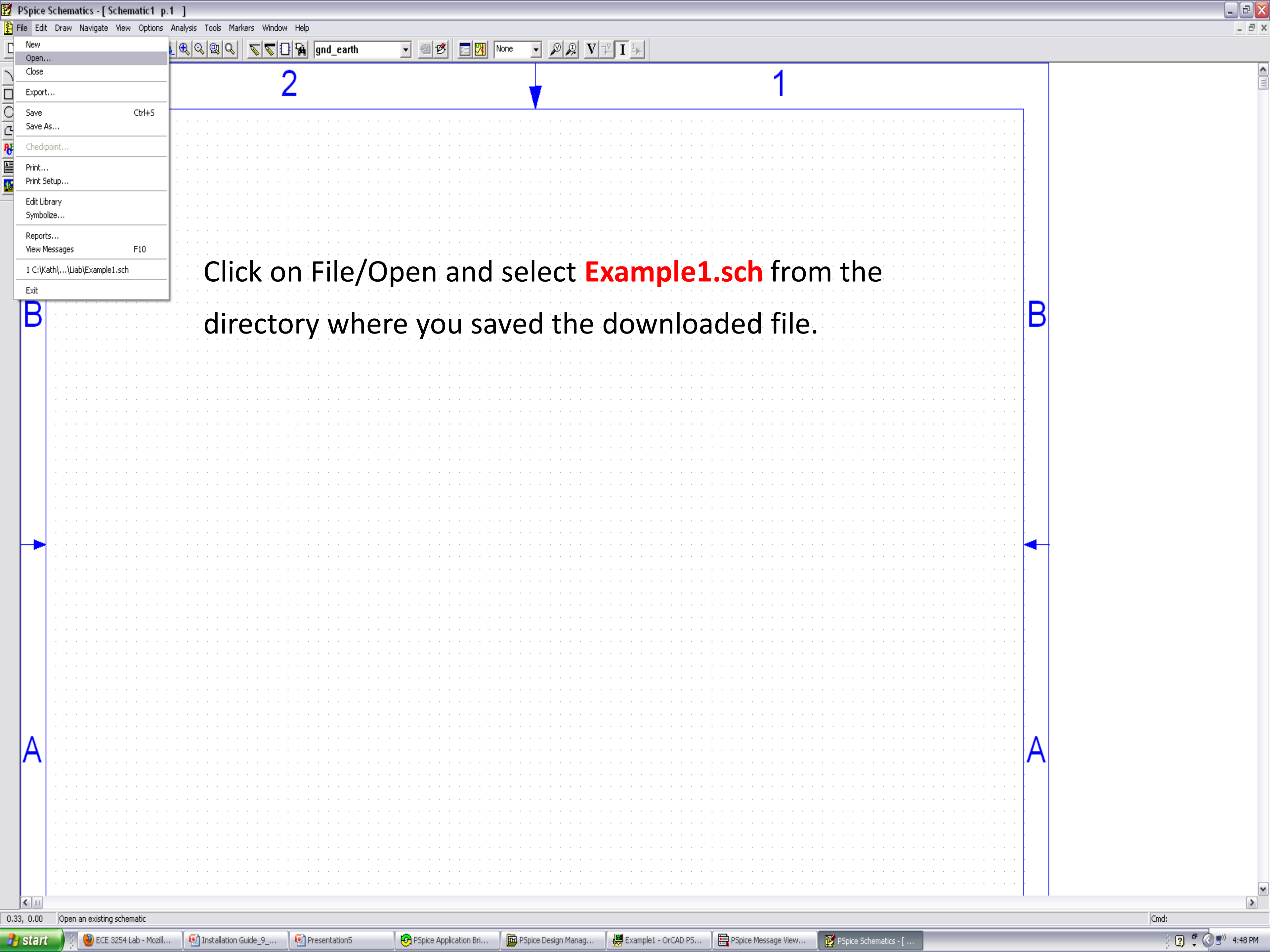
Verification of Proper Installation of 9.1 Schematics

File for Simulation Test

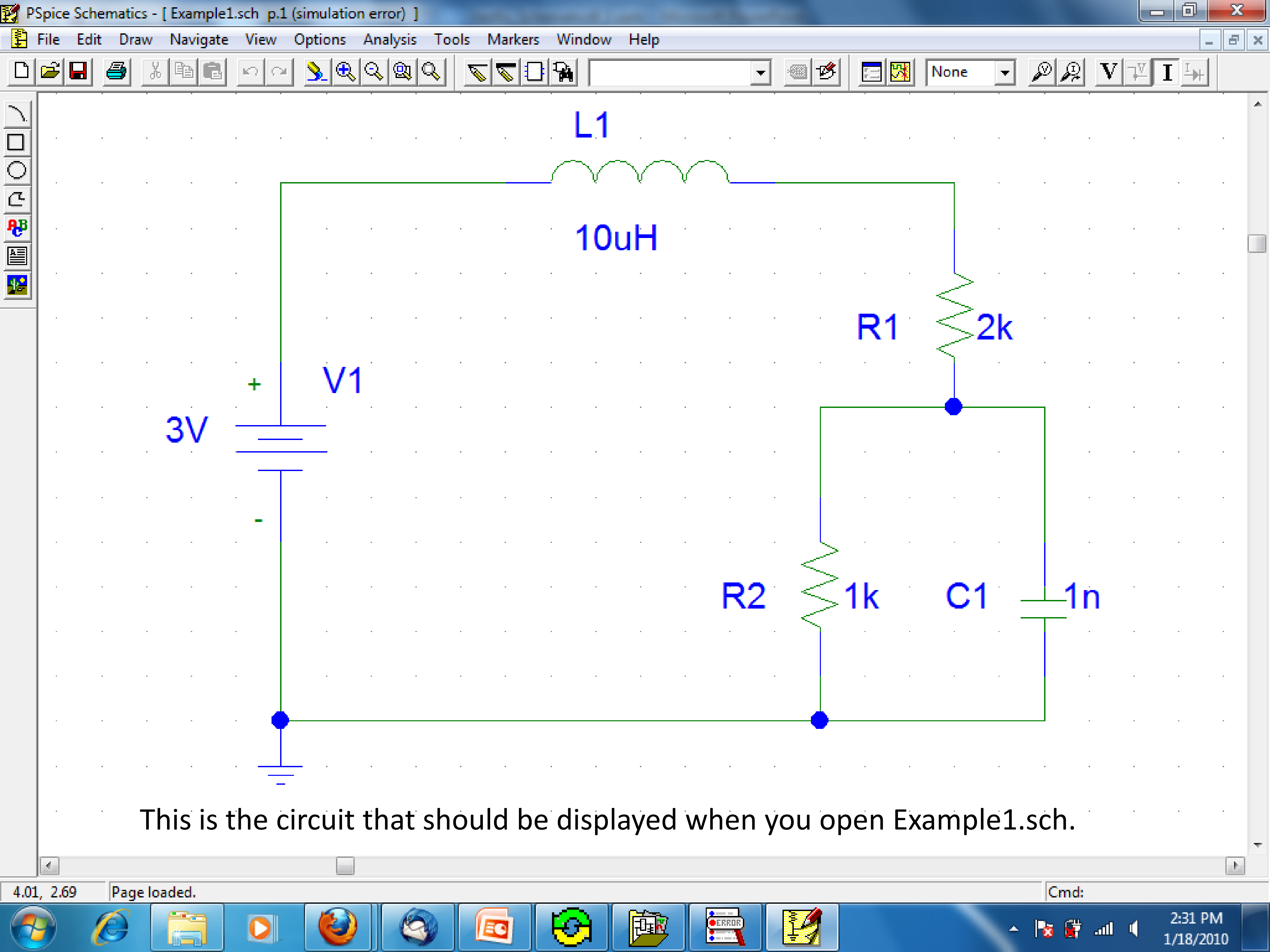
- Download Example1.sch from the ECE 2074 Scholar site under Resources in the folder called:

  Technical Support: Circuit Simulation/Version 9.1 /Schematics/Installation Verification

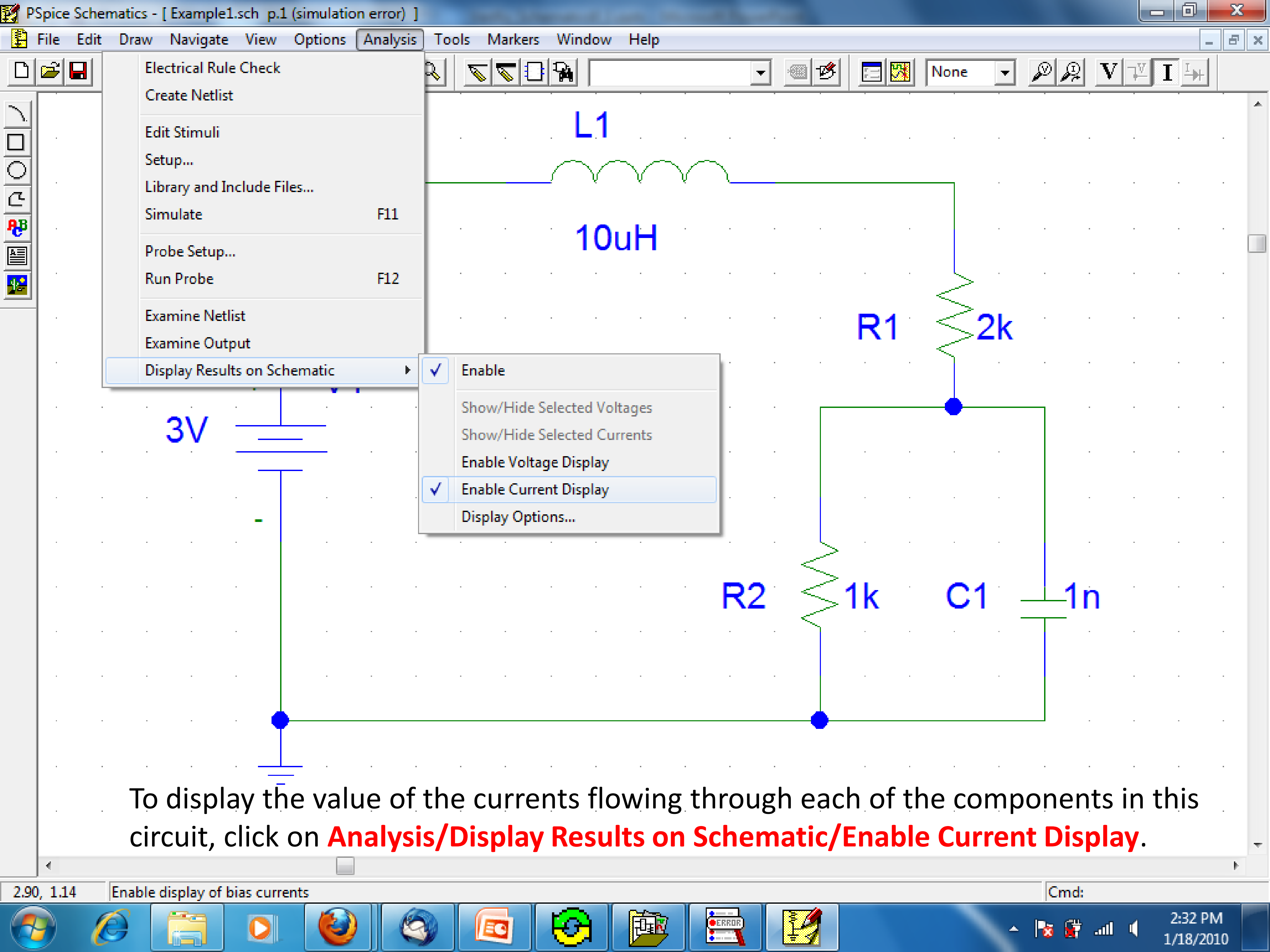
- Then, run PSpice Student/Schematics.
  - Several applications should launch
    - PSpice Application Bridge
    - PSpice Device Manager
    - PSpice Message Viewer
    - **PSpice Schematics**
      - **The screen for this application automatically be the one that you are viewing.**



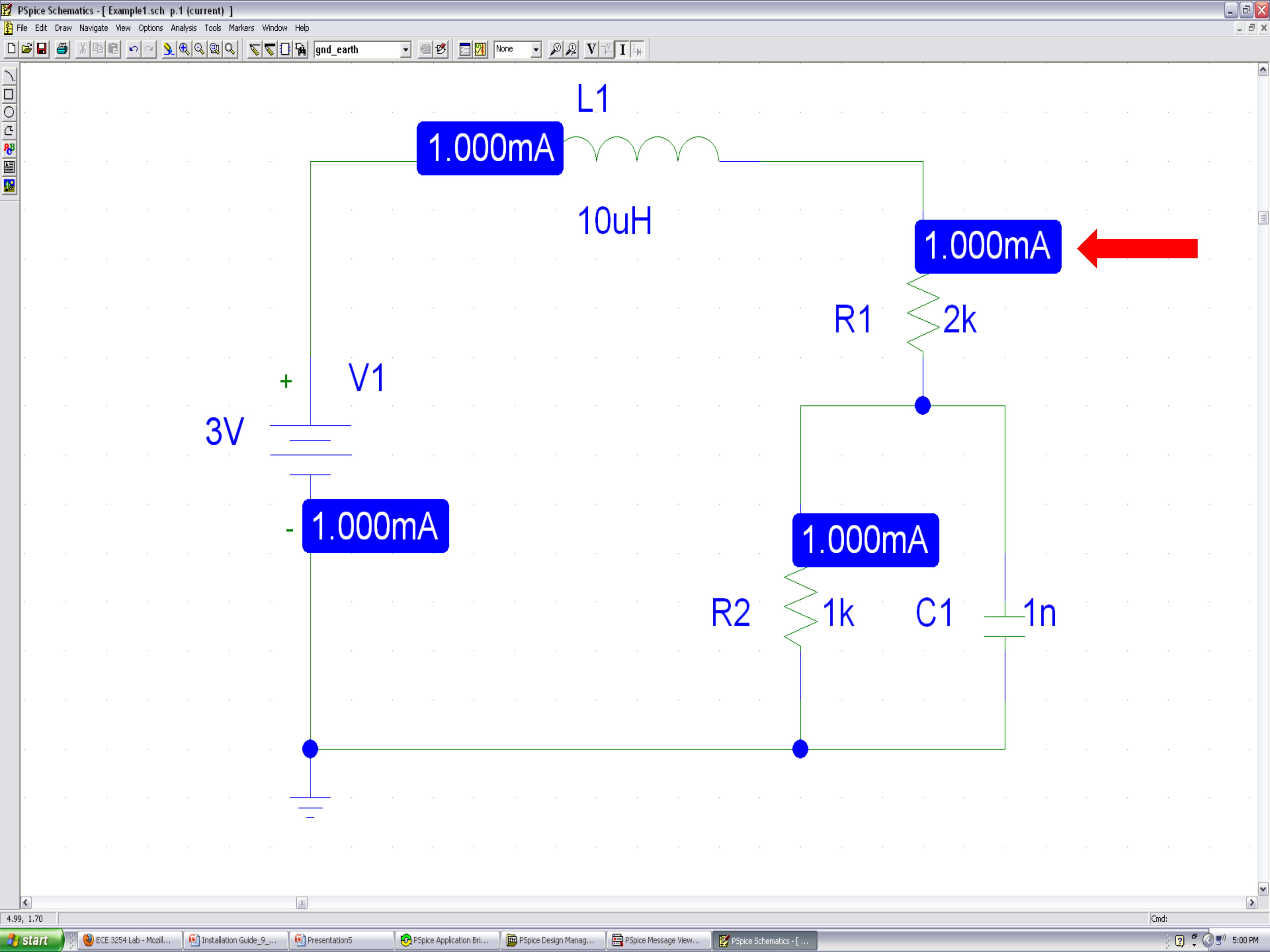
Click on File/Open and select **Example1.sch** from the directory where you saved the downloaded file.



This is the circuit that should be displayed when you open Example1.sch.

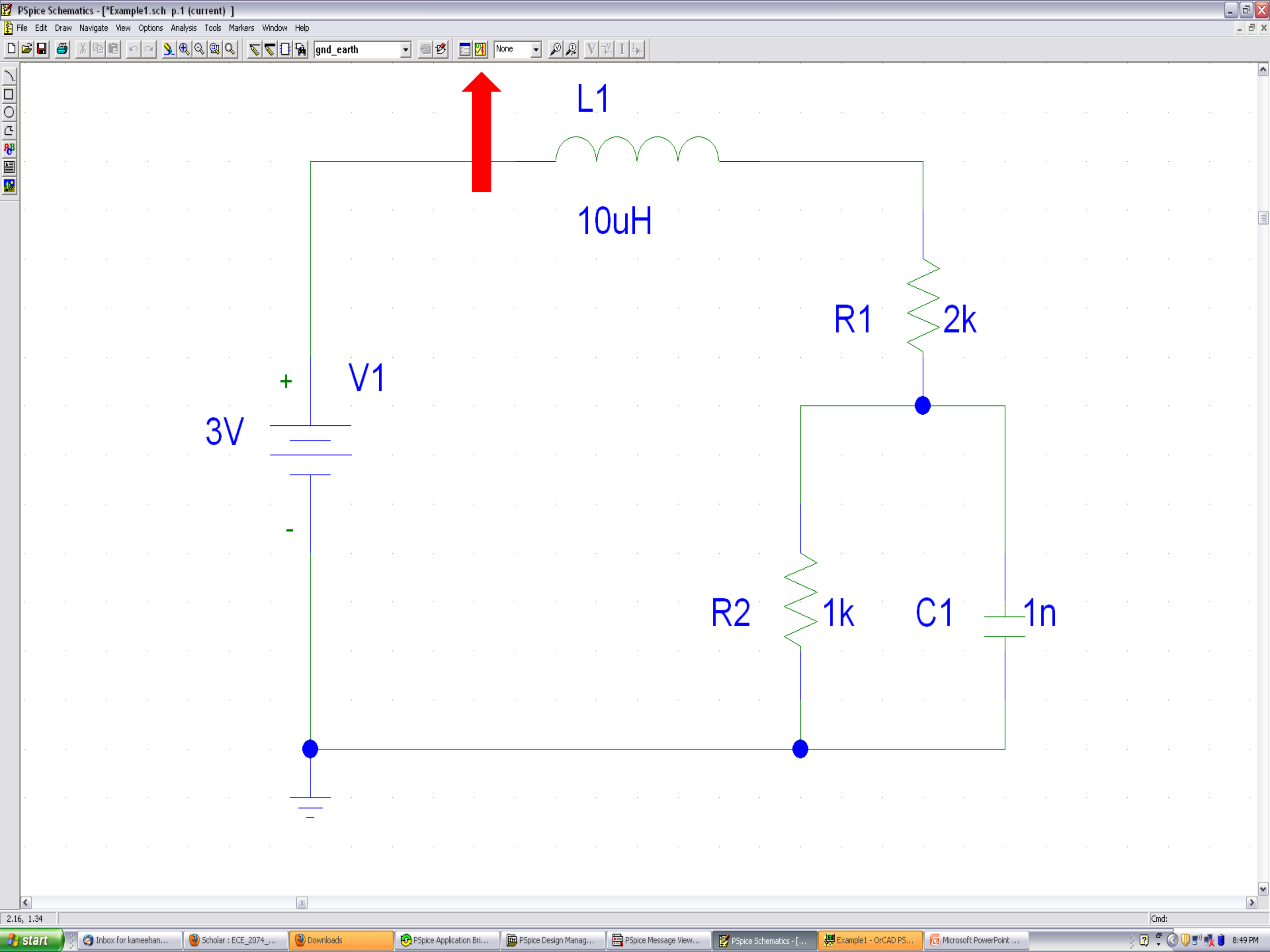


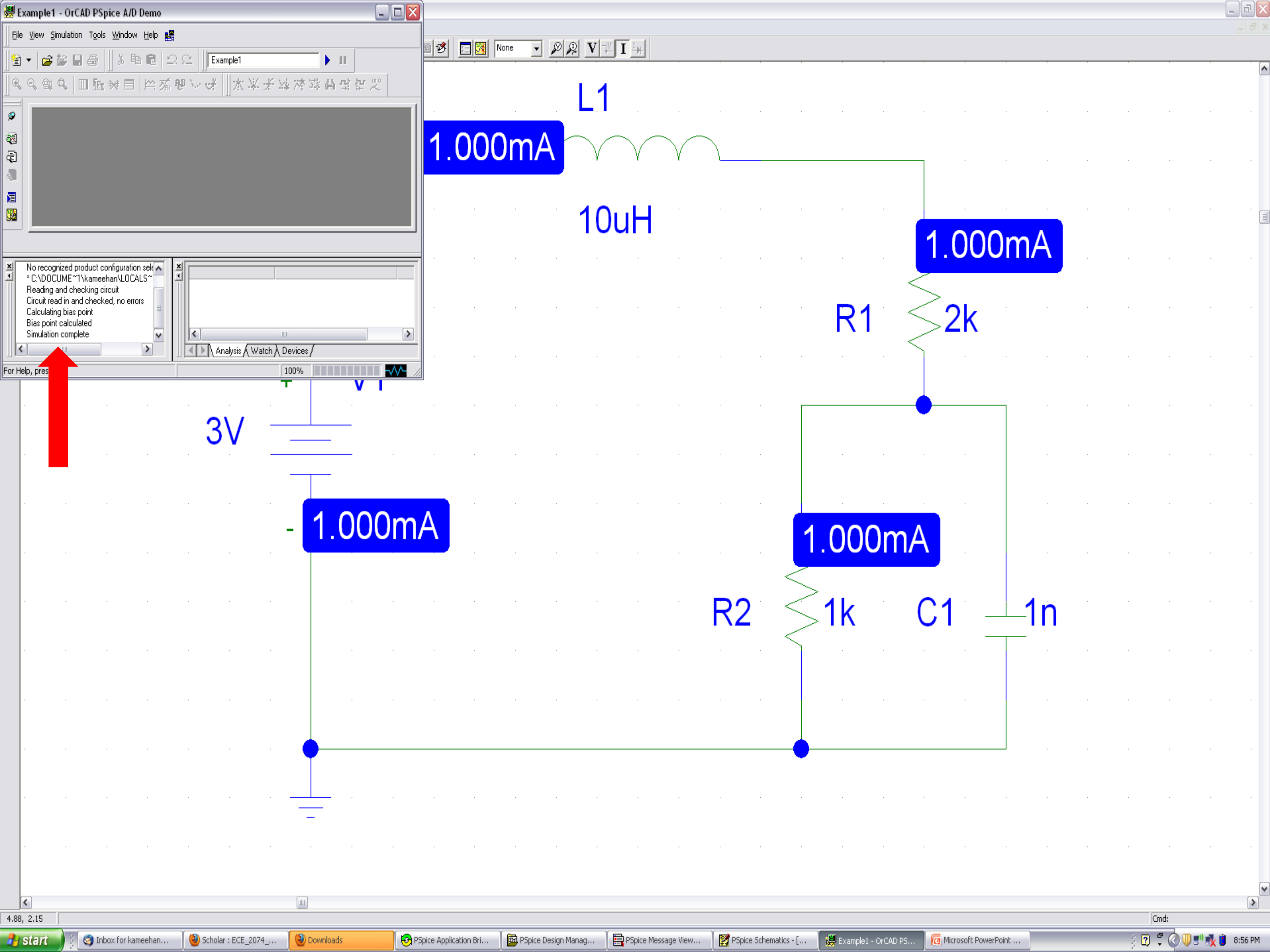
To display the value of the currents flowing through each of the components in this circuit, click on **Analysis/Display Results on Schematic/Enable Current Display**.


PSpice Schematics - [Example1.sch p.1 (current)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
1.000mA
10uH
1.000mA
R1
2k
V1
3V
1.000mA
1.000mA
R2
1k
C1
1n
4.99, 1.70
Cmd:
start
ECE 3254 Lab - Mozil...
Installation Guide_9_...
Presentation5
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
5:00 PM

If the Currents aren't Shown

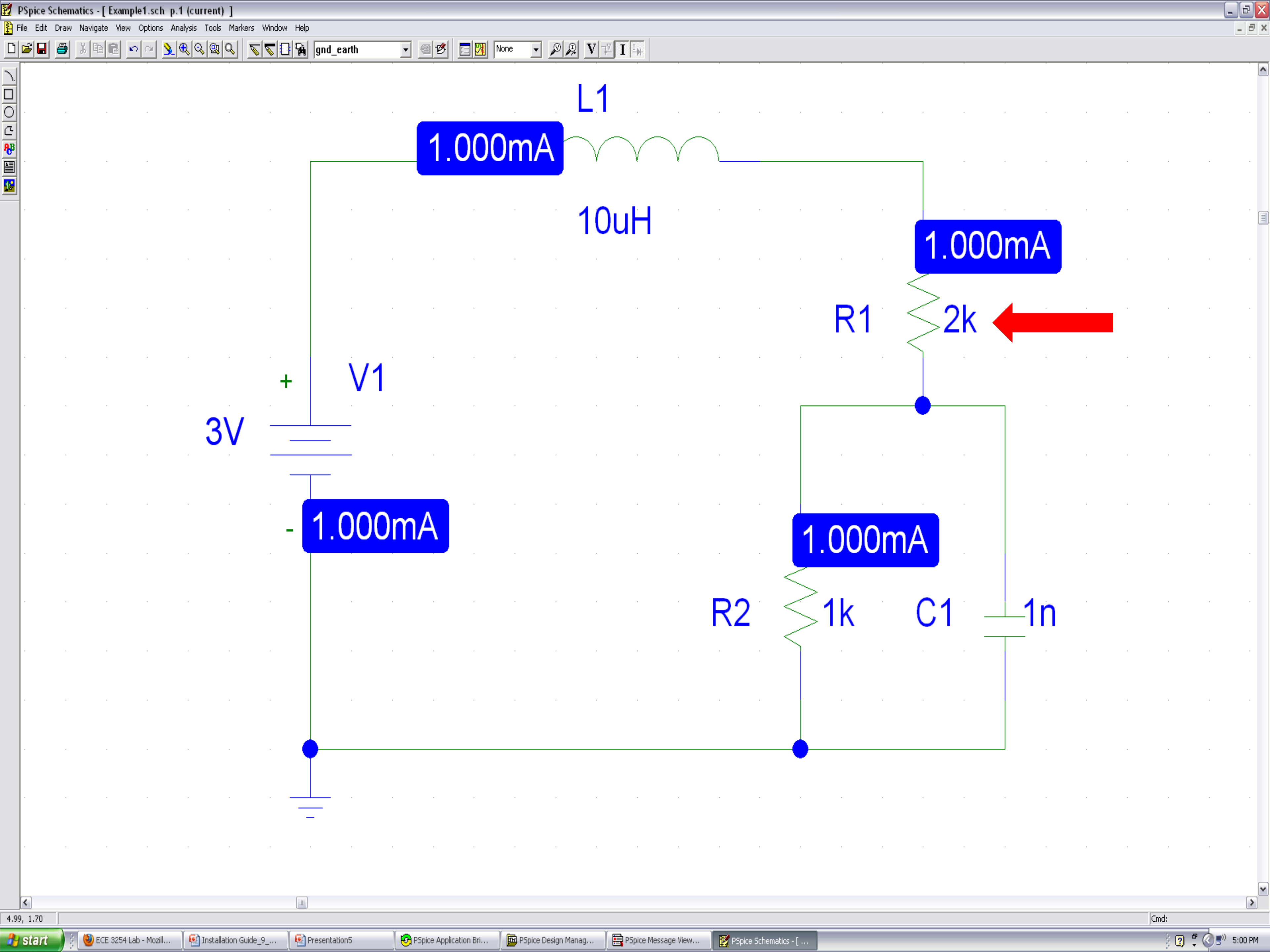
- Click on the Simulate button to run the calculations.
  - The values for the currents through the components should appear on the circuit after **Simulation complete** appears in the pop-up box called Example1 – ORCAD A/D DEMO.

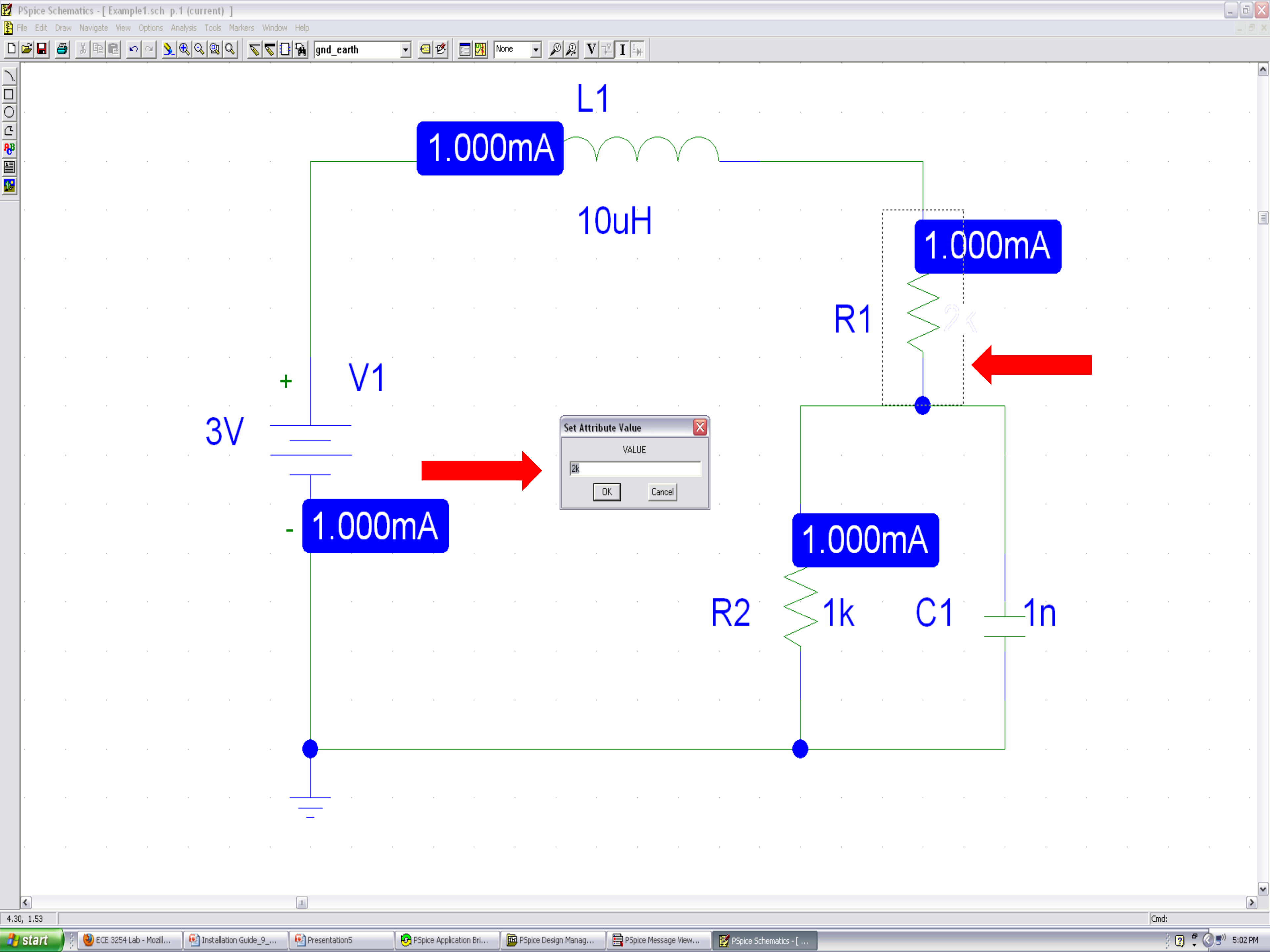

PSpice Schematics - [*Example1.sch p.1 (current)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
10uH
R1
2k
V1
3V
+
-
R2
1k
C1
1n
2.16, 1.34
Cmd:
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
Example1 - OrCAD PS...
Microsoft PowerPoint ...
8:49 PM

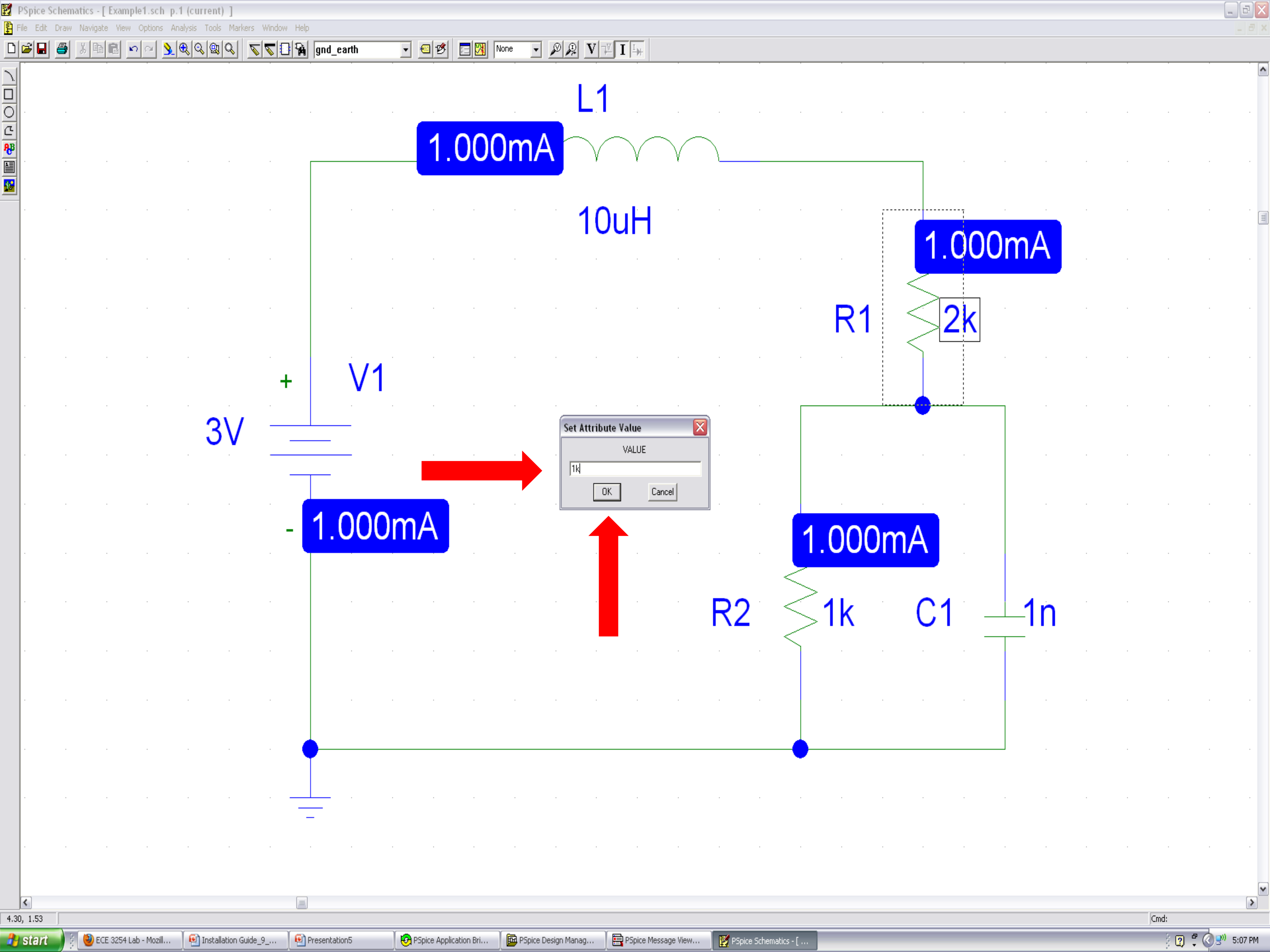

Example1 - OrCAD PSpice A/D Demo
File View Simulation Tools Window Help
Example1
No recognized product configuration sele
* C:\DOCUME~1\kameehan\LOCALS~
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point
Bias point calculated
Simulation complete
Analysis Watch Devices
For Help, pres
100%
None
L1
1.000mA
10uH
1.000mA
R1
2k
3V
1.000mA
1.000mA
R2
1k
C1
1n
4.88, 2.15
Cmd:
start
Inbox for kameehan...
Scholar : ECE_2074_...
Downloads
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
Example1 - OrCAD PS...
Microsoft PowerPoint ...
8:56 PM

Change the Value of Resistor R1

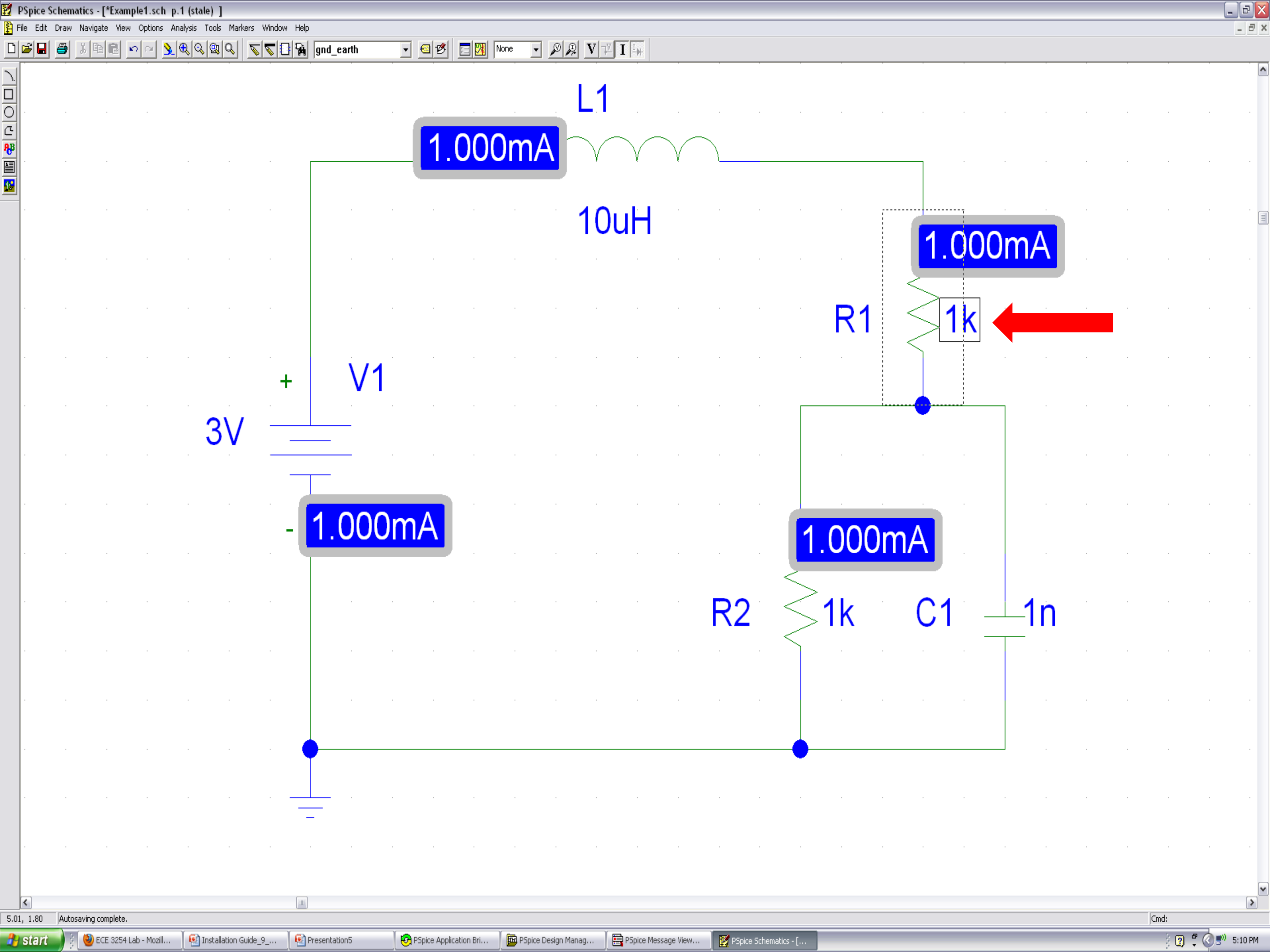
- Double click on the **2k** that is located to the right of **R1**.
  - A dashed box will surround the resistor symbol by R1.
  - A pop-up window called Set Attribute Value will open. The VALUE, 2k, will be highlighted.
  - Type 1k in the box and click OK.
    - The value of R1 printed to the right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


PSpice Schematics - [Example1.sch p.1 (current)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
1.000mA
10uH
1.000mA
R1
2k
V1
+
3V
-
1.000mA
1.000mA
R2
1k
C1
1n
4.99, 1.70
Cmd:
start
ECE 3254 Lab - Mozill...
Installation Guide_9_...
Presentation5
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
5:00 PM


PSpice Schematics - [Example1.sch p.1 (current)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
1.000mA
10uH
1.000mA
R1
V1
+
3V
-
1.000mA
Set Attribute Value
VALUE
2k
OK
Cancel
1.000mA
R2
1k
C1
1n
4.30, 1.53
Cmd:
start
ECE 3254 Lab - Mozil...
Installation Guide_9_...
Presentation5
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
5:02 PM

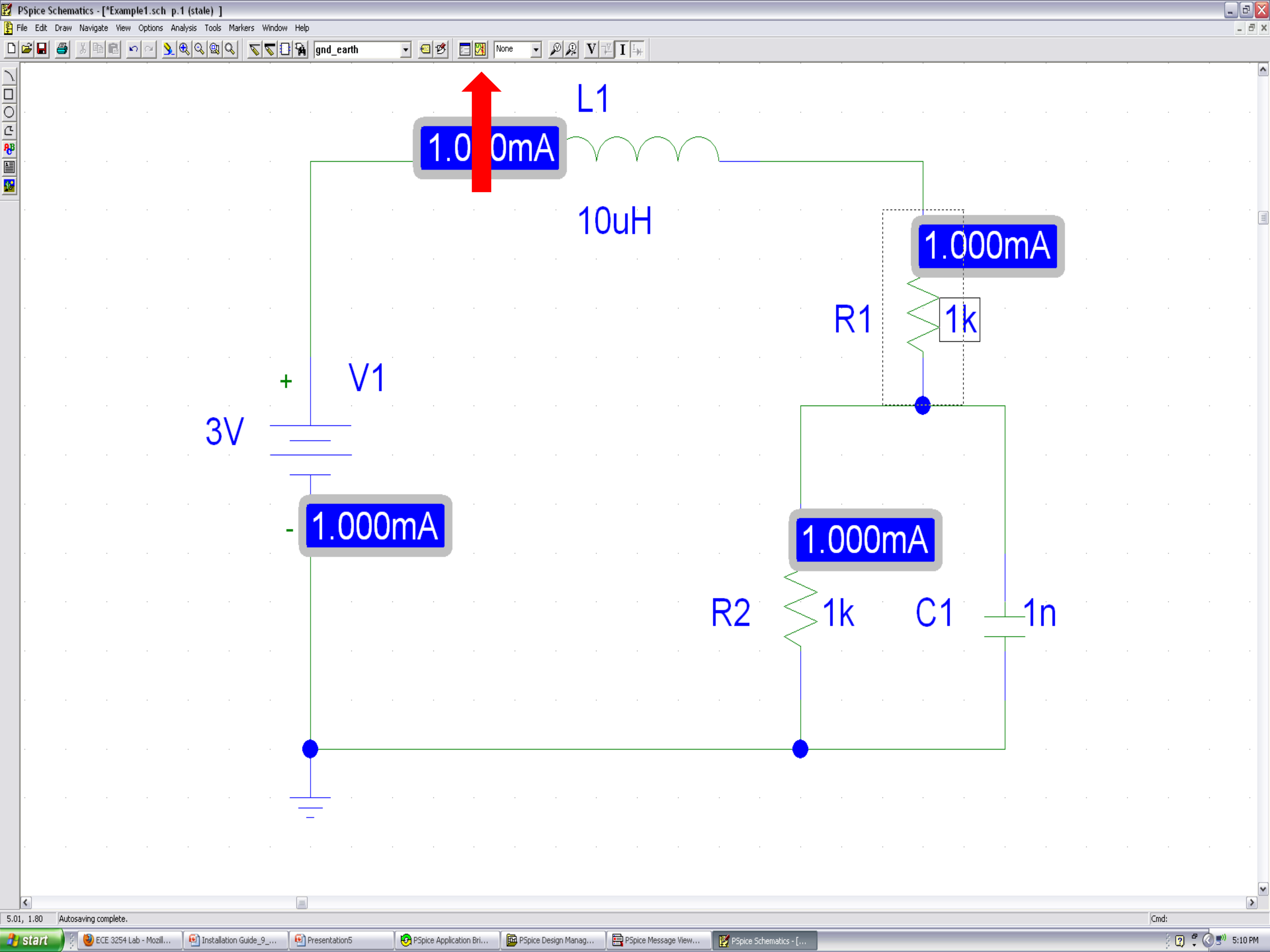


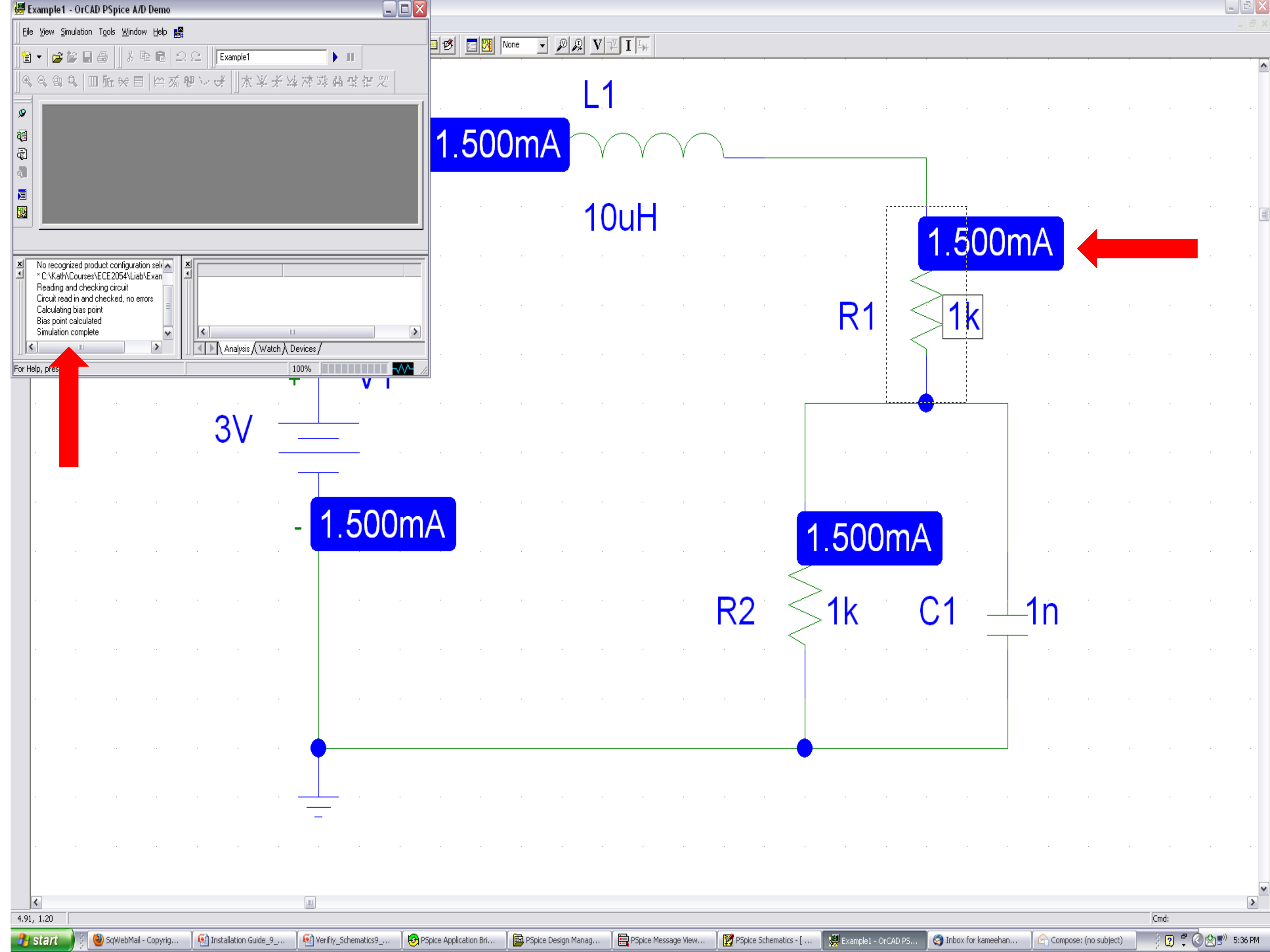
PSpice Schematics - [Example1.sch p.1 (current)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
1.000mA
10uH
1.000mA
R1
2k
V1
+
3V
Set Attribute Value
VALUE
1k
OK
Cancel
-
1.000mA
1.000mA
R2
1k
C1
1n
4.30, 1.53
Cmd:
start
ECE 3254 Lab - Mozil...
Installation Guide_9_...
Presentation5
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
5:07 PM


PSpice Schematics - [*Example1.sch p.1 (stale)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
1.000mA
10uH
1.000mA
R1
1k
V1
+
3V
-
1.000mA
1.000mA
R2
1k
C1
1n
5.01, 1.80
Autosaving complete.
Cmd:
start
ECE 3254 Lab - Mozil...
Installation Guide_9_...
Presentation5
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
5:10 PM

Run a Simulation with New Value

- Click on the yellow icon that is located roughly in the middle of the toolbar at the top of the screen.
  - If the program has installed properly:
    - A pop-up window called 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**.


PSpice Schematics - [*Example1.sch p.1 (stale)]
File Edit Draw Navigate View Options Analysis Tools Markers Window Help
gnd_earth
None
L1
1.0 0mA
10uH
1.000mA
R1
1k
V1
+
3V
-
1.000mA
1.000mA
R2
1k
C1
1n
5.01, 1.80
Autosaving complete.
Cmd:
start
ECE 3254 Lab - Mozil...
Installation Guide_9_...
Presentation5
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
5:10 PM


Example1 - OrCAD PSpice A/D Demo
File View Simulation Tools Window Help
Example1
No recognized product configuration sele
* C:\Kath\Courses\ECE2054\Liab\Exan
Reading and checking circuit
Circuit read in and checked, no errors
Calculating bias point
Bias point calculated
Simulation complete
Analysis Watch Devices
For Help, pres
100%
L1
1.500mA
10uH
1.500mA
R1
1k
3V
1.500mA
1.500mA
R2
1k
C1
1n
4.91, 1.20
Cmd:
start
SqWebMail - Copyrig...
Installation Guide_9_...
Verify_Schematics9_...
PSpice Application Bri...
PSpice Design Manag...
PSpice Message View...
PSpice Schematics - [...
Example1 - OrCAD PS...
Inbox for kameehan...
Compose: (no subject)
5:36 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