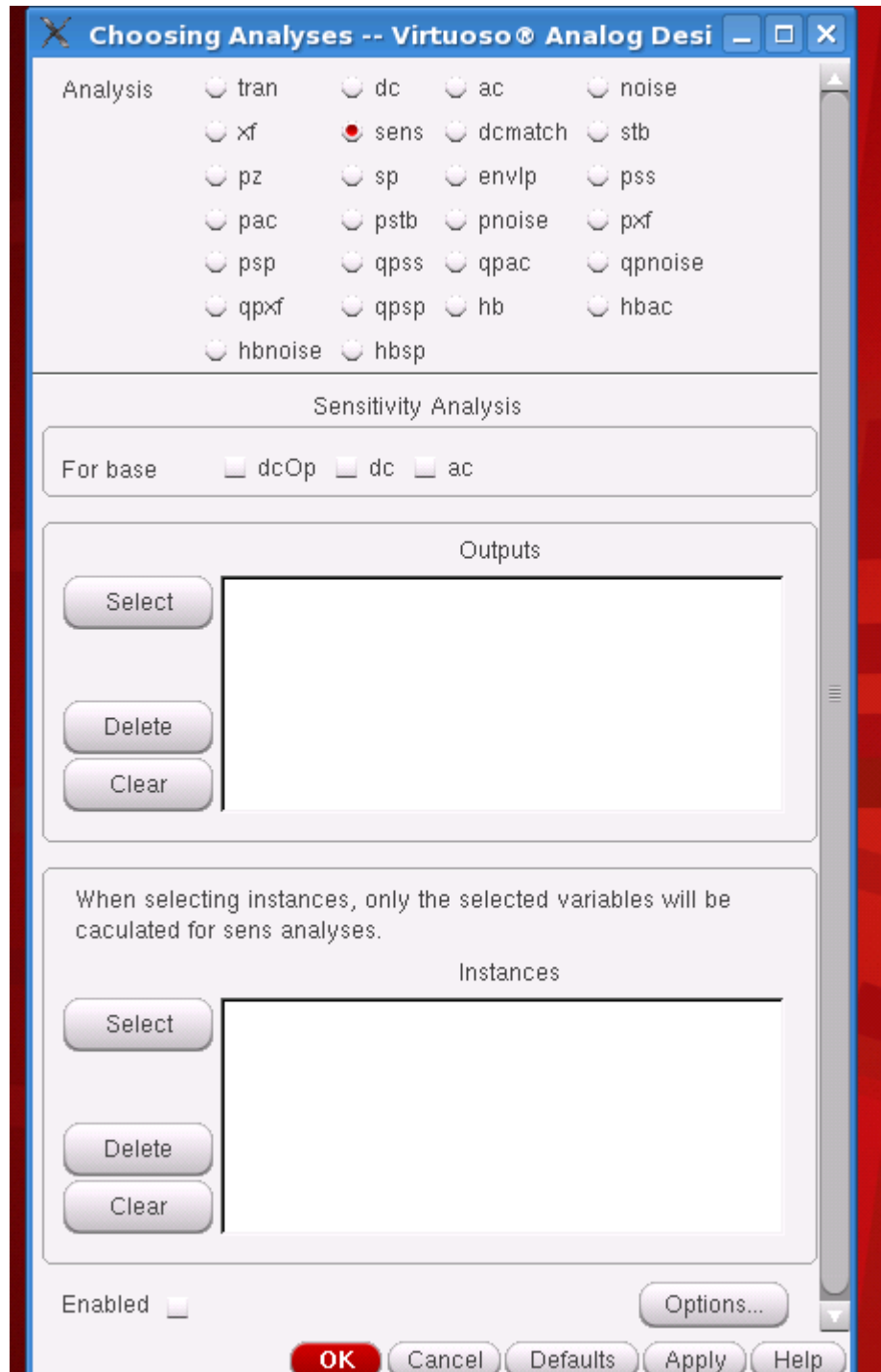


## Sensitivity Analysis

Sensitivity analysis lets a designer see which parameters in a circuit most affect the specified outputs. It is typically used to tune a design to increase or decrease certain design goals. You might run a sensitivity analysis to determine which parameters to optimize using the optimizer.

- Choose the sens radio button on the Choosing Analyses form. The form redraws:



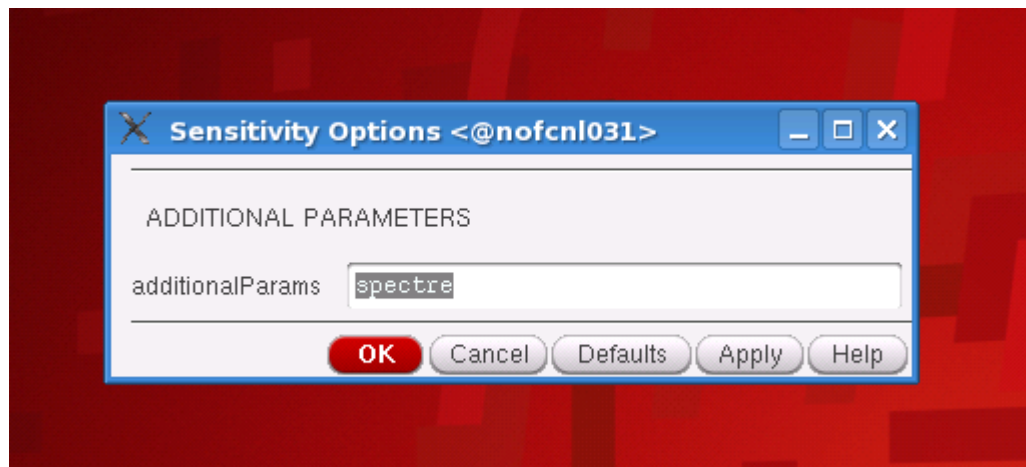
- Choose which types of sensitivities you want to calculate.

In the 'For base' field, choose any of the analyses on which you want to perform a sensitivity analysis. The available analyses are dcOp (DC operating point), dc, and ac. Before you run a sensitivity analysis, you must run the corresponding base analysis.

- Click Select to select the outputs you want to measure.

Select prompts you to select outputs by clicking on their instance in the schematic. Outputs can be any nets or ports. When you click Select, the Schematic window moves to the front of the screen. The Schematic window must be open before you can select any outputs. Use the Esc key to end selection.

- Click Options to set the spectre options controlling sensitivity analysis.



Type a filename in the sensfile field to specify a filename for the Spectre sensitivity results. This file is in ASCII format, and is generated in the psf directory. If you do not specify a value, the file is named sens.output by default.

- View your results.

The results will be saved in sens.output file, Which will be inside the below shown path.

Homedirectory→simulation→Inver\_test→spectre/schematic/psf.

Inside psf folder you can find the sens.output file which shows the sensitivity of your circuit.