

Procedures to install the Design Architect IC Tool

- Extract eldo.tar.gz and daic.tar.gz to any location. They occupy 1.4 GB and 1.1 GB respectively when extracted. It should correspondingly create folders with names "AMS_2005.3" and "icflow_home".
- Copy the files 'mgc_location_map' and 'startdaic' to your home directory.
- Open the file 'mgc_location_map' in a text editor. Insert the full path(no symbols or aliases) of the folder "icflow_home". There are 9 variables and you should set path for all of them. The commented lines should help you.

For example:

If 'icflow_home' directory's path was /mnt/home/xxxx/ then insert "/mnt/home/xxxx/" before what is already

The file before editing :

```
MGC_LOCATION_MAP_2
```

```
# MGC IC sources_lib
```

```
$MGC_IC_SOURCES_LIB icflow_home/pkgs/utilities_ic.ixc/mgc_icstd_lib/sources_lib
```

```
# MGC IC generic_lib
```

```
$MGC_IC_GENERIC_LIB icflow_home/pkgs/utilities_ic.ixc/mgc_icstd_lib/generic_lib
```

```
.  
. .  
. . .
```

The file after editing :

```
MGC_LOCATION_MAP_2
```

```
# MGC IC sources_lib
```

```
$MGC_IC_SOURCES_LIB /mnt/home/xxxx/icflow_home/pkgs/utilities_ic.ixc/mgc_icstd_lib/sources_lib
```

```
# MGC IC generic_lib
```

```
$MGC_IC_GENERIC_LIB /mnt/home/xxxx/icflow_home/pkgs/utilities_ic.ixc/mgc_icstd_lib/generic_lib
```

```
.  
. .  
. . .
```

- Open the file "startdaic" in a text editor. You have to set two environmental variables here. There is a commented instruction which variable to set.

Third line : Insert the path where the directory "icflow_home" is (extracting daic.tar.gz creates this folder.) After adding the path the third line looks like this

```
setenv MGC_HOME /mnt/home/xxxx/icflow_home
```

Thirteenth line : Insert the path where the "AMS_2005.3" folder is (extracting eldo.tar.gz creates this folder) After adding the path the thirteenth line look likes this

```
setenv anacad /mnt/home/xxxx/AMS_2005.3
```

- Change your shell to TC Shell with the command 'tcsh' and invoke the tool with the command ./startdaic
-

Important: Changes to be made in the library file before simulation

Download the tsmc018.lib from the link <http://www.ee.iitm.ac.in/~nagendra/cadinfo.html>. Open it in a text editor, make the following change.

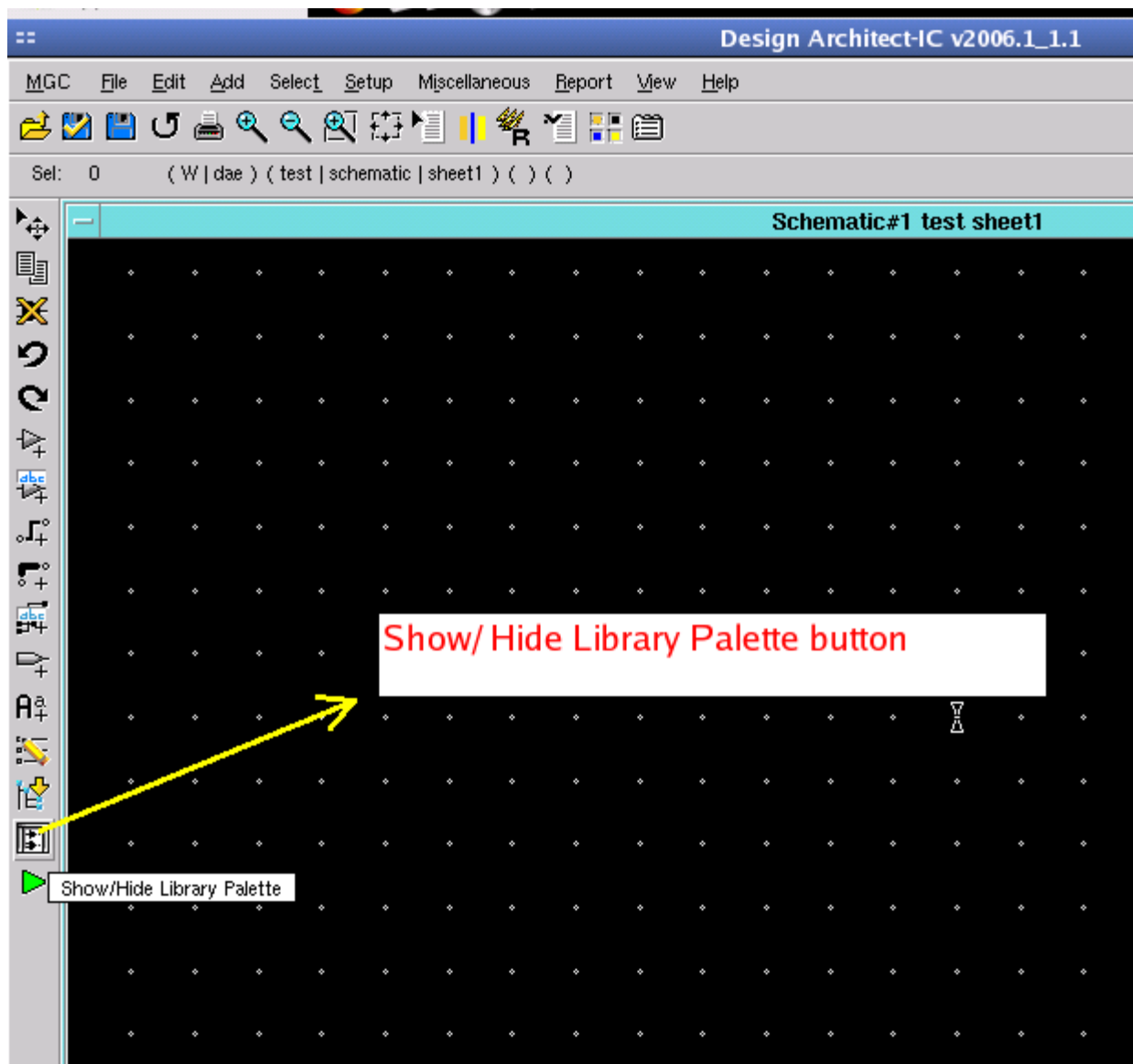
- ◆ Change the value of level to 53 in lines Nine and Fifty One.

If you use any other library like UMC library, it should be in .lib format. And you ensure that the level is 53.

Working around with DAIC

- Create a folder (say 'Mydesign') where you will save each project. Create sub folders(say 'Inverter') inside this folder ('Mydesign') for each design. The tool Design Architect IC creates further sub folders and files inside this work folder ('Inverter').
- Lets see an example of how to simulate an CMOS inverter. Create a folder named 'Inverter' inside the 'Mydesign' folder.
- Click File --> Open --> Schematic. Select browse and point to the folder 'Inverter'. Click Options and select 'New sheet'. Click 'OK' twice.
- A new blank sheet will be opened. The right corner button in the title bar of the sheet maximizes the window and lets you see the status bar. This will help you see the progress of your work and errors if any will be reported here.
- Select the 'Show/Hide Library palette' (penultimate button in the left hand side tool bar). A new panel will be opened in the right side. The 'devices library' has the

Transistors and other components like resistors, capacitors... The 'Generic Library' has the Vdd, ground and other symbols. The 'Sources Library' has various sources.



- Select components and place it in the sheet. To rotate the component press the 'r' key. Select a component and press the key 'q' to get its properties window where you can change various attributes. Press 'w' to get the wire and connect the components.

- When using transistors, make sure you change the ASIM_MODEL parameter in the properties dialog box of each transistor to the model name mentioned in the library. By default ASIM_MODEL will be NCH/PCH. To know what to enter in the ASIM_MODEL field, open the library file in a text editor. The picture below shows the details for tsmc018.lib. Apply similar attributes to the PMOS transistor if necessary.

```

* T58F SPICE BSIM3 VERSION 3.1 PARAMETERS
*
* SPICE 3f5 Level 8, Star-HSPICE Level 49, UTMOST Level 8
*
* DATE: Oct 31/05
* LOT: T58F           WAF: 9005
* Temperature_parameters=Default

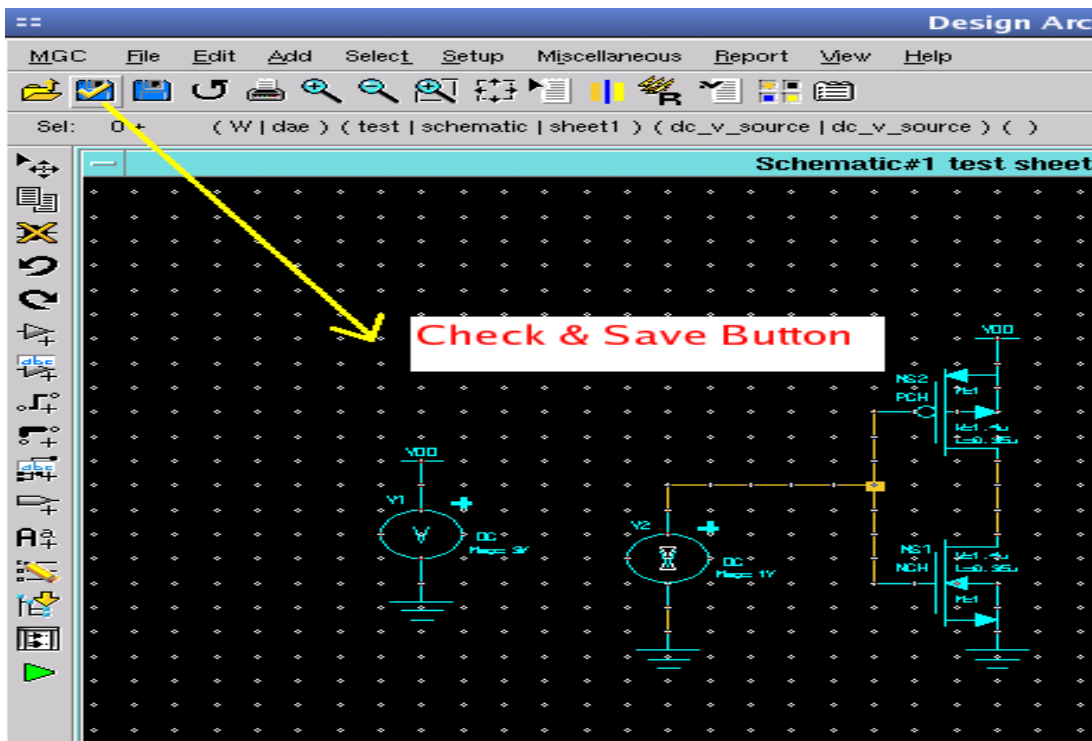
.MODEL CMOSN NMOS (
+VERSION = 3.1          TNOM   = 27          TOX   = 4.1E-9
+XJ      = 1E-7         NCH   = 2.3549E17    VTHO  = 0.3662473
+K1      = 0.5864999   K2    = 1.127266E-3   K3    = 1E-3
+K3B     = 0.0294061  WO    = 1E-7         NLX   = 1.630684E-7
+DVTOW   = 0          DVT1W = 0           DVT2W = 0
+DVTO   = 1.2064649   DVT1  = 0.4215486   DVT2  = 0.0197749
+UO      = 273.8094484 UA    = -1.40499E-9   UB    = 2.408323E-18
+UC      = 6.504826E-11 VSAT  = 1.355009E5   AO    = 2
+AGS     = 0.4449958  B0    = 1.901075E-7   B1    = 4.99995E-6
+KETA    = -0.0164863 A1    = 3.868769E-4   A2    = 0.4640272
+RDSW   = 123.3376355 PRWG  = 0.5          PRWB  = -0.197728
)

```

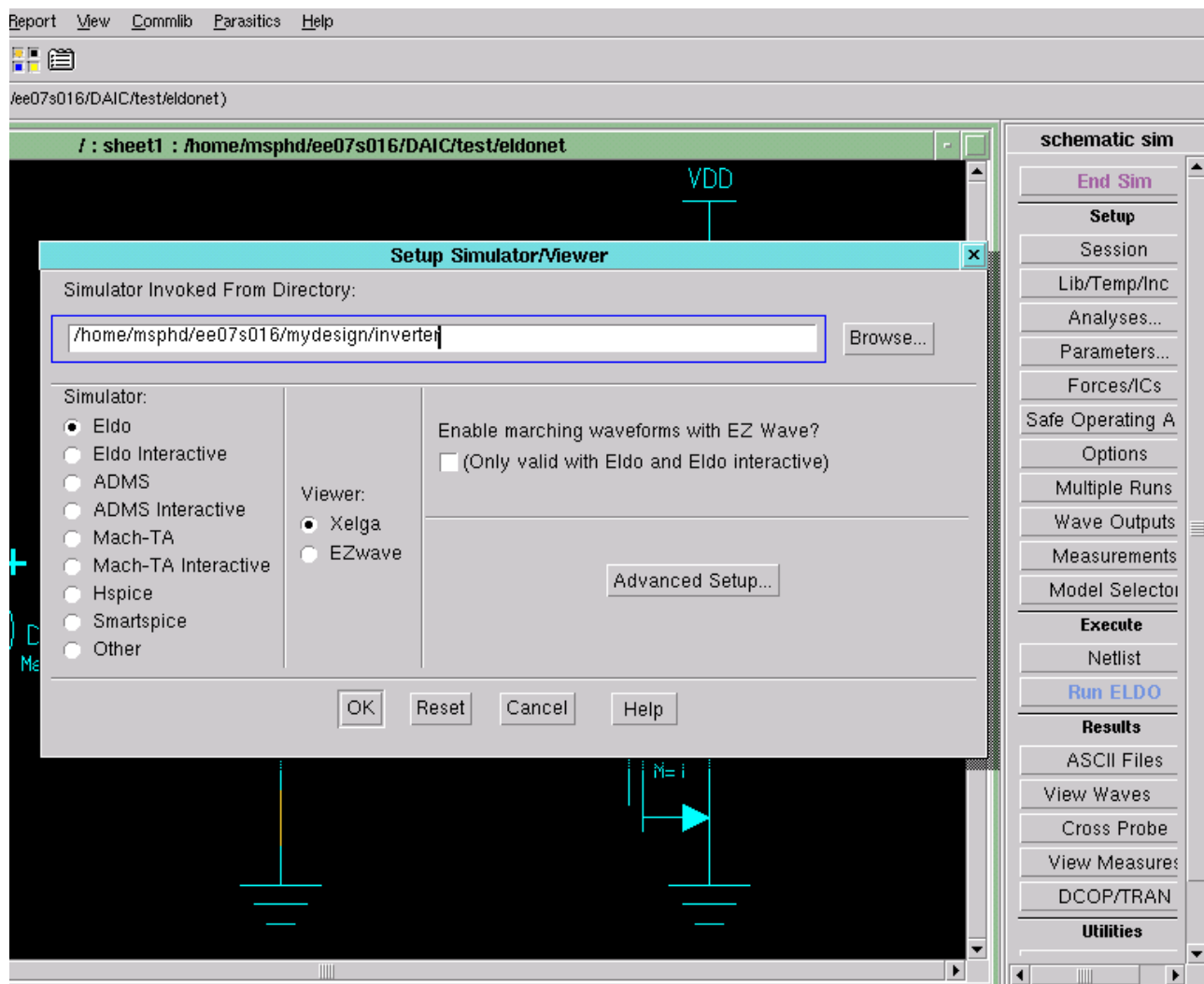
Change it from 49 to 53

Model Name = CMOSN

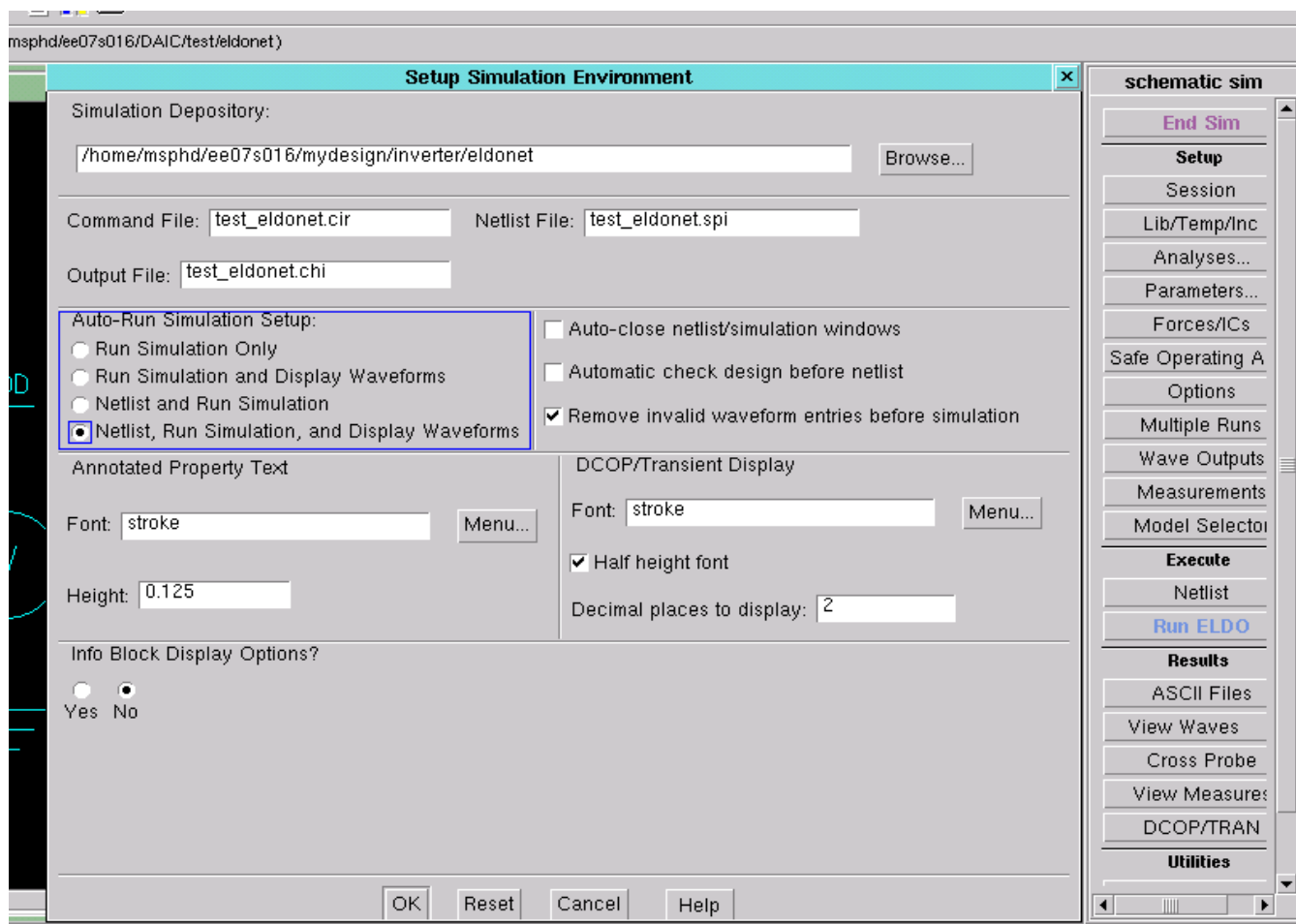
- After connecting various components click 'check and save schematic' button (Second button in the top tool bar). Errors if any will be shown in a separate window.



- Click the 'Simulation' button in the right side panel and click 'OK' in the dialog box that pops up. You enter simulation mode.
- On the right side panel, click the button session and then simulator/viewer. Select a folder where you want to store the files that are produced during simulation. Better choose the working folder 'Inverter.' Select Xelga as the waveform viewer.

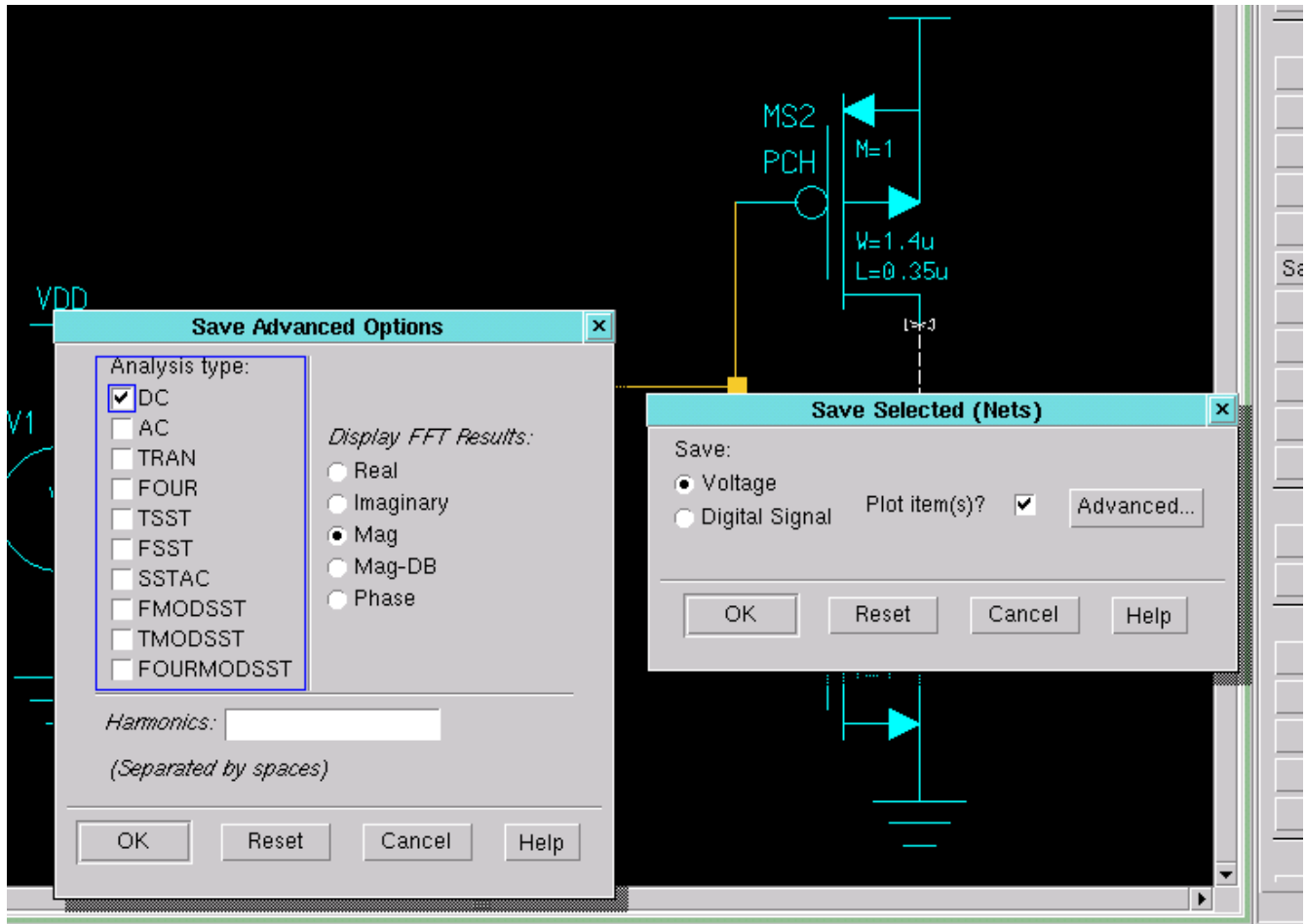


- In the right side panel, click the button 'session' and then 'netlister' in the pop menu. Enter the name of the nets where it should be ground. By default only the nets where ground label is attached becomes ground.
- In the right side panel, click the button session and then 'environment'. You can ask the simulator to display the waveforms directly after running simulation. In this case the information on nets which are saved will be plotted. The next point describes how to save various nets for waveforms.

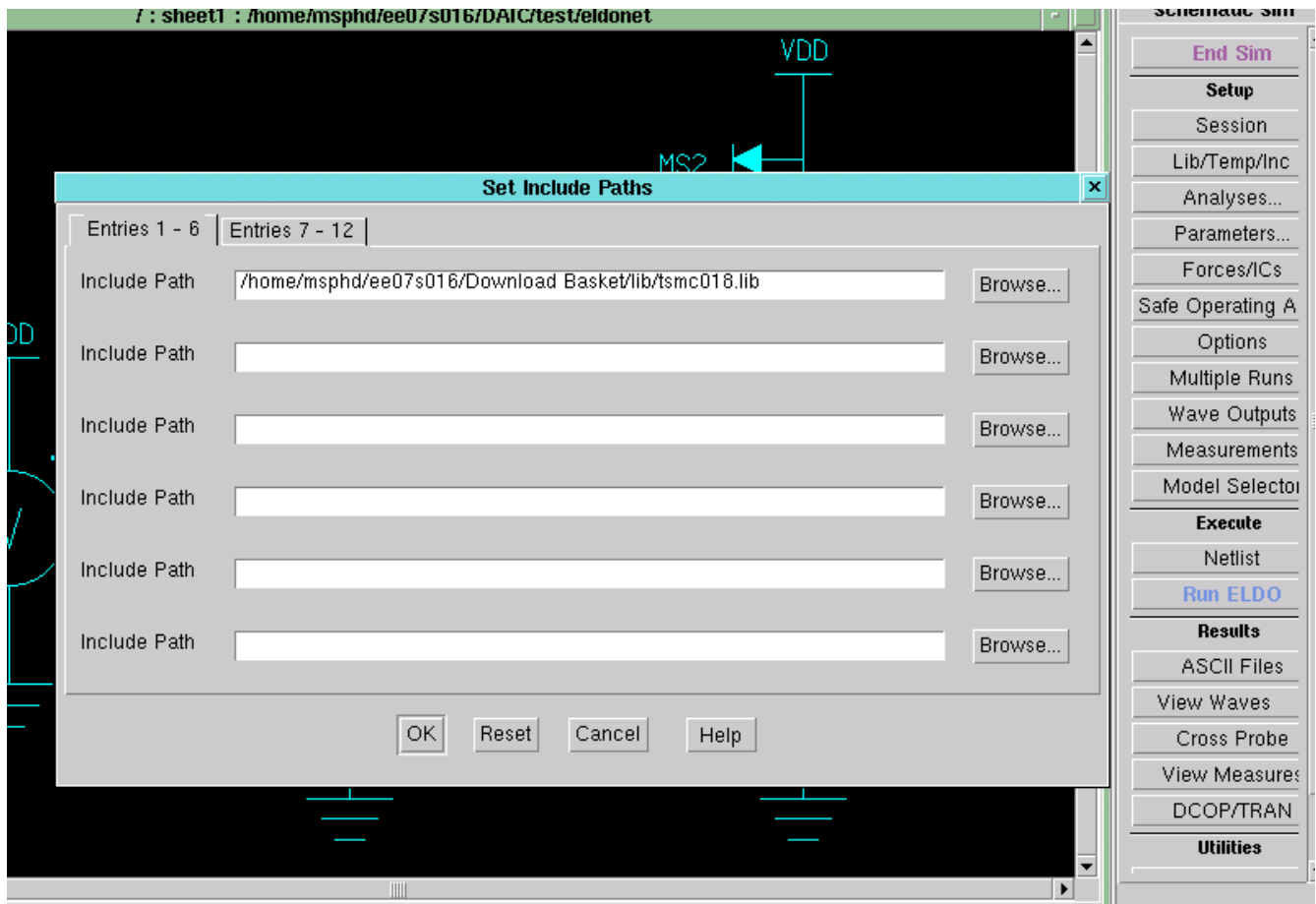


- Before you save nets you have to choose the analysis. Here we want a DC analysis with the input voltage being swept from 0 to Vdd. Click 'Analysis' button in the right side panel and in the pop up box select DC simulation. Click 'Setup' button and select the source for sweep and enter the sweep range.

- Select a Net where you want to results to be plotted. In the right panel click 'Wave outputs' button. Click Save selected. In the pop up menu select voltage. Click the advanced menu. Choose the Analysis type (here DC) and the result type (here magnitude) To plot the current, select a node and repeat the above steps.



- To Add library files corresponding to the FET, Click 'Lib/Temp/Include' button in the right panel and click 'Add Path' Point to the library file. The library file should have the extension as '.lib' Here we add the tsmc018.lib file. There are some important changes to be made in the library file before adding them. Please see the note above this documentation.



- Click 'Run Eldo' button and wait for the results to be plotted. To make changes and simulate, Click 'End Simulation' button and repeat the steps.

