Usage

Execute (from the CIW) the command

- hitkit Utilities -> Corner Analysis

This creates a form.

1. Select 'spectre' as your target simulator.
2. Check the 'Root Model Path' corresponds to the process you are using.
3. Enter the 'Circuit Name' for the design you wish to simulate.
4. Specify the temperature and voltages ranges which you wish to use. You must also specify the name of the design variable associated with the voltage supply, this has a default name of "Vsupply" and this name (or another of your choice must be specified on the properties form of your voltage supplies.
5. Select the MOS model you wish to use.
6. Select the Bipolar model you wish to use.
7. Click on 'Add Corner' and set the corner parameters you require for each device in your circuit (CMOS devices, RESistors, CAPacitors & BIPolar devices) Also select the TEMPerature and SUPPLY voltage conditions. The field NAME is an optional comment and only provided for convenience.
8. Repeat this procedure until you have defined all the corners you require.
9. The button 'Auto-Corner' may be used as an alternative to manually selecting all the desired process variations. This function automatically creates data for all the specified process conditions - a count of the number specified is also shown.

The completed form should look similar to the one shown below:-

10. When you are satisfied with the selected data click on the 'OK' button. The system should respond with the message (in the CIW) "Corner definition file has been written!". If you get any error messages, please correct the problem and repeat this step.

11. Open your schematic (if not already opened) and start Analog Artist with the spectre simulator (other simulators will NOT work).

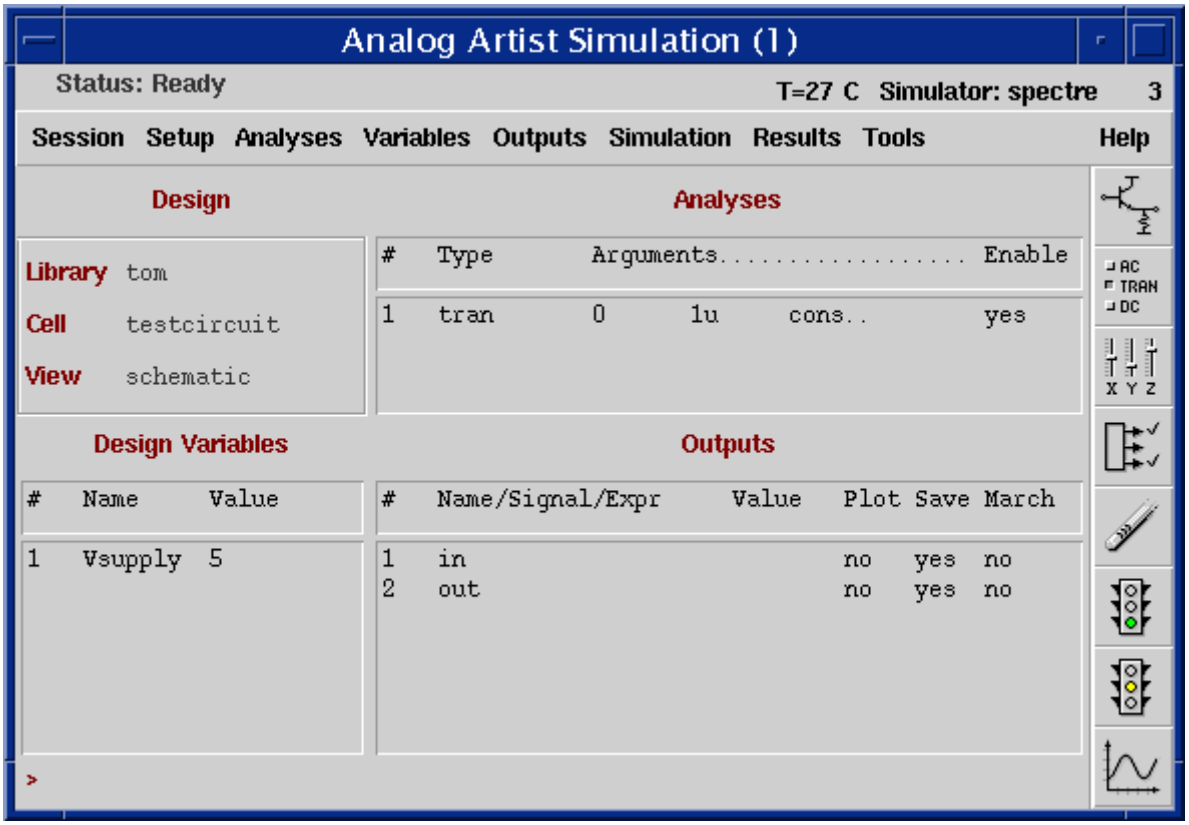
12. Specify the analysis you require, the nodes you wish to save, etc as for a normal simulation run.

13. Check the variable for the voltage supply (Default 'Vsupply') is on the form and give it a default value. If not then simply add this by selecting 'Variables->Edit...' from the simulation window,

a. Add the 'New variable' :-

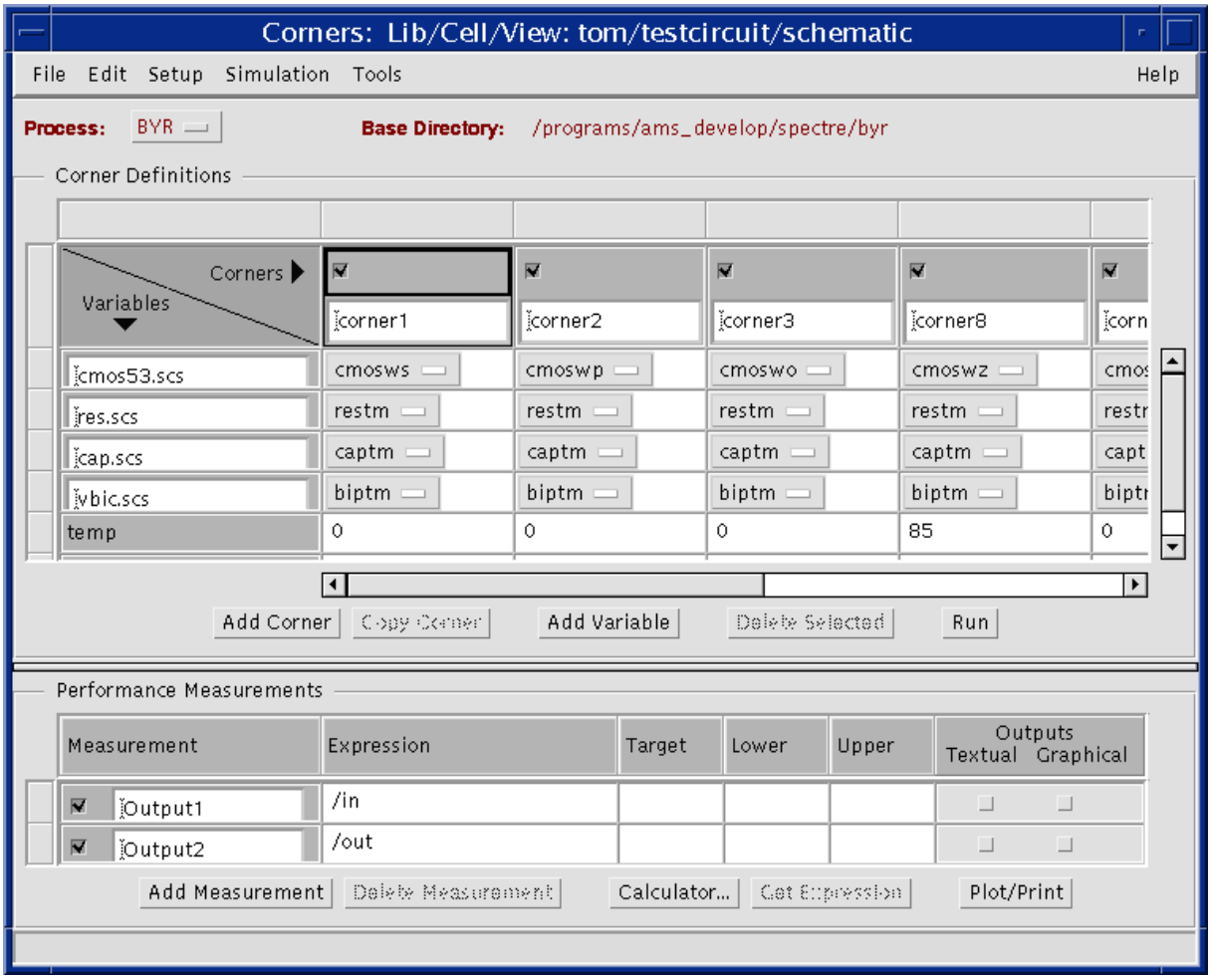
b. Name : Vsuppply

c. Value (Expr) : 5



14. Select 'Tools->Corners ...' from the simulation window.

If you have defined your Corners just before starting Analog Artist, then your corner definition file will be loaded automatically. If you want to reuse a previously defined corner definition file, then select "File->Load ..." from the Corners window. The file to load should have the name of your circuit (the one you supplied in the hitkit Utilities Corner Analysis Form) with the extension ".dcf" e.g. testcircuit.dcf. The Corner window should finally look like the following:



If you want to add an additional design variable use the "Add Variable" button.

For an additional corner use the "Add Corner" button.

For adding an additional measurement, use the "Add Measurement" button. The expression can be typed in directly, or you can make use of the Calculator.

15. When everything is setup according to your needs, hit the "Run" button.

16. The results can be displayed directly if you enable the textual or graphical ouput in the Corners window, or you can use the Calculator after the simulation, to postprocess the simulation data.