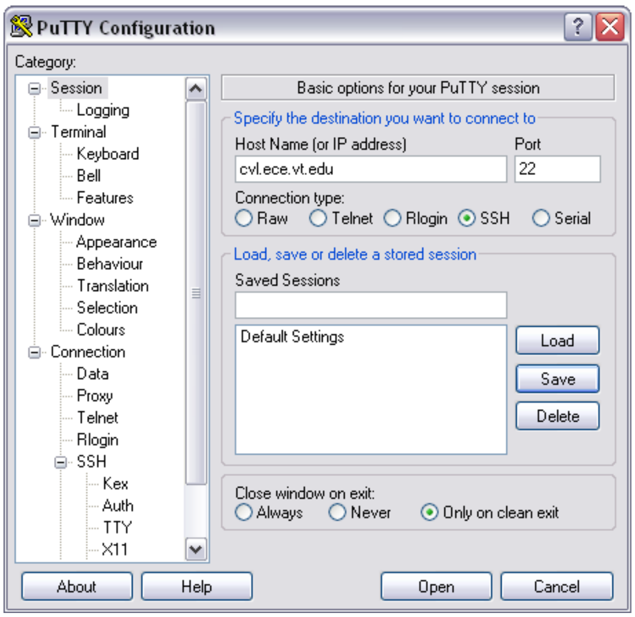
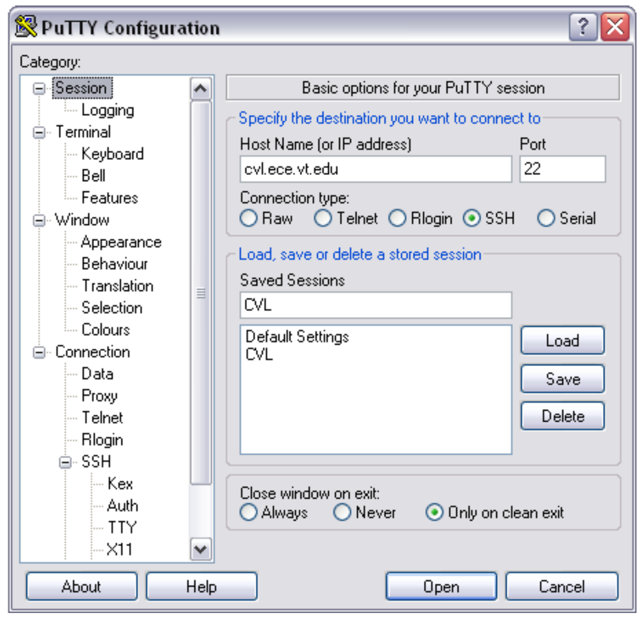
Using Cadence Virtuoso, a UNIX-based OrCAD PSpice-like program, Remotely on a Windows Machine

A. Launch **PuTTY**.

1. Load the **Saved Session** that has **Enable X11 forwarding** and the **Host Name** is **cvl.ece.vt.edu**.
    a. In the left image, the changes were made to the **Default Settings** session. In the right image, a session called **CVL** was saved, which has **Enable X11 forwarding** and the **Host Name** is **cvl.ece.vt.edu**.





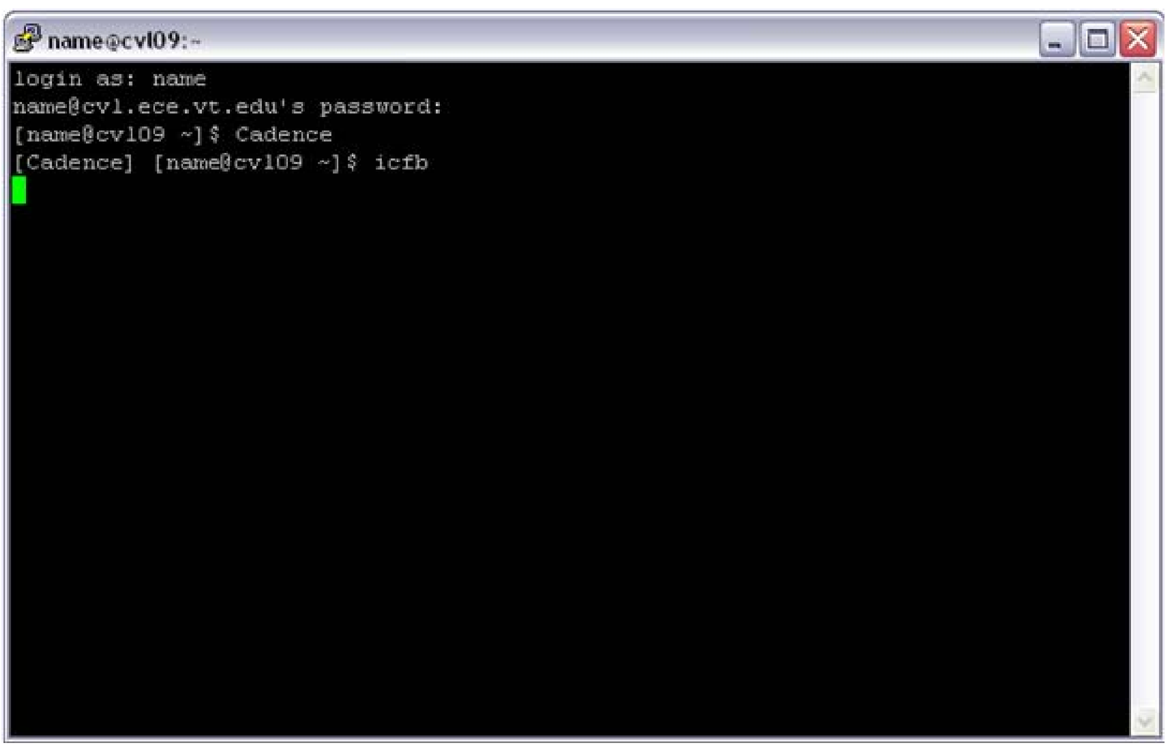
    b. If you did not save these settings when you set up the remote access, follow the steps 1c – e listed in the Remote Access.docx.

2. Click **Open**.

B. A terminal emulator window will open.

1. Type in your CVL user name and hit Enter.
    a. If you do not have a CVL account with user name and password, please go to the [CVL main page](). Click on [CVL Accounts]() under **Getting Started** and follow the instructions from there.
2. Type in your CVL password and hit Enter.

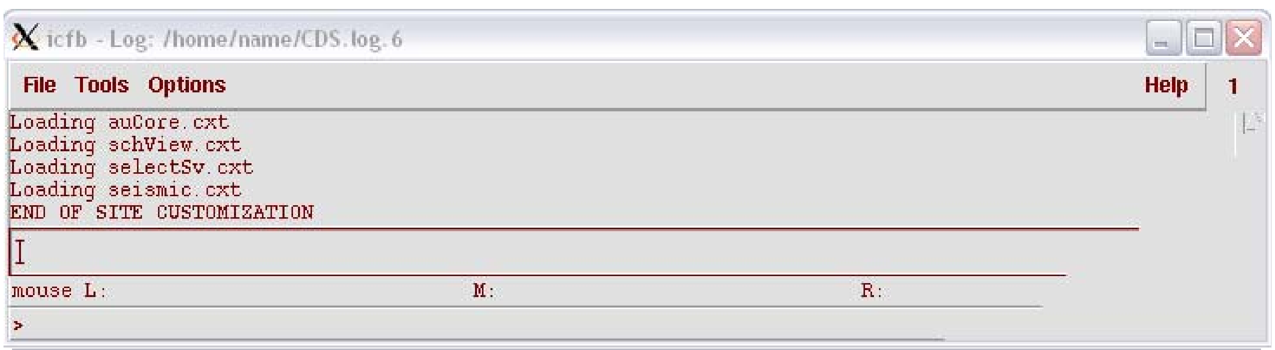
C. Once logged in, do the following to launch, Virtuoso, the Unix version of PSpice:
   1. Type **Cadence** on the next line and hit Enter.
   2. Type **icfb** on the next line and hit Enter.



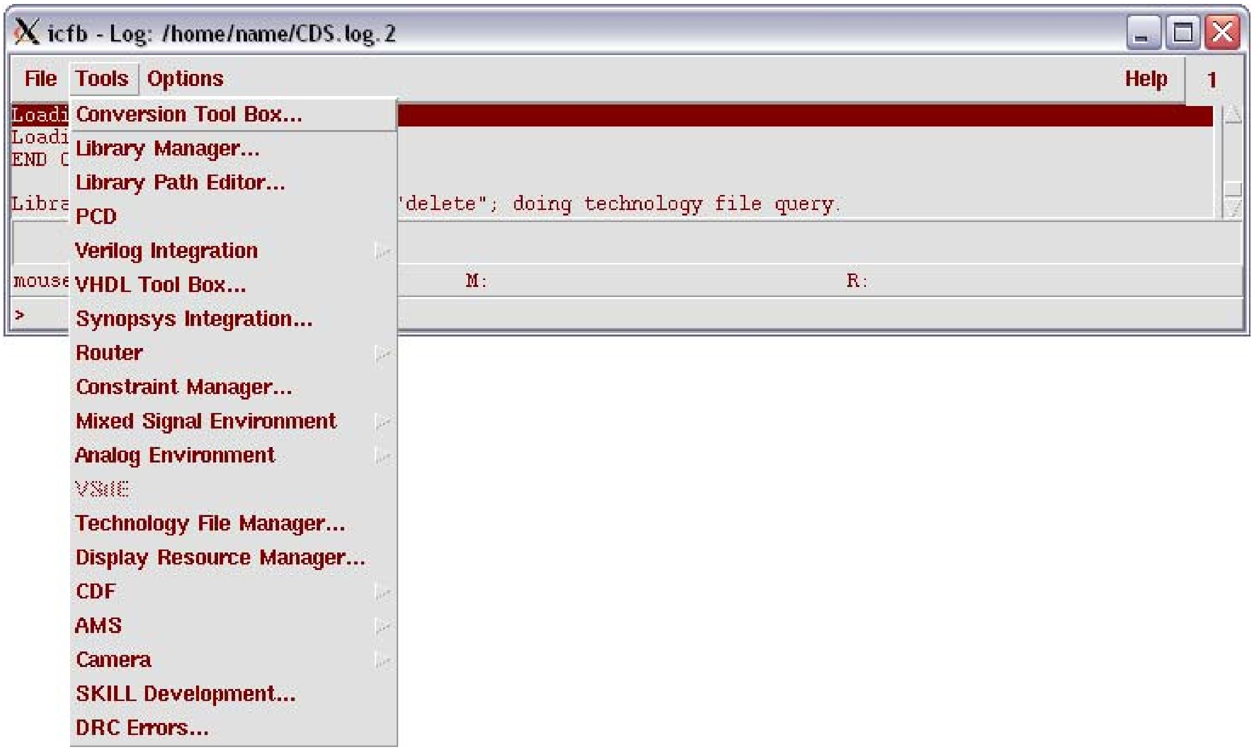
a. If an error message appears, type **exit** and then close PuTTY by typing **logout**. Run ***startxwin.bat***. Then, restart from the beginning of these instructions.

3. Shortly afterwards, an image with will pop up and then disappear indicating that the Cadence Virtuoso has been launched.

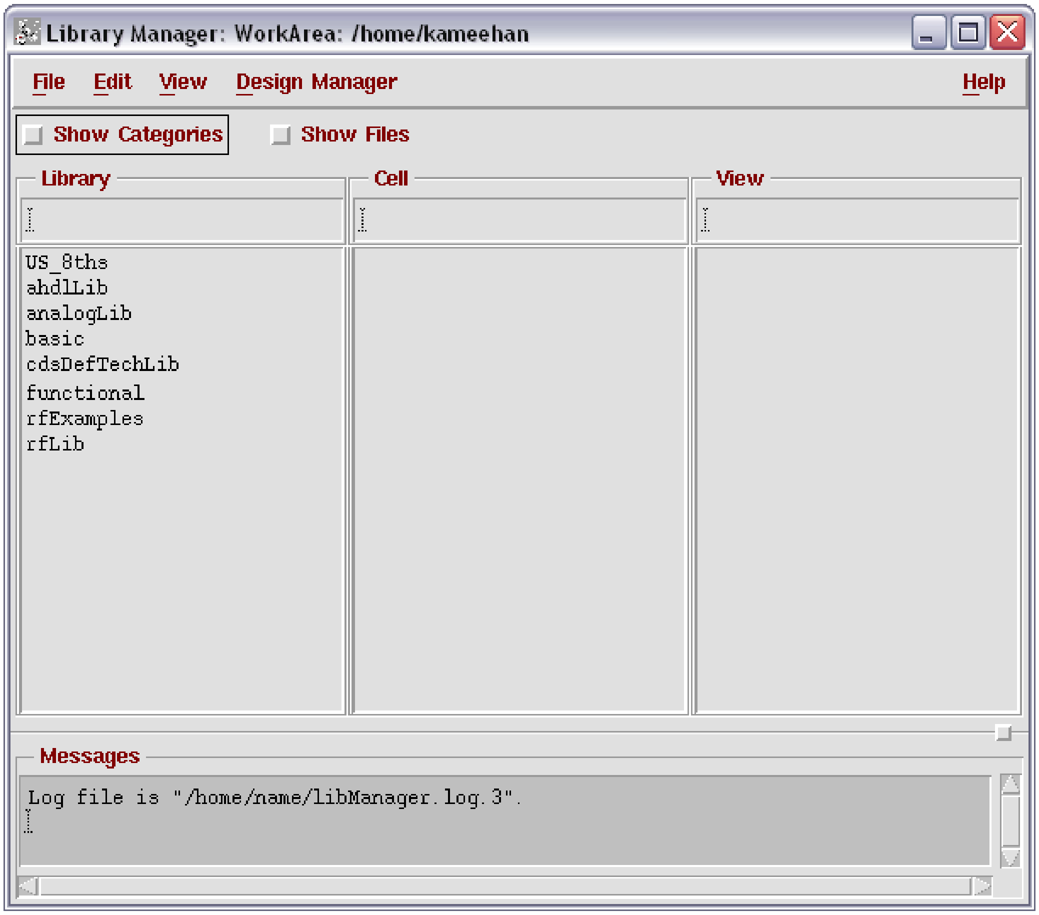
4. Almost immediately after this, two **cygwin** windows will open automatically. One is the **Log** (a Command Interpreter Window) and the second is **What's New**. Keep the **Log** window open. You may close the **What's New** window.



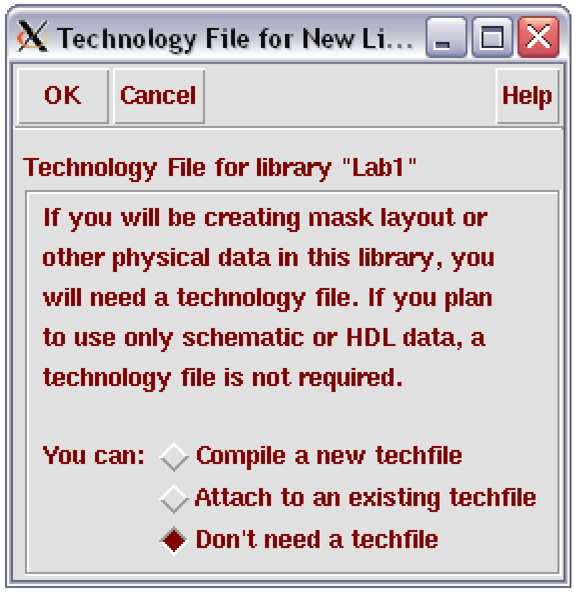
5. In the **Log** window, click on **Tools** and select **Library Manager**.



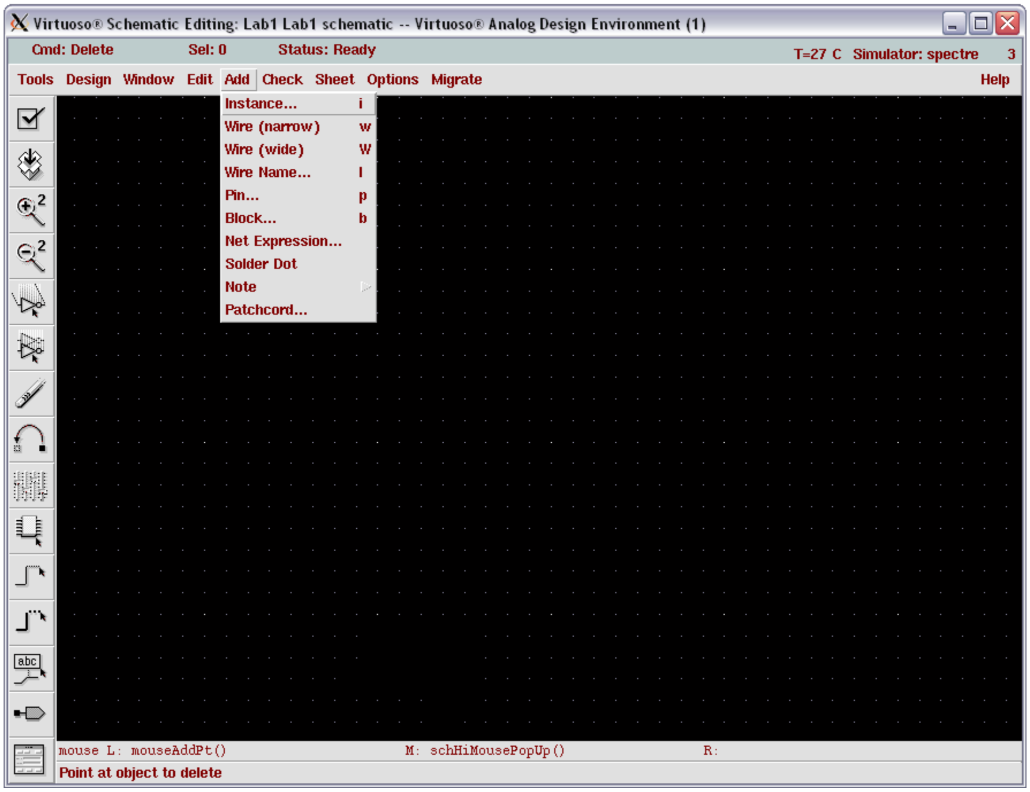
6. The Library Manager pop-up window contains a list of all of the parts libraries. You can make new libraries as you import PSpice models from the web, generate new device models yourself, or form a library using the parts available in the other existing libraries as you draw a new circuit schematic.



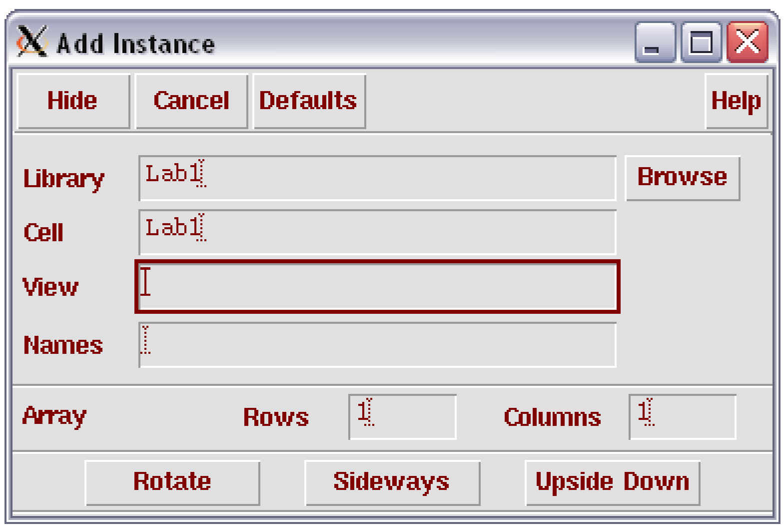
a. To create a new library,
   i. Select **File** →**New** → **Library**.
   ii. Enter the name you wish to use for the library in the **Name** field
   iii. Click **OK**.
      a. The name will now show up on the list in the **Library Manager** window.
   iv. In the **Technology File for New Library** window that will now appear, select the option **Don't need a techfile** and click **OK**.
      a. The **Technology File** window will then close.



7. Open a **Cell View** in one of two ways
   a. Technique 1:
      i. Click on one of the names in the list of libraries in the **Library Manager**. The name should now appear in the box directly under the word **Library**.
      ii. Next, select **File** →**New** → **Cell View**.
      iii. **Create New Cell View** window opens.
         a. Type a name into the box named **Cell Name**, which is outlined in red.
         b. Select **Composer-Schematic** in the box by **Tool** if it is not the default.
         c. Click **OK**.
      iv. In the **Library Manager** window, the box under **Cell** has been filled with the **Cell Name** entered in step 7.a.iii. a. and schematic has been written in the large box under **View**.
   b. Technique 2: will be described at some later date.
8. **Virtuoso@ Schematic Editing** window is launched.
   a. To add a component to the schematic, go to **Add → Instance** on the upper toolbar or click on the DIP package icon  on the left side of the window.



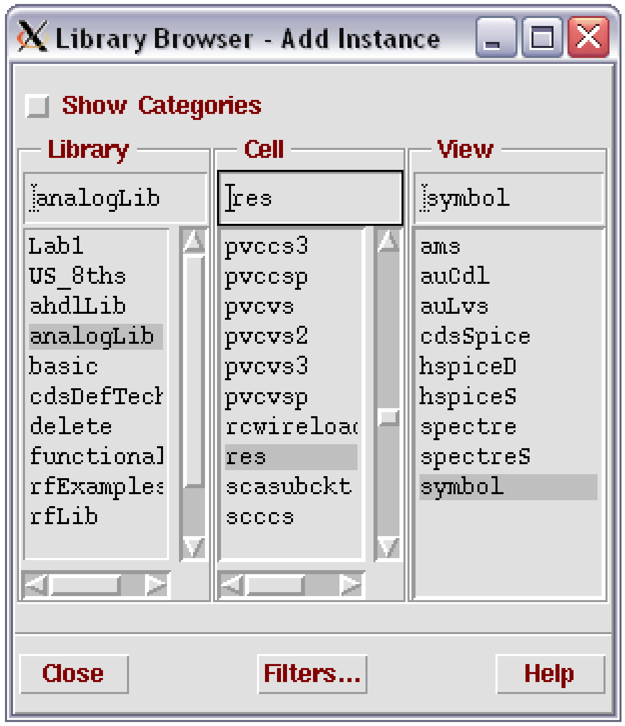
b. The **Add Instance** window will open.



c. Click on **Browse** in the **Add Instance** window to select the **Library** in the **Library Browser** window that now opens that contains the part that you want to add to the schematic. Most of the parts that will be used in ECE 2074 and 3074 are in **analogLib**.
   i. Click on the part from the list that is in the large box under **Cell**.

a. **res** is a resistor, **cap** is a capacitor, **ind** is an inductor, **gnd** is ground.

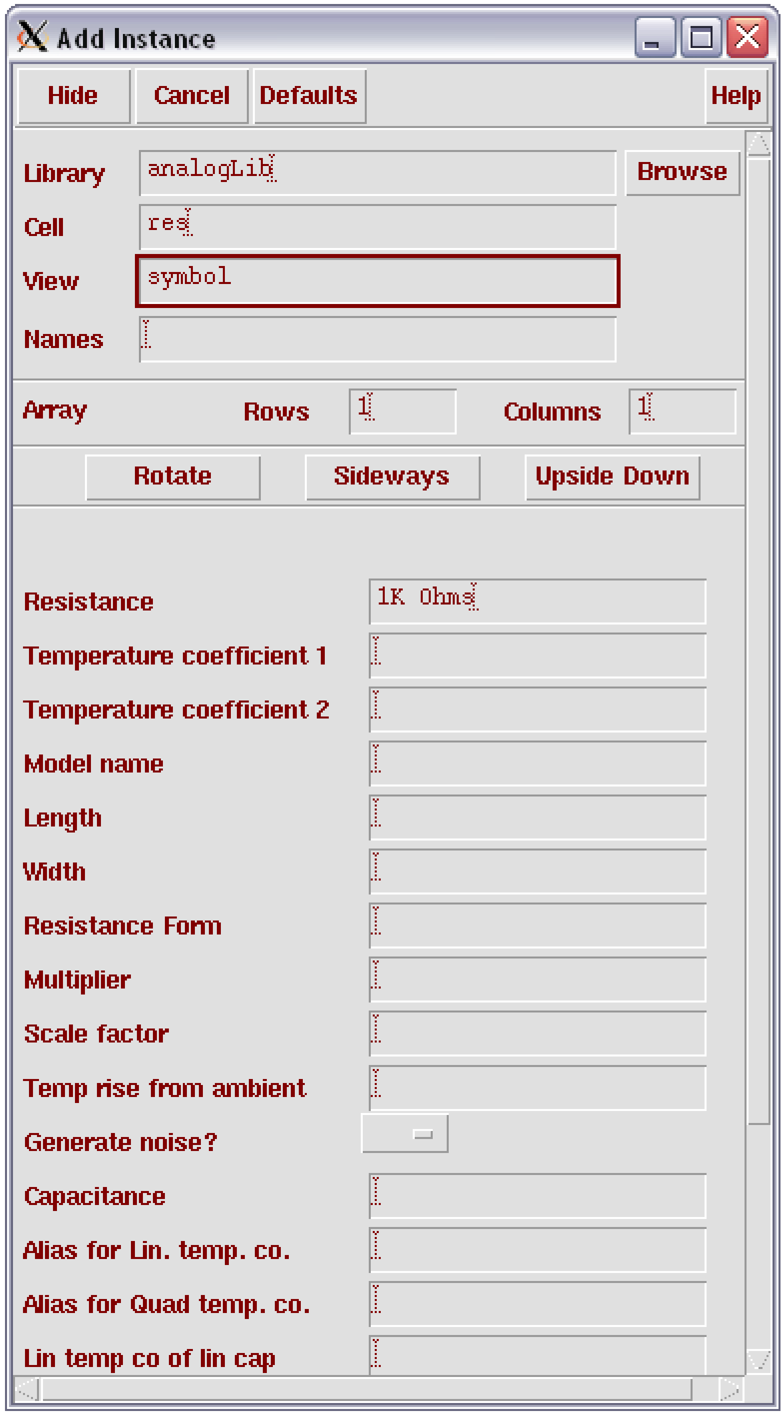
ii. Click on **symbol** in the list that then opens in the large box under View.



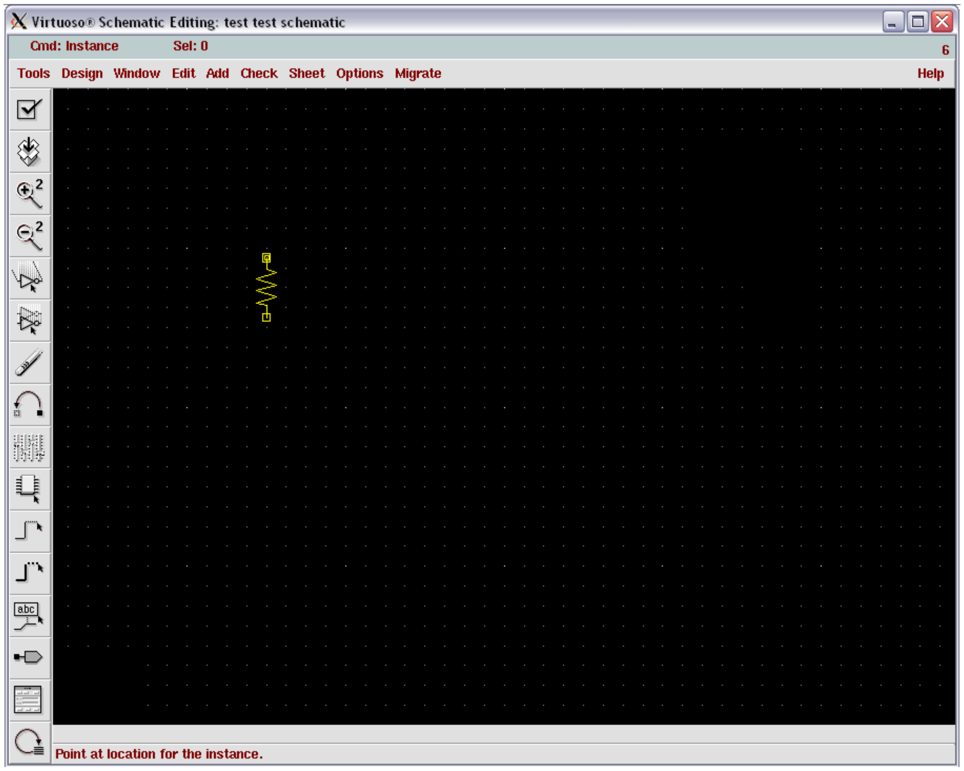
d. Alternatively, you can type the part name into the box to the right of **Cell** in the **Add Instance** window to call the part from the library selected and type **symbol** into the box to the right of **View**.

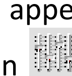
   i. The part name is case sensitive.

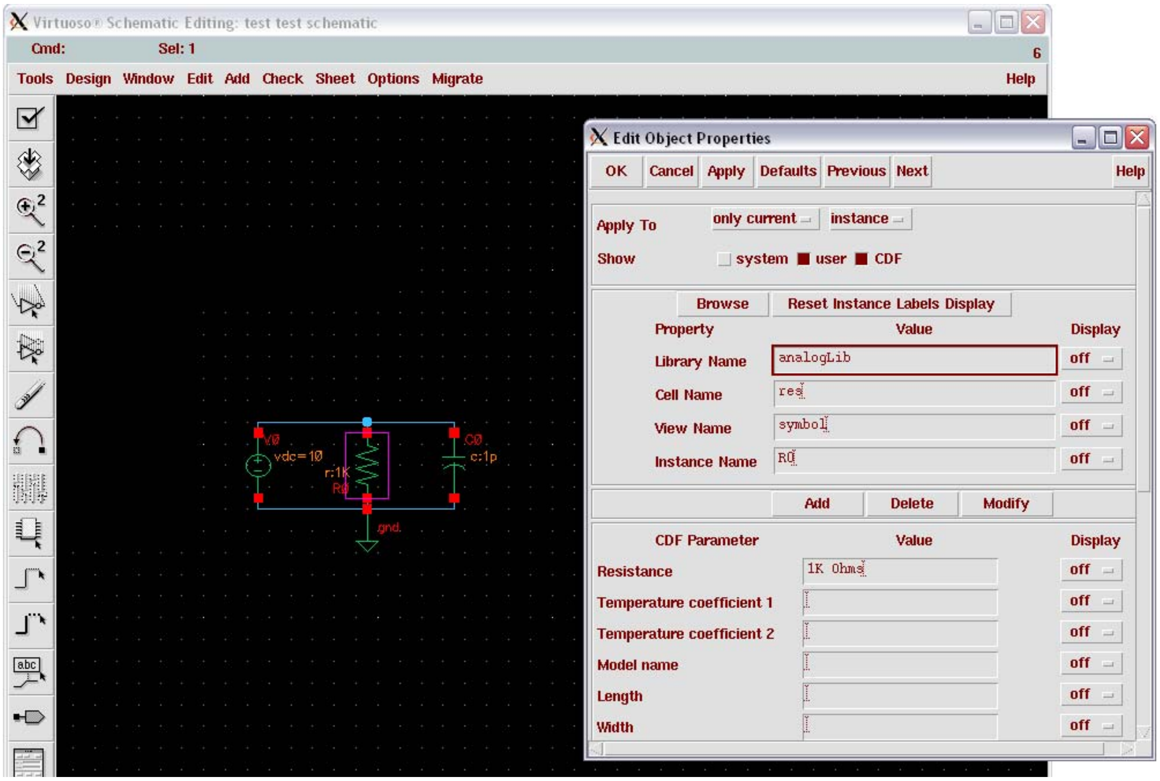
e. The **Add Instance** window should expand to include boxes where you can **Name** the component and enter values for its properties, such as its resistance if you have selected res.



f. Now move your mouse on to schematic in the **Virtuoso Schematic Editing** window. The symbol of the part that you selected should appear.



i. Right click to rotate the symbol.
ii. Left click to place it on the schematic.
   a. To rotate a part after it has been placed, you can either select the component by clicking and press **r** or you select the component and select **Edit → Rotate**
   b. To change the value of a part after it has been place, click on the symbol section of the part once. A box should appear around the part. Then, click on the Properties icon  or select **Edit →Properties → Object**, or press the **q** key. The **Edit Object Properties** window will open. After modifying the part's properties, close the **Edit Object Properties** window.



iii. Additional parts of the same type may be added to the schematic by continuing to left click.

iv. Click the **Esc** key on your keyboard to end the placement of a part or close the **Add Instance** window.

g. To wire the part, click on the narrow wire icon  or select **Wire (narrow)** from the list on **Add** in the upper toolbar.

h. To save your design, click on the Save  or the Check and Save  icons. Alternatively, you can select **Design → Save** or **Design → Check and Save** on the upper toolbar.

i. The **Check and Save** option will check your design for any errors or warnings that may be present. You will be notified of errors by a visual indicator within your design as well as the error or warnings information within the **Log** window. If you created your schematic properly, you should see the following message in the CIW window shown here.

a. If there are errors, you will see small flashing squares within the schematic editing window. Read the error and warning messages in the **Log** window and the correct the errors/warnings as necessary.

b. Once correct, check and save your design by selecting **Design → Check and Save** or the **Check and Save** icon .

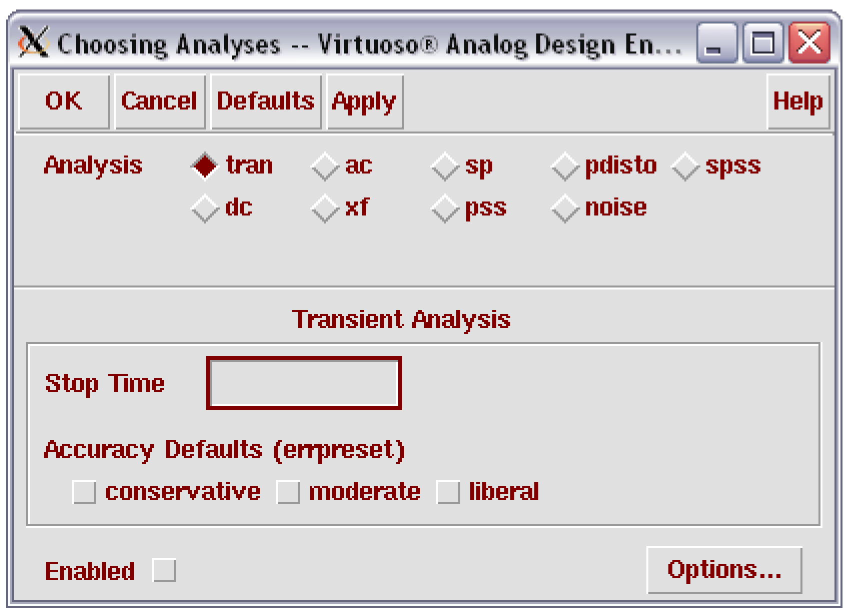
i. To run a simulation, you first have to select **Tools → Analog Environment** in the **Virtuoso Schematics Editing** window.

   i. In the **Analog Environment** window, select **Setup → Simulator/Directory/Host**.

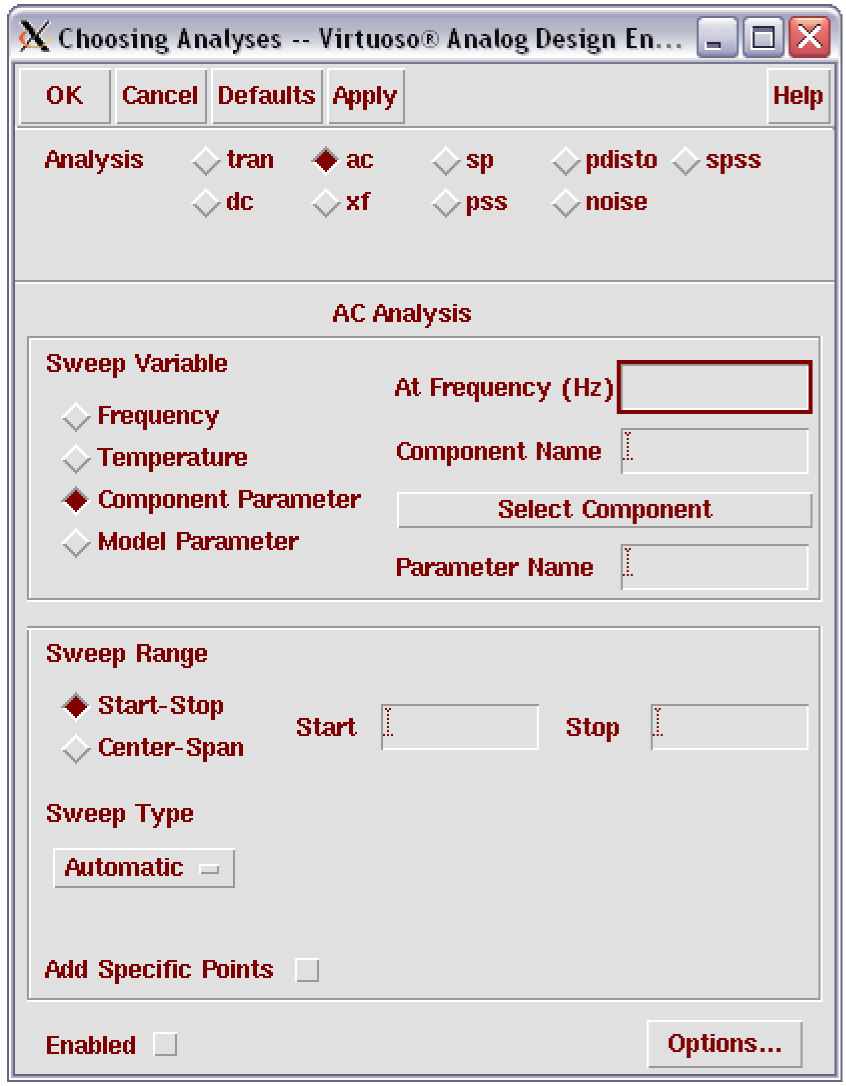
      a. In the Choosing Simulator/Directory Host window, select the **Simulator** as ***spectreS.*** Note that the default is ***spectre***; however, this simulator currently does not run properly.
      b. Make sure that the **Project Directory** is ***~/cadence/simulation***.
      c. Click on the OK button.
      d. If you are prompted to **Save Current State**, select **Yes** and enter the state name as ***state1***.

   ii. Back in the **Analog Environment**, select **Analyses → Choose**.

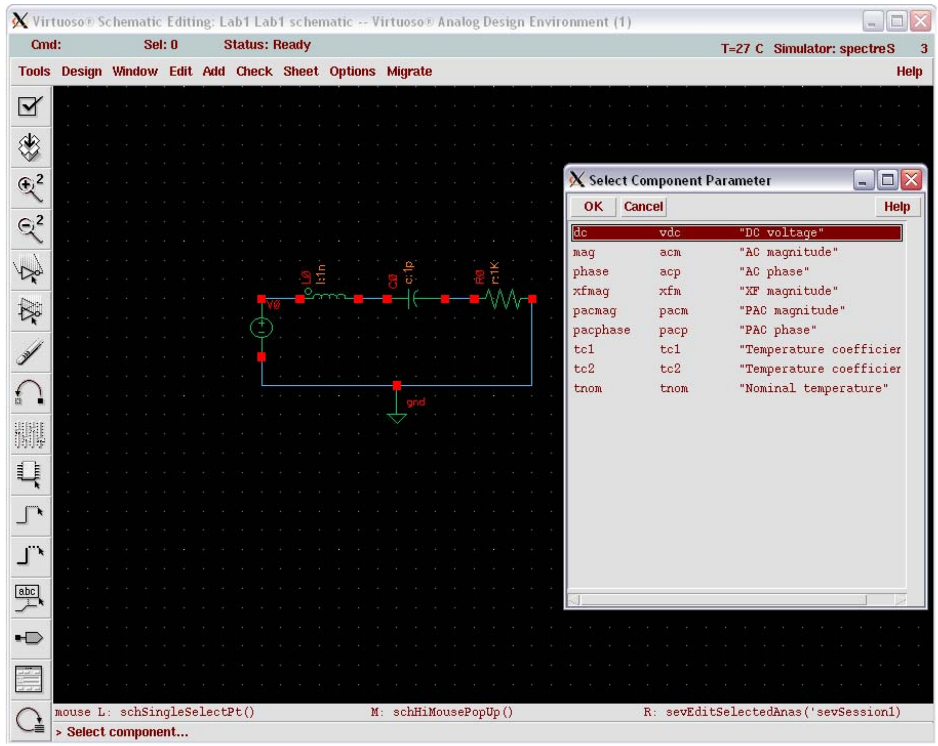
      a. In the **Choosing Analyses** window, click on the type of analysis that you want to run and then complete the boxes required for that analysis and then click **Enabled** at the lower left corner of this window. Click **OK** if this is the only analysis that you would like to run. Otherwise, click **Apply** and then select the additional analyses to be performed.



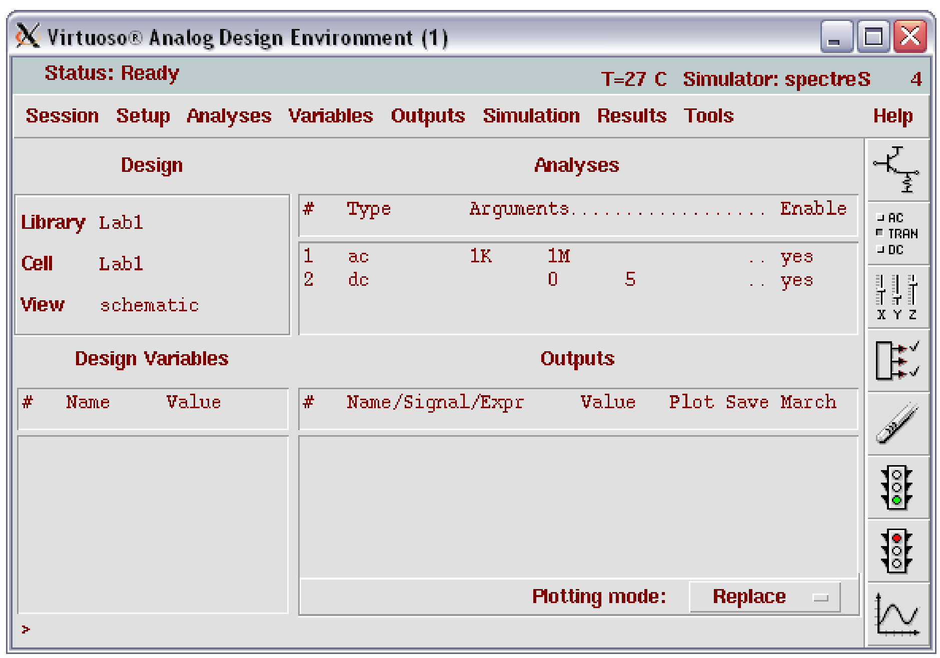
         i. If you chose the **dc** or **ac** analysis, which are equivalent to the DC or AC Sweep in OrCAD PSpice and you chose to sweep a **Component Parameter**, you will then have to **Select Component**. Click on this button and then click on the component symbol in the **Virtuoso Schematics Editing** window.



b. Chose the **Component Parameter** that you would like to vary during the simulation by clicking to highlight it in the **Select Component Parameter** pop-up window. Then click **OK**.

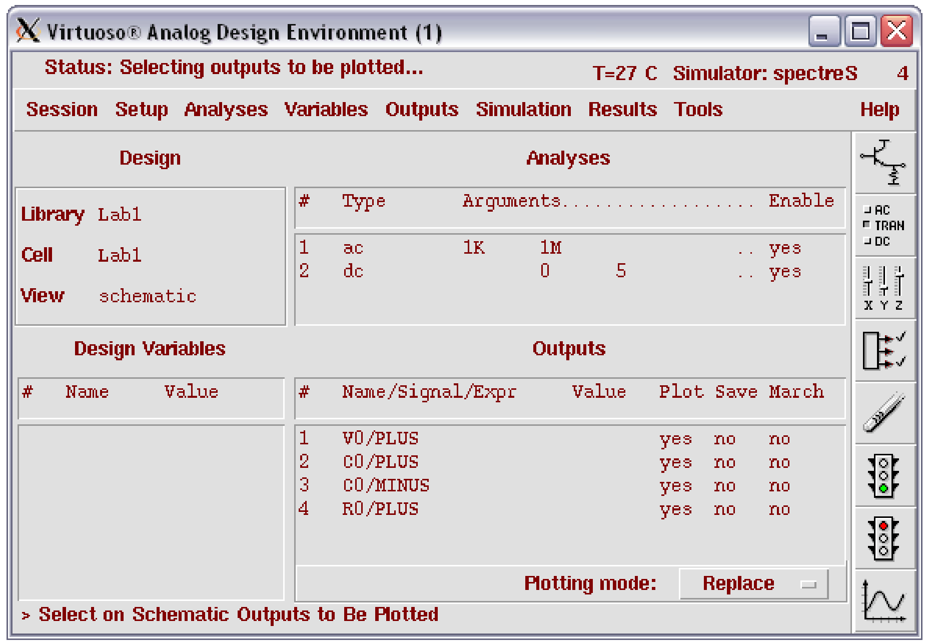


c. A list of the analyses that you have selected appear in the **Analog Design Environment** window.



d. To identify which outputs should be plotted at the end of the simulation, click on **Outputs → To Be Plotted → Select On**

Schematic in the **Analog Design Environment** window. Go to **Virtuoso Schematics Editing** and click on the nodes where you would like the voltages and currents displayed in the output plot. A list of these nodes will appear in the **Analog Design Environment** window.



e. To run the simulation, select **Simulation → Run** on the upper toolbar of the **Analog Design Environment** window or click on the  icon on the right. A plot of the outputs should be displayed at the end of the simulation.