Application Note

SPICE Models:
ROHM Voltage Detector ICs

BD48□□G/FVE,BD49□□G/FVE,BD52□□G/FVE,BD53□□G/FVE,
BD45□□□G,BD46□□□G,BU48□□G/FVE/F,BU49□□G/FVE/F,
BU42□□G/FVE/F,BU43□□G/FVE/F,BD47□□G

No.10006EAY01

1. INTRODUCTION

1.1 SPICE

SPICE is a general-purpose circuit-simulation program for nonlinear DC, nonlinear transient, and linear AC analysis. It solves network equations for node voltages and was designed to solve both linear and nonlinear electrical circuits. SPICE models simulate a variety of circuits with high accuracy, from switching power supplies to RAM cells and sense amplifiers. A variety of circuit components are supported, including resistors, capacitors, inductors, mutual inductors, dependent/independent voltage/current sources, transmission lines, and common semiconductor devices such as diodes, bipolar junction transistors (BITs), junction field effect transistors (JFETs), metal-oxide-semiconductor field effect transistors (MOSFETs), and metal-semiconductor FETs (MESFETs).

1.2 PSpice®

PSpice® is a SPICE analog circuit and digital logic simulation software that runs on personal computers.

1.3 OrCAD® PSpice 9.1

OrCAD® PSpice simulates analog-only circuits. After preparing a design for simulation, OrCAD Capture generates a circuit file set, which contains the circuit netlist and analysis commands. The circuit file set is read by PSpice A/D for simulation. PSpice A/D formulates these into meaningful graphical plots, which can be marked for display directly from the schematic page.

OrCAD PSpice 9.1 (Student Version) is PSpice simulation software that is available for free but with certain limitations imposed on the libraries and functionality. A fully functional version can be purchased from Cadence Design Systems, Inc.

1.4 SPICE Models: ROHM Voltage Detector ICs

ROHM SPICE model is made by typical data, and the manufacturing dispersions are not included.
Moreover, it does not guarantee all of the simulation result of execution by using this.
This model is being supplied as a aid to confirm the validity of a design approach and help to select surrounding component values.
While it reflects reasonably close to similarity to the actual device in terms of performance, it is not suggested as a replacement for breadboarding.
Simulation should be used as a forerunner or a supplement to a traditional lab testing.

Note:
OrCAD is a registered trademark of Cadence Design Systems, Inc.
PSpice is a registered trademark of Cadence Design Systems, Inc.

2. SIMULATION CIRCUIT SETUP

2.1 Model File

A model defines the electrical behavior of a part. On a schematic page, this correspondence is defined by a part's implementation property, which is assigned the model name.

Depending on the device type that it describes, a model is defined as one of the following:

- a model parameter set
- a subcircuit netlist

Both ways of defining a model are text-based, with specific rules of syntax.

2.2 Model Libraries

Device model and subcircuit definitions are organized into model libraries. Model libraries are text files that contain one or more model definitions. Typically, model library names have a .LIB extension.

Most model libraries contain models of similar type. For vendor-supplied models, libraries are also partitioned by manufacturer.

Two files are used, the LIB file and the OLB file. The LIB file holds the characteristics of the model while the OLB file contains the model symbols.

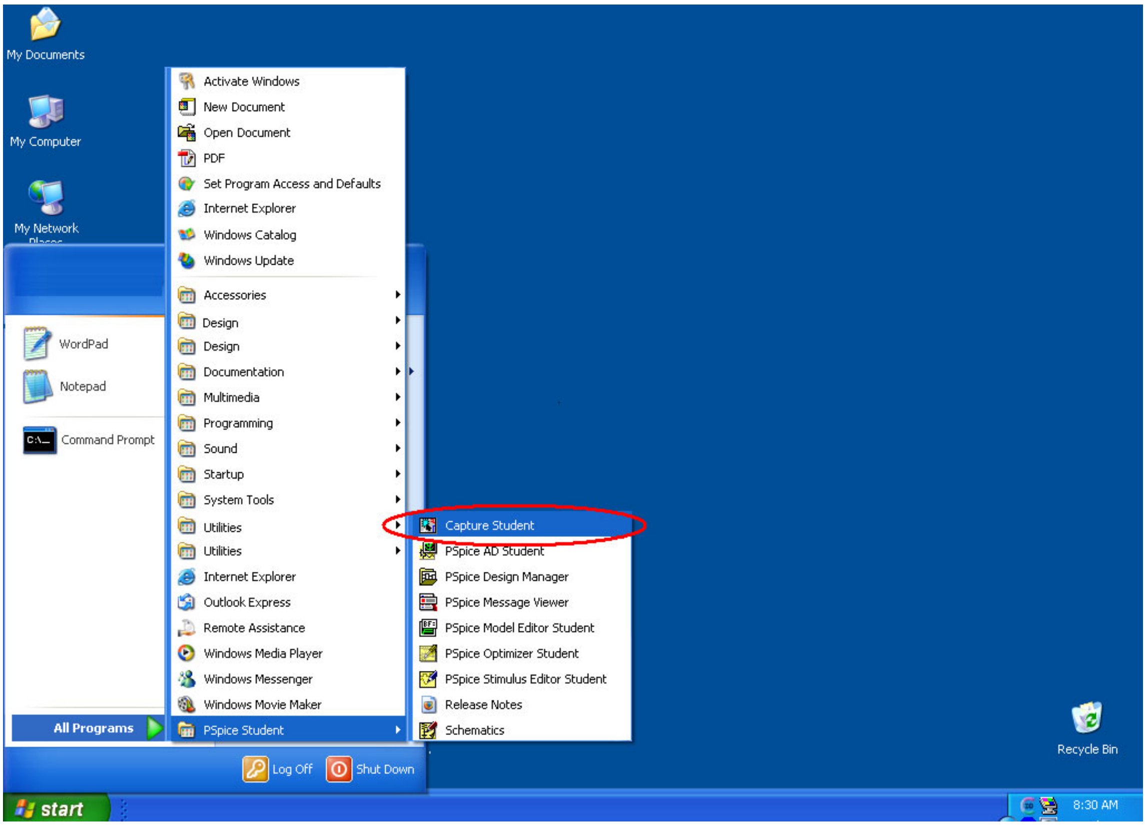
Although the LIB file alone is enough, circuits cannot be generated using OrCAD Capture. Instead, the netlist file must be used in PSpice A/D.

In contrast, having an OLB file of the model enables the user to make circuits in the OrCAD Capture, resulting in greater convenience and better circuit conceptualization.

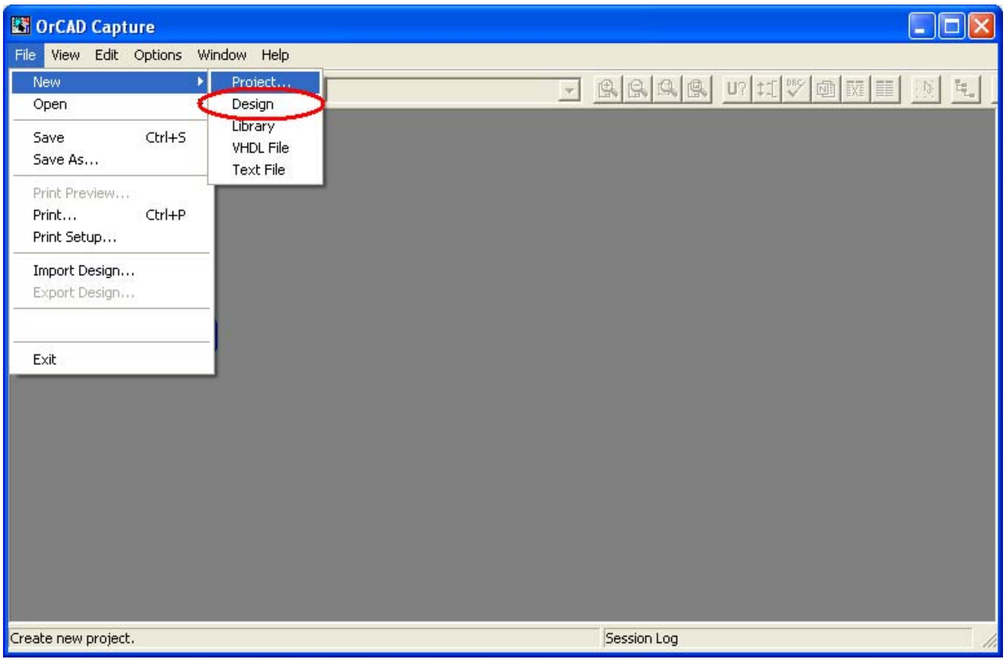
The sample LIB and OLB model files used in the setup, simulation, and evaluation examples with OrCAD PSpice are BU4229.LIB and BU4229.OLB. These are PSpice models of a voltage detector IC manufactured by ROHM.

2.3 Simulation Setup

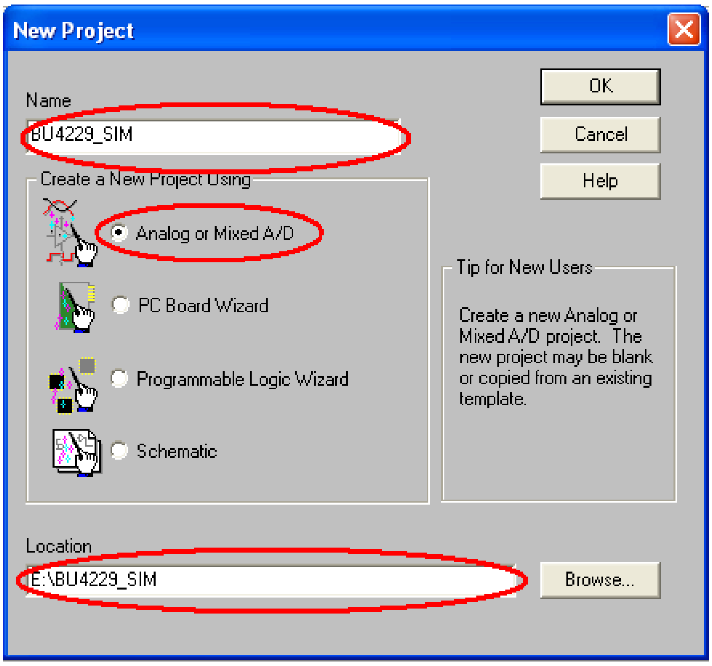
To start simulation setup, select the OrCAD Capture application.



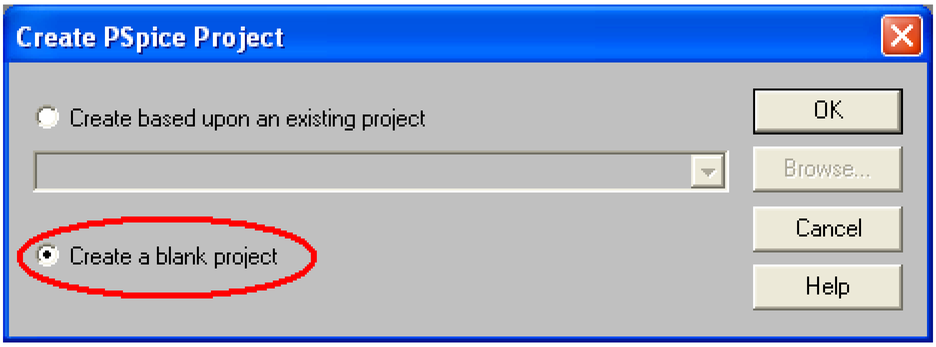
2.3.1 Starting the Design Project



Create a new project and enter the necessary parameters such as project name, project type, and project directory. In this example, the project name is BU4229_SIM_test and the type of project is Analog or Mixed A/D.

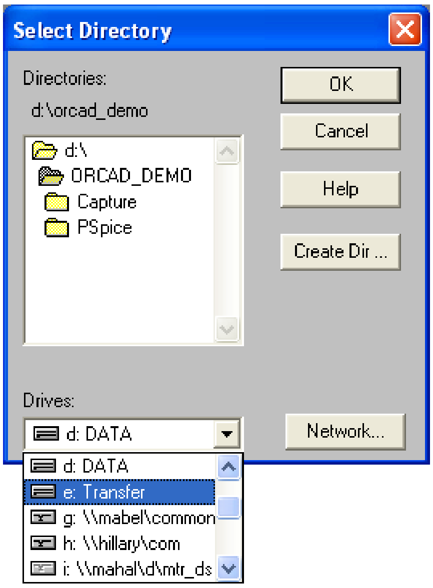


Next, create a blank project

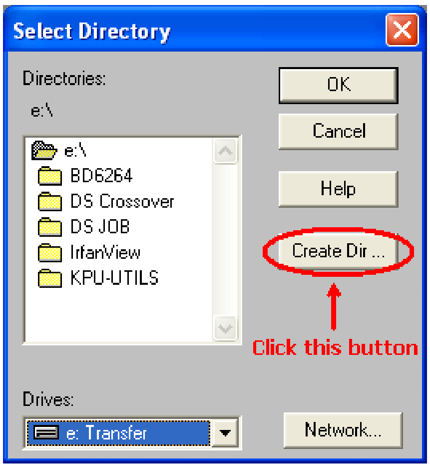


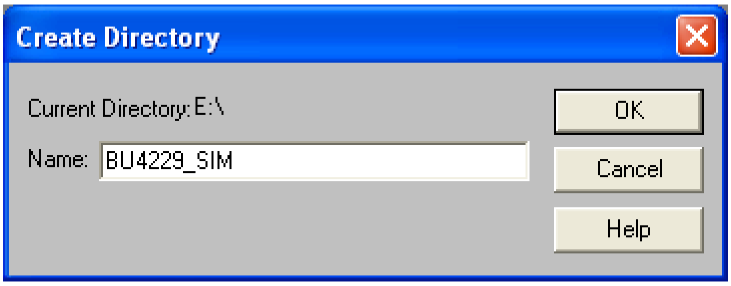
2.3.2 Project Directory

Select the target directory for saving the project. In this example we will be saving our project on drive E:

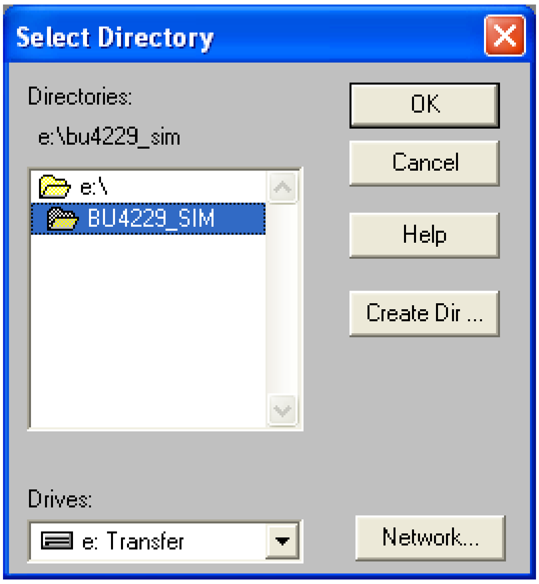


Create a project folder, such as BU4229_SIM.

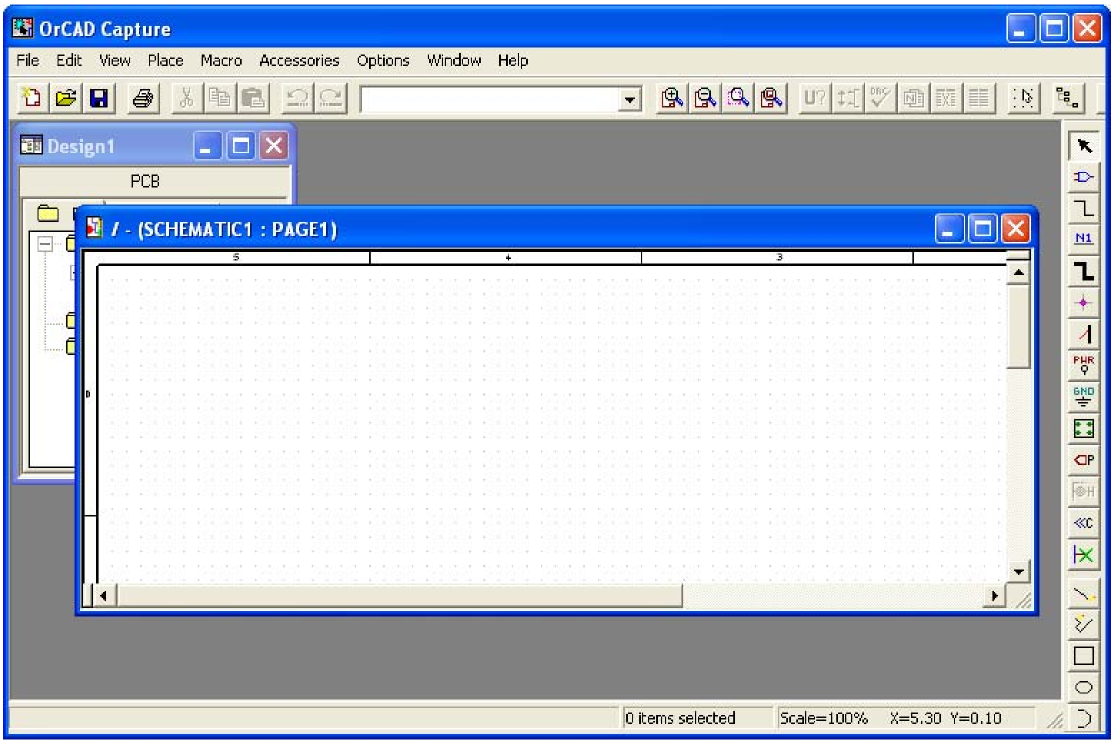




After creating the folder, select that folder then press OK

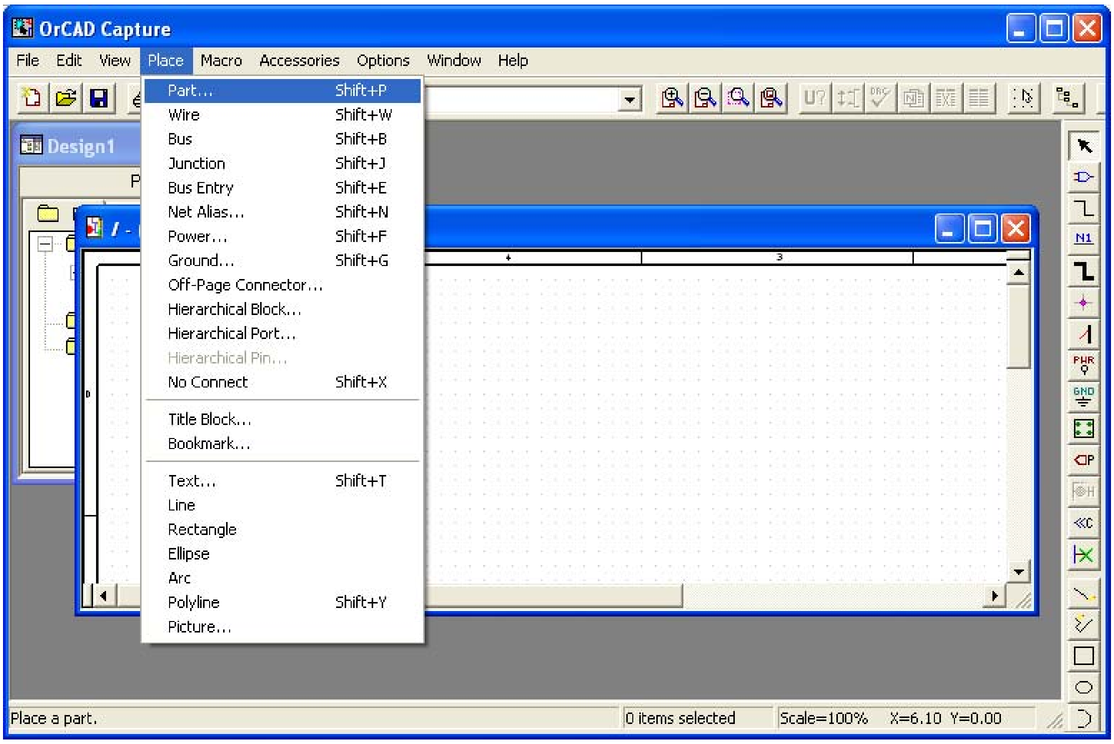


2.3.3 The OrCAD Capture Environment

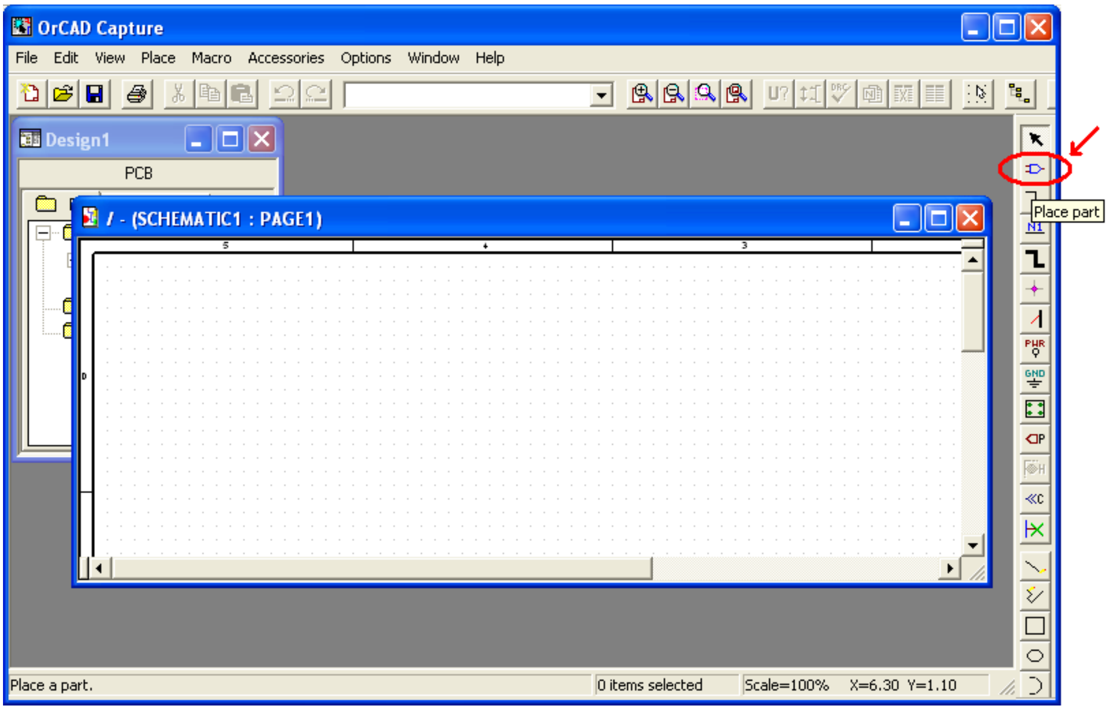


This is how OrCAD Capture will look like after successfully setting the project directory. The next step is to add parts to the circuit for simulation.

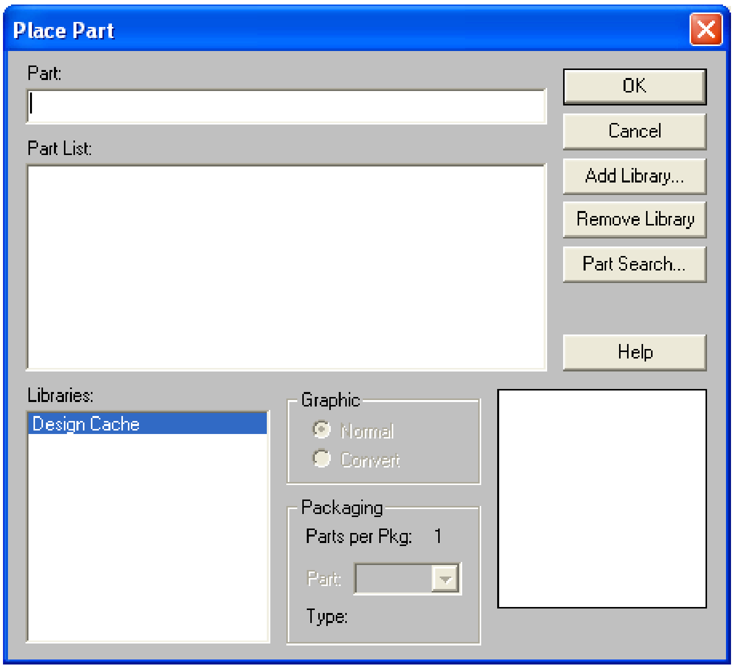
2.3.4 Adding Parts to the Schematic



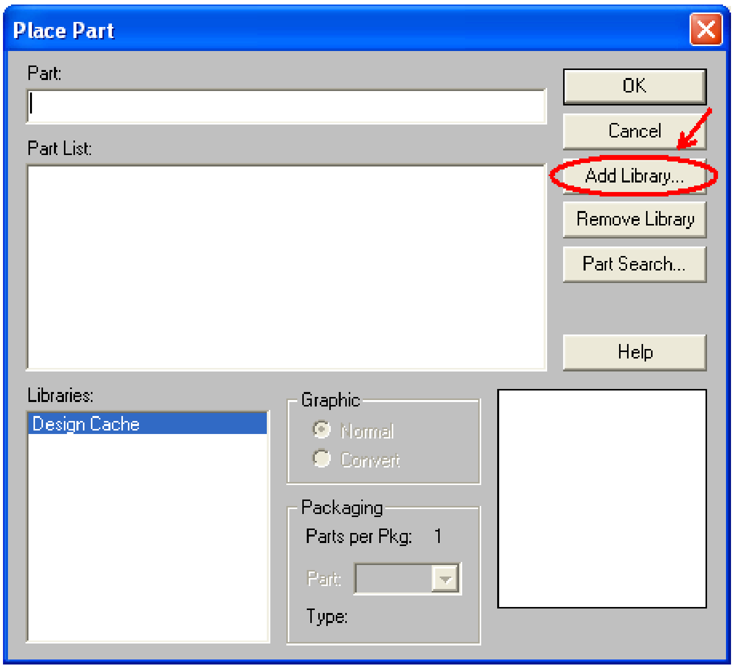
Parts may be added as shown or by simply pressing Shift+P. Parts can also be added by clicking on the Place Part button on the right tool bar.



This is the Place Part window. Initially no libraries are included. Libraries need to be added for setup (i.e. the PSpice models for the BU4229).

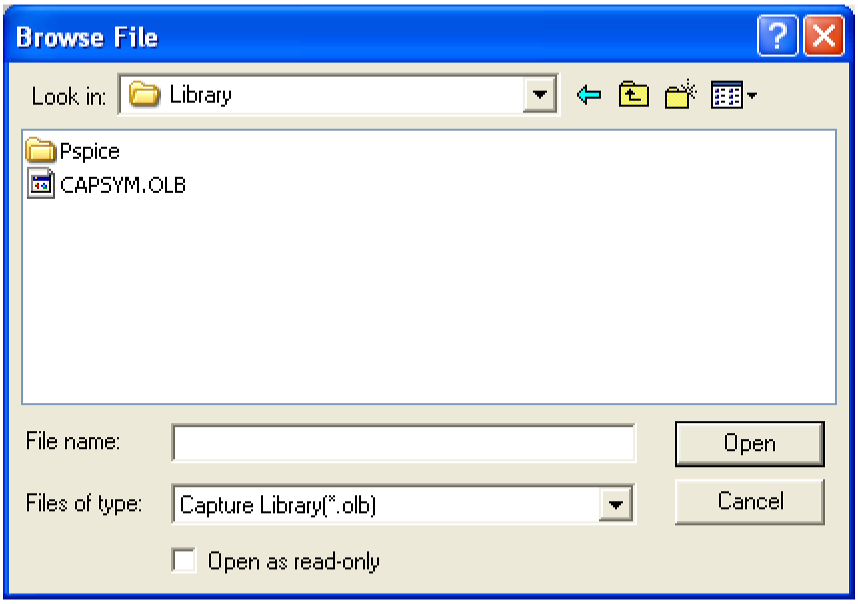


2.3.5 Adding Libraries

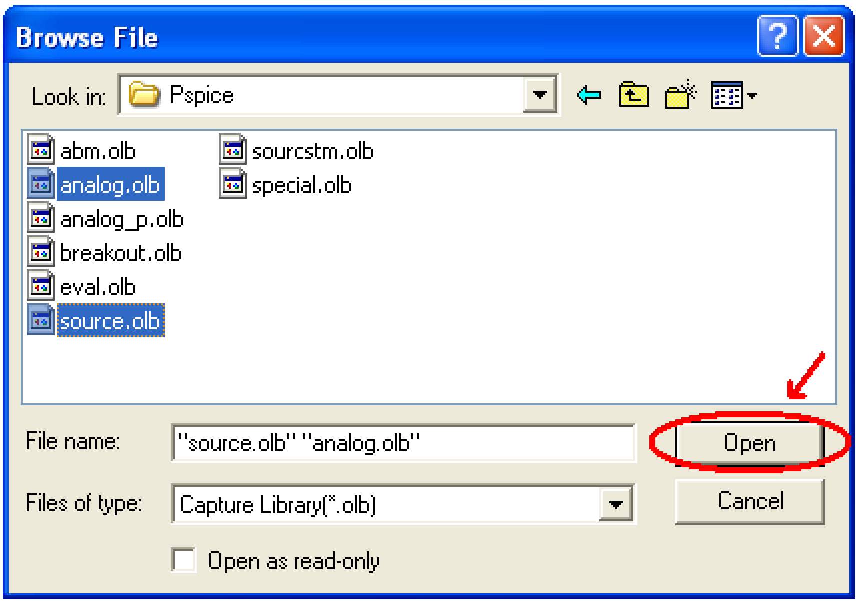


To add Libraries, click on the Add Library button as shown above.

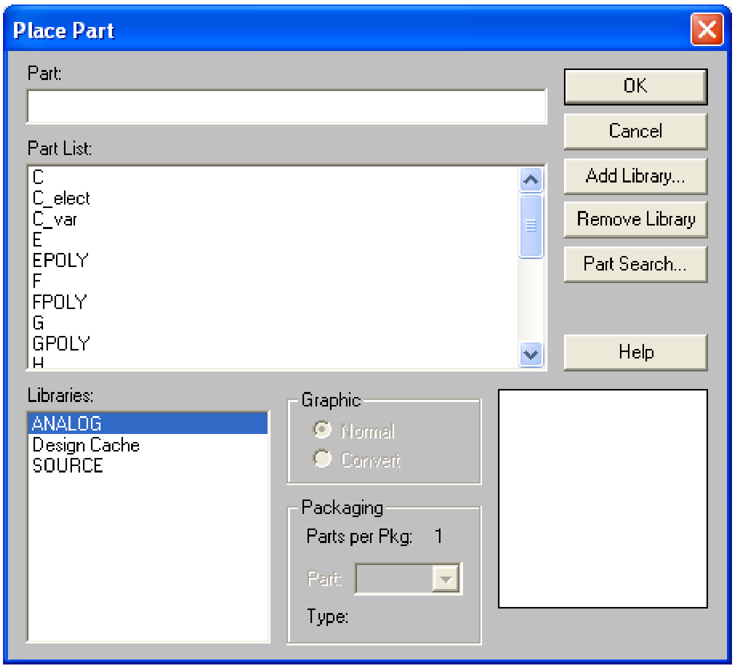
Select the OLB files required for simulation. Normally, sources.OLB and analog.OLB are necessary, which are a shared OLB file of OrCAD containing voltage/current sources and analog devices such as resistors and capacitors. Go to the Capture/Library/PSpice folder (default) in the OrCAD PSpice directory or to the relevant folder.



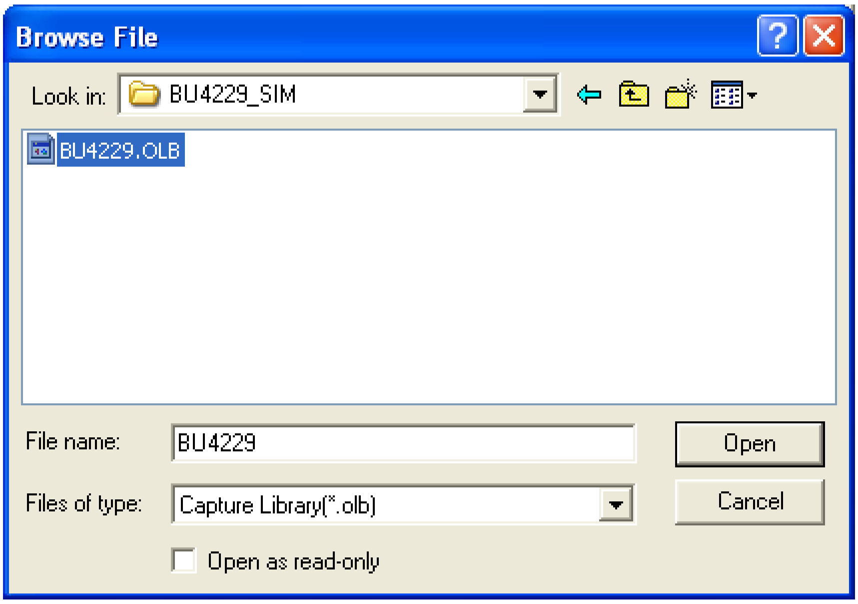
From the Capture/Library/PSpice folder, select analog.OLB and source.OLB then press the Open button. (Press Ctrl and click on the files to select them both)



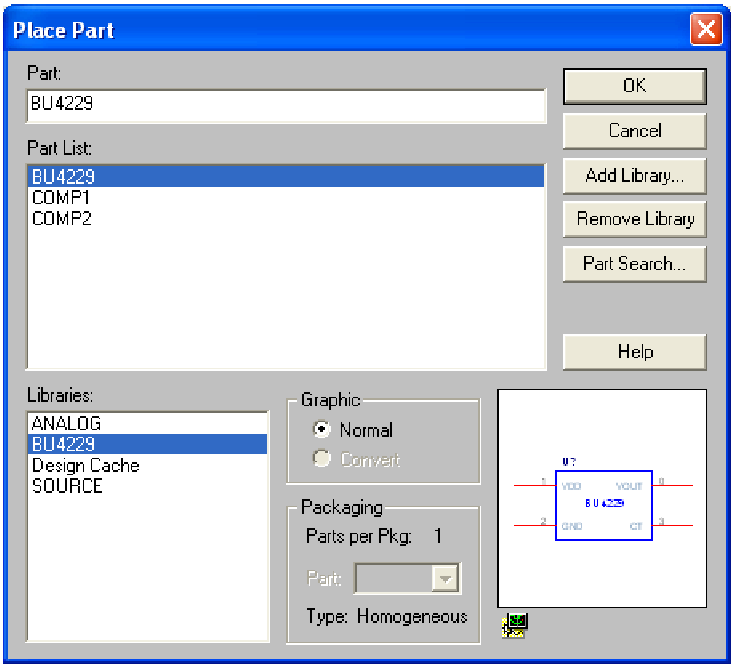
This is what the Place Part window will look like after adding the libraries. You may see a list of parts from each library.



Next, add the model library of the PSpice model for simulation. It is easier if the model files are located in the Project Folder. (E.g. BU4229.LIB and BU4229.OLB in E:¥BU4229_SIM) Then add the BU4229.OLB library.



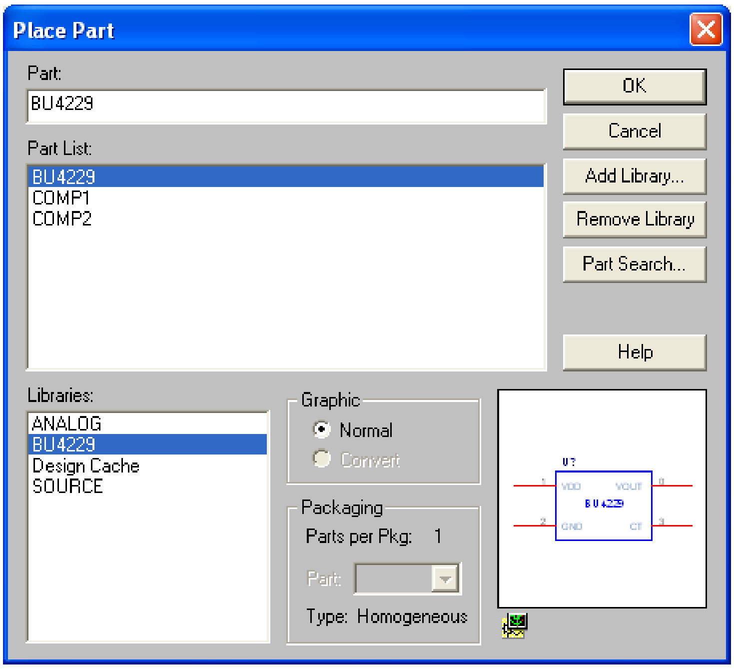
The parts list of the BU4229 library should now be displayed, along with (possibly) other associated parts.

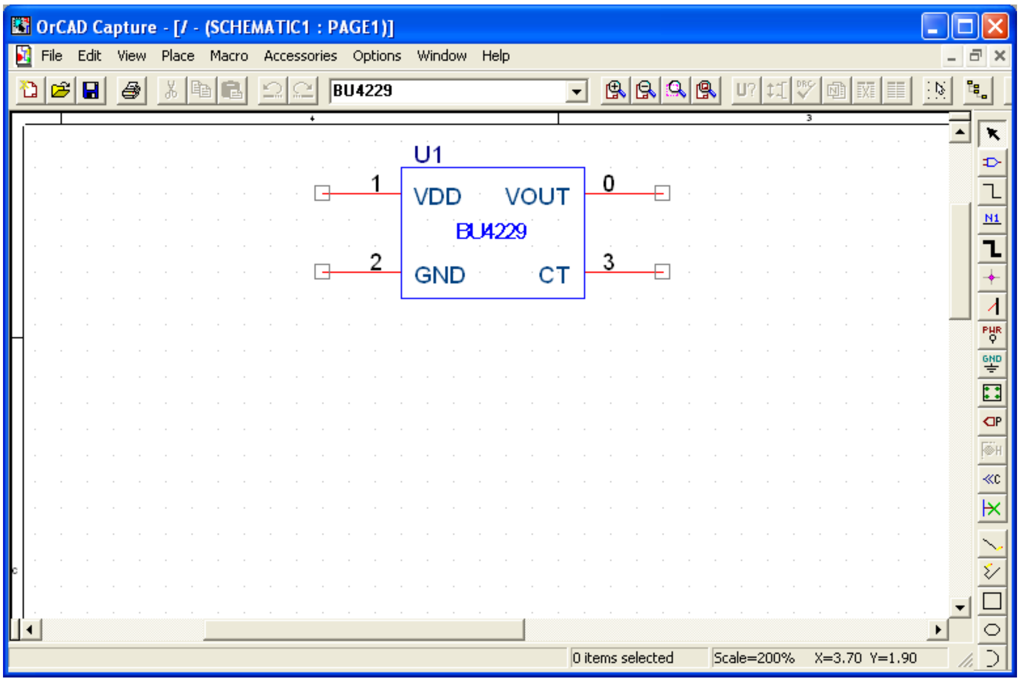


Simulation setup can now begin.

2.3.6 Simulation Setup

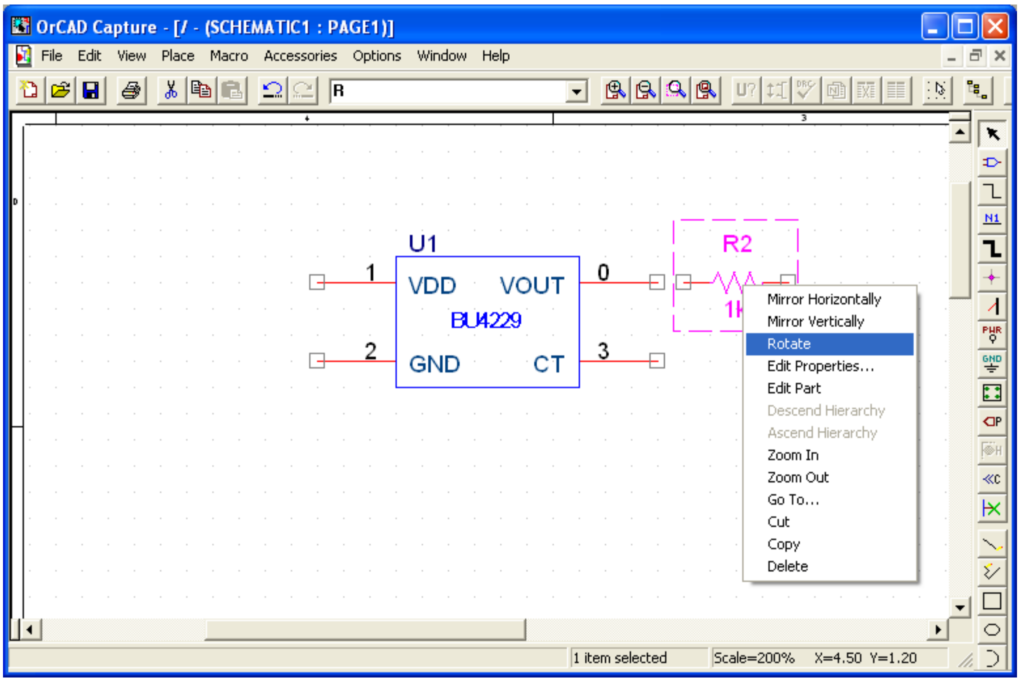
Add parts in the OrCAD Capture Environment.



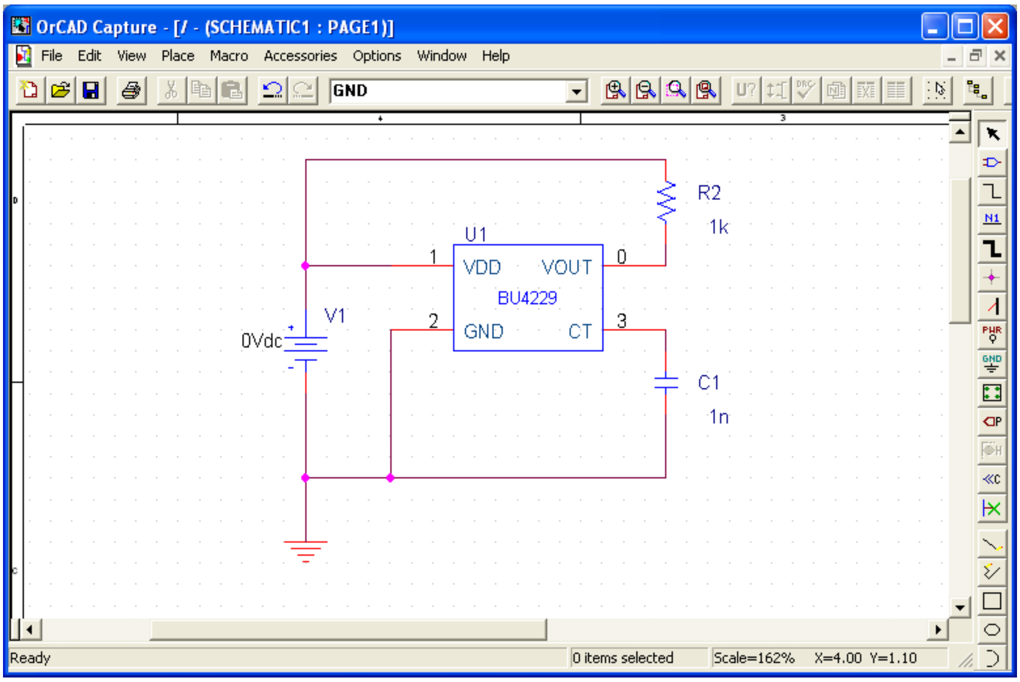


Next, add voltage sources and other components like resistors and capacitors. DC Voltage (VDC) and DC Current (IDC) sources can be found in the Sources library while passive devices like resistors (R) and capacitors (C) are located in the Analog library.

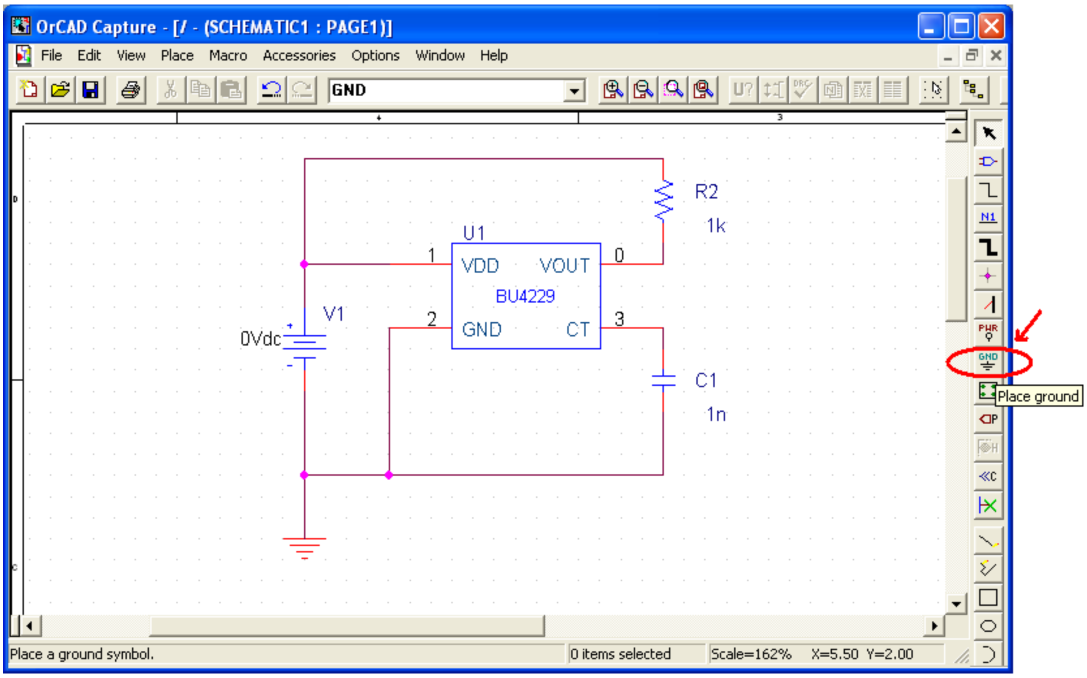
Place the parts. To rotate the device, right-click on the device and select Rotate.



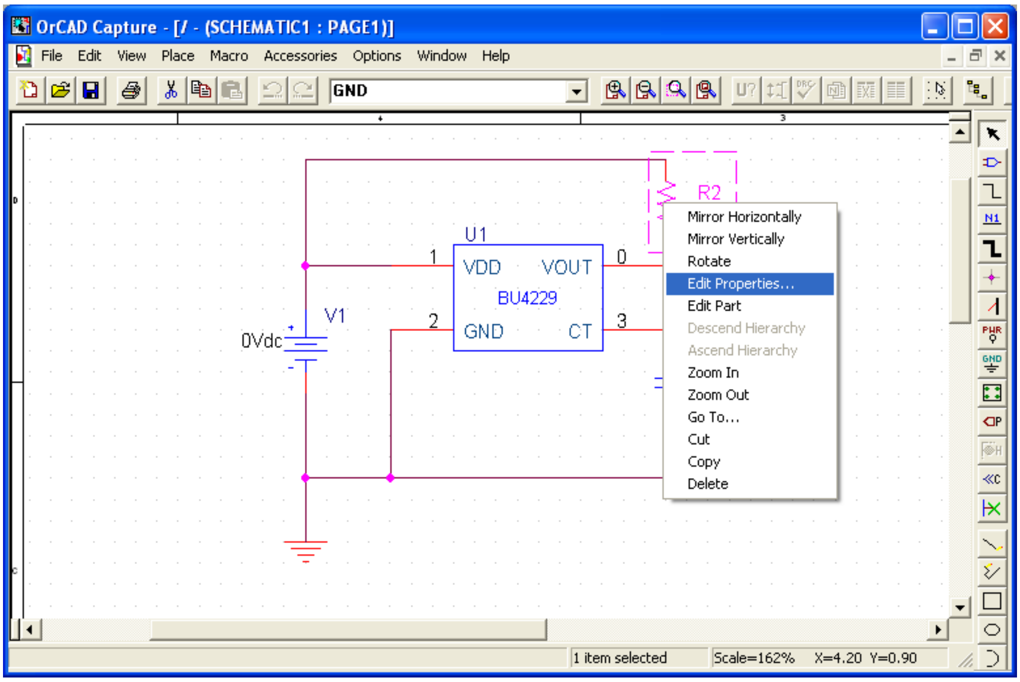
Complete the necessary wiring as shown below. In this case the simulation will be run to determine the detection voltage. (Detection Voltage Parameter)



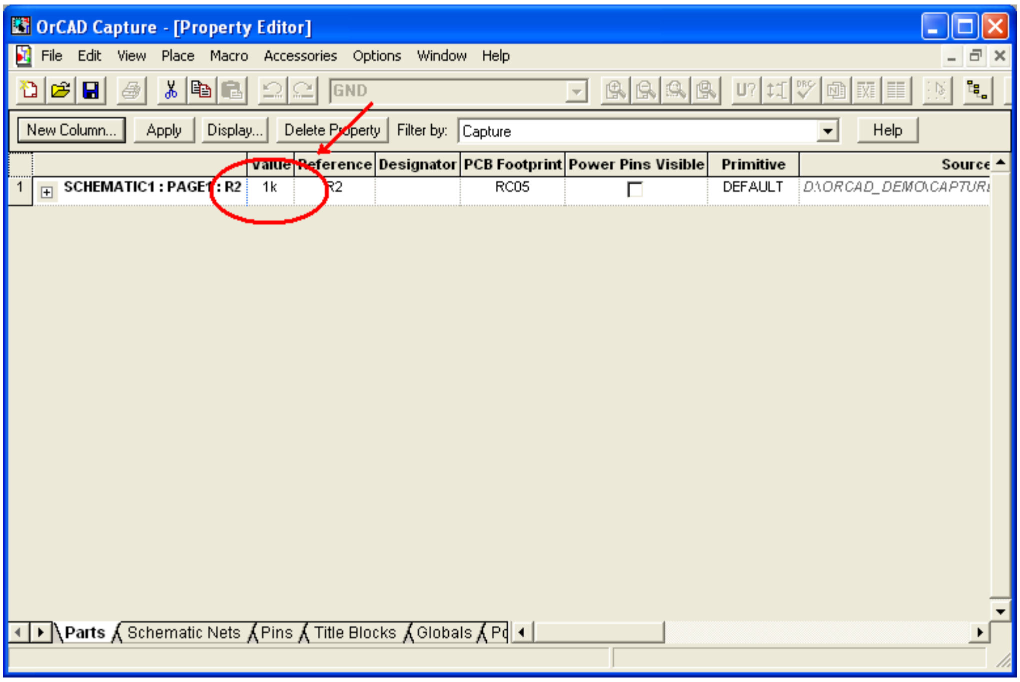
Place a ground by clicking on the GND button in the right tool box.



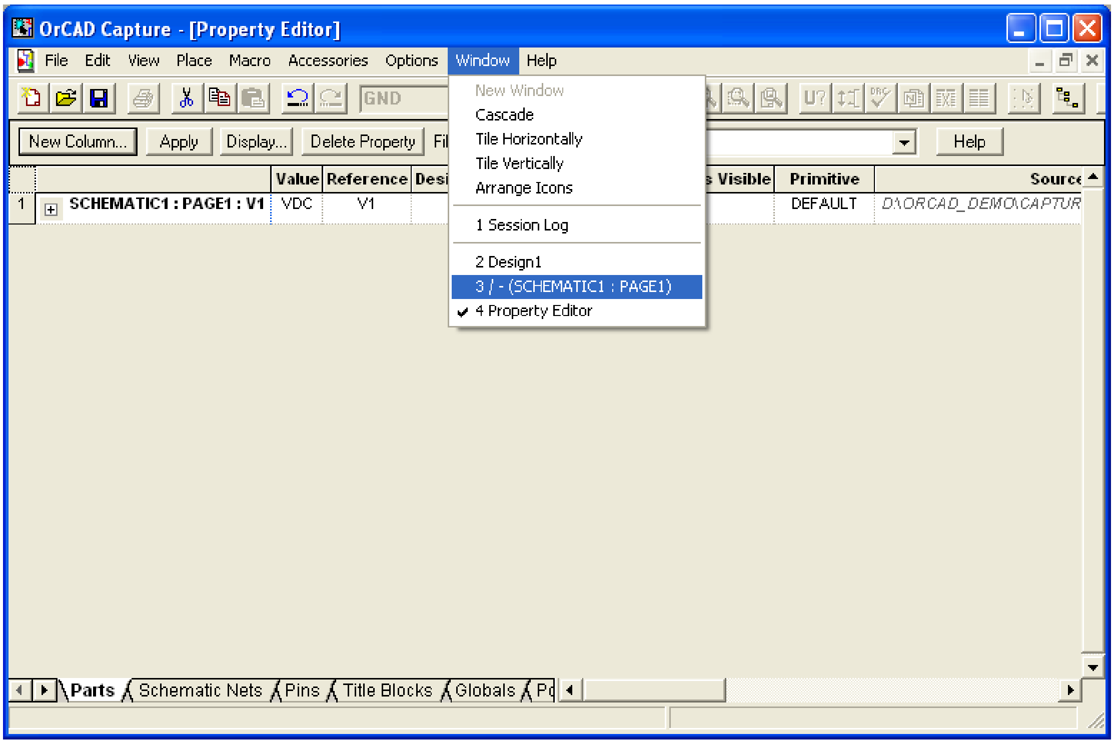
To set the values of the external components (i.e. resistor, capacitor, DC voltage source), click on the device then right-click and choose Edit Properties. For this example, R2 will be set to 470Kohms, C1 to 100nF, and V1 to 5V.



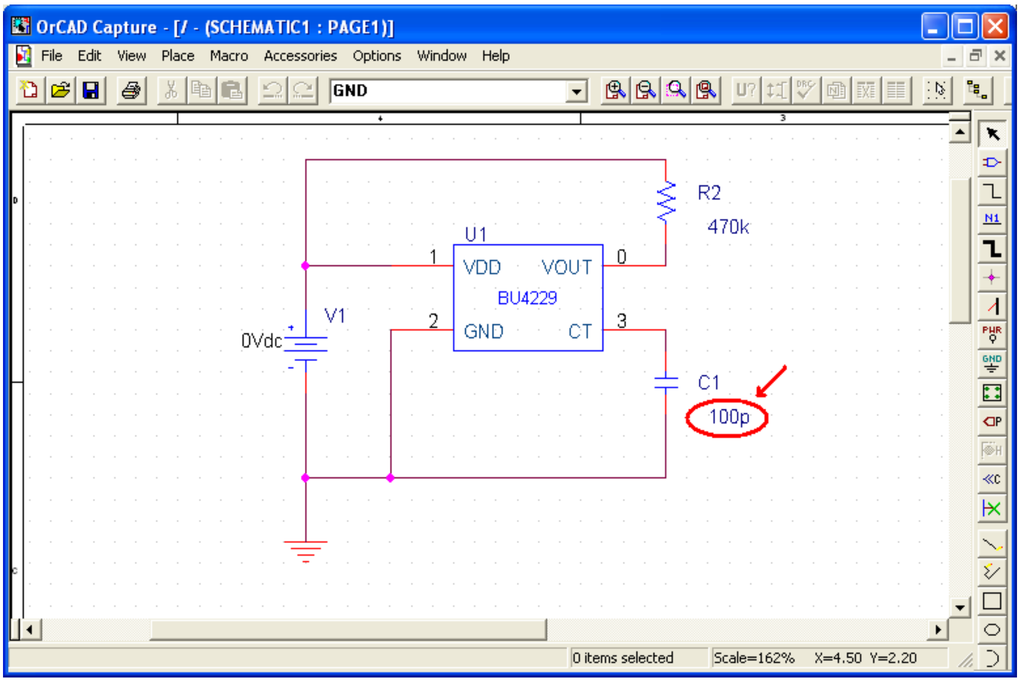
The Property Editor window will appear. Change the value of the part by typing the value with the proper prefix unit (i.e. k for kilo and p for pico).



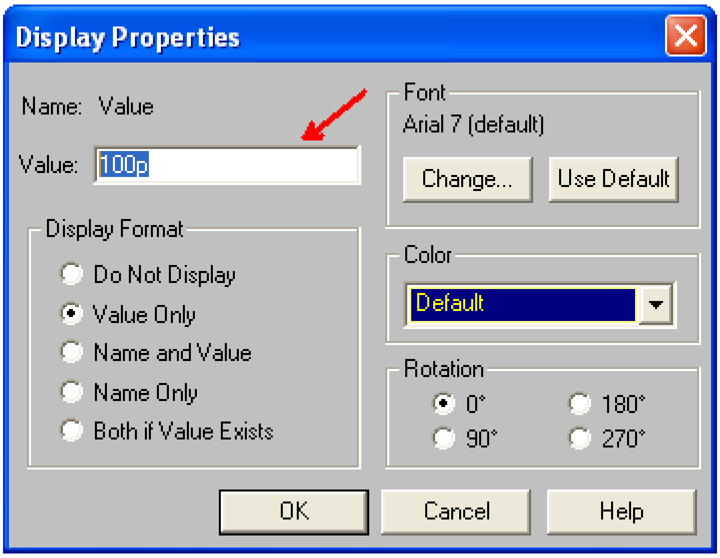
Return to the schematic page by clicking on Window tab in the main pull-down menu.



The values can also be changed by double-clicking on the value displayed.

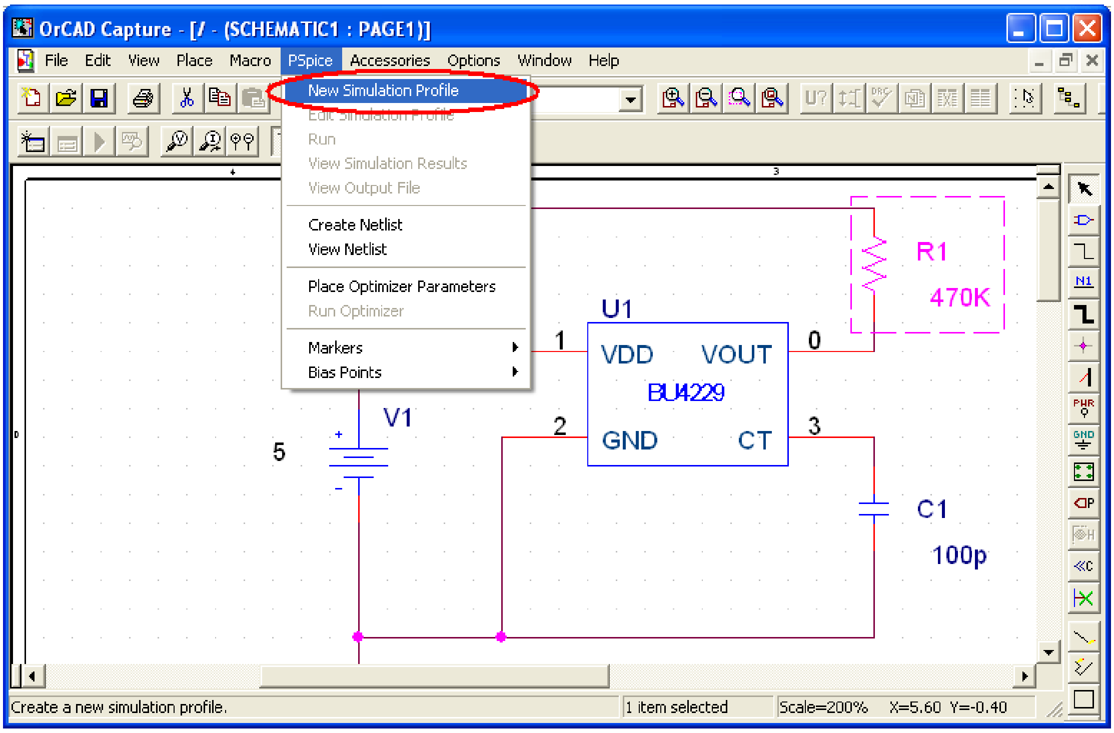


Enter the value in the popup box, then press OK.

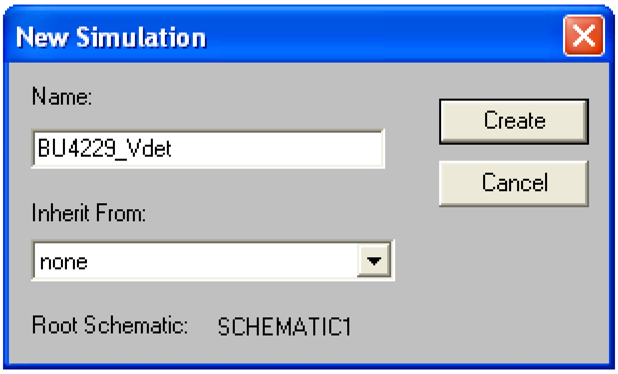


2.3.7 Creating a Simulation Profile

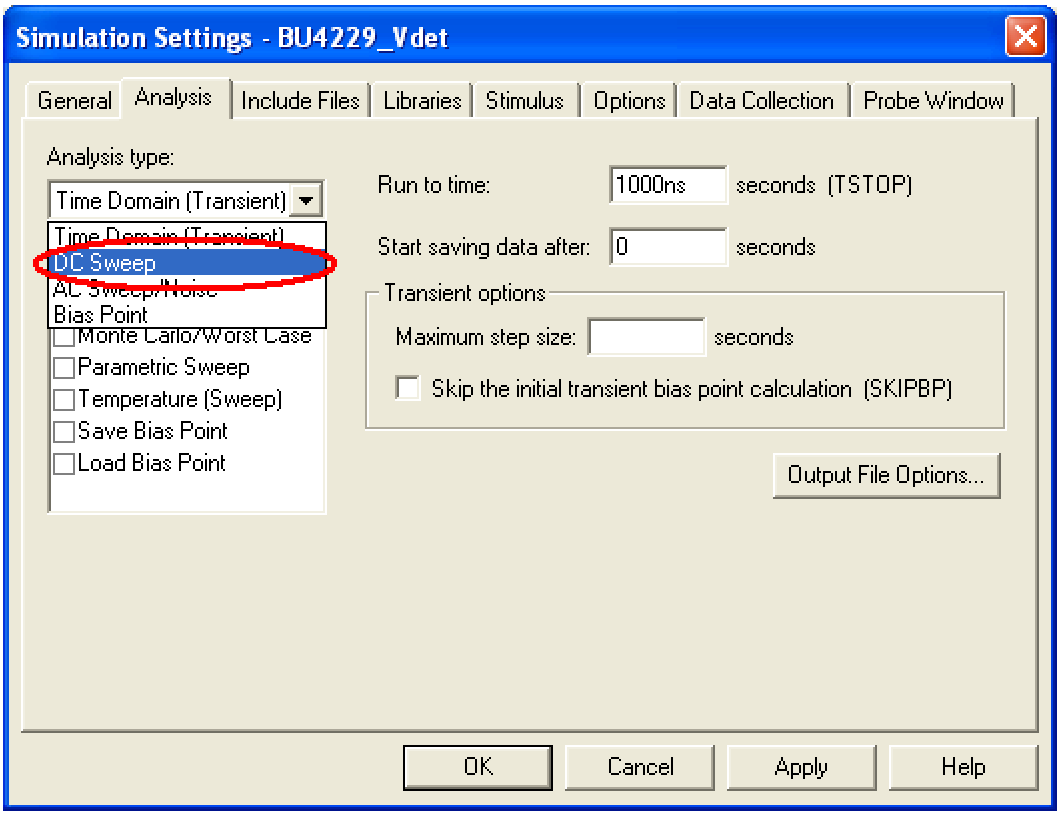
To create a simulation profile, click on the PSpice tab in the main menu and select New Simulation Profile.



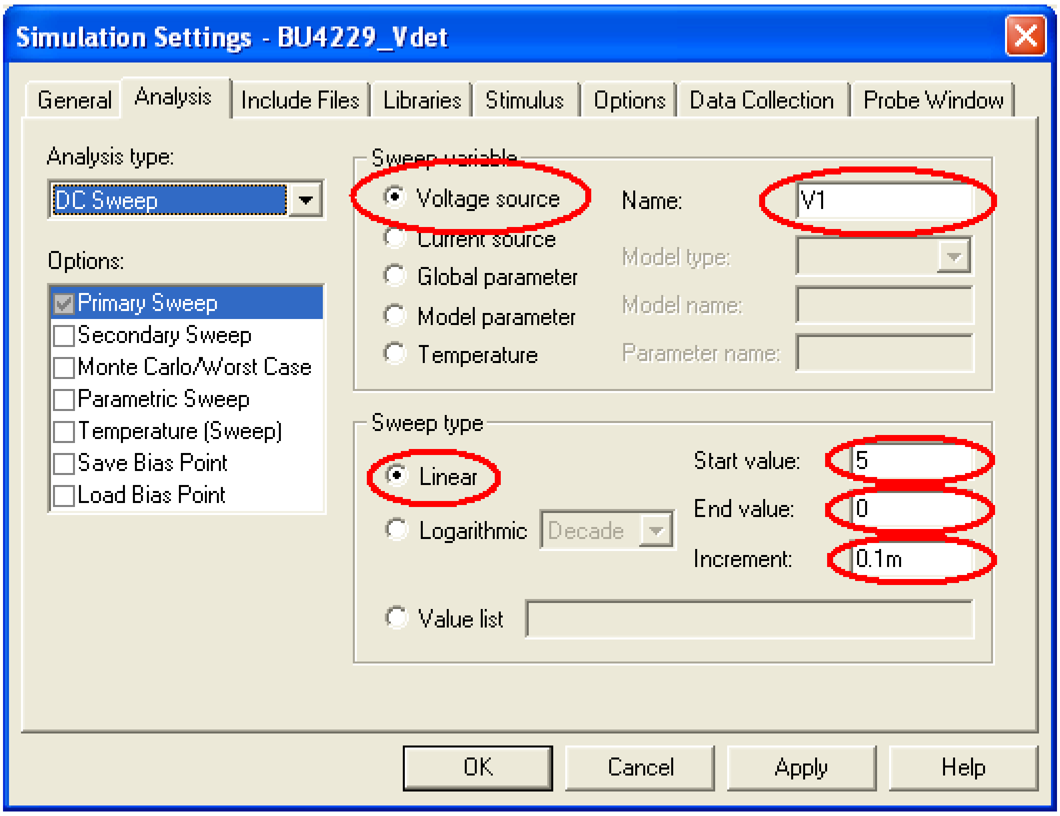
Enter a name for the simulation profile. In this example the name is BU4229_Vdet.

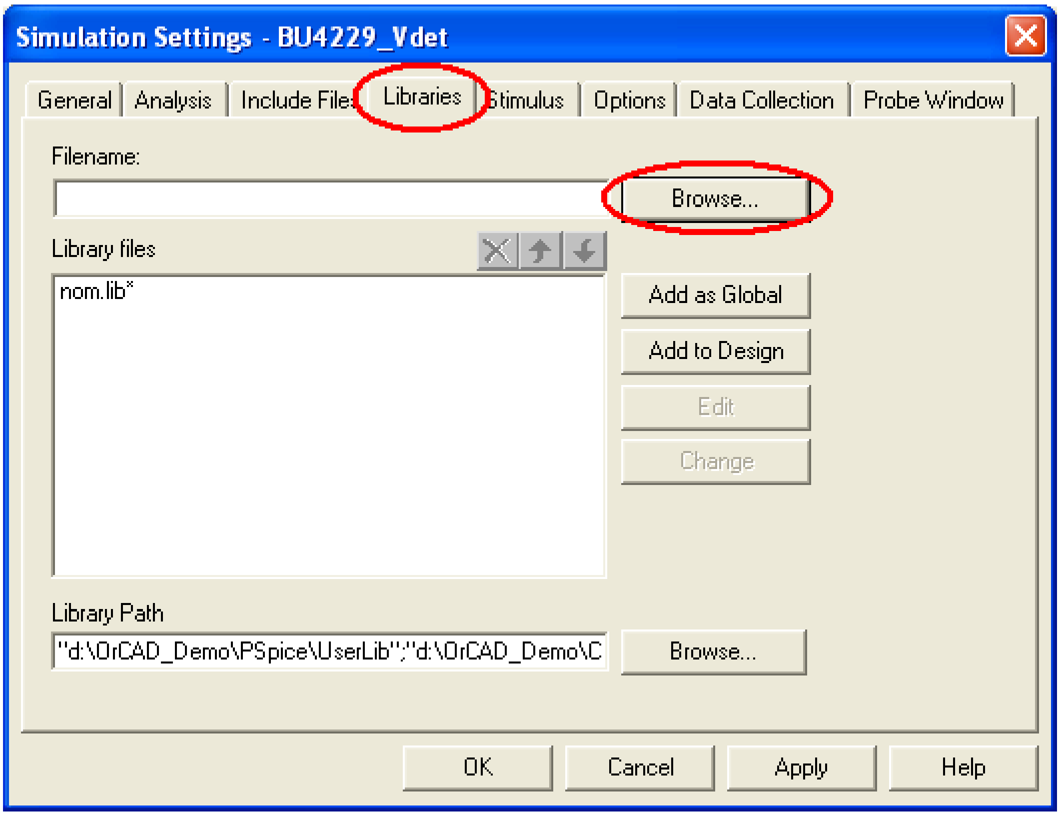


In the Simulation Settings window, select DC Sweep under Analysis type in order to evaluate Vdet.

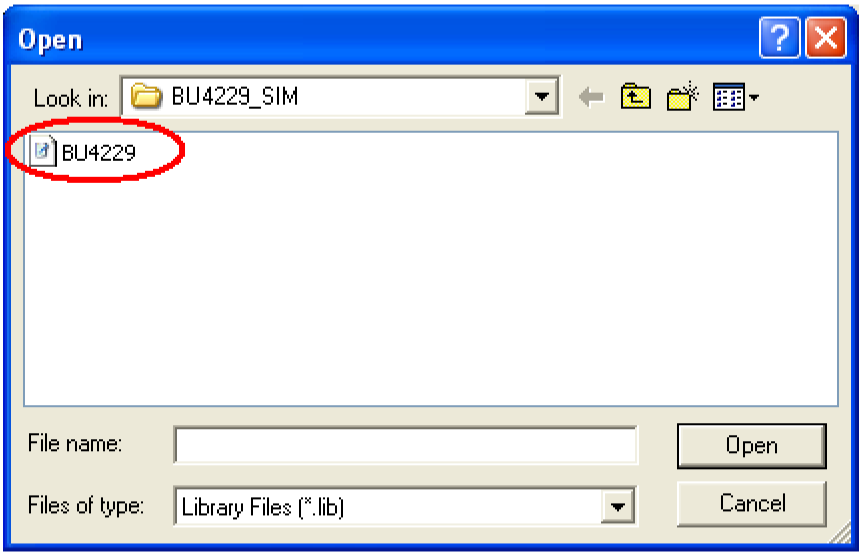


Enter the required settings. In this case the DC voltage source V1 will be swept linearly from 5V to 0V with a stepping voltage of 0.1mV.

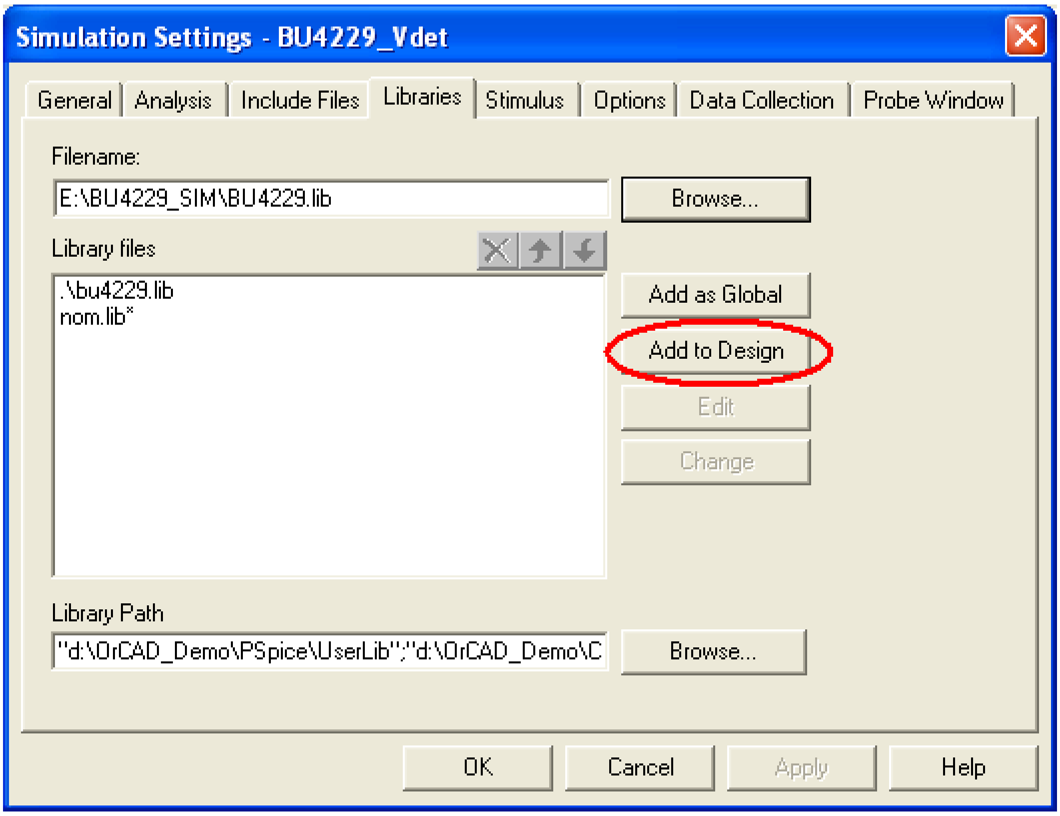




Locate the model file BU4229.LIB



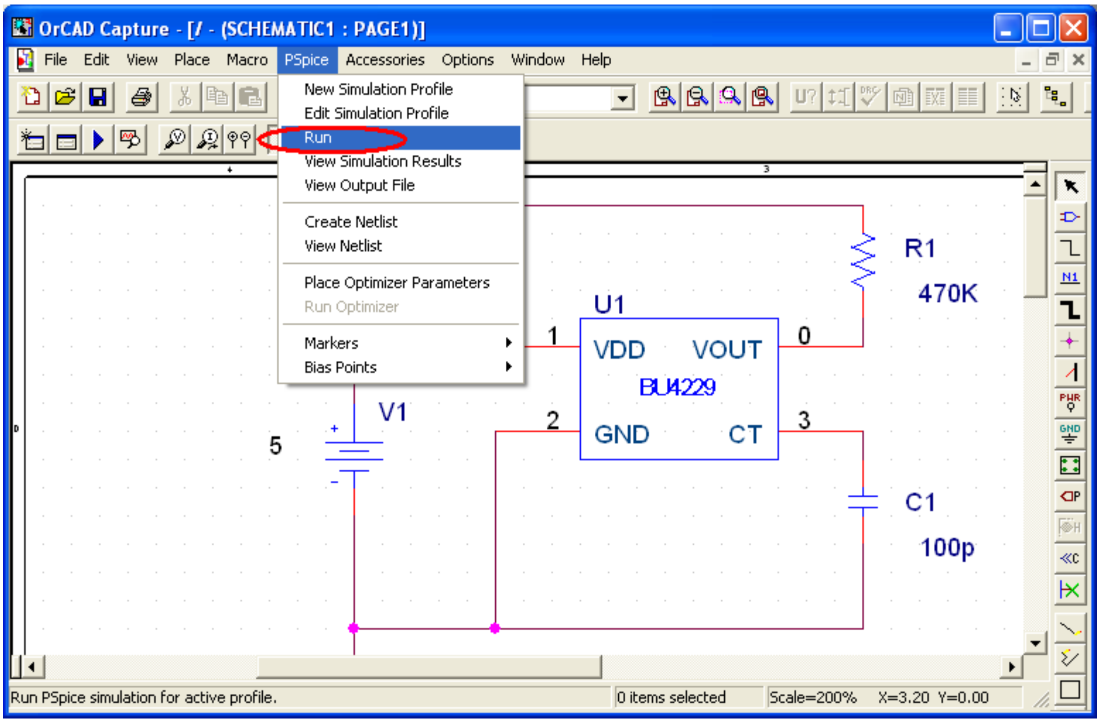
Add the library to the design then press OK.



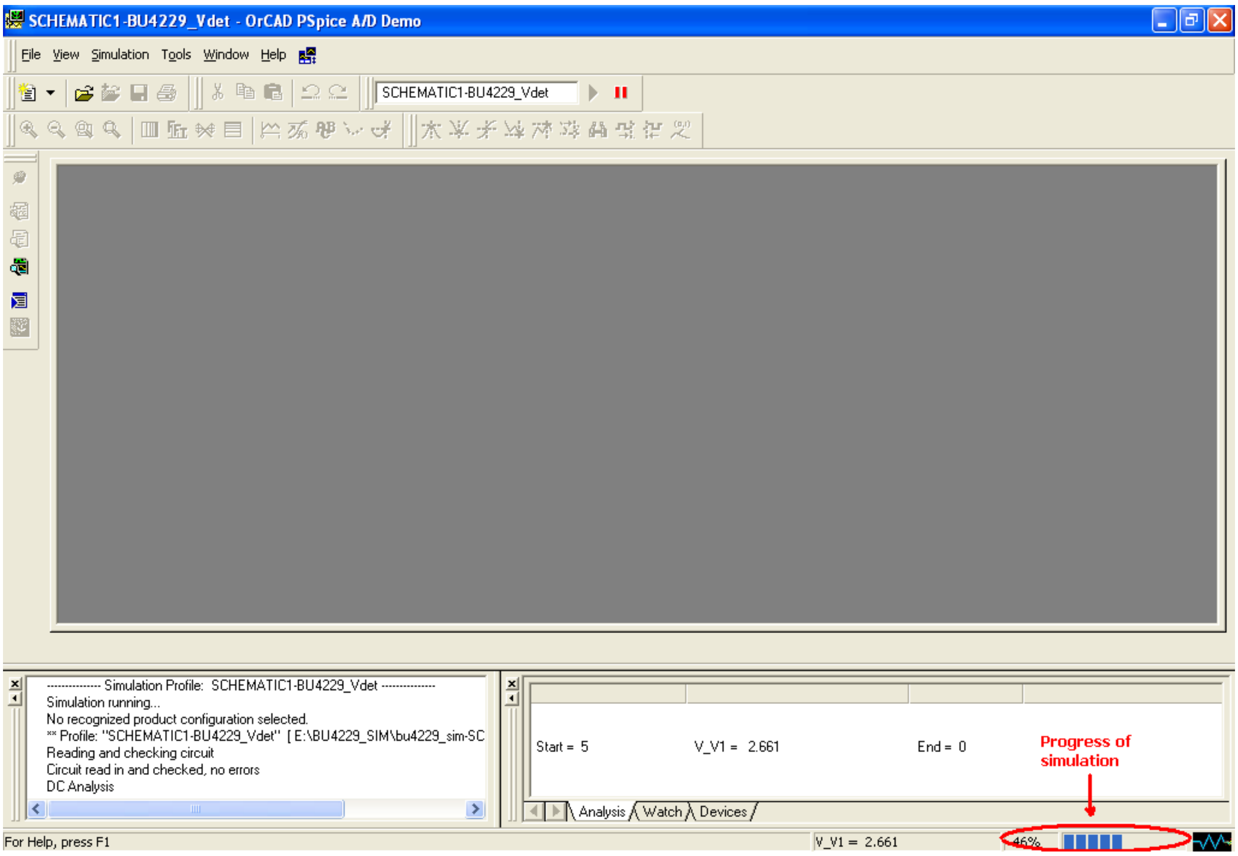
Simulation is now ready to begin.

2.3.8 Running the Simulation

To run the simulation, go to PSpice tab in the main menu and select Run.

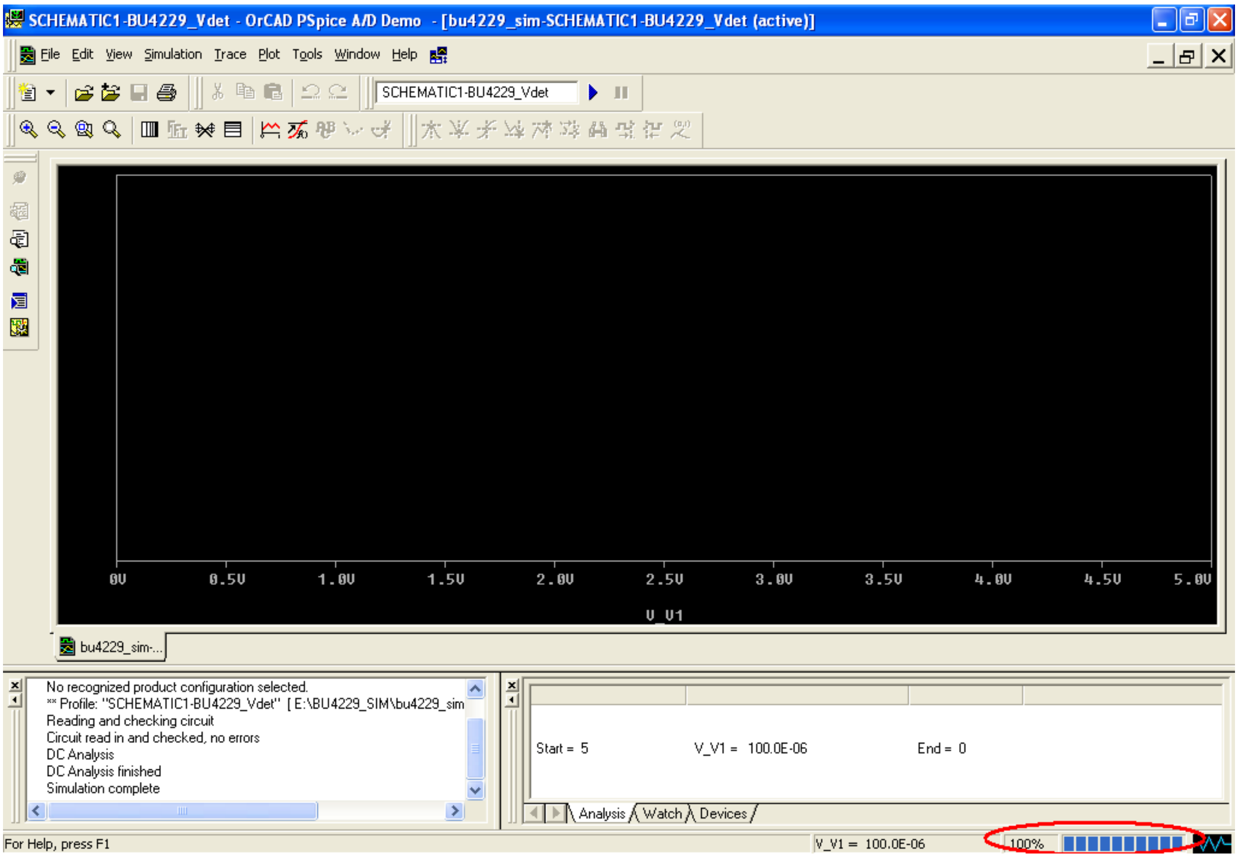


OrCAD PSpice A/D will automatically load and run the simulation.

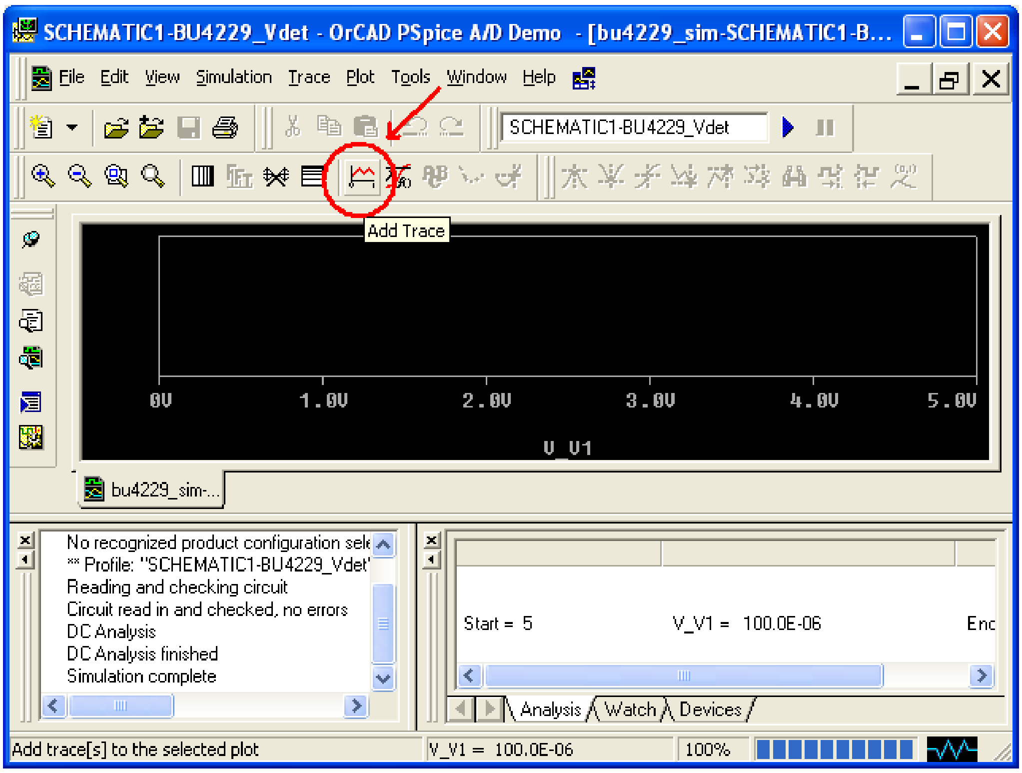


3. EVALUATION OF RESULTS

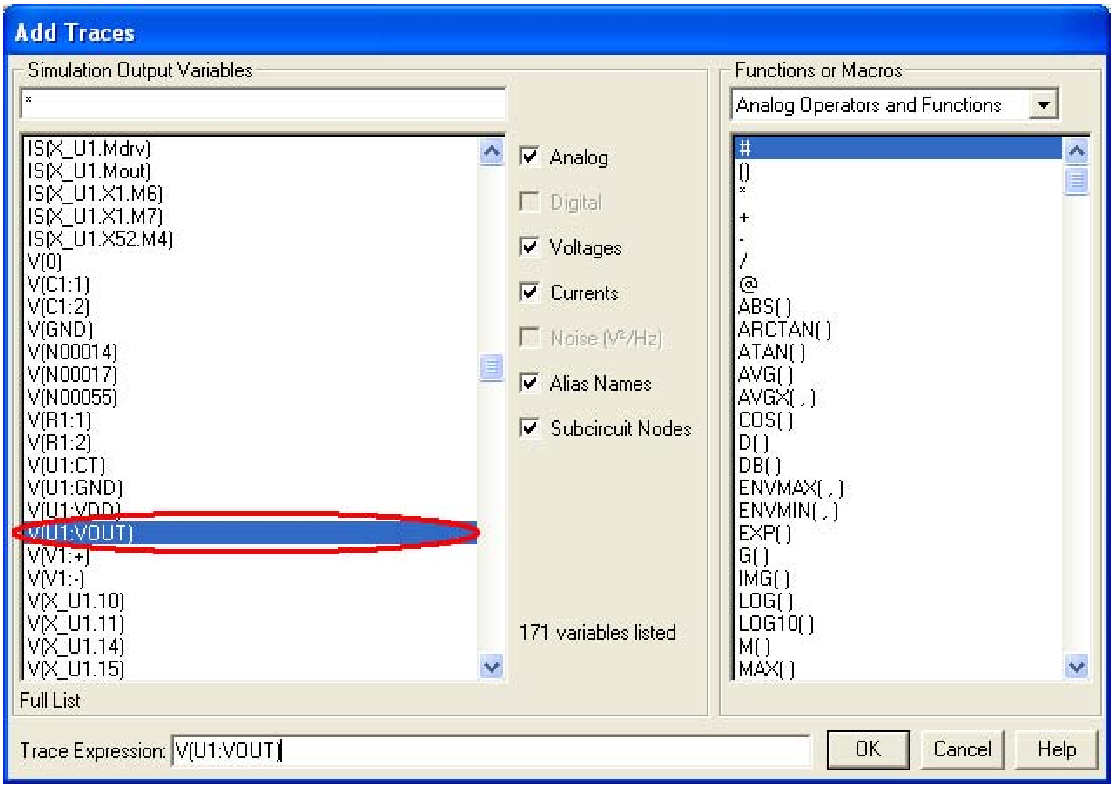
A completed simulation will look like this. Note the 100% shown on the progress bar.



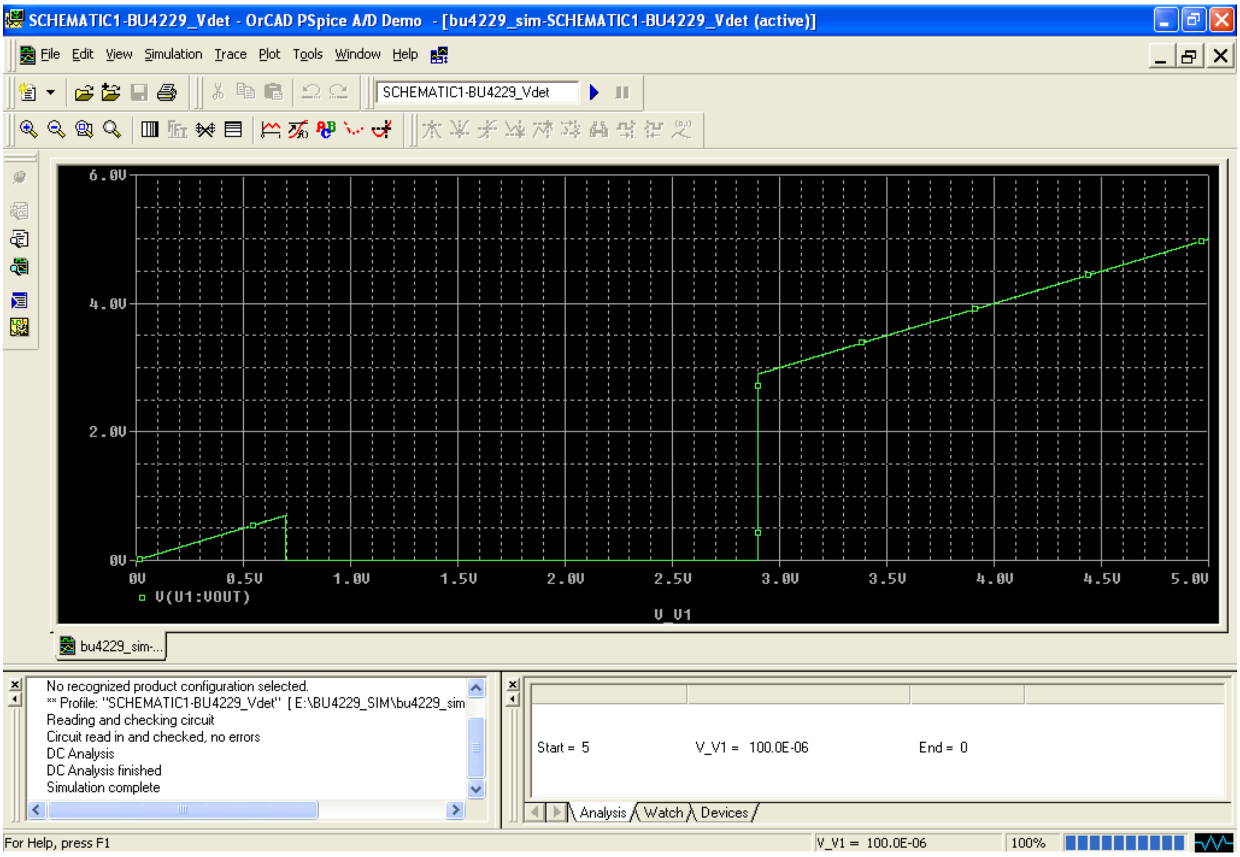
3.1 Adding a Trace or Output



Select the pin, node, or branch for monitoring/measurement. In this case the VOUT pin is selected [V(U1:VOUT)].

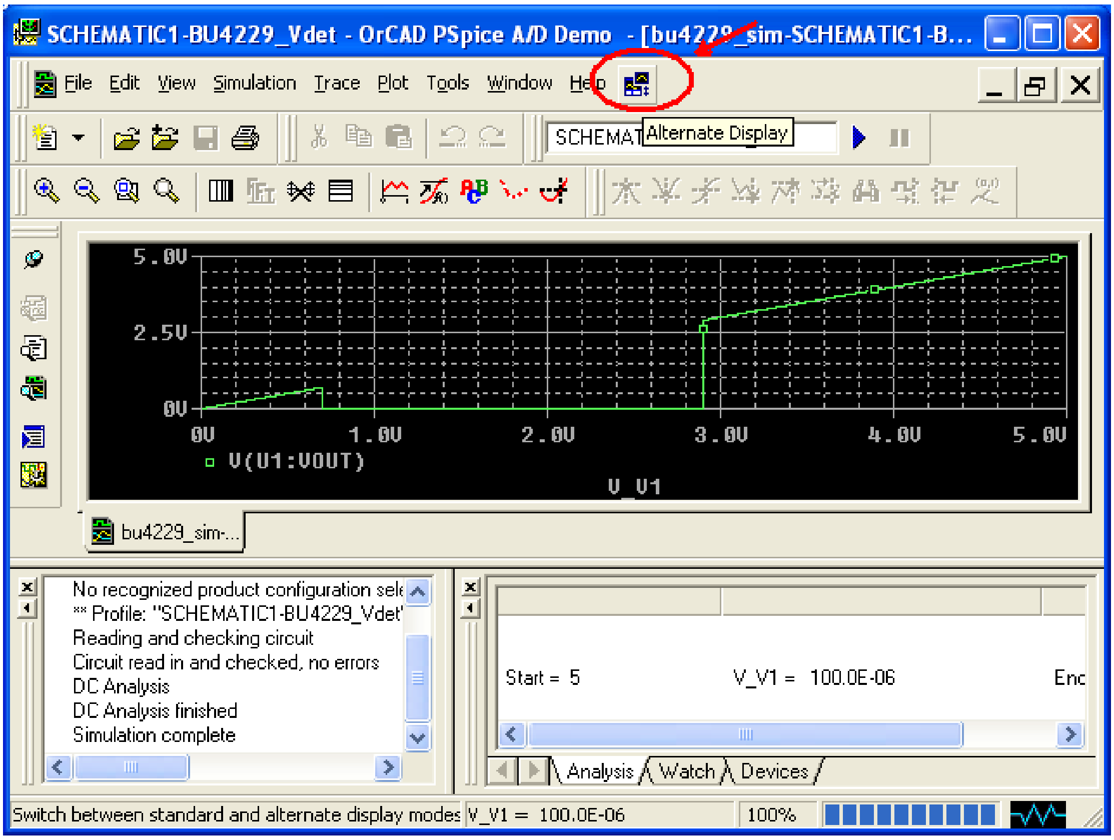


The OrCAD PSpice A/D should look like this.

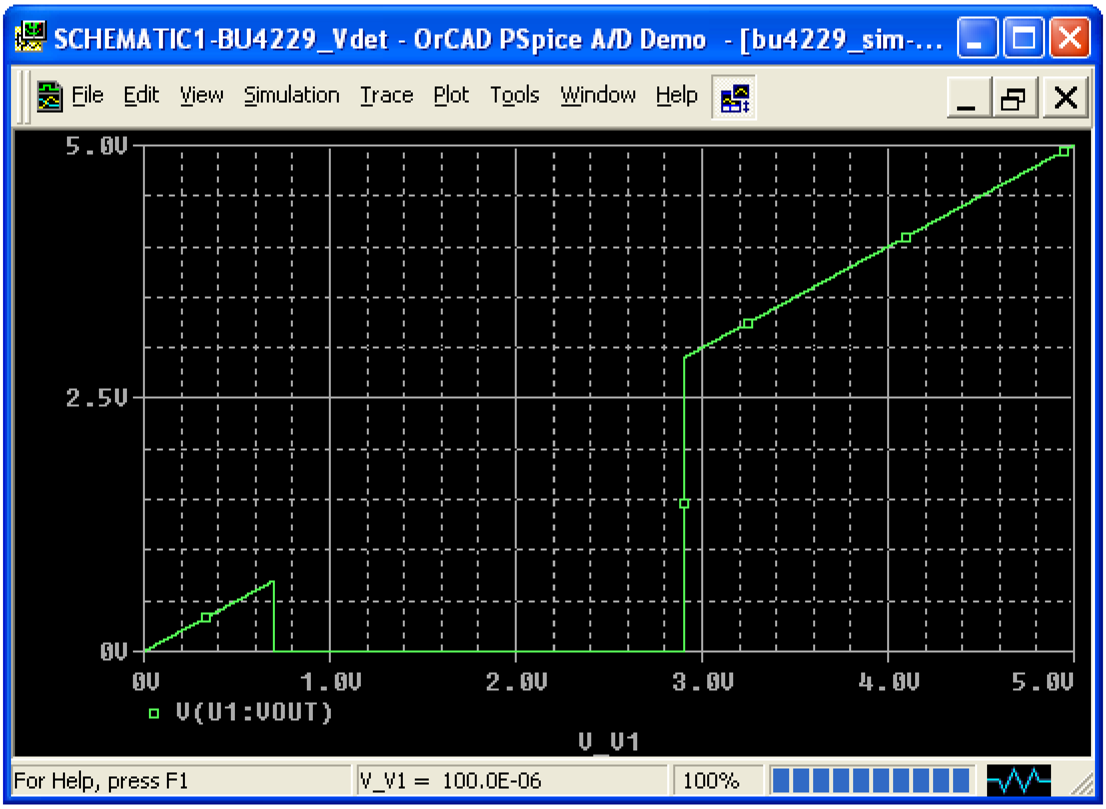


3.2 Alternate Display

Enlarge the graph to full screen by clicking on the Alternate Display button.

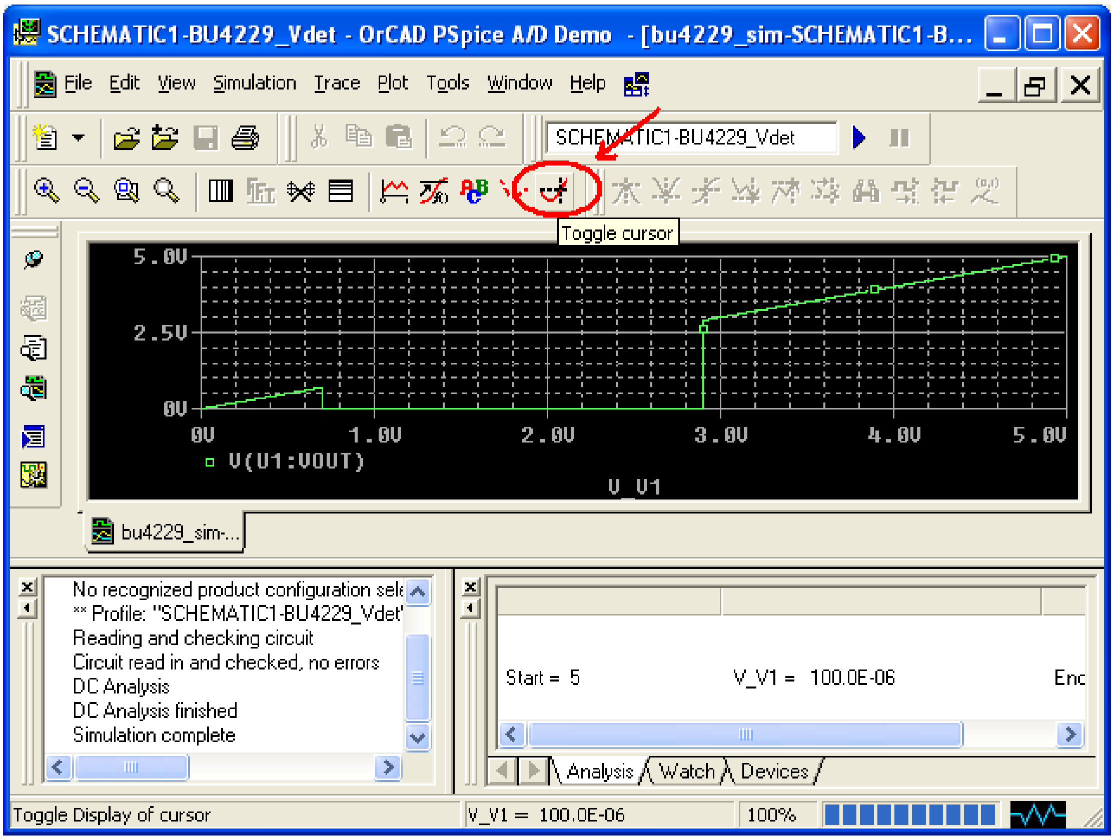


It should look like this.

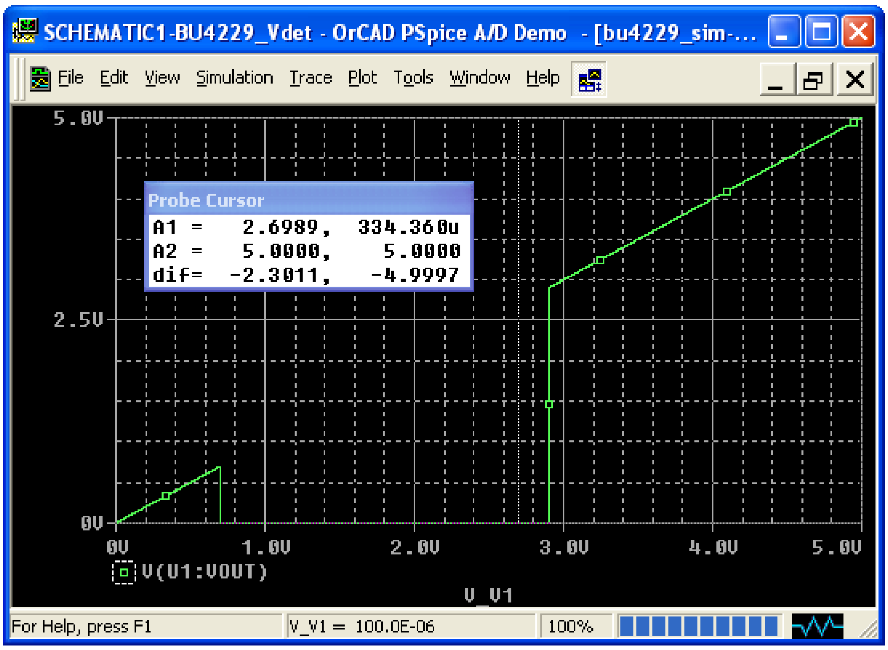


3.3 Measurement

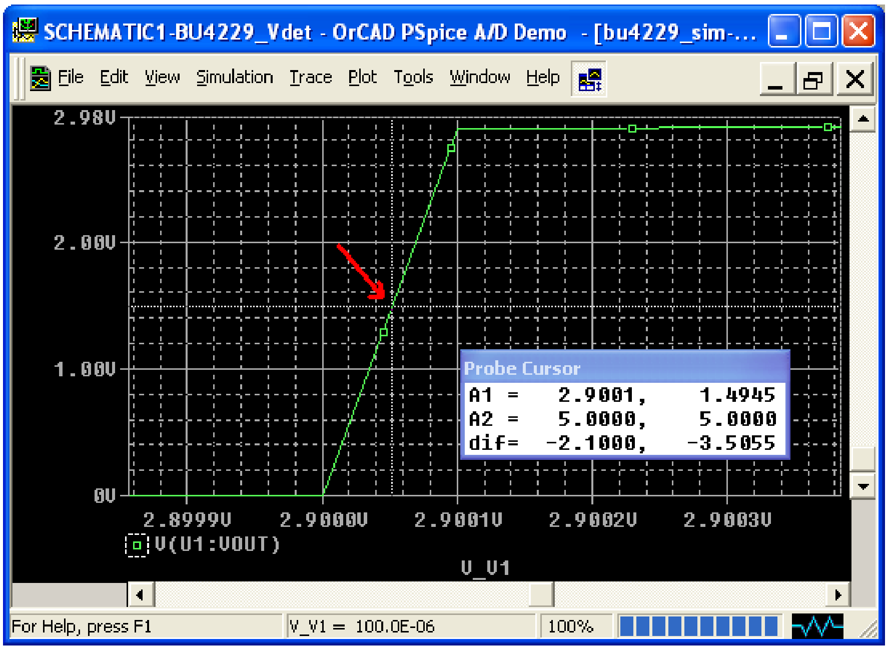
To begin measurement, first enable the Cursor by clicking on the Toggle cursor button (shown below).



The cursor will appear, with the coordinates shown in a small window. Navigate to the desired point. In this example we will measure the threshold voltage.

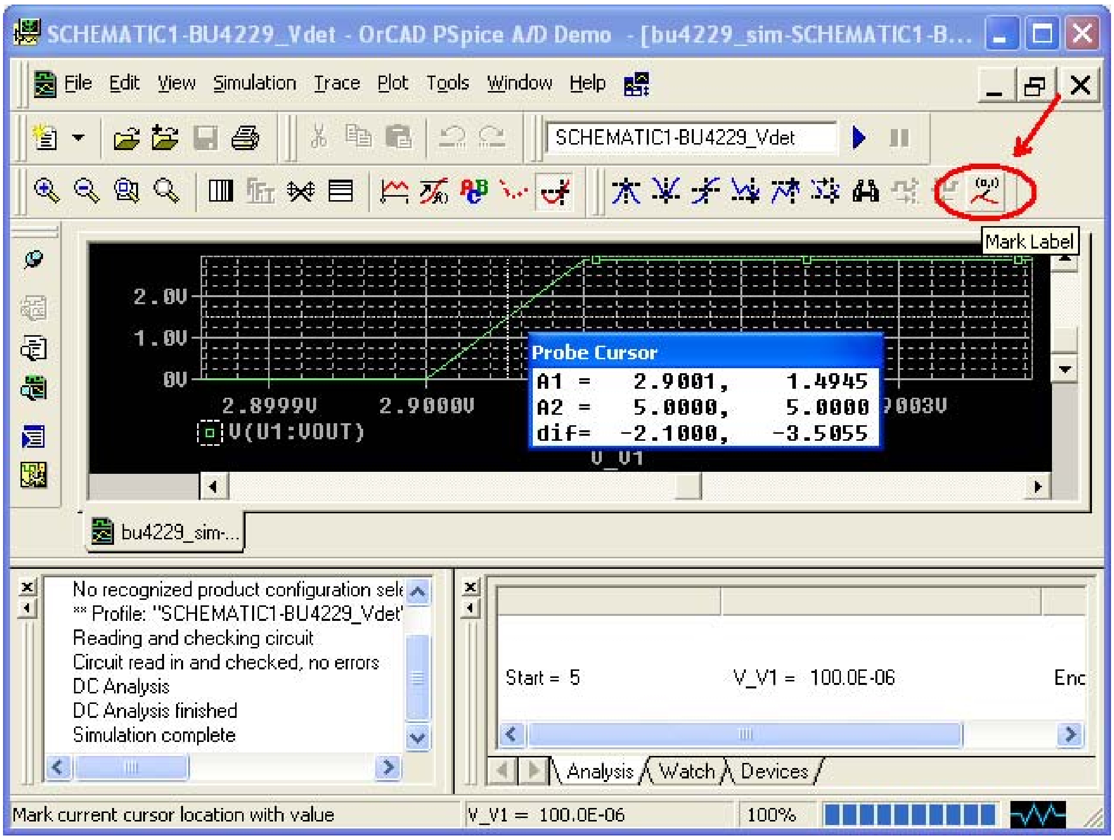


Zooming is possible by selecting the area and pressing Ctrl+A. For this measurement the cursor is placed midway on the VOUT drop (VDD sweep from 5V to 0V).

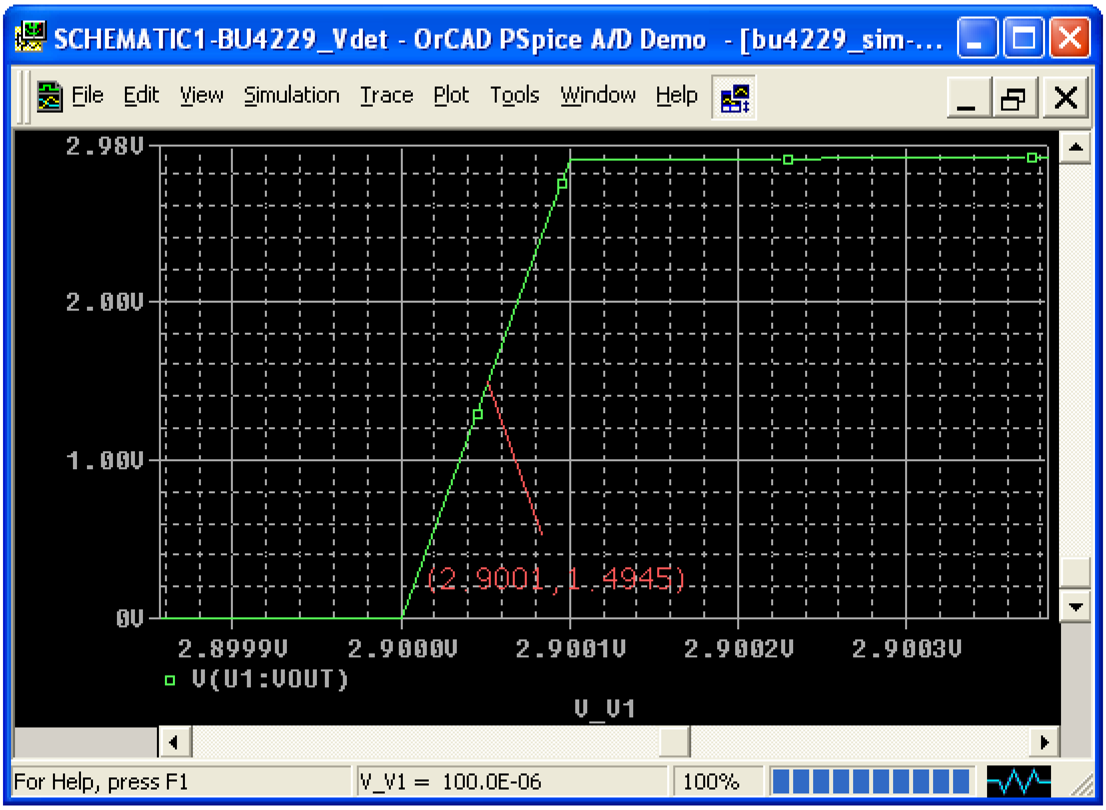


The threshold voltage can be found by looking at the coordinates of the cursor. In this example the threshold voltage is at 2.9001V of VDD.

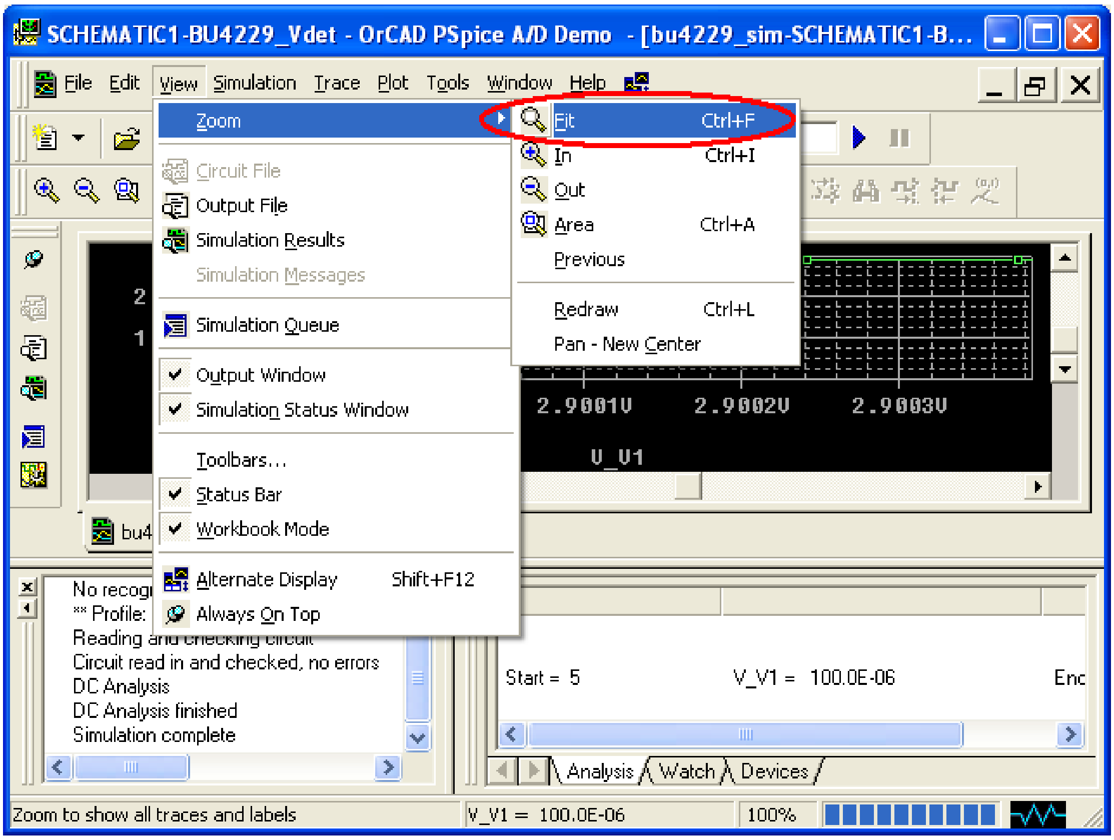
3.4 Annotate Measurement Values



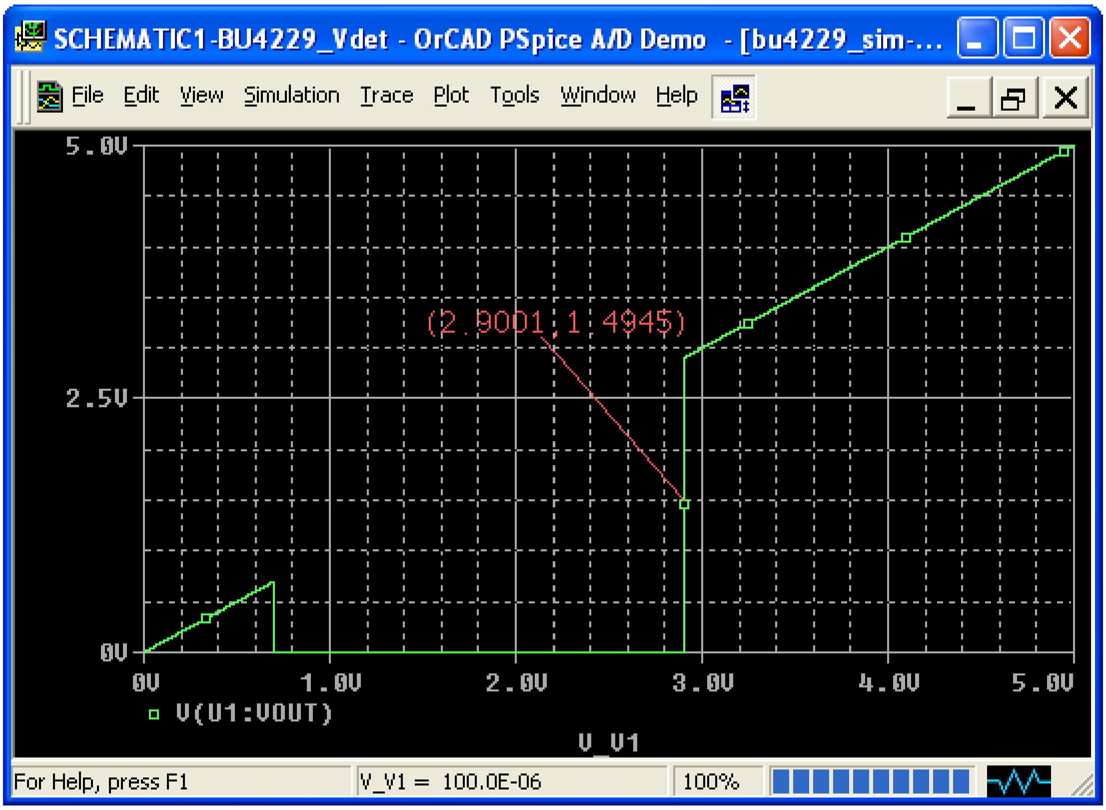
To add an annotation of the measurement, click on the Mark Label button shown above. Ensure that the cursor is at the desired measurement point before making the annotation. Otherwise, the annotation will be incorrect. The annotation will display the coordinates of the cursor.



To fit the graph for viewing, go to View tab then select Fit under Zoom.



A fit view of the graph is shown below.



From the simulation results, VDET of the BU4229 is verified to be 2.9V with VOUT=470kΩ pulled up to VDD, CT=100pF, with a VDD sweep from 5V to 0V.

Notes

No copying or reproduction of this document, in part or in whole, is permitted without the consent of ROHM Co.,Ltd.

The content specified herein is subject to change for improvement without notice.

The content specified herein is for the purpose of introducing ROHM's products (hereinafter "Products"). If you wish to use any such Product, please be sure to refer to the specifications, which can be obtained from ROHM upon request.

Examples of application circuits, circuit constants and any other information contained herein illustrate the standard usage and operations of the Products. The peripheral conditions must be taken into account when designing circuits for mass production.

Great care was taken in ensuring the accuracy of the information specified in this document. However, should you incur any damage arising from any inaccuracy or misprint of such information, ROHM shall bear no responsibility for such damage.

The technical information specified herein is intended only to show the typical functions of and examples of application circuits for the Products. ROHM does not grant you, explicitly or implicitly, any license to use or exercise intellectual property or other rights held by ROHM and other parties. ROHM shall bear no responsibility whatsoever for any dispute arising from the use of such technical information.

The Products specified in this document are intended to be used with general-use electronic equipment or devices (such as audio visual equipment, office-automation equipment, communication devices, electronic appliances and amusement devices).

The Products specified in this document are not designed to be radiation tolerant.

While ROHM always makes efforts to enhance the quality and reliability of its Products, a Product may fail or malfunction for a variety of reasons.

Please be sure to implement in your equipment using the Products safety measures to guard against the possibility of physical injury, fire or any other damage caused in the event of the failure of any Product, such as derating, redundancy, fire control and fail-safe designs. ROHM shall bear no responsibility whatsoever for your use of any Product outside of the prescribed scope or not in accordance with the instruction manual.

The Products are not designed or manufactured to be used with any equipment, device or system which requires an extremely high level of reliability the failure or malfunction of which may result in a direct threat to human life or create a risk of human injury (such as a medical instrument, transportation equipment, aerospace machinery, nuclear-reactor controller, fuel-controller or other safety device). ROHM shall bear no responsibility in any way for use of any of the Products for the above special purposes. If a Product is intended to be used for any such special purpose, please contact a ROHM sales representative before purchasing.

If you intend to export or ship overseas any Product or technology specified herein that may be controlled under the Foreign Exchange and the Foreign Trade Law, you will be required to obtain a license or permit under the Law.

ROHM SEMICONDUCTOR

Thank you for your accessing to ROHM product informations.
More detail product informations and catalogs are available, please contact us.

ROHM Customer Support System

http://www.rohm.com/contact/