Corner Simulations

This document illustrates the approach to perform corner simulations for the OPAMP/System. Before starting with these steps, you are expected to have met the specification under nominal conditions.

Corners are used to verify the circuit performance under the extreme case conditions. If the performance is met in there, the confidence is high that the circuit will meet the requirements under all possible circumstances. A commercial used design is expected to be successful in all corners. Within the scope of this lab, your design must meet the specification for the nominal case and the majority of corners. It is not mandatory to reach the specifications in all corners, although a high number of successful corners (ideally all) will be beneficial for the evaluation of your work. In general, the following corners need to be considered for the extreme cases:

1. Process Corners:
    - MOS (ss, snfp, tt, fnsp, ff)
    - Capacitor (ss, typ, ff)
    - Resistor (ss, typ, ff)
2. Voltage Corners: 1.62 V, 1.8 V, 1.98 V (± 10% variation)
3. Temperature Corners: -25°C, 25°C, 85°C

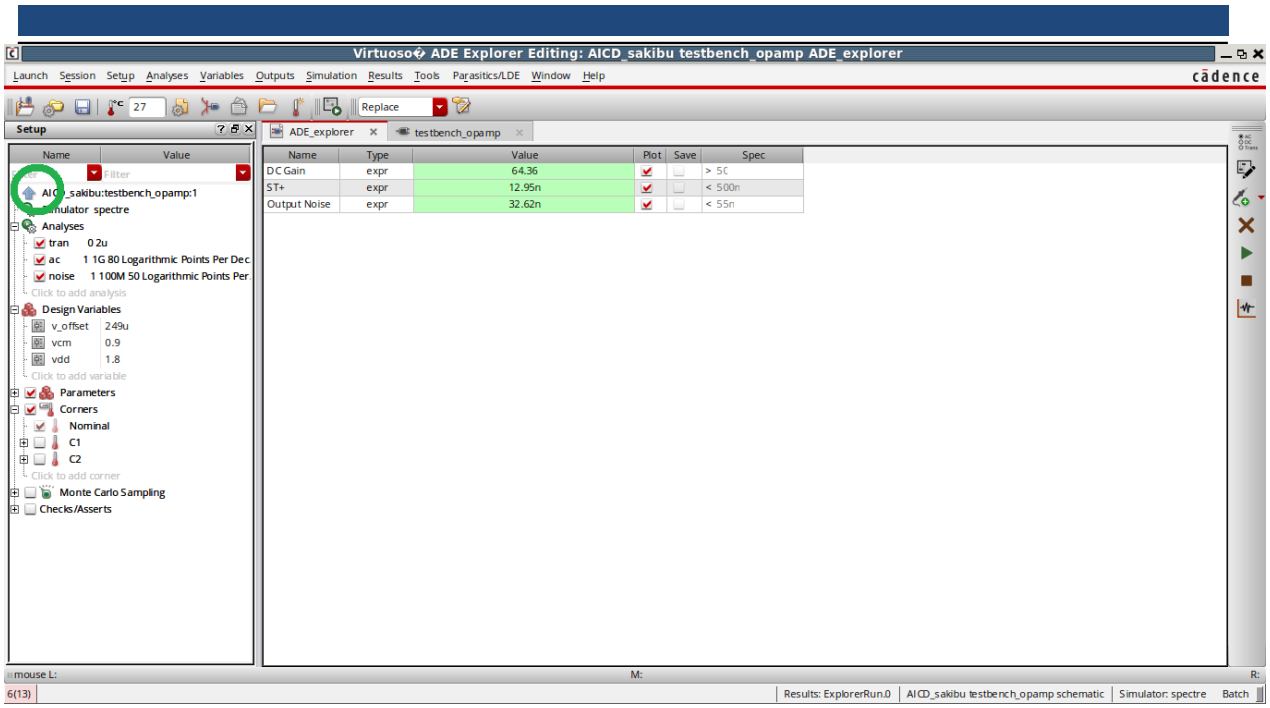
Overall this leads to 5*3*3*3*3 = 405 possibilities. To reduce the simulation time for this lab, we will only use the following extreme corners: MOS (ss, snfp, fnsp, ff); Cap (min, max); Resistor (min, max); Voltage (1.62, 1.98); Temperature (-25°C, 85°C). The following Outputs are to be calculated for the Corner Simulation runs:

1. DC Gain
2. Gain Bandwidth
3. Phase Margin
4. Slew Rate +ve
5. Slew Rate –ve
6. Settling Time (high)
7. Settling Time (low)
8. Offset

Step 1

From library manager of cadence, open your ADE Explorer file that has the tests on which you want to run the corner simulation.

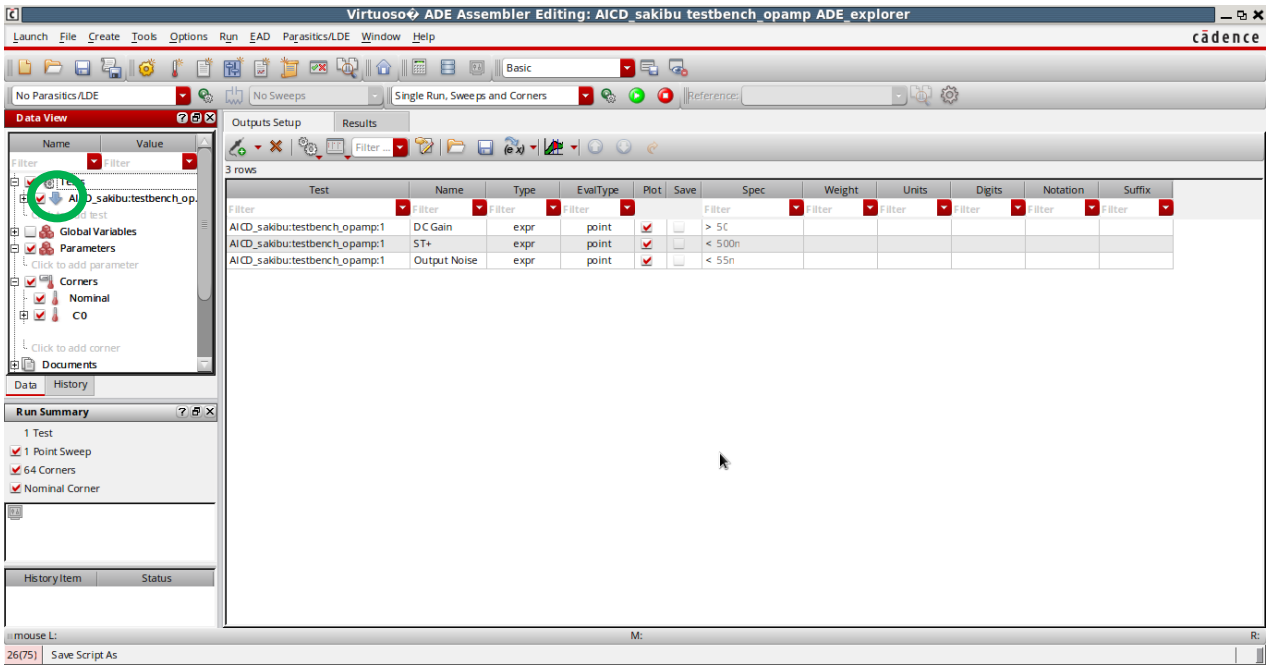
The ADE Explorer will look similar to the example below (In this example we have ac analysis: DC Gain, transient analysis: Settling time low to high and noise analysis for measuring Output noise):



Now click the blue arrow (marked in green circle).

Step 2

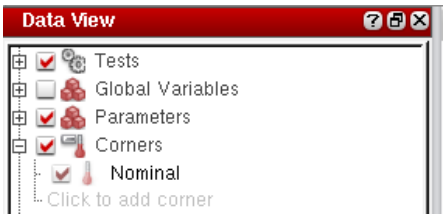
It should now bring the ADE Assembler Editing window similar to the one below.



Note that, you can always go back to ADE Explorer. If you want to go back, just from the menu, click on the blue arrow besides the test (marked in green circle). You can also access the explorer via the **Launch -> ADE Explorer** menu.

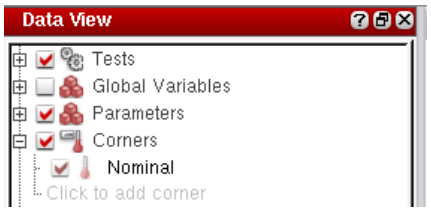
Step 3

Now in the **Data View tab** uncheck the **Global Variables** section.

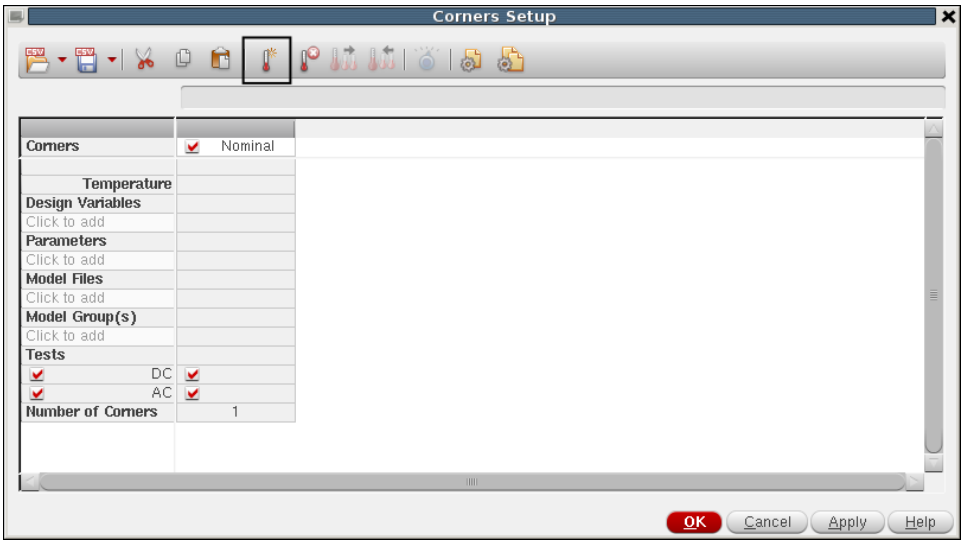


Now, click on the **'+' option** available before the **Corners section** to setup the required corners for the corner simulation simulations, in the following way:

+ -> Corners -> Click to add corner

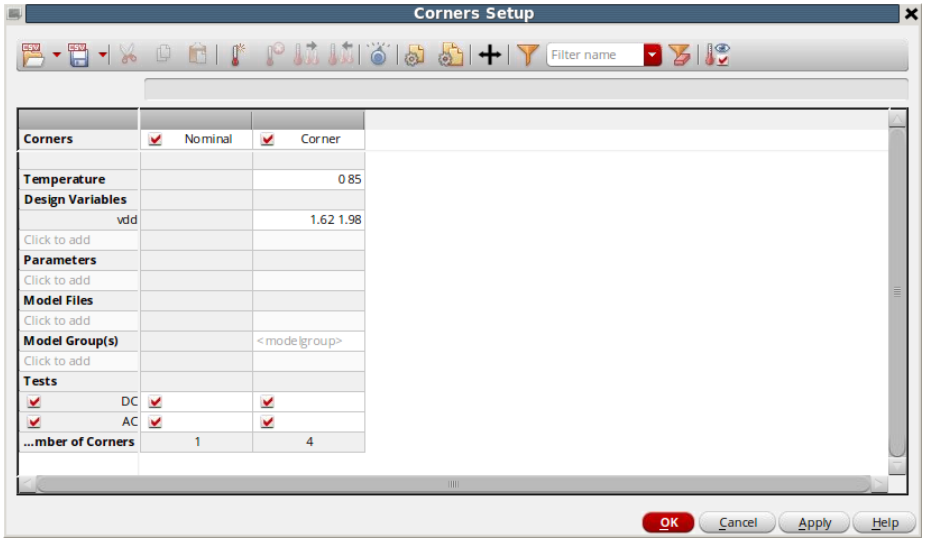


Once you click on the **Click to add corner** option the **Corners Setup** window opens up:



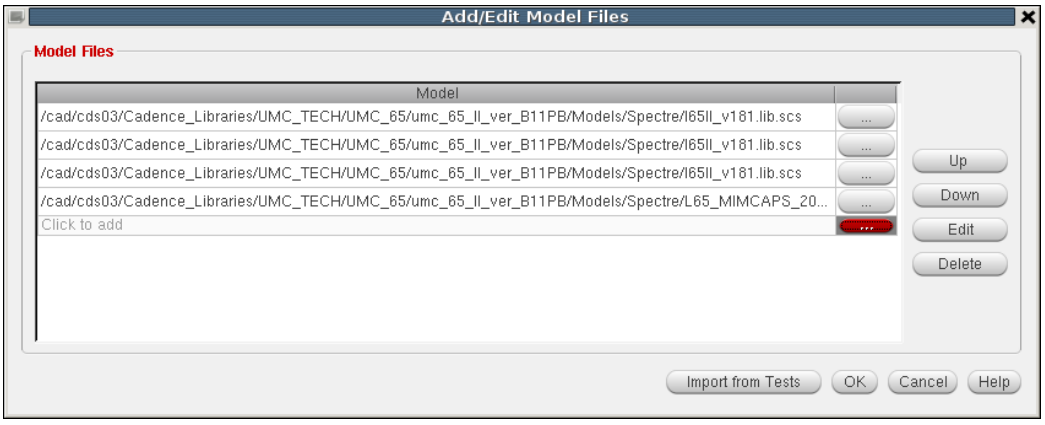
Click the highlighted option to start adding corners. Next, click the **click to add** option under **Design Variables** section and in the drop down menu select **All** to add all the variables at once.

Once all the required variables are added, setup the window as follows for the temperature and voltage corners:



Step 4

Next, to setup the process corners we will first have to add the model library files available from UMC. For this select the **click to add option** available under the **Model Files** which opens up the following window.

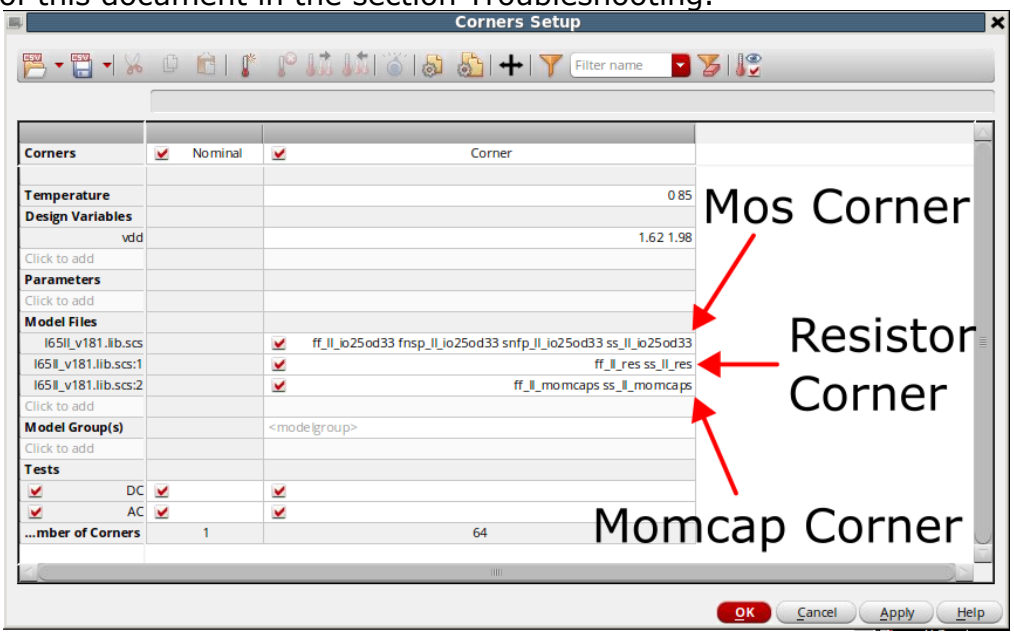


Click on the **'...'** option to browse and add model files. Both following paths are relevant:

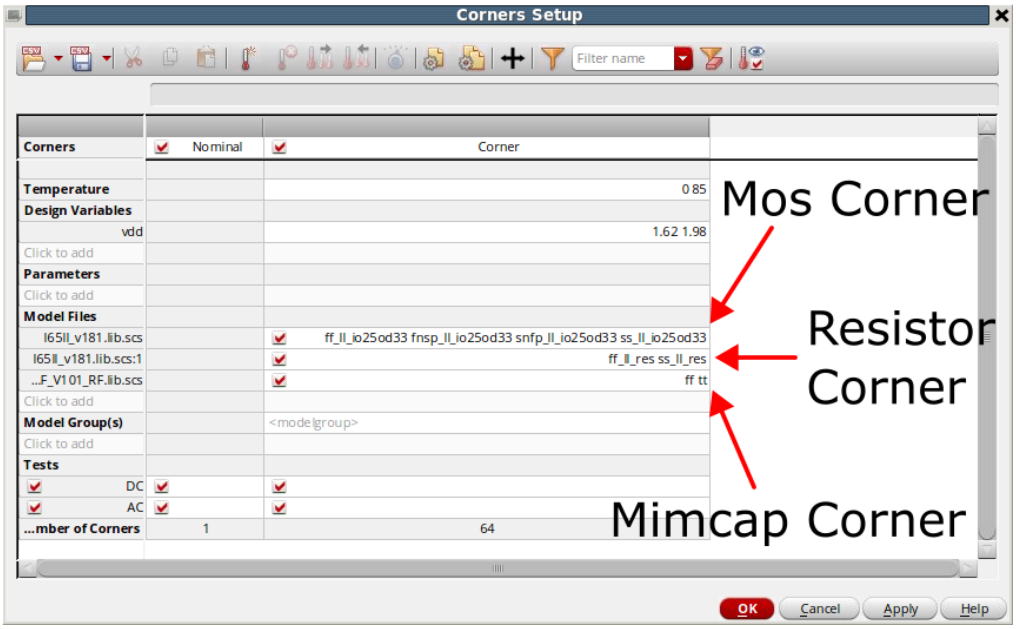
1. MOS/RES/MOMCAP:
   /cadstud/cds03/Cadence_Libraries/UMC_TECH/UMC_65/umc_65_ll_ver_B11PB/Models/Spectre/l65ll_v181.lib.scs

2. MIMCAP:
   /cadstud/cds03/Cadence_Libraries/UMC_TECH/UMC_65/umc_65_ll_ver_B11PB/Models/Spectre/L65_MIMCAPS_20F_KF_V101_RF.lib.scs

Here you have to choose the files according to your own design. All elements used in your design must be specified. So you will need to add the first path multiple times, for the MOS, RES and CAP each. For the cap, just use the type you have chosen (either MOMCAP or MIMCAP). If your design does not use a Resistor, it can be omitted too.
The added model files contain the internal model parameters for different devices and corners. So the device and corner which should be used by the simulator must be specified explicitly. For the MOSFETs add "**ss_ll_io25od33 ff_ll_io25od33 fnsp_ll_io25od33 snfp_ll_io25od33**" in the row of the "l65ll_v181.lib.scs model". For the resistor add "**ff_ll_res ss_ll_res**" to the next "l65ll_v181.lib.scs:1" line (the index :1 at the end indicates, that this is a duplicate entry, which is ok here, DO NOT ADD THIS AT THE MODEL PATH. Depending on your design, you can use either MOMCAPS or MIMCAPS. For the MOMCAPS add next to the third line of the "l65ll_v181.lib.scs:2" add "**ff_ll_momcaps ss_ll_momcaps**". For the MIMCAPS use the "L65_MIMCAPS_20F_KF_V101_RF.lib.scs" file instead and "**ff ss**". A compact overview of the relevant files and corners definition is provided at the end of this document in the section Troubleshooting.

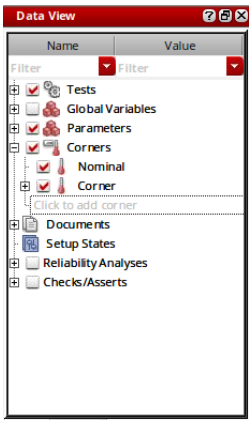


Example for MOMCAPS



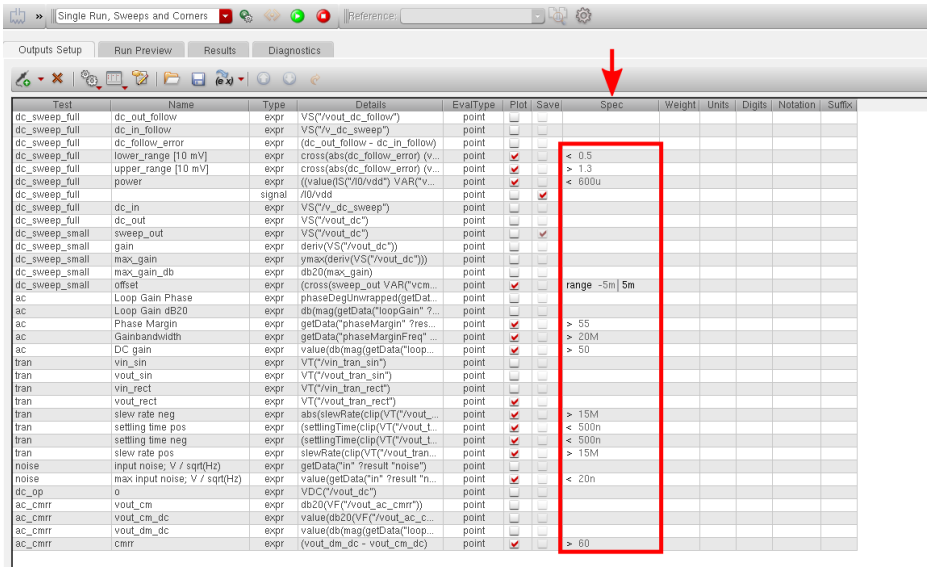
Example for MIMCAPS

These are just examples, depending on your individual design, the configuration might vary. For individual setup, please refer to the Troubleshooting section at the end.
Next, click ok and the setup corner shows up as follows:



Step 5

To make the interpretation of results easier, click on the **Outputs Setup** tab and add the required specifications for the parameters being simulated.



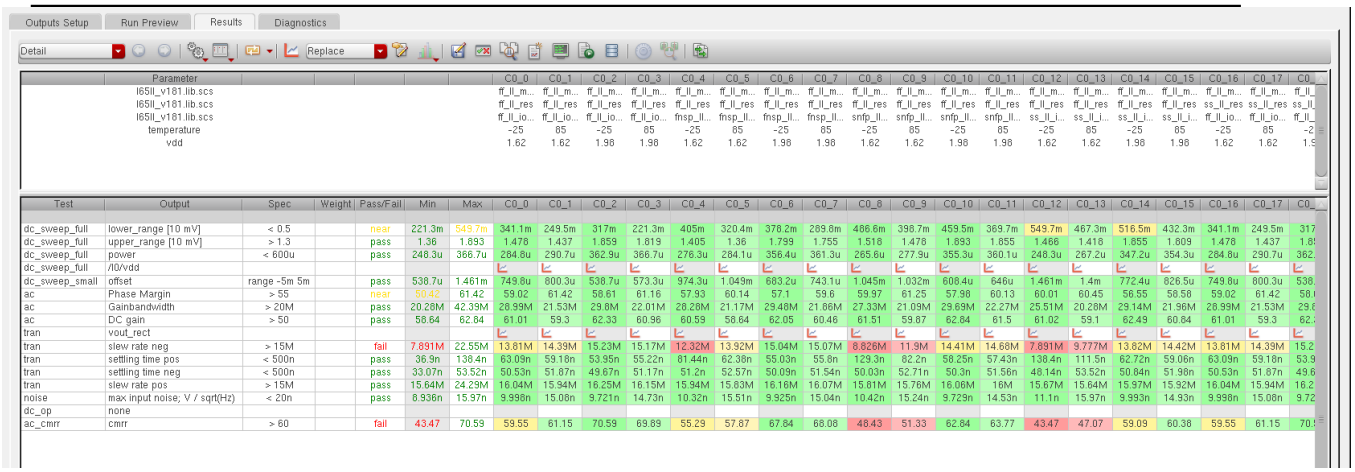
The addition of these specifications under the **Spec** tab would allow you to easily distinguish between passed (green), failed (red) and near (yellow) corners.

With this the setup for corner simulations is done and the analysis can now be run using,

Run -> Single Run, Sweeps and Corners …

Sample corner simulation results:

The color highlighting provides a quick overview on the critical corners and design targets.



Troubleshooting:

Since you are manually specifying the Model Files for the simulator, the default settings are ignored. So you have to make sure, that **all** elements used by your design are specified within your Model File list. If the setup with the model libraries is incomplete, you might encounter a similar error message:

> ERROR (SFE-23): "input.scs" 54: The instance `R0' is referencing an undefined model or subcircuit, `RNHR_LL'. Either include the file containing the definition of `RNHR_LL', or define `RNHR_LL' before running the simulation. Error found by spectre during circuit read-in.

In this example an element **R0** of the type **RNHR_LL** is specified in the schematic, but no definition for the resistor type **RNHR_LL** is found by the simulator. To solve this issue, you have to specify at least one case for the resistor (**tt_ll_res**, **ff_ll_res** or **ss_ll_res**). For other possible devices, please refer to the following table.

Element	Model File	Options
RNHR_LL	[…]/l65ll_v181.lib.scs	tt_ll_res ff_ll_res ss_ll_res
mimcaps_20f_mm	[…]/L65_MIMCAPS_20F_KF_V101_RF.lib.scs	tt ff ss
momcaps_sy_mmkf **or** momcaps_as_mmkf	[…]/l65ll_v181.lib.scs	tt_ll_momcaps ff_ll_momcaps ss_ll_momcaps
n_25_ll **or** p_25_ll	[…]/l65ll_v181.lib.scs	tt_ll_io25od33 ss_ll_io25od33 ff_ll_io25od33 fnsp_ll_io25od33

		snfp_ll_io25od33
The following elements should not be used by your design, but will be present in the given SAR logic		
n_12_llhvt **or** p_12_llhvt	[…]/l65ll_v181.lib.scs	tt_ll_hvt12
n_12_llrvt **or** p_12_llrvt	[…]/l65ll_v181.lib.scs	tt_ll_rvt12
n_12_lllvt **or** p_12_lllvt	[…]/l65ll_v181.lib.scs	tt_ll_lvt12
n_12_llnvt **or** p_12_llnvt	[…]/l65ll_v181.lib.scs	tt_ll_nvt12
If you experience other missing elements, you will likely use the wrong devices in your schematic		

Replace […] with /cadstud/cds03/Cadence_Libraries/UMC_TECH/UMC_65/umc_65_ll_ver_B11PB/Models/Spectre