In this tutorial, we will learn how to access and use Cadence. The outline of this tutorial is as follows:

1. Accessing the UG machines
2. Accessing Cadence and adding the 45-nm CMOS library
3. Learning basic simulation setup, including DC and AC for a low-pass filter using ADE-L
4. Learning how to use the 45-nm CMOS technology files and performing simulations including transient simulations and DC sweep
5. Case study: Simulating a common-source amplifier

1. Accessing the UG machines

Please refer to the following files uploaded on Quercus for a detailed explanation on how to access the UG machines using different methods: "VNC-Scripted.pdf", "VNC Connection.pdf" and "ECE Workstation Fall 2025.txt". Here we will demonstrate a quick demonstration of accessing UG machines using PUTTY on a PC using the approach outlined in VNC-Scripted.

Open Putty and enter the host name as shown in Fig. 1, using the workstation numbers listed in ECE Workstation Fall 2025.txt. Click save and then open the connection.
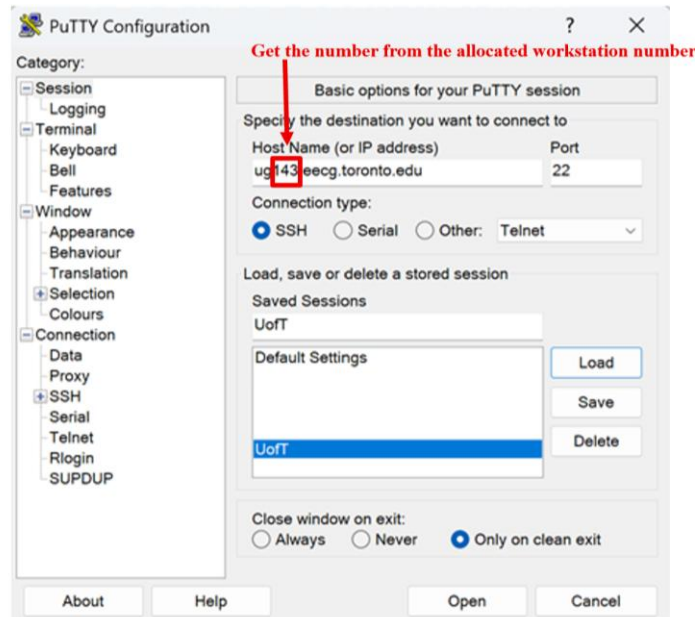

Fig. 1. Putty setup

Login using your UTORid and student number. Then use the "ece297vnc start" command as shown in Fig. 2 to start a new session. You can also used "ece297vnc list" and "ece297vnc stop" to list or terminate your active sessions as explained in VNC-Scripted.pdf. Once you successfully start a new session, go to putty settings as shown in Fig. 3, add the source port and destination. Click add, then go back to session and save and finally click apply.

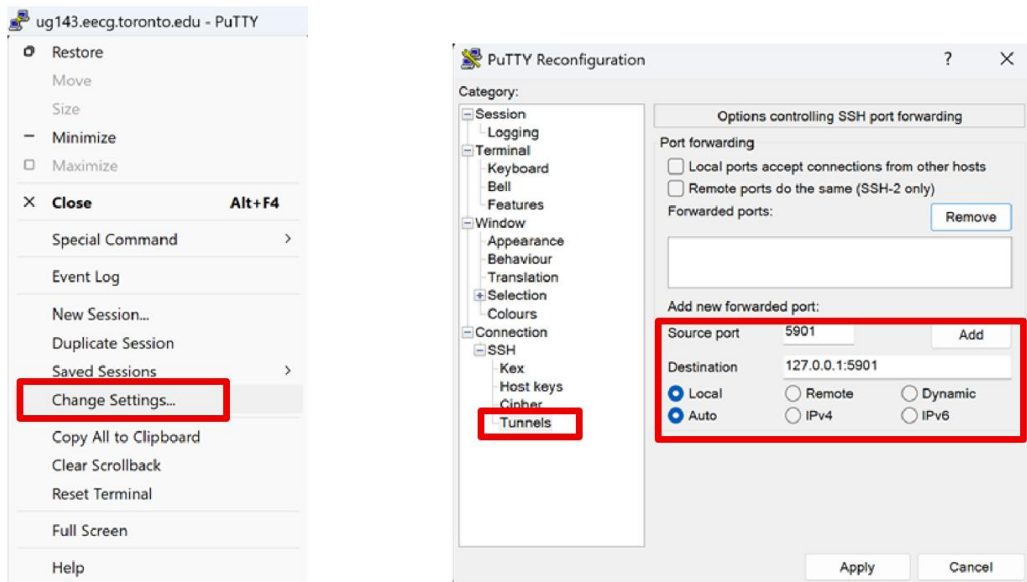

Fig. 2. Starting a new VNC session

Fig. 3. Changing the connection settings in Putty

Use a VNC (tightVNC, TigerVNC) to connect to your sessions using the remote host in Fig. 2, as shown in Fig. 4 and will now have access to your session desktop and files.
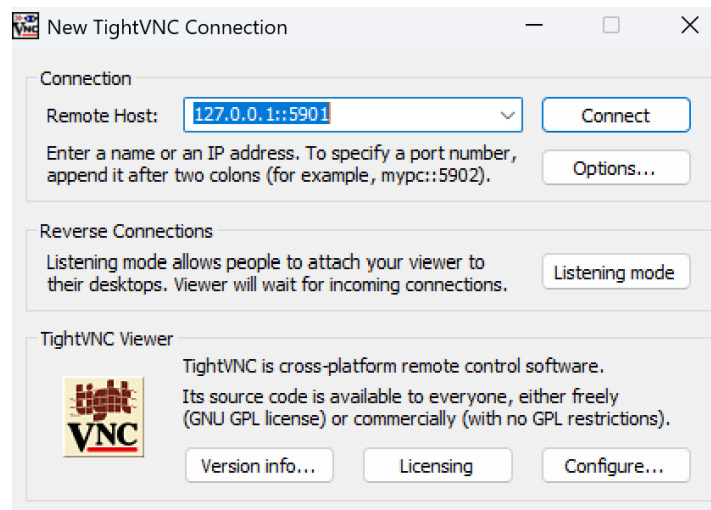
Fig. 4. Connecting to VNC

2.   **Accessing the 45-nm Technology File and Cadence**

Download the "example_45n_cadence.zip" file from the course website and move it to your directory folder. For this, you can either use the VNC internet browser and download it directly to your VNC or download the file on your own laptop and then use tools such as WinSCP to move any file to your VNC. Then unzip the file by going to the terminal and using the command shown in Fig. 5.

Fig. 5. Unzip example_45n_cadence.zip

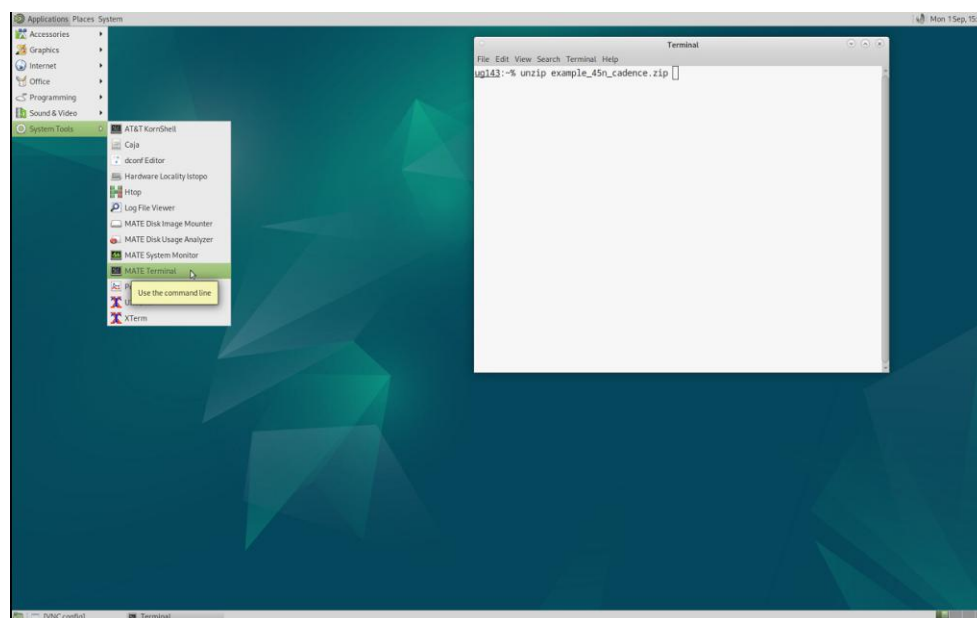Then we would have to load the directory and run Cadence (Fig. 6). We must also manually add the library files the first time, so run the pwd as shown in Fig. 6 to get the library path for adding the technology file. Once the virtuoso window opens up, go to tools, library path editor and add the library path and save (Fig. 6). Note that sometimes there might be a problem with saving the new library path as there are no other libraries yet. In this case, delete the new library path you added, then from file, create a new library (called init) (the settings don't matter as we will delete this library later), then add the library path. After this step, delete the (init) library.
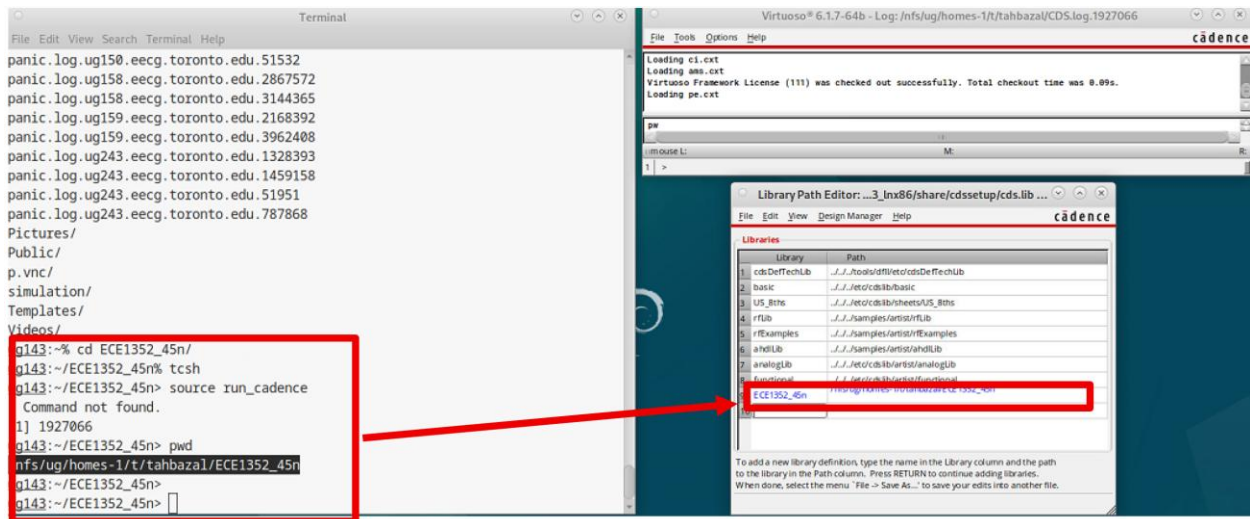

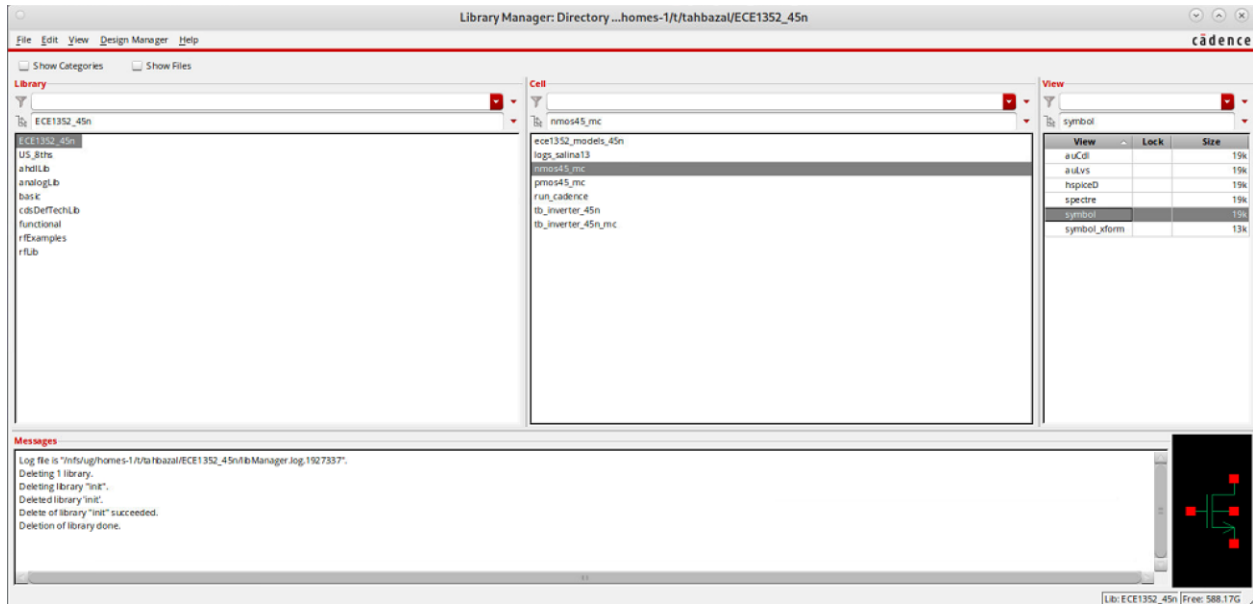**Fig. 6.** Opening Cadance and adding the library path

Fig. 7. Opening Cadance and adding the library path

Note that the window shown in Fig. 6 is called Command Interpreter Window (CIW), which is the main window of the Cadence software. This window enables us to have control over different parts of Cadence. Using this window, you can view warnings, errors (if any!), and other informational messages. **So anytime you face a problem (the schematic cannot be saved, the simulation gets stuck) you should take a look at this window.**

From the libraries shown in Fig. 7, we will use the following:
- analogLib: contains basic analog components (resistor, capacitor, inductor, voltage and current sources, ground, and etc.)
- Technology file (ECE1352_45n): the technology file you use during this course (It contains transistor models). We will learn how to use this technology file in Part 4 of this tutorial.

3. Simulating a Low-Pass Filter Using ADE-L

In the Library Manager click on the library you just added and then "File >> New >> Cell View". A new window pops up, but it might be at the background (Fig. 8). Enter the cell name. Since we want to design and simulate a simple low pass filter, I choose "Low_Pass_Filter" (spacing is not accepted in the naming). You can select the type of the cell view (schematic, layout, etc). Here, we choose Schematic (the "View" will be filled automatically). In the Application section you can choose the Cadence tool you want to use. Here, we use Schematics L. After clicking on OK, the schematic editing tool will open (Fig. 8). This is the main window we draw our circuits. Generally, we do not use menus, but keyboard hotkeys which some of them are summarized in Table I. Learning hotkeys in Cadence will speed up the design process dramatically. One of the most important hotkeys is "F3". If you hit a hotkey, and then hit "F3" immediately afterwards, it brings up a window detailing the options associated with that particular hotkey (for example for copying, you can press c then F3 to copy something multiple times).
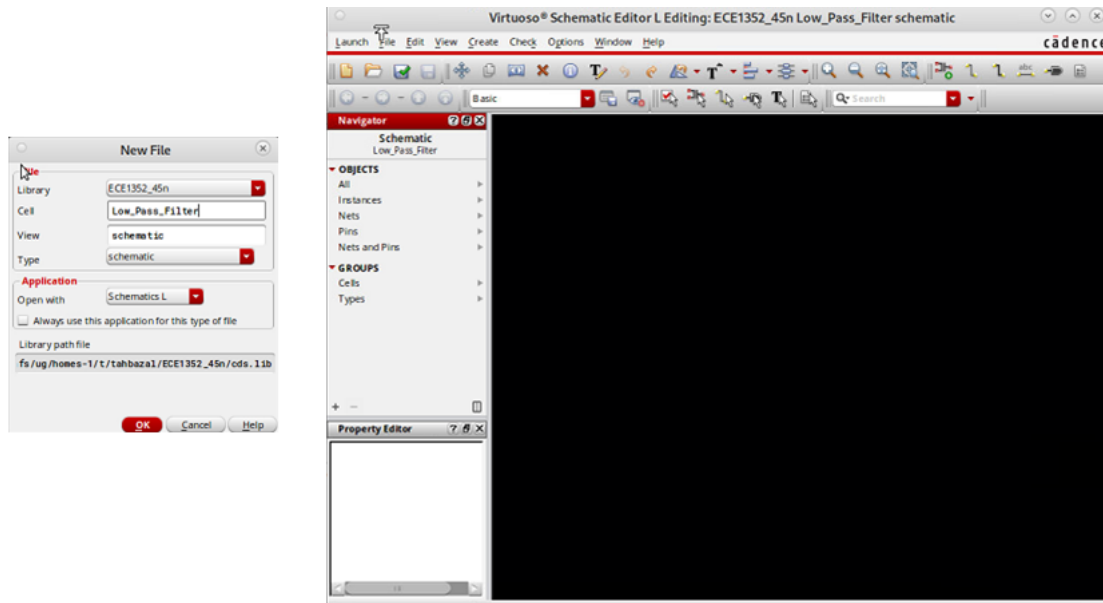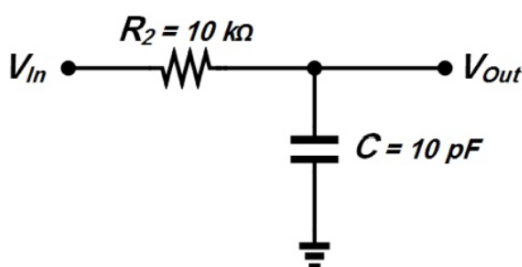

Fig. 8. Creating a new cell view and schematic editor

Table I Hotkeys used in schematic editor

| Hotkey	Function
i	Add instance
w	Wire up
f	Fit all
q	Edit Properties
c	Copy
m	Move objects and drags their wiring with them
M	Move objects after disconnecting their wires
p	Add pin
l	Wire label
r	Rotate
u	Undo
del	Delete
Esc	Cancel previous command

Fig. 9. Designed low-pass filter

We want to simulate the low pass filter shown in Fig. 9 using Cadence. First, we start by adding the resistor. To add instance press "i" or equivalently select "instance" from the "Create" menu. The window shown in Fig. 10 will pop up. You can type the library, cell, and view names or click Browse (left side of Fig. 11). Since we use "analogLib" frequently, it is better to memorize Table II. Select "analogLib" library, "res" cell, and "symbol" view. "Add instance" window will pop up (right side of Fig. 11). You specify the component parameters here. For resistor we only enter its value (there is no need to write units (Ohm,V,A), just enter the value). Here we put the value as a variable Rf, where we could have just entered the value. Putting the value would allow us to easily sweep the resistor value in the next steps.

Go back to the schematic and place the resistor wherever you want by left click. You can add multiple resistors by simply left clicking in different places of the schematic. Pressing "ESC" on the keyboard will end the add instance mode. If necessary, click on "Rotate", "Sideways", or "Upside Down" in the "Add instance" window. Do the same steps for the capacitor and ground. If you want to change the component parameters after you placed them, select the component on the schematic by left clicking on it and pressing "q" on keyboard. To connect the components with wire, press "w" on keyboard. For drawing wires, left click on each terminal (red square) and move the mouse to the other desired terminal. To create label for wires, press "l" on keyboard, then type out the desired name and left click on the wire. We use "Vout" and "Vin" for output and input node, as shown in Fig. 12. For the input, we have chosen "vsource" that has DC, AC and transient components as explained in Fig. 13.

Table II Frequently used analoglib components

Cell Name	Component
res	Resistor
cap	Capacitor
ind	Inductor
gnd	Ground
vdc/idc	DC voltage/current source
vsin/isin	Sinusoidal voltage/current source
vpulse/ipulse	Pulse voltage/current source

Fig. 10. Add instance window
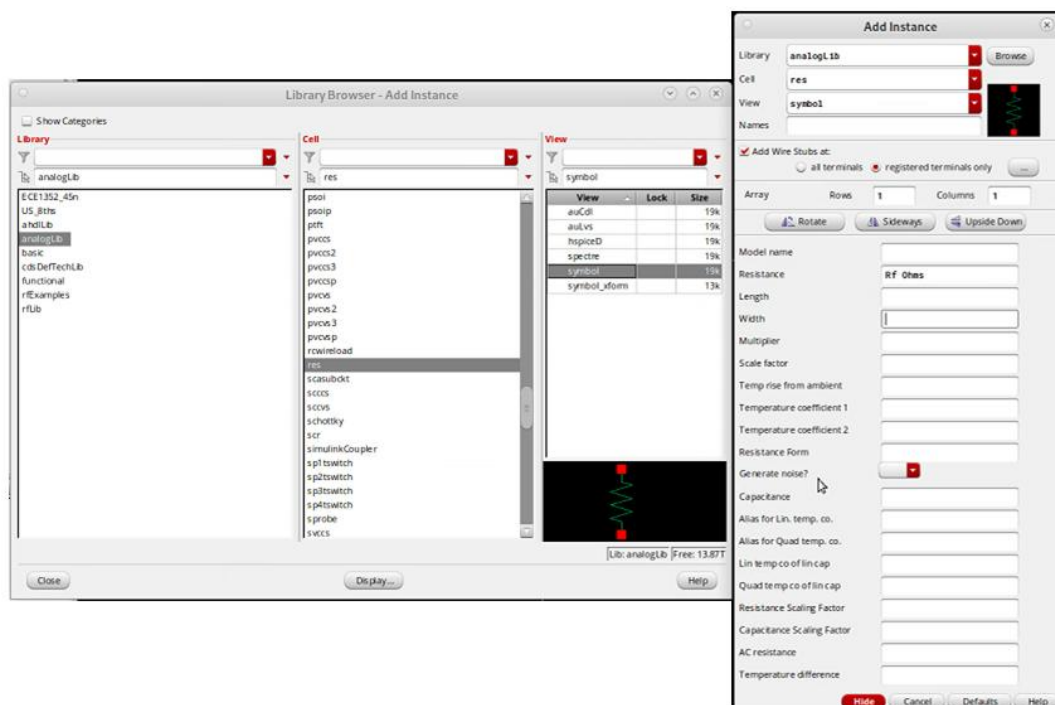
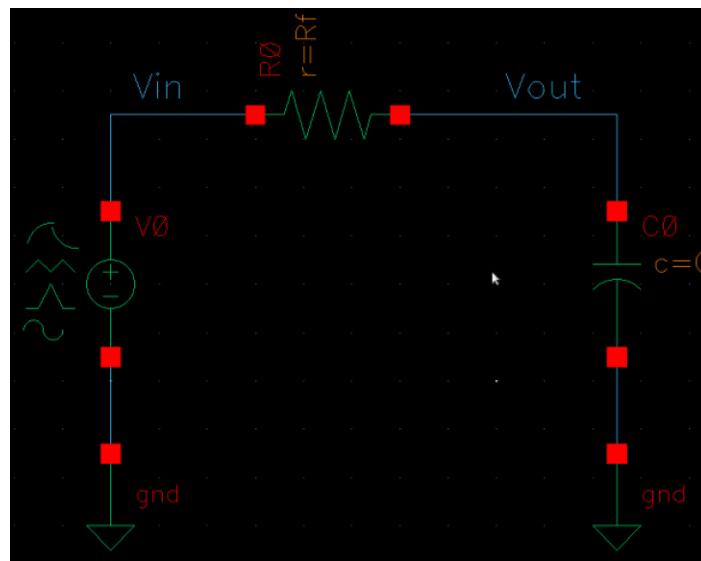**Fig. 11.** After selecting Browse in Fig. 10

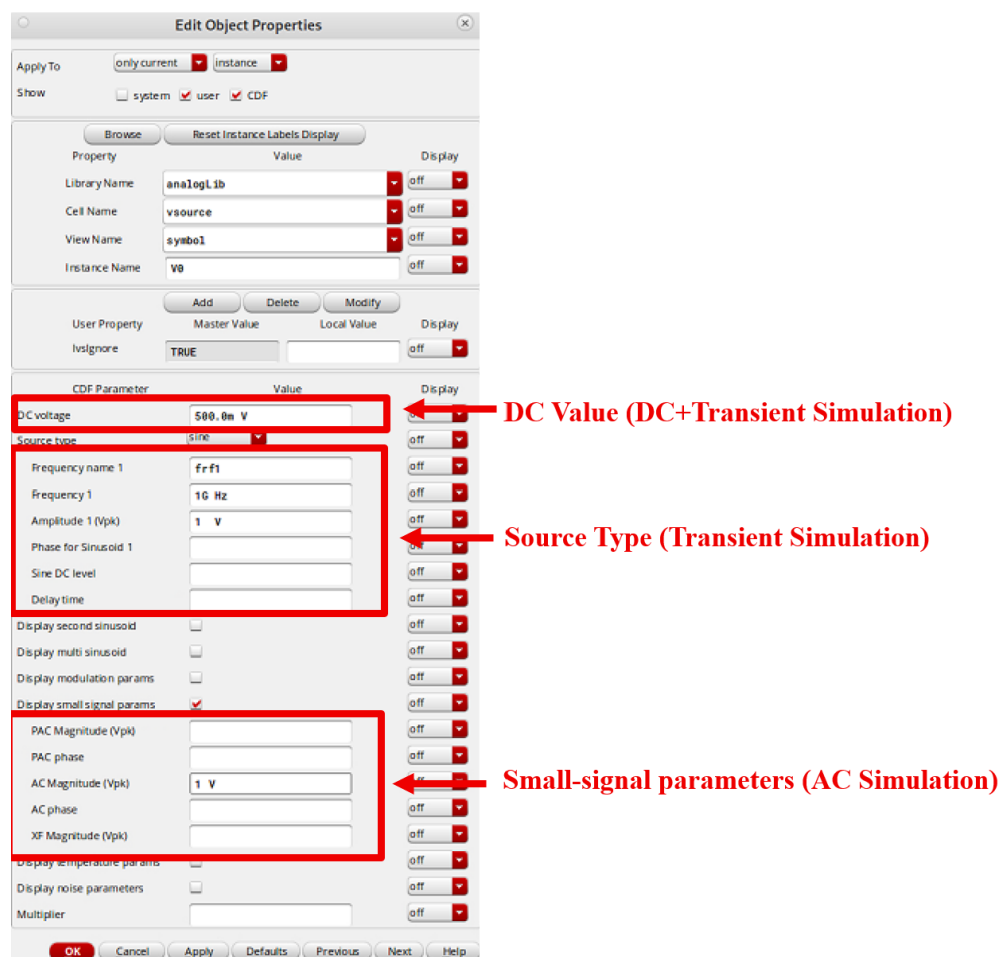**Fig. 12.** Designed low-pass filter in Cadence

Fig. 13. Vsource details

Press "Shift+X" to check and save the schematic or alternatively click on the check and save sign on the schematic. If there is any error or warning, check CIW for details. **Every time you change the schematic, you must check and save it again, otherwise you will get an error while you want to run simulations.**

To open the simulation window, click on "Launch >> ADE L" in the main schematic window. The ADE window pops up (Fig. 14).

- Through the "Session" menu you can save and load your simulation setting. So, you do not need to setup again next time you open ADE L.
- You can choose different analyses to enable or disable from "Analyses" menu. You can click on the "choose analyses" from right side of the ADE L window too.
- "Variables" menu helps you to configure different variables you used in your design. To see a list of all variables you used click on "Variables >> Copy From Cellview". The list of variables will be shown in "Design Variables" box (Fig. 14).
- Generally, all node voltages will be saved unless you are running a complicated system and limit the number of saved node voltages to save memory. You can control which node voltages and terminal currents to be saved using "Outputs >> Save All".
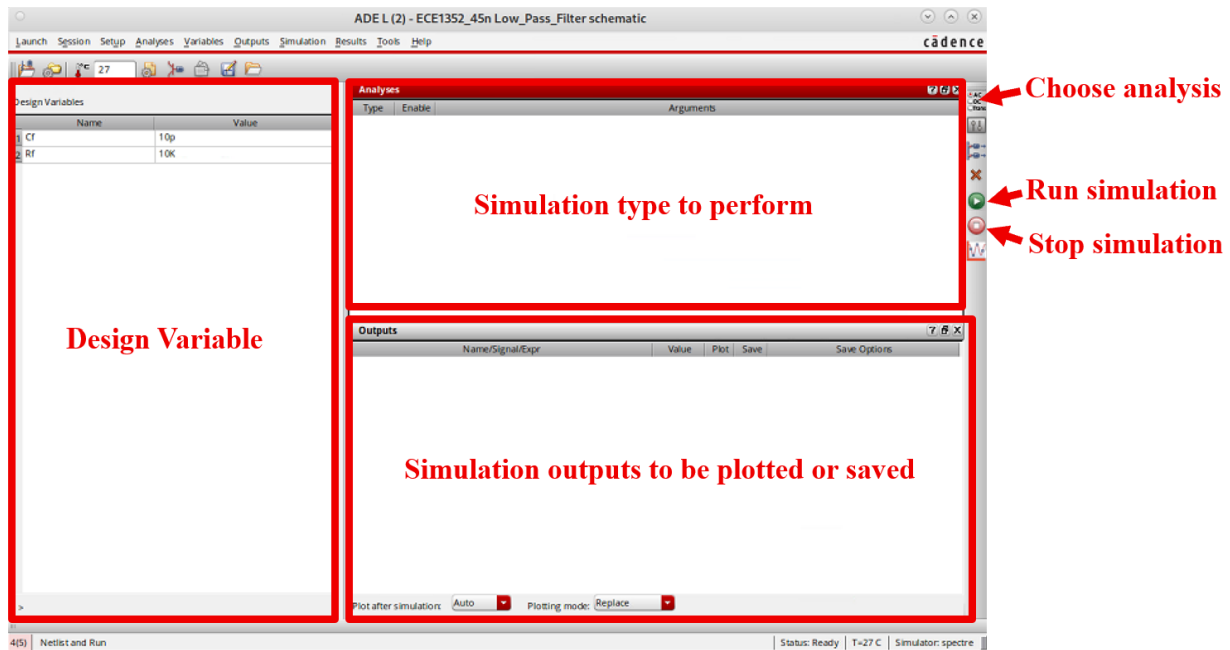

Fig. 14. ADE window

DC Simulation

To perform DC simulation, click on the "choose analyses" from right panel and select dc (Fig. 15). Check "Save DC Operating point" and click on OK. You can perform a DC sweep here, as will be explain in Part 4 of this tutorial and is often used to characterize transistors. Also note that by saving DC operating points, you can check the operating point of transistors as well (Part 4). Run the simulation. To see the DC Node Voltages, DC operating points, and etc. on the schematic you can use "Results >> Annotate" options on top of ADE menu (Fig. 16). The results will be shown in the schematic as Fig.16. Note that the results can also be printed by by going through the print option and clicking on the desired node voltage.
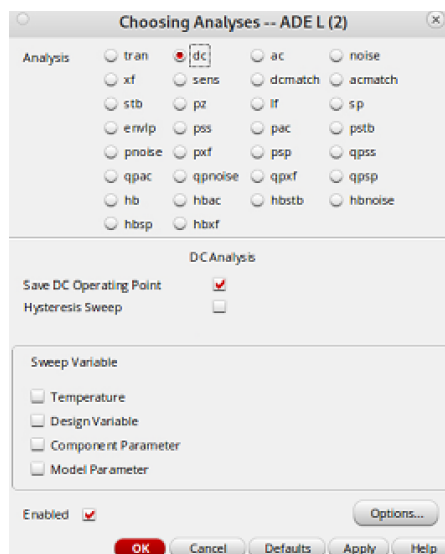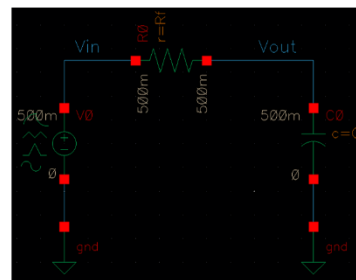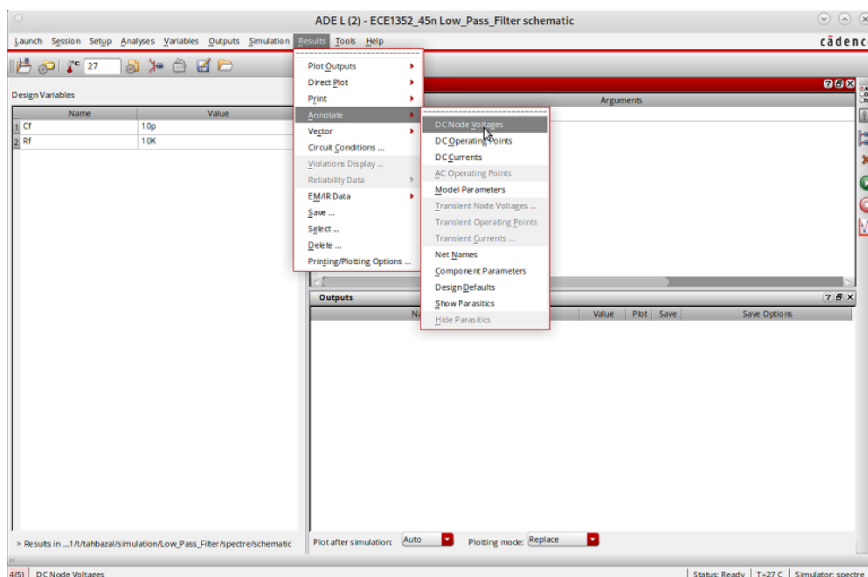

Fig. 15. DC analysis

Fig. 16. Using "Annotate" option to see the DC operating point on the schematic and the node voltages

AC Simulation

To perform AC simulation, click on the "choose analyses" from right panel and select AC (Figure 14). We will sweep the frequency in a logarithmic scale from 10k to 10G. Fill the "Sweep Range" box as shown in Fig. 17. Click on OK and then run the simulation. Since we chose the input AC magnitude to be 1, simply the output voltage will show the transfer function of the low pass filter.
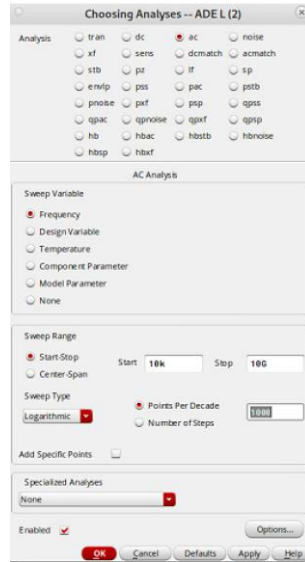

Fig. 17. AC analysis

To see the output voltage waveform, click on "Results >> Direct Plot >> Main Form" in the ADE window (Fig. 18). Direct Plot Form window will pop up and ask you to select a net on schematic (left side of Fig. 19). You can choose voltage or current, magnitude (dB10 or dB20) or phase of the node. Put the modifier on dB20 (20 log) and select the output net on the schematic. You should see the waveform of Fig. 9.
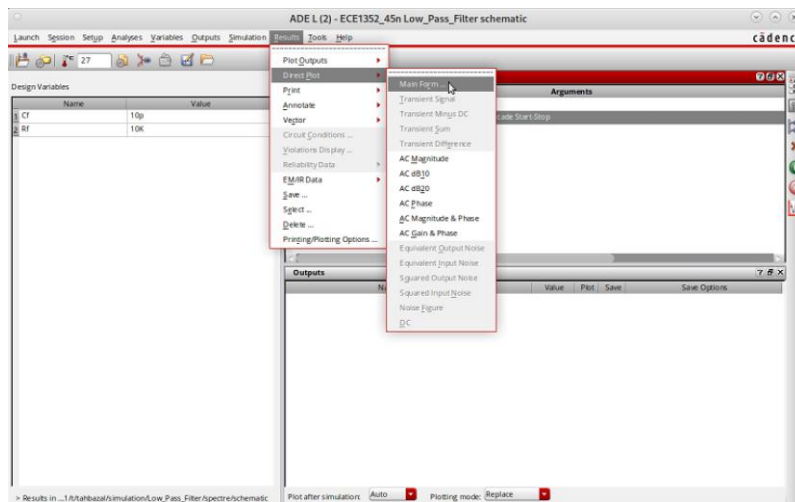

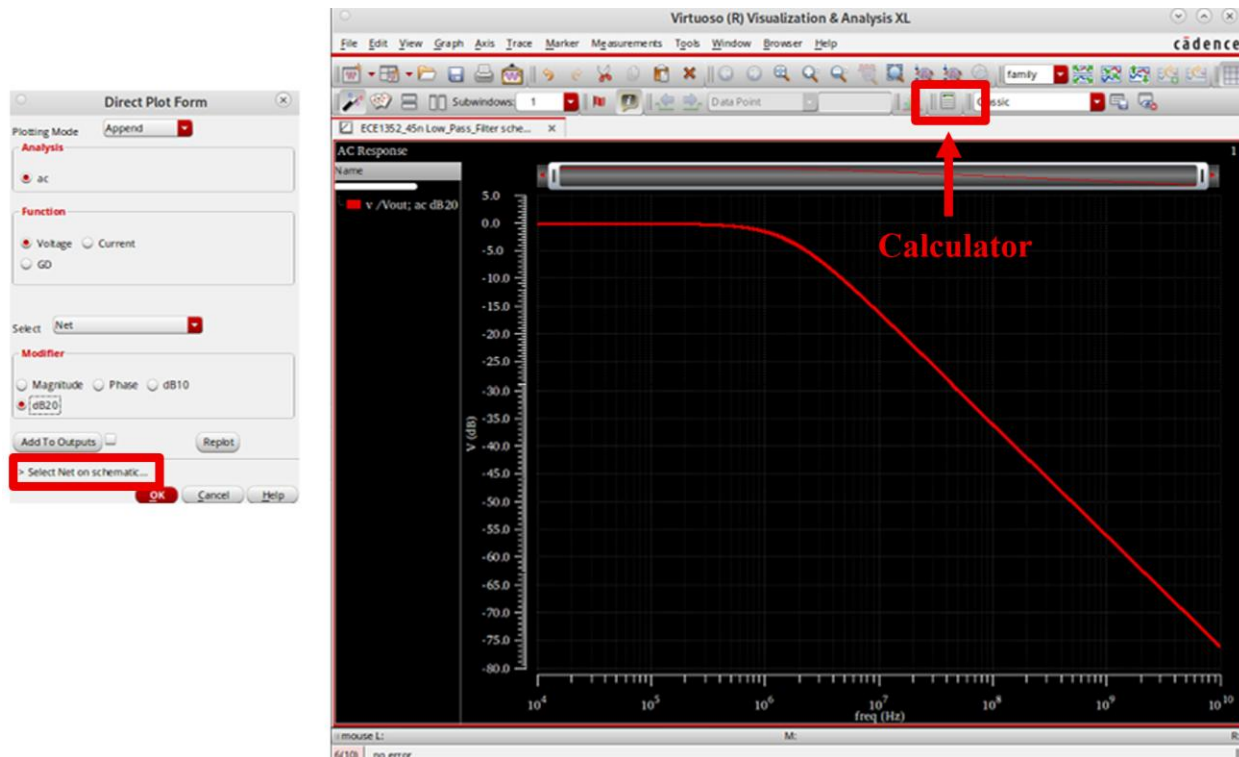**Fig. 18.** Using "Direct plot" option to plot the transfer function

Fig. 19. AC simulation result

For measurement purposes on the waveform, you can:
- Press "v" for vertical bar or "h" for horizontal bar and drag the bar to the place you want with mouse.
- Press "m" to place a marker you want. Then press "d" to place another marker (you can change marker place by dragging it with mouse). In this case, you will be able to see the dx, dy and the slope between two points.
- To zoom in a specific part of the waveform hold right click down and determine the part you want.
- To zoom in horizontally/vertically hold down shift/ctrl and use mouse wheel.

Calculator Setting

One of the most useful tools in Cadence is calculator. If the waveform is open, you can access to calculator by clicking on its sign on the top menu (Fig 20). At the bottom of the calculator you can see special functions that perform different functions on the selected waveform (multiply, divide, THD, DFT, maximum and minimum, etc).
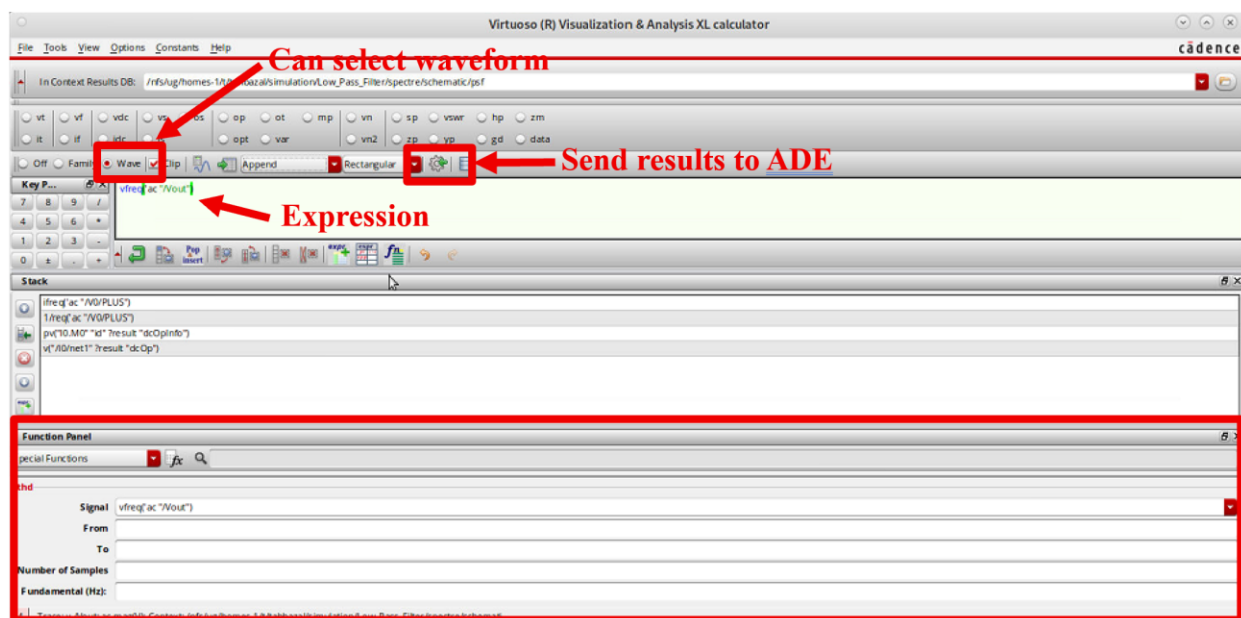
Fig. 20. Calculator settings

Here we want to find the input impedance using calculator which cannot be directly calculated using AC simulation. To this end we define the input voltage (which is 1 V) divided by input current as an output. Now, plot the current of the input source by clicking on the positive terminal of the input source and take the results to calculate the input impedance. By this, the expression of input current will appear in the calculator as shown in Fig. 21. It is important to note that, only source (DC, AC, etc.) terminal currents are saved automatically. If you want other terminal currents to be saved, you have to change the setting in "Outputs>> Save All" in ADE window. You can also add the result to the ADE output (will appear in the ADE window) so that every time you run the simulation, this expression is calculated. The obtained Zin in this case is shown in Fig. 22. Also note that the axis scale can be changed by clicking on it and then changing the scale as desired.
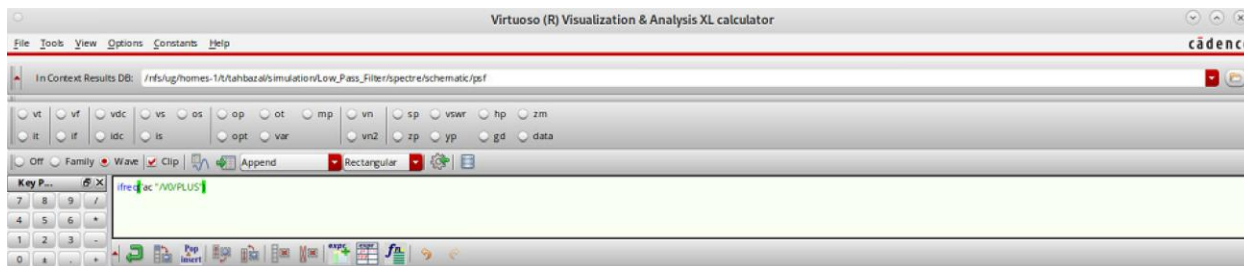

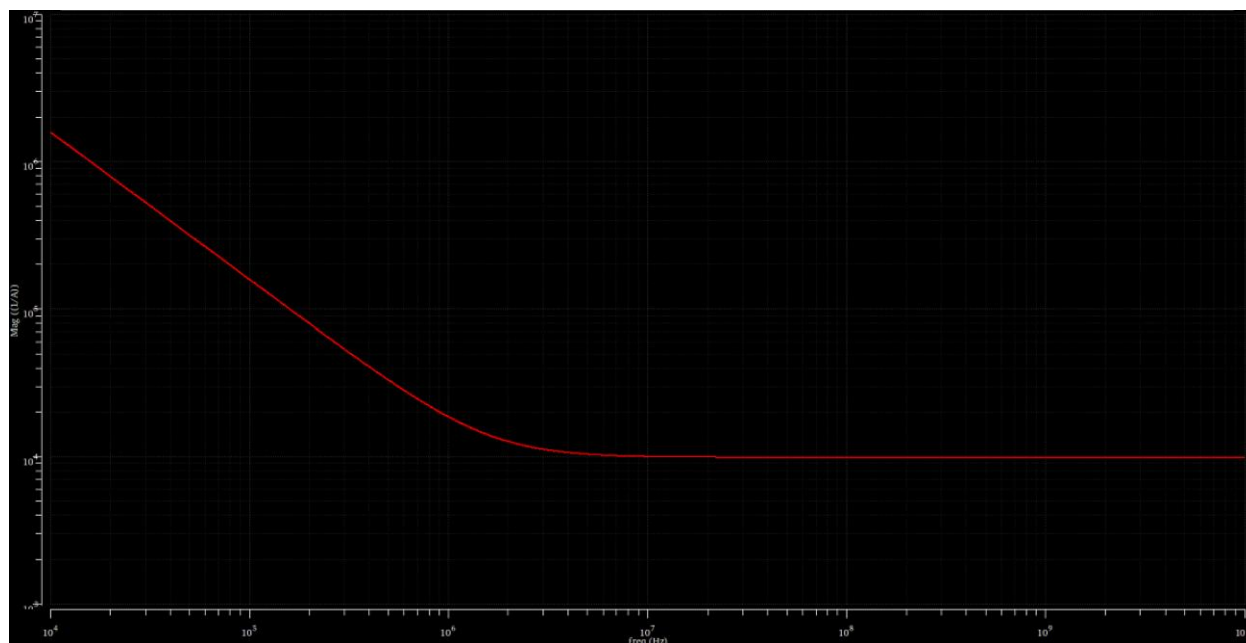**Fig. 21.** Calculator settings for obtaining the input impedance

13

Fig. 22. Obtained input impedance of the low-pass filter using AC simulation

Parametric Sweep

Here we want to sweep the capacitor value and see its effect on output voltage. In the ADE window click on "Tools >> Parametric Analysis" and fill the sections as Fig. 23 (Sweep from 10p to 100p in 10 linear step). To run the sweep, click on the green button (play button) on the parametric analysis window. At the end, in ADE window, click on "Results >> Direct Plot >> Main Form" and select output voltage, where the transfer function is illustrated for different capacitor values as shown in Fig. 24.
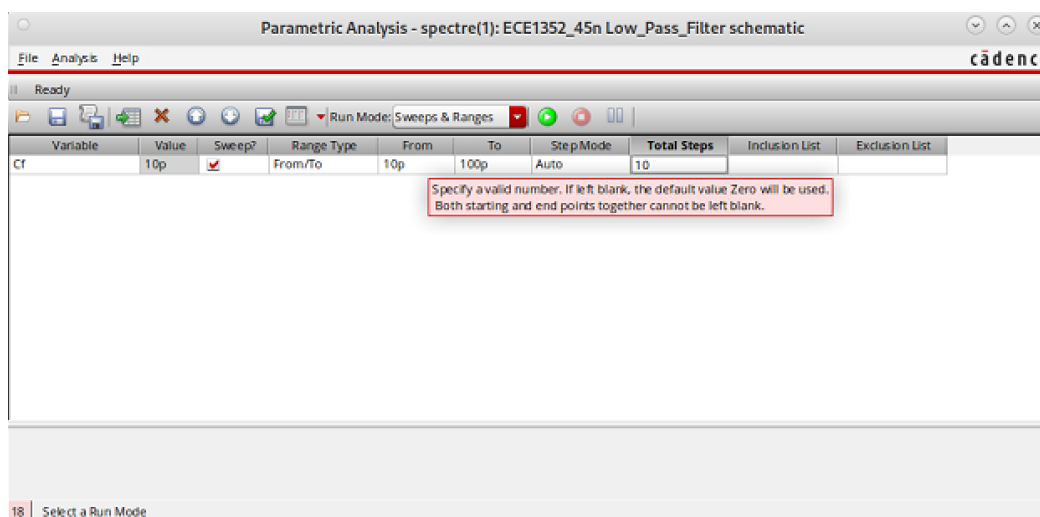

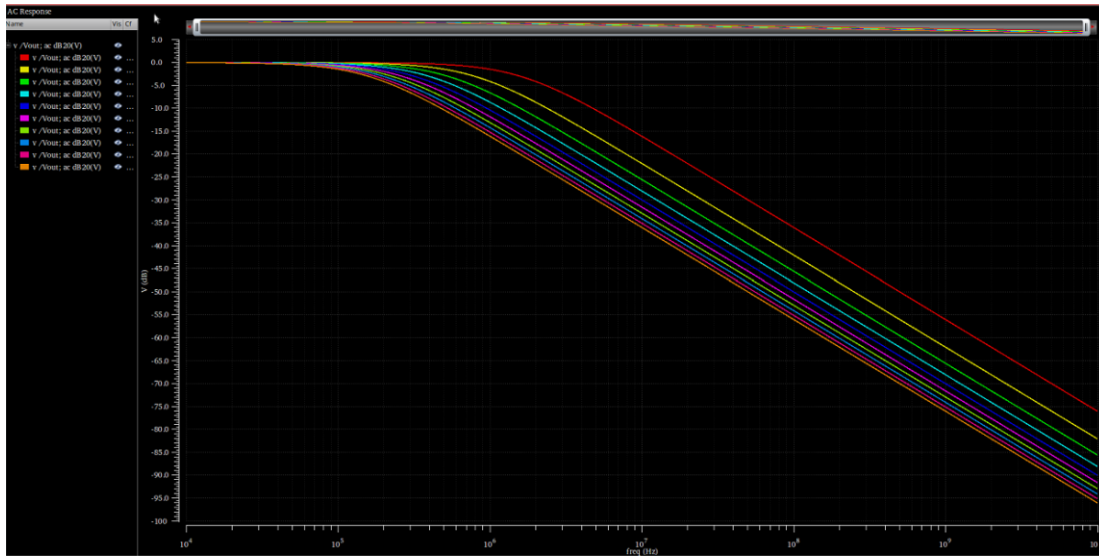**Fig. 23.** Parametric analysis setup

Fig. 24. Parametric sweep AC simulation results

As an exercise, we want to plot the 3-dB bandwidth versus the capacitor value. For this purpose, we need to use the calculator and the parametric sweep function. Plot the transfer characteristic as done in Fig. 19 for a capacitor of 10pF (or any other value), but this time in magnitude (not dB). Then take the results to the calculator. Then use the special function "bandwidth" (which calculate the 3-dB bandwidth) as shown in Fig. 25 in the calculator and send the results to the ADE. The results will not be shown in the ADE window (Fig. 26). Now if we run a parametric sweep as shown in Fig. 23, and then plot bandwidth from the ADE window, we will get bandwidth versus capacitor value as shown in Fig. 27.
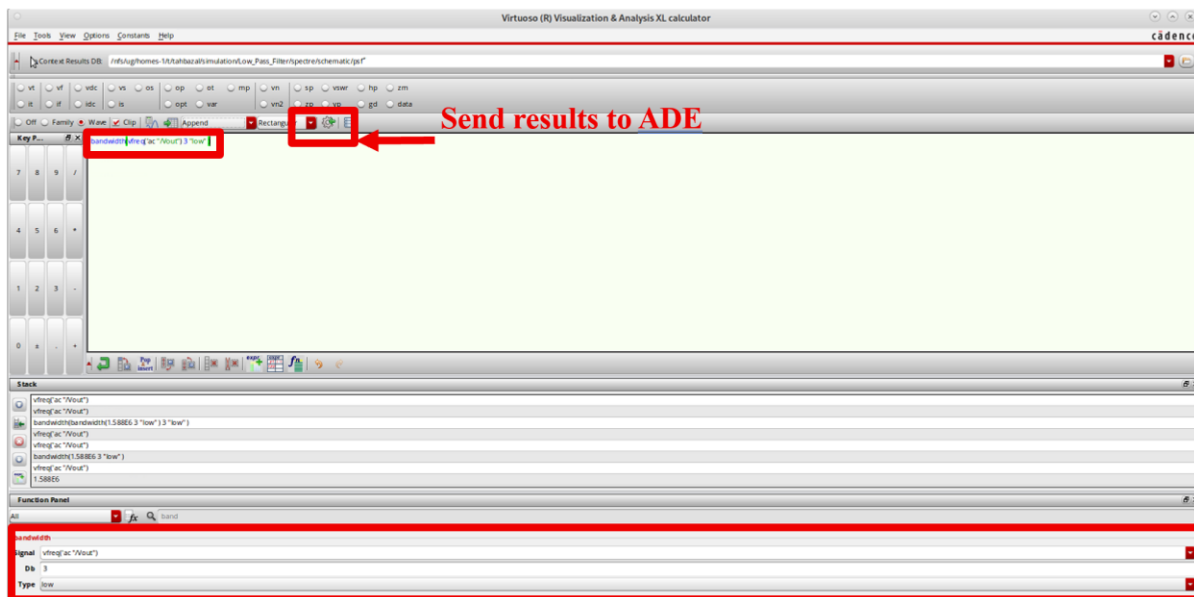

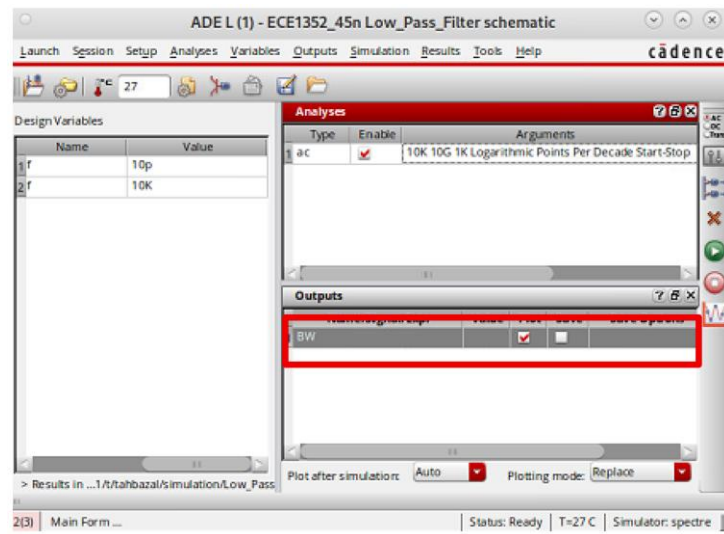**Fig. 25.** Using bandwidth function in the calculator

Fig. 26. Output of the calculator is added to the ADE output
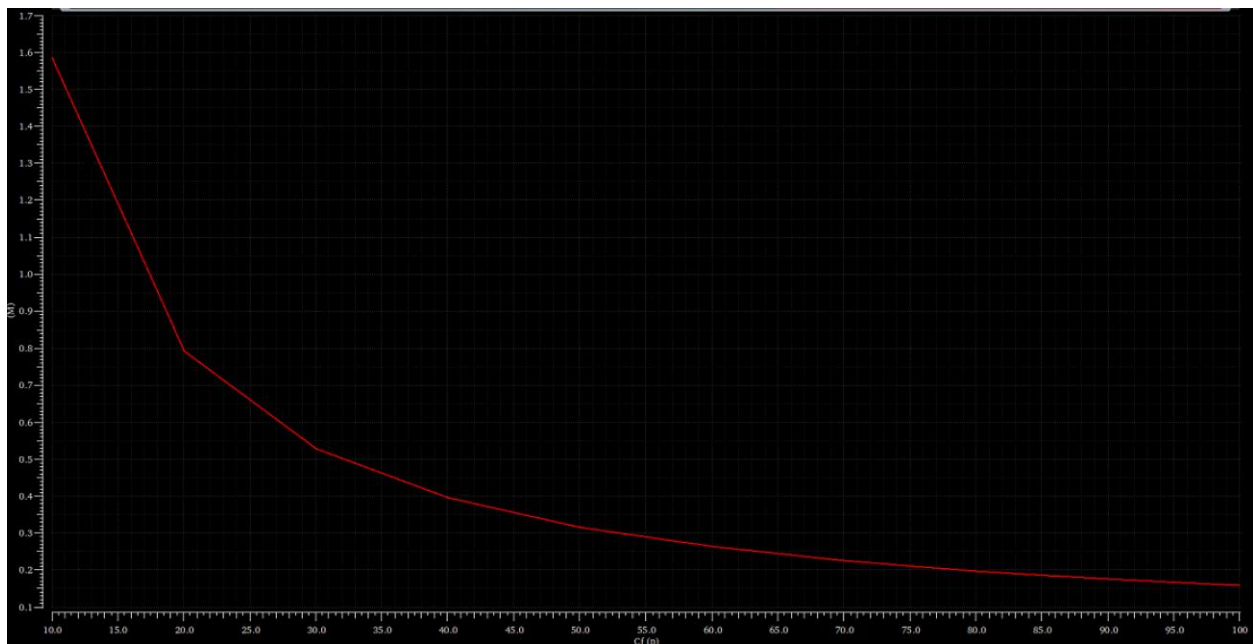

Fig. 27. 3-dB bandwidth versus the capacitor value

Create Symbols

There might be a need to create a symbol for a designed circuit. This can be because you aim to use multiple of these circuits or that you have a more complicated overall circuit. For this reason, we typically remove all of the source and then put pins on the inputs, outputs and supply connections by pressing "p" on the schematic and then placing pins accordingly (It is recommended that pins all be input/output pins). For the low_pass filter, the updated schematic with pins is shown in Fig. 28. Also, we cannot have component values left as variable, rather we can put their design values or use "pPar("ValueVariable")" as the component value as shown in Fig. 28, and later set the design value at the property of the instance.

After putting the pins, go to Create>>Cellview>>From Cellview to create the symbol for your design (Fig. 29). Finally you can move the pin connection in the symbol editor and draw your preferred symbol. Once you check and save, you can place this instance in another schematic similar to how we placed resistors and capacitors from the design library instead of analoglib. As an example, Fig. 30 shows a schematic using two of the designed low-pass filters cascaded with the first filter having Rf1 and Cf1 values with input source connected to the input. Note that in a top schematic level, you can use "shift+e" to descend into a symbol and "ctrl+e" to ascend.
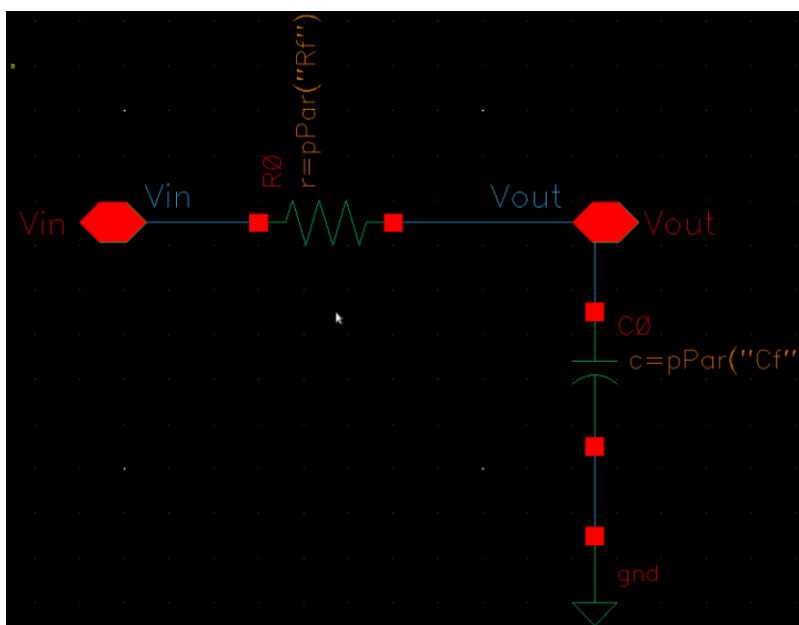

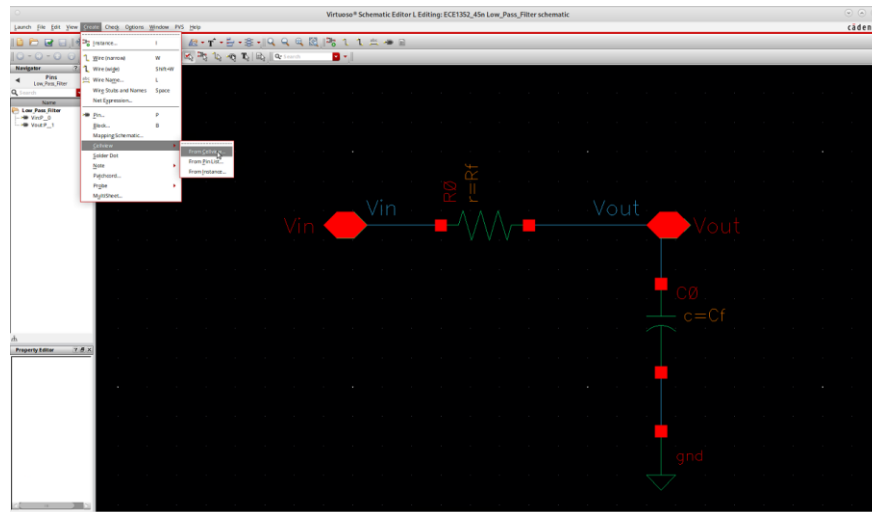**Fig. 28.** Updated low-pass filter schematic with pins

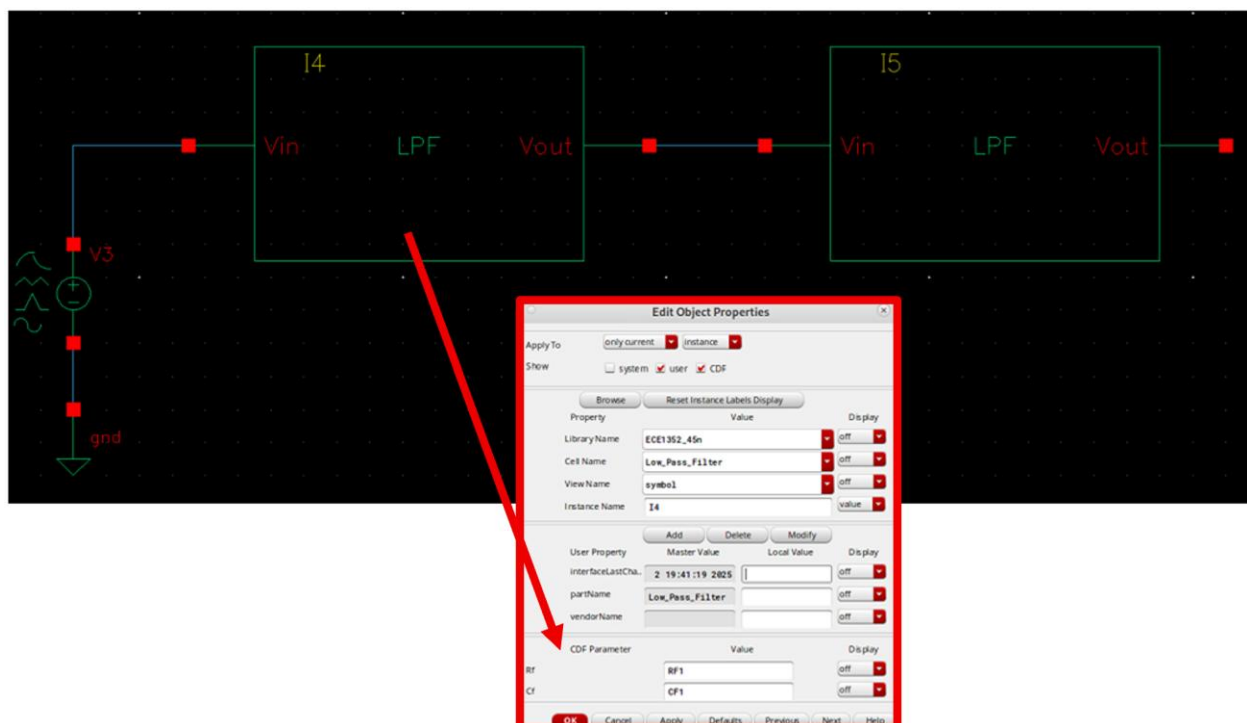**Fig. 29.** Updated low-pass filter schematic with pins

Fig. 30. Cascode of two low-pass filters with the first filter having Rf1 and Cf1 component values

Transient Simulation

We want to see the effect of input pulse train at the output. Replace the vsource at the input with a "vpulse" from "analogLib" and fill the necessary parameters as shown in Fig. 31. Please note that a vsource is also capable of generating a pulse train. However, vpulse is typically used for generating pulse trains. In the ADE window, add transient simulation with 1µs and choose the mode depending on the complexity of the circuit and the accuracy that you want (Fig. 32). Higher accuracy slowdowns the simulation. Run the simulation and plot the results using direct plot and you should see the results as Fig. 33.
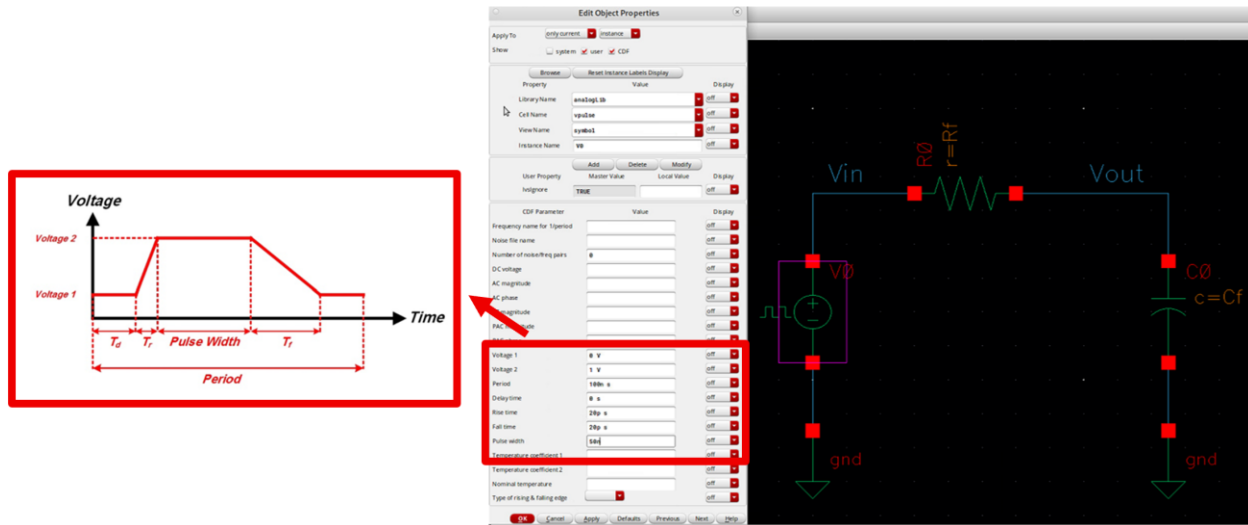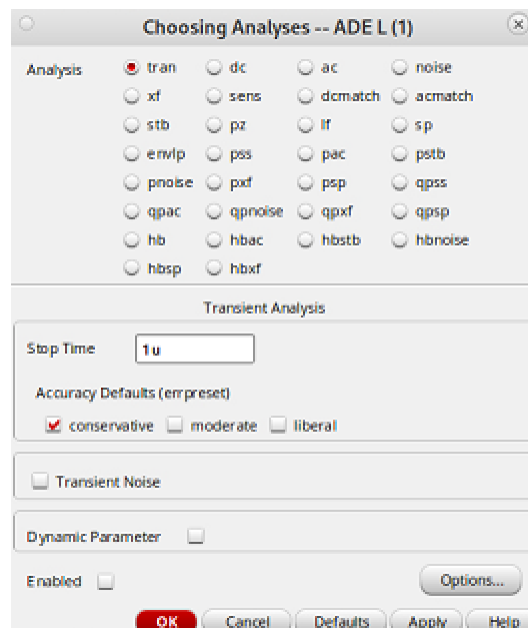

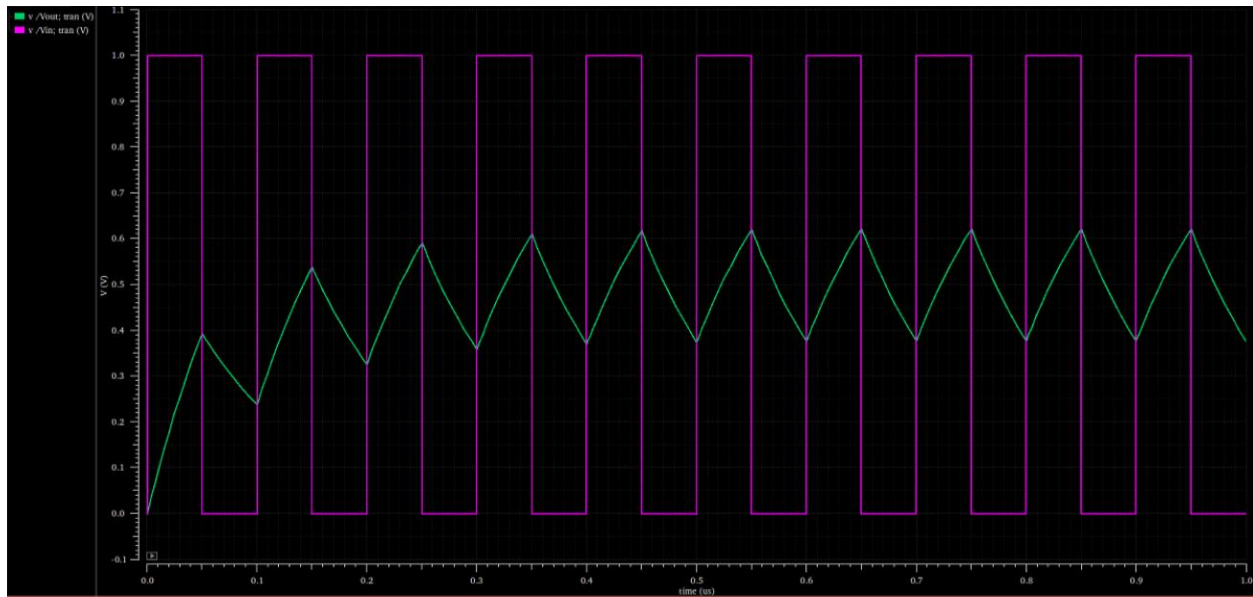**Fig. 31.** Vpulse setups

Fig. 32. Transient analysis

Fig. 33. Transient simulation results

4. Accessing 45-nm Technology File and Simulating an Inverter

Open the inverter schematic "tb_inverter_45n" from the library path added in Part 2 (Fig. 34). As can be seen from here, we use the devices "pmos4" and "nmos4" from the "analogLib" library and add the width and length for each transistor according to our design (Fig. 34). However, in the next steps we would need to setup the model library for the simulations to run. Launch ADE-L and copy the variables. Set "per" as 1ns and choose a transient analysis up to 20nsec.
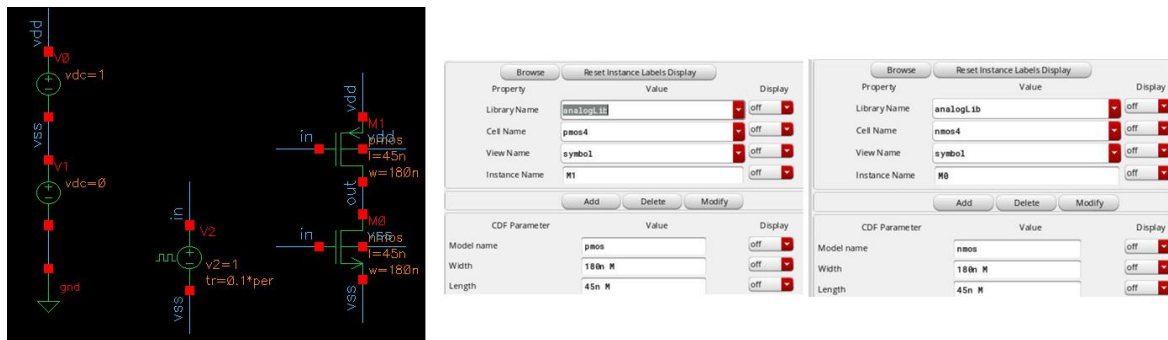

Fig. 34. Designed inverter using 45-nm CMOS transistors

Before running the simulation, we need to add the model library. For this, from the ADE-L window, go to Model Libraries, and then add the "library_file.scs" from the added library path. After that, you can select the process corner which you are trying to simulate with (Fig. 35). Afterwards, you can run your simulation. The transient input and output waveforms of the inverter are plotted in Fig. 36.

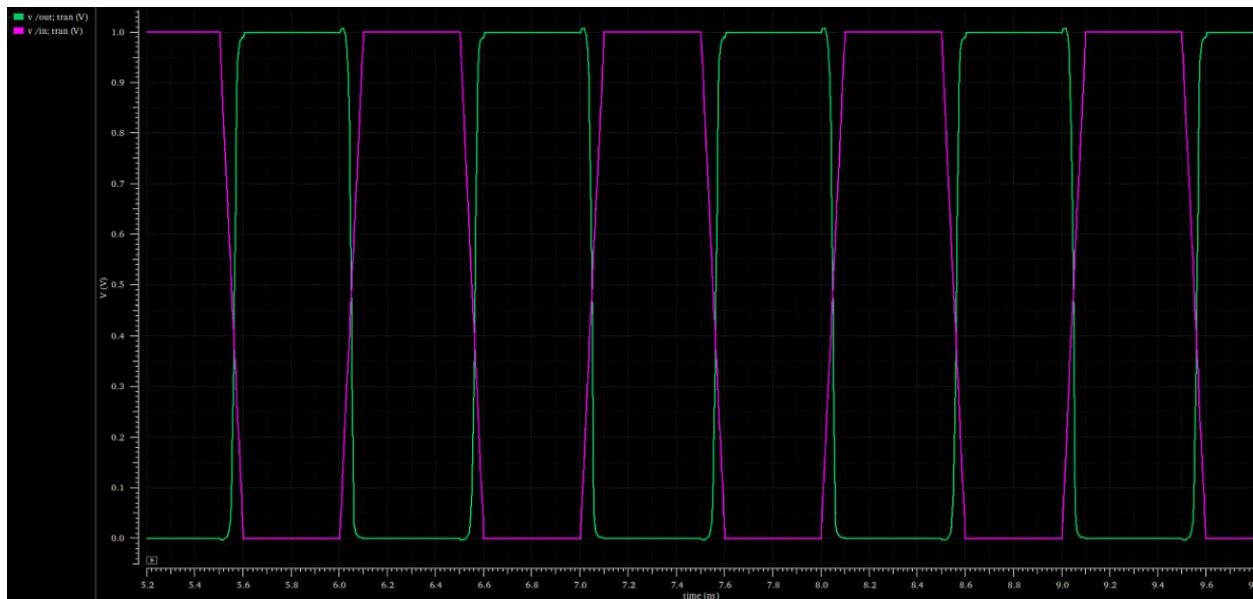**Fig. 35.** Adding the 45-nm model library

Fig. 36. Inverter transient simulation results

DC Sweep

Another important simulation setup is the DC sweep, where we find the transfer characteristics of a circuit or a transfer parameter versus different input DC voltages. For this simulation, we have to modify the input source as shown in Fig. 37, where we model the input DC voltage as Vin and plan to sweep this in the simulation. After check and save, go back to ADE-L window and copy the variable. Give it a value (such as 0, it doesn't matter as we plan to sweep it). Then open the DC analysis and set it up as shown in Fig. 38, where we sweep Vin from 0V to 1V in 1000 steps.
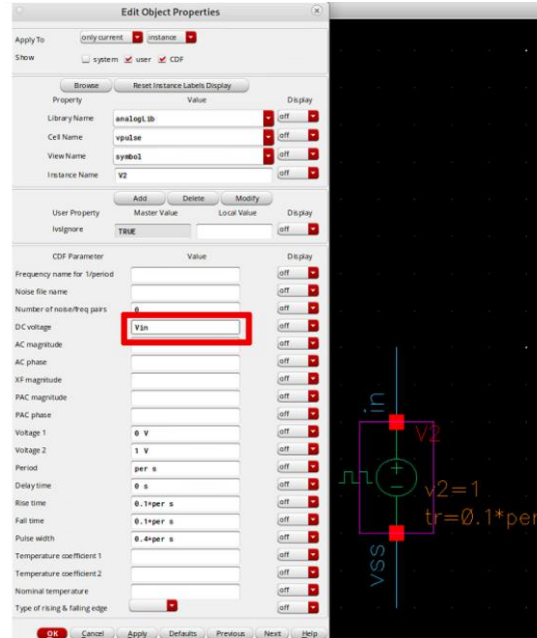

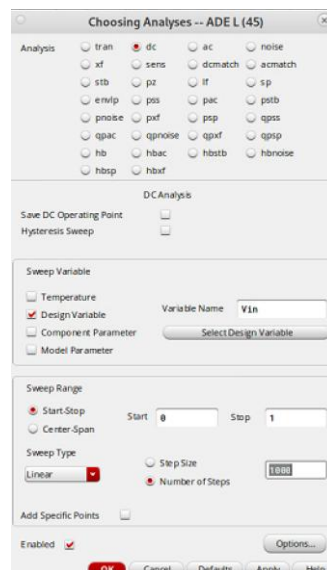**Fig. 37.** Setting up the input source for a DC sweep

Fig. 38. DC sweep analysis

After running the simulation, use direct plot and plot the output. Fig. 39 shows the transfer characteristics of the designed inverter which matches our expectations. Note that we can still use the calculator to perform special functions such as finding the derivative of the transfer characteristics in Fig. 39. The calculator also has an option label "op" where it allows us to find the operating point of a component and add it to ADE output. Then you can perform a parametric sweep to plot a certain device parameter versus input voltage. This is left as an exercise. Note that in this case you don't need a dc sweep rather a parametric sweep (DC sweep only gives the op value for the final swept input value). For example, Fig. 40 shows the simulated current of the NMOS transistor versus the input voltage.
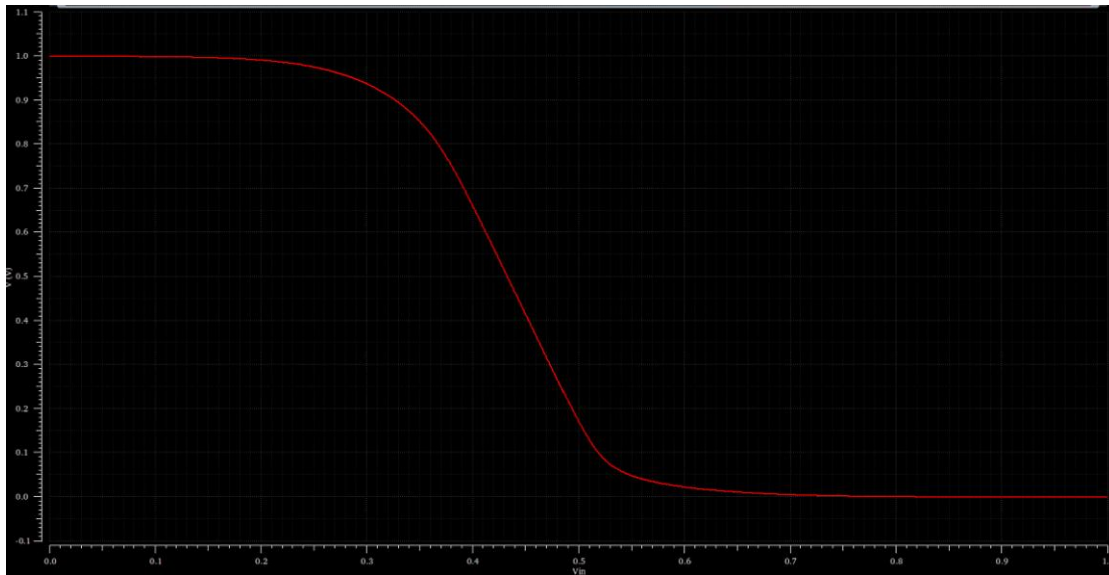

Fig. 39. Obtained inverter characteristics of the inverter using DC sweep.
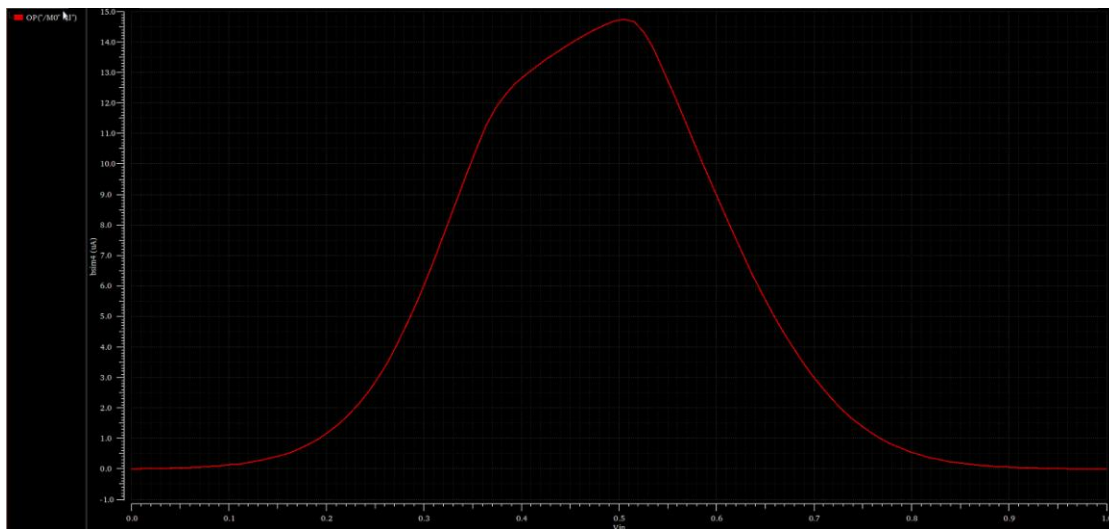

Fig. 40. Simulated current of the NMOS transistor versus the input voltage using the OP option in the calculator and a parametric sweep

5. Designing and Simulating a Common-Source Amplifier

For simulating analog circuits, we typically follow the procedure listed below:

- Perform a DC simulation to make sure that the transistors are in their appropriate region and that the DC node voltages have a proper value
- Perform an AC simulation to find the gain and phase response of the circuit (Transfer function)
- Perform transient simulations and dft (outside the scope of this tutorial) to see the large-signal behavior and settling time of the circuit, as well as the nonlinearity distortion
- Might need to perform a noise simulation to see the total integrated output noise

As an example, lets design and simulate a Common-source amplifier shown in Fig. 41 using the procedure outlined above and validate our understanding of simulating analog circuits in Cadence.
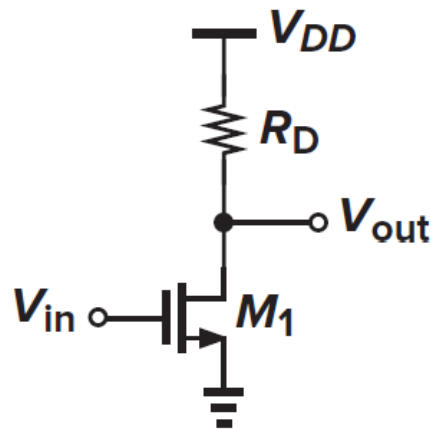

Fig. 41. Common-source amplifier architecture