ENGI 5131 --- Monte Carlo Analysis

Lakehead University

Wai Wong, Carlos Christoffersen

Winter 2013, updated May 2017

Objective:

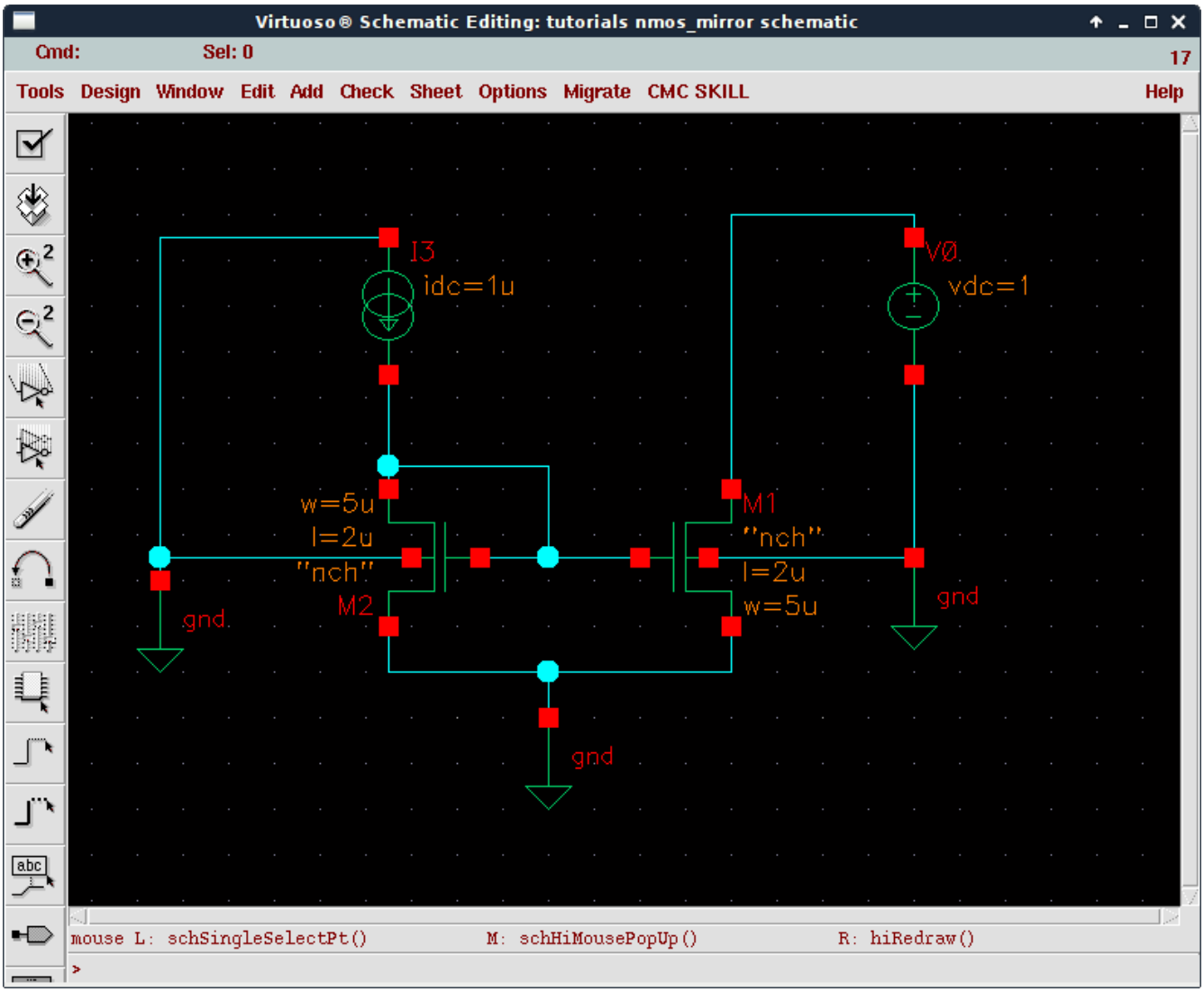
To gain familiarity with basic setup of the Cadence Spectre Analog Design Environment and the Monte Carlo analysis available there in.

1. Introduction

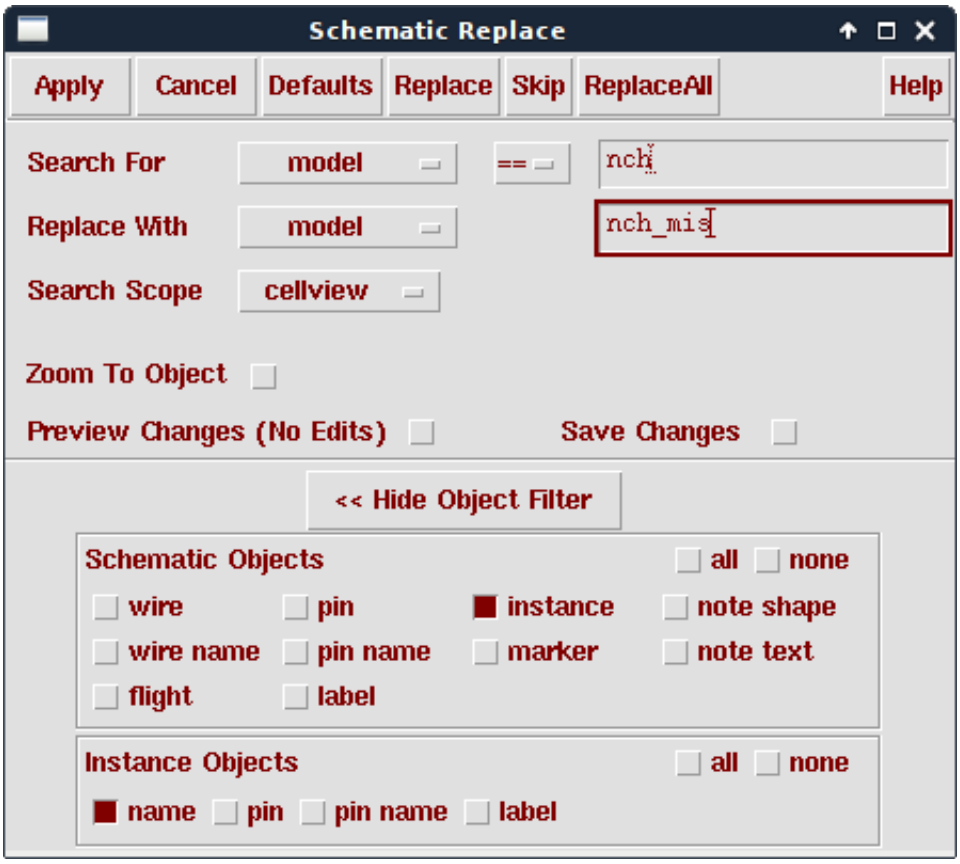
Monte Carlo analysis is commonly used to predict the effect of random variations of CMOS process parameters in the performance of a new design. For example, the tolerance rating of some capacitors and resistors can range as much as ±15%. Often it is impractical to include the tolerance of every component in hand calculations; fortunately, the Monte Carlo analysis incorporated in a circuit simulator can effectively predict the performance of a circuit in various working conditions. If the performance of a circuit does not meet expectations, consider to replace some components or redesign the circuit.

2. Schematic

Create a simple current mirror schematic as shown below:



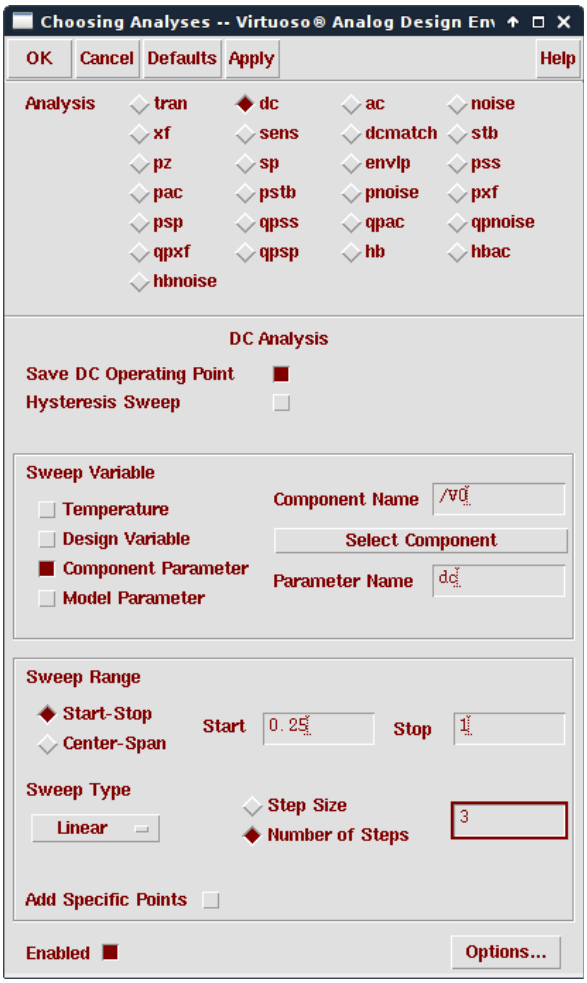
Replace the transistor standard models with the statistical models with mismatch information. In the schematic editor select Edit->Search->Replace:



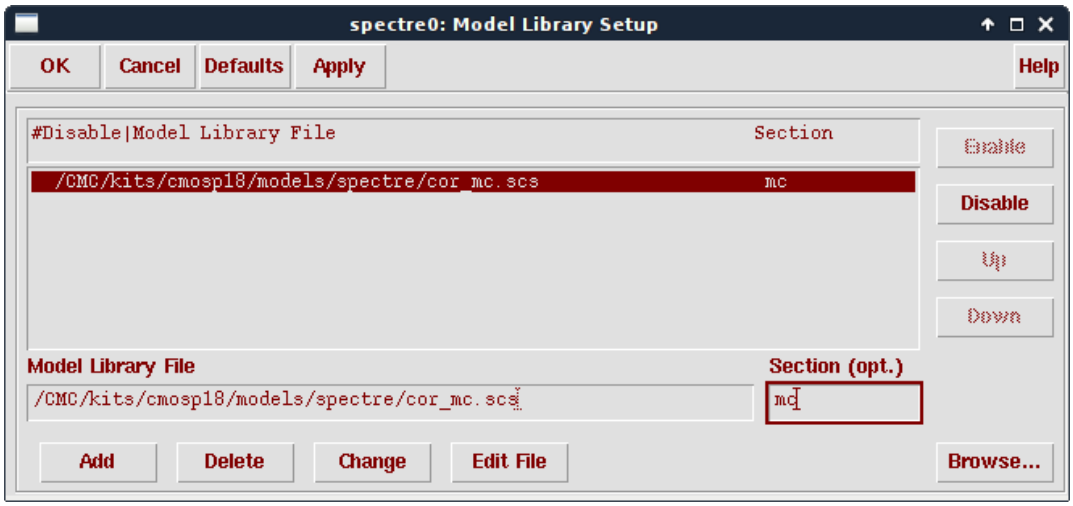
Initially, select “other” in the “Search For” list ant type “model”. Show the object filter and select the fields shown above. Then press “Replace” once for every instance. Close the Replace window and press “F6” to redraw the schematic. The new mode name appear next to each transistor. If your design has additional transistor models, perform a similar model replacement for each model (for example, use pch_mis for thin-oxide PMOS).

3. Monte Carlo Set up

1. Start the ADE and set up a DC sweep as shown below. Make sure to check the “save DC operating point” box.



2. Open the **Model Library Setup Form** by clicking **Setup** -> **Model Libraries** in the ADE menu and update the library path of the ADE as shown below.



That library only includes the models for low-voltage standard-Vth MOSFETs. Note that the section is set to “mc” (Monte Carlo).

Corner Simulations

Corner simulations are an alternative to Monte Carlo analysis. Only manually-chosen combinations of parameters are considered in corner simulations. The default section with typical parameters is called “tt”. In addition, there are 4 sections with corner parameters available for transistor libraries: "ss", "ff", "sf" and "fs". The first letter refers to NMOS and the second for PMOS transistors (fast and slow). Just change the section to one of the corners in the Model Libraries window, for example from “mc” to “ff” and run simulations as usual.

For BJT, resistor and capacitor libraries, only “tt” (typical), “ss” (slow) and “ff” (fast) are available.

You'll have to include a separate line in the Model Libraries window for each section selected. For example, one line with “ss_na” for slow low voltage non-adjusted MOSFETs and “ss_bip” for slow bipolar transistor models. The process documentation and the following file can be used to guide your selection:

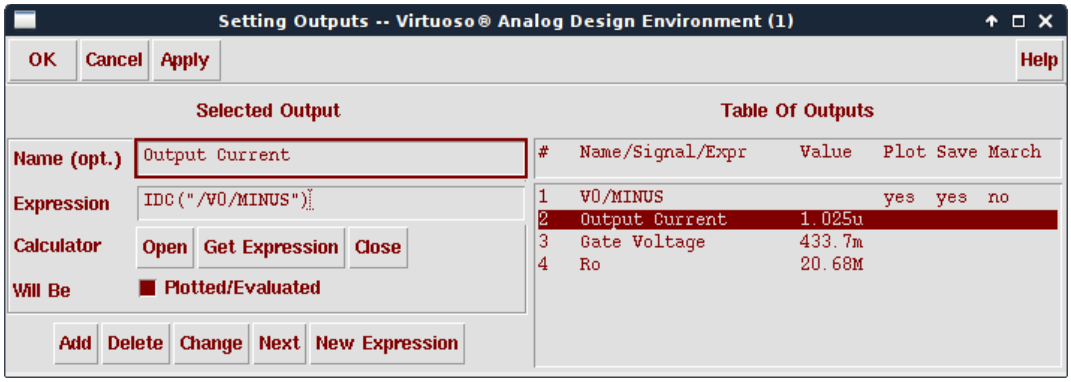
/CMC/kits/models/icfspectre.init

For example if your design is using both 1.8V and 3V transistors in a Monte Carlo simulation, you have to also include an additional library section in your model library.

```
/CMC/kits/cmosp18/models/spectre/cor_mc_3v.scs section=mc_3v
```

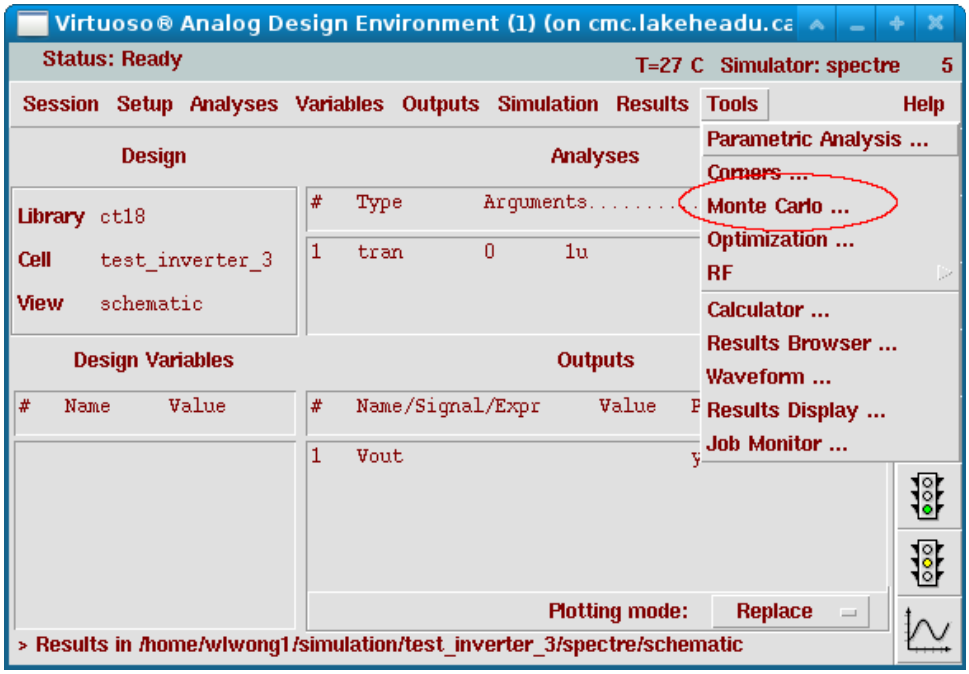
Note: in this process, statistical models are only provided for regular MOSFETs (1.8 V and 3.3V). For non-adjusted MOSFETs, BJTs and resistors only corner parameters are provided with this kit. Corner analysis can be performed with those devices.

4. Click OK to continue. Add the following outputs to the analysis: gate voltage, output current and output resistance (you have to figure out how to do that):

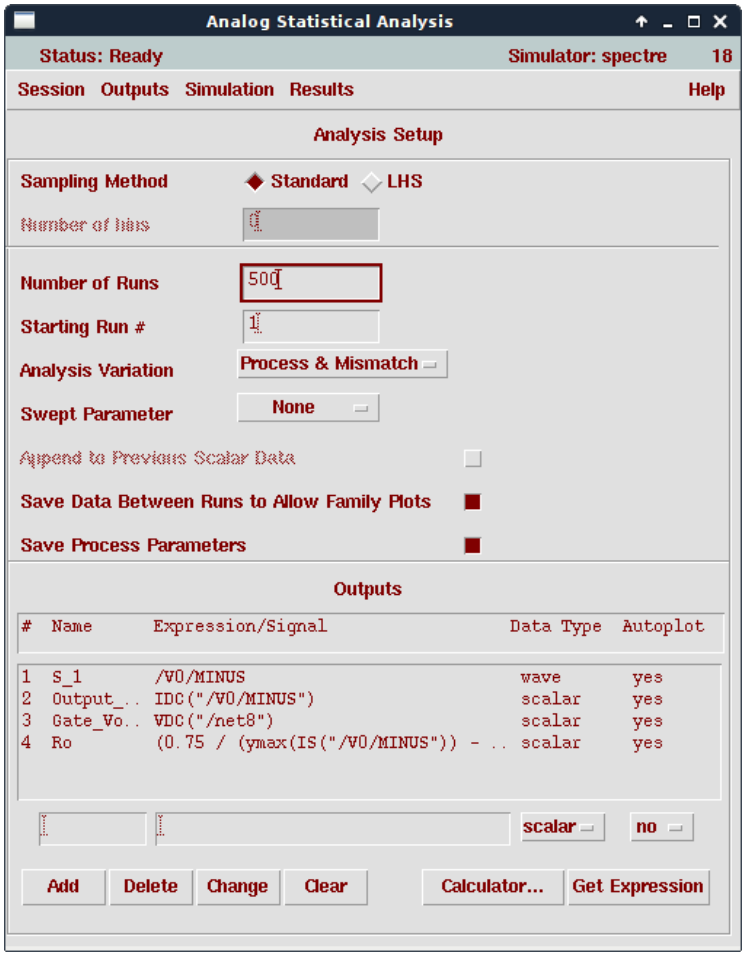


V0/MINUS should be saved (to estimate output resistance).

5. Open the MC's **Analog Statical Analysis Form** by clicking **Tools -> Monte Carlo.**



6. Fill in the form as shown below. Remember to check the box to save data between runs to allow family plots.



Once you have finished the data entry, click **Simulation -> Run.** Initially you can try a smaller number of runs to make sure everything works as intended. Normally 1000 simulations are required to produce meaningful statistical results.

If Analysis Variation is set to “Process Only”, parameters change but are the same for all devices in each run. If “Process and Mismatch” is selected, each device is assigned a different set of random parameters.

4. Results