

All,

Pages 3-9 outline the noise analysis simulation set-up.

At the part where it talks about adding flicker noise, switch back to the following below (pages 1-2).

1. Open your ADE L
2. Go to setup/ model files
3. Look at the pathway for the ami06N.m file already in place (click the (...), double click the ami06N.m
4. Highlight the pathway
5. Open a terminal
6. mkdir ~/cadencelin/models
8. cp "model pathway" ~/cadence/models
9. repeat 3,4 and 8 for the amio6P.m
10. cd cadencelin
11. cd models
12. gedit ami06P.m
13. add the following to the end of it, be sure to replicate the syntax already in use, this is very important.

```
KF=0.466E-26 &  
AF=1.61 &  
)
```

add

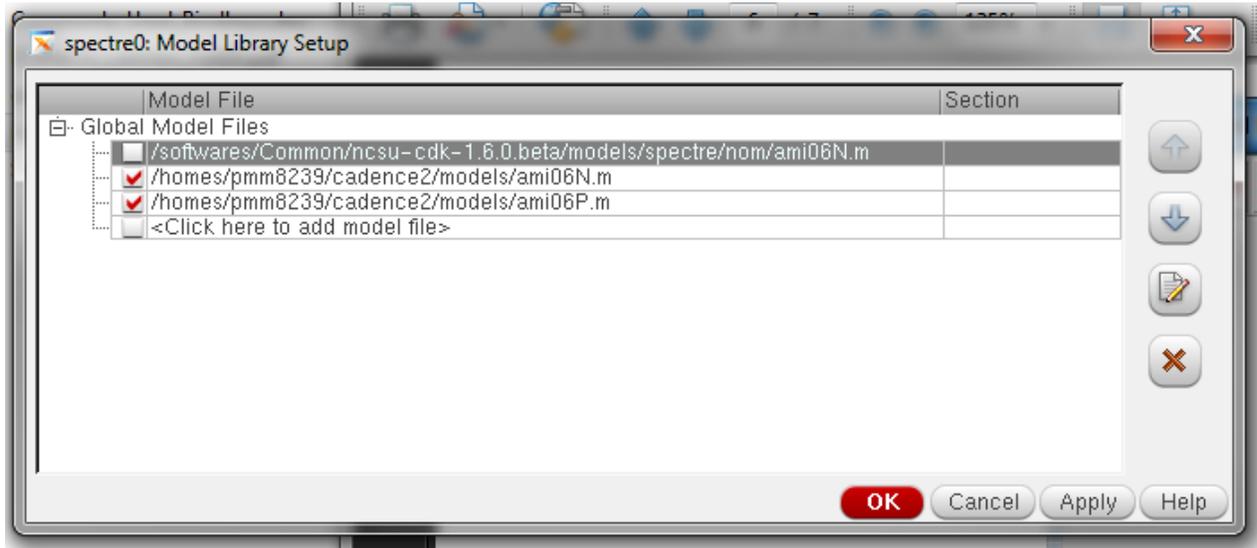
```
KF=0.276E-25 &  
AF=1.53 &  
)
```

to the end of the NMOs model file.

save both.

14. open the model files back up in the ADE

15. copy in the models files onto the list, and uncheck the model files that are there by default. the window should look like it does below. with your username in place of pmm8239



this will add the flicker noise components to the circuit and you will be able to run noise simulations that include flicker noise.

Noise Analysis

In order to do noise analysis on a differential system, you first need to have the correct input source configuration. This is shown in Figure 1. The VCVS values have gains of ± 0.5 to provide the differential gain to the circuit. If you want to run a common mode gain simulation, just change these values from ± 0.5 to 1. The source V_{IN_CM} is there to provide the common-mode input voltage to the circuit. In this case it is non-zero, but that is not always the case so we put it there in case we want a non-zero input common-mode voltage. It is a good idea to always use this method for your input when you have a differential input.

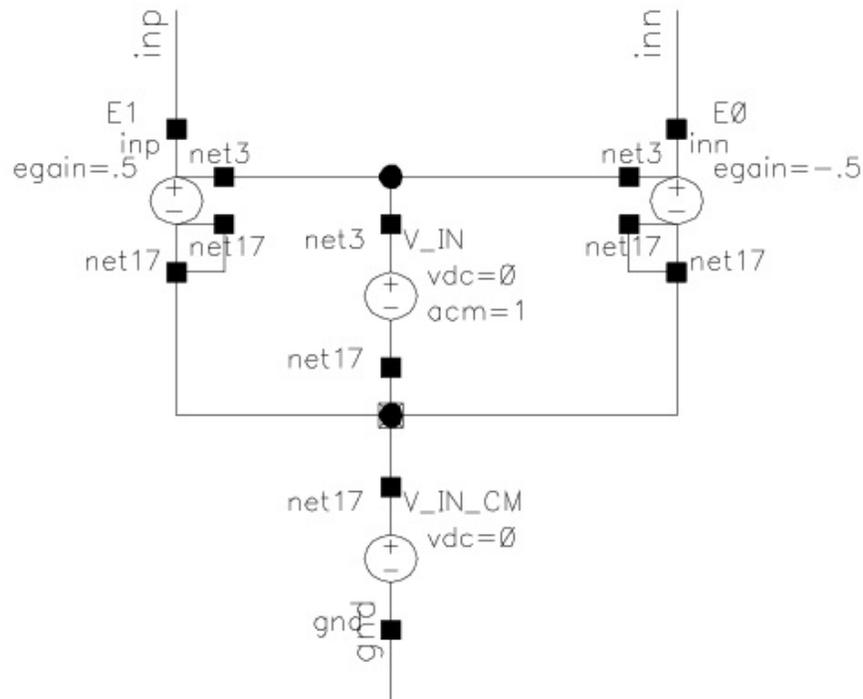


Figure 1: Differential Input

Figure 2 shows an example of a differential which I will do the noise analysis on. To run a noise analysis, click Analysis and then choose noise. Input the frequency range that you would like to use, for example 1 Hz to 1 GHz. Next we need to select the output node, so click “Select” next to “Positive Output Node” and then go to your schematic and click on your output node. Go back to your Analysis window and click “Select” next to “Input Voltage Source” and then go to your schematic and click on the input voltage source. When you are finished, your Analysis window should be similar to that of Figure 3. Click “OK” and then run the simulation.

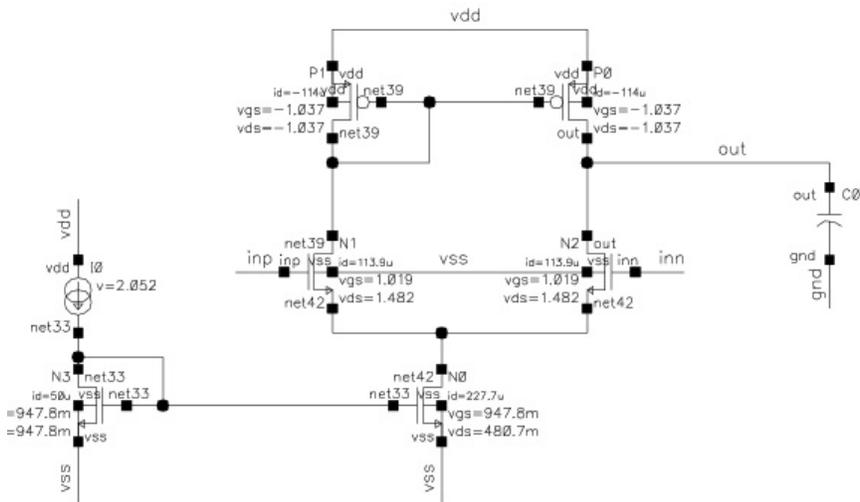


Figure 2: Single Ended Differential Amplifier

To plot the output noise and input referred noise, refer to Figure 4. You can plot either the noise, or the squared noise. Usually the squared noise is more useful when doing analog circuit design. As an example, I plotted the squared output noise in Figure 5. As you can see, there is no flicker noise. The flicker noise coefficients are not included by default in the models that you are using. We will have to modify our model files in order to get the flicker noise to appear. This is demonstrated in the next section.

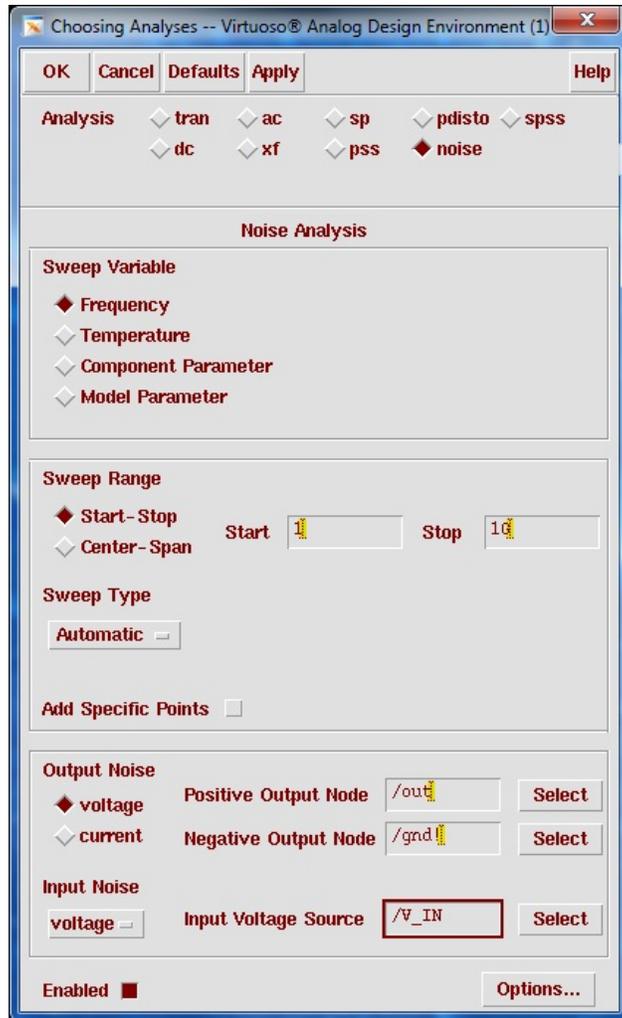


Figure 3: Noise Analysis Window

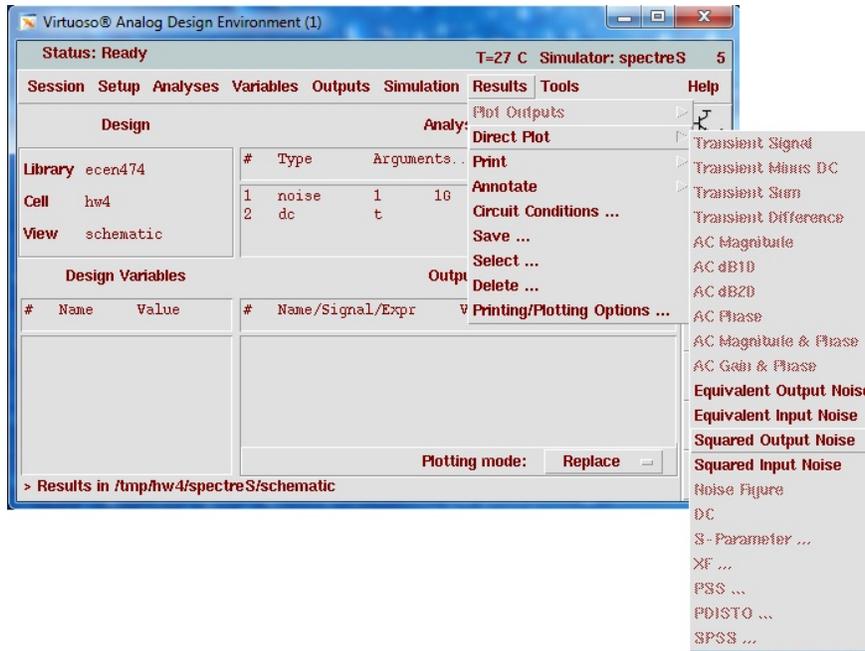


Figure 4: How to Plot Input and Output Noise

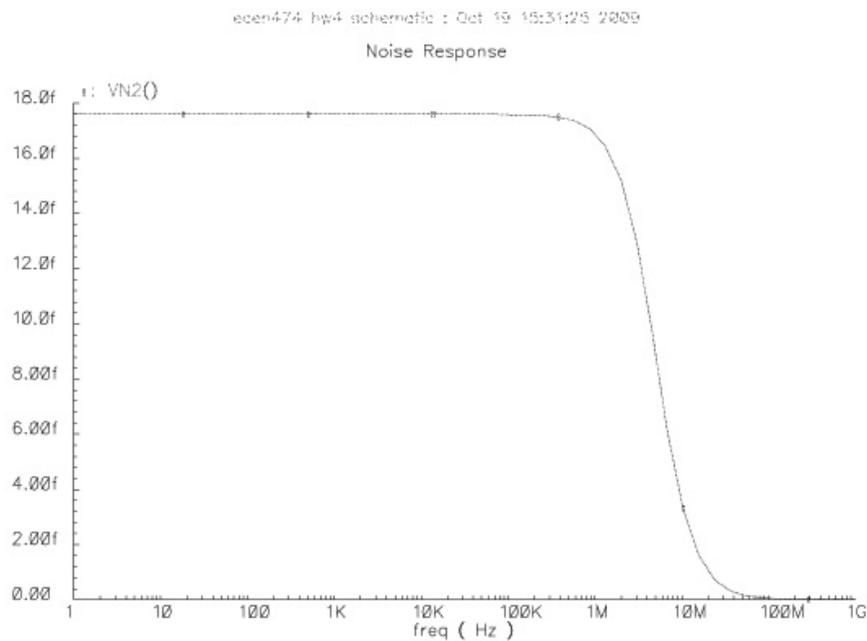


Figure 5: Squared Output Noise

Adding Flicker Noise Parameters to Model Files

We need to edit the model files and add flicker noise coefficients to them. To do this, we need to first make a directory to store the model files in, and then copy the default models into this directory. From your terminal type:

```
mkdir ~/cadence/models
cp /baby/cadence/ic50/local/models/spectre/nom/ami06N.m ~/cadence/models/
cp /baby/cadence/ic50/local/models/spectre/nom/ami06P.m ~/cadence/models/
```

Now change directories in your terminal to your model directory that you created. Use a text editor such as gedit to modify the model files. Type “gedit ami06N.m” to edit the model file.

For NMOS model, add the following to the end of your model file:

```
KF=0.276E-25 &
AF=1.53 &
)
```

For PMOS model, add the following to the end of your model file:

```
KF=0.466E-26 &
AF=1.61 &
)
```

For example, Figure 6 shows the model file for the NMOS after being modified. It is very important to follow the format exactly. You need to have no spaces in the model parameter, then one space followed by the &.

Next, we need to tell the Analog Environment where to look for the model files. Click “Setup” then “Model Path” in the Analog Environment. In the Model Path window that opens, type “/cadence/models” in the New Directory field, then select Add Above. This is demonstrated in Figure 7.

Now if you run the noise analysis, and replot the output, you should see flicker noise in your spectrum as seen in Figure 8.

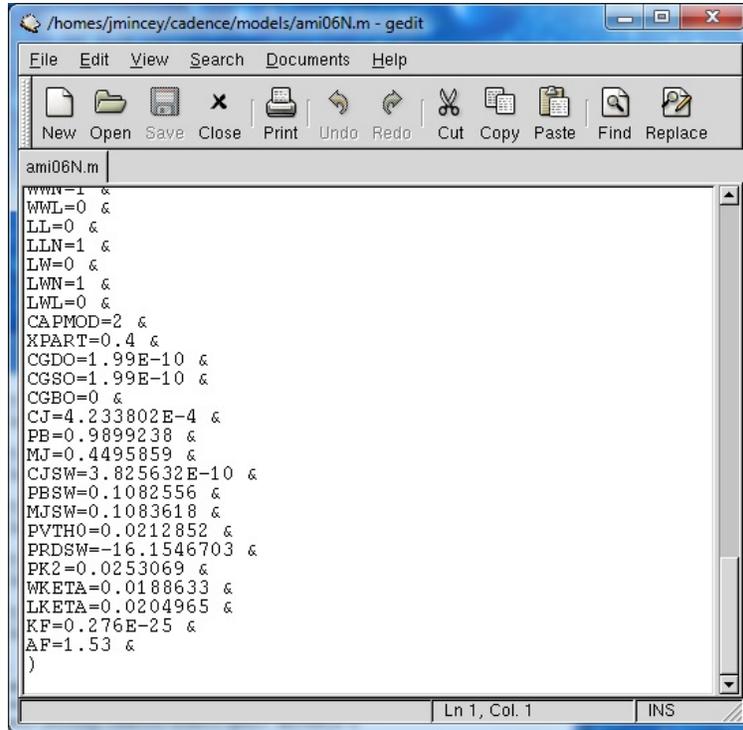


Figure 6: Example of Model File

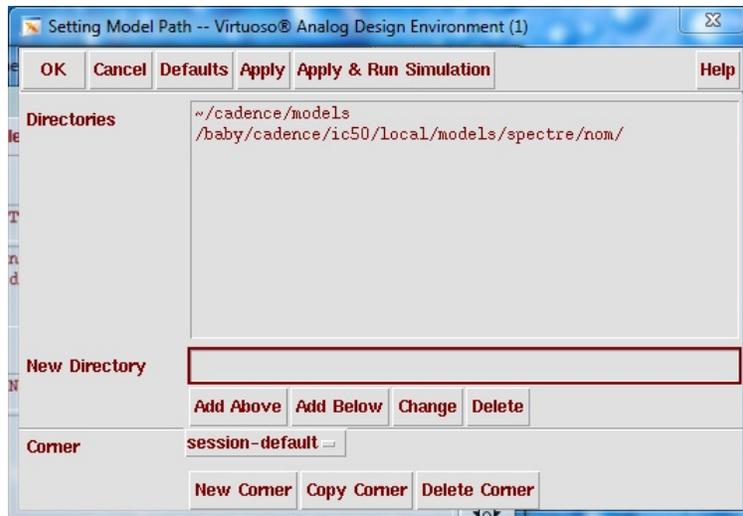


Figure 7: Editing the Model Path

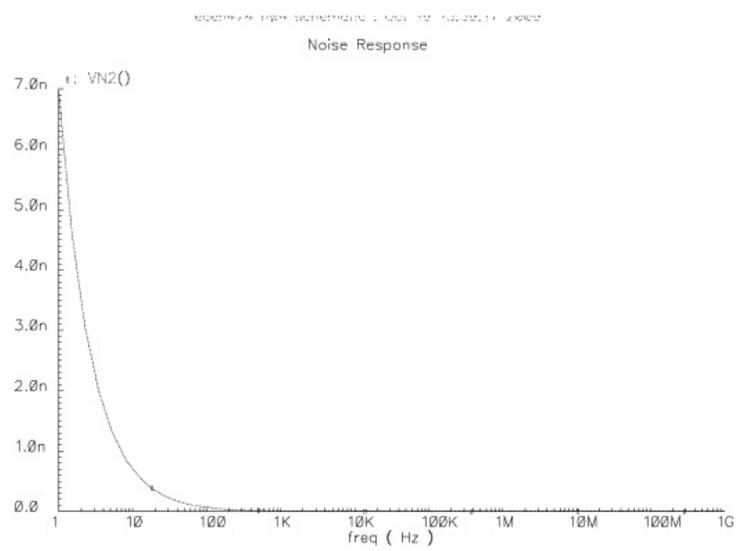


Figure 8: Squared Output Noise with Flicker Noise