

## Objectives

1. Create a new project in OrCAD.
2. Design a circuit in OrCAD.
3. Run simulations on the designed circuit.

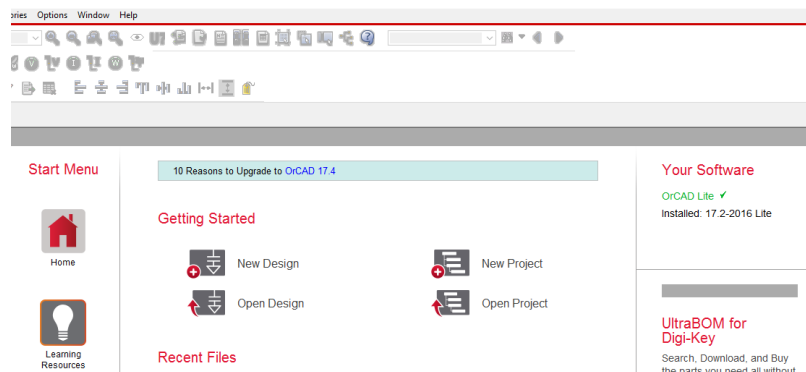
## Preparation work

You need to have access to OrCAD. The software is available in computer lab T41. You can access it physically or remotely via <https://remotelab.sys.kth.se/Labs/Account/Login>.

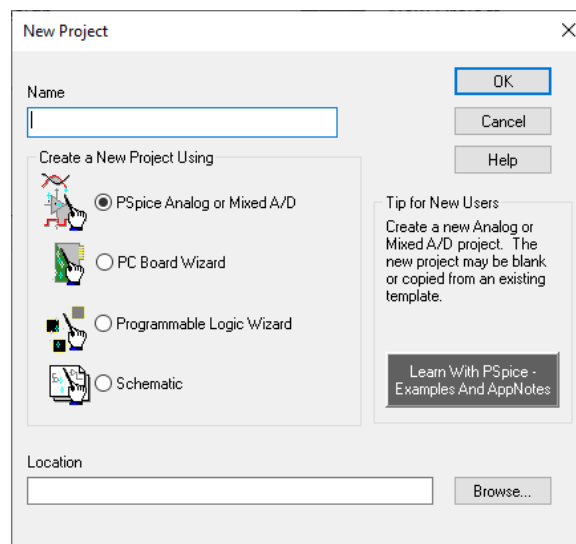
## Part 1. Create a new project in OrCAD

Open OrCAD CIS Lite (inside folder Cadence) from the Start Menu.

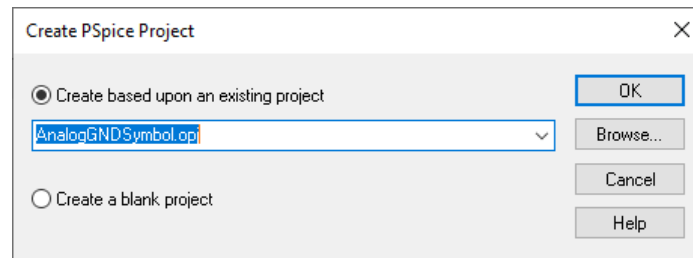
Create a new project by selecting “**File**”, then “**New**”, then “**Project**” or choosing **New Project** from the start page.



When you have selected the project, the following panel will be displayed. In this panel, you need to assign a project a **Name**, select **Analog or Mixed A/D**, and choose a **Location** where to save your file.



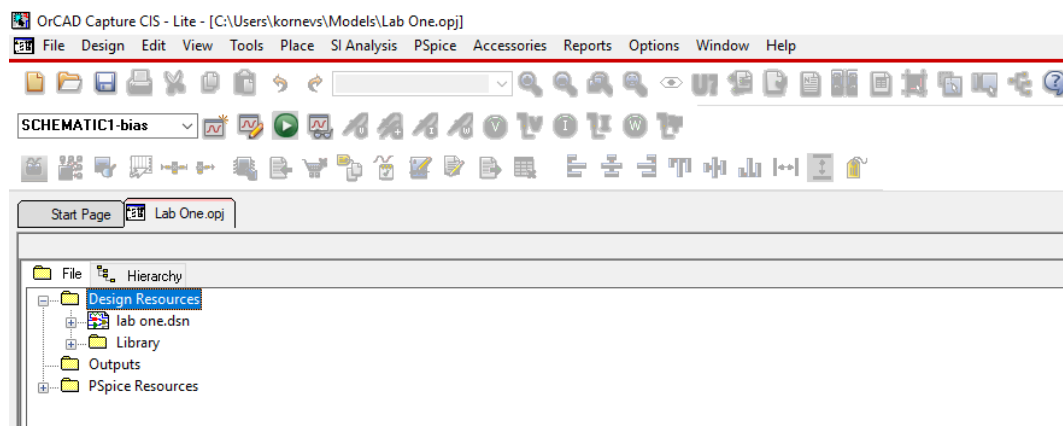
Give a name to your project and chose a location for it. Click “**OK**” and the following panel will appear.



If the project is not based in full or in part on another project, then select **“Create a blank project”** (at the bottom). If a copy of some other completed project is wanted as the starting point for the new project, select **“Create based upon an existing project”**, then select it from the “drop-down list or “Browse” for the location of that project (it will have a “.opj” extension). Note that none of the original project contents will be altered by any changes made to the new (copied) project.

In some cases, a blank project might be the selection; however, in most cases, the schematic will almost always require a ground, so selecting the existing project **“AnalogGNDsymbol.opj”** will start the schematic with a ground. If you select the “blank project” option, the ground will have to be placed separately.

Select “AnalogGNDsymbol.opj” and click OK. It might ask for a Cadence Product Choice, in this case, use OrCAD Lite – Pspice. The following display will be presented:



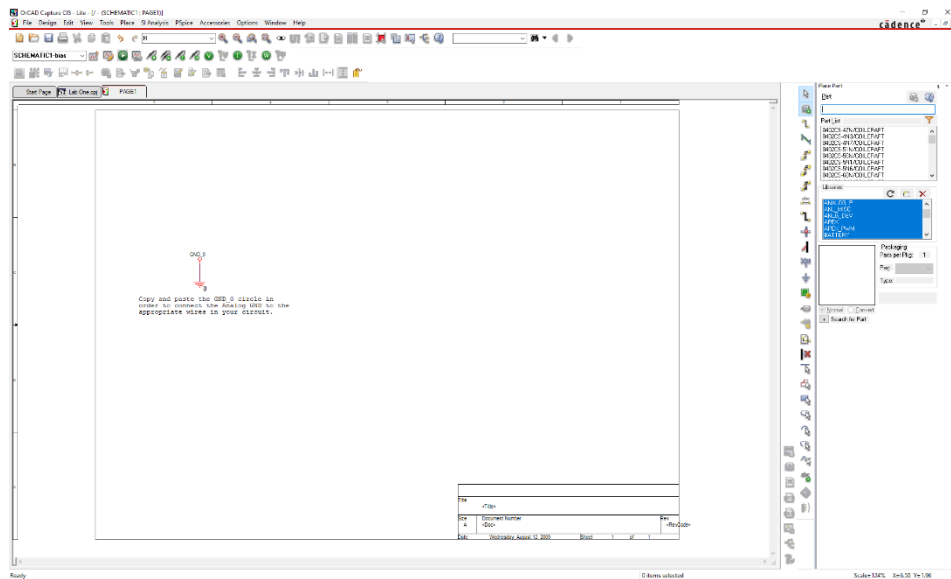
## Part 2. Design a circuit in OrCAD

Open the subfolder “XXXX.dsn”, where XXXX is your project name then subfolder “SCHEMATIC1” to see subfolder “PAGE1”. Click on “PAGE1” and get the page where you can draw your circuit.

Texts and parts can be selected by clicking on them. If a right-click is done within the box or highlighted area, a menu of options appears (Cut, Copy, Delete, Rotate, etc.).

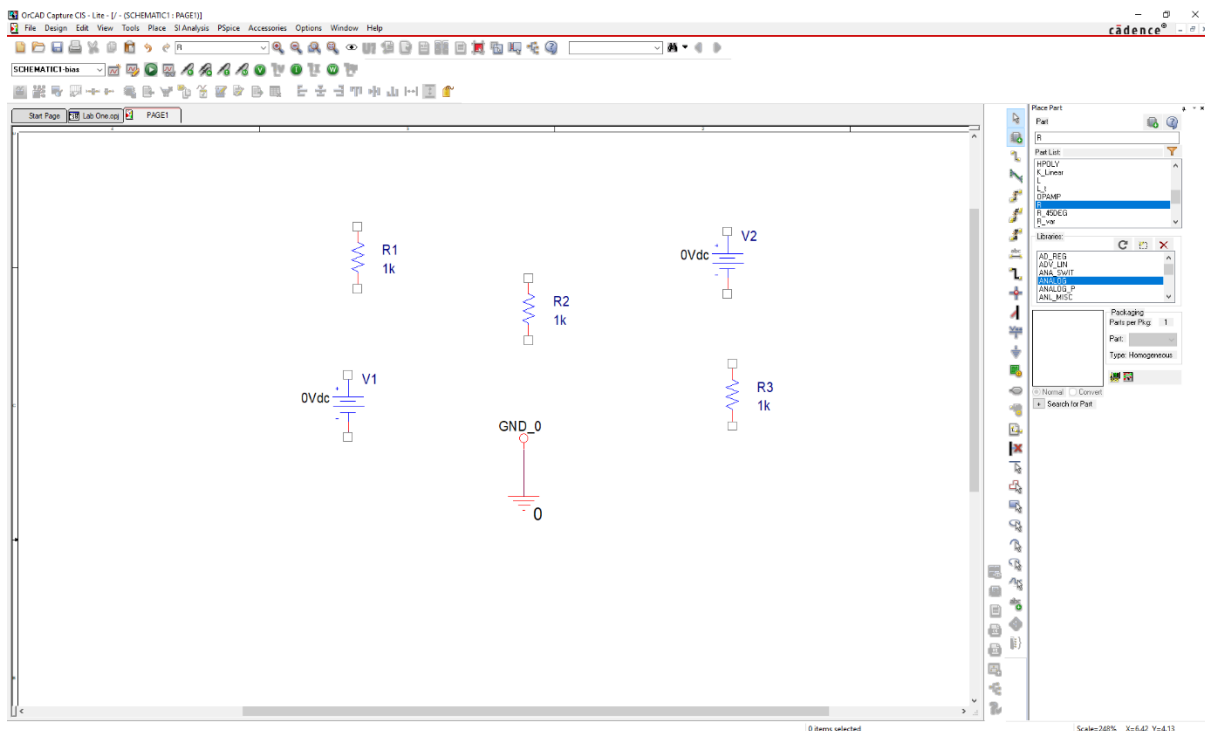
Select the text below the ground symbol and delete it.

Go to the top toolbar and select the “Place” tab, then the “Part” option. Explore those part options for the various libraries shown.

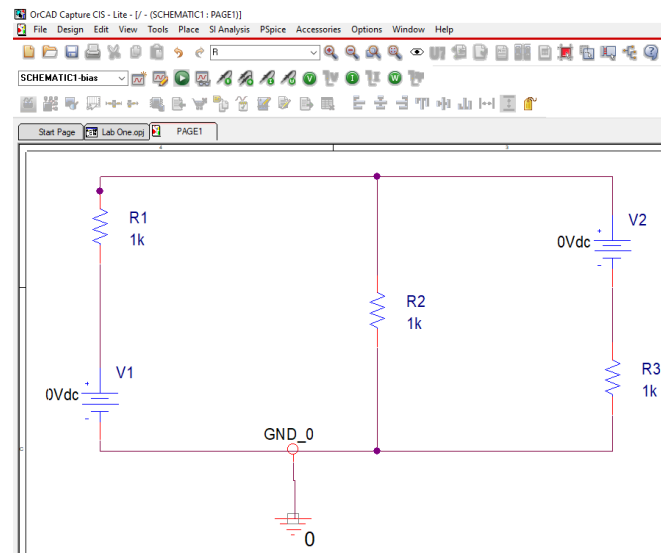


In the “SOURCE” library “Part List”, scroll down to find the “VDC” option and select it, then use either the “Enter” key from the keyboard or double click to select. drag the part to schematics and put two voltage sources. When parts are placed, use the “Esc” key to terminate the placement(s). If more than one was placed, highlight the extras and delete them. From the “Analog” library select the “R” and place three resistors in the schematic.

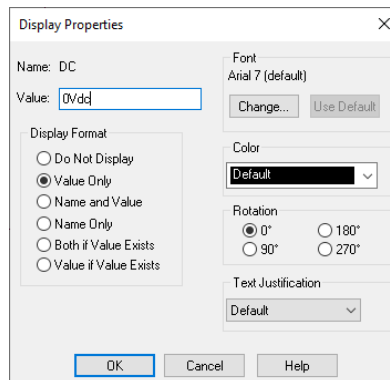
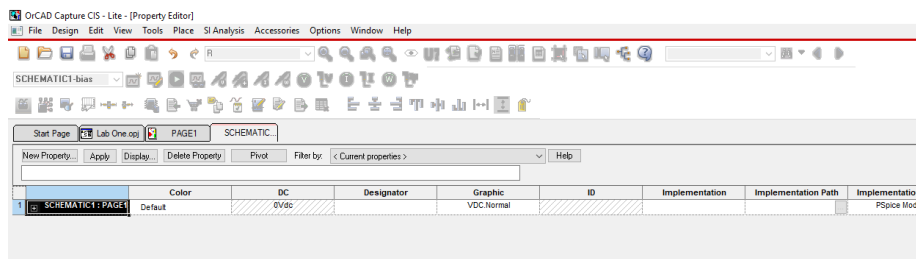
Now rotate one of the resistors (click on it, then right-click on the highlighted region and select “Rotate”), then move the parts and text around until it looks like the following display (page in a picture was zoomed in).



The last part of the process is to “wire” the parts together. Do this by going to the “Place” tab in the top toolbar, and selecting the “Wire” option. Use “click and drag” from part-tab to part-tab for each connection. Use “Esc” to end the process. If a mistake is made, highlight it, “Delete” and try again. To make sure that you have connected elements, you can try to drag them around – the wires should follow if they are connected properly. If it is hard to see the circuit, “click and drag” a box around the whole circuit, right-click in the highlighted area and select “Zoom In”. The display should be similar to the following.

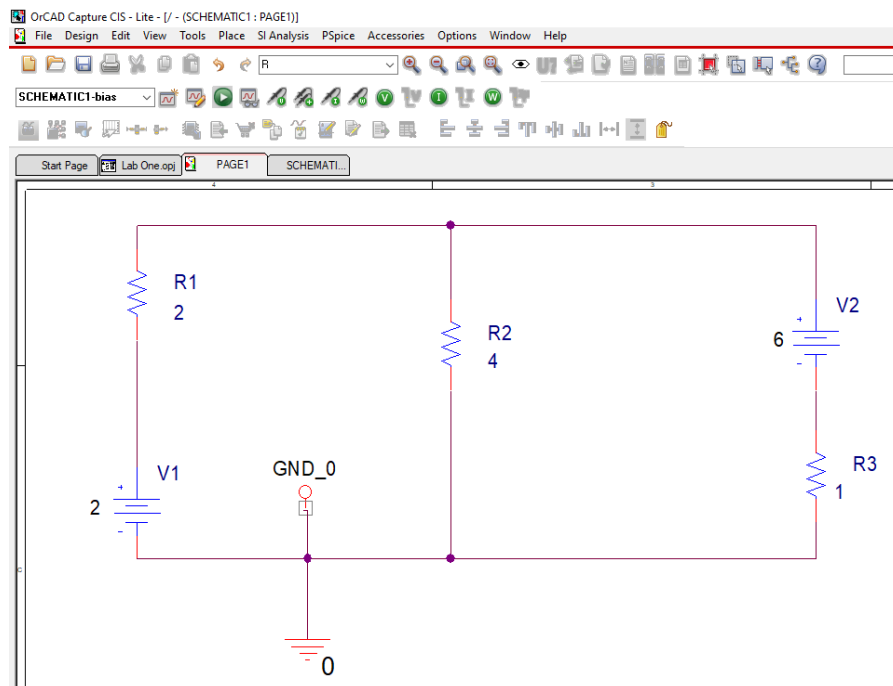


Double-click on each component to edit its value. You might have two types of menus: the first one is if you click on a text, for example, “0Vdc”. In this case, write a value in a **Value** field. The second menu is opened if you click on the element itself. It provides more options to change, find the field that holds the current value (**DC** for voltage sources or **Value** for resistors) and write proper value in it.

	Color	DC	Designator	Graphic	ID	Implementation	Implementation Path	Implementation
1 SCHEMATIC1: PAGE1	Default	0Vdc		VDC.Normal				PSpice Model

Set values as follows:  $R1=2\Omega$ ,  $R2=4\Omega$ ,  $R3=1\Omega$ ,  $V1=2V$  and  $V2=6V$ . The display should be similar to the following.

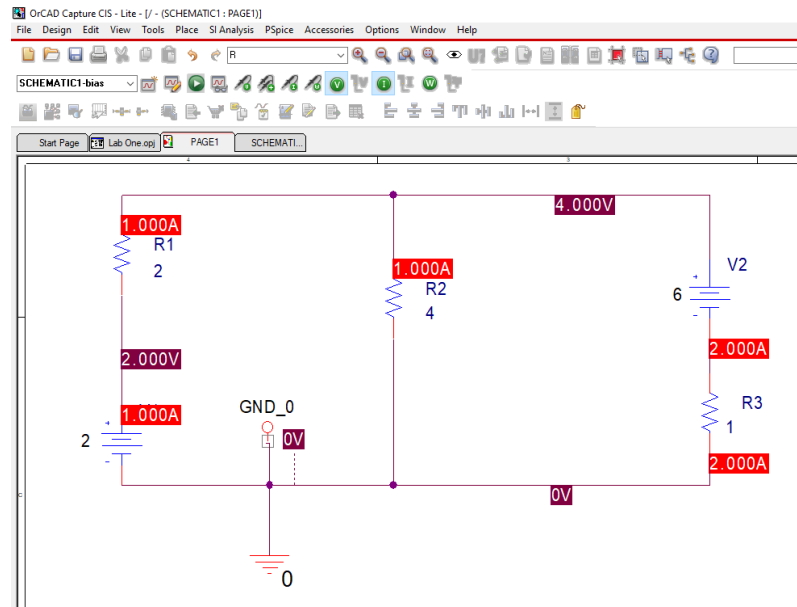


### Part 3. Run simulations on the designed circuit

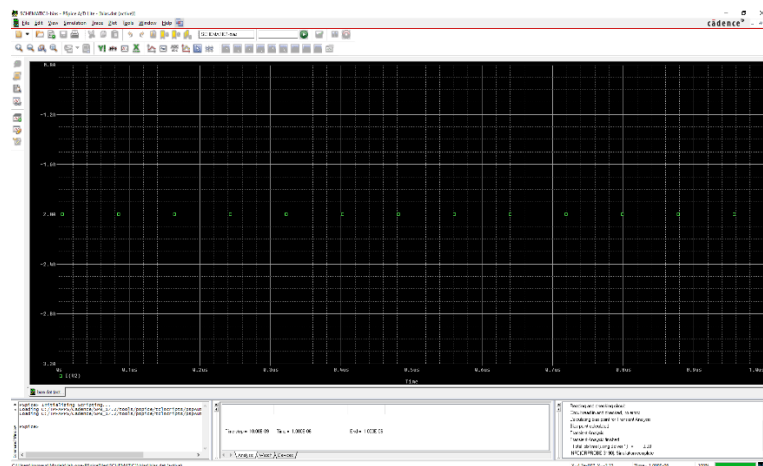
If it is desired to display the bias voltages and currents after the simulation has been run, select the “V” (Enable Bias Voltage Display) and/or “I” (Enable Bias Current Display) from the toolbars at the top. Alternatively, they can be selected using the “PSpice” drop-down tab in the top toolbar, then the option “Bias Points”. If the values displayed after simulating are in the way, they can be individually deleted or moved, or they can be collectively turned off by means of the “V” or “I”, etc., in the toolbar.



Select the “PSpice” drop-down tab from the top toolbar, then the option “Run”. If there are problems, make sure that points aren’t “floating”, meaning not fully connected. Make certain that the “o” above the ground is as shown above. To get back to the schematic, select the icon in the desktop toolbar for the schematic as shown below. When done, the schematic should look like the following.



Bias Point is valid for circuits where the values do not change over time. If we want to look for a value that can change over time, we need to open **Edit Simulation profile** and select Time Domain from **Analysis type**. Then press OK. Select a probe and connect it to the circuit. Run simulation. You should see a graph with selected values.



## Assignment

Simulate all circuits in Lab 2 and record all results in the Lab 2 report. OrCad does not have an ohmmeter to measure resistance, so you cannot directly measure  $R_{Th}$  or  $R_N$ .