

Setting Up and Running a Simulation

Once the test bench is setup successfully, for running the simulations click on 'Launch' tab and choose ADE L (analog environment).

In Analog Environment Window:

1. To attach and use the required models and parameters, click 'setup' tab and choose 'Model Libraries...', In the Model Library Setup window, add following model files

/Kits/IBM_PDK/cmrf7sf/reIAM/Spectre/models/allModels.scs (section tt)

/Kits/IBM_PDK/cmrf7sf/reIAM/Spectre/models/design.sc

Click Apply.....Ok

2. To choose the type of Analysis, click on 'Analysis' tab or the second button on the right hand side corner, and choose the desired type of Analysis [transient/DC/AC] .

a) If it's a transient analysis, click on 'tran' and type the stop time and the choose accuracy defaults (usually conservative). And click ok.

b) If it's a DC analysis, select 'dc' and click on component parameter (the parameter or the source that you would like to sweep) then select the component name, say a dc source in the test bench that has been named 'vin'. Then type 'dc' in the parameter name. Next select the sweep range and give the start and stop values (say, -0.5 to 0.5) and in the sweep type select linear, type the desired number of points (1000 usually) in the space near the 'total points' and click ok.

c) If it's an AC analysis, select 'ac', for the 'sweep variable' select 'frequency' and in the 'sweep range', type the desired range of frequencies (say 1Hz to 10 MHz). You can leave the sweep type in 'automatic' or choose 'logarithmic' and type in the number of points per decade.

3. Next select the 'variables' tab on the top or from the right side corner, and enter the names of the variables in the test bench (vdd and vss) and enter the desired values or Click variables Or you can Copy (variables) from cellview.

4. Next select the 'outputs' tab from the top or right side, select 'To be plotted' and then select 'Select on Schematic' . As soon as you do this, the test bench schematic will pop up and you can select the wires and nodes that need to be plotted (Tip: click on a wire if you want to plot a voltage and a node to plot a current). Using the setup outputs tab in the right side, you can also evaluate expressions like crossing points, phase margin, gain margin...

5. If you want to delete a variable or an output that you have selected by mistake, you can highlight the corresponding one and delete using the 'delete' button (fifth one from the top in the right side).

6. To run a simulation click on 'Simulation' tab and click on 'run' or simply click on the green light button in the right side. To stop a simulation in the middle, click on the red light button or choose 'Stop' by clicking the 'Simulation' tab.

7. You can change the process corner by
/Kits/IBM_PDK/cmrf7sf/reIAM/Spectre/models/allModels.scs (section ss/ff/sf or fs)

8. You can vary the temperature by clicking Tools..... Parametric Analysis.....variable.....Choose temperature and give a value to temperature.