Virtuoso 23.1
Module 3 – Basic Simulations

American University of Beirut
Lebanon

Contents

1. Creating the Test Bench using ADE Explorer
   a) Defining the Variables
   b) Defining the Analyses: transient, dc, ac
   c) Defining the Outputs
   d) Defining the Model Library
2. Running the Single Point Simulation
3. Running the AC Simulation

Module Objective

In this module, we will learn how to:

- set up different types of analysis using ADE Explorer
- add outputs to be plotted
- determine the small-signal voltage gain using transient analysis
- write the expression for the voltage gain using the tool Calculator
- determine the small-signal voltage gain using ac analysis

1. Creating the Test Bench using ADE Explorer

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

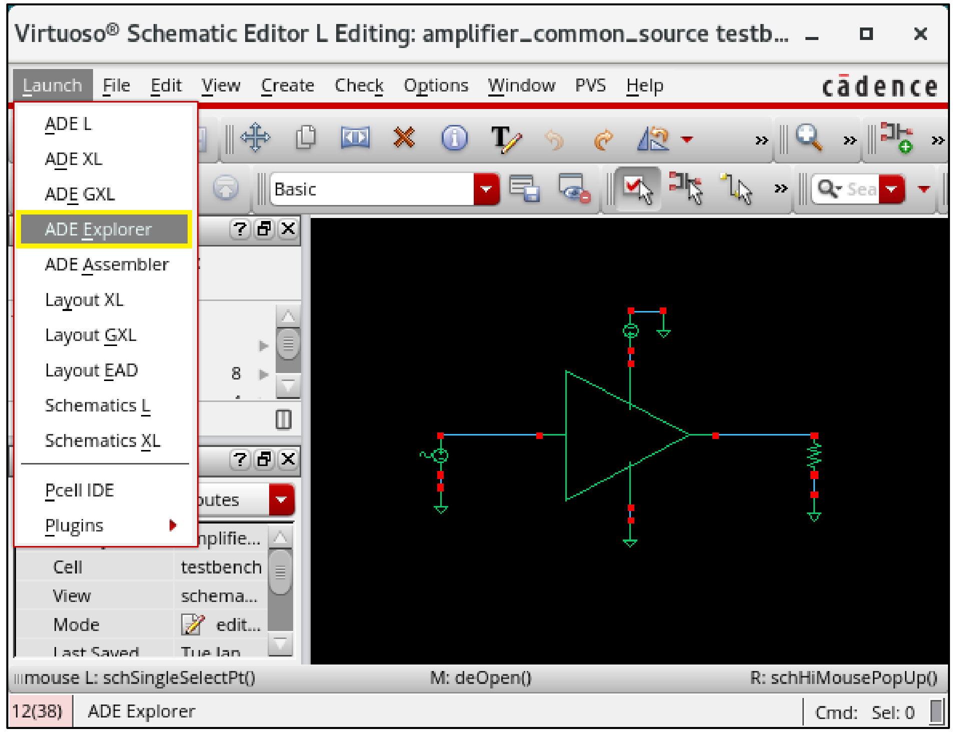
Defining the Model Library

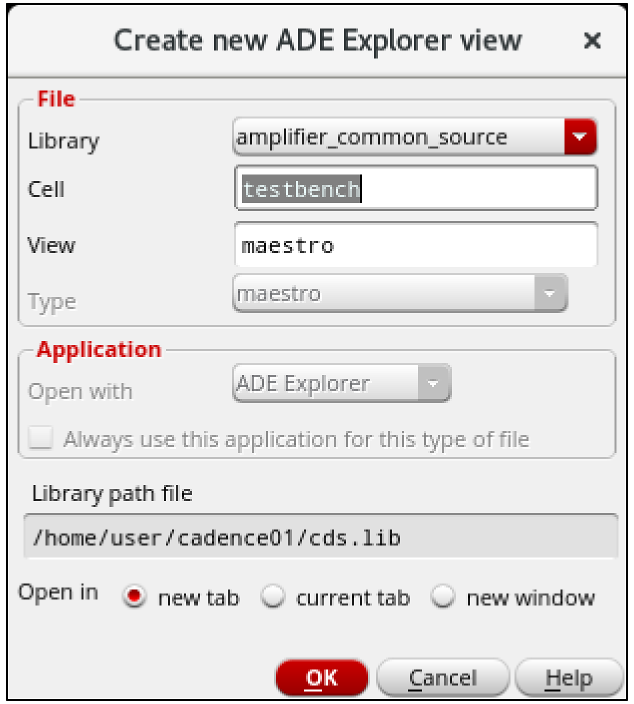
Single Point Simulation

AC Simulation

1. Creating the Test Bench using ADE Explorer

- Open the schematic view of the “testbench” cell that was created in Module 2.
- Launch ADE Explorer from the Schematic Editor (Launch → ADE Explorer).
- Select **Create New View** and click OK.
- The ADE Explorer Editing window opens. This is the environment where we will run our simulations.





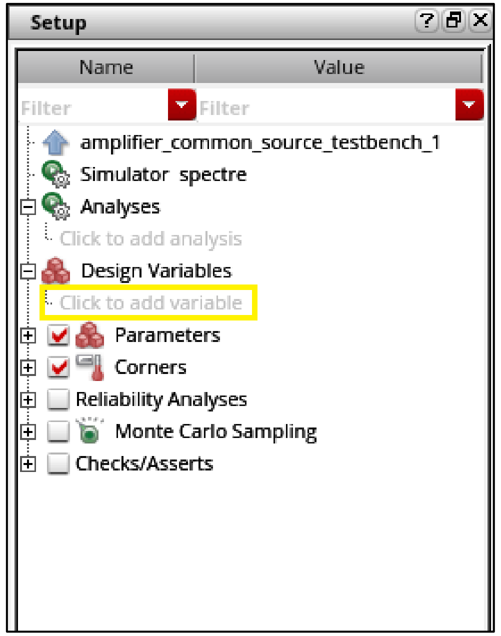
- The simulator is the engine of the tool.
- The simulator requires parameters to run.

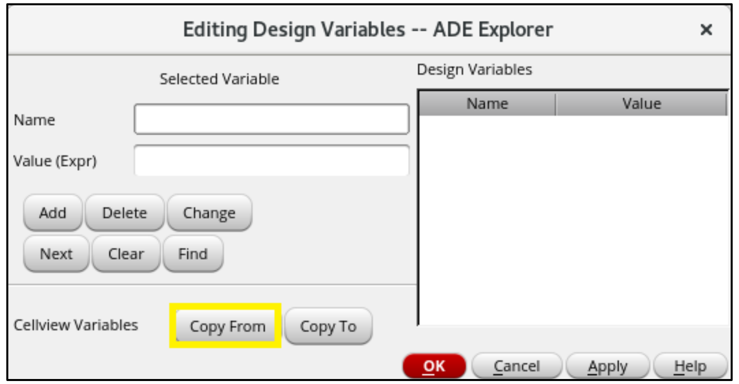
AMERICAN UNIVERSITY OF BEIRUT

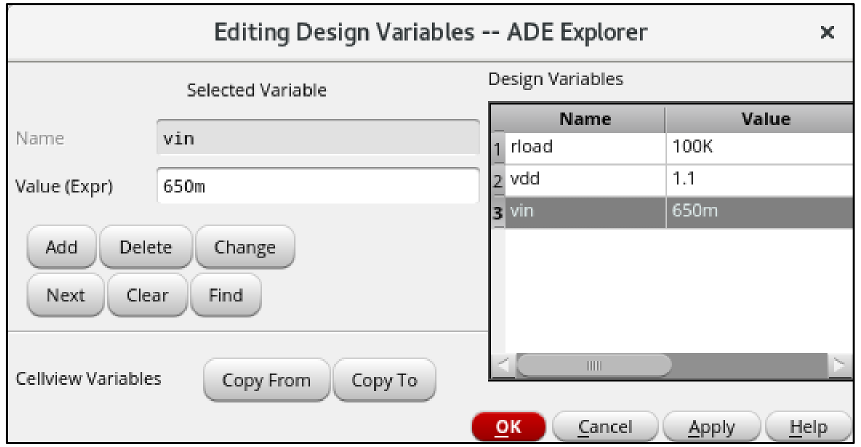
- ADE Explorer
- **Defining the Variables**
- Defining the Analyses
- Defining the Outputs
- Defining the Model Library
- Single Point Simulation
- AC Simulation

1.a. Defining the Variables

- The “Design Variables” is at the left side of the window, click on **Click to Add Variables**, then select **Copy From**.
- Set the values to each of your design variables as shown below by typing the value and clicking **Change** afterwards.
- Set vin to 650 mV, vdd to 1.1 V, and rload to 100 KΩ.







- Design variables allow you to change the parameters from the simulator without changing the schematic.
- Design variables are useful when the same parameter needs to be changed in several components.

cādence®

AMERICAN UNIVERSITY OF BEIRUT

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation

1.b. Defining the Analyses: transient, dc, ac

- In this example, we will run transient, dc, and ac analysis. Spectre is the simulator performing the analyses.
- Spectre creates a set of equations, which is a result of performing a KVL and a KCL on your linearized circuit.
- The simulator solves this set of equations by direct matrix solution.
- The solution of this matrix is compared to device model equations, if the two solutions are far from each other, it iterates.
- It keeps on iterating, until the solutions are close enough to each other - this is defined by the accuracy setup of the simulation.

- Transient analysis is the most fundamental analysis where you can see your signals versus time.

AMERICAN UNIVERSITY OF BEIRUT

ADE Explorer
Defining the Variables
Defining the Analyses
Defining the Outputs
Defining the Model Library
Single Point Simulation
AC Simulation

1.b. Defining the Analyses: transient, dc, ac *(continued)*

- You can set the accuracy (known internally as the "errpreset Parameter") to one of the following:
  - Conservative: The simulation runs at its slowest pace; however, it is the most accurate one (usually employed for sensitive analog circuits).
  - Moderate: The simulation runs at a moderate pace and moderate accuracy.
  - Liberal: The simulation runs at its fastest pace, but it is less accurate (usually employed for digital and analog circuits that have only short time constants).

- Therefore, when choosing the errpreset setup of the analysis there is always a trade-off between speed and accuracy; hence, the choice depends on the preference of the designer and the size of the circuit.

- If the design is large, it is a good practice to start with "Moderate" and then move to "Conservative" when the design is more mature.

cadence

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

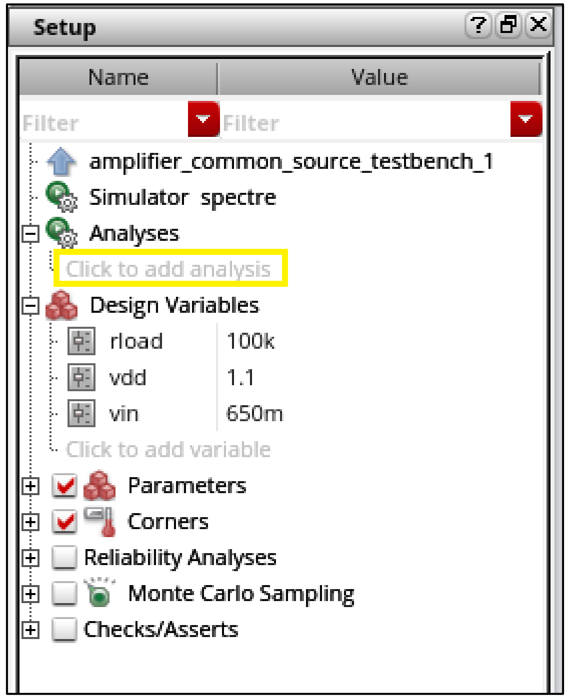
Defining the Model Library

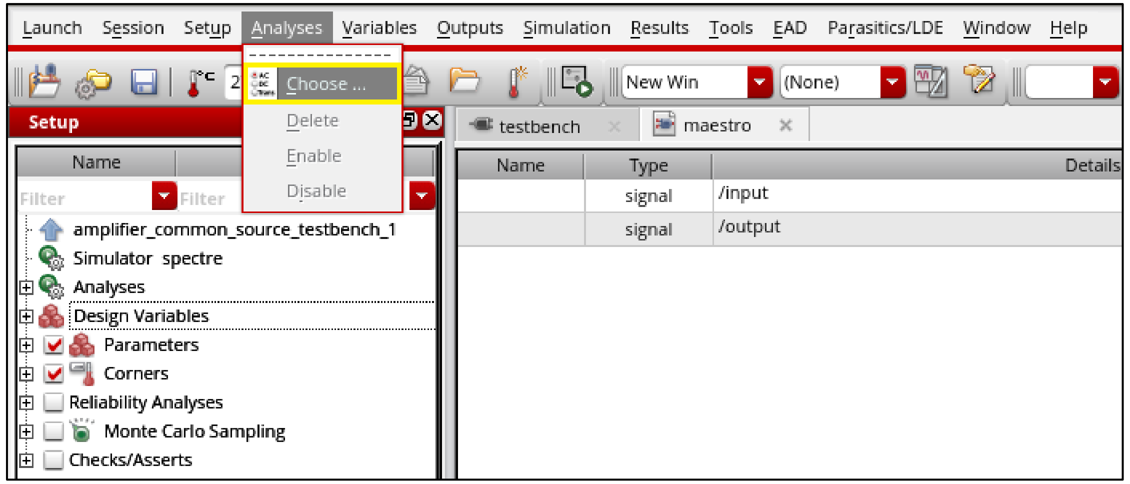
Single Point Simulation

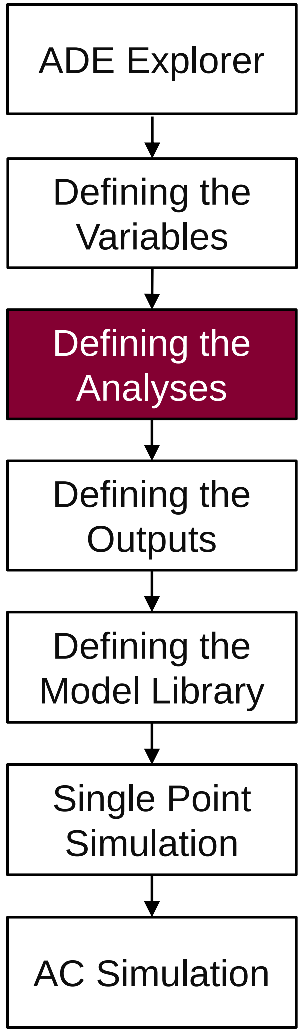
AC Simulation

1.b. Defining the Analyses: transient, dc, ac *(continued)*

- Before running your simulation, choose the type of analysis you want to perform on your test bench, to do so, select **Analyses → Choose...** (or simply click on **Click to add analysis**).

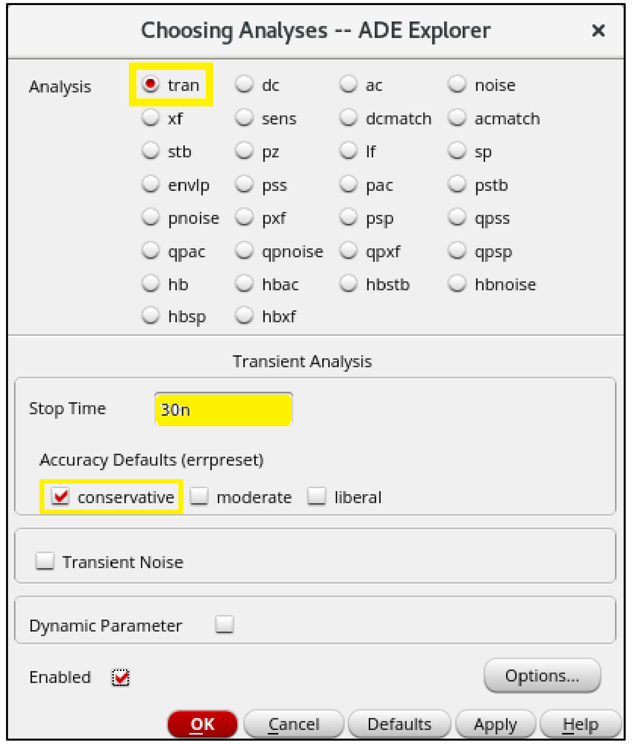


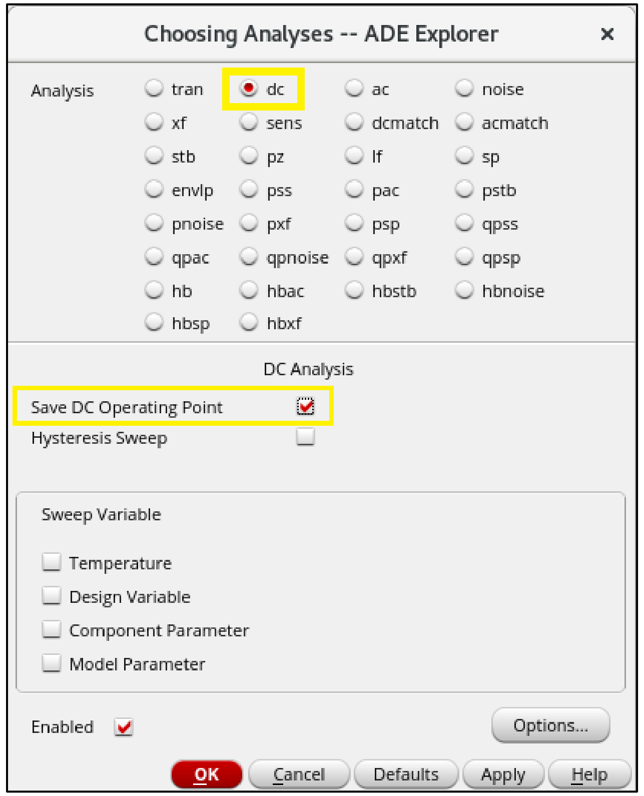


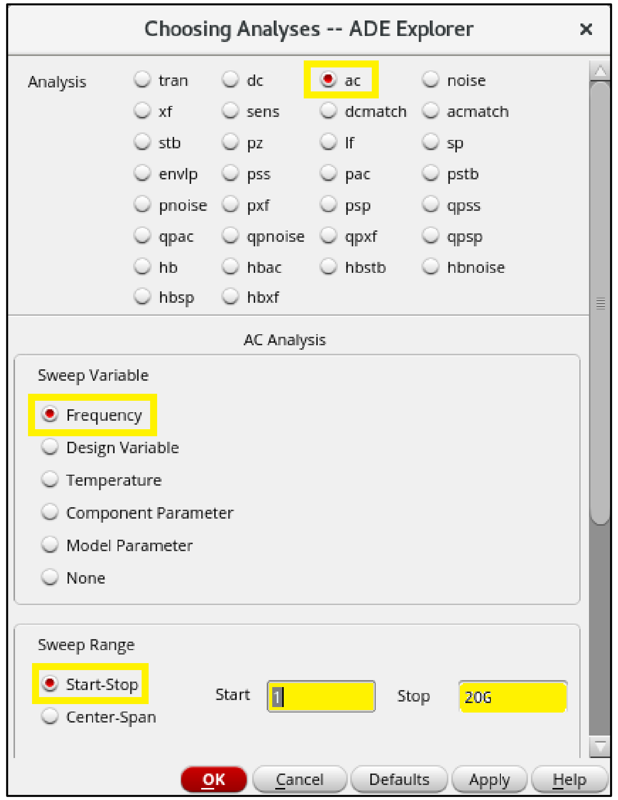


1.b. Defining the Analyses: transient, dc, ac *(continued)*

- First, to set up the transient analysis select **tran**, a “Stop Time” of 30ns, and a **conservative** accuracy (to be more accurate in the analysis).
- Second, to set up the dc analysis, simply select **dc** and check **Save DC Operating Point**.
- Finally, to set up the ac analysis, simply choose frequency and set the sweep from 1 to 20G Hz.



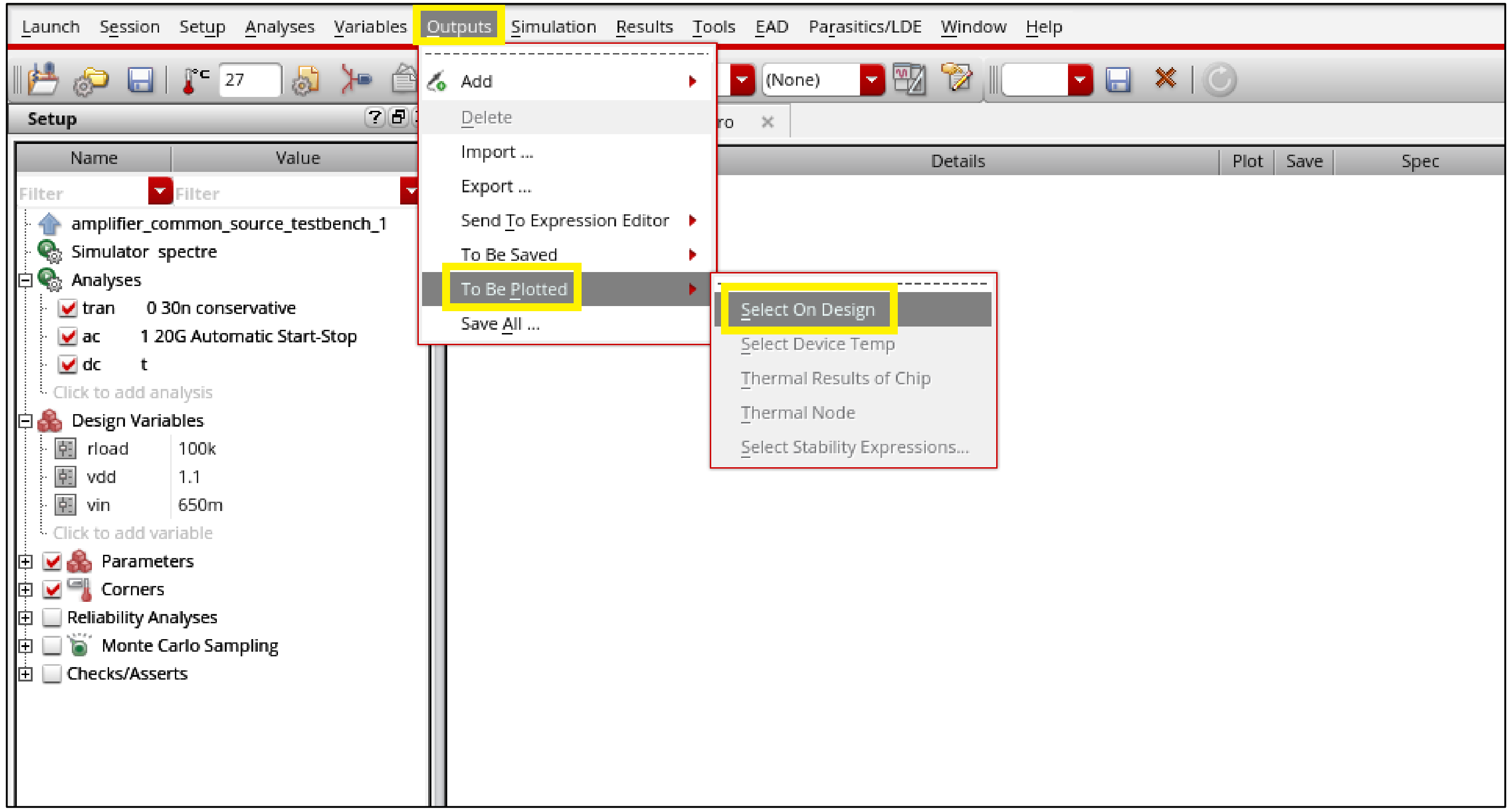




- The analyses are chosen in this step.

1.c. Defining the Outputs

- In order to define the outputs of the simulation select: **Outputs → To Be Plotted → Select on Design**.



- Having only the needed outputs conserves storage space which is important with large designs.

cadence

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

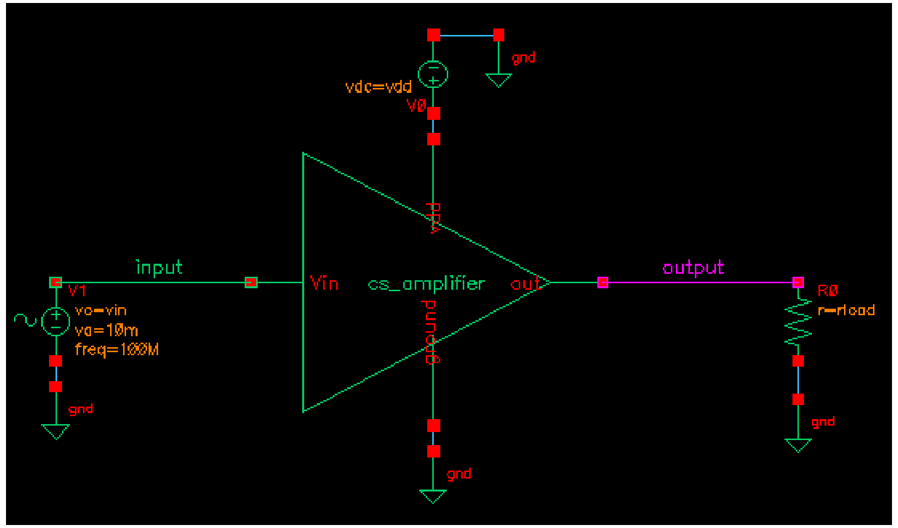
Defining the Model Library

Single Point Simulation

AC Simulation

1.c. Defining the Outputs *(continued)*

- Select each of the nodes corresponding to your output and input (to choose the outputs to be plotted), to stop selecting, press ‘Esc’.
- Note that selecting a red square will correspond to the current going into the component.
- As shown in the figure below, each net is now highlighted in a certain color (purple and green in our case), this will be the color of each plot when the simulation is run.



- It is a good practice to always display the input along with the output to make sure that the design is being driven by the intended waveform.

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

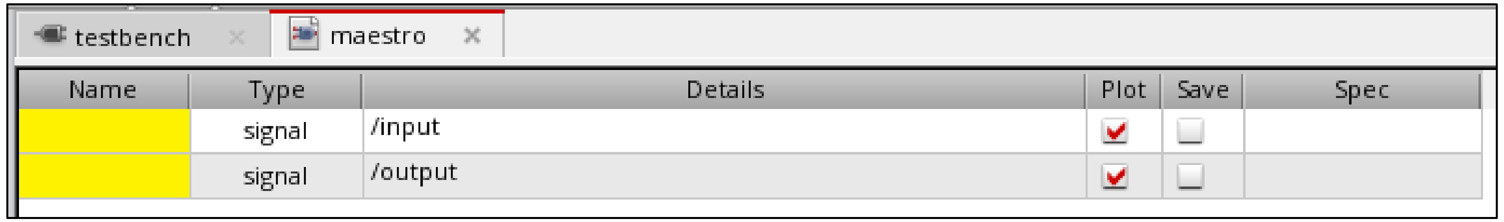
Defining the Model Library

Single Point Simulation

AC Simulation

1.c. Defining the Outputs *(continued)*

- You can also give a meaningful name to each of your outputs as shown below (this is useful when you have many outputs to be plotted to avoid confusion).
- We will skip this step, since we already named the connections “input” and “output” previously.



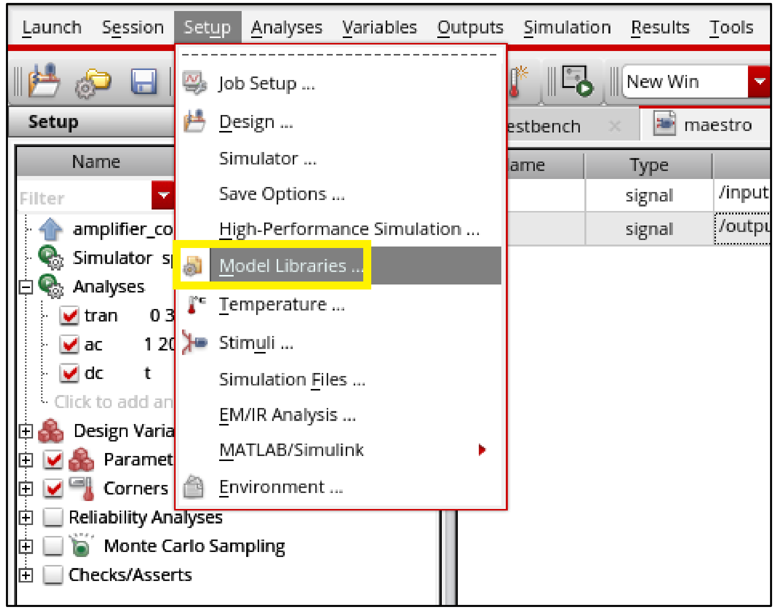
Name	Type	Details	Plot	Save	Spec
	signal	/input	☑	☐	
	signal	/output	☑	☐	

- Naming your outputs will help you identify the plot that represents a specific output.

cadence

1.d. Defining the Model Library

- You still need to add the model files (these files define the parameters of the components used in your circuit, thus implicate your circuit behavior).
- Select **Setup → Model Libraries…**



- The model libraries need to be set.

cādence®

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

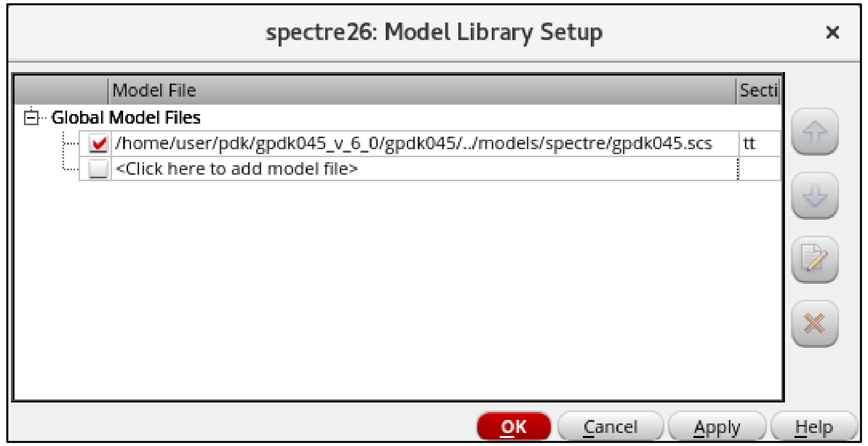
Defining the Model Library

Single Point Simulation

AC Simulation

1.d. Defining the Model Library *(continued)*

- The tool may have already defined the Model Library.
- If this is the case, the file “gpdk045.scs” will be selected as shown below.
- Click on the **Section** field and **select** “tt” from the drop-down list. It stands for “Typical NMOS Typical PMOS” process conditions.
- Click **OK** to save your “Model Library Setup”.



- **If the tool hasn’t defined the Model Library, refer to slide 26 of Module_02.**

- Spectre files have the “scs” extension. They can be read using a text editor.

2. Running the Single Point Simulation

AMERICAN UNIVERSITY OF BEIRUT

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

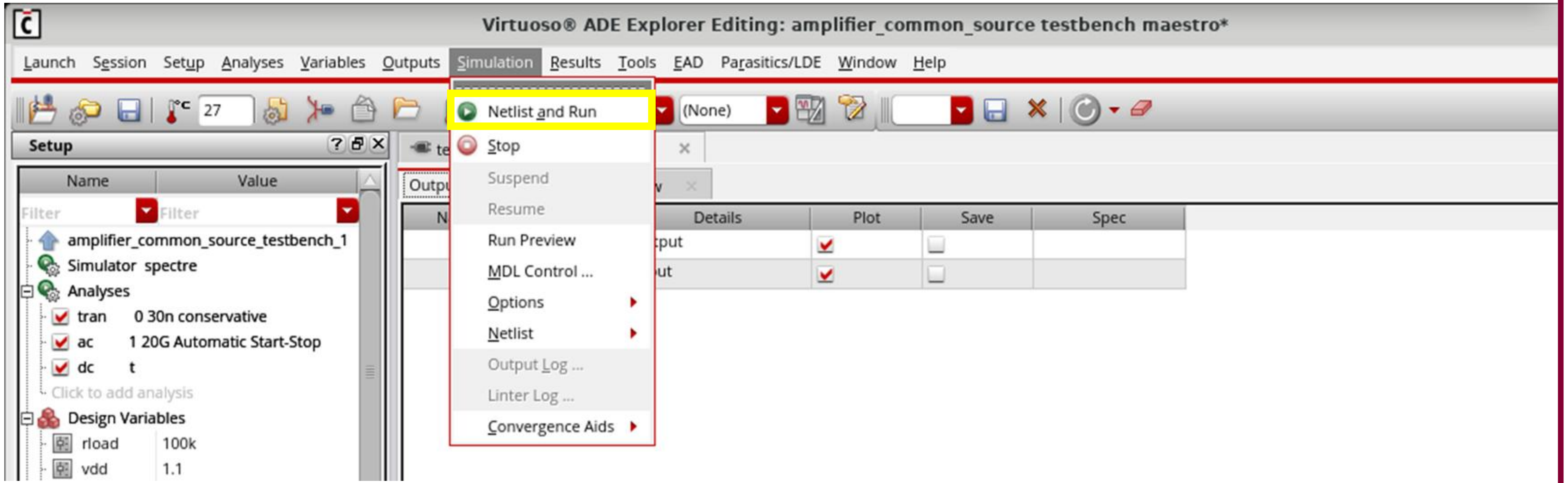
Defining the Model Library

Single Point Simulation

AC Simulation

2. Running the Single Point Simulation

- The setup of your analyses is now complete, to run the simulation, click on the **Netlist and Run** button.



- Always make sure to “Check and Save” the design right before running the simulation.
- Now, the simulation will run.

cadence

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

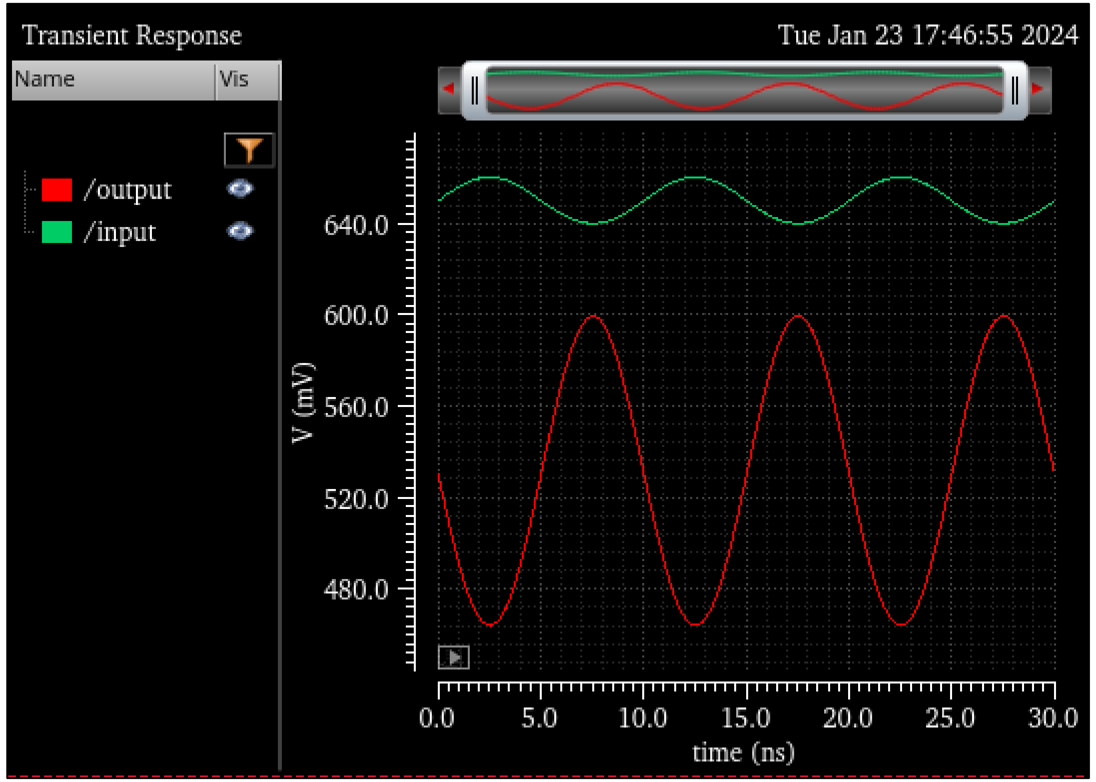
Defining the Model Library

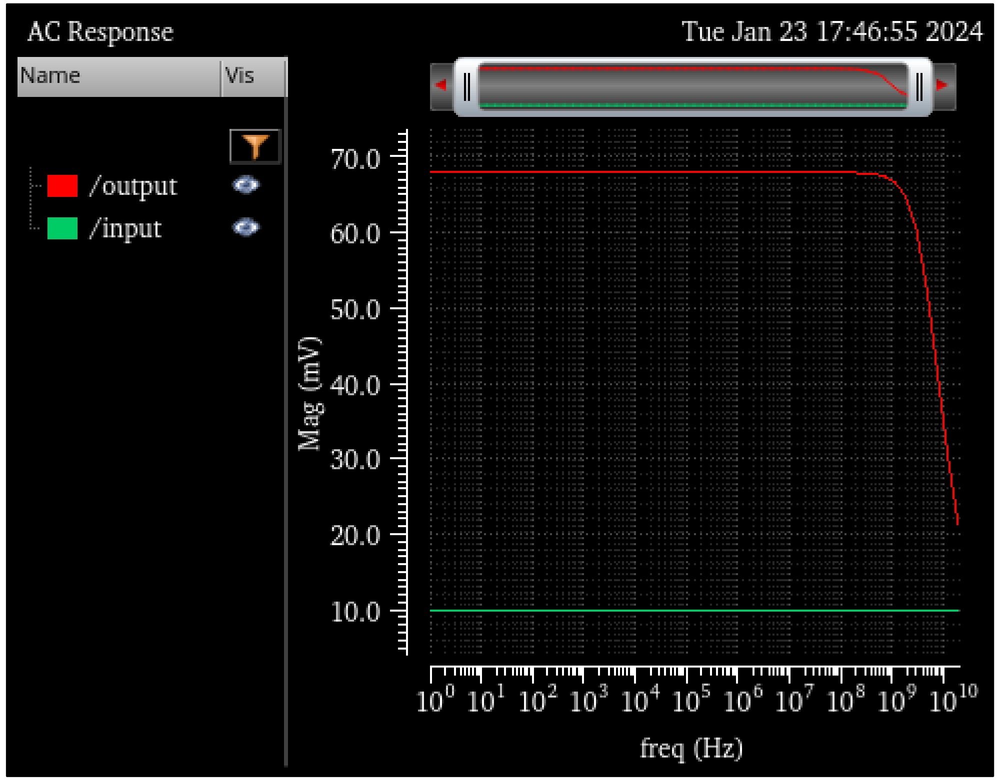
Single Point Simulation

AC Simulation

2. Running the Single Point Simulation *(continued)*

- The results of the transient and ac analysis are shown in the following plots.





- You can use the “Calculator” tool within the waveform viewer to create additional waveforms from the ones you have, e.g. vin2-vin1.
- We will use this tool to be able to view the gain in the ac analysis later.

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation

2. Running the Single Point Simulation *(continued)*

- Observe that the transient analysis's output peaks at 600 mV. In addition, the DC average is 530 mV rather than the original 550 mV. This little adjustment took place because the simulation tool's internal models and formulae for carrying out all the intricate calculations differ from those used for manual calculations, which oversimplify the process.
- The designer should primarily use the simulations as their major source of reference, with the manual calculations serving just as a guide for the trend he should be following.

AMERICAN UNIVERSITY OF BEIRUT

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation

2. Running the Single Point Simulation *(continued)*

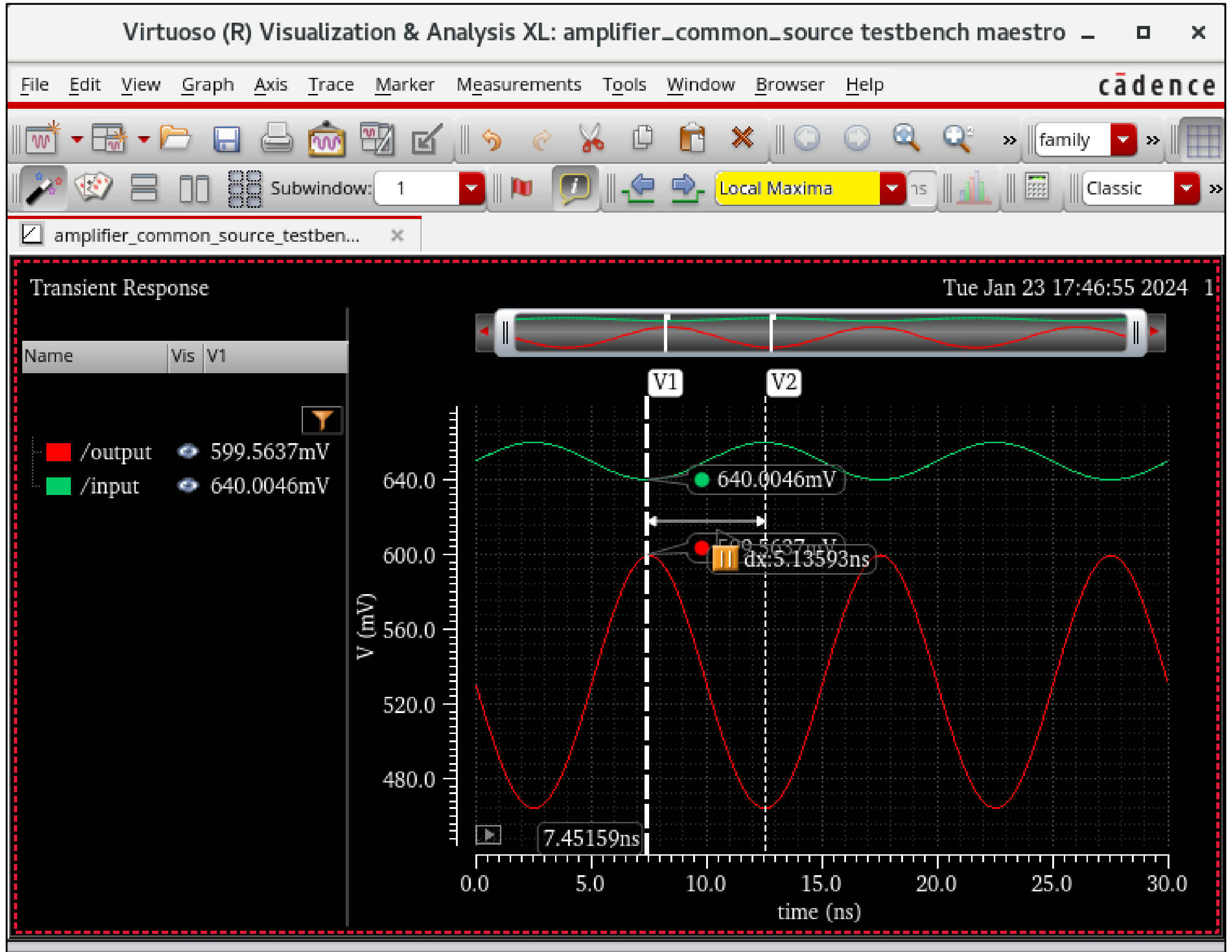
- Below is a method to find the small-signal voltage gain using the transient analysis.
- The method is as follows:
  - Create two vertical markers. Select the first marker and set it at the Local Minima and the second at the Local Maxima (shown next slide). The vertical marker intercepts for all the traces are displayed.
  - Select the two vertical markers by using the *Ctrl key* and choose Marker → Create Delta Marker.
  - To display the vertical marker intercepts in a table, do the following: *Select Window → Assistants → Vert Marker Table.* The Vert Marker Table assistant appears at the bottom of the window, displaying all vertical marker intercepts for each trace.
  - The delta value for both Voutput and Vinput is displayed in the marker table.
  - To calculate the voltage gain, use the following equation: 20log(delta(Voutput)/delta(Vinput)).
  - The equation would be: 20log(135.241/19.98) = 16.61 dB
  - The small-signal voltage gain is then 16.61 dB.
- Later, we will use the ac analysis to find the voltage gain and the 3dB bandwidth.

- The delta value displayed in the vertical marker table is always an absolute value.

cadence

2. Running the Single Point Simulation *(continued)*

- ADE Explorer
- Defining the Variables
- Defining the Analyses
- Defining the Outputs
- Defining the Model Library
- **Single Point Simulation**
- AC Simulation



Vert Marker Table

	V1	V2	|V2-V1|
x	7.45159ns	12.5875ns	5.13593ns
/output	599.5637mV	464.3222mV	135.2415mV
/input	640.0046mV	659.9849mV	19.98026mV

2. Running the Single Point Simulation *(continued)*

ADE Explorer

Defining the Variables

Defining the Analyses

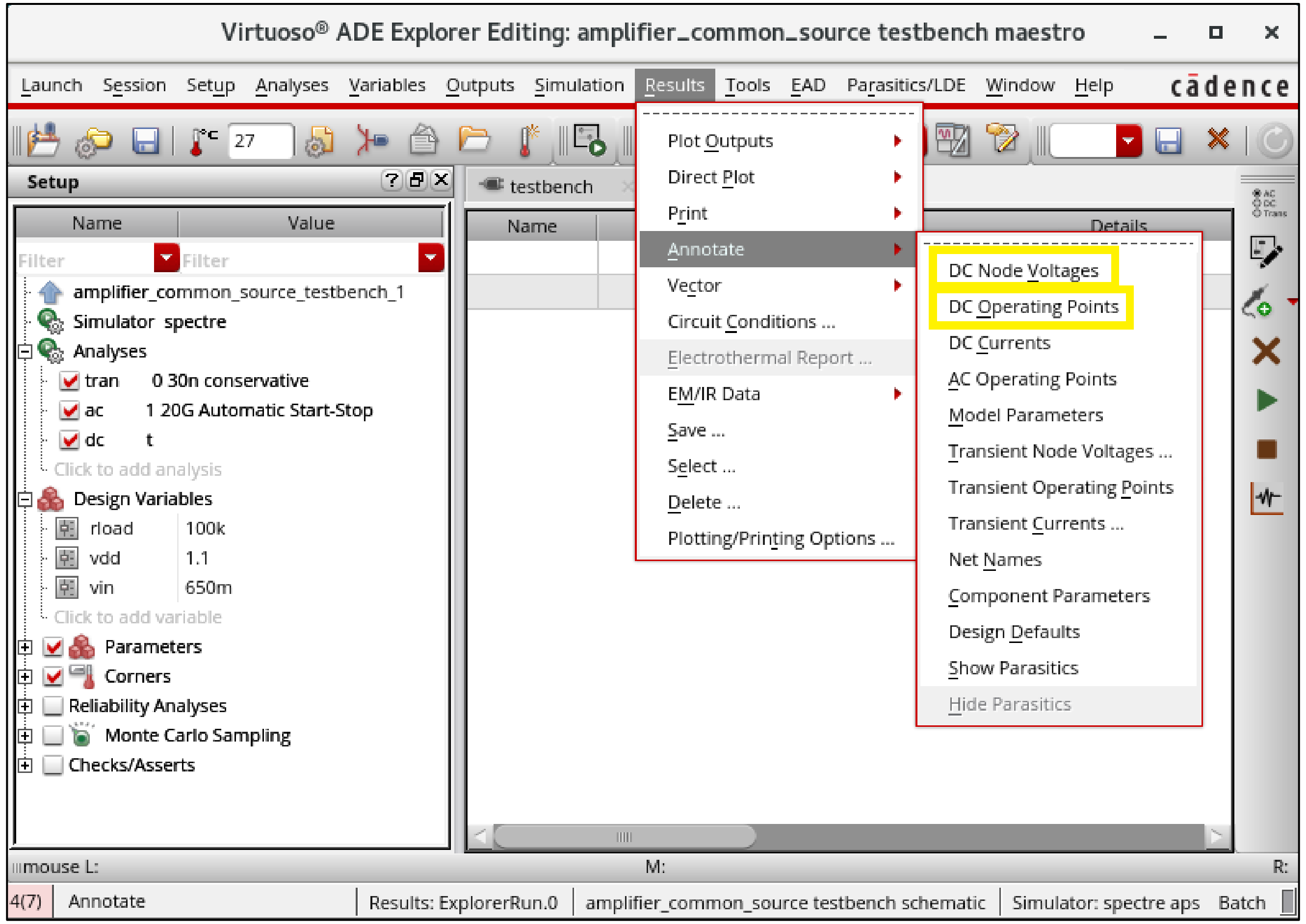
Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation

- To view your DC Operating points, and DC node voltages; select **Results → Annotate → DC Node Voltages and DC Operating Points**.



ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

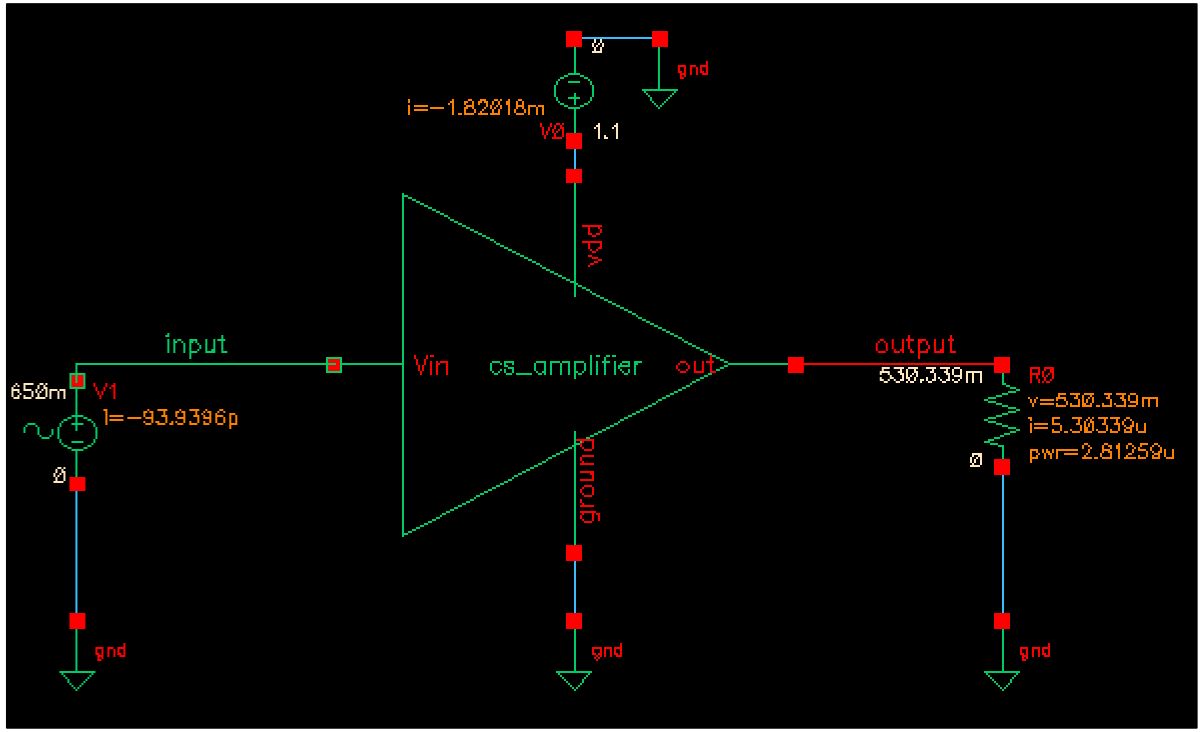
Defining the Model Library

Single Point Simulation

AC Simulation

2. Running the Single Point Simulation *(continued)*

- The DC operating points and node voltages are annotated on the schematic.



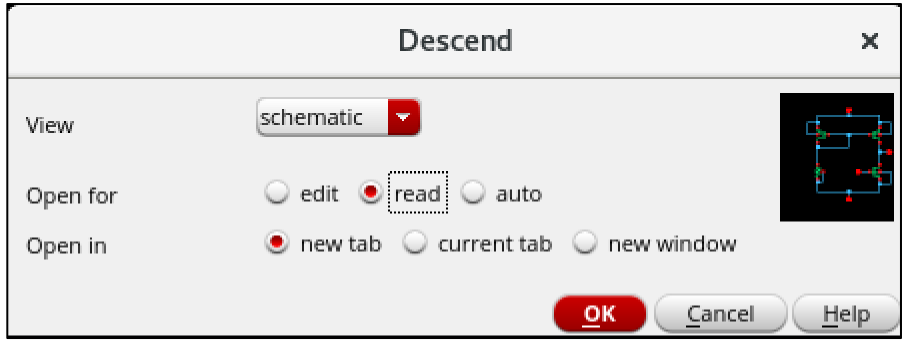
ADE Explorer
Defining the Variables
Defining the Analyses
Defining the Outputs
Defining the Model Library
Single Point Simulation
AC Simulation

2. Running the Single Point Simulation *(continued)*

- To check **DC Node Voltages and/or DC Operating Points** inside your OpAmp, descend to your OpAmp in Hierarchy by either method:

  1. Select the OpAmp, then right click → Descend Read…
  2. Double Click on the OpAmp

- The "Descend" window will pop up, you can choose either to **edit**, or **read** only your schematic. Click **OK** to proceed.

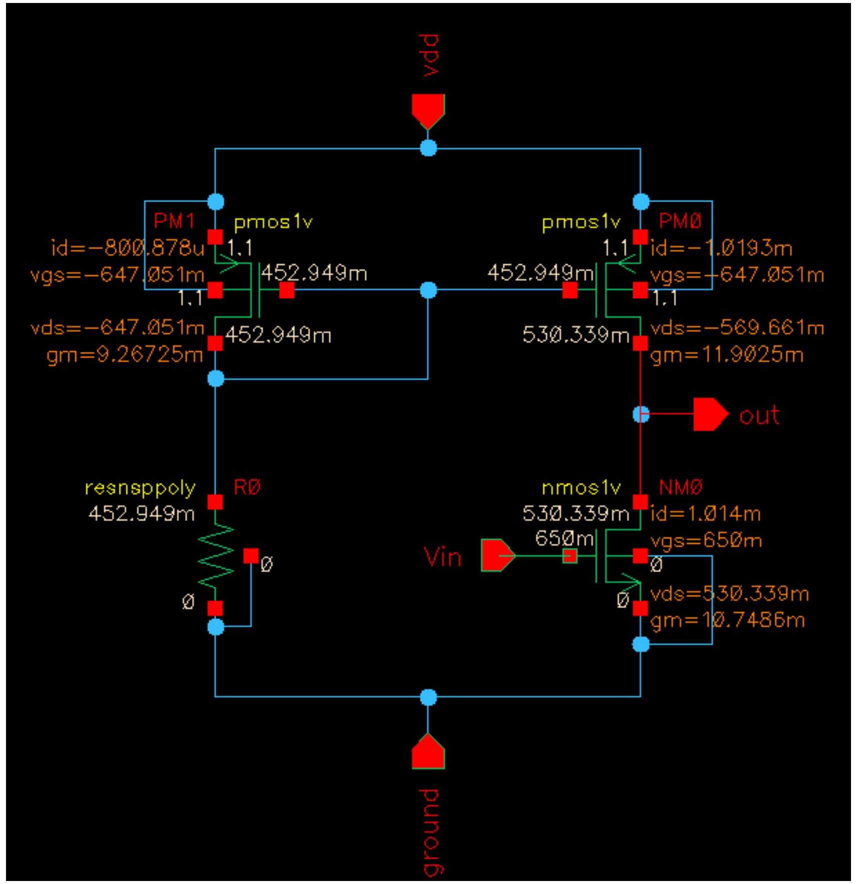


cadence

AMERICAN UNIVERSITY OF BEIRUT

2. Running the Single Point Simulation *(continued)*

- The schematic will pop up in a new tab with the **DC Node Voltages and DC Operating Points** annotated, as shown below.



- Before exiting your simulation, make sure to save ADE Explorer.

cadence

3. Running the AC Simulation

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation

3. Running the AC Simulation

- We will use the tool Calculator to be able to manipulate waveforms.
- We will also use another method to find the AC gain and Bandwidth of the design.
- From the maestro tab, select Tools → Calculator…
- We will first calculate the gain.
- The ac gain equation is 20log(Vo/Vin) and the answer will be in dB. The 20log function in the Calculator is “**dB20**”.
- Now for the (Vo/Vin) part, we have to plot the AC output voltage divided by the AC input voltage vs. frequency.
- The expression of the AC output voltage vs. frequency is **vf(output)** and the expression of the AC input voltage vs. frequency is **vf(input)**.
- Our final expression will be:

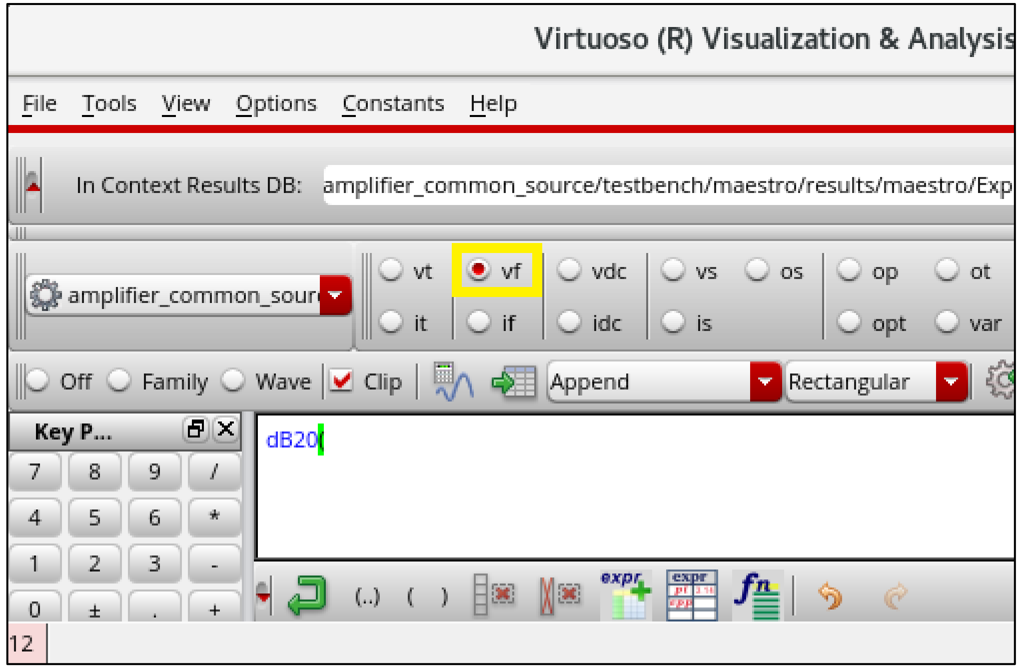
  dB20(vf(output)/vf(input))
- The next few slides explain how to plot the gain.

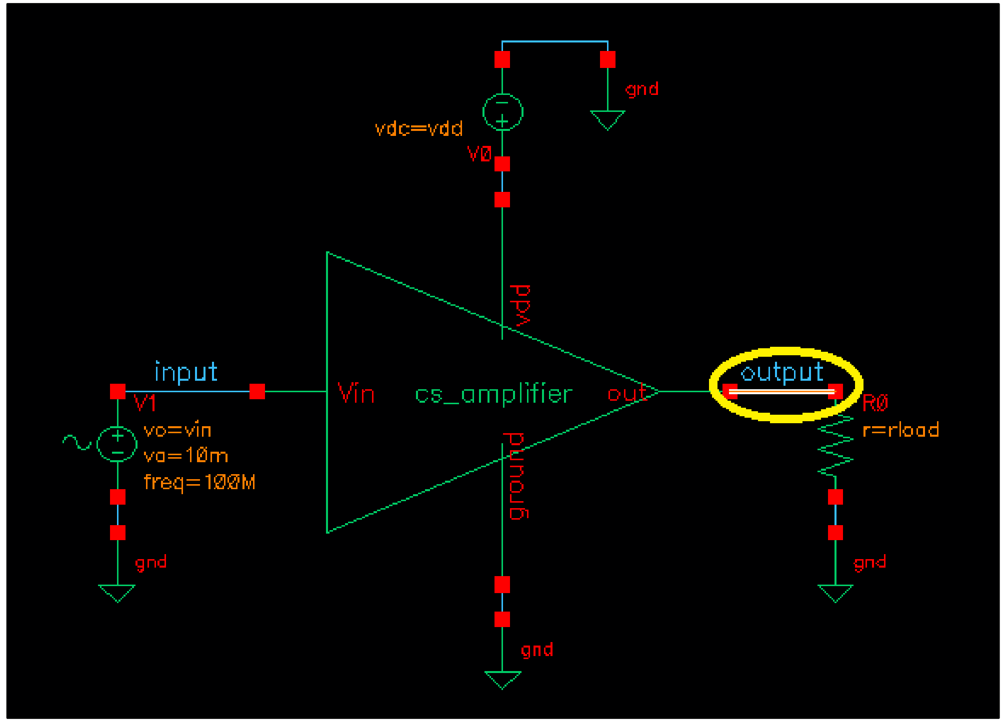
AMERICAN UNIVERSITY OF BEIRUT

- ADE Explorer
- Defining the Variables
- Defining the Analyses
- Defining the Outputs
- Defining the Model Library
- Single Point Simulation
- **AC Simulation**

3. Running the AC Simulation *(continued)*

- Start by typing “**dB20(** ” in the field shown below. As you can see this writing will be colored in blue which indicates that this is a built-in function in the calculator.
- Now click on the “**vf**” button. This will take you to your testbench tab where you will choose your output voltage. Remember that clicking on a red square means choosing current and clicking on a connection means choosing voltage.
- Click on the connection that represents your output voltage.





- Note that dB20 is case sensitive. db20 or DB20 or any other form won’t work.

cadence

ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

Defining the Model Library

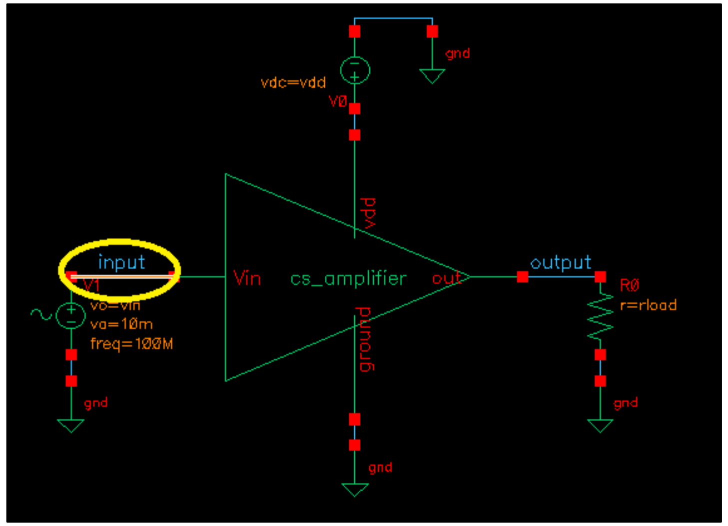
Single Point Simulation

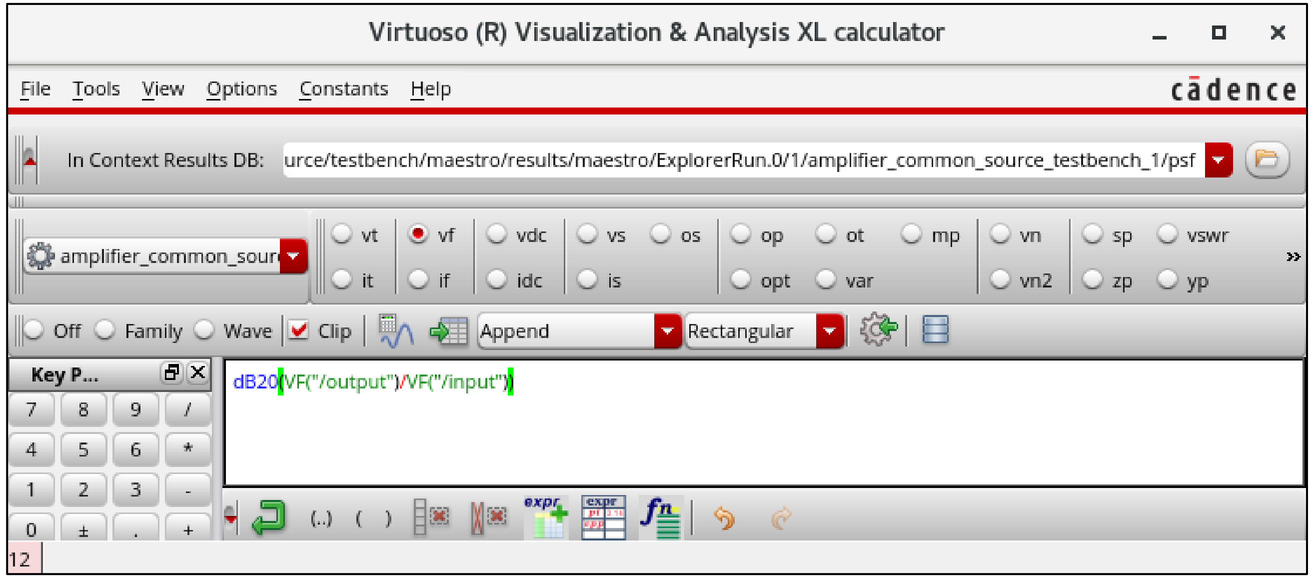
AC Simulation

3. Running the AC Simulation *(continued)*

- Now the expression we have is ‘dB20(VF(“/output”)’.
- We have to add the division sign “/”.
- Click on “**vf**” again and choose the connection that represents your input voltage.
- After that, **close** the parenthesis of **dB20** and your expression should now be:

dB20(VF("/output")/VF("/input"))





ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

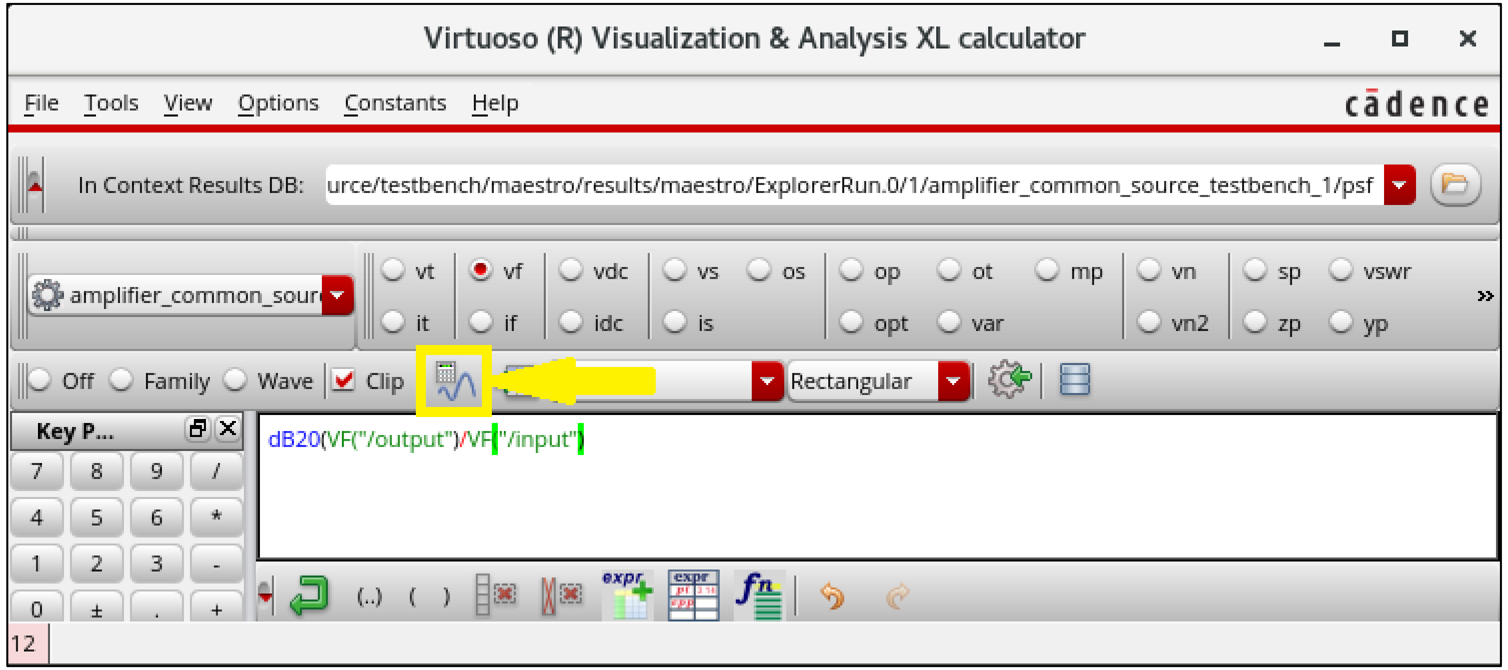
Defining the Model Library

Single Point Simulation

AC Simulation

3. Running the AC Simulation *(continued)*

- To plot the waveform, click on the highlighted button.
- This button will evaluate the written expression. It will generate a plot if needed or display a scalar value if this is the desired output.
- The generated output is shown in the next slide.



3. Running the AC Simulation *(continued)*

ADE Explorer

Defining the Variables

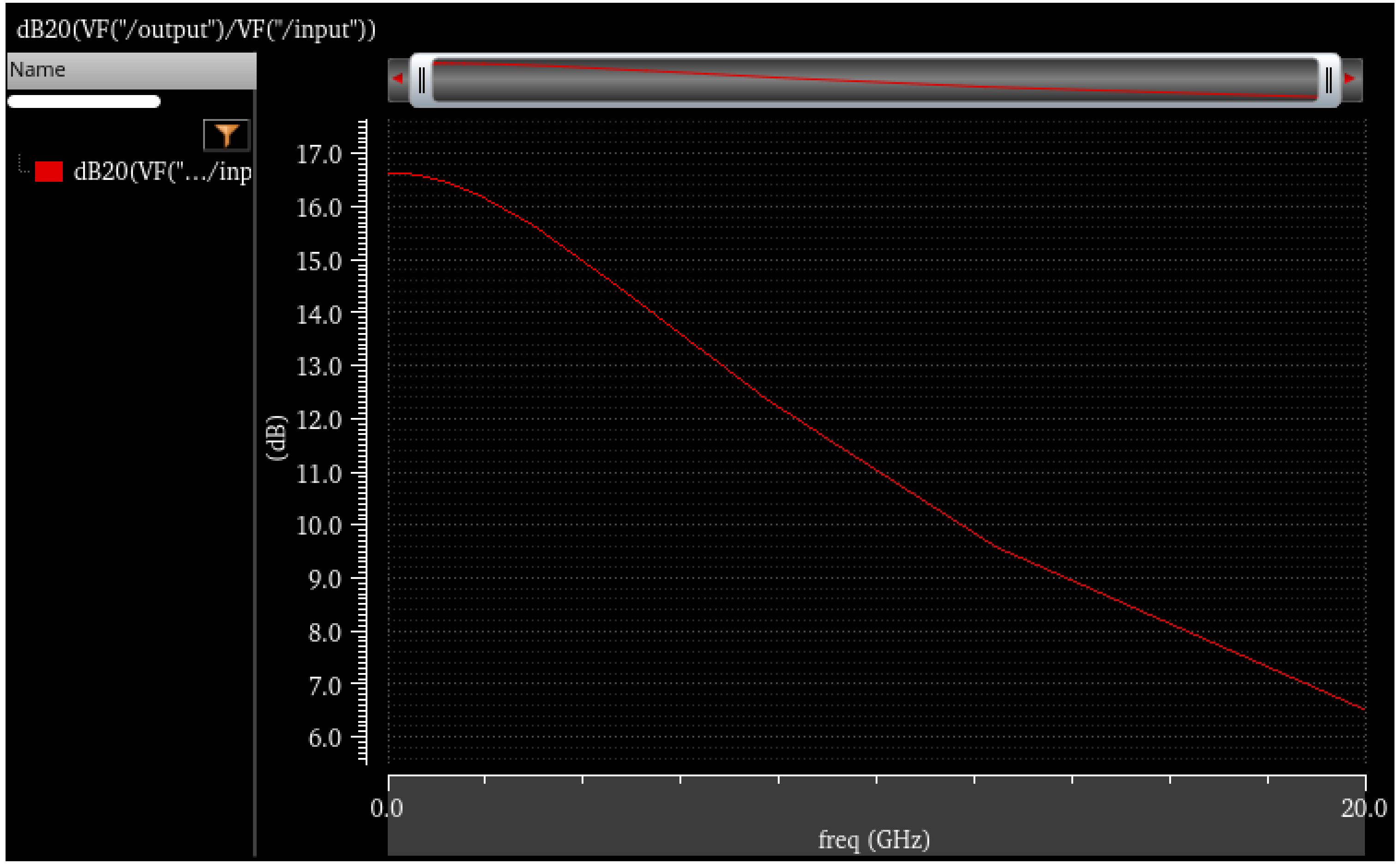
Defining the Analyses

Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation



ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

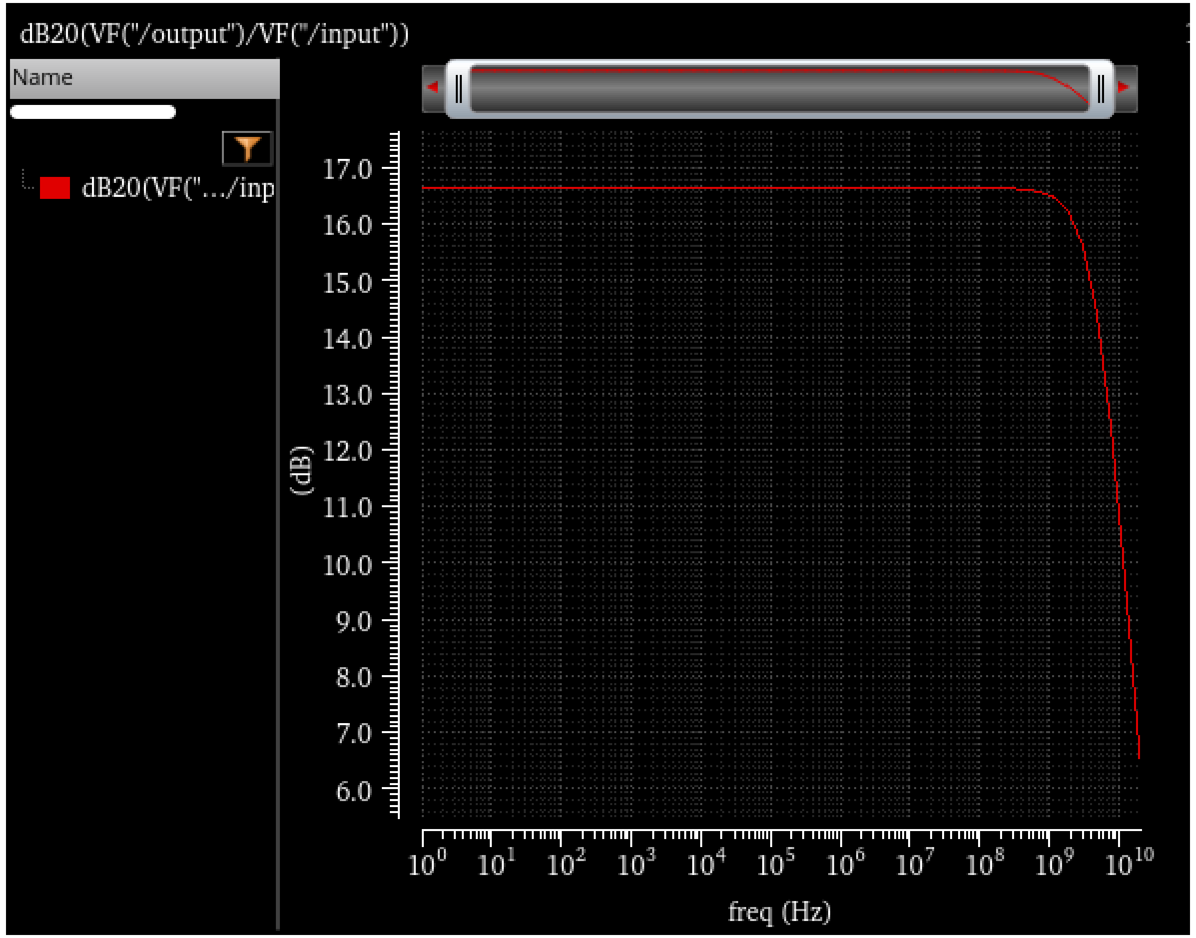
Defining the Model Library

Single Point Simulation

AC Simulation

3. Running the AC Simulation *(continued)*

- To be able to pinpoint the gain and the bandwidth, change the scale of the x-axis to logarithmic by double clicking on the x-axis → **Scale** → **Log.**
- The plot is shown below.



ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

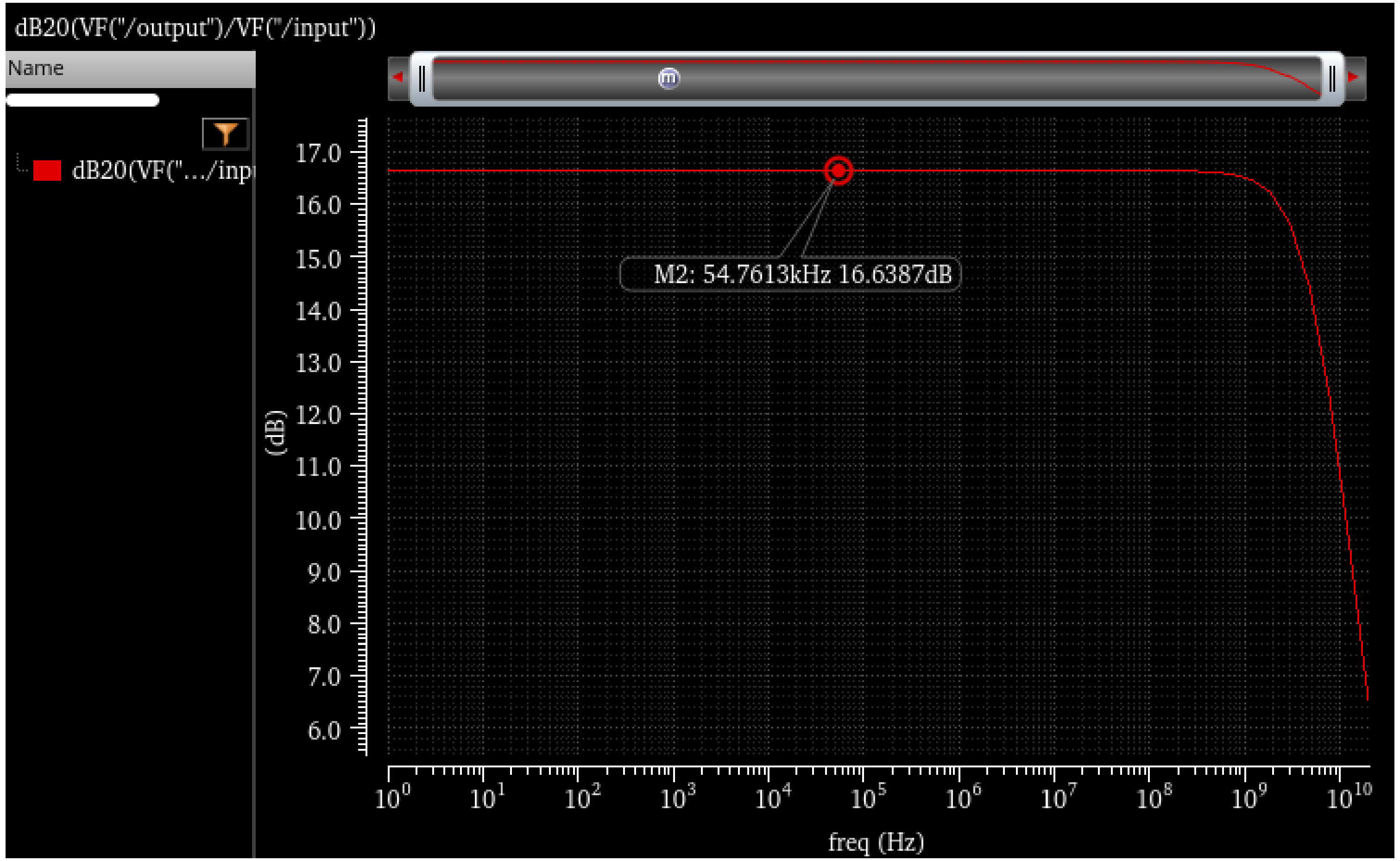
Defining the Model Library

Single Point Simulation

AC Simulation

3. Running the AC Simulation *(continued)*

- The gain is in the stable part of the graph. Add a marker by going to **Marker → Create Marker → OK** and put it on a point in the stable part.
- We can see that our gain is 16.63 dB.



ADE Explorer

Defining the Variables

Defining the Analyses

Defining the Outputs

Defining the Model Library

Single Point Simulation

AC Simulation

3. Running the AC Simulation *(continued)*

- The 3dB bandwidth is at the frequency where the curve drops by 3dB.
- The maximum of the curve is at 16.63 dB, so the 3dB frequency will be at 13.63 dB.
- To find this frequency, add a horizontal marker and set the Y Positions to 13.63.
- The 3dB bandwidth is approximately 6 GHz.