Analog IC Simulation

Mentor Graphics 2006

Santa Clara University

Department of Electrical Engineering

Date of Last Revision: March 29, 2007

Table of Contents

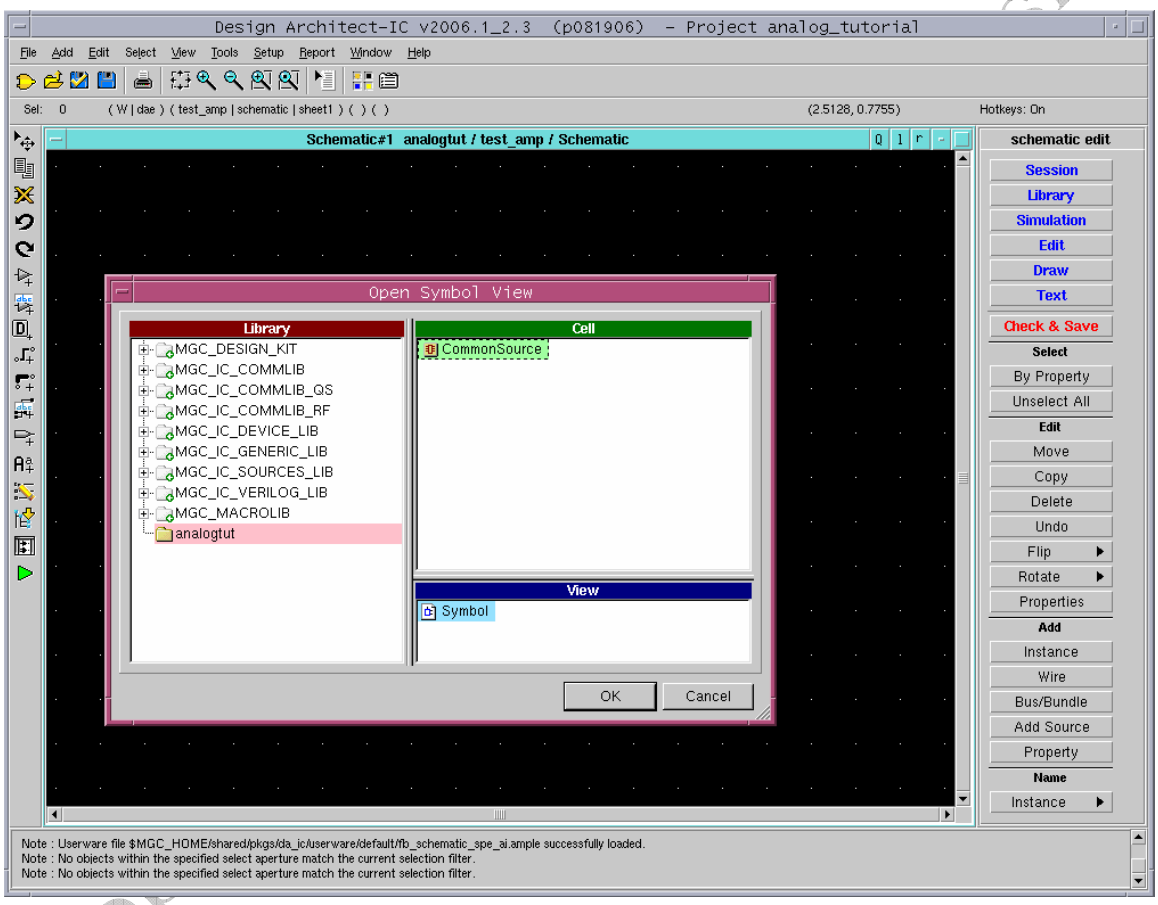
1. Objective

This Tutorial contains a step-by-step procedure for simulating a common source amplifier circuit in the Mentor Graphics Design Architect tool. It covers DC Analysis, Transient Analysis, DC Operating Point Analysis and AC Analysis simulation of the amplifier.

This tutorial assumes that the schematic of the amplifier has already been created. For help with creating schematic and symbol, please refer to the Analog IC Schematic Entry tutorial.

2. Basic Test Circuit Creation

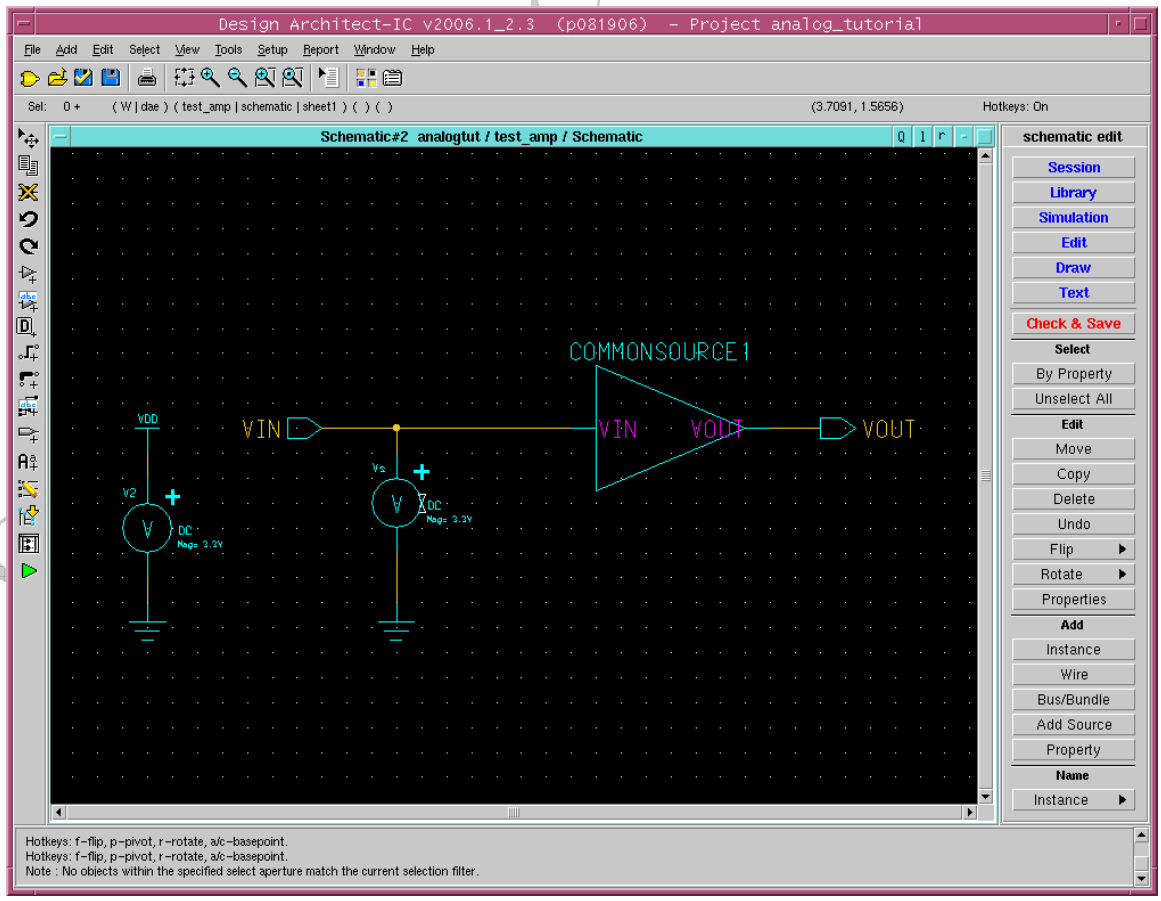
- Open **icstudio** from the command prompt as in the Schematic Entry tutorial
- Create a **new view** in the created library (**analogtut**) and name it **test_amplifier** with **view type** as **Schematic**. We will use this sheet to create basic test circuit required for the simulations.



1. Adding an Instance to the test circuit

A new schematic entry sheet opens in DA-IC. In the test_amp schematic:

- Add the symbol for the amplifier that you generated in the schematic entry tutorial.
  - Click **Add > Instance** from **schematic_edit** palette on RHS.
  - In the **Choose Symbol** popup window, click **Browse**, navigate and select **CommonSource** and click OK. Place this symbol of the inverter in the sheet as shown in the figure below.
- Add **PORTIN, PORTOUT, VDD** and **Ground** from the **Generic Lib**
- Name the ports as **VIN** & **VOUT,** and wire them to the **VIN** and **VOUT** pins of the amplifier symbol as shown in the figure.
- Add a **DC** source by selecting **IC_Library > Sources Library > DC**.
- Highlight the DC source, right click mouse**, Properties > Edit** to modify the voltage value of the source from **1V** to **3.3V**.
- Connect the positive node of the **DC source** to **VDD** symbol and negative terminal to the **Ground** symbol.
- Add second **DC** source for the input, and connect to **VIN**. Edit the properties to set **INST = Vs**
- Click **Check & Save**. If there are any errors, correct them before moving on.



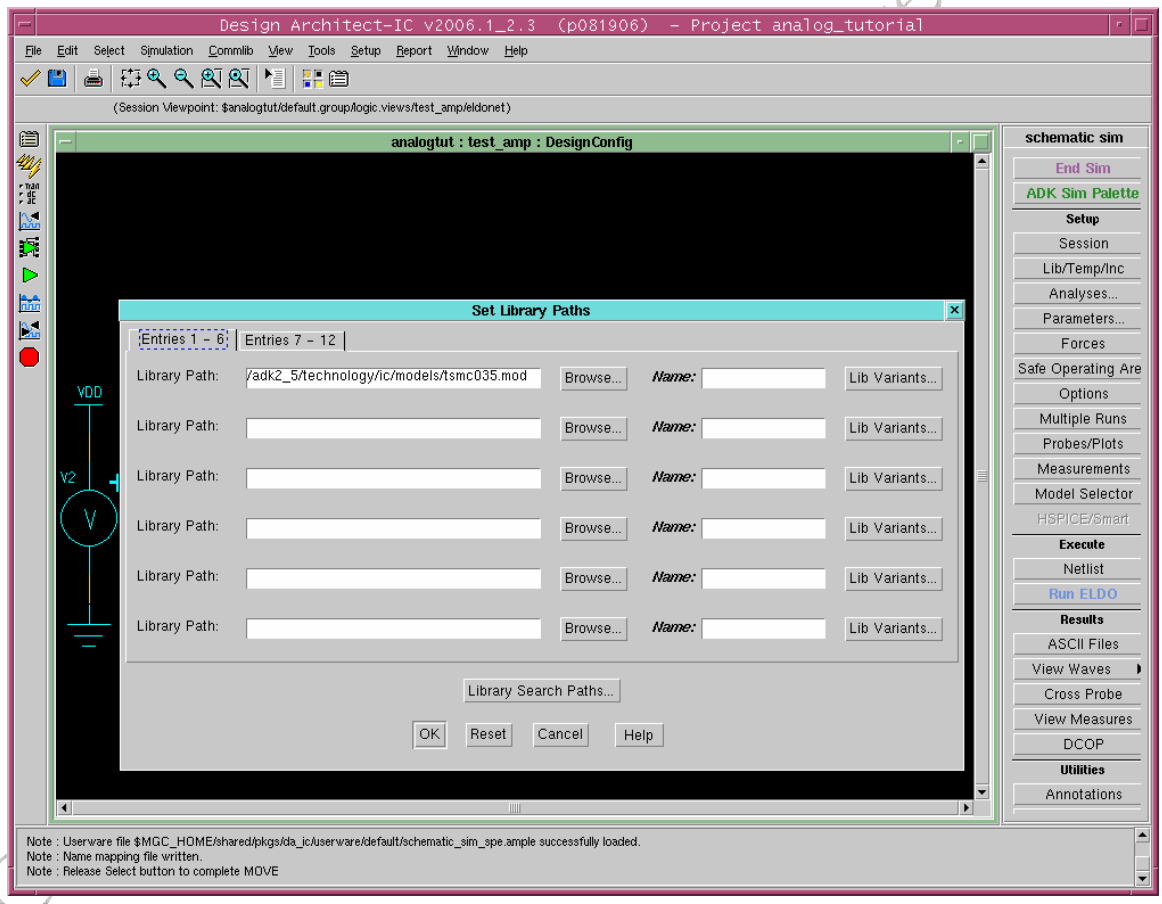
2. Setting up the simulation parameters

Click on **simulation** from the **schematic_edit** palette on the RHS to enter the simulation mode. A popup window appears with a Warning message, indicating that the schematic will be closed. Click **OK** to accept default options.

In the simulation mode (schematic_sim palette):

- On the simulation palette, click the button **Lib/Temp/Inc > Library.**
- Type **/opt/mentor-2004.3/sol/adk2_5/technology/ic/models/tsmc035.mod** in **Library path** box and click **OK**.
  This includes the TSMC0.35 micron BSIM model for the simulations.

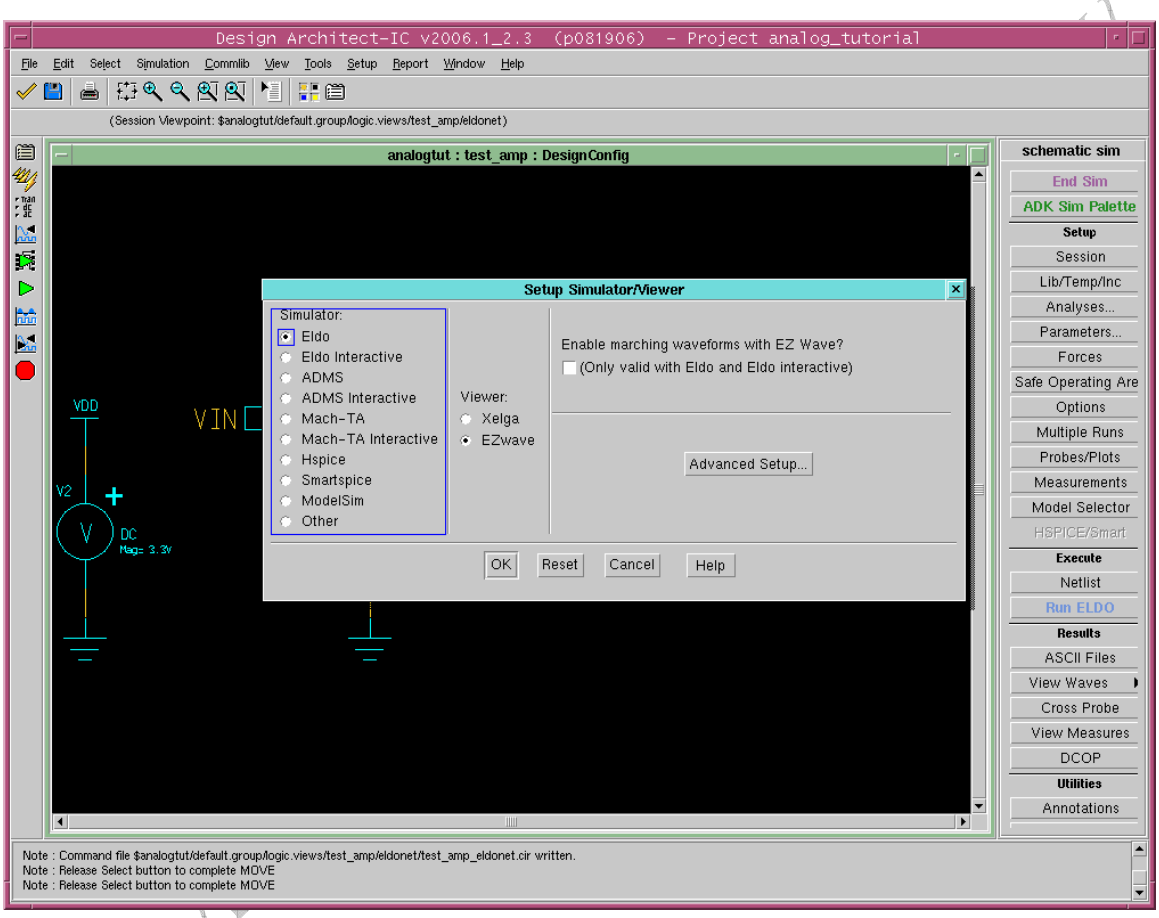
  The screen should look like the figure below:



3. Setting up the Simulator / Viewer

In the simulation mode (schematic_sim palette),

- Click **Session > Simulator / Viewer**.
- Select **Eldo** under Simulator and **EZwave** under Viewer and click **OK**.



Click on **Check & Save**. You are now ready to run an analysis on the amplifier.

2. DC Analysis

For a DC analysis, we will sweep the input voltage, **Vs**, from 0V to 3.3V and view its effect on the output.

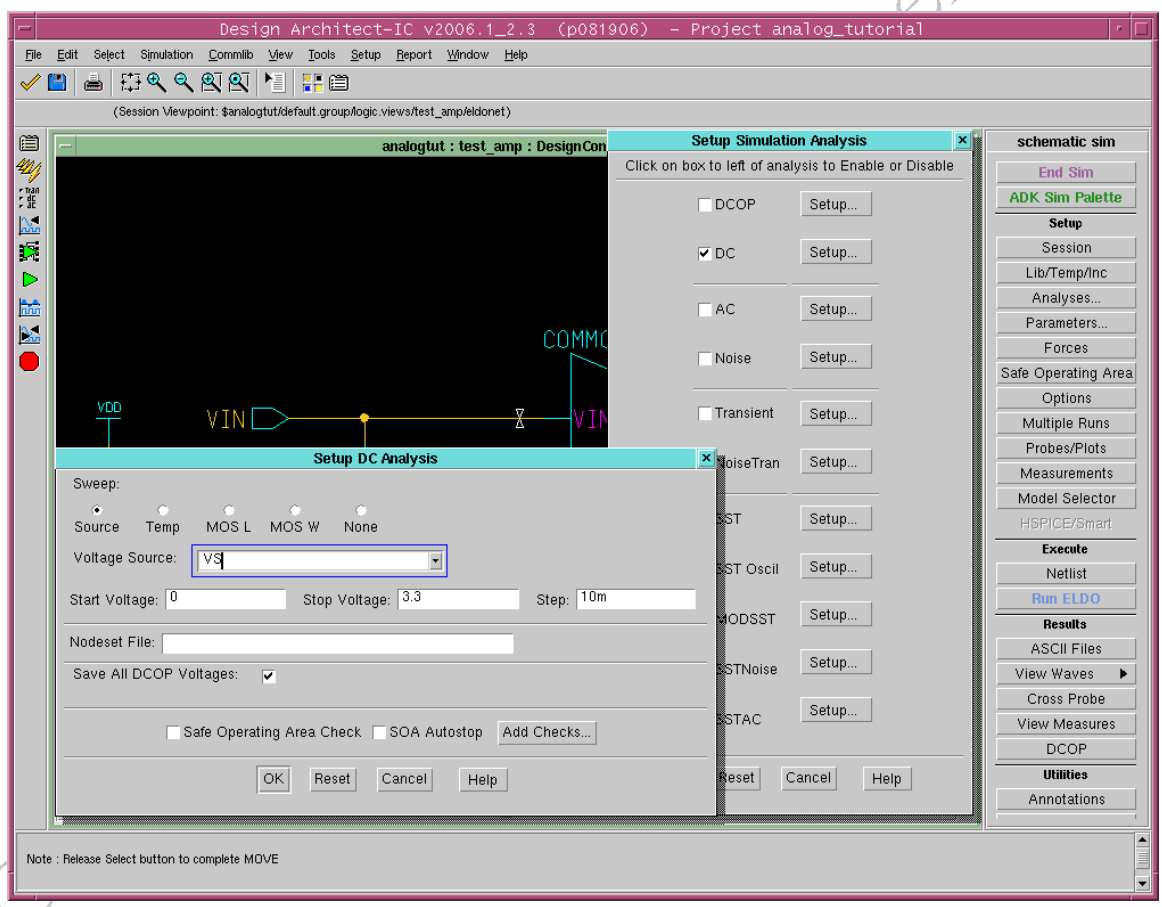
1. Editing and simulating the test circuit

- Open the **test_amp** schematic sheet and make sure that you are in **schematic_edit** mode.
- From the steps above, the input to the amplifier should already be a **DC** voltage source between **VIN** and Ground. If not, change the input source to a **DC** voltage source. The source should be named **Vs**. (Note that the value of the DC voltage source is 1V, by default. For the DC sweep we will be doing, this value does not matter and so you can leave it as it is.)
- Click **Check & save** the sheet and then enter the **simulation** mode by clicking on **simulation** from the **schematic_edit** palette on the RHS. Click **OK** to accept the default options in the Warning popup window that appears.

2. Setting up the simulation parameters

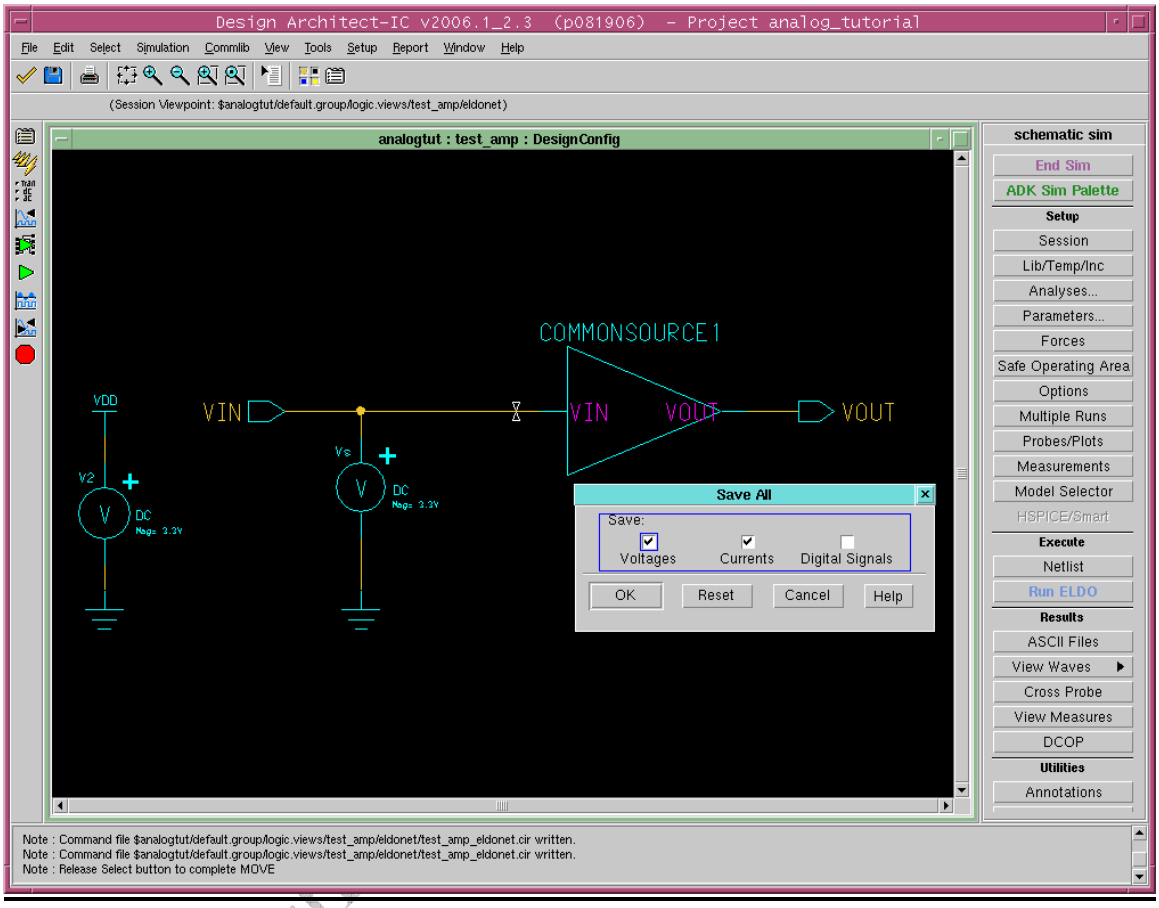
On the **schematic_sim palette:**

- Click on **Lib/Temp/Inc > Library** to make sure that **/opt/mentor-2004.3/sol/adk2_5/technology/ic/models/tsmc035.mod** appears in **library path** box and click **OK**.
- Click **Setup > Analyses**.
- In the **Setup Simulation Analysis** window that appears, select **DC** and click on **Setup** associated with **DC**. In the **Setup** box that appears, select **Source** and select **VS** for **Voltage source**, put **0** in the **start field**, **3.3** in the **stop** field and **10m** in the **step** field and then click **OK**.



2. Setting up the simulation parameters

- Click **Probes/Plots > Save All…**from the **Palette**. This opens a dialog box for choosing which signals should be saved for later analysis.
- Make sure the **Voltages** and **Currents** boxes are checked, and click **OK**.



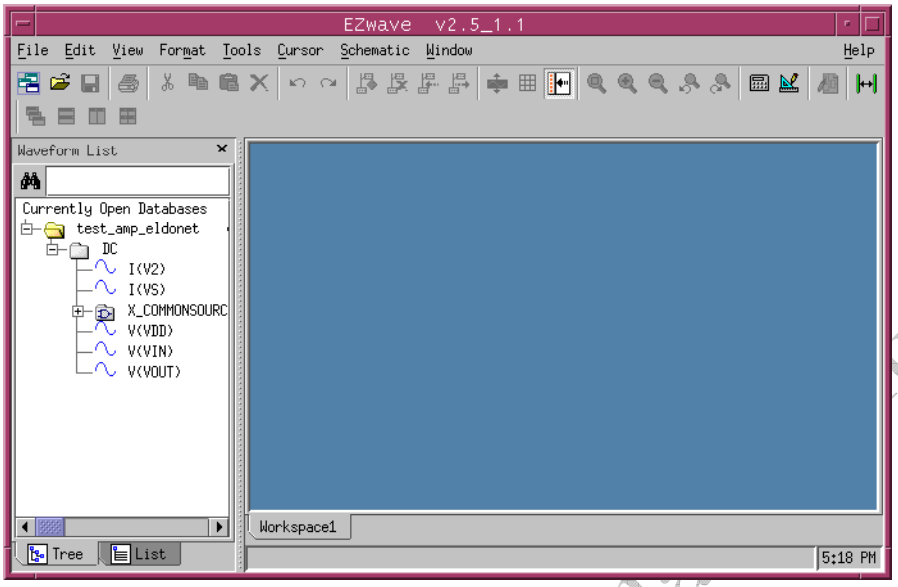
3. Executing the simulation setup

On the **schematic_sim palette:**

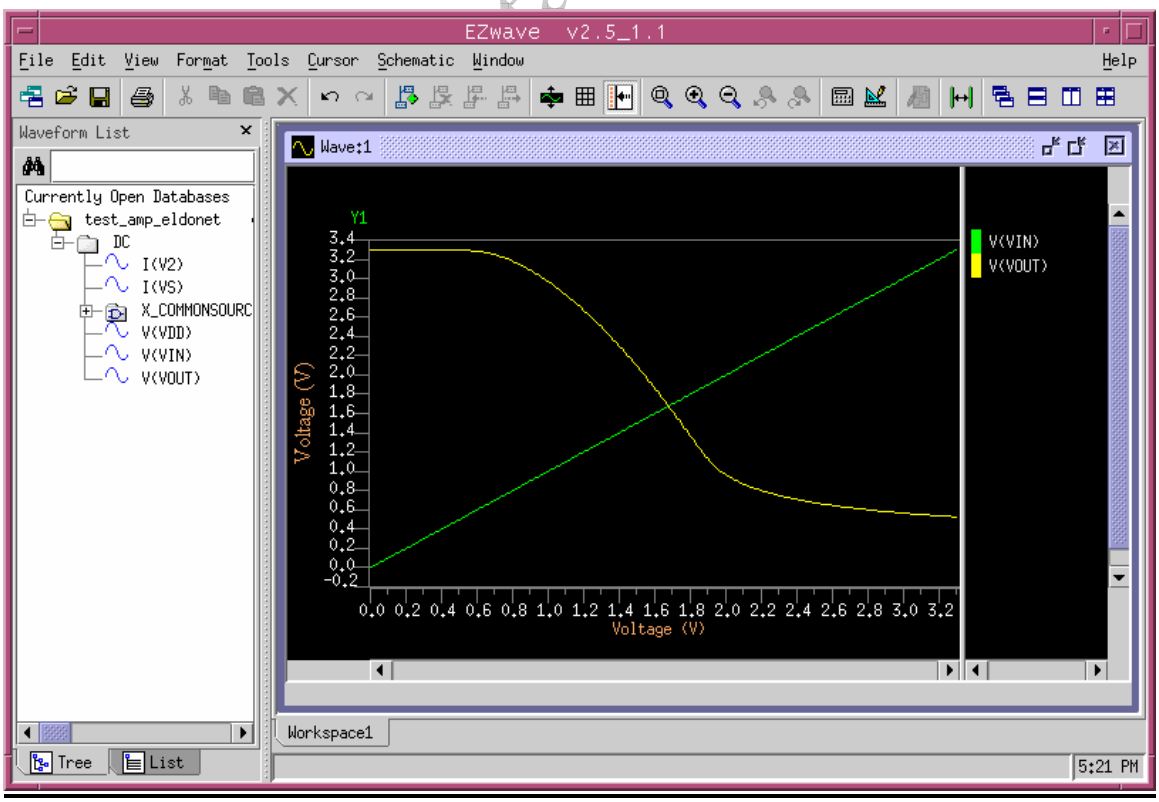
- Click **Execute > Netlist**. This opens up a new window that starts netlister. Check the netlist created for any errors. If there are any errors, correct them before proceeding further. Else close the window by pressing **Enter** key.
- Click **Execute > Run Eldo**. This starts Eldo in a new shell window and it may take few seconds before the simulation is complete. You may scroll down the window to see the DC simulation results. Press **Enter** key to close the Shell Window.

4. Viewing the results using EZWave viewer

To view the results of your DC analysis in EZWave, do the following:
On the **schematic_sim palette:**
Click **Results > View Waves**. A new window will open as shown below.



Under **test_amp_eldonet > DC > X_COMMONSOURCE,** Highlight both **V(VIN)** and **V(VOUT)**, then right click, and select **Plot (Overlaid)**

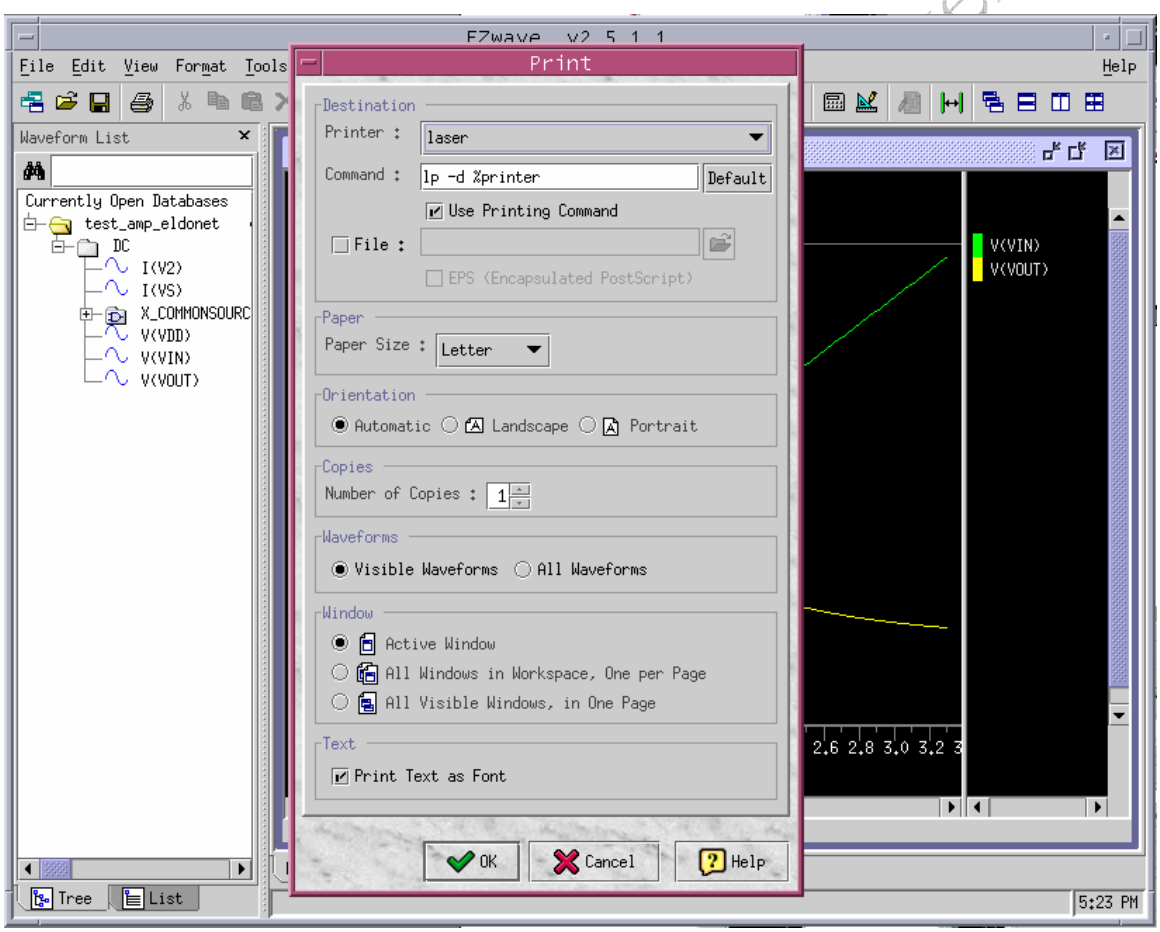


This DC analysis shows you where to set the DC bias point of the transistor. From the input/output plot, we see that the high gain region corresponds to a DC input voltage of about 1.0V to 1.8V. In the subsequent analyses, you will set a DC bias voltage of 1.5V at the gate of the transistor, ensuring that the device is operating in the the correct region.

5. Printing the plots

To Print your waveform:

- Click **File > Print**.
- Type `**laser**` for the printer name & Click **OK**.



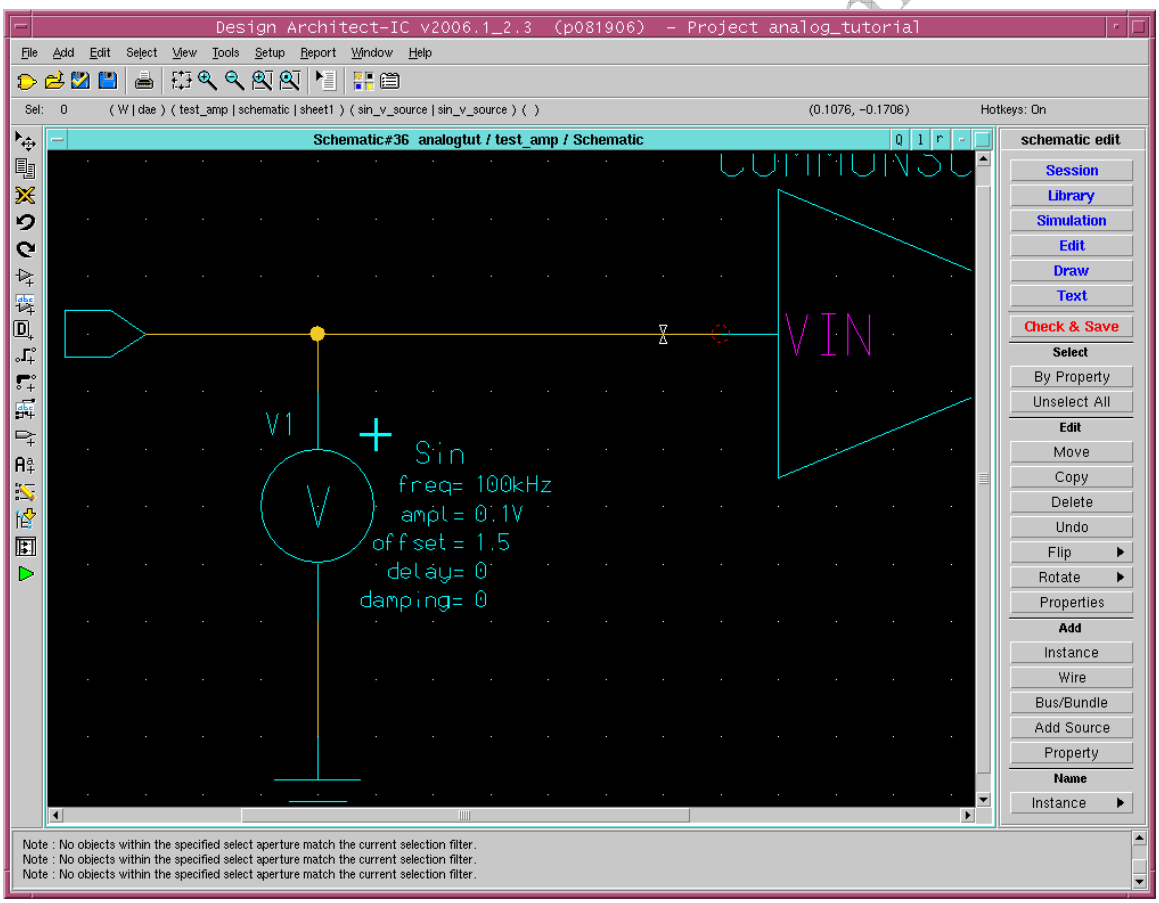
4. Transient Analysis

In this section, you will provide a sinusoid at the input and view the input & output waveforms.

1. Editing and simulating the test circuit

- Open the **test_amp** sheet that you have created with the symbol of your amplifier and make sure that you are in **schematic_edit** mode.
- Remove any existing input sources and add a **SIN** voltage source between the **VIN** pin and **Ground** as shown in the following figure.
- Name this sourse as **Vs** and change the attributes as:

ampl	0.1V	(AC amplitude of the input)
freq	100kHz	
offset	1.5V	(DC bias point, as found in the DC analysis)

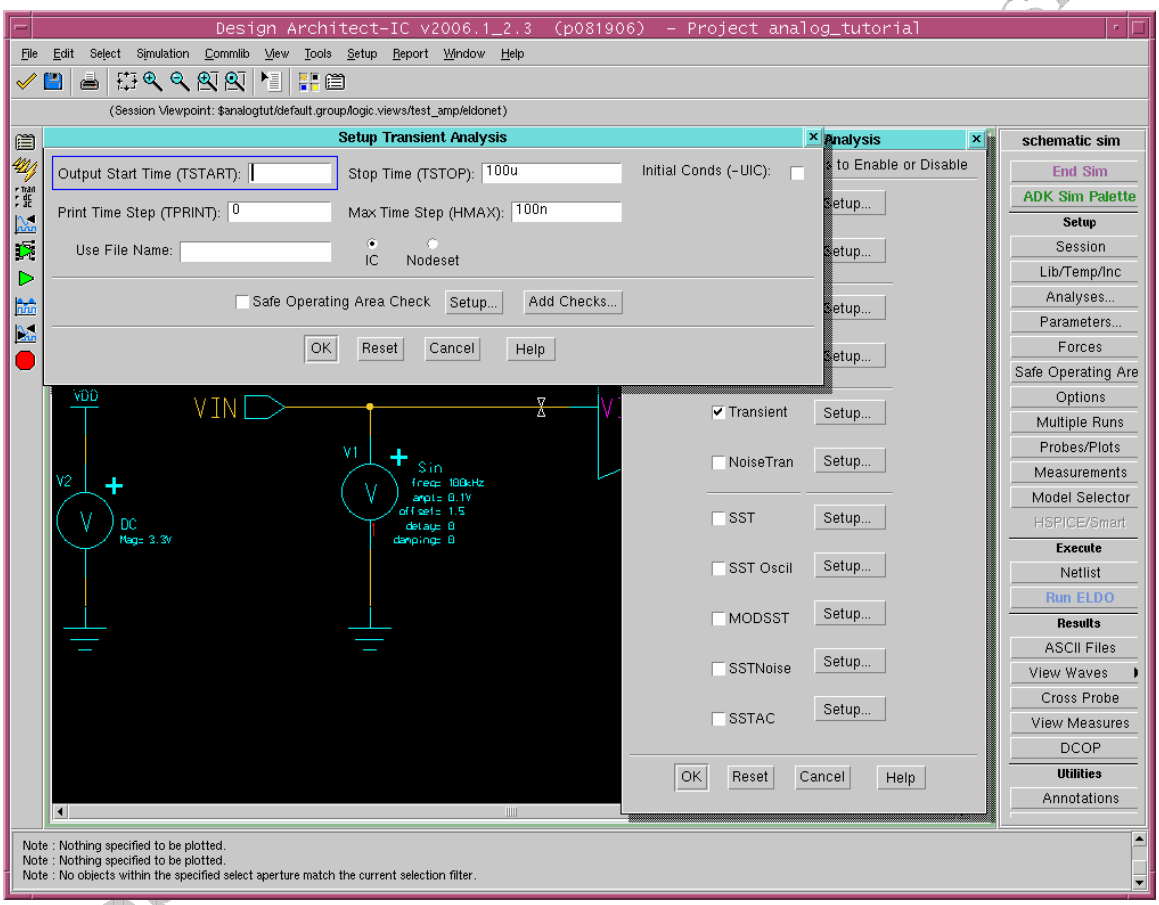


- **Check & save** the sheet and then enter the **simulation** mode by clicking on **simulation** from the **schematic_edit** palette on RHS. Click **OK** to accept default options in the Warning popup window appeared.

2. Setting up the simulation parameters

On the **schematic_sim palette:**

- Click on **Lib/Temp/Inc > Library** to make sure that **/opt/mentor-2004.3/sol/adk2_5/technology/ic/models/tsmc035.mod** appears in **library path** box and click **OK**.
- Click **Setup -> Analyses**. In the **Setup Simulation Analysis** window that appears, select **Transient** and click on **Setup** associated with it.
- In the **Setup** box that appears type **100u** in the **Stop Time** field and **100n** in **Max Time Step** field. Then click **OK**.



3. Setting up the signals to be probed

- Click **Probes/Plots > Save All…**from the **Palette**. This opens a dialog box for choosing which signals should be saved for later analysis.
- Make sure the **Voltages** and **Currents** boxes are checked, and click **OK**.

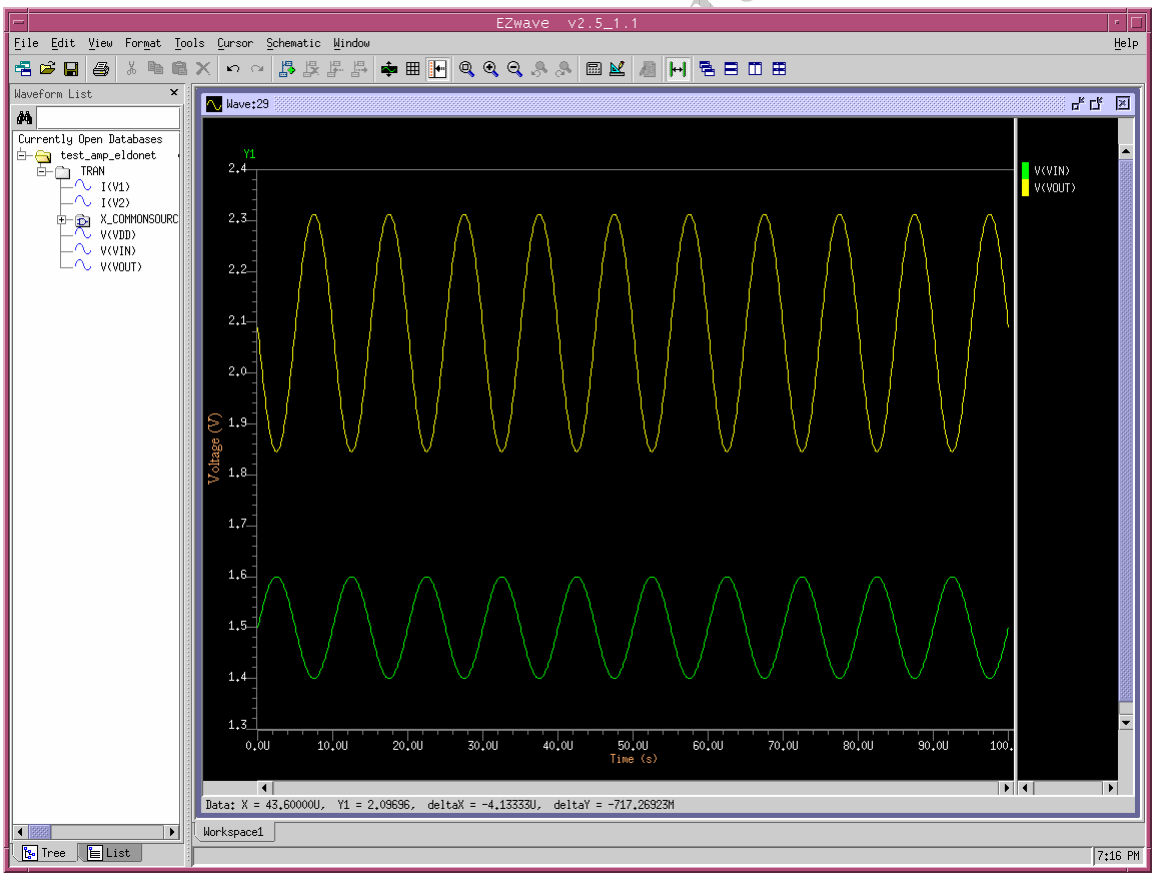
4. Executing the simulation setup

- Click **Execute > Netlist**
- Click **Execute > Run ELDO**

5. Viewing the results using EZWave viewer

To view the results of your transient analysis in EZWave, do the following:
On the **schematic_sim palette:**

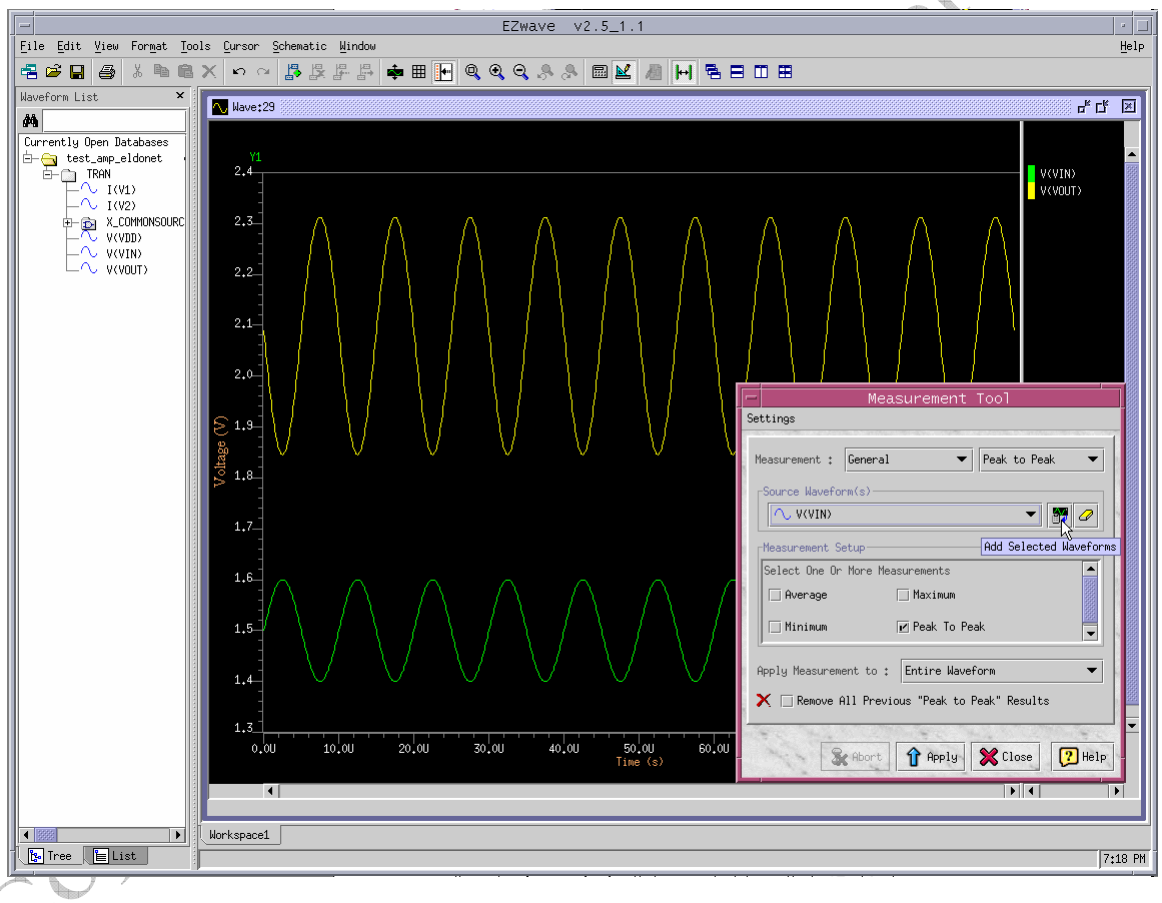
- Click **Results > View Waves**. A new window will open as shown below.
- Under **test_amp_eldonet > TRAN > X_COMMONSOURCE :**
    - Highlight both **V(VIN)** and **V(VOUT)**, then right click, and select : **Plot (Overlaid)**

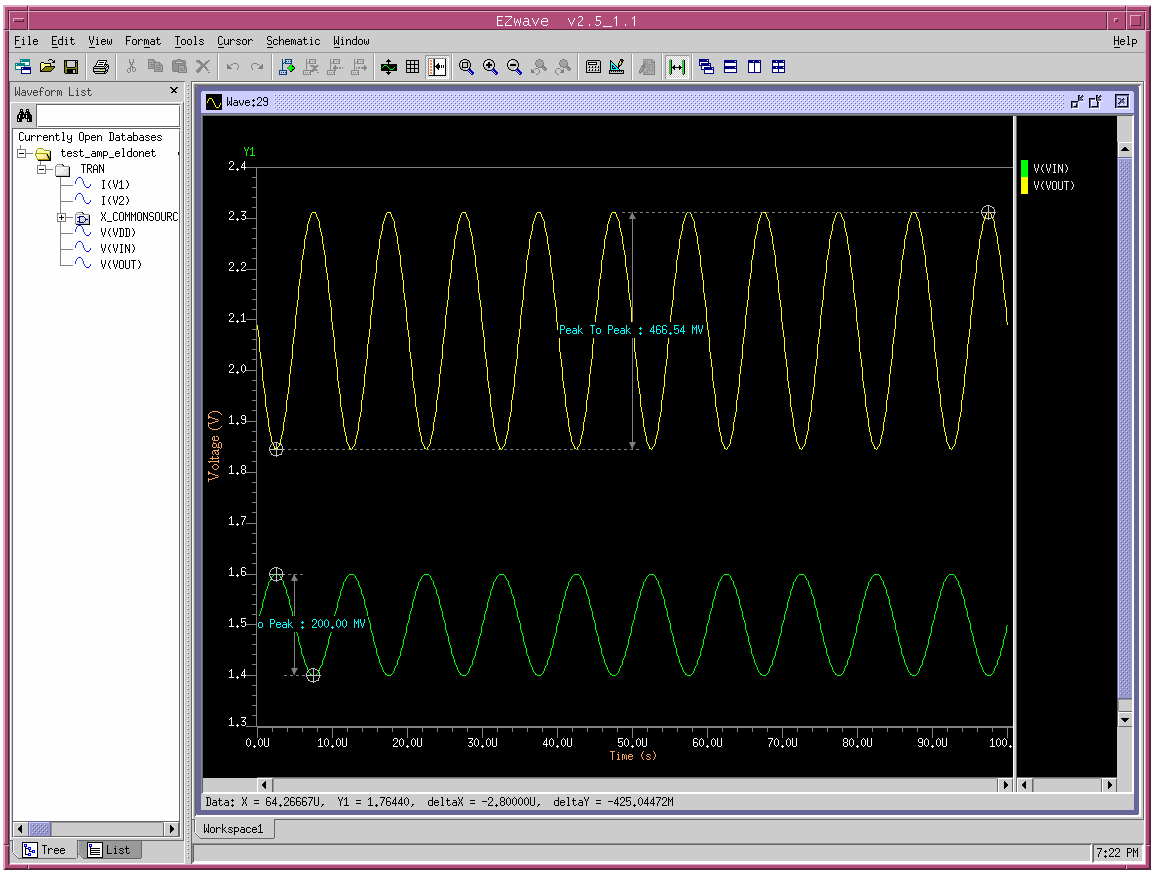


6. Measuring Gain with the EZWave Measurement Tool

To measure the gain of the circuit from the transient analysis in EZWave, do the following:

- Select **VOUT** by clicking the yellow trace.
- From the menu, select **Tools > Measurement Tool**
- Click the **Add Selected Waveform** button
- Select **V(VOUT)** as the **Source Waveform**
- Select **General > Peak-to-Peak** as the Measurement
- Click **Apply**
- Repeat for the **VIN** waveform





Many other measurements can be made with the EZWave Measurement Tool.

5. DC Operating Point Analysis

In this section, you will find the DC operating point of several nodes in the circuit. The input of the amplifier should be configured as in the above transient analysis.

1. Editing and simulating the test circuit

- Open the **test_amp** sheet that you have created with the symbol of your amplifier and make sure that you are in **schematic_edit** mode.
- Remove any existing input sources, and add a **SIN** voltage source between the **VIN** pin and **Ground** as shown in the following figure.
- Name this sourse as **Vs** and change the attributes as:

ampl	0.1V	(AC amplitude of the input)
freq	100kHz	
offset	1.5V	(DC bias point, as found in the DC analysis)

2. Setting up the simulation parameters

On the **schematic_sim palette:**

- Click **Setup > Analyses.** Select **DCOP** in the dialog box that appears, and then click **OK.**

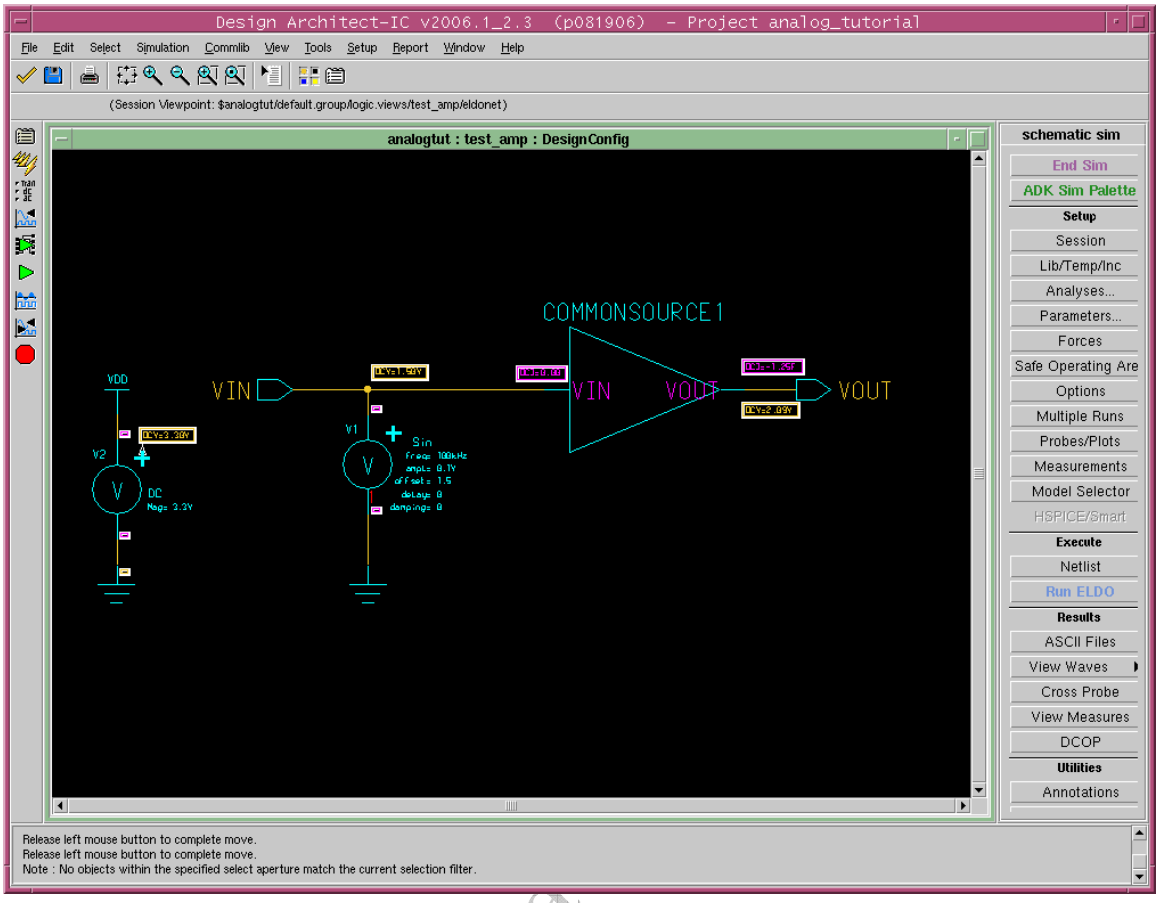
3. Executing the simulation setup

- Click **Execute > Netlist**
- Click **Execute > Run ELDO**

4. Viewing the results using Monitors

The next step is to view the results of your simulation run.
To see all the DC voltages and currents in your schematic,

- Click **Results > DCOP > Add monitor > All Nets – Voltages**
- Again click **DCOP > Add monitor > All Pins - Current**



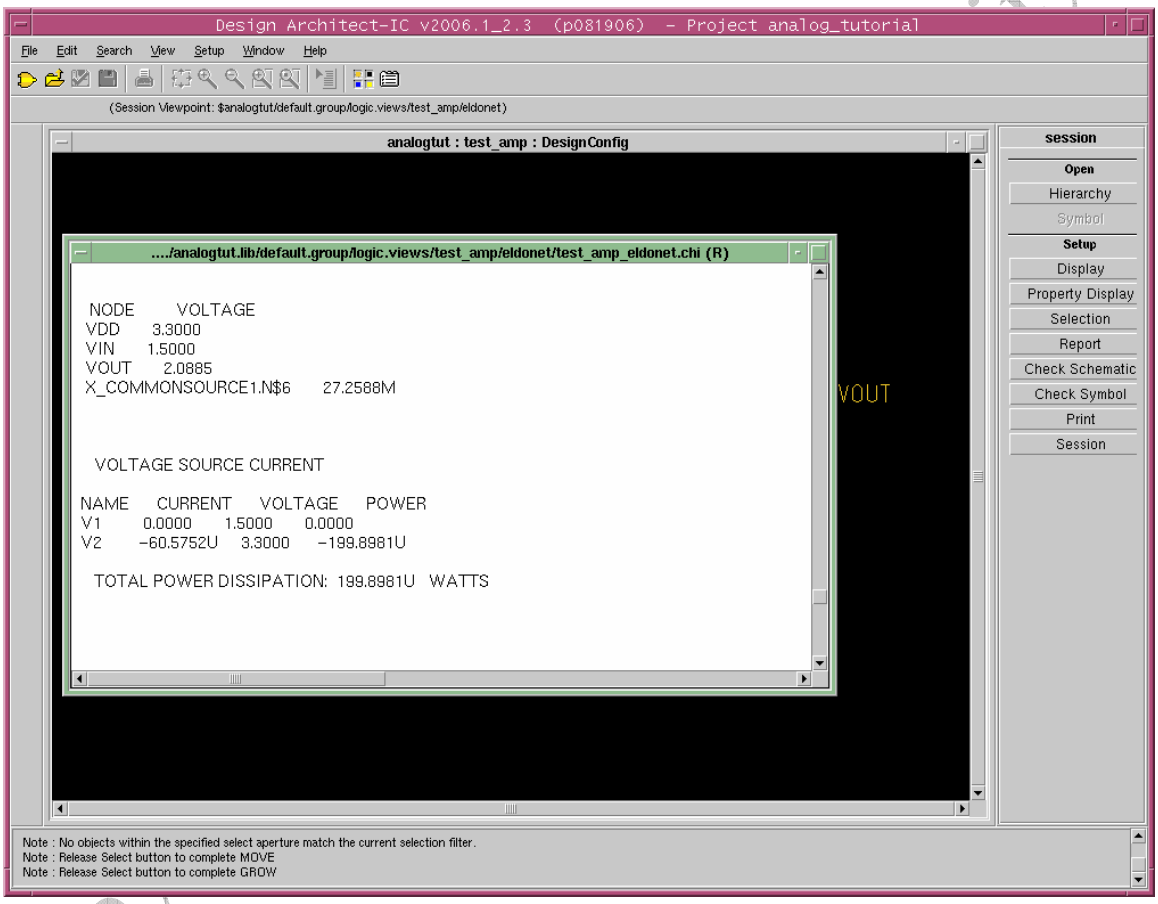
Current and Voltage values at various nodes will be displayed as shown in the above figure.

Alternately, you can select voltages or current option, if you want to see only voltages or currents.

5. Static Power consumption of the circuit

The static (DC) power consumption of the circuit for the given operating point is calculated during the DCOP simulation.

- Click on **Results > ASCII Files, right click mouse button > View Log**.
- Scroll down in the log window to find the information about the power consumption as shown in the figure below:



6. AC Analysis

In this section, you will run AC analysis and trace and measure your results. In AC Analysis, the input signal is swept over a range of frequencies.

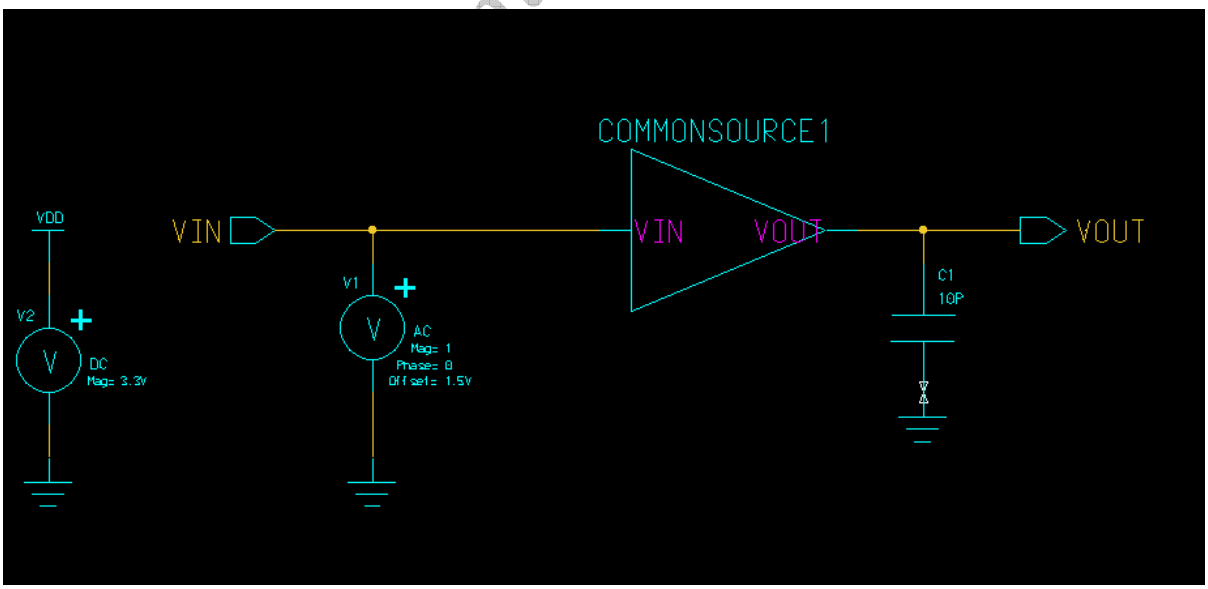
1. Editing and simulating the test circuit

- Open the **test_amp** sheet that you have created with the symbol of your amplifier and make sure that you are in **schematic_edit** mode.
- Remove any existing input sources, and add an **AC** voltage source between the **VIN** pin and **Ground**
- Name this sourse as **Vs** and change the attributes as:

mag	1V	(AC amplitude of the input)
offset	1.5V	(DC bias point, as found in the DC analysis)

Note: While 1V can not be considered a small signal input for this circuit, it is acceptable in the AC simulation since SPICE does not use a large-signal model for the AC simulation. SPICE first runs a DC operating point analysis to construct the small signal model, then uses the specified AC voltage in the linear model. By setting the input to 1V, the output voltage equals the gain directly. (A_V = Vout/Vin)

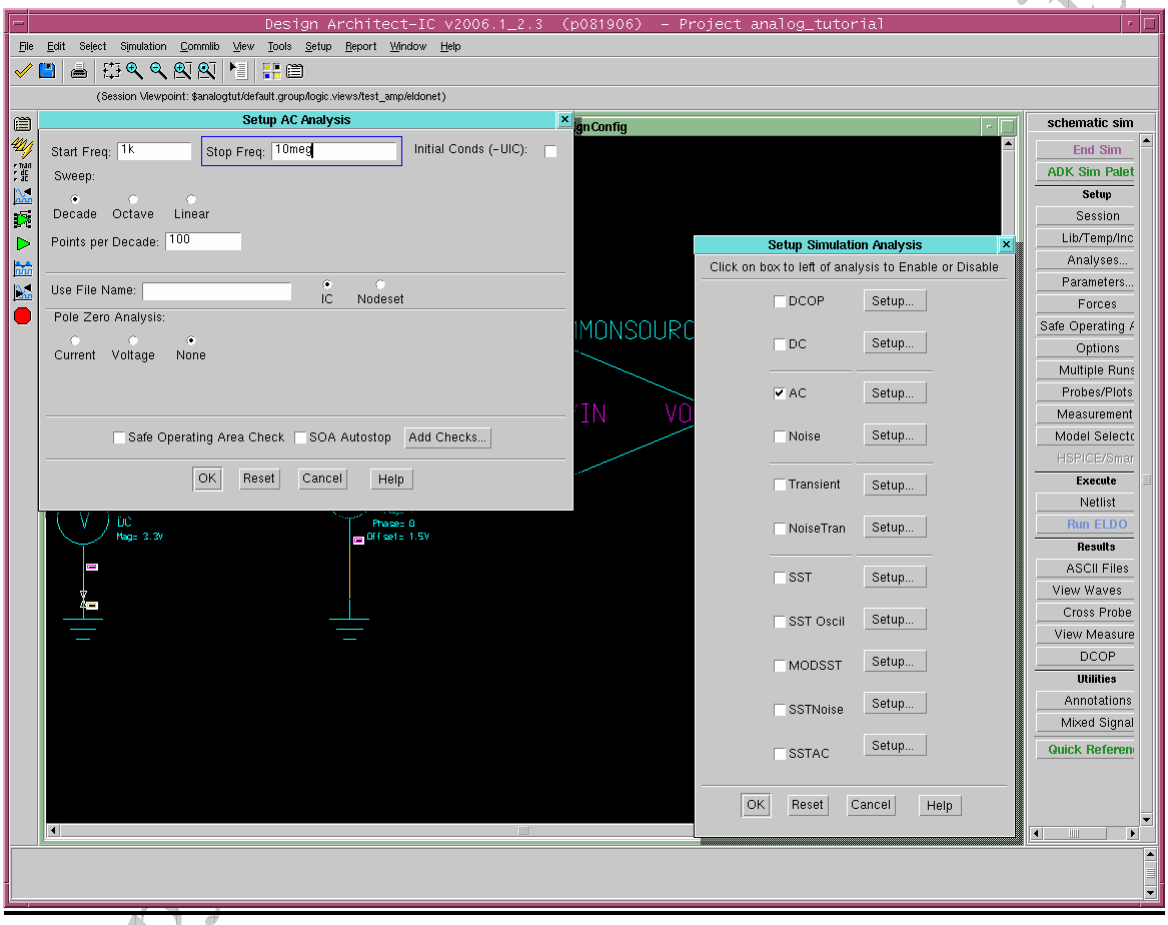
- Add a 10pF capacitor at the output of the amplifier as shown below
- Click **Check & save** the sheet and then enter the **simulation** mode by clicking on **simulation** from the **schematic_edit** palette on RHS. Click **OK** to accept default options in the Warning popup window appeared.



2. Setting up the simulation parameters

On the **schematic_sim palette:**

- Click **Setup > Analysis.** Check **AC** and click on the **Setup** box associated with it. Select the **Decade** button for **Sweep type.** For this circuit you will perform an AC sweep of input frequency from **1k** to **10MEG**. Enter 100 in the **Points per Decade** box. Enter these values in the **Start Freq** and **Stop Freq** respectively. Click **OK**



3. Setting up the signals to be probed

- Click **Probes/Plots > Save All…**from the **Palette**. This opens a dialog box for choosing which signals should be saved for later analysis.
- Make sure the **Voltages** and **Currents** boxes are checked, and click **OK**.

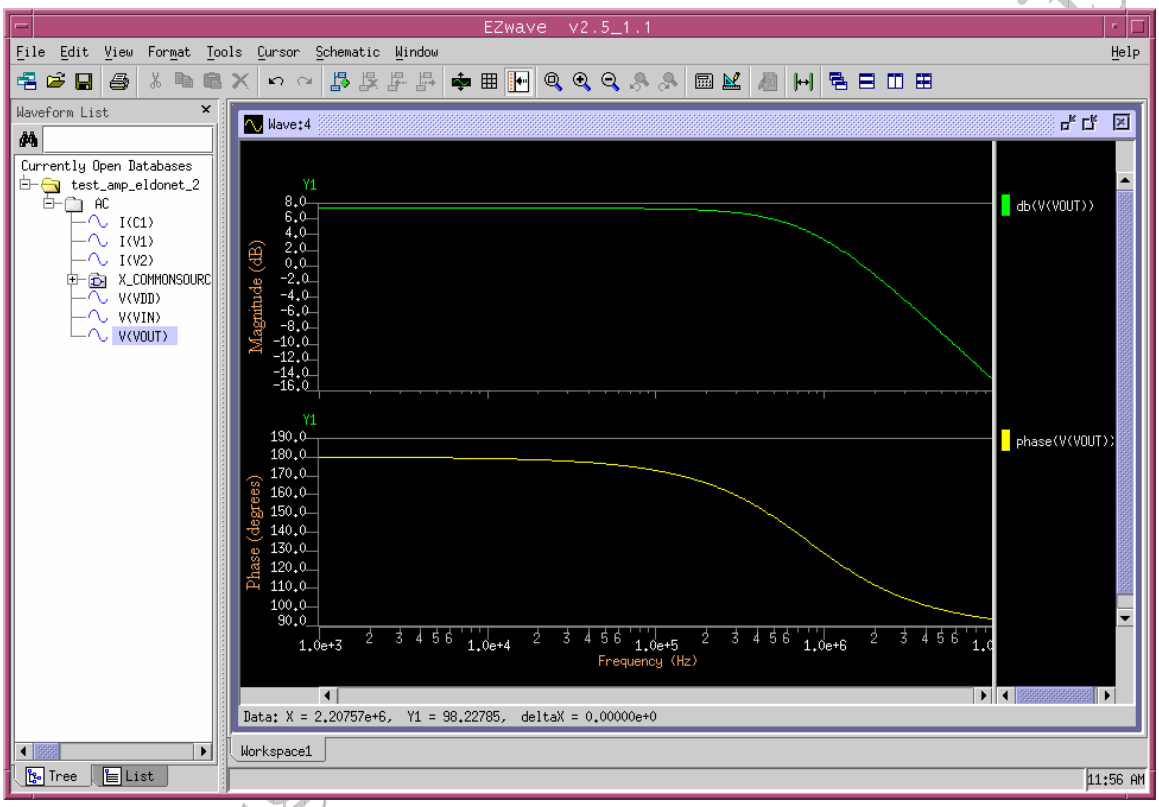
4. Executing the simulation setup

- Click **Execute > Netlist**
- Click **Execute > Run ELDO**

5. *Viewing the results using EZWave viewer*

To view the results of your transient analysis in EZWave, do the following:
On the **schematic_sim palette:**

- Click **Results > View Waves**. A new window will open as shown below.
- Under **test_amp_eldonet > TRAN > X_COMMONSOURCE :**
- Highlight **V(VOUT)**, then right click, and select : **Plot**

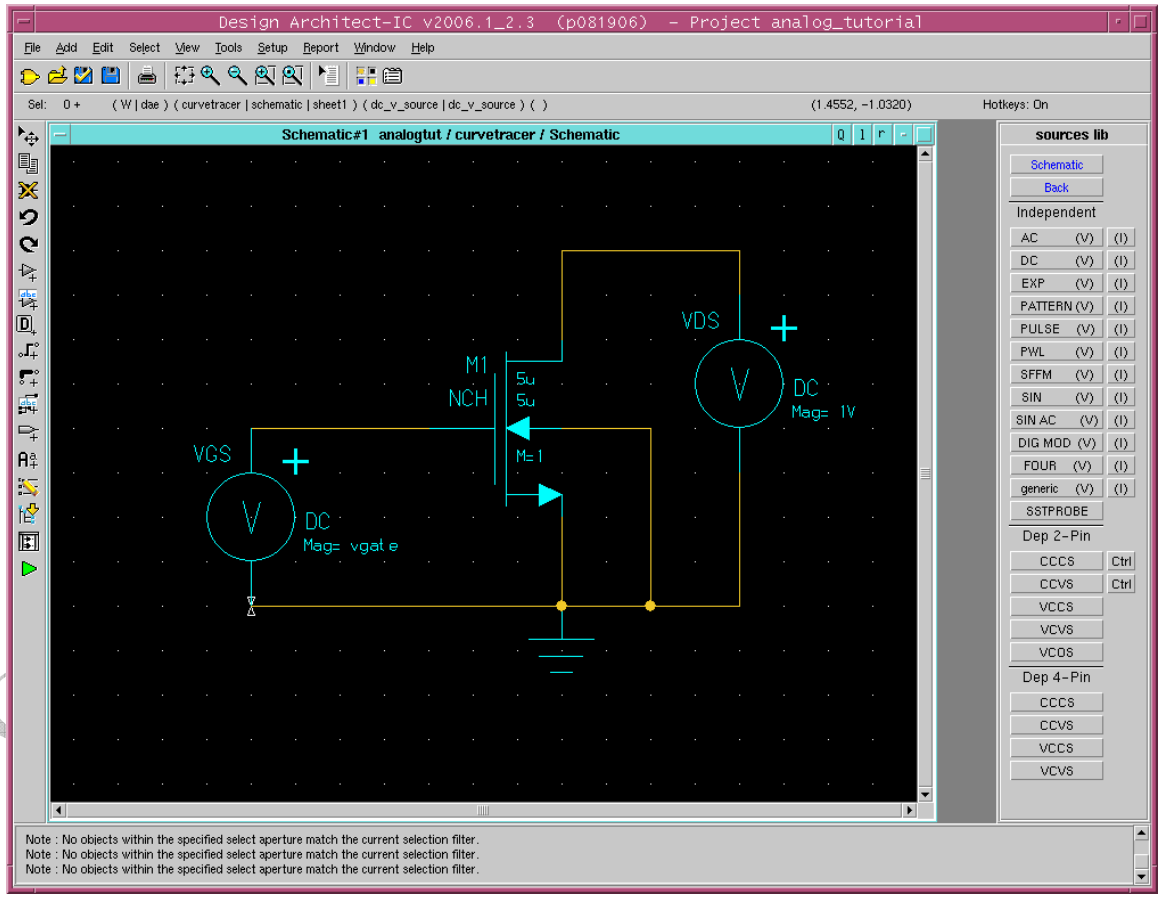


7. Parameter Sweeps

It is often helpful to perform an analysis on one parameter while simultaneously sweeping another parameter. This concept will be demonstrated by generating the ID vs VDS curves for a MOSFET.

1. *Editing and simulating the test circuit*

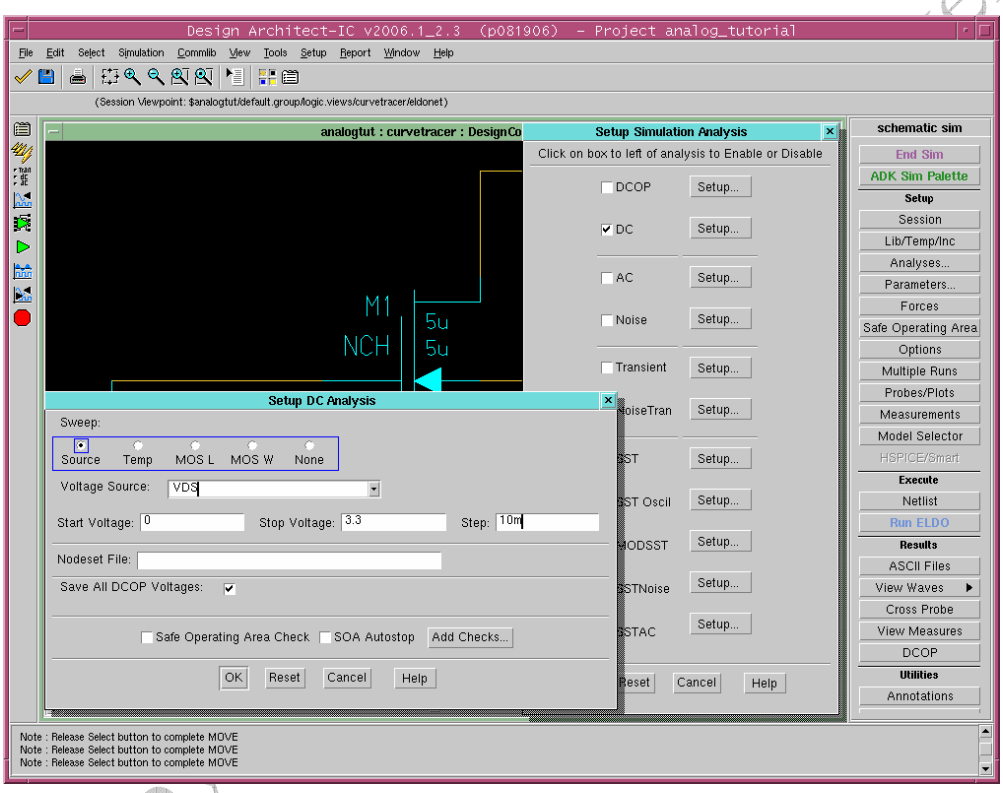
- Create a new schematic view and draw the circuit shown below. (Name the gate voltage source **VGS** and the drain voltage source **VDS**. Both sources are **DC**)
- For **VGS**, set **Mag = vgate**
- Click **Check & save** the sheet and then enter the **simulation** mode by clicking on **simulation** from the **schematic_edit** palette on RHS. Click **OK** to accept default options in the Warning popup window appeared.



2. Setting up the simulation parameters

On the **schematic_sim palette:**

- Click on **Lib/Temp/Inc > Library** to make sure that **/opt/mentor-2004.3/sol/adk2_5/technology/ic/models/tsmc035.mod** appears in **library path** box and click **OK**.
- Click **Setup > Analyses**.
- In the **Setup Simulation Analysis** window that appears, select **DC** and click on **Setup** associated with **DC**. In the **Setup** box that appears, select **Source** and select **VDS** for **Voltage source**, put **0** in the **start field**, **3.3** in the **stop** field and **10m** in the **step** field and then click **OK**.

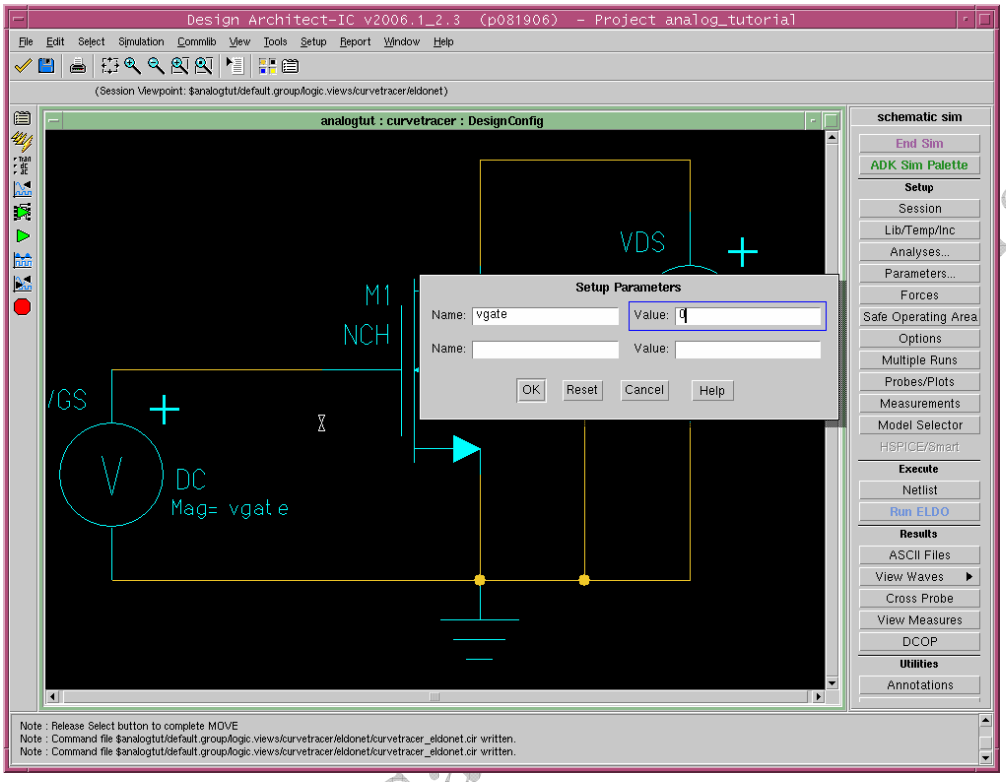


- Click **Probes/Plots > Save All…**from the **Palette**. This opens a dialog box for choosing which signals should be saved for later analysis.
- Make sure the **Voltages** and **Currents** boxes are checked, and click **OK**.

3. Defining Parameters

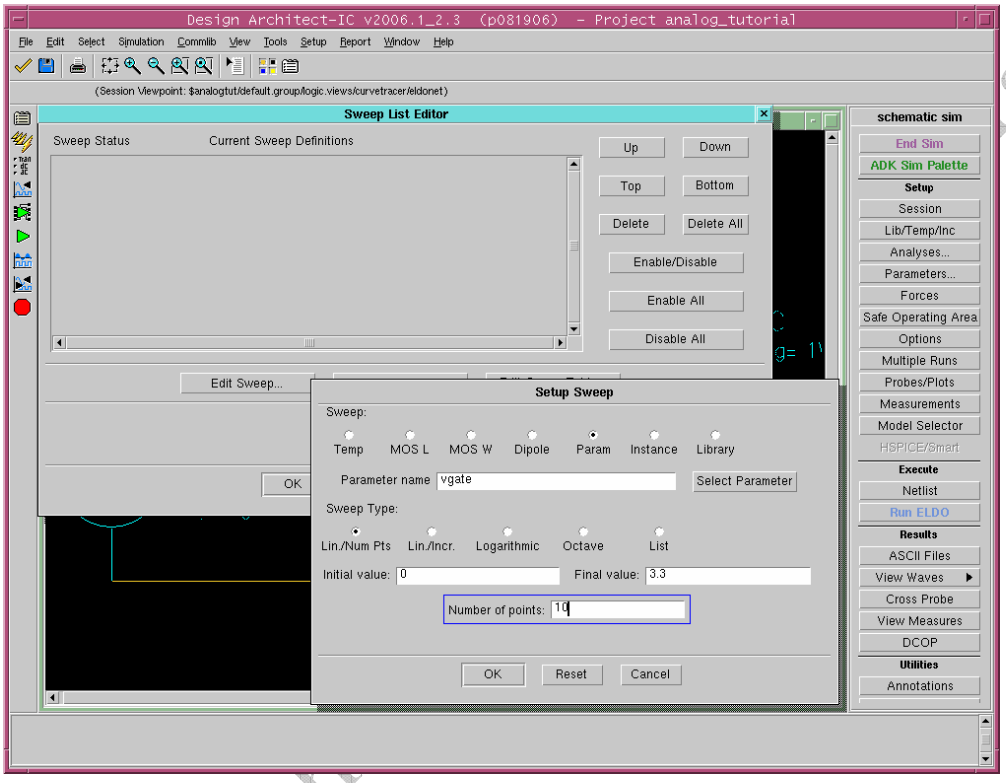
- On the simulation palette, click **Parameters**, and enter into the first empty box:

Vgate 0



4. Setting up Multiple Runs

- Click **Multiple Runs > Sweep** to bring up the Sweep List Editor
- Under **Multi-Run Mode**, select **Sweep**
- Click **Add Sweep**
- Select **Param**, and choose **vgate** as the Parameter name
- Setup the sweep for **Lin/Num Pts**, **0** to **3.3**, with about **15** steps. Click **OK**



5. Executing the simulation setup

On the **schematic_sim palette:**

- Click **Execute > Netlist**.
- Click **Execute > Run Eldo**

6. Viewing the results using EZWave viewer

To view the results of your DC analysis in EZWave, do the following:
On the **schematic_sim palette:**

- Click **Results > View Waves**. A new window will open as shown below.

Under **curvetracer_eldonet > DC > M1 :**

- Highlight **I(D)** then right click, and select **Plot**