Schematic Design and Simulation in Cadence ICFB6

Last updated: 1/22/2018 by Sam Lightbody
Spoorthi Nayak, Sudip Shekhar
Originally prepared by Ahmadreza Farsei in 2014 for ICFB5
ELEC 404/571F RFIC Design Course, University of British Columbia

In this tutorial you will learn how to run Cadence and how to draw a simple schematic using basic components from analogLib, and then run ac simulations.

1. Connection to ECE servers

I would recommend running the Cadence simulation from the undergraduate lab (MCLD 348 or 358) on a wired (Ethernet) connection if you find the simulations/GUI running slow. Or if you have access to faster computers in your own research lab (EECE), you may use those computers too.

Otherwise, you can use X2go or MobaXterm on Windows/Mac to run Cadence simulations on a server. X2go usually runs faster than MobaXterm.

Note: in the following instructions, userID/username refers to your EECE user account id (not your CWL).

X2go [Windows]:

Please see instructions here: http://sudip.ece.ubc.ca/cad-access/

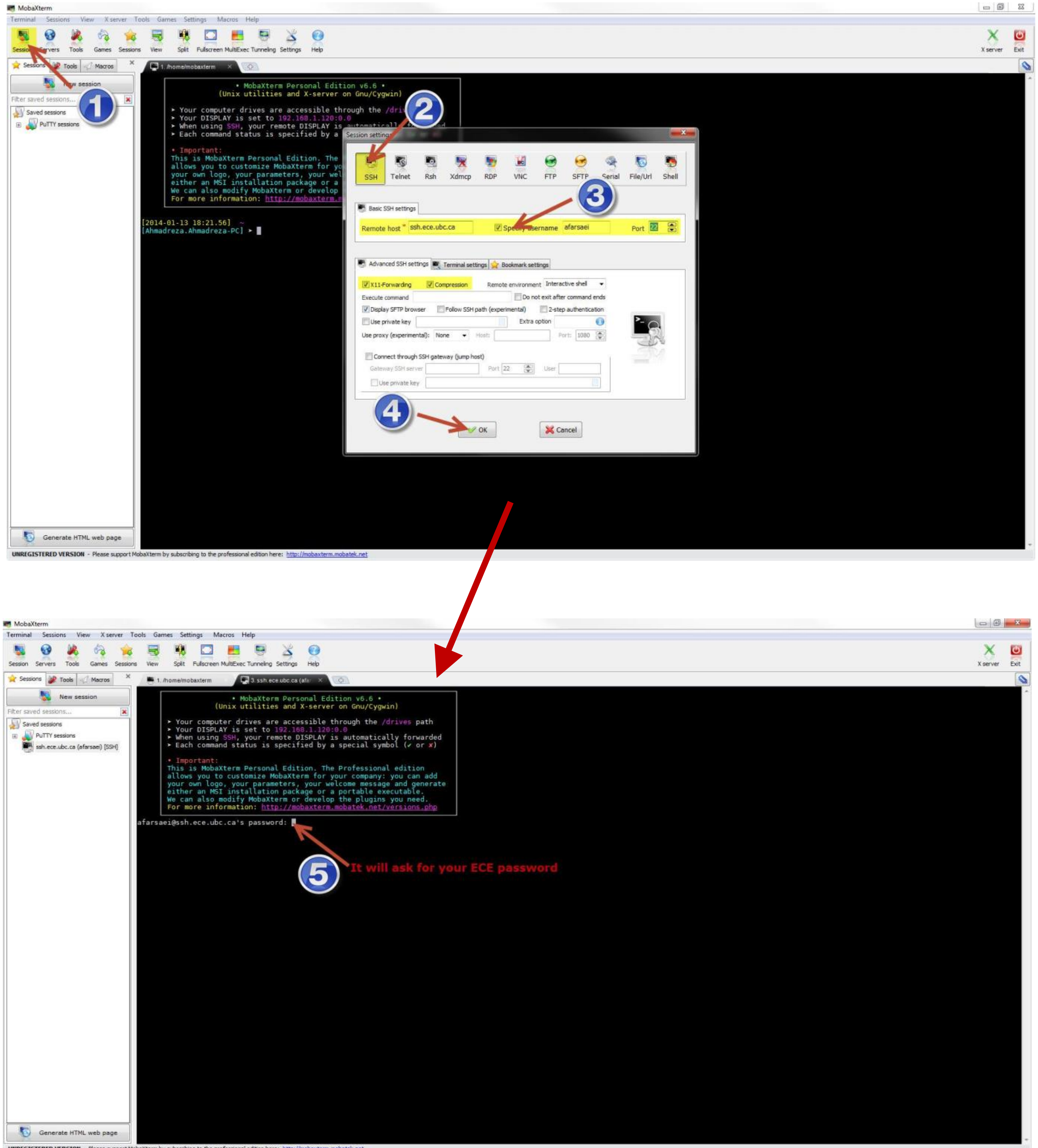
MobaXterm [Windows]

Download from http://mobaxterm.mobatek.net/. There is a tutorial on the main page of their website. After installing the software use the steps shown in the following picture to setup your *ssh connection*. For the rest of this tutorial pay special attention to the highlight areas in the pictures.

You can login to **ssh.ece.ubc.ca** to run your simulations. If connecting via UBC wireless services or from home, on MobaXterm, you can start a new session:

→ SSH

→Remote host: ssh.ece.ubc.ca "specify username:" <your userID>



Alternatively, you can use our dedicated compute server, **ug-compute1.ece.ubc.ca**, after going through the ssh.ece.ubc.ca gateway. To start such a new session:

→ SSH

→Remote host: ug-compute1 "specify username:" <your userID>

Connect through SSH gateway: ssh.ece.ubc.ca User: <your userID>

Linux (Ubuntu, Red hat, Mac)
You can easily run Cadence without any extra GUI interface, you just open your terminal and type the following command to connect to *ssh machines* in ECE.
> **ssh -X <userID>@ssh.ece.ubc.ca** # Note -X enables X11 forwarding and -x disables it.

To jump directly to ug-compute1:

> **ssh -X -tt <userID>@ssh.ece.ubc.ca ssh -X -tt <userID>@ug-compute1** # It will ask you for password twice.

Mac Users: you need a version of X window system for your Mac, one of your options is *XQuartz* that you can find http://xquartz.macosforge.org/landing/.

Connecting to Different Machines:
If you are a graduate student and have your own computer in your research Lab, please *ssh* to your own computer using the above method to reduce the overload on the ECE machines. In order words, after connecting to ECE machines using one of the above method, use the following command to connect to your ssh machine. (Thanks)

Command Prompt >> ssh -X [yourusername]@[your computer name]

2. Useful Linux commands to know for this course, known problems and solutions

First of all, the machine ug-compute1 was deployed to allow students in ELEC404/571F to do their work, so please keep it private.

<u>**"no more process"**, or **"cannot fork"** or your cadence is crashing :</u> The number of processes per user is limited to **100**. If you are receiving a message "no more process", or "cannot fork" or your cadence is crashing etc., you might have reached your limit.
You can use the following command to check for the number of your processes:

> ps aux |grep <your userID> | wc -l # this gives a count of the number of outputs from the PS command that are associated with your userID
e.g.
ssh-linux5:~> ps aux|grep q4t7|wc –l
10 # this user could start 90 more processes

> ps aux|grep <your userID>

> ps aux | grep f0e7 # for example

```
root   24071  0.0  0.0  70464  3220 ?   Ss   16:48   0:00  sshd: f0e7 [priv]
f0e7   24074  0.0  0.0  70464  1648 ?   S    16:48   0:00  sshd: f0e7@notty
f0e7   24075  0.0  0.0  12636  1300 ?   Ss   16:48   0:00  tcsh -c /usr/lib/openssh/sftp-server
f0e7   24084  0.0  0.0  3904   748  ?   S    16:48   0:00  /usr/lib/openssh/sftp-server
f0e7   31525  0.0  0.0  11880  428  ?   Ss   Feb13   0:00  ssh-agent
```

For example, the above jobs are created when you connect via Mobaxterm.
In case the number of processes you are running is close to 100, you will have to close one or more or all applications so you can run any new commands.

To **kill** a specific process use the command:

> kill <PID> # for example 24075 will kill the sftp-server which you only need when you are transferring files.

To kill all your processes (which also logs you out) and start again:

> pkill -u <userID> # you can only kill your processes, for other IDs you will get a permission denied message

File is locked. Cannot edit. Open for read only?
Or upon starting Cadence, it takes a long time to startup, and then gives the warning:
***WARNING* file /home/ugrad/(yourusernamehere)/CDS.log Connection refused**
Or it takes a long time to open a schematic:
If you Cadence session crashed previously while a schematic was being edited, it left a lock file before the crash. [Cadence automatically creates this temporary lock file to prevent another user from opening up the same file for edit.] You will need to delete this lock file before editing the schematic again. Look for the file with an extension .cdslck and delete it.

3. Launching Cadence

You need to make a directory where you can create your projects. Use the following commands to create the folder and change your current directory to the created folder.

- ✓ [Creating the Folder] Command Prompt >> **mkdir RFCourse2018**
- ✓ [Change Current Directory] Command Prompt >> **cd RFCourse2018**

Now you are ready to run Cadence using the following commands.
[Both undergraduate and graduate students should run Cadence using these commands for now (for homework assignment #1). **Please do not share, distribute, upload any process information or design files outside the classroom because of strict confidentiality clauses.]**

- ✓ [Setting Up Environment for Cadence]
  Command Prompt >> source **/CMC/kits/AMSKIT616_GPDK/underg_install.csh**

You only need to run the above command once to set up the directory. After running the above command you will see a bunch of messages in your X2go/MobaXterm Terminal window. It will create some folders and generate a file setup_local.csh. Everytime you want to run Cadence source this file in the **RFCourse2018** directory by typing:

Command Prompt >> source **setup_local.csh**

Then, to open Cadence, descend into the cds folder by typing:

Command Prompt >> cd **cds**

and finally:

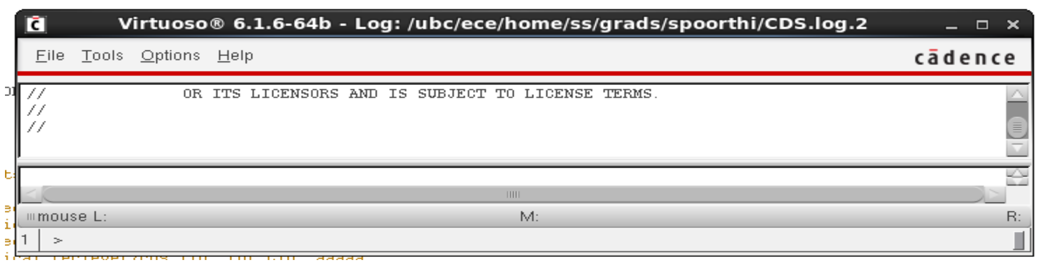
- ✓ **virtuoso -log virtuoso.log &** # The & will run the command in the background. So you can use your shell to give more commands.

(To source Cadence using TSMC 65nm libraries, if you have access, use the command:

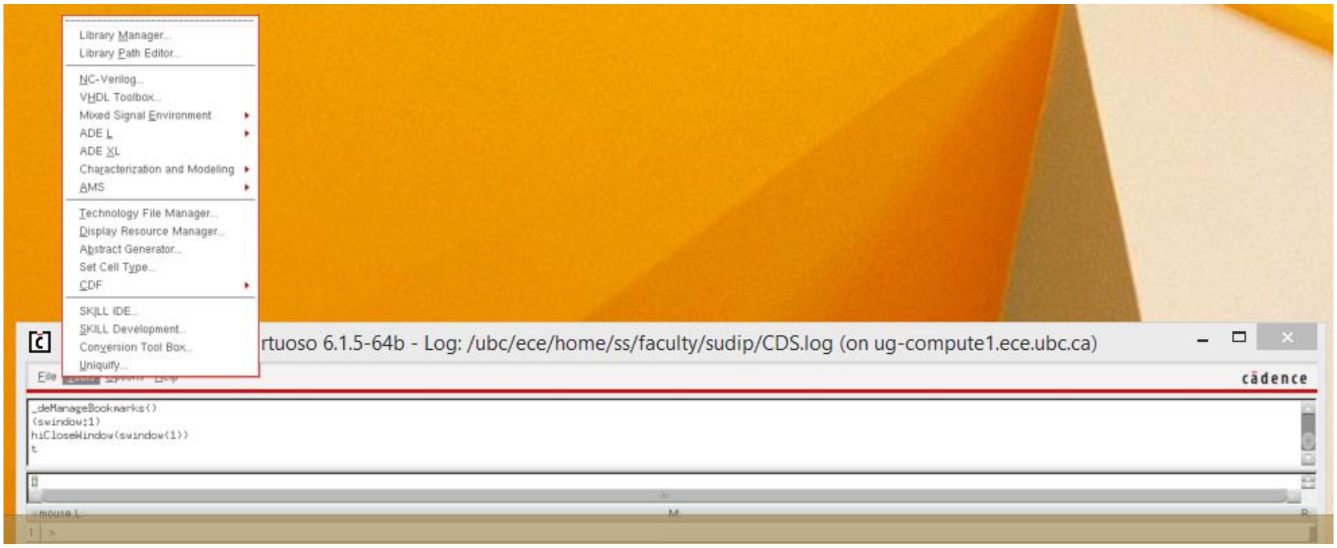
Command Prompt >> source **/CMC/kits/tsmc_65nm/CRN65GP/PDKOA/kit.tsmc65nm_gp_OA.2.0.csh**

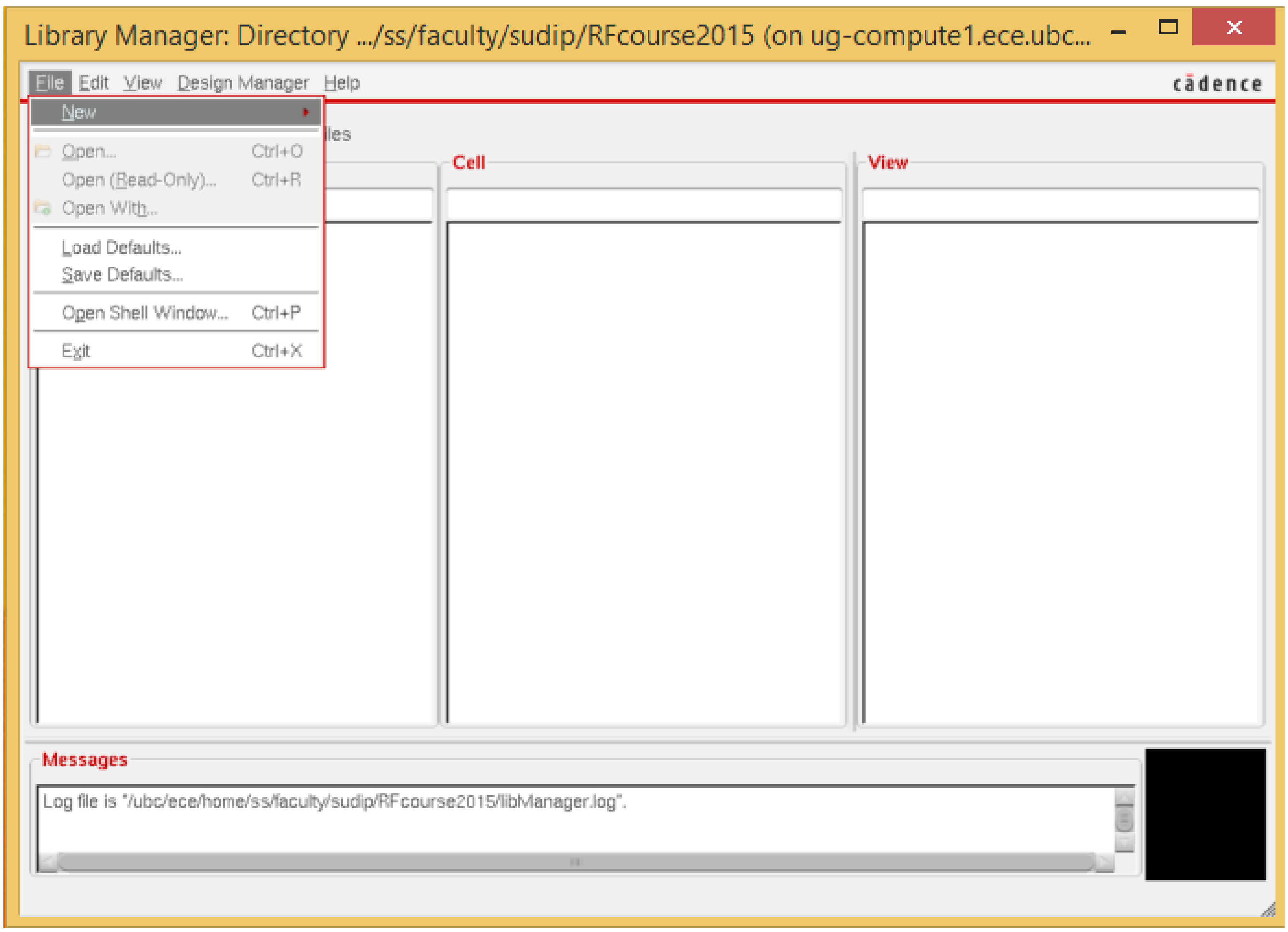
and then run the **virtuoso** command as normal)

You should be able to see a GUI window pop up now; this windows is called CIW (Command Interpreter Window). If there is any error or warning message, this is the first place to take a look.



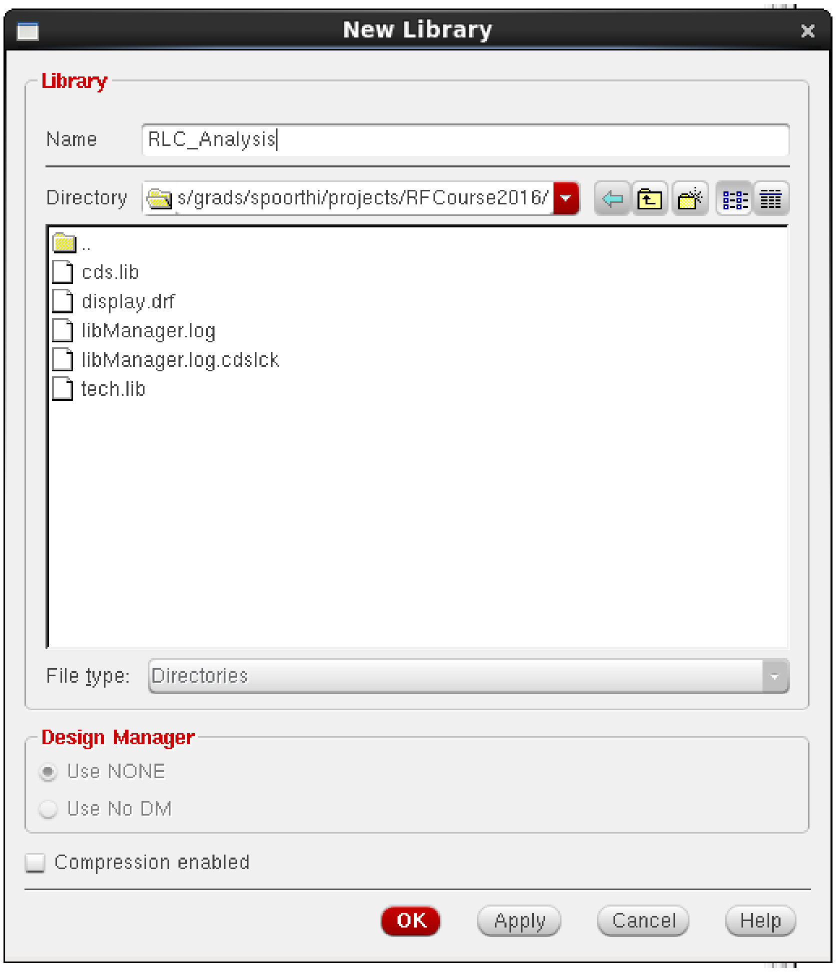
In Cadence you have to first create a library (think of it as a container which includes different designs for your project). For this, start the Library Manager (*Tools → Library Manager*), then in the Library Manager window, *File → New →Library*:

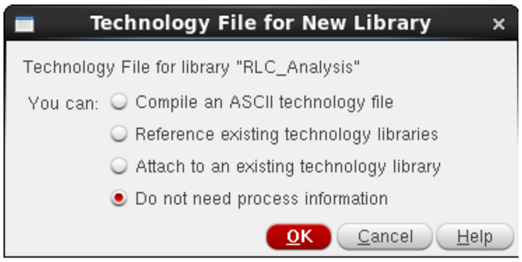




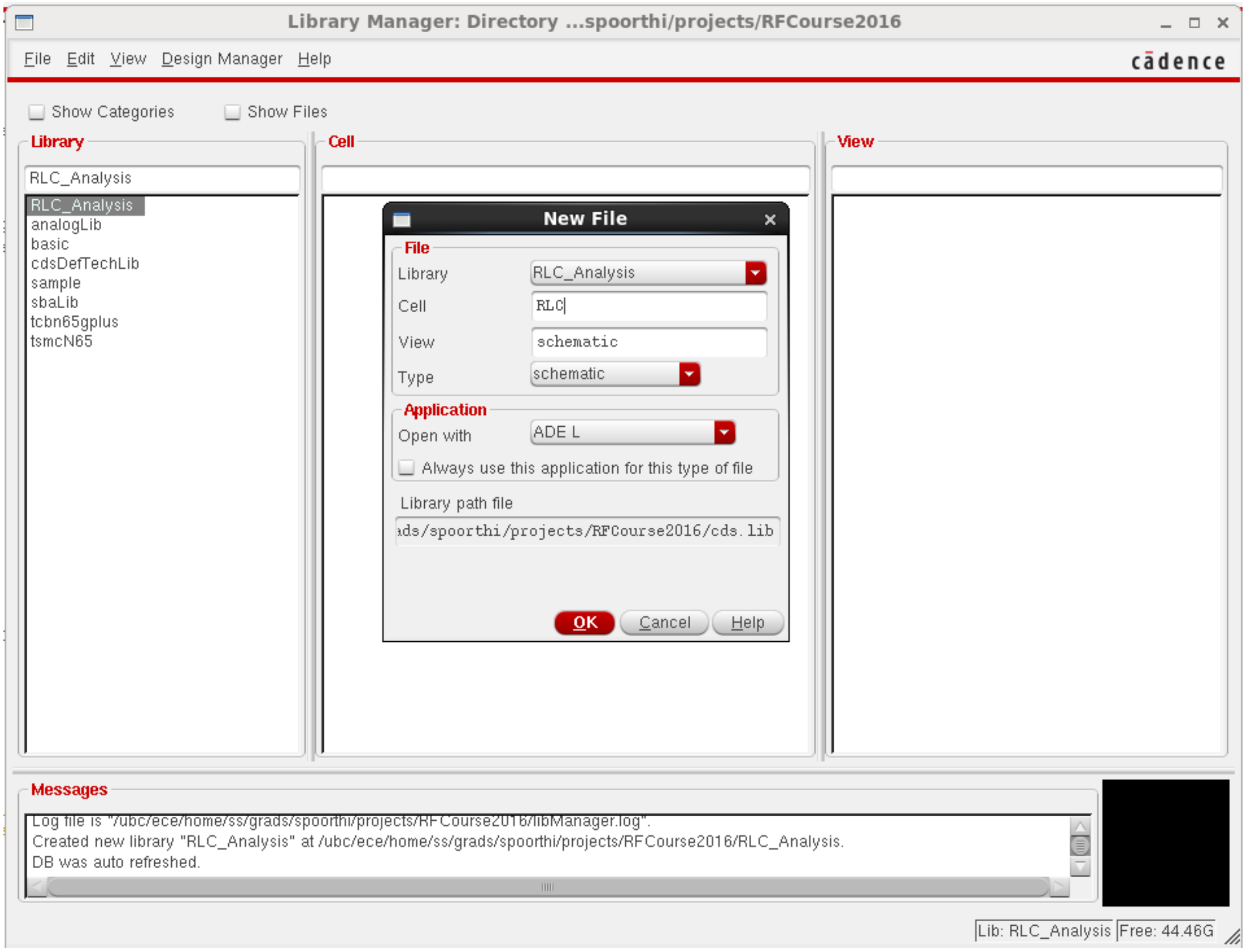
Specify the Name of the Library (e.g. it could be *RLC_Analysis*). When prompted, select *"Do not need process information".*

[Later in the course, students may choose *"Attach to an existing technology library"* – which specifies that the library is going to use a specific technology, say, 45nm GPDK technology, **gpdk045** , or, 65nm TSMC Technology, tsmcN65.]

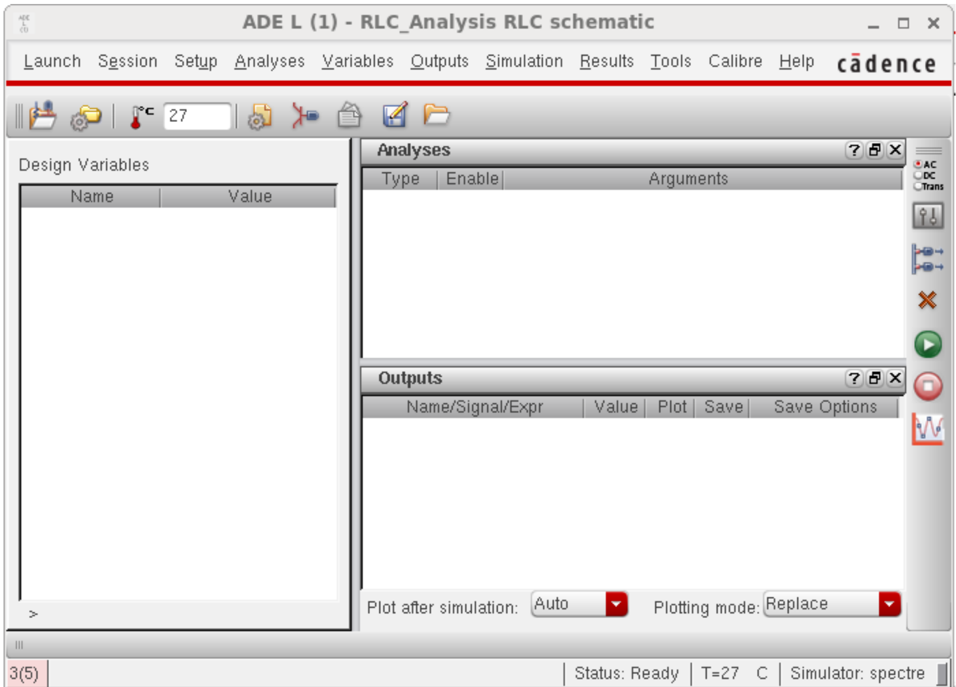


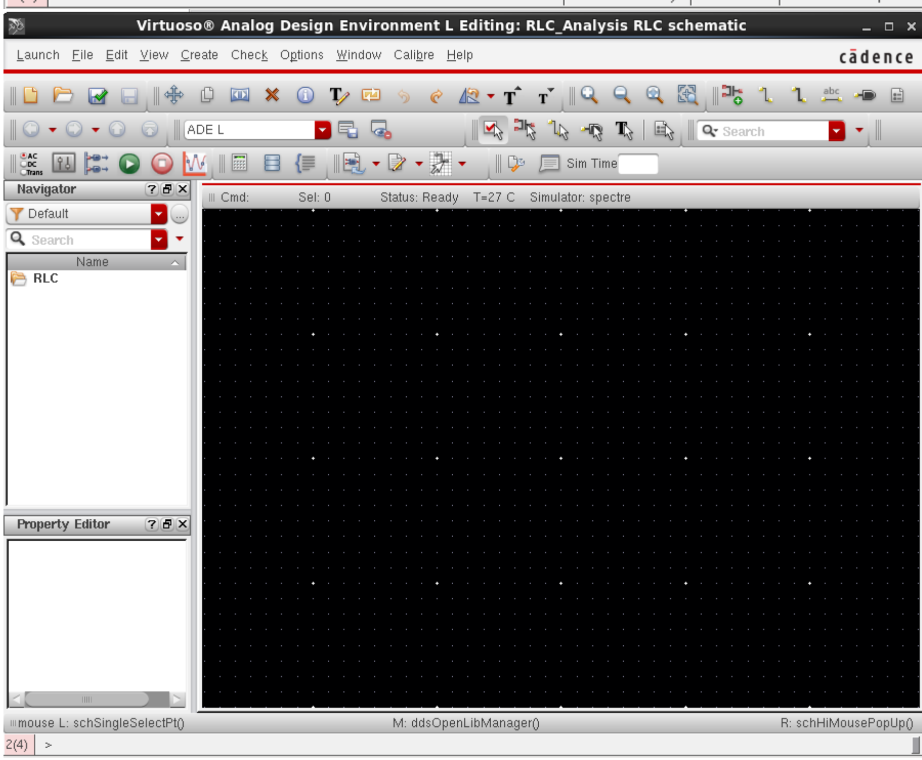


After Creating Library, you have to create a Cellview; Cellview contains your designs. Click on "Assignments" in the Library Manager, then go to *File* → *New* → *Cellview*.
In this window, type the Cell Name(e.g. RLC), and View Name (e.g. schematic) that specifies which view of the cell you are going to work on (- for now we are working on the Schematic). Ensure that the Tool automatically chooses schematic as "Type". For the application, choose *Open with* ADE L. Check "Always use this application for this type of file" if you want. [Sometimes, the software may upgrade you to ADE XL for license reasons].



Below is the GUI of the Cadence Virtuoso Schematic Editor (bottom) as well as the Analog Design Environment (ADE) Editor (top). Let's focus on the Schematic Editor first – this is the place where you are going to draw your schematic.

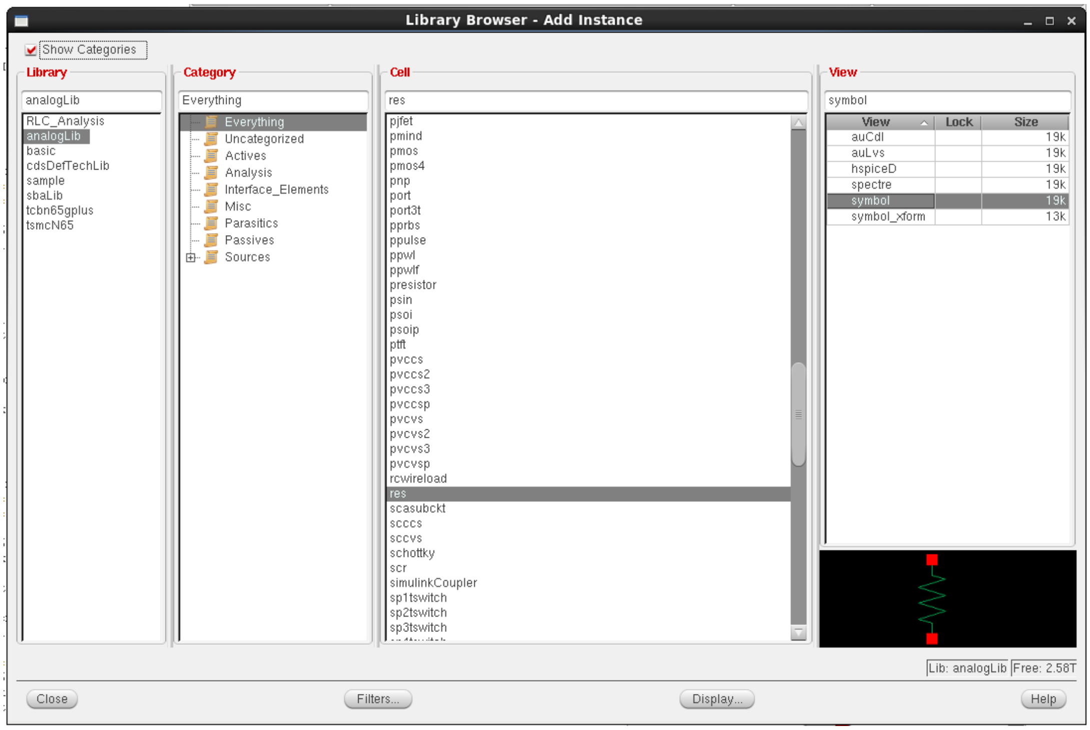




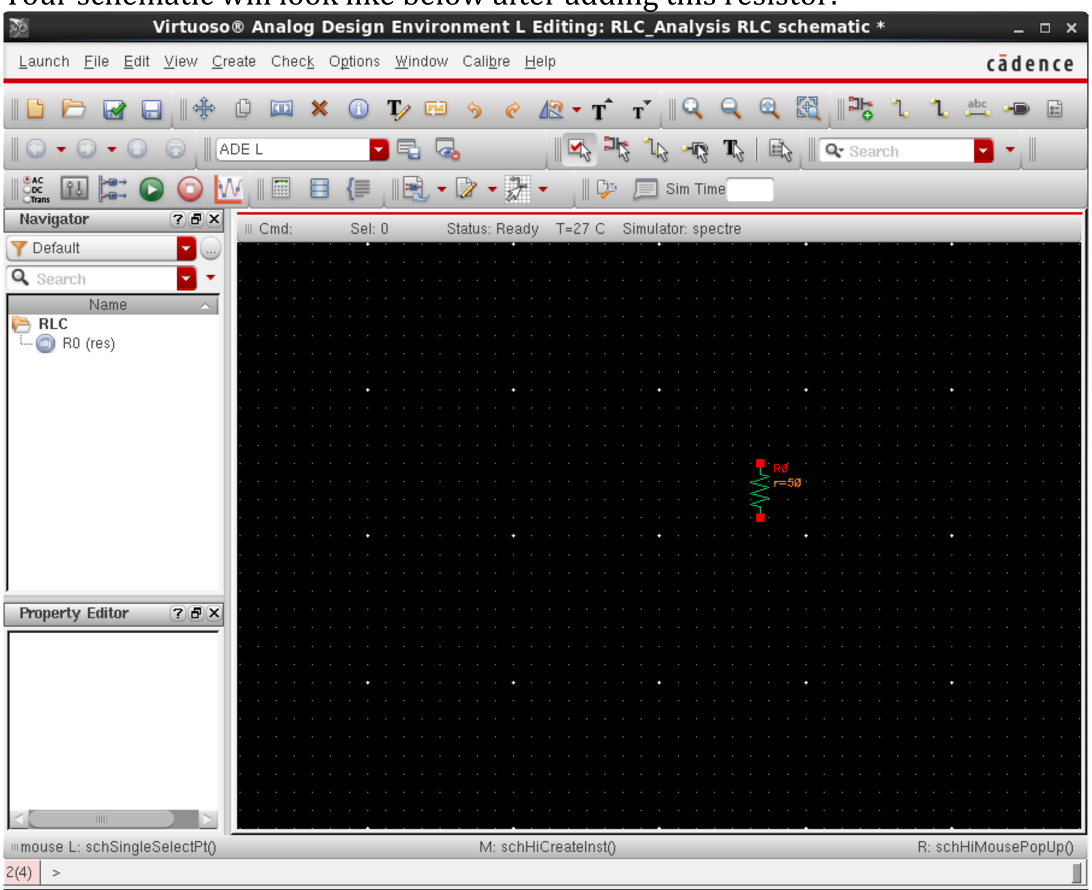
To draw schematics you need components, which you can add to your schematic as follow: Create → Instance. **Or Just use the shortcut "i".**

Let us add a resistance (res) from analogLib. Also, we will change the resistance to 50 Ohms from default of 1K Ohms.

Most of the components that you will be using in this course will be from analogLib library (some from "basic" library). If you don't remember the name of the Library, Cell and View, you can click on Browse button and then find your component using Library Browser, which is shown below:

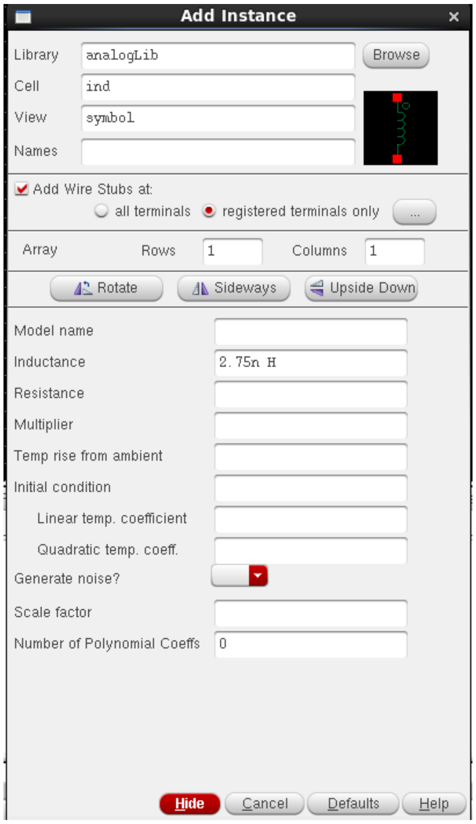


Your schematic will look like below after adding this resistor:

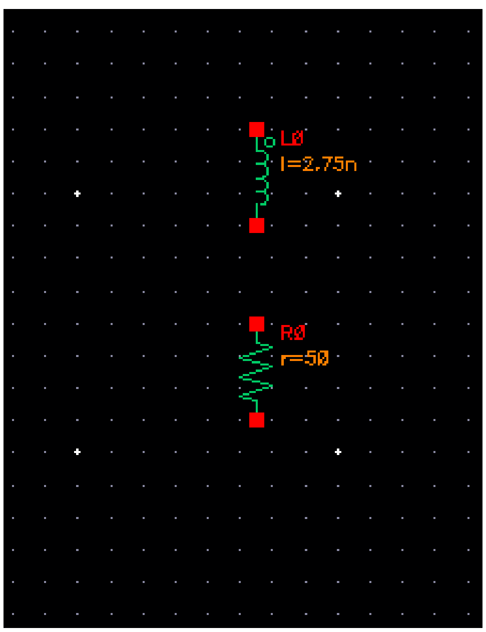


Use the keys "[" and "]" to zoom in and out.

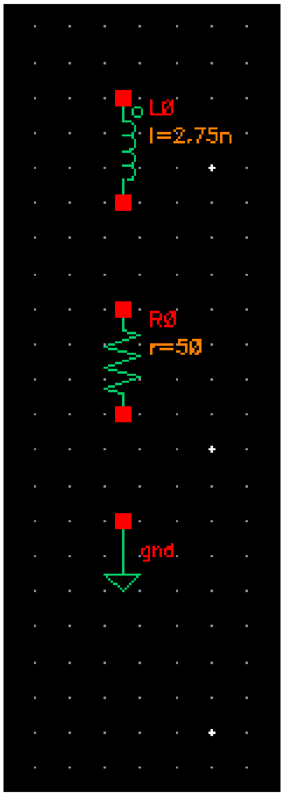
For the RLC circuit, we also need an inductor, which can be added to your design either using Library Browser or by typing the Library Name (analogLib), Cell Name (ind) and View Name (symbol) in the corresponding boxes directly after using "i". [Cell name for a capacitor is *cap*.] Choose 2.75nH for the inductance in our RLC circuit example.



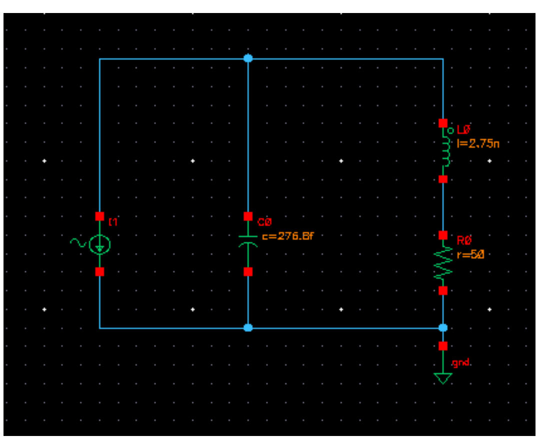
Schematic after adding the inductor:



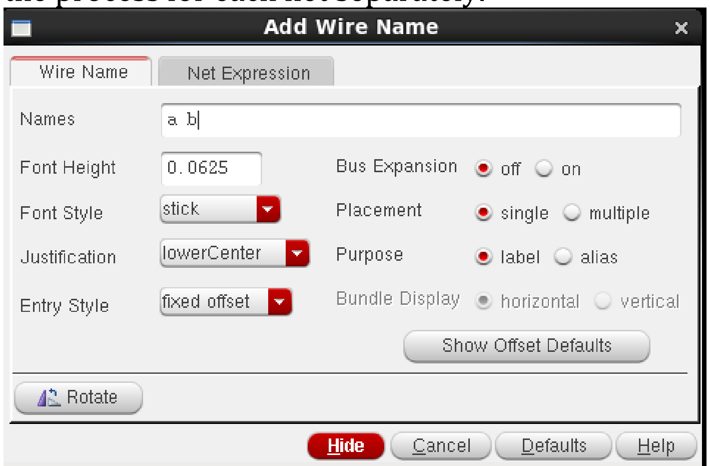
Every electronic circuit needs a ground which will be considered as the reference point, 0 volt, for the potential. Schematic after adding the ground symbol (gnd) from analogLib:



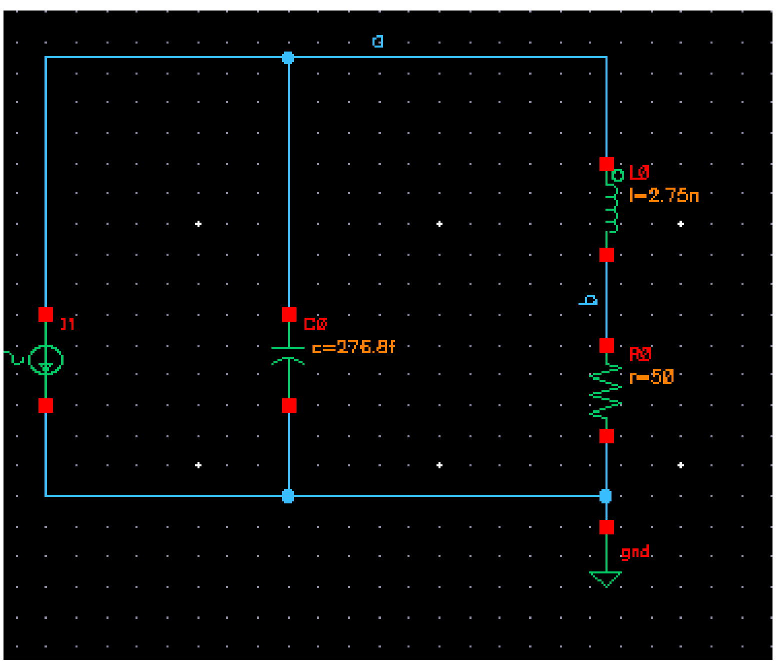
Next, add a capacitor (cap) from analogLib with a value of 276.8fF.
Finally, you need to apply an ac source (voltage “vsin” or current “isin”) to find the ac impedance of this circuit. As an illustration, we will use the current source component called isin with an AC magnitude of 1 A for simulation. Now you have to connect symbols using wire as follow **[use the shortcut “w”]**:



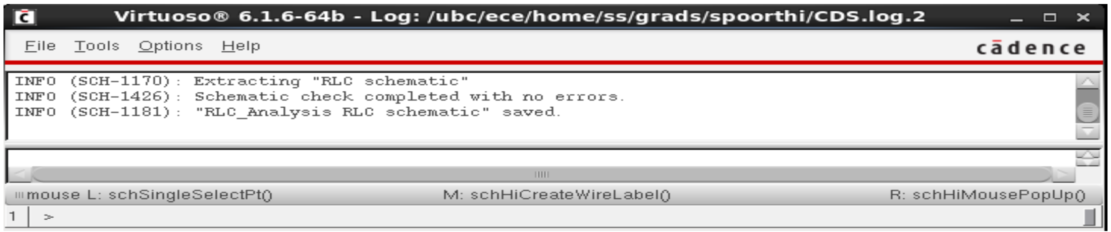
It is always good to label the nets that are important for your design because you can find them easily after the simulation in the results. We do the labeling as follow (*Create* → *Wire Name*... or **shortcut "l"**):
Below we choose to add multiple wire labels in the GUI, and then click on the respective nets on the schematic in the proper sequence to label them. It saves some time compared to repeating the process for each net separately.



Final schematic after adding the Labels is as follow.
Don't forget to check and save your design before any kind of simulation. Click on the floppy drive with check mark symbol on the top left corner, or just use the **shortcut "shift+x".**



Always look for warnings and errors in the CIW window and fix them. You *may* be able to ignore the warnings. In our example, we see no errors or warnings.

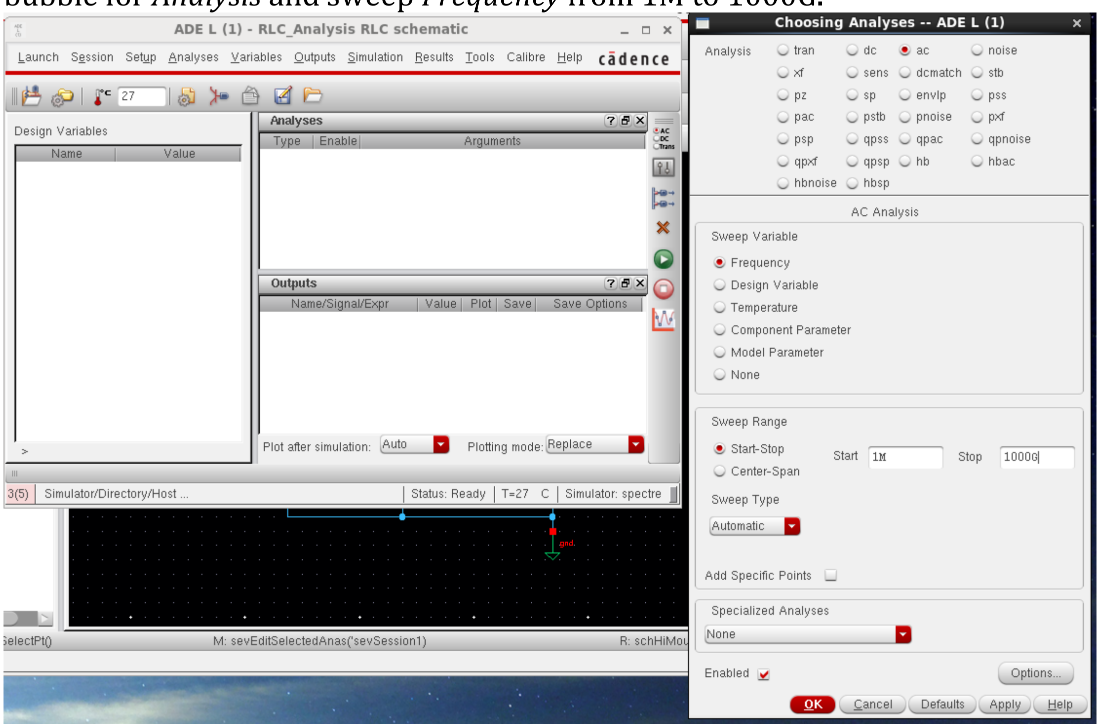


After clicking the *check and save* (shift+x) button and making sure that there is no errors and warning, we are ready to run the simulation using Analog Design Environment

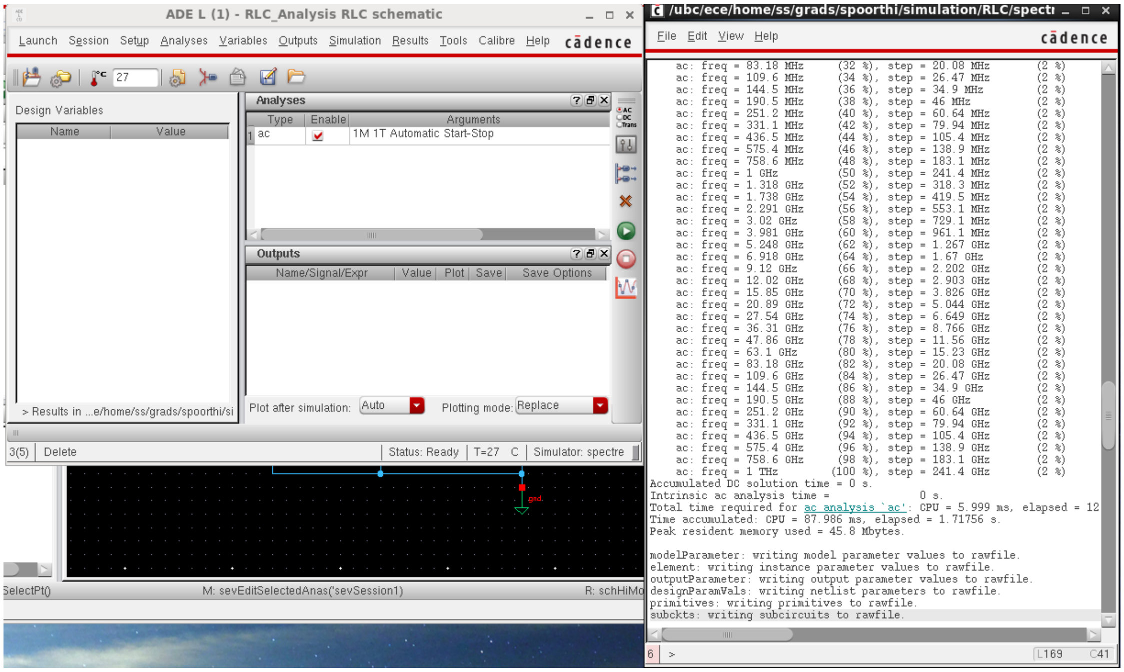
---

4. Running Simulation

We want to run a frequency response to obtain the input impedance from 1MHz to 1000GHz. In order for this, in the Virtuoso Analog Design Environment, Choose *Analyses* → then click on *ac* bubble for *Analysis* and sweep *Frequency* from 1M to 1000G.



After clicking Ok, your ADE window will look like below. Click on the green arrow to run simulation. If you get an error, go back and ensure that you “check and save”d the schematic! If everything is okay, the simulation should be successfully completed, and you will see the success notification on the CIW as well as “spectre.out” window.



---

5. Plotting Results

Go to Results → Direct Plot → AC Magnitude in the ADE simulation window for choosing ac simulation. Next, go to your schematic editor to select the node "a" and hit <Escape> on your keyboard. You will then obtain a plot for the input impedance [Why?] of the RLC network.