

HEPIC Summer week - Instructions for practical session

Description

The exercises will be conducted using Cadence Virtuoso, running on a dedicated UNIX machine.

In order to access the machine, you will need a laptop with an SSH client and X11 forwarding: e.g., Putty and Xming are popular choices on Windows.

Online you can find many tutorials for your OS, for example:

- <http://ocean.stanford.edu/courses/ESS141/PuTTY/>

Please make sure to install the software prior to your visit.

Connecting to the UNIX server

Hostname: r7cad-slac.stanford.edu

User and password: will be sent to you through a separate e-mail or given during the summer week

Running virtuoso on the system

1. Log in through SSH to the UNIX server
2. Type the following commands in the terminal, in your home directory:

```
csH
source /home/hepicSW/HEPICsetup.csh
```

3. You should now be able to start Cadence Virtuoso by running in the terminal:

```
virtuoso &
```

4. In case you get the following error, check your X11/Xming/Putty configuration:

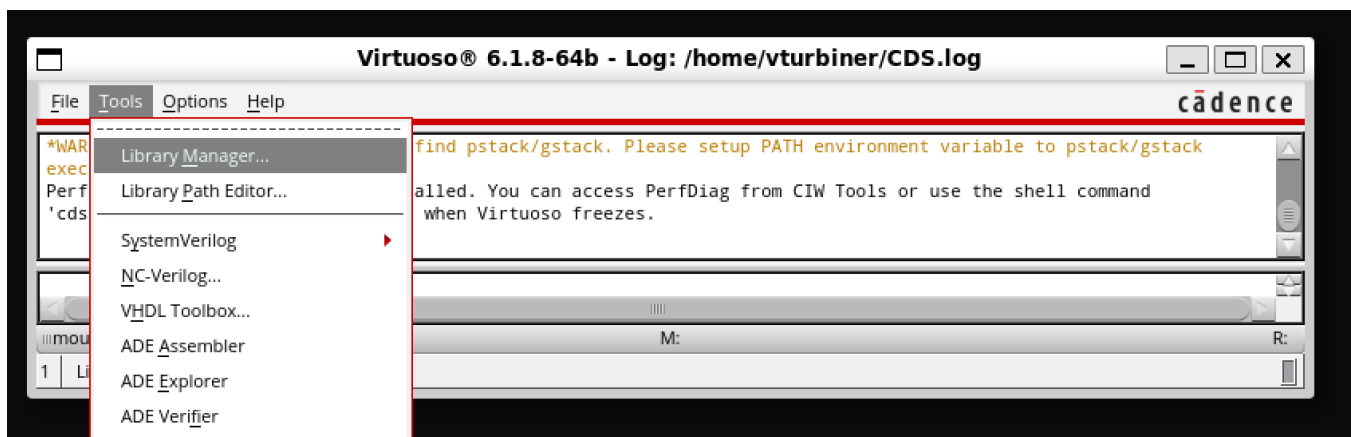
```
PuTTY X11 proxy: Unsupported authorisation protocol
*WARNING* X Window Display Initialization failure
*WARNING* (DISPLAY "localhost:10.0")
```

PDK

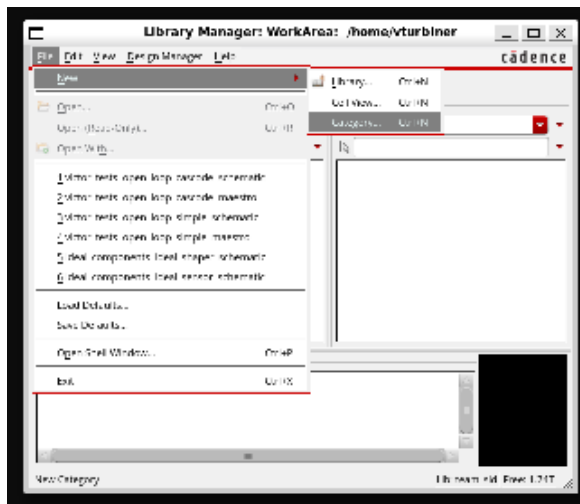
We will use the a Generic PDK (GPDK) 180nm from Cadence. The PDK is installed in /home/hepicSW/gpdk180_v3.3

Creating a schematic with Virtuoso

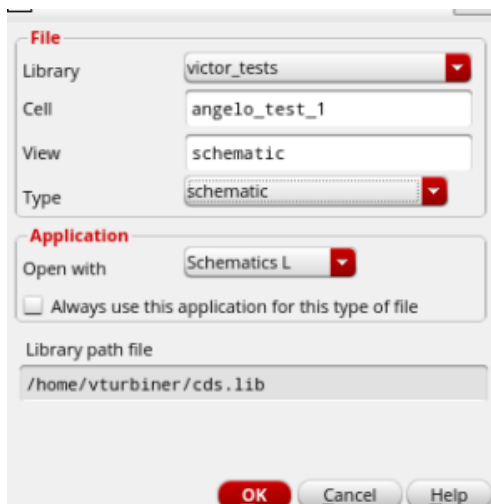
Open library manager:



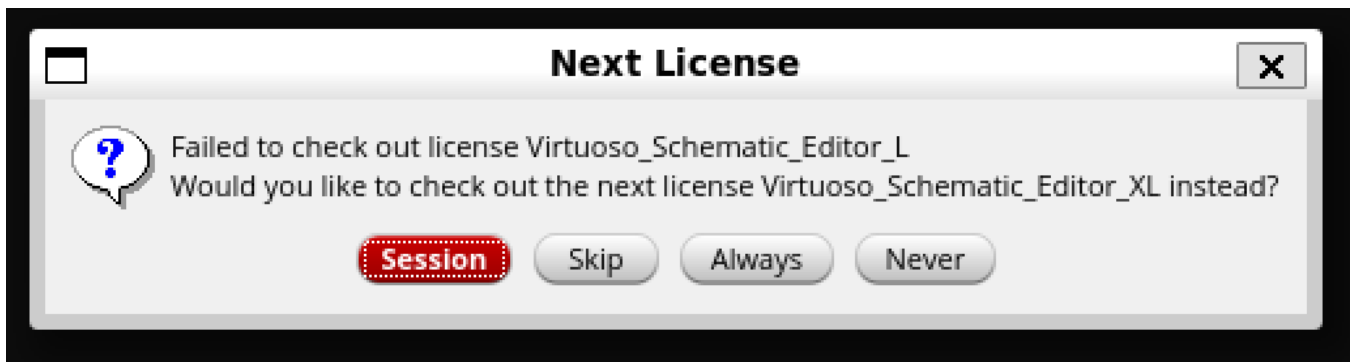
Select your team's library. Go to File New Cell View:



Create a schematic for your circuit:



If the following menu pops up, press "Session":



Getting Started With Virtuoso Schematics

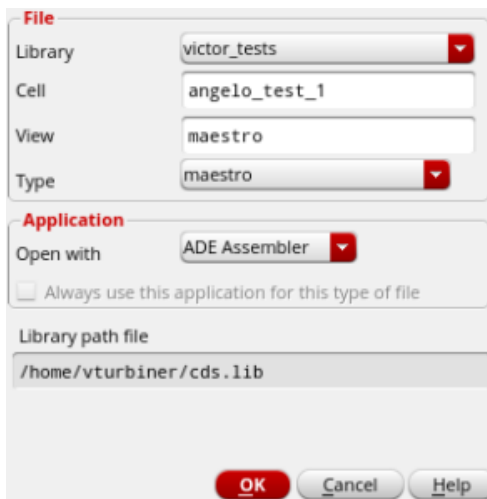
- Create a new component by pressing the "i" key.



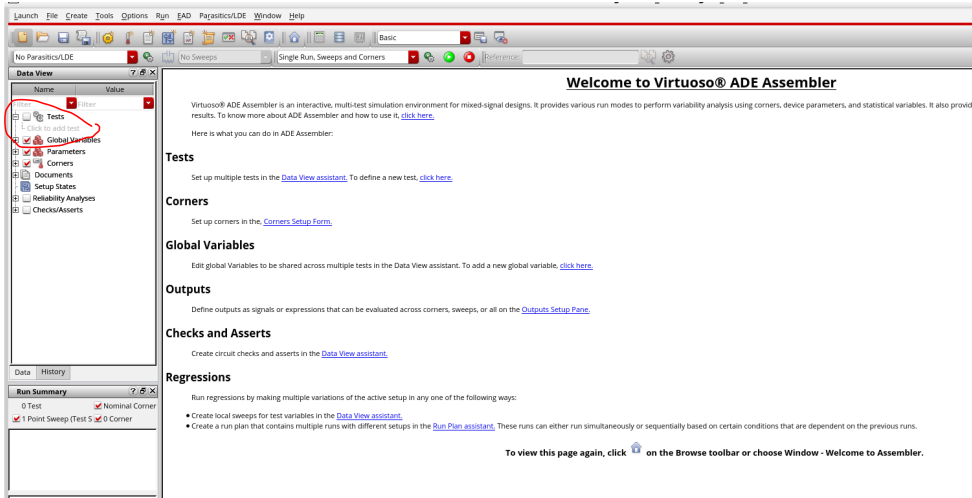
- When you press the "i" key, the following menu pops up:
- Ideal components can be found in the Library "analogLib".
- For transistors, use "nmos" and "pmos" from the "gpd180" library.
- Don't forget to have at least one ground from the analogLib library.
- Once you have configured the component, press the "Hide" button and you can place the part.
- Change an object's properties by left-clicking on it and then pressing the "q" key.
- Draw a wire by pressing the "w" key.
- Label a wire by pressing the "l" key, typing the label name, and then pressing the "Hide" button.

Testing a schematic with Virtuoso

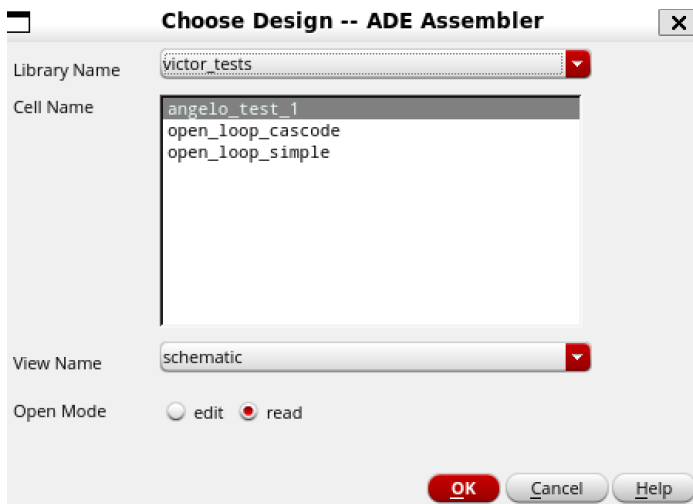
From Library Manager, go to file New Cell View. Create a "maestro" file.



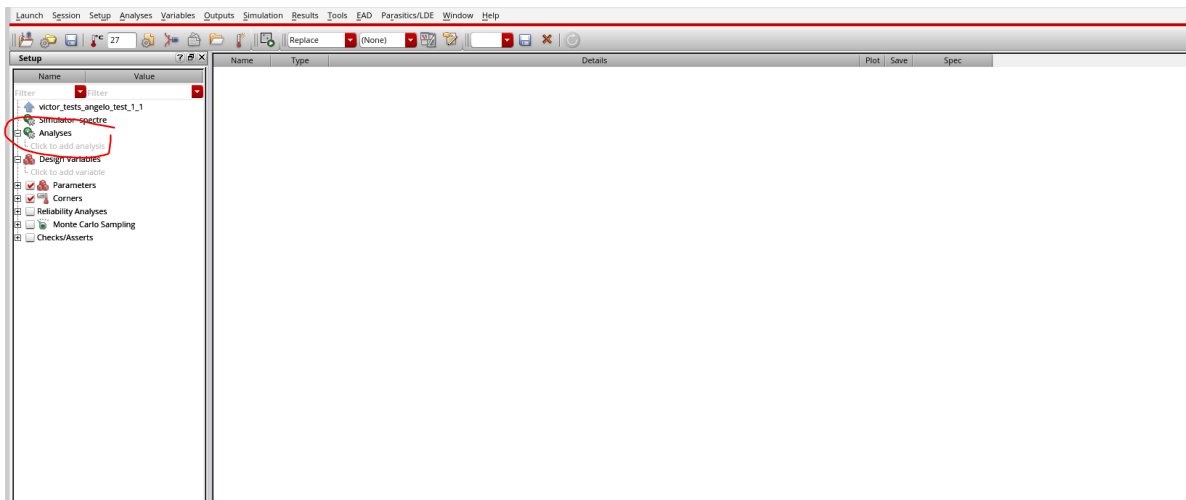
ADE Assembler will open. Click "Add new test":

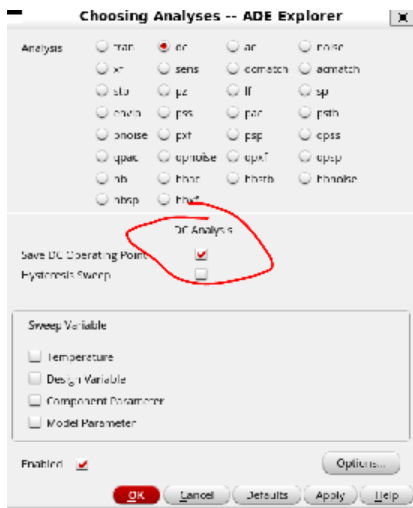


Choose your design:

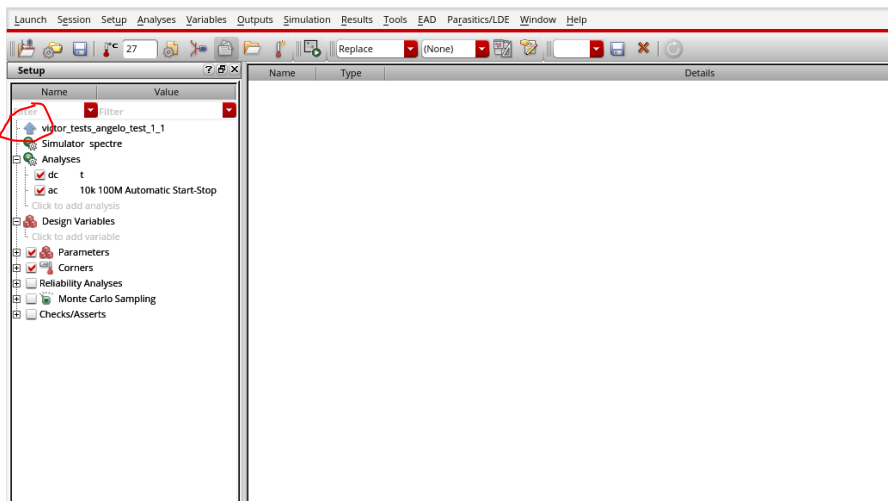


Add a DC Analysis with operating point and an AC Analysis:

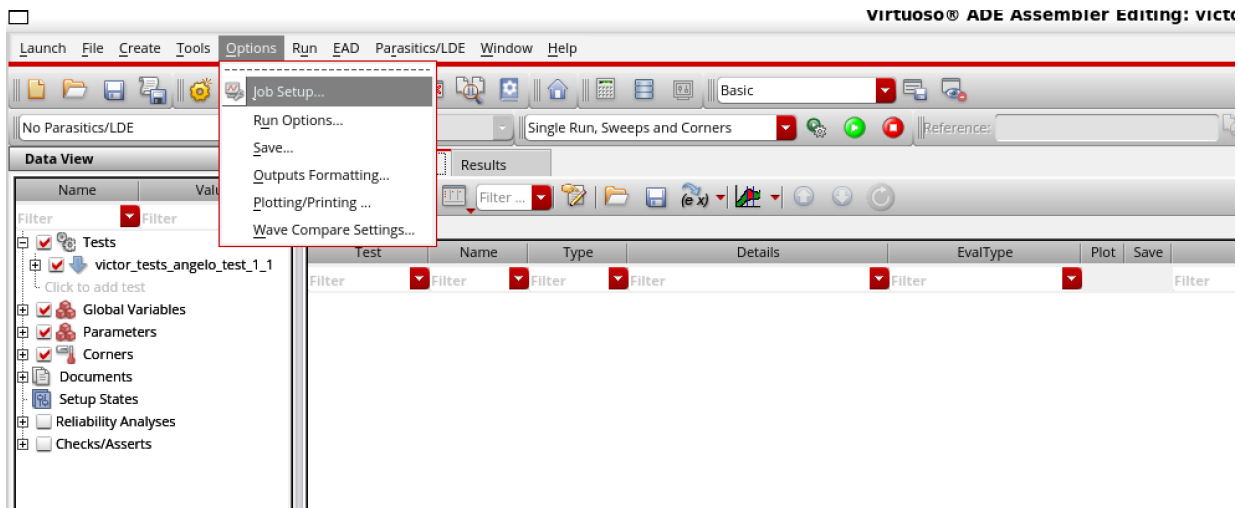




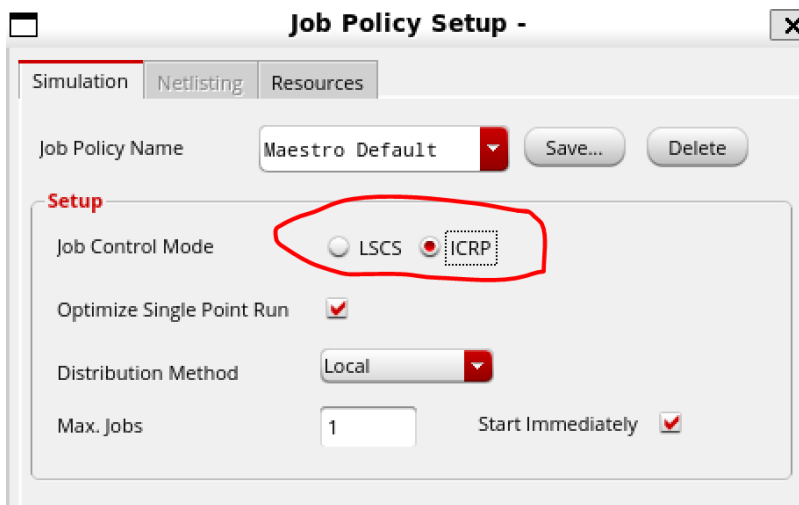
Press the blue arrow to return to the main page:



Before running tests, you may have to set up jobs. Go to Options Job Setup:



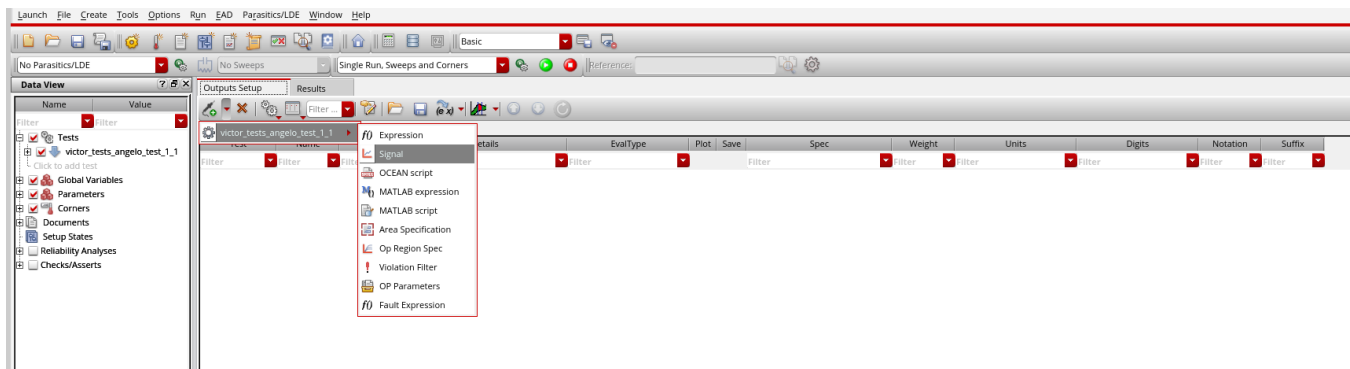
Set Job Control Mode to ICRP:



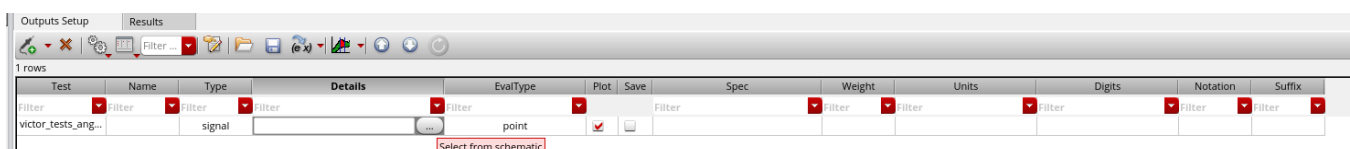
You are now ready to run tests by pressing the green play button!

Measuring voltages and currents

In the main page, you can add which signals should be plotted:

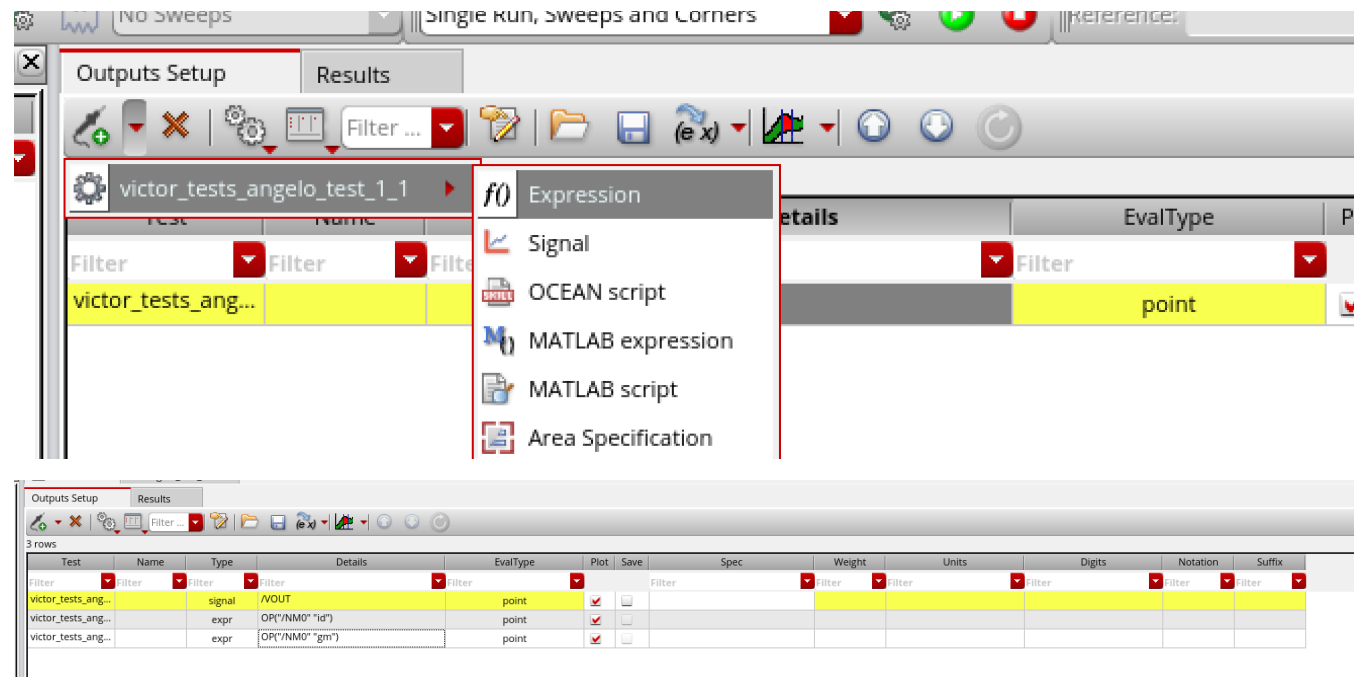


Easily pick an expression by clicking "Select From Schematic" in the "Details" column:



Measuring operating point parameters

To obtain operating point parameters, we need to add expressions.



Write the expression in the "Details" tab. Possible expressions include:

```
OP("/NM0" "gm")
OP("/NM0" "gds")
OP("/NM0" "id")
OP("/NM0" "cgg")
OP("/NM0" "cgs")
```

You may also have math, such as

```
(OP("/NM0" "gm") / OP("/NM0" "id"))
```

Another convenient tool is DC annotations. Right click on any cell in the outputs table, and select **annotate DC Node Voltages**. The annotations are in white/yellow.

