

Corners

Corners define differences due to process inaccuracies, temperature and other parameter variations. It is clear that simulations that take these differences into consideration will differ one from another.

Corners that describe differences due to process inaccuracies (such as doping variations) are supplied with the process kit and usually are located in models library. For example the kit can include corners for: Fast nmos Fast pmos, Slow nmos Slow pmos, Fast nmos Slow pmos, Slow nmos Fast pmos, Typical nmos Typical pmos.

There is also possibility that corners will describe IC's behavior in different temperatures and other parameter variations, such as vdd variations (in this case vdd has to be a variable in schematic).

Each corner that will be simulated can contain one technology corner, one temperature value and one value for every other parameter. During corner simulation all available corners are simulated and thus influence of parameter variations on IC can be checked.

It is important to perform such simulations, because if a design meets all requirements for all technology corners available in the kit during simulation stage, the likelihood that all requirements will be met during chip test increases.

Right now there are two different ways to simulate corners in VLSI center. First one is manual however it's much easier to activate.

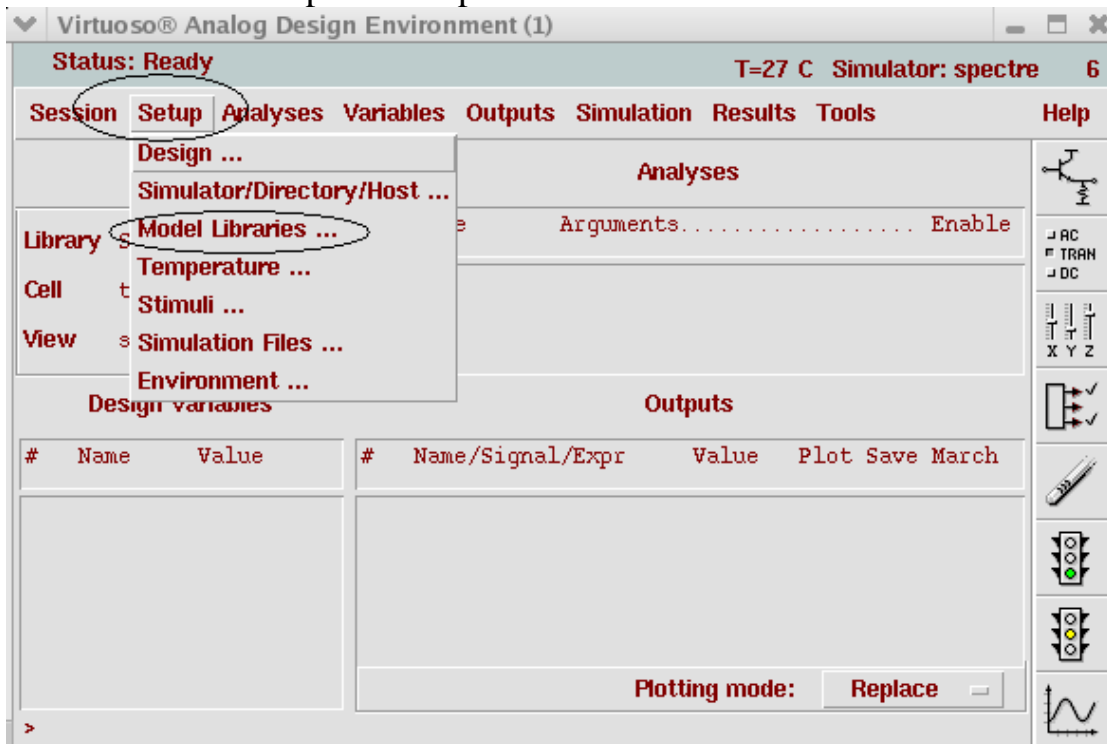
Its biggest disadvantage that you can get only one result each time.

The second way is using Vsde tool. On the one hand it's more difficult to activate ,on the the other hand the possibilities are much wider, and we can see much more than one corner result by running the simulation one time. I'll explain the ways in details.

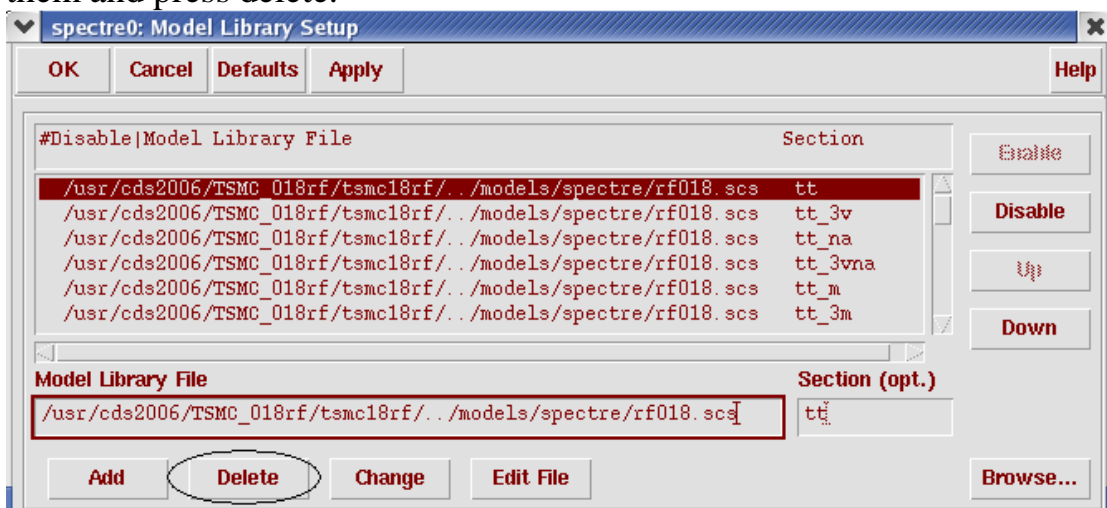
Performing corners manually

As I explained in previous section there are different models for every device for different type of simulation (ss/tt/ff/sf/fs).

In order to define the model files for simulation you should enter Model Library Setup. In order to do it you should enter Analog Design Environment. Then press Setup->Model Libraries.



In the window that will open you should erase all the files by choosing them and press delete.



In the next stage you should add five files (by pressing Add button)

incl.scs : for typical simulation

incl_ss.scs : for slow slow simulation

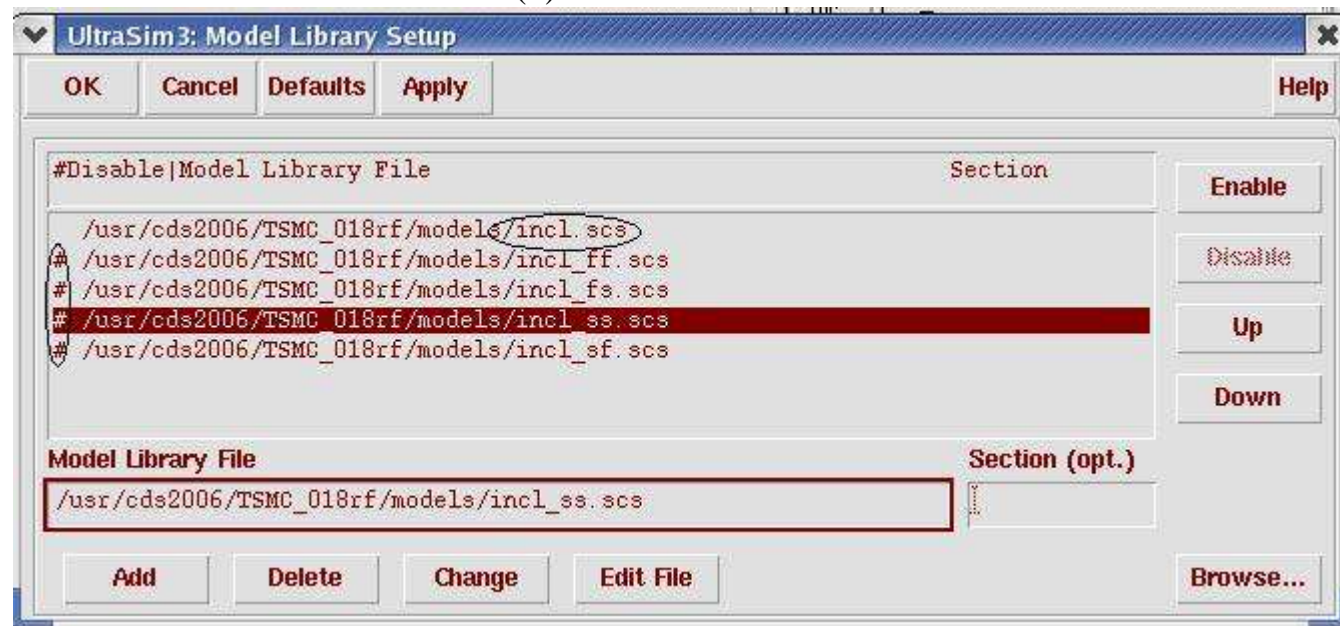
incl_sf.scs : for slow fast simulation

incl_fs.scs : for fast slow simulation

incl_ff.scs : for fast fast simulation

All these files are exist in **/usr/cds2006/TSMC_018rf/models** directory

All you have to do is to choose the incl you want for the type of the simulation and disable the others (#)



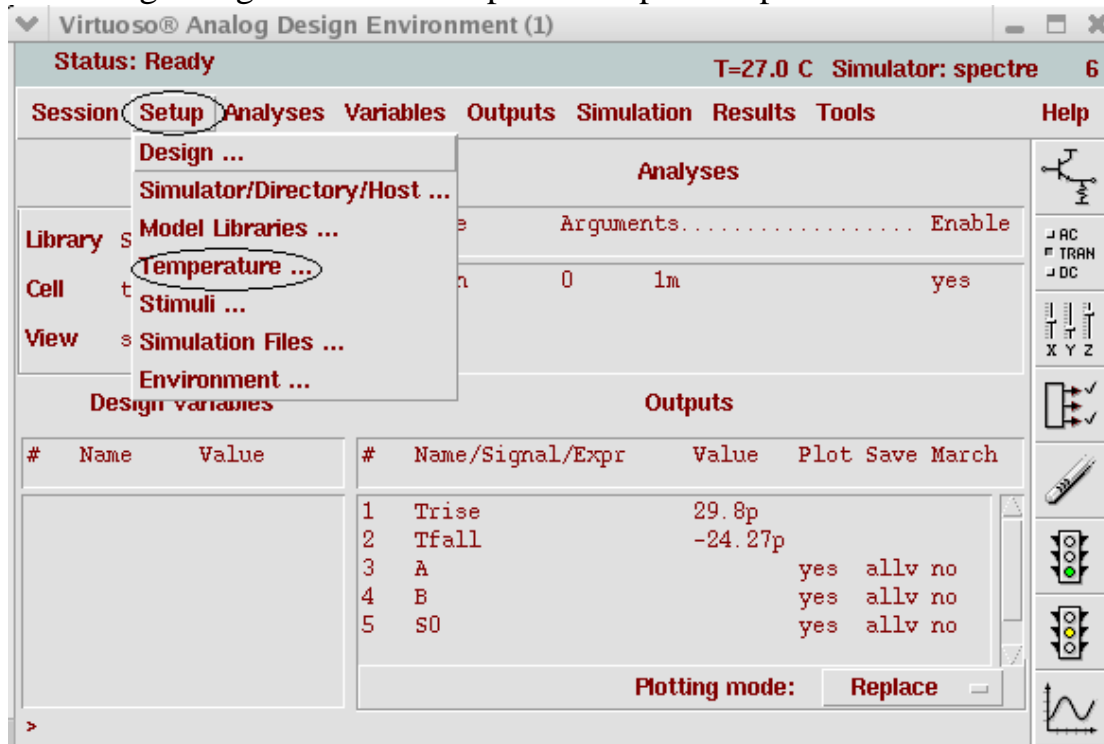
In the example above typical simulation is running.

Comment: for slow fast simulation and fast slow simulation I defined mimcap and resistors as typical since they do not support sf/ fs.

In order to expand the possibilities of corners you can change manually the temperature and supply voltage for every simulation.

Changing temperature

In Analog Design Environment press Setup->Temperature



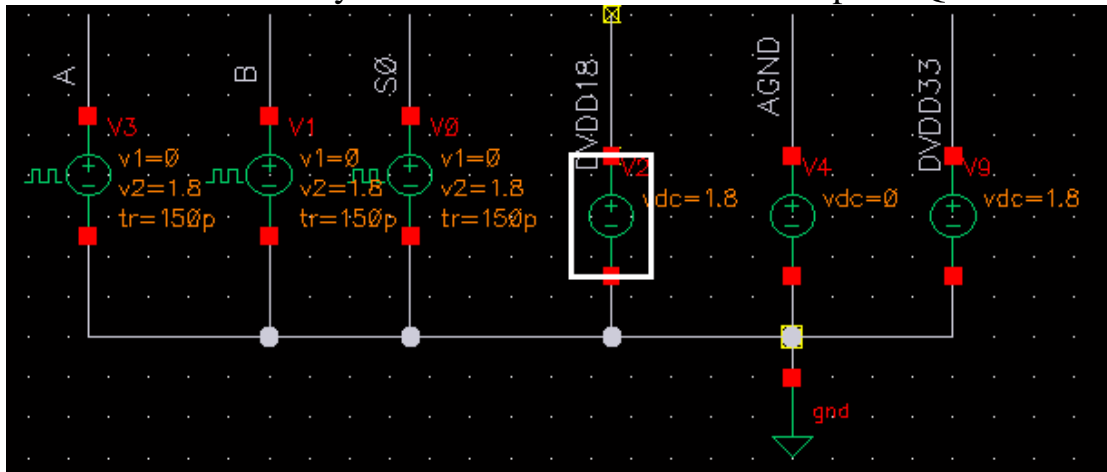
The following window will appear



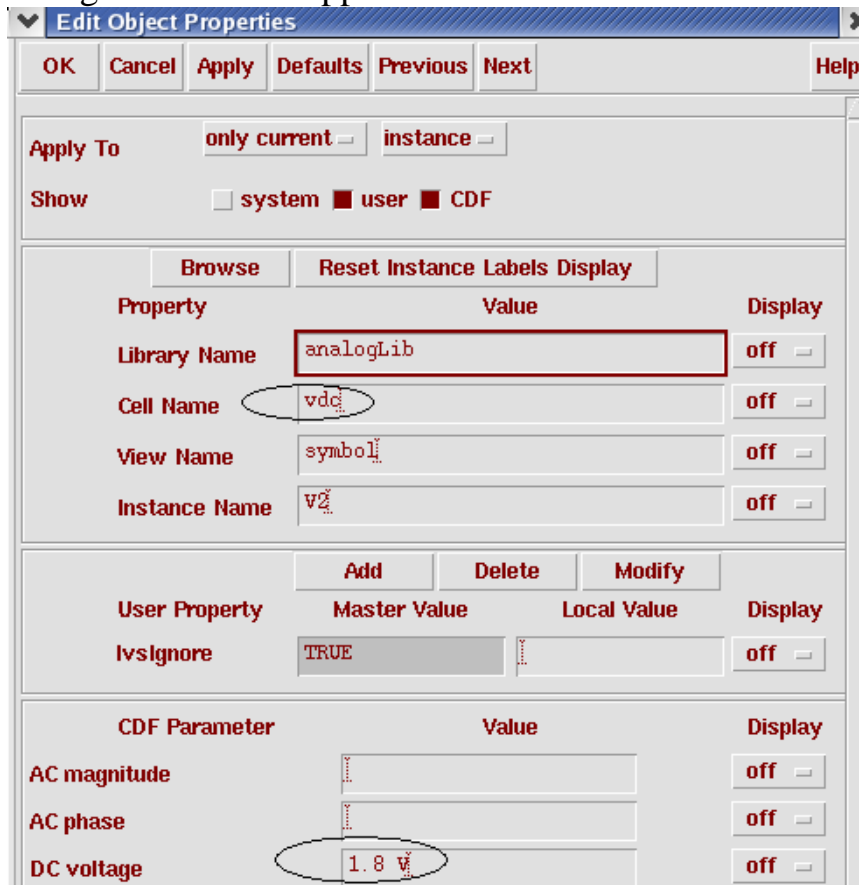
If you want you can change the scale and the Degrees. Since the purpose of corners is to accept the extreme of extreme results you should define **highest** temperature for slow slow simulation and **lowest** temperature for fast fast simulation.

Changing supply voltage

In Schematic view of your test find the **vdc** source and press Q



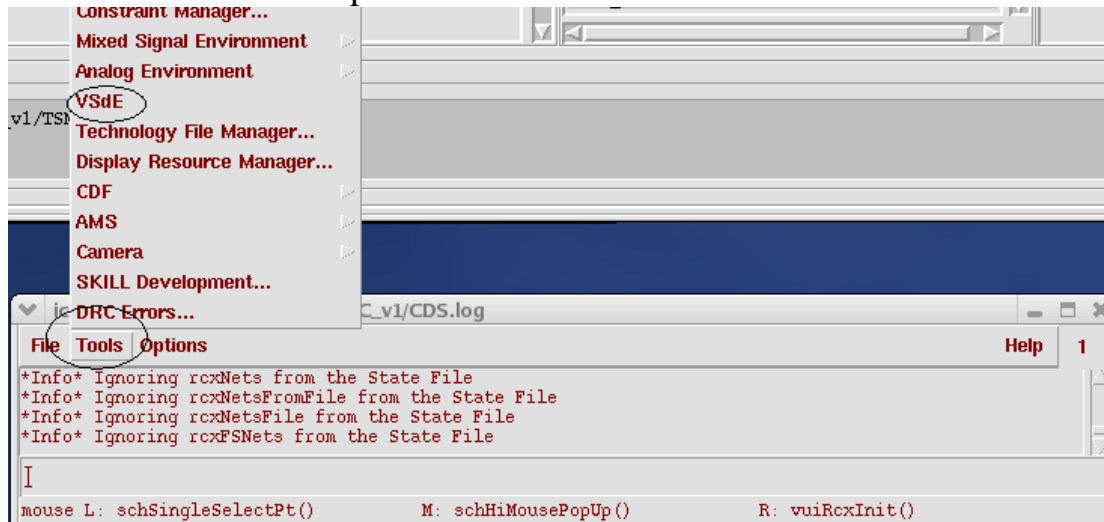
The following window will appear



Since the purpose of corners is to accept the extreme of extreme results you should define **lowest** supply voltage with **highest** temperature for slow simulation and **highest** supply voltage with **lowest** temperature for fast simulation.

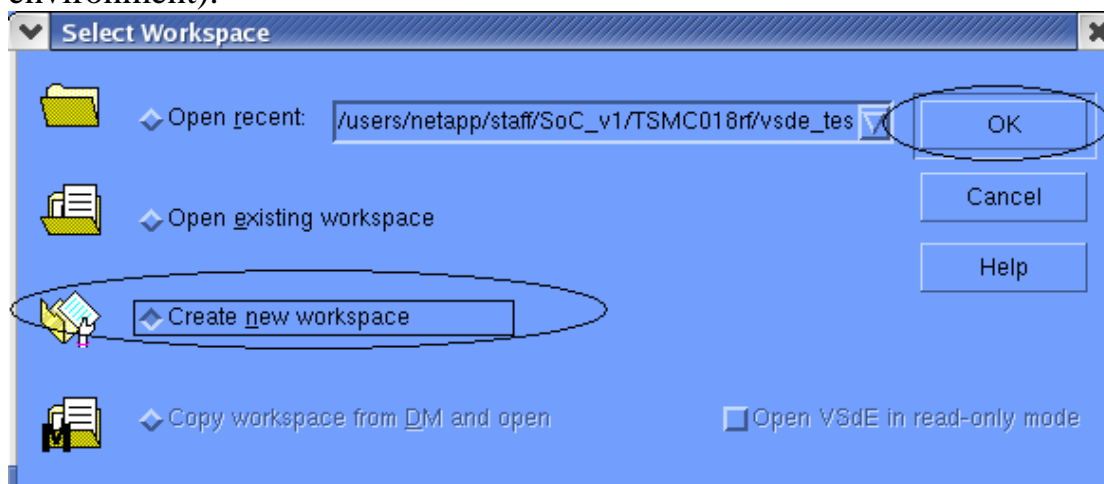
Performing corners with Vsde

From the icfb window press Tools->Vsde

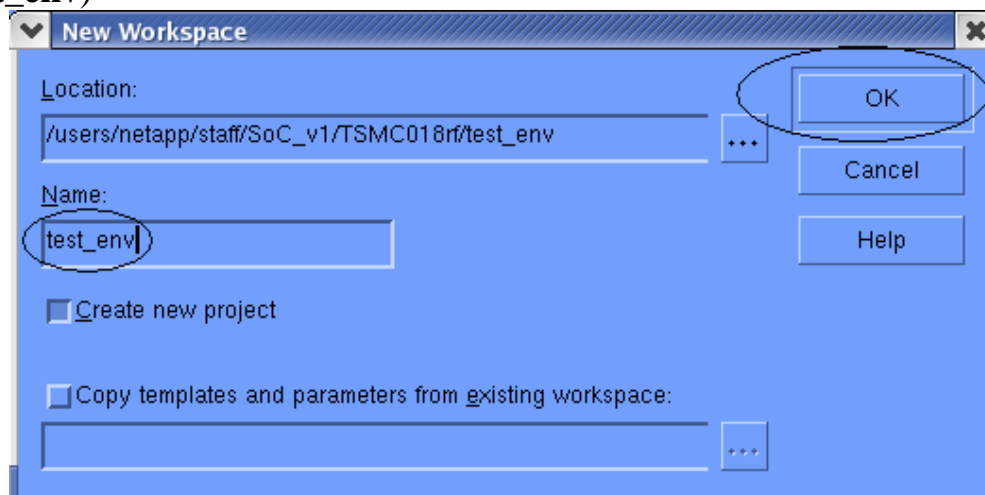


It will take about a minute to upload the tool.

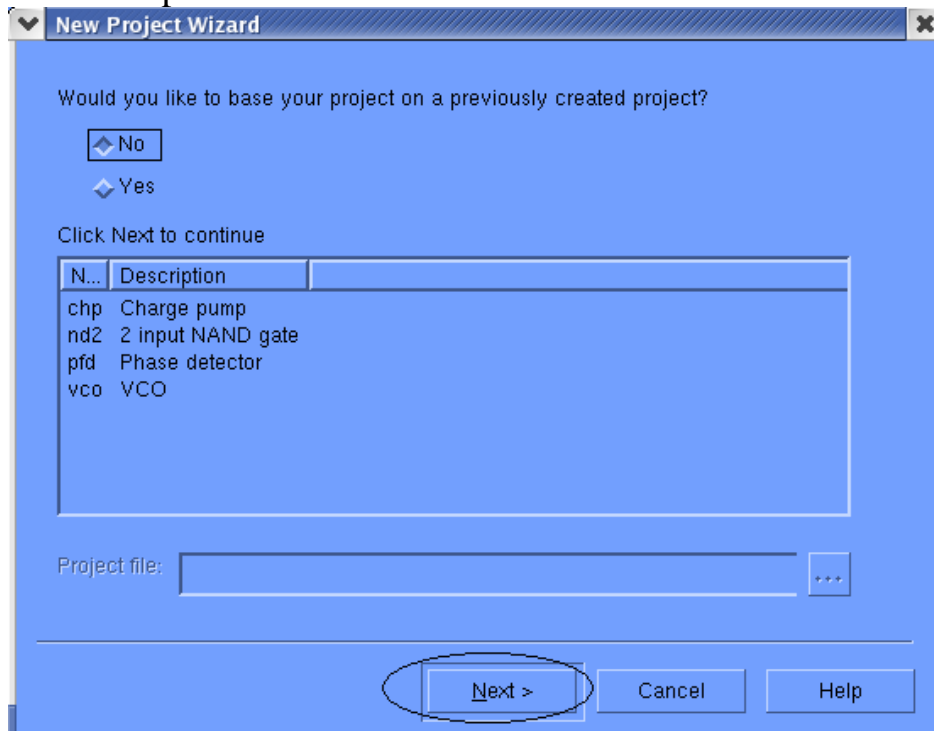
In order to start working you should create a new workspace (work environment).



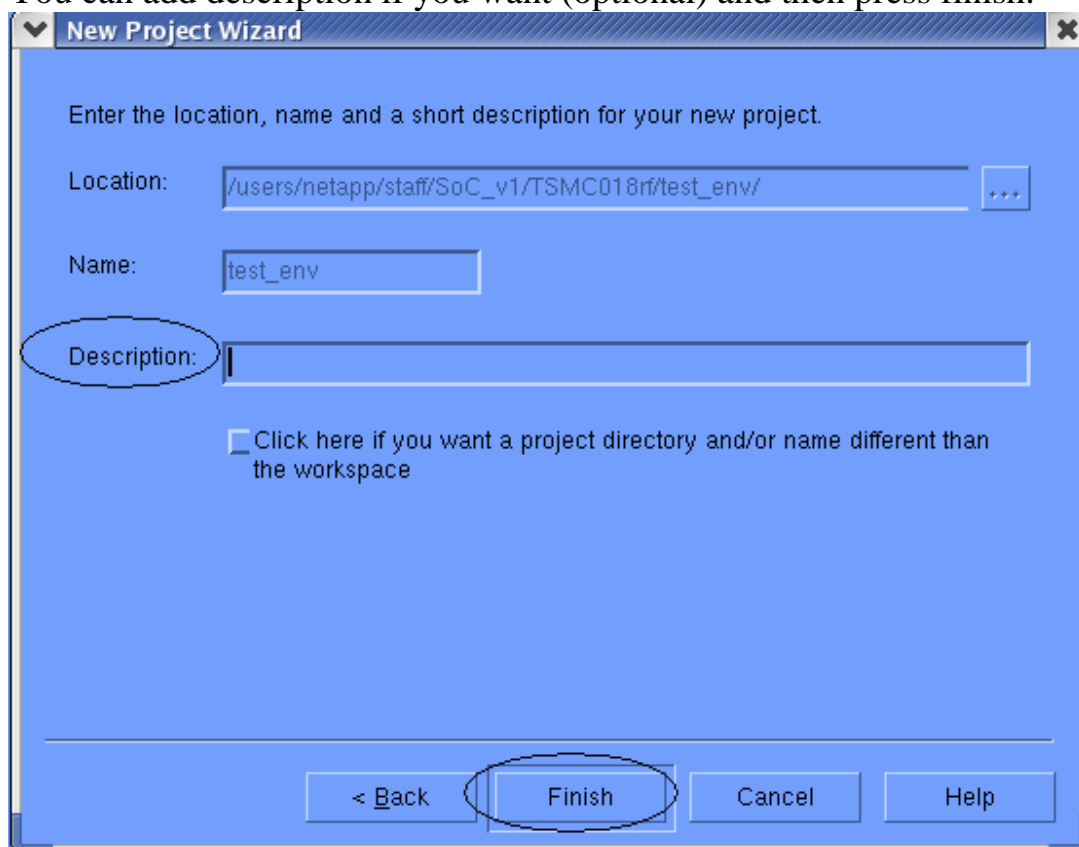
Create it under your working directory (in this example I called it test_env)



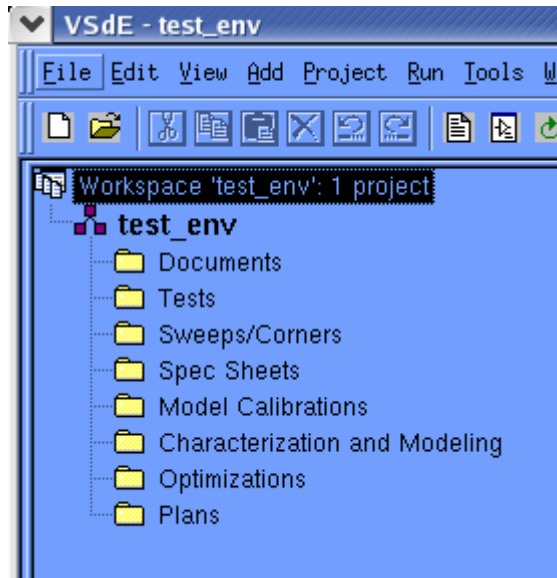
Choose No and press next



You can add description if you want (optional) and then press finish.

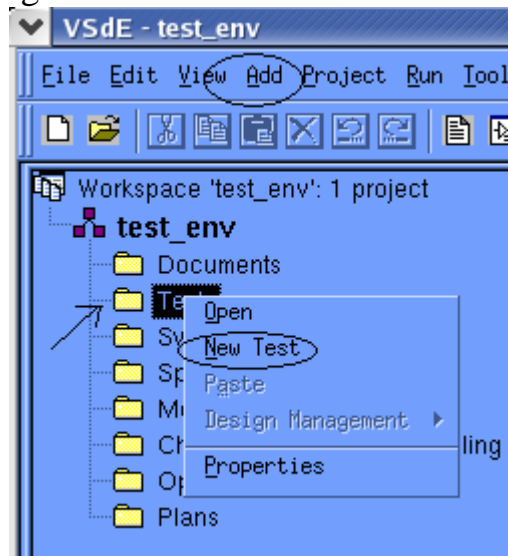


Now you've build an environment to simulate in Vsde:



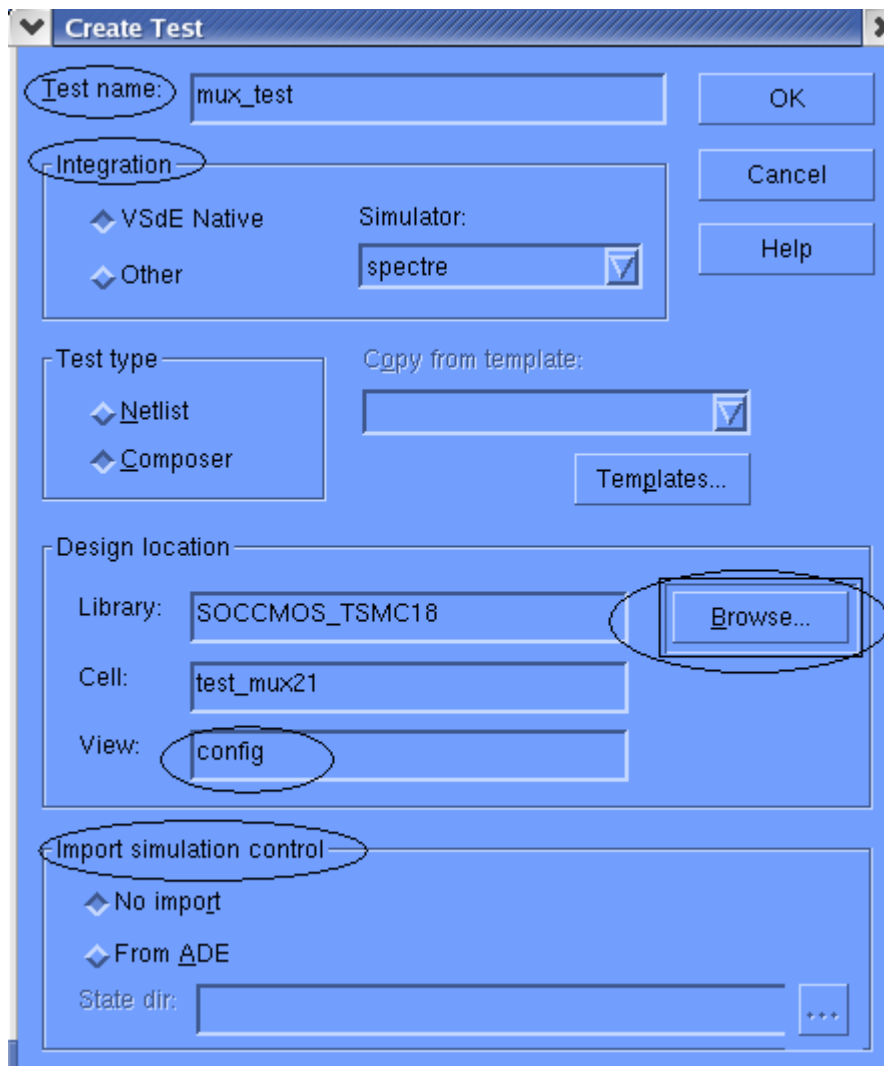
In the next stage you'll create a **new test**.

You can do it by pressing Add->New Test or just click with mouse on test directory, then right click and choose New Test

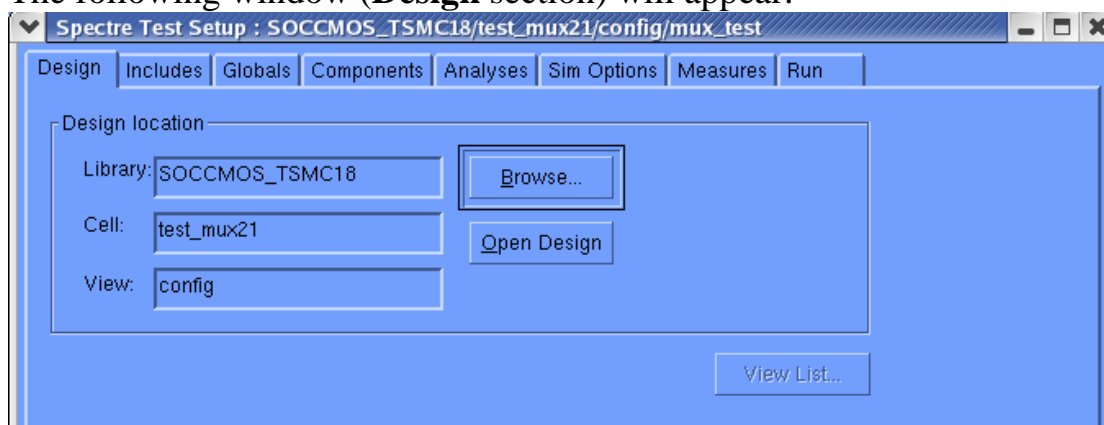


You should name your test under Test name section

In **Integration** section if you want other simulator other than spectre press other and choose other simulation (for example ultrasim). However most of the time we will work with spectre. Under **Design location** choose the cell that you built for testing. It's important to choose config view if you want to simulate post layout. Also you have an option to import your state from Analog Environment by choosing From ADE in Import simulation control and pressing ... for browsing. In this manual we'll build a test inside Vsde environment therefore we'll choose No Import in this section.

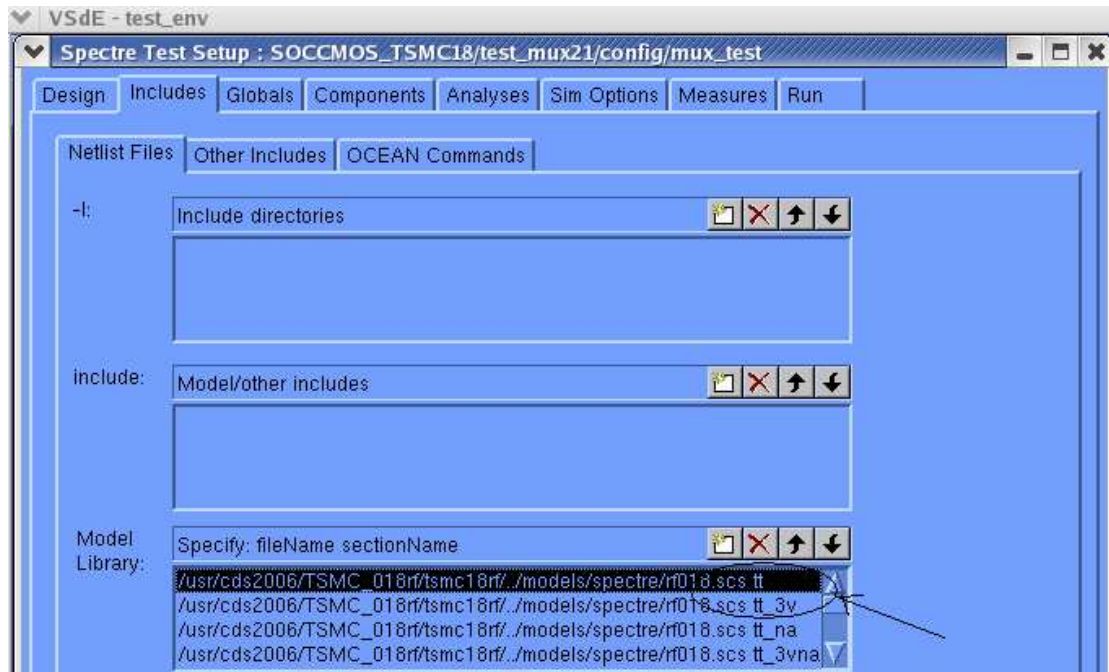


The following window (**Design** section) will appear:



By pressing Browse you can choose another cell view for testing.

The next section is **Includes**:

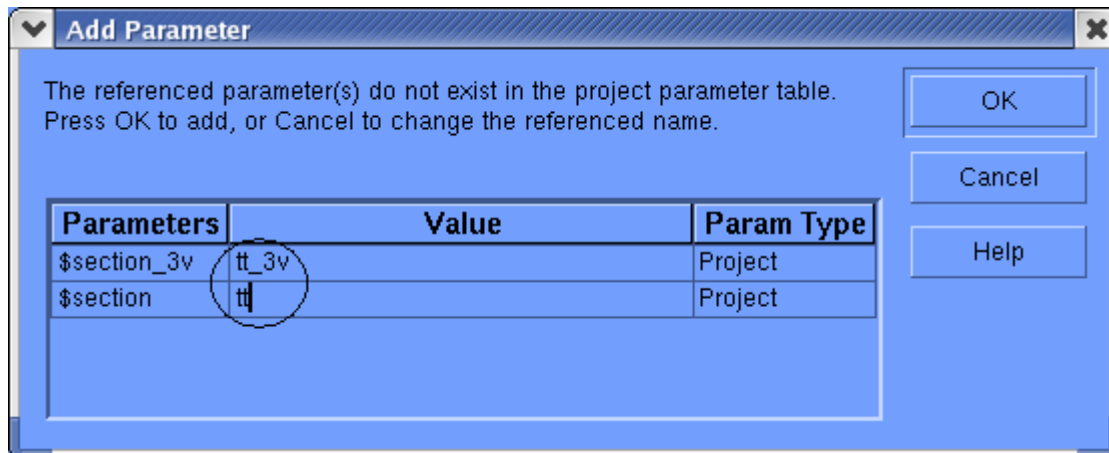


For creating corners we need to create global parameters and change the section name as following: tt->\$section, tt_3v->\$section_3v. Afterwards we will be able to change \$section, \$section_3v according to our needs.

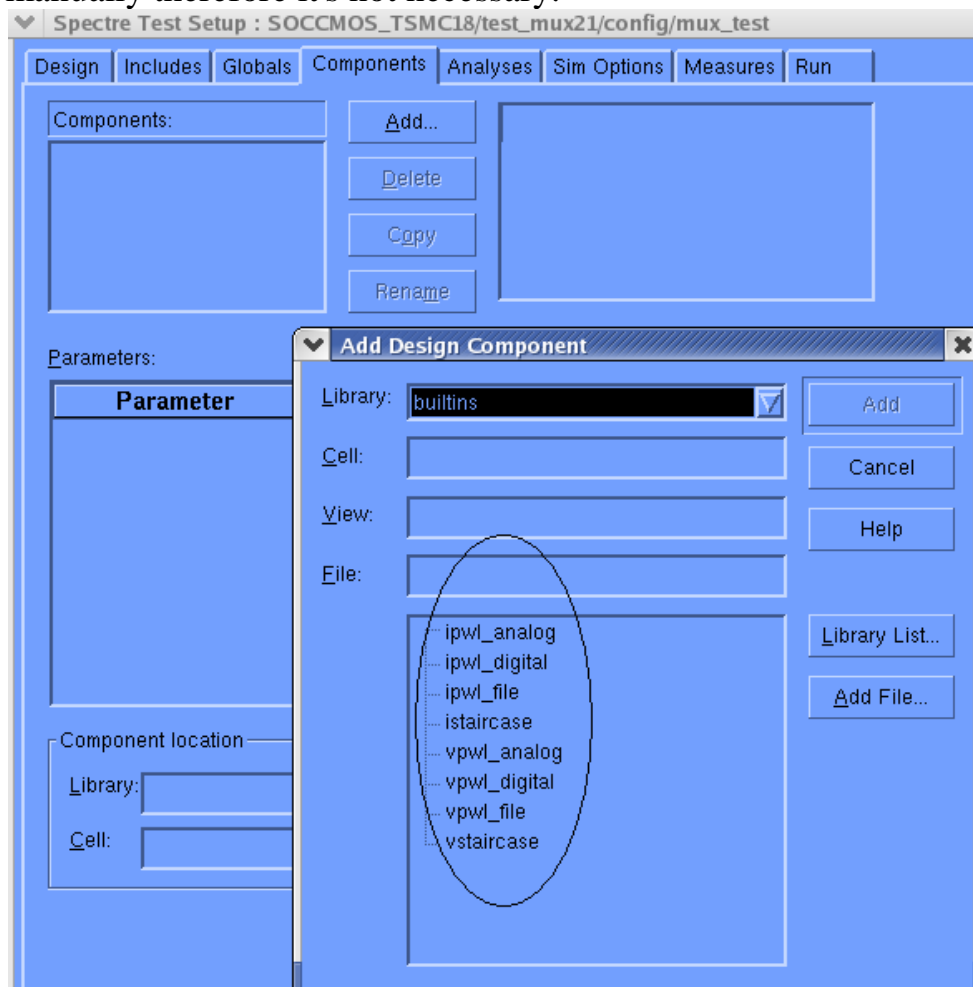


Press ... to edit each section.

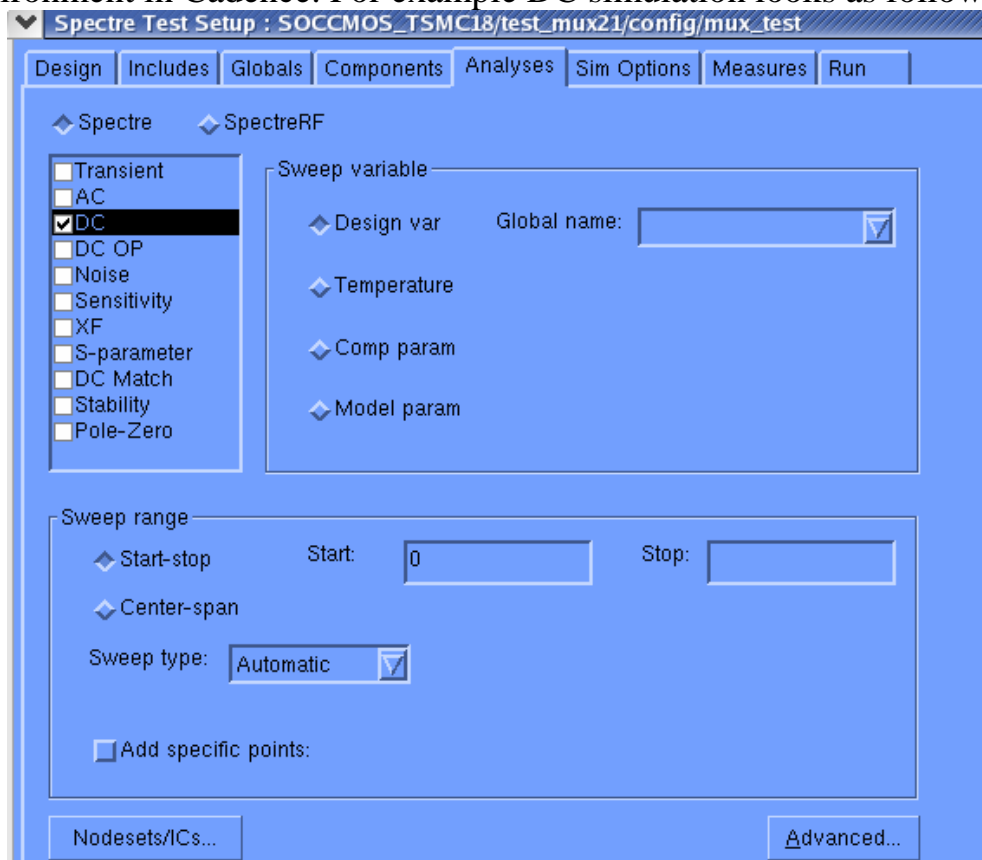
Once you'll press **Globals** a window with global parameters will appear. You have to define it's and press Ok value before you'll be able to continue.



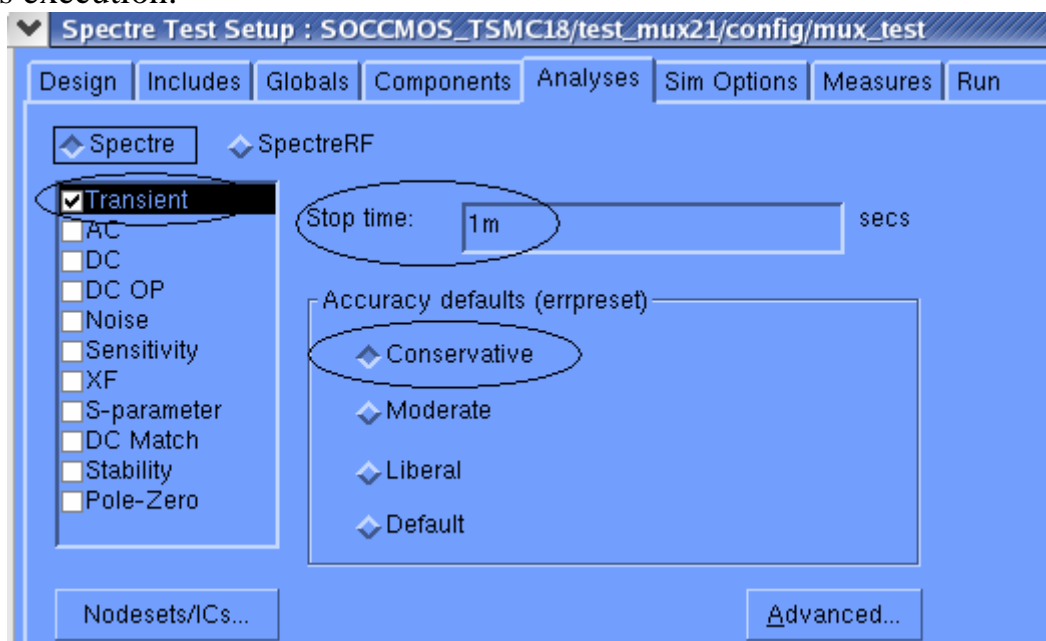
The next section **Components** is used to add different voltage and current sources to the design. However in this manual I built the schematic if the test manually therefore it's not necessary.



Under section **Analyses** we will define all the tests which are needed for simulation. The environment is very similar to regular analog environment in Cadence. For example DC simulation looks as following:



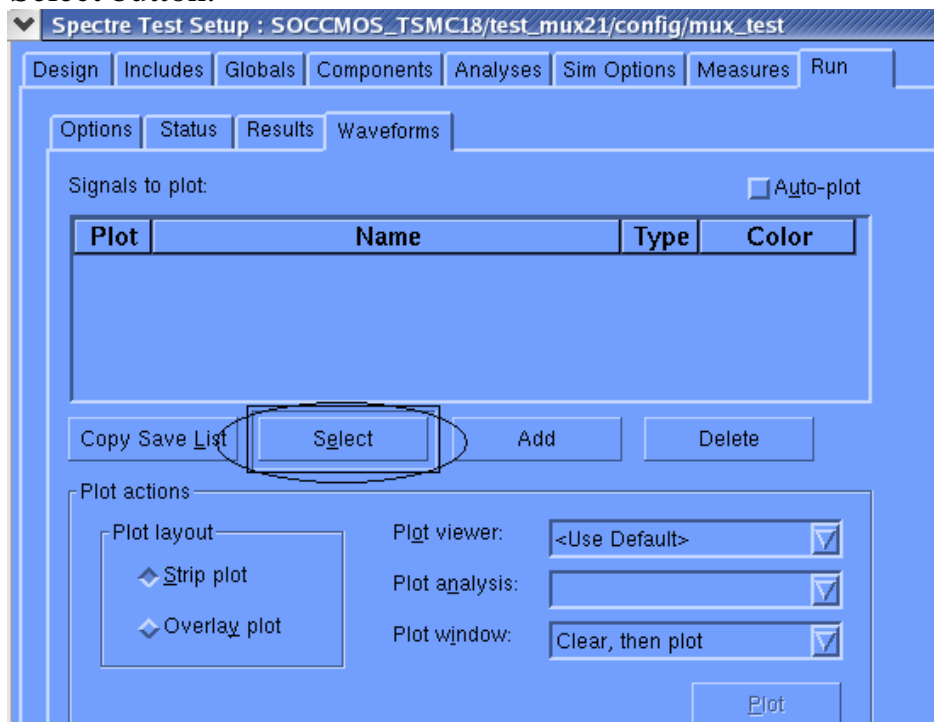
By pointing the simulation (on the left side at the top) we can run more than one simulation in parallel exactly as in Analog Environment. However we'll define Transient Simulation only for this test to simplify its execution.



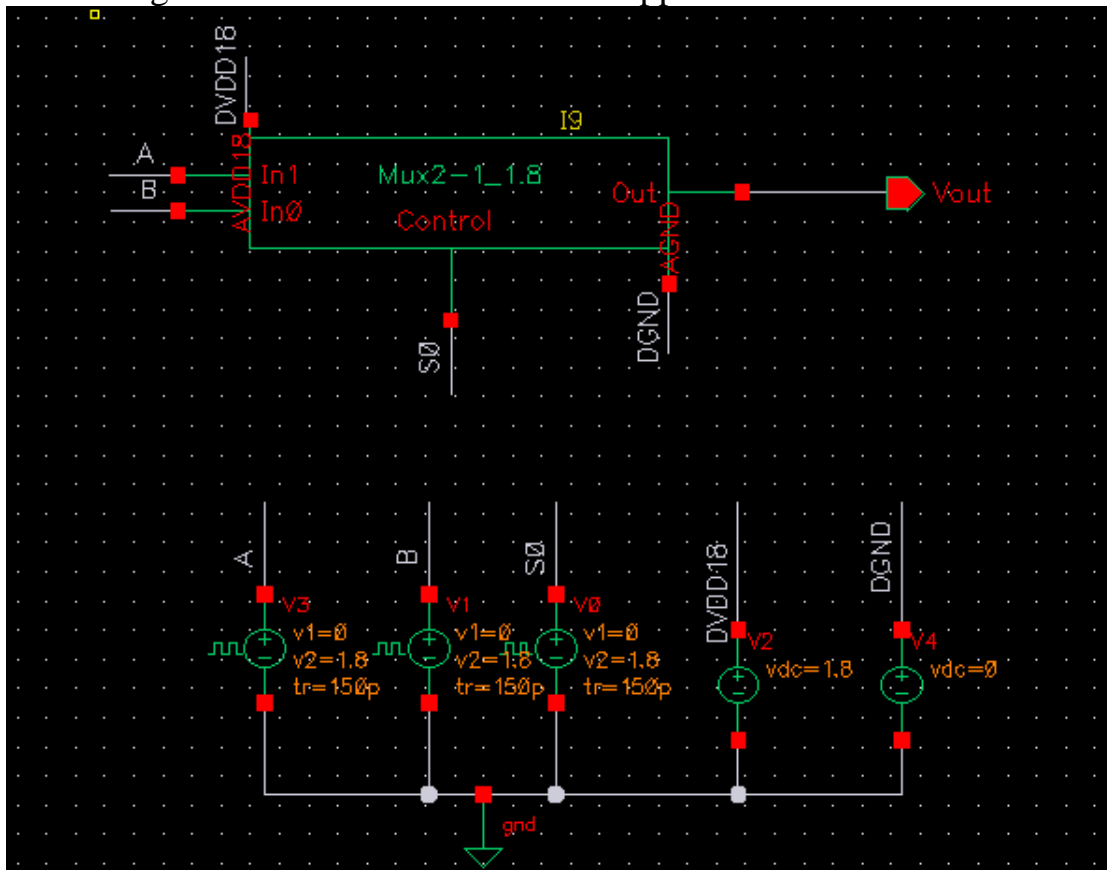
Under Sim options we can see the default temperature



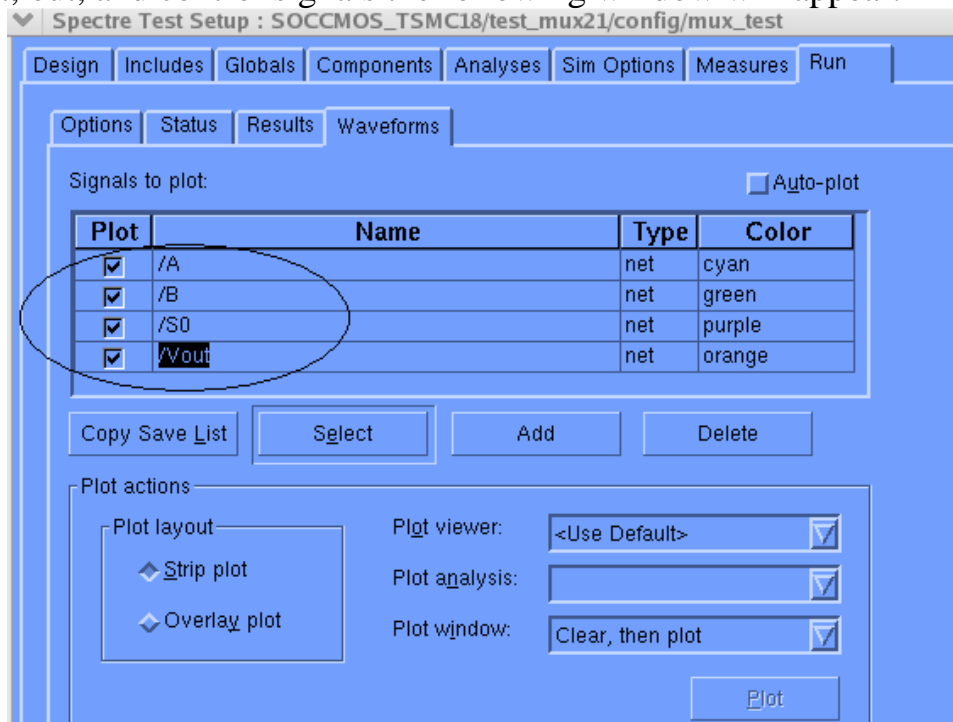
Under **Run** section, choose sub section called **Waveforms**.
Press Select button.



The config view of the test we build will appear:



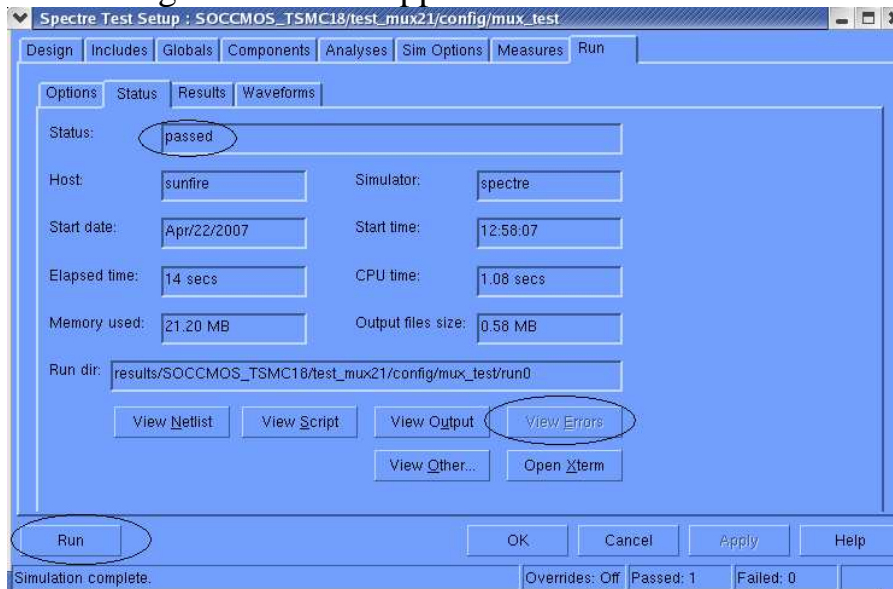
Select the signals exactly as in Analog Environment. After you'll choose input, out, and control signals the following window will appear.



You can choose in the window which signals do you want to see.

Also after running the simulation the Plot option in the right bottom of the screen will appear so you will not have to run the simulation in order to see the waveform.

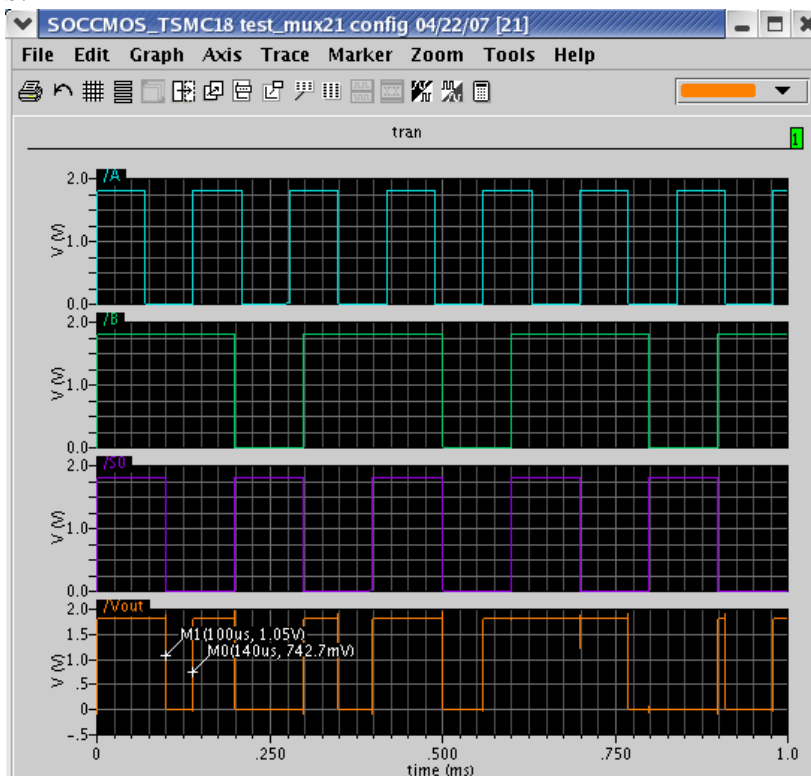
Pass to sub section called **Status** and press Run. If the simulation will pass the following window will appear:



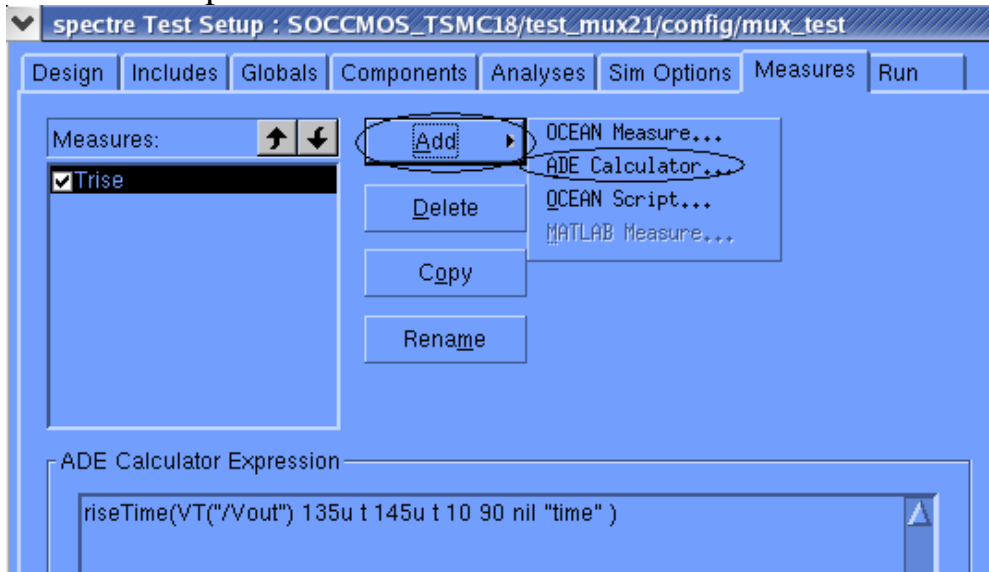
If in the Status: will appear failed the icon View errors.

Also during simulation under icon Run you can see different stages of Simulation then the final state is Simulation complete.

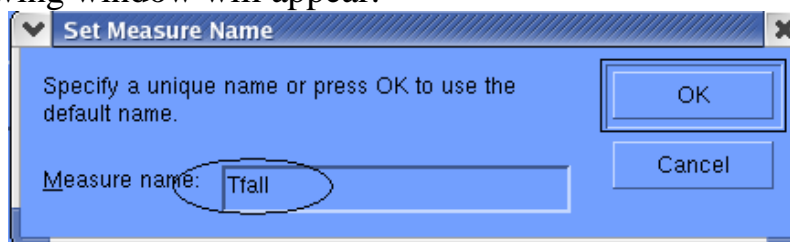
By passing to sub section **Waveforms** and pressing Plot you can view all the signals:



Now we will go back to **Measures** section and we will add Trise and Tfall. First of all press Add->ADE Calculator



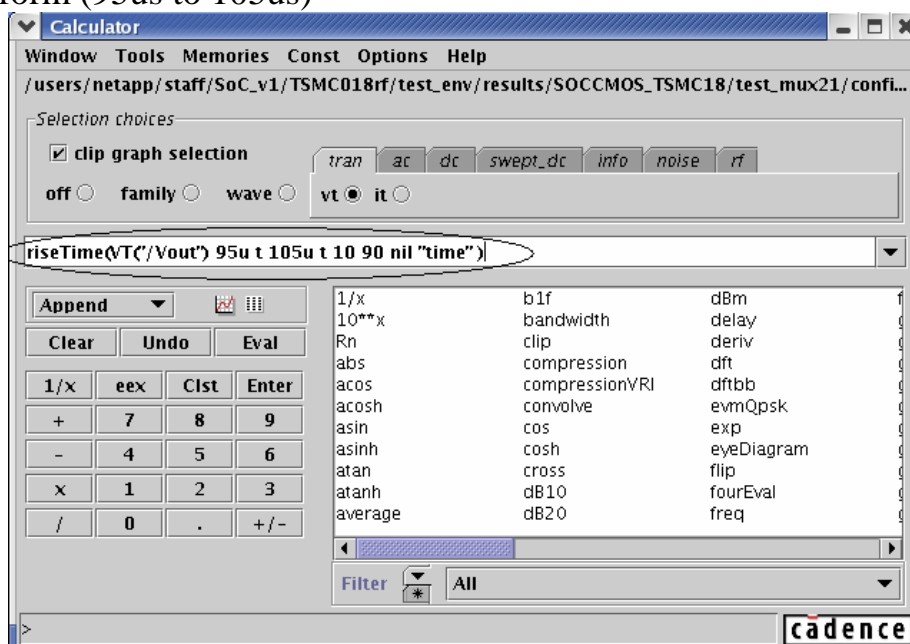
The following window will appear:



After pressing Ok we will open calculator.

In the calculator we'll generate an expression to calculate Tfall (the same as Trise) of Vout signal (from 90 % -> 1.62v to 10 % -> 0.18v).

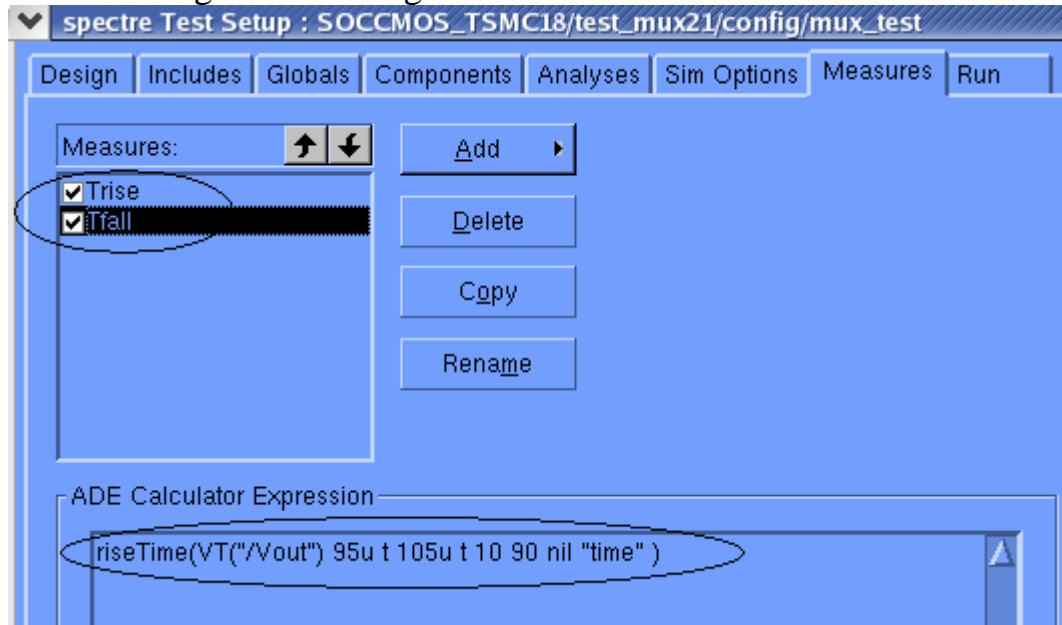
Notice that the range in which to measure Tfall I took from the Waveform (95us to 105us)



Next copy the expression to Ocean script (encircle the expression with mouse, and then copy by pressing the middle button of mouse).

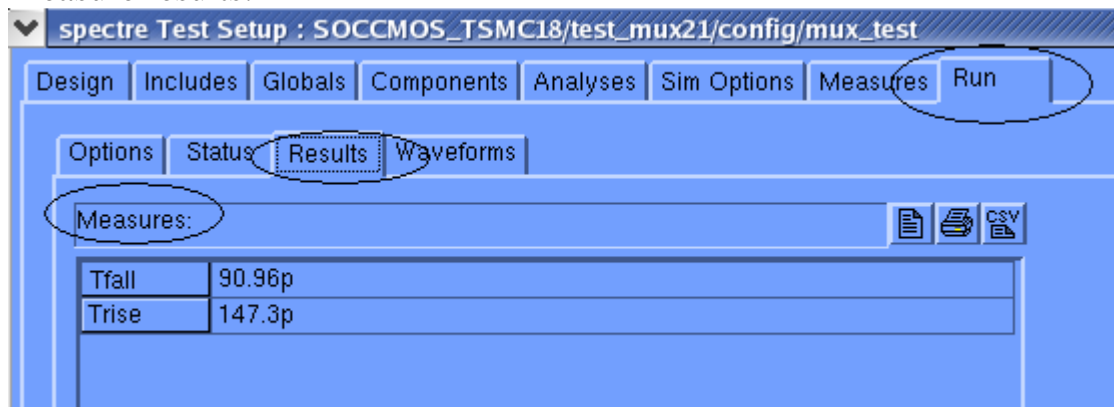
Repeat the process for Trise.

After receiving the following window:



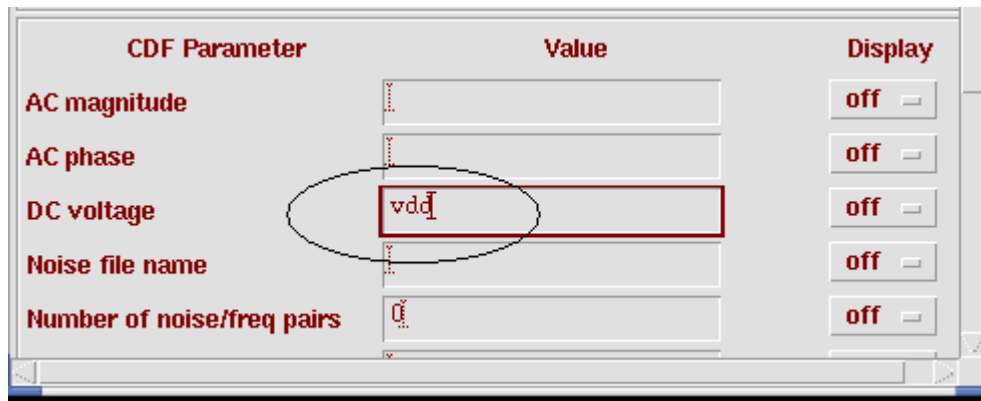
Press Run button again.

Under Rub section in Results subsection you'll receive the following Measure results:

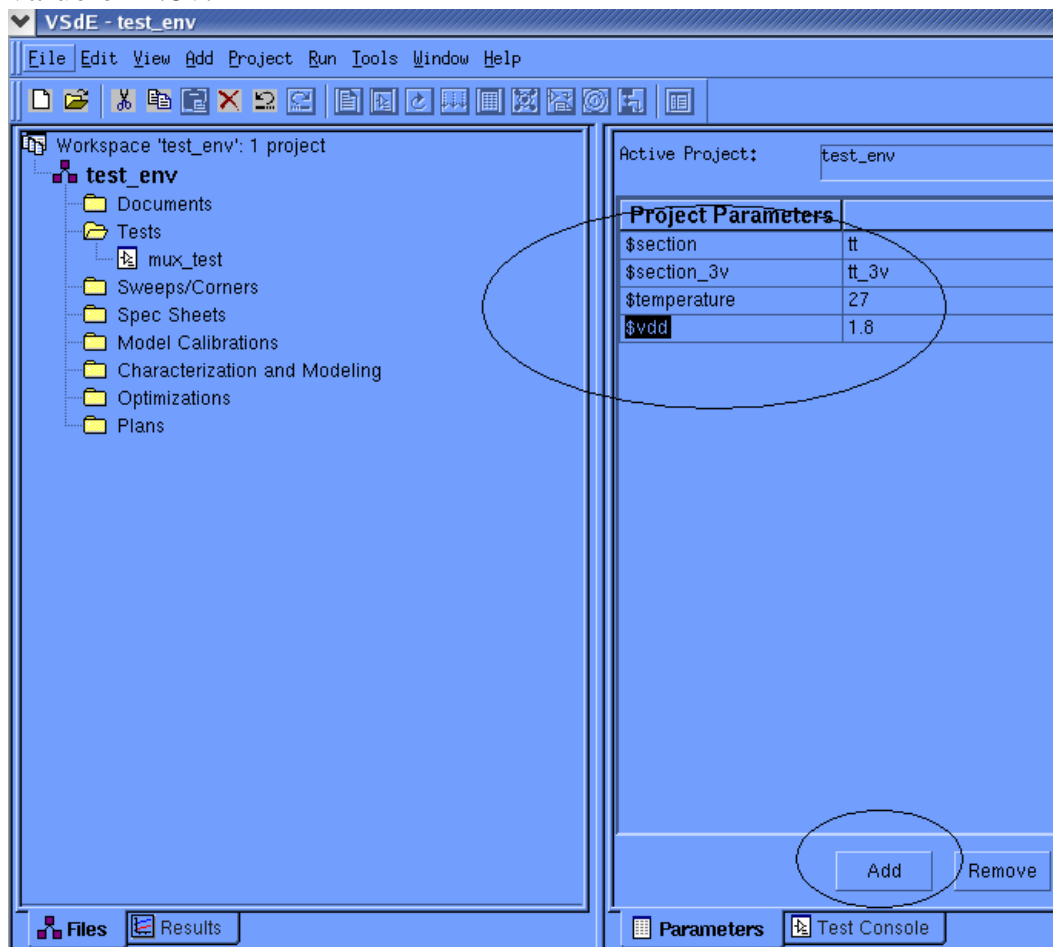


Creating corner simulation in Vsde

After building a test and running it the next stage is to build a corner test. First stage press q to enter the properties of Vdc and change DC voltage to vdd.

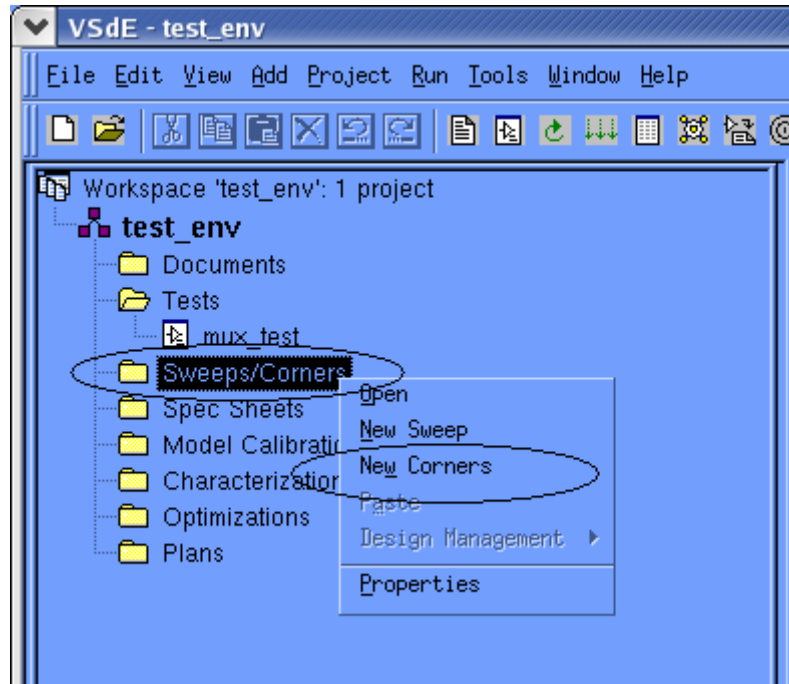


In the main window of vsde press Add and write \$vdd. Give it a default value of 1.8v.

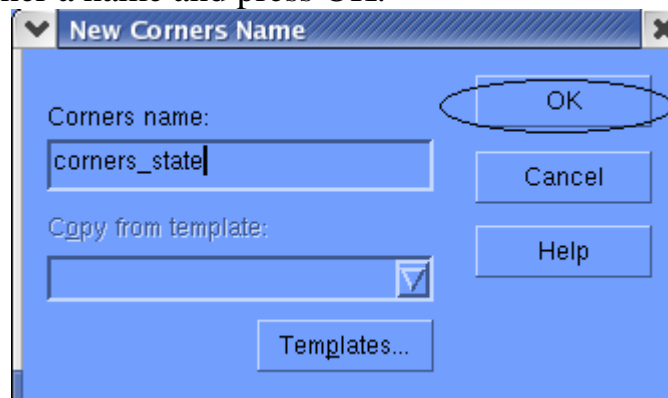


Also notice that Project Parameters already exist \$section, \$section_3v We added before and \$temperature which is added automatically once the simulation is built. All this parameters are necessary for corners.

In the main window under Sweeps/Corners click right button of the mouse and choose New Corners

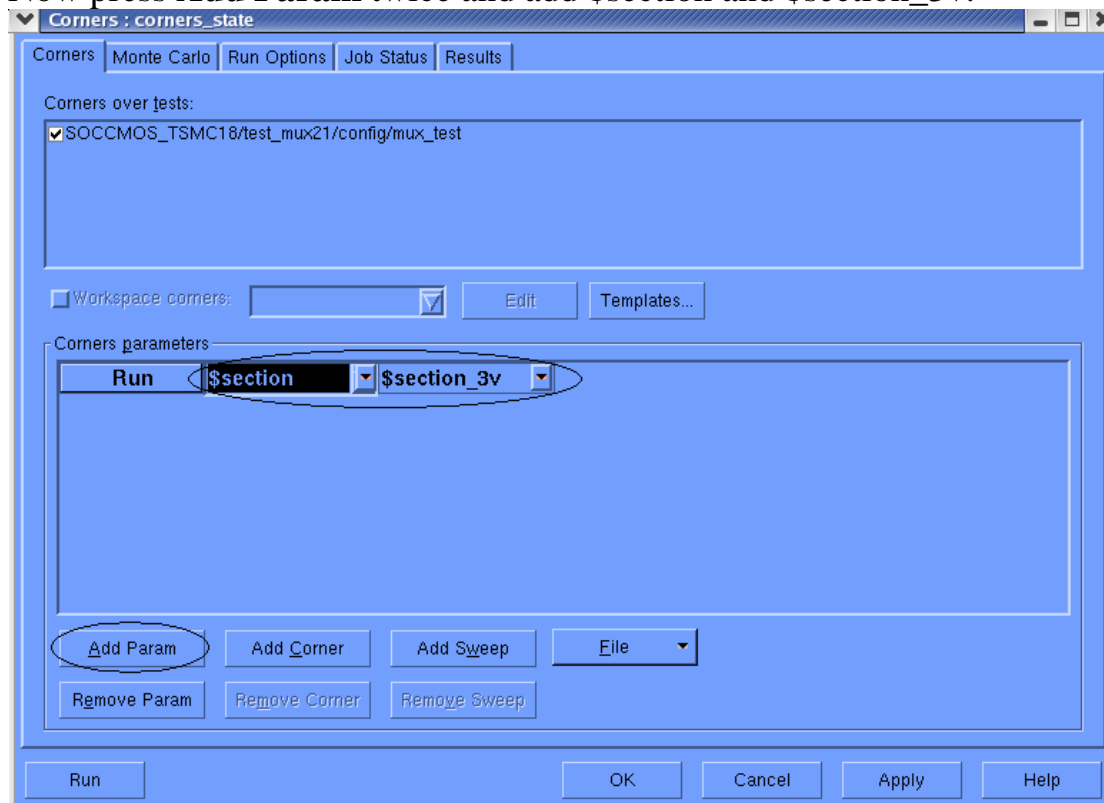


Give your corner a name and press OK.

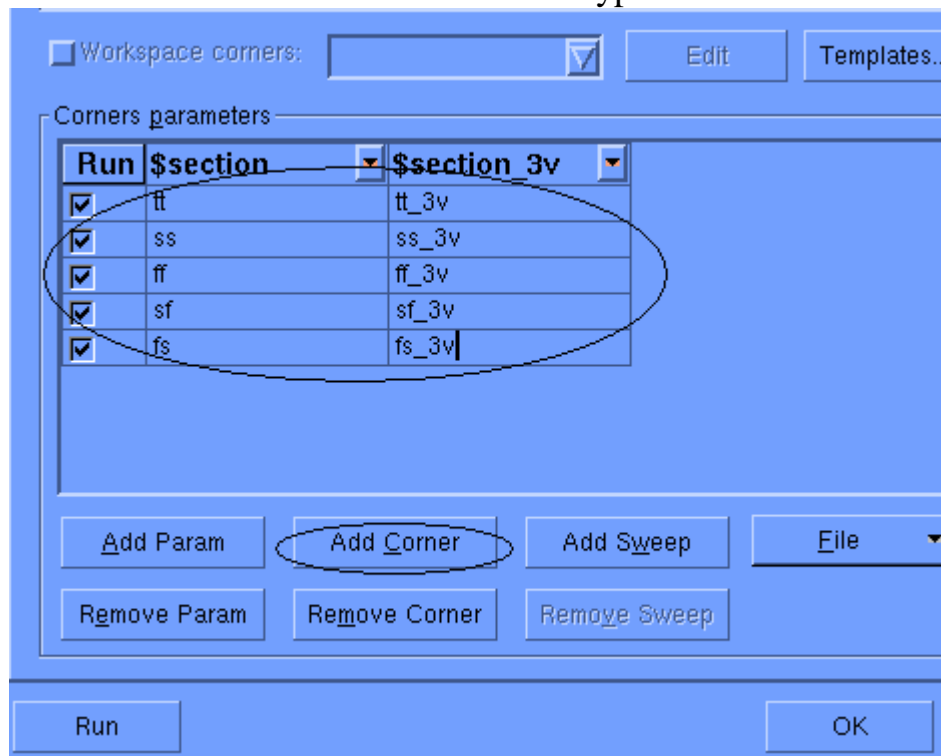


In the window notice that in Corners over tests the test I built before is marked. The point is that you can build one Corners template and choose on which test to run it!

Now press **Add Param** twice and add \$section and \$section_3v.

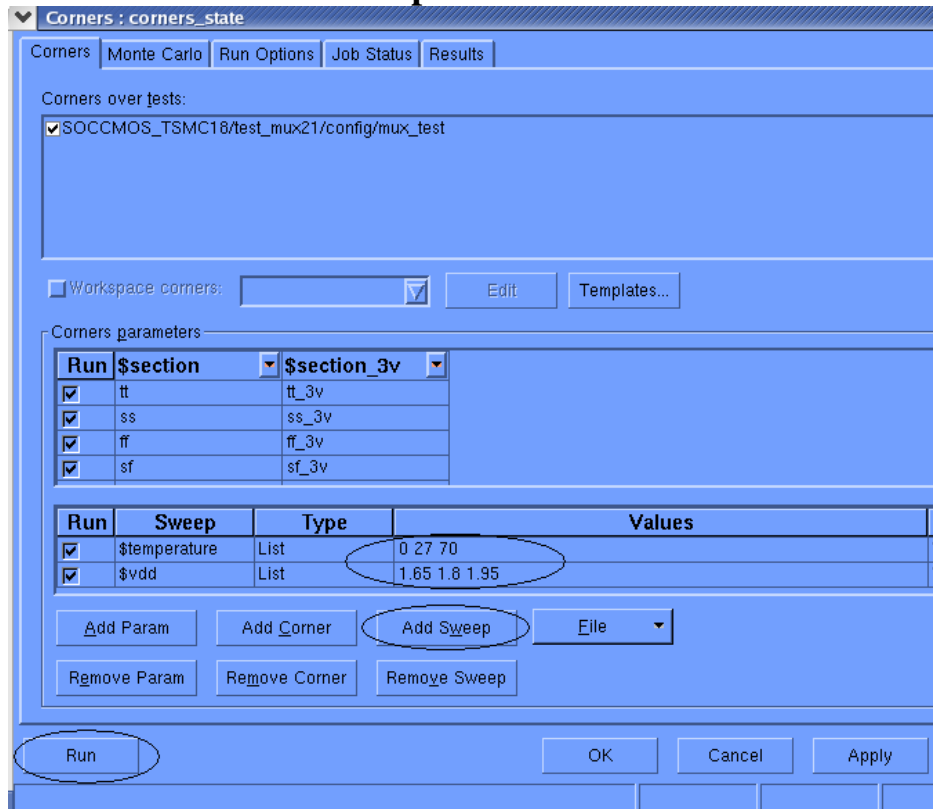


Press **Add Corner** 5 times and add module types of transistor

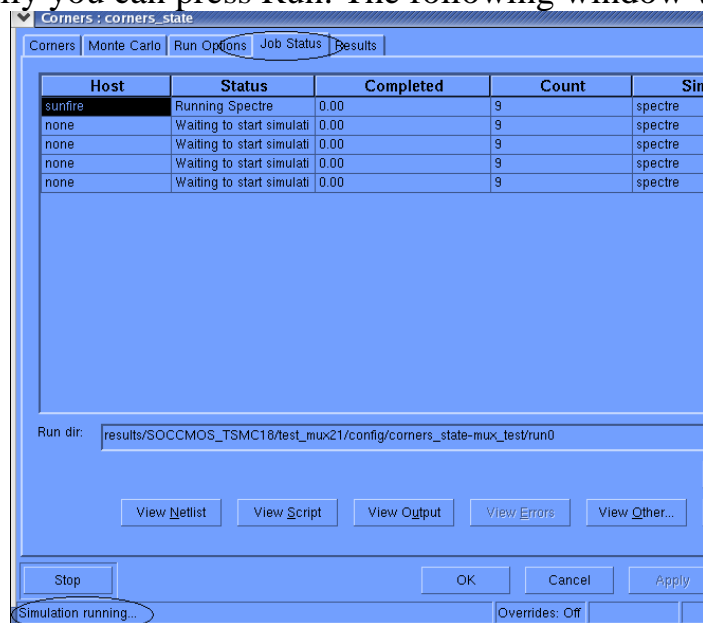


In \$section we define module types (parameters) for 1.8v transistors.
 In \$section_3v we define module types (parameters) for 3.3v transistors.

Press two times **Add Sweep** .And fill value section as following:

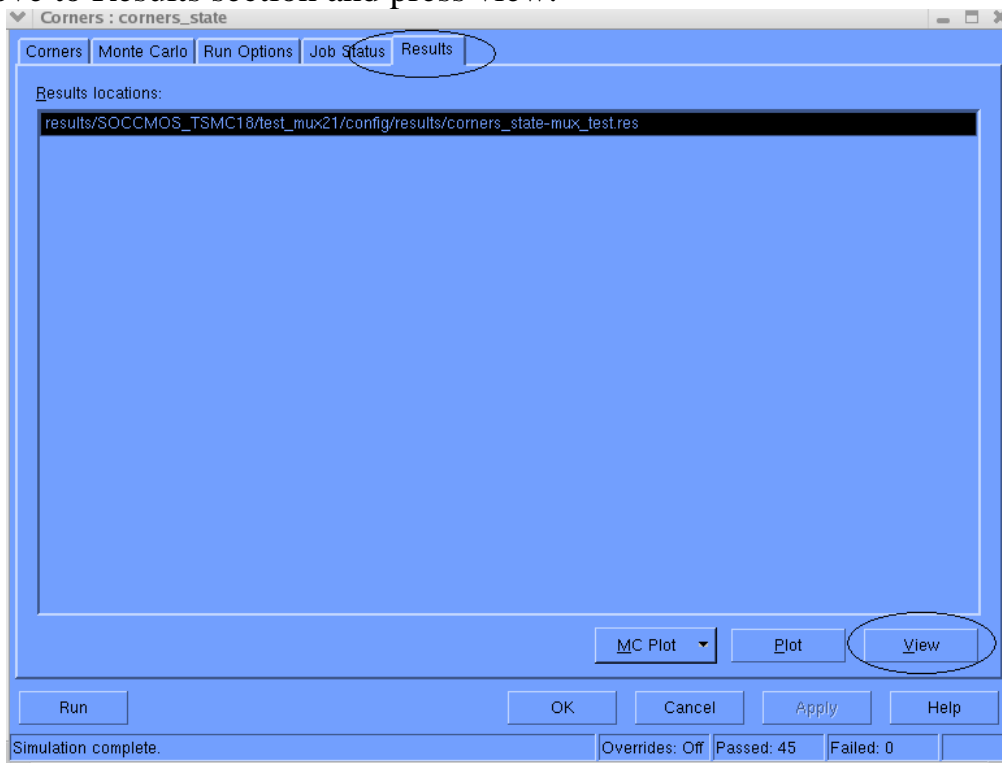


You can see that I've chosen one typical temperature (0, 27 and 70 Celsius) and two extreme. For voltage supply we usually take approximately 90 % (1.65v) 100 % (1.8v) and 110 % (1.95v). You can see that we suppose to receive $5 \times 3 \times 3 = 45$ different results after running corners. Finally you can press Run. The following window will be open:

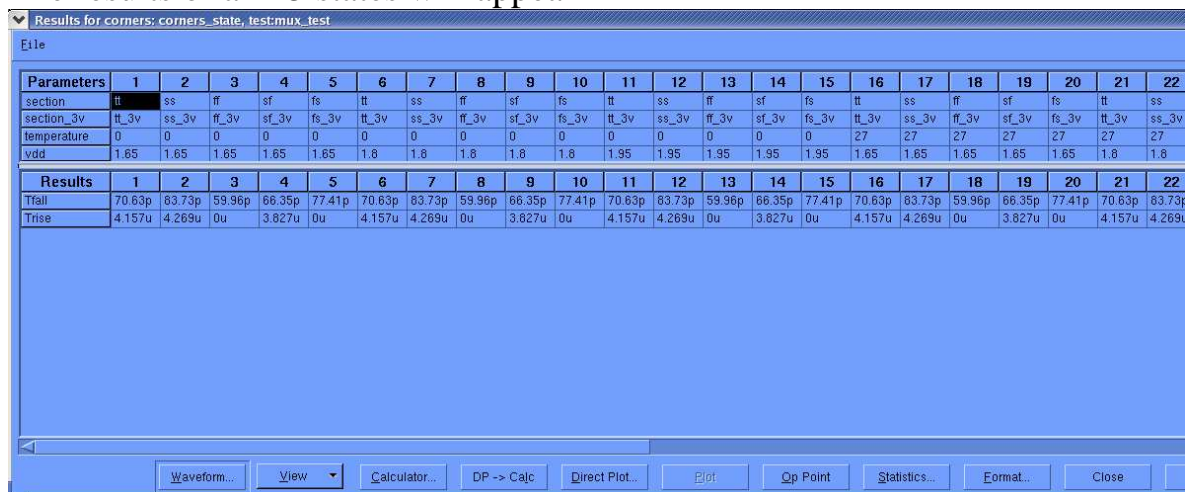


If you forgot something by pressing Stop you'll stop the process. Down on the left side of the screen you can see the status of the simulation.

Move to Results section and press view.



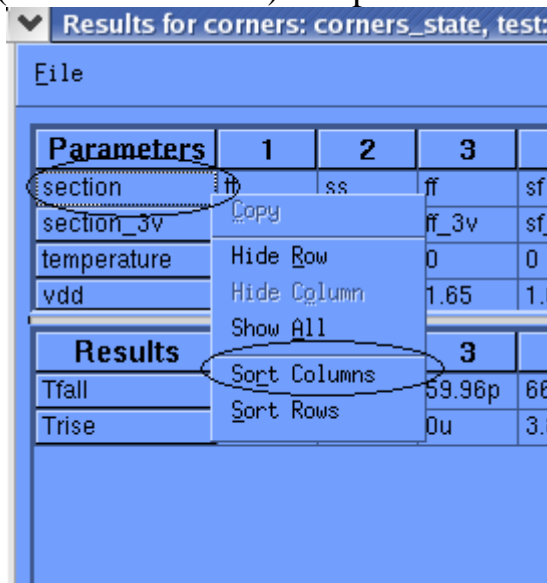
The results of all 45 states will appear



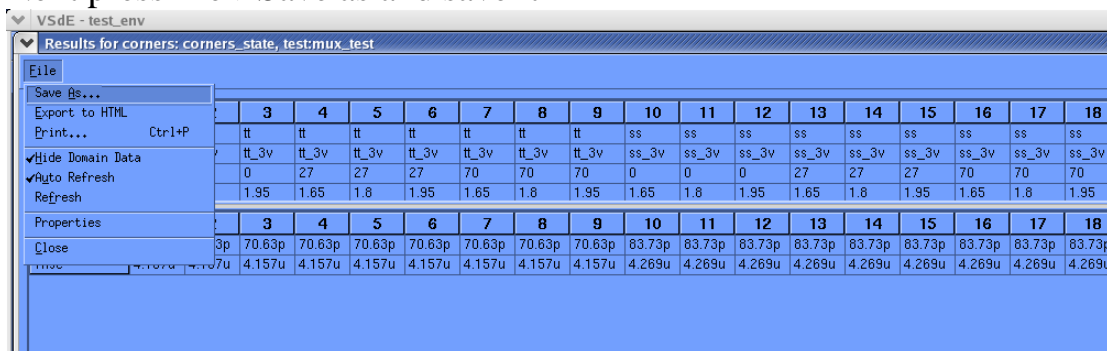
Parameters	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22
section	tt	ss	ff	sf	fs	tt	ss	ff	sf	fs	tt	ss	ff	sf	fs	tt	ss	ff	sf	fs	tt	ss
section_3v	tt_3v	ss_3v	ff_3v	sf_3v	fs_3v	tt_3v	ss_3v	ff_3v	sf_3v	fs_3v	tt_3v	ss_3v	ff_3v	sf_3v	fs_3v	tt_3v	ss_3v	ff_3v	sf_3v	fs_3v	tt_3v	ss_3v
temperature	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	27	27	27	27	27	27
vdd	1.65	1.65	1.65	1.65	1.65	1.8	1.8	1.8	1.8	1.8	1.95	1.95	1.95	1.95	1.95	1.65	1.65	1.65	1.65	1.65	1.8	1.8
Results	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22
Tfall	70.63p	83.73p	59.96p	66.35p	77.41p	70.63p	83.73p	59.96p	66.35p	77.41p	70.63p	83.73p	59.96p	66.35p	77.41p	70.63p	83.73p	59.96p	66.35p	77.41p	70.63p	83.73p
Trise	4.157u	4.269u	0u	3.827u	0u	4.157u	4.269u	0u	3.827u	0u	4.157u	4.269u	0u	3.827u	0u	4.157u	4.269u	0u	3.827u	0u	4.157u	4.269u

Down the screen there are different options. By pressing **Waveform** you can create different kinds of waves as a function of voltage supply or temperature. By pressing **Calculator** you'll open it. Pressing **Direct Plot** will allow you to check the signal of any node exactly as in Analog environment. By pressing **Format** you can choose the type of display of numbers.

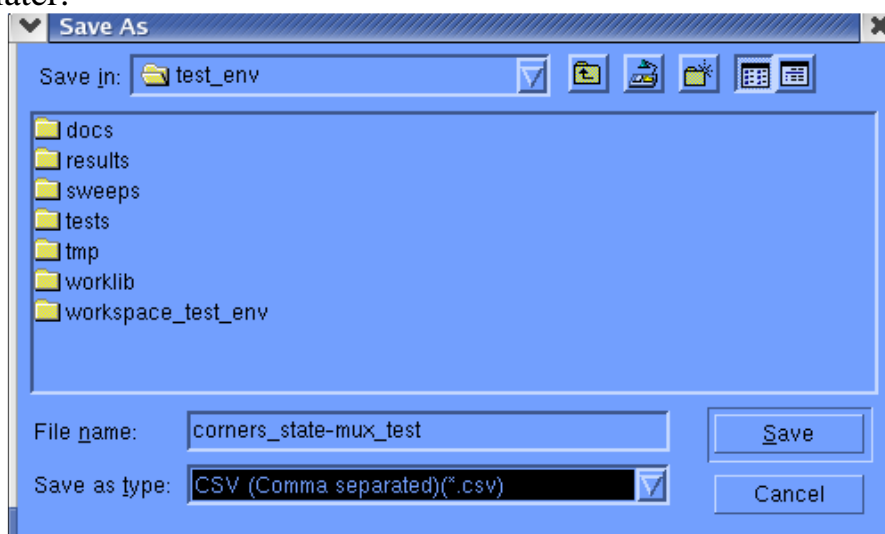
However the main purpose is to save the results so we will be able to process them later. Therefore first of all press with the right button of the mouse on section (under Parameters) and press Sort Columns



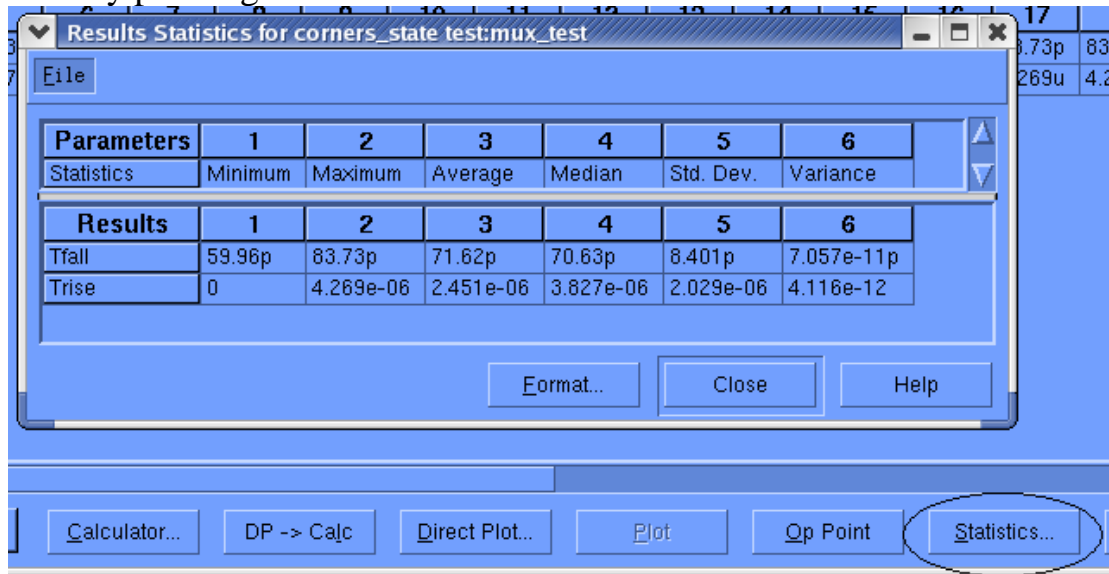
It will allow us to arrange data by types of corners tt/ss/sf/fs/ff
 Next press file->Save as and save it



While saving the file under CSV format you'll be able to open it with Excel later.



Also by pressing Statistics



Parameters	1	2	3	4	5	6
Statistics	Minimum	Maximum	Average	Median	Std. Dev.	Variance

Results	1	2	3	4	5	6
Tfall	59.96p	83.73p	71.62p	70.63p	8.401p	7.057e-11p
Trise	0	4.269e-06	2.451e-06	3.827e-06	2.029e-06	4.116e-12

You can get some more information about your results.

Also by pressing File ->save as you'll be able to save all the statistic data in Excel file.

One comment: Since Cadence is working under Linux/Unix Environment you might need to use FTP application to move the file to Windows environment.