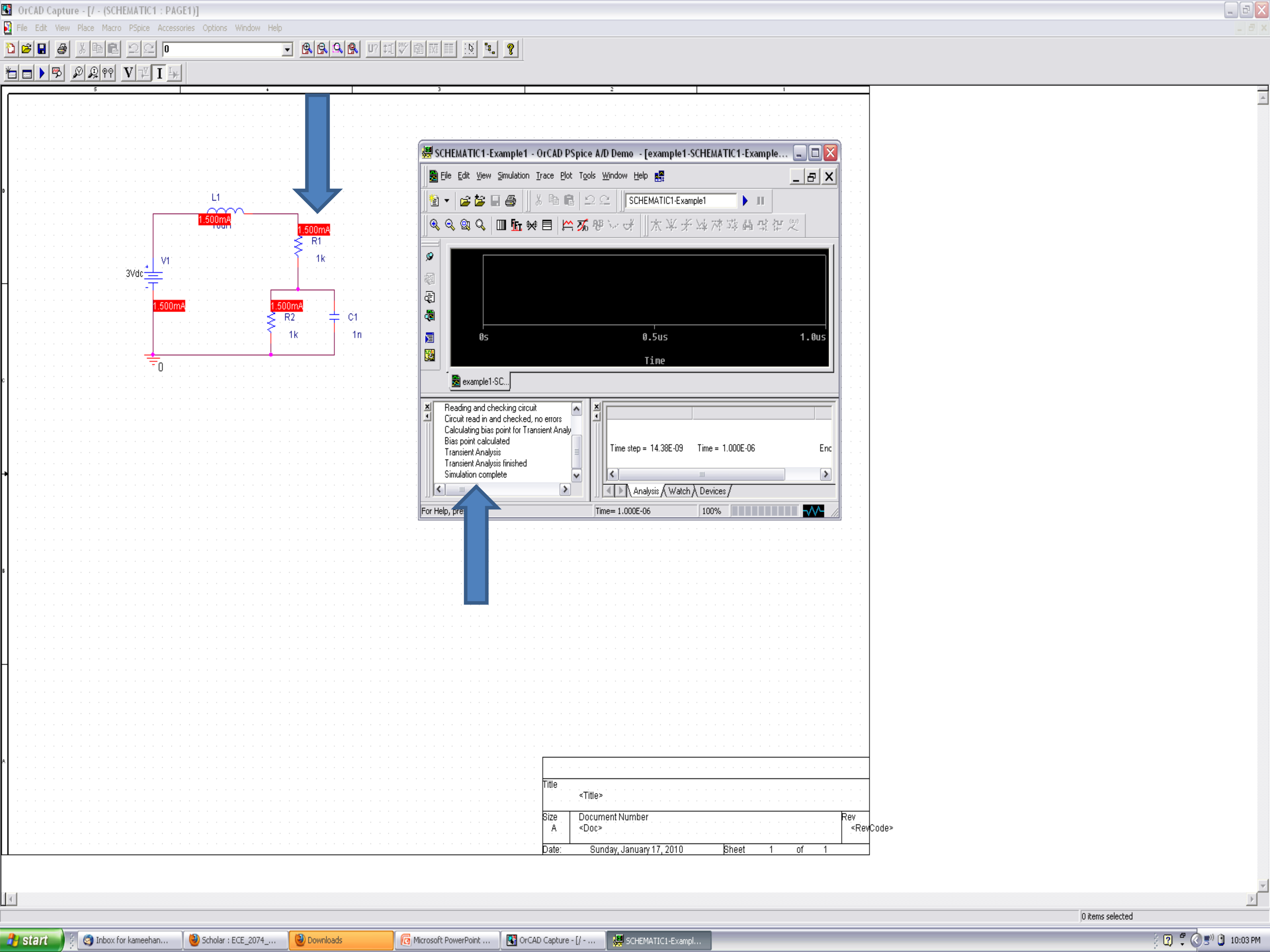
- Double click on the **2k** that is below **R1**.
  - A dashed box will surround the **2k** by **R1**.
  - A pop-up window called Set Attribute Value will open. The VALUE, 2k, will be highlighted.
  - Type 1k in the box and click OK.
    - The value of R1 printed to the right of the resistor symbol should now be **1k**.


OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
L1
1.000mA
R1
V1
3Vdc
R2
1k
C1
1n
0
Display Properties
Name: Value
Value: 2k
Font
Arial 7 (default)
Change...
Use Default
Display Format
Do Not Display
Value Only
Name and Value
Name Only
Both if Value Exists
Color
Default
Rotation
0°
180°
90°
270°
OK
Cancel
Help
Title
<Title>
Size
A
Document Number
<Doc>
Rev
<RevCode>
Date: Sunday, January 17, 2010
Sheet 1 of 1
1 item selected
Scale=137%
X=3.50 Y=1.50
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
Microsoft PowerPoint ...
OrCAD Capture - [/ - ...
SCHEMATIC1-Exampl...
9:52 PM


OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
L1
1.000mA
1.000mA
R1
V1
3Vdc
1.000mA
1.000mA
R2
1k
C1
1n
0
Title
<Title>
Size
A
Document Number
<Doc>
Rev
<RevCode>
Date: Sunday, January 17, 2010
Sheet 1 of 1
1 item selected
Scale=137%
X=4.30 Y=1.70
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
Microsoft PowerPoint ...
OrCAD Capture - [/ - ...
SCHEMATIC1-Exampl...
10:00 PM

Run a Simulation with New Value

- Click on the Run PSpice icon (blue arrowhead) that is located on the second row of buttons, third from the left.
  - If the program has installed properly:
    - A pop-up window called SCHEMATIC1-Example1 – ORCAD A/D DEMO should open and the words **Simulation complete** should be printed in the lower left.
    - The values for the currents that are flowing through the components in the circuit should change to **1.500mA**.


OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Place Macro PSpice Accessories Options Window Help
L1
1.500mA
R1
1k
V1
3Vdc
1.500mA
1.500mA
R2
1k
C1
1n
0
SCHEMATIC1-Example1 - OrCAD PSpice A/D Demo - [example1-SCHEMATIC1-Example...
File Edit View Simulation Trace Plot Tools Window Help
SCHEMATIC1-Example1
0s
0.5us
1.0us
Time
example1-SC...
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point for Transient Analy
Bias point calculated
Transient Analysis
Transient Analysis finished
Simulation complete
Time step = 14.38E-09
Time = 1.000E-06
Analysis
Watch
Devices
Time= 1.000E-06
100%
Title
<Title>
Size
A
Document Number
<Doc>
Rev
<RevCode>
Date:
Sunday, January 17, 2010
Sheet
1
of
1
0 items selected
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
Microsoft PowerPoint ...
OrCAD Capture - [/ - ...
SCHEMATIC1-Exampl...
10:03 PM

If There was No Change in the Current

- If the value of the current did not change from 1.000mA, then the installation of PSpice was not completed.
  - Close Schematics and make sure that all of the other PSpice applications have also closed.
    - A few of them will need to be closed manually.
  - Using the option under Control Panel, uninstall PSpice and then reinstall it using the Setup file that was unzipped from 91pspstu.exe. Then, repeat the instructions in this PowerPoint file.

ECE IT Staff

- If the program fails to run properly a second time, please see Mr. Branden McKagen of the ECE IT group for assistance.
  - His office is 346 Whittemore Hall.
  - He is generally available from:
    - 9am – noon on Monday through Friday
    - 1:15pm – 5pm on Monday, Wednesday, and Friday