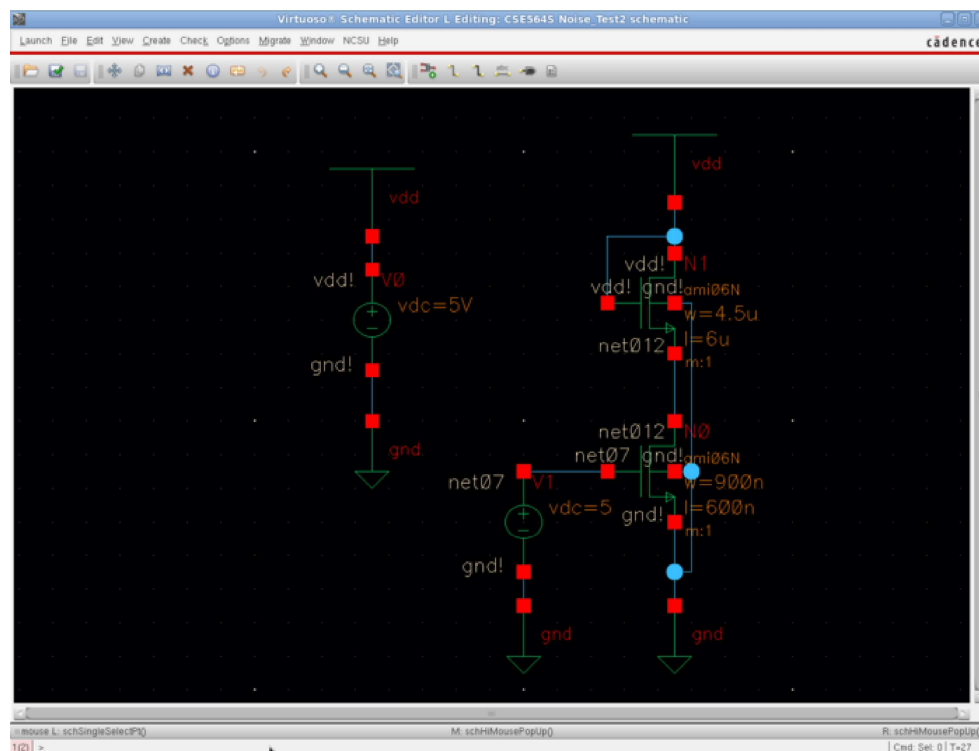


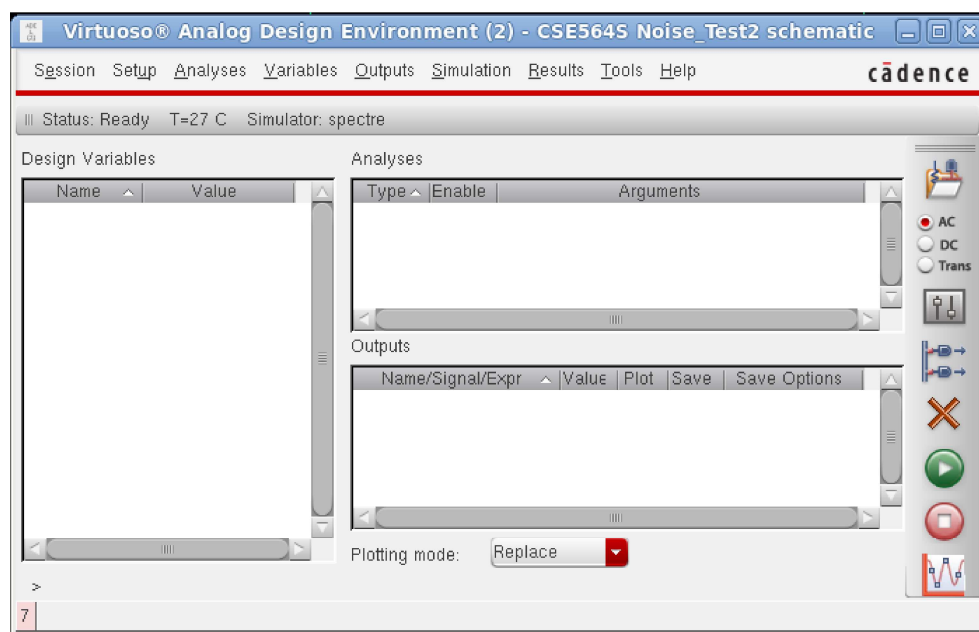
## How to Perform a Noise Simulation in Cadence

From EDA Wiki

First open the Cell View of the circuit you wish to perform the noise analysis on.



Start the ADE by going to **Launch -> ADE L**.



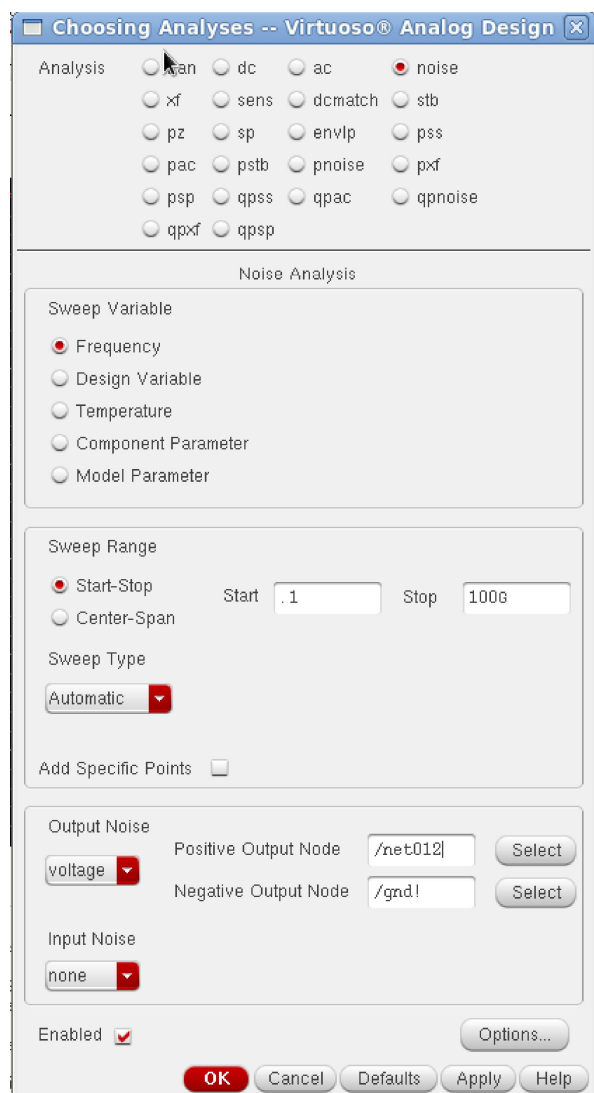
To set up the simulation click on **Analyses -> Choose....**

Select **noise** as the analysis type, and enter the range of frequencies you'd like to simulate in the **Start** and **Stop** boxes.

There are two ways to set up what to use for the output of the simulation. Choosing the **probe** option from the drop down allows you to select a transistor from your circuit to be analyzed.

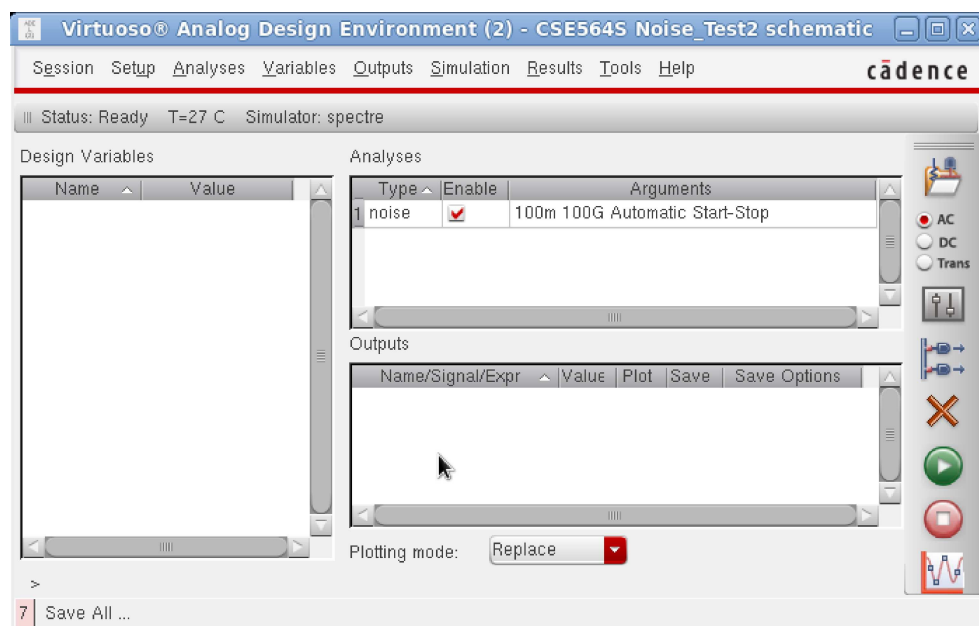


You can also choose to monitor the output voltage noise by picking **Voltage** from the drop down menu and selecting the wires in your schematic that represent the positive and negative voltage output.

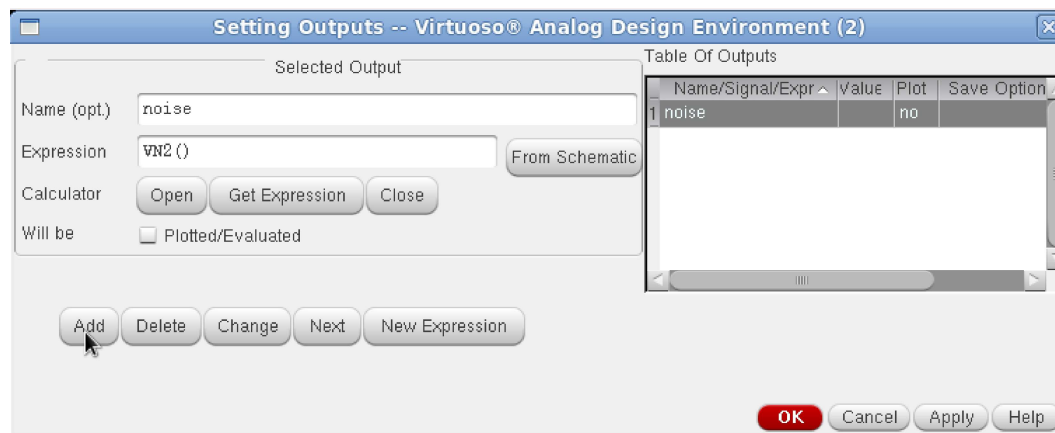


Finally, for the input make sure to select **none** from the drop down menu and then click **OK**.

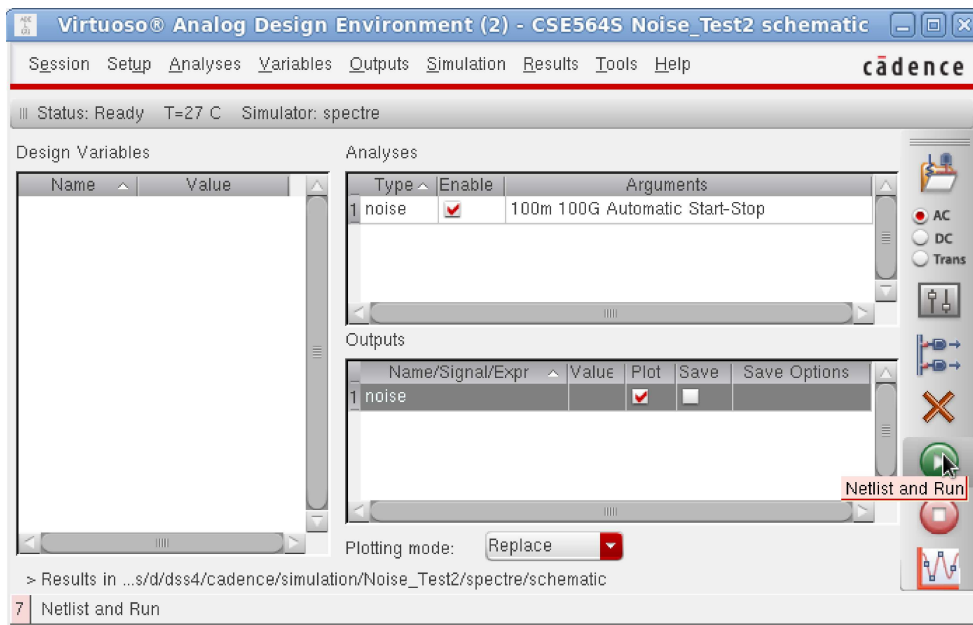
To set up a data plot, right click the box under **Outputs** (shown by the mouse in the below figure) and select **Edit**. This will bring up the output dialog.



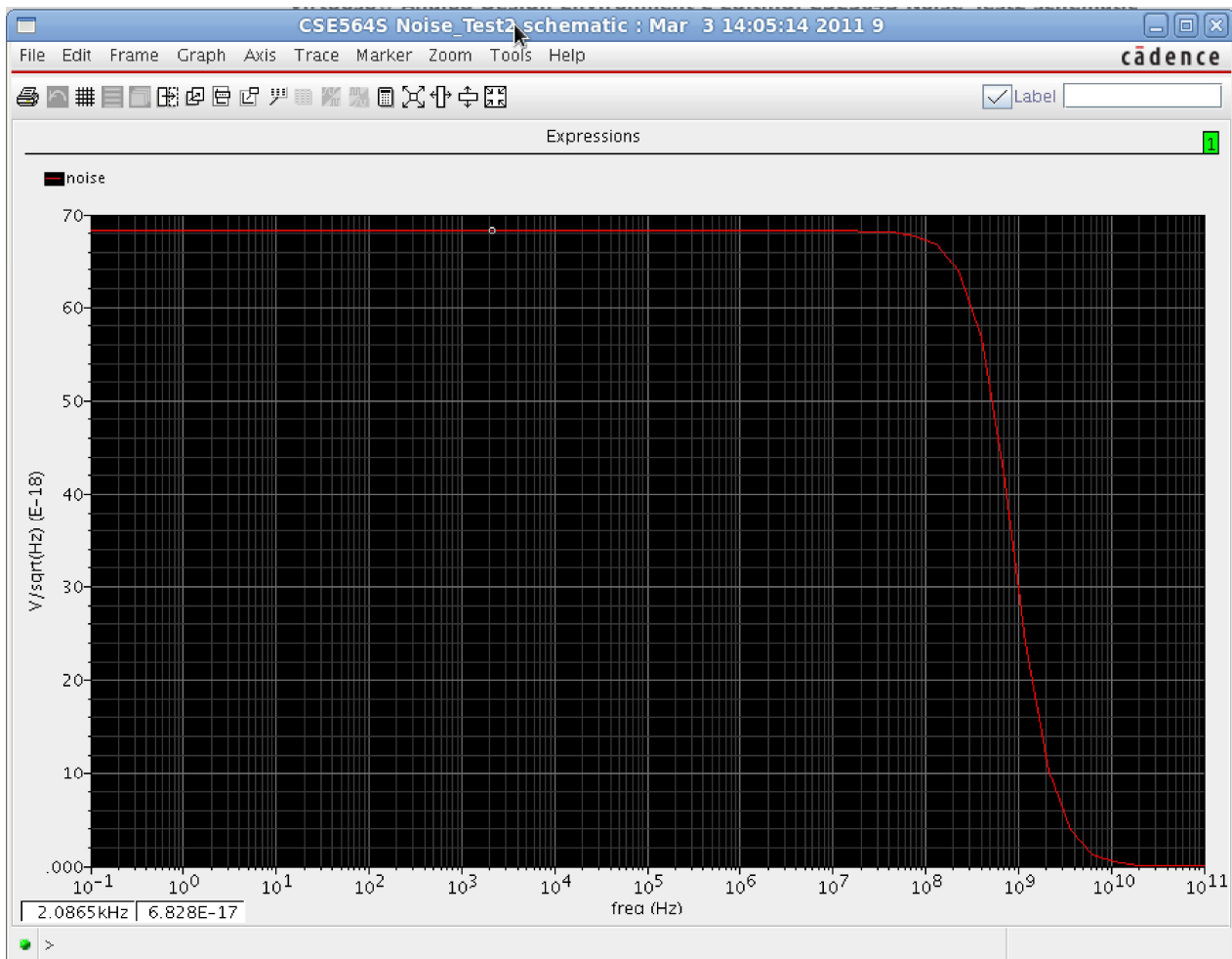
In the **Expression** box enter VN() if you would like your results to be V/sqrt(Hz) or VN2() if you'd prefer the results in V<sup>2</sup>/Hz. The name can be anything you wish. Click **Add** and then **OK**.



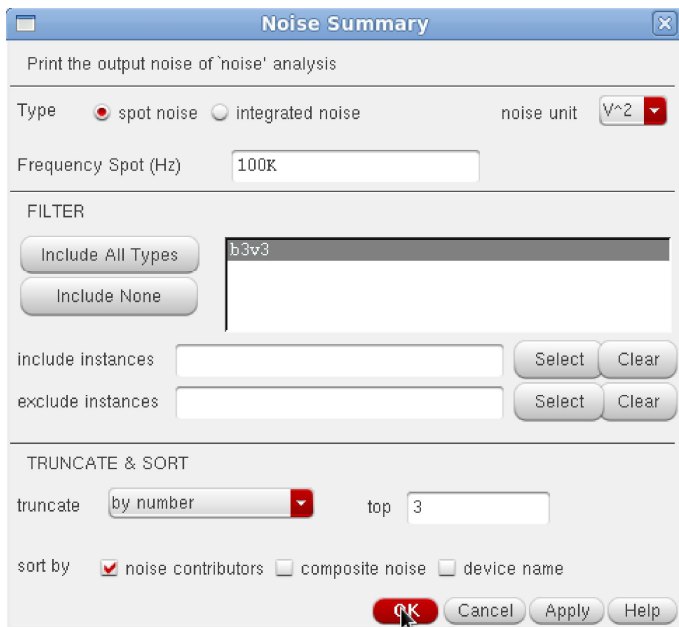
Your simulation is ready to run. Click the green play button to start the simulation and display the results.



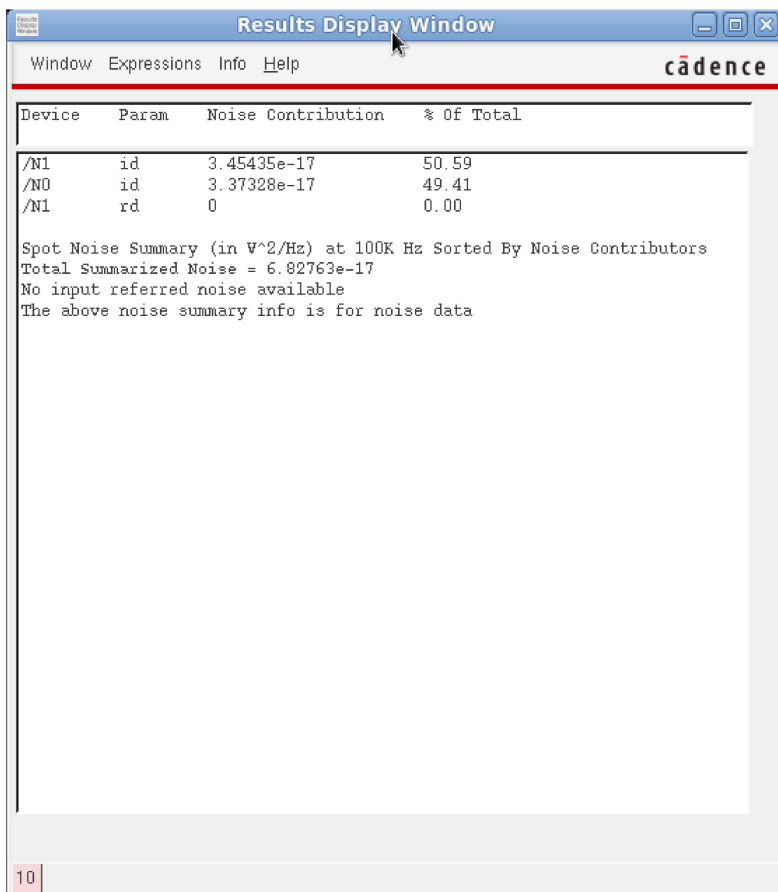
There is an error in cadence when displaying the plot. If VN2() is the expression you chose for your result the units on the graph will still read V/sqrt (Hz) (as shown below). The data that is plotted, however, is correct and in the proper units (you can plot and compare both expressions to check for yourself).



Finally, more in depth information can be gained by looking at the results for one particular frequency. To do this, on the ADE window select **Results -> Print -> Noise Summary**. In the box that pops up, select the frequency that you're interested in and click **Include All Types** then **OK**.



A window will display the total noise at the given frequency and which parts of the circuit contribute to that noise.



Retrieved from "[http://eda.engineering.wustl.edu/wiki/index.php?title=How\\_to\\_Perform\\_a\\_Noise\\_Simulation\\_in\\_Cadence&oldid=705](http://eda.engineering.wustl.edu/wiki/index.php?title=How_to_Perform_a_Noise_Simulation_in_Cadence&oldid=705)"