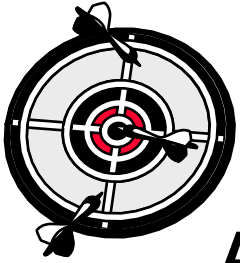# Introduction to Cadence

Dr Basel Halak
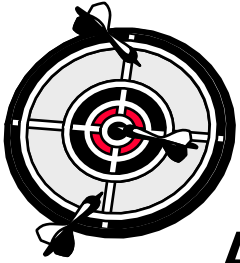
Learning Outcomes

After completing this unit, you should be able to:

1. **Set up a directory to run Cadence**

2. **Install the design files for the AMS 0.35um PDK**

3. **Run the Cadence Design Manager (ICFB)**

4. **Create a Cadence Project**

5. **Create a simple Cadence Cell View (Schematic)**

6. **Link to an external library in Cadence**

7. **Use the external blocks in your design**

Learning Outcomes

After completing this unit, you should be able to:

1. **Set up a directory to run Cadence**

2. **Install the design files for the AMS 0.35um PDK**

3. **Run the Cadence Design Manager (ICFB)**

4. **Create a Cadence Project**

5. **Create a simple Cadence Cell View (Schematic)**

6. **Link to an external library in Cadence**

7. **Use the external blocks in your design**

Introduction

- **The first thing we need to do is to create a directory that has the correct setup and config files to run:**
  - A: Cadence
  - B: the AMS PDK (Process Design Kit)

- **Log onto a Unix Machine (hind) using NX Client or Exceed**
  - See instructions on:
  - http://users.ecs.soton.ac.uk/bim/notes/cad/guides/begin.html

- **Create a new directory in your home directory**
  - /home/*username*/EDA/lab1
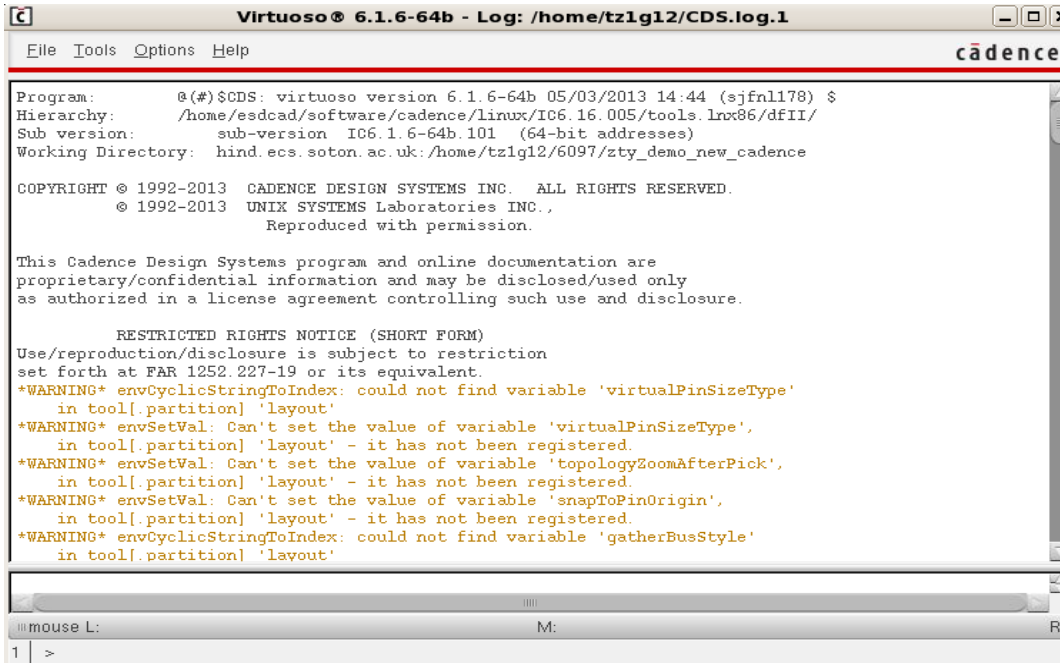
Set up the current working directory

- **The first step is to set up the local working directory for the AMS 0.35μm process.**

- **open a new terminal and type the following:**

  tcsh

  source /opt/esdcad/scripts/ams_v400_tcshrc

Run Cadence Design Manager

- **This will bring up the Cadence Design Manager:**

Learning Outcomes

After completing this unit, you should be able to:

1. **Set up a directory to run Cadence**

2. **Install the design files for the AMS 0.35um PDK**

3. **Run the Cadence Design Manager (ICFB)**

4. **Create a Cadence Project**

5. **Create a simple Cadence Cell View (Schematic)**

6. **Link to an external library in Cadence**
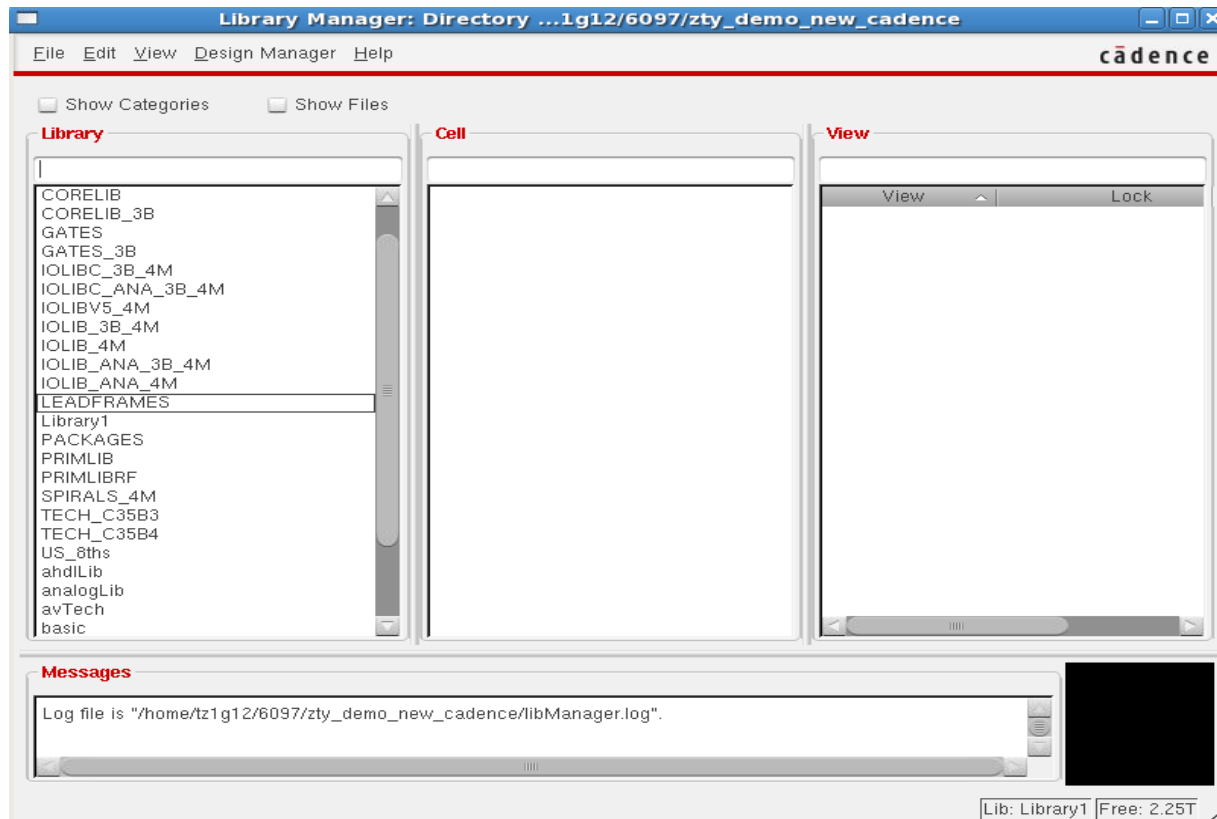
7. **Use the external blocks in your design**

Create a new library

- **Cadence works by creating multiple libraries in separate directories**
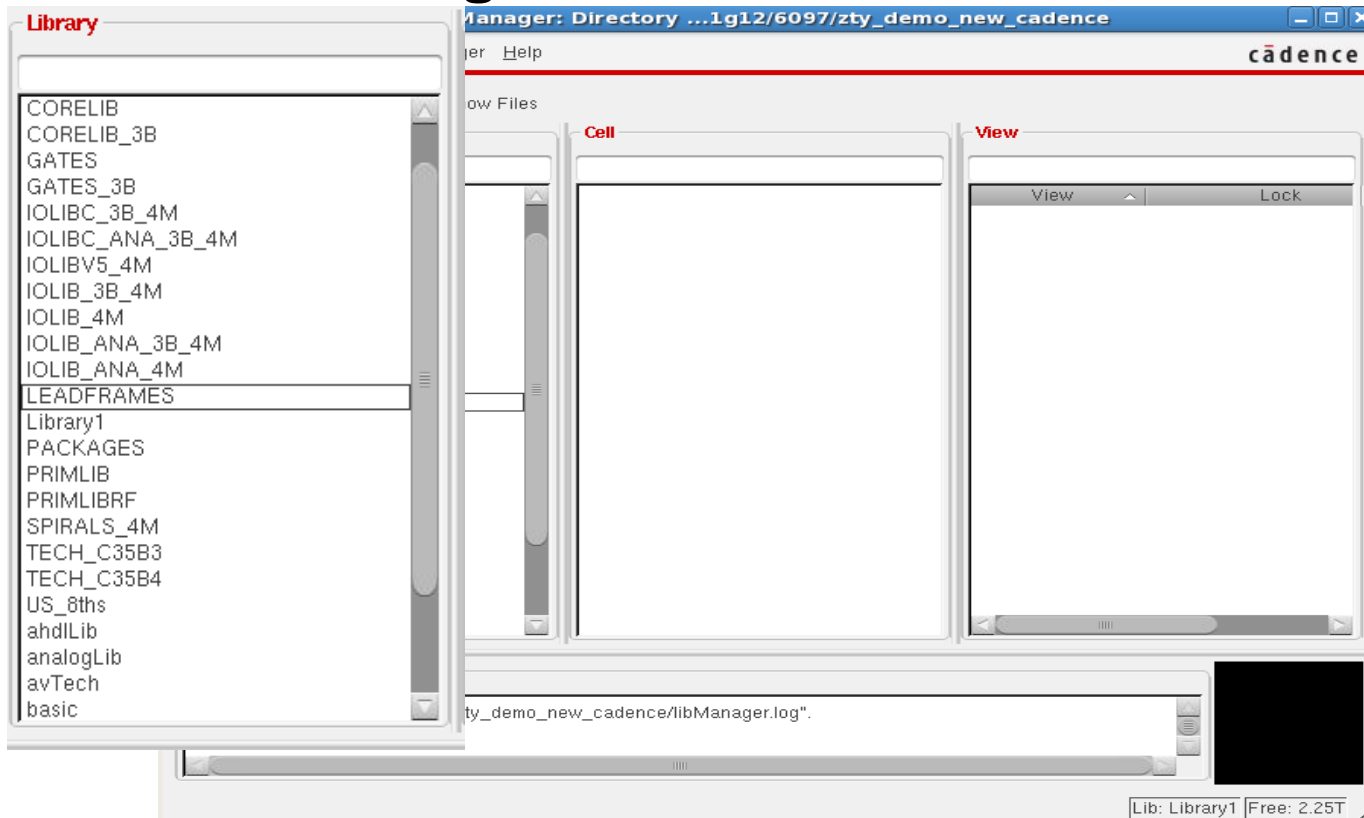
- **Using the Tools Menu – select the library manager**
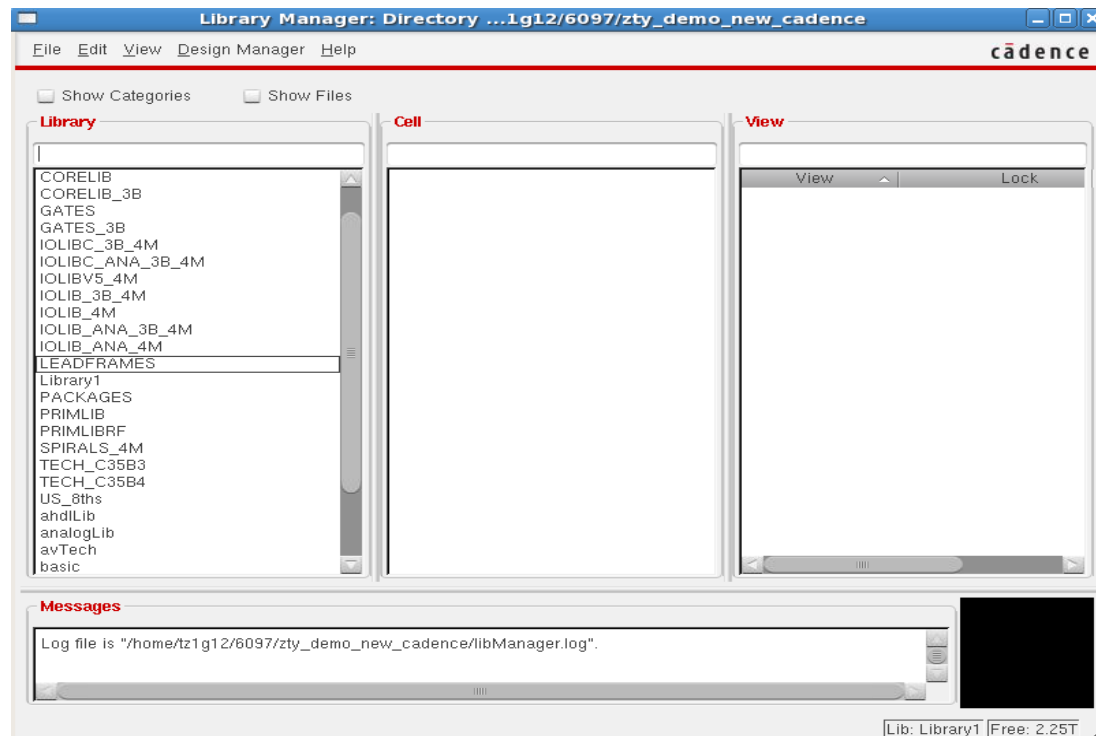
The Library Manager Explained

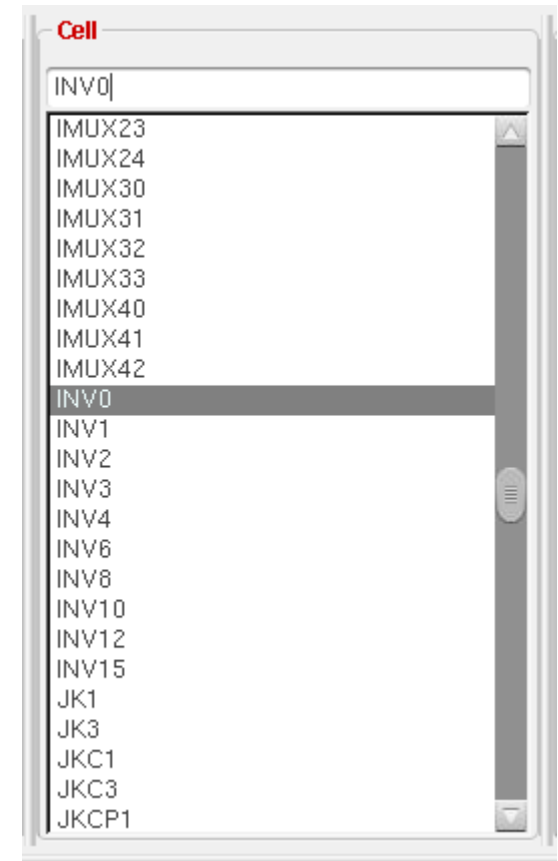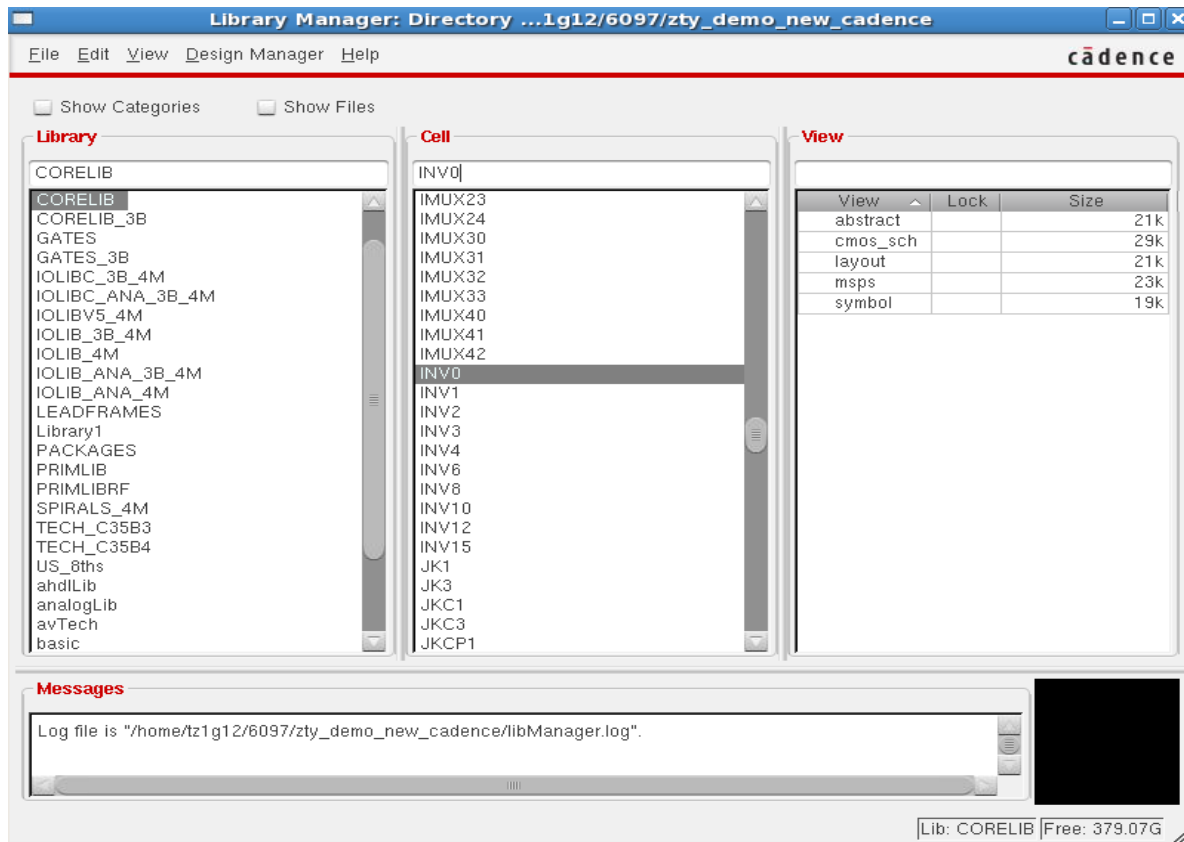- **The Library manager handles "libraries", "cells" and "views"**

The Library Manager Explained

- **Libraries are the containers for all the design elements. For example a sandard cell library may contain all the gates available in the PDK**
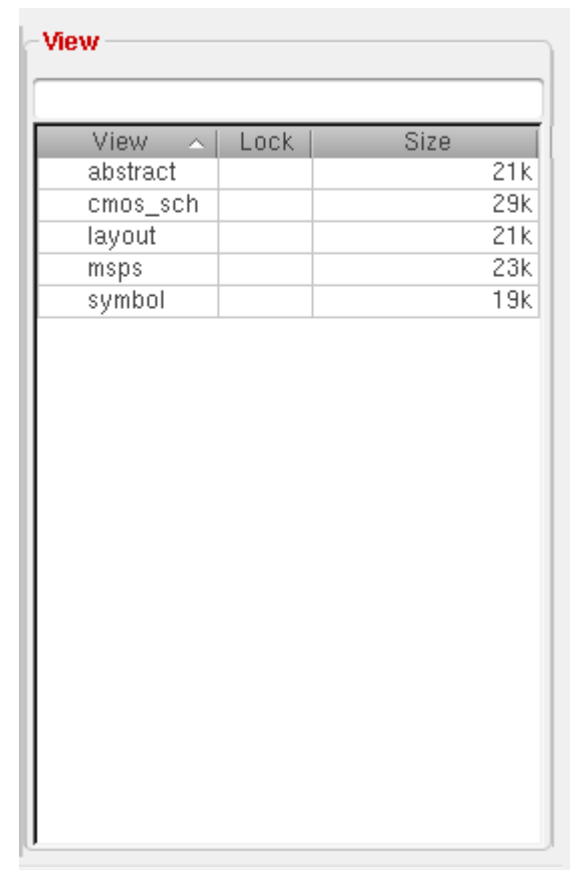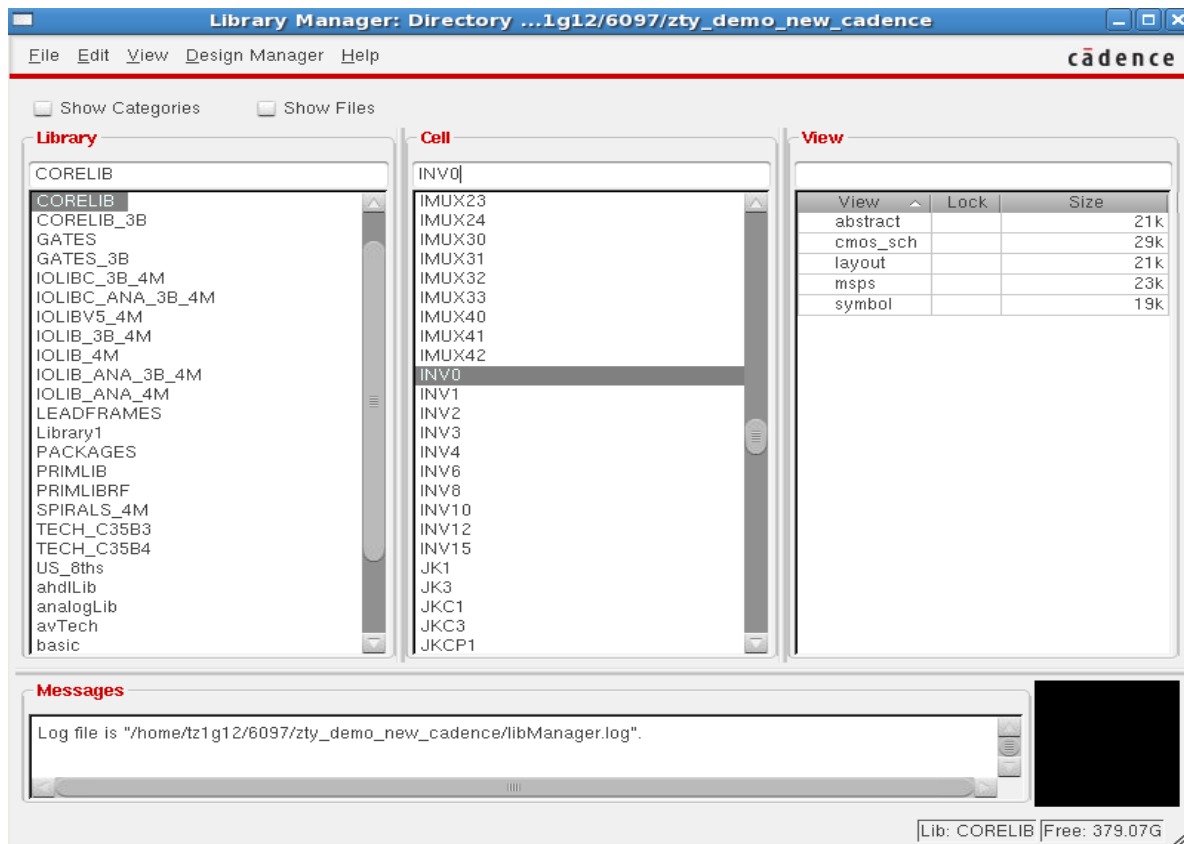
The Library Manager Explained

- **Each element, such as a gate, in the library is called a cell – an example might be an inverter gate.**

The Library Manager Explained

■ **Finally, each cell has a set of different views such as schematic, or layout, or a model**

Libraries in Cadence

- **The library definition in Cadence consists of two parts:**
    - 1. The library NAME – what you see in the library manager
    - 2. The library DIRECTORY – what is actually there in the file system

- **DO NOT just copy the library around, as this does not handle all the links properly.**
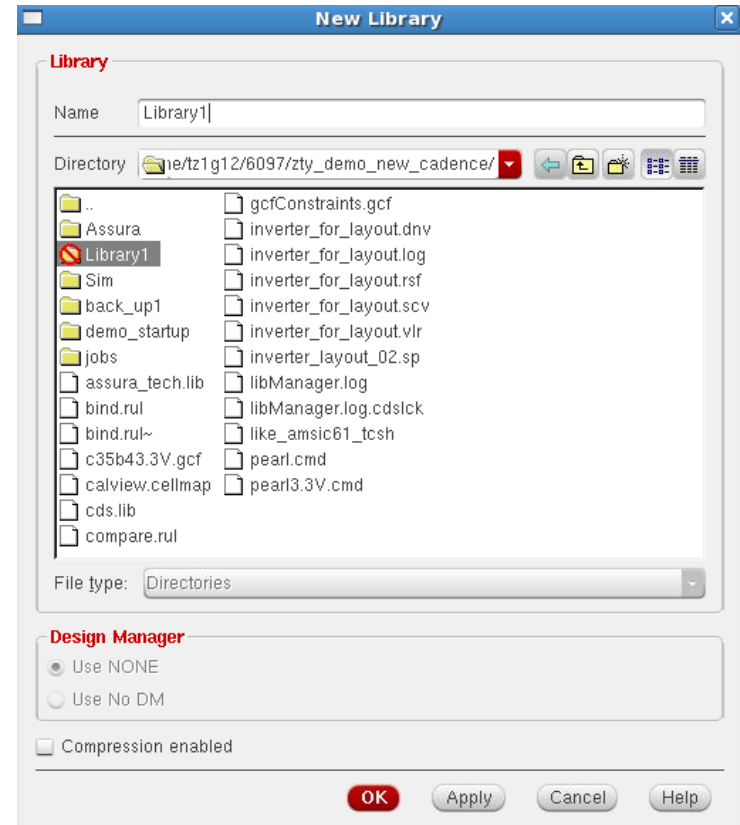
Create your first library

- **To create new libraries, you can use the library manager (LM) to create a new library from scratch**
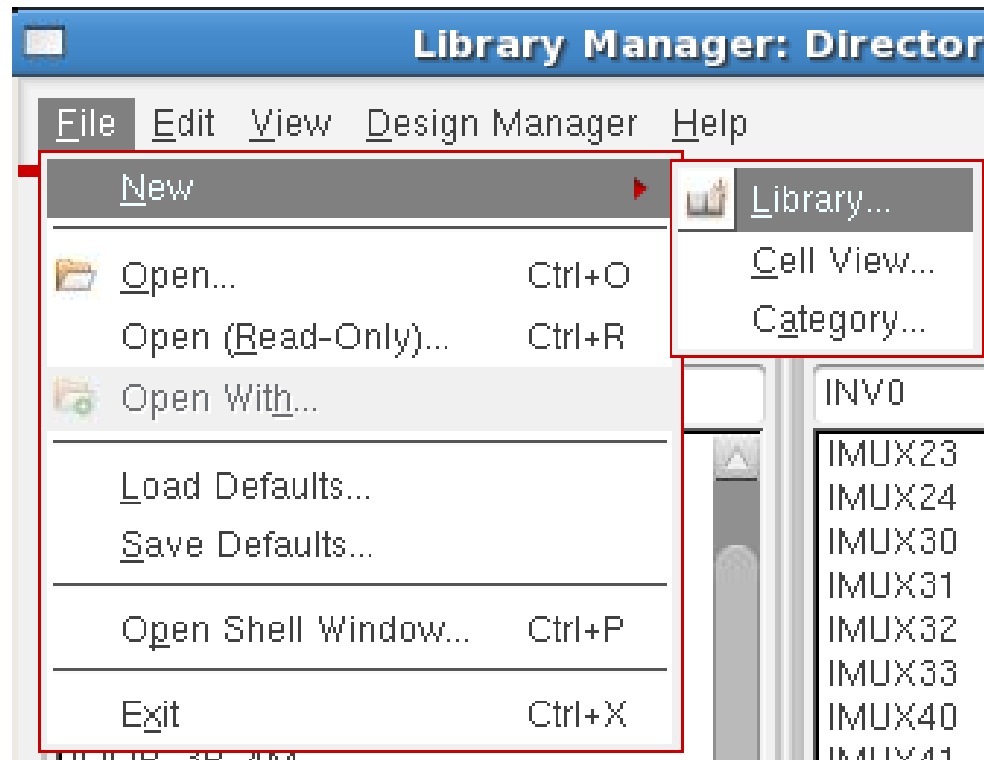
Create your first library

- **The library manager will default to your current directory, and ask you for a library name:**
  - enter *library1* or something similar
  - remember:
    - Follow UNIX rules
    - Don't use spaces
    - Don't start with a number

Creating your first library

- **When you create a library, you will be asked whether you want to attach to a technology file**

- **What does this mean?**

- **When you create a design, you are working with either NO process, or a SPECIFIC process (this implies a certain number of layers, design rules, models etc)**

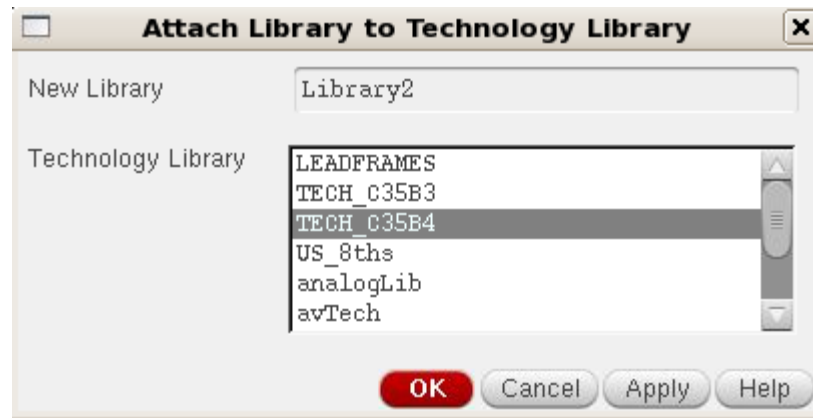- **In this case we are working with the AMS C35B4C3 process.**

Creating your first library

- **Choose "attach to an existing techfile"**
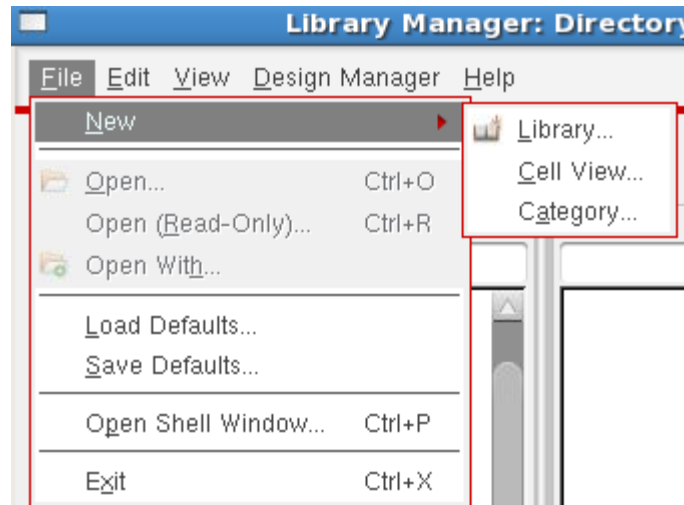

- **And choose TECH_C35B4**

Creating your first library

- **Congratulations!, You will now have your first Cadence Library**

Creating your first cell view

- **Now you have your first library, you can create your first cell view.**

- **Start from the library manager and choose New Cell View.**

Creating a cell view

- **Notice the library defaults to the one you have just created**

- **If you *don't* have this selected, make sure that it is – the standard libraries will be READ ONLY**

- **Notice the View Name – schematic**

- **The Tool pulldown should be Schematics XL**
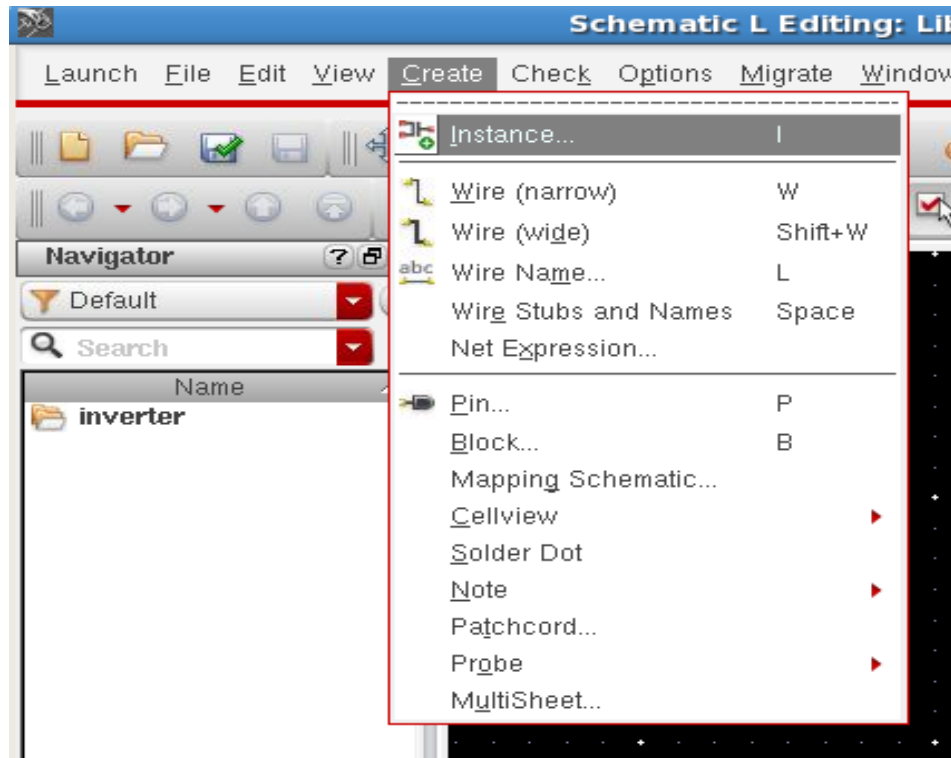
- **Click OK**
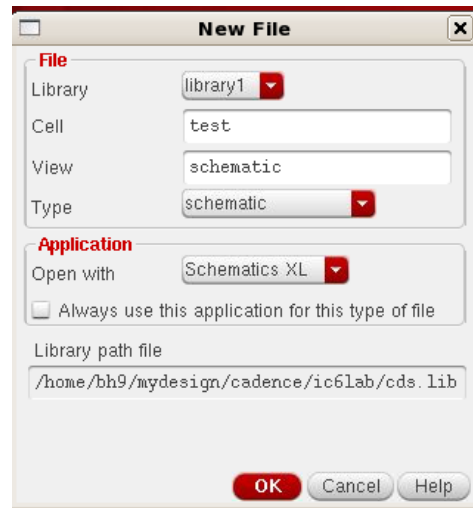
Creating a cell view

■ **When you create a new cell view, the Cadence Design manager will open the correct editing tool – in this case Schematic L (schematic editor) automatically**
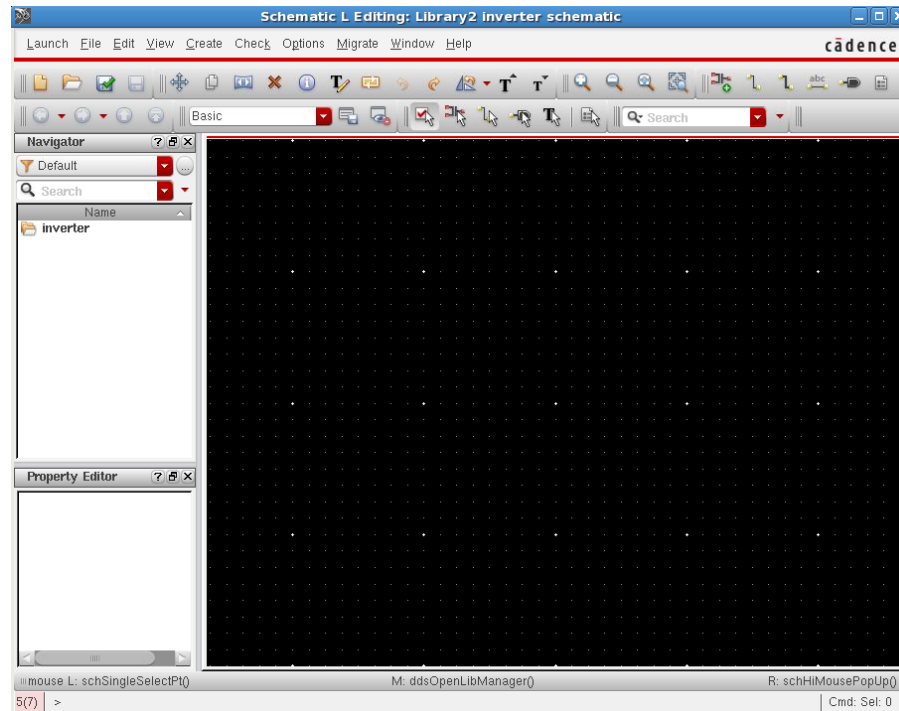
Adding Components

■ **You can now add components to the schematic**

■ **To add components (transistors, passives, sources etc) you must use the Create -> Instance option**

Adding Instances

- **Whenever you add an instance you need to use the *same* library structure (library, cell, view) idea as when you created your own cells.**

- **You need to use the symbol view**

- **Luckily, Composer allows you to Browse for the correct cell using the library manager**

Check and Save

- **Don't forget to check and save your schematic**

- **Once you have saved the schematic, you can close it using File -> Close**
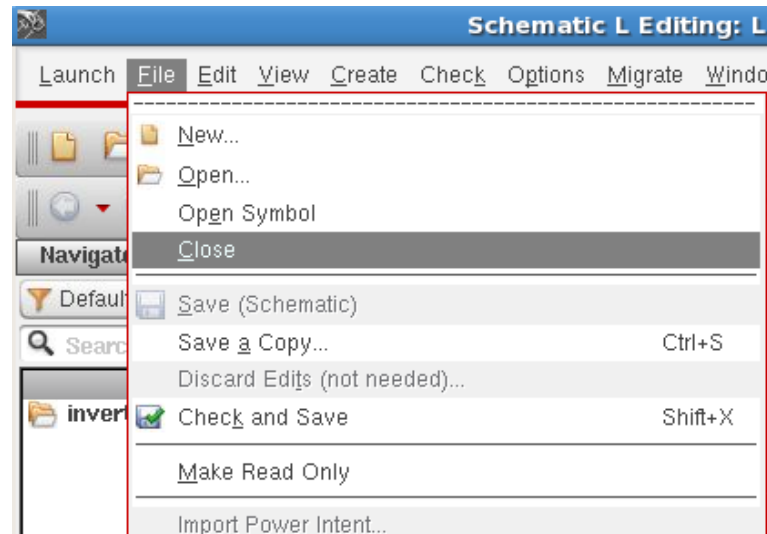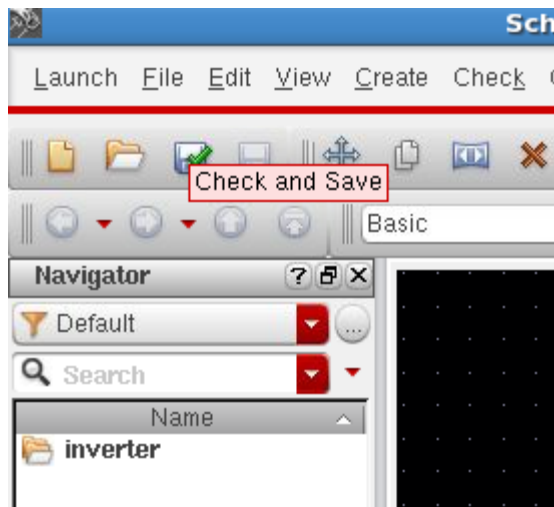
Create a simple circuit

- **Cadence provide a simple library of electronic *primitives* such as voltage sources, resistors, capacitors etc**

- **These can be constructed into circuits and simulated, *however they do not have the information to be physically fabricated***

- **We can use them to build a simple test circuit to learn how the design software works**

Create a simple RC circuit
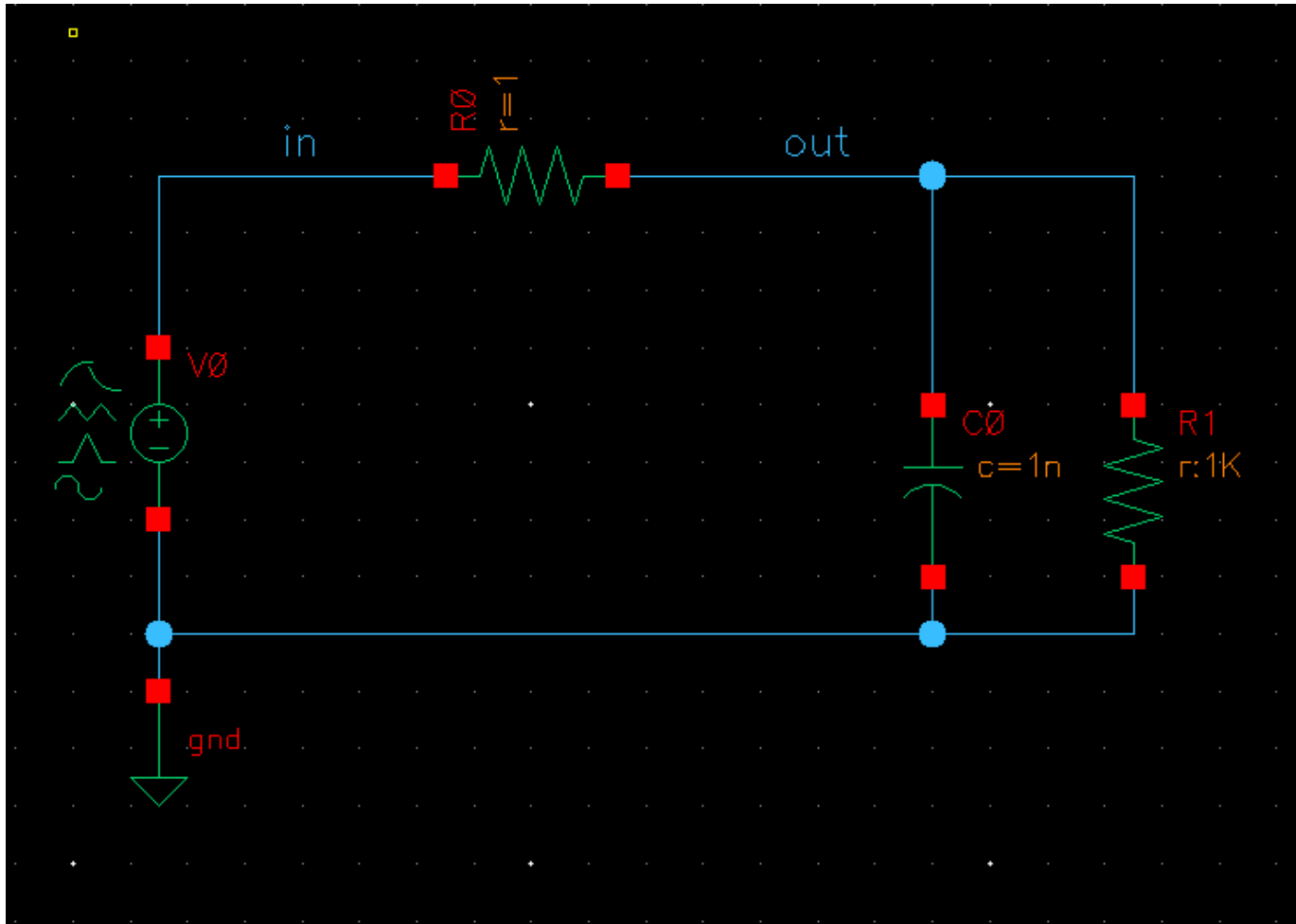
Add a pulse voltage source

■ **AnalogLib->vsource**

- Source type: pulse

- DC voltage=1

- Zero value: 0

- One Value: 1

- Period of waveform:20ns

- Pulse width: 10ns

- Rise time: 100ps

- Fall time: 100ps

Reference to ground

- **AnalogLib -> gnd**

- **No parameters**

- **Connects globally to ground**
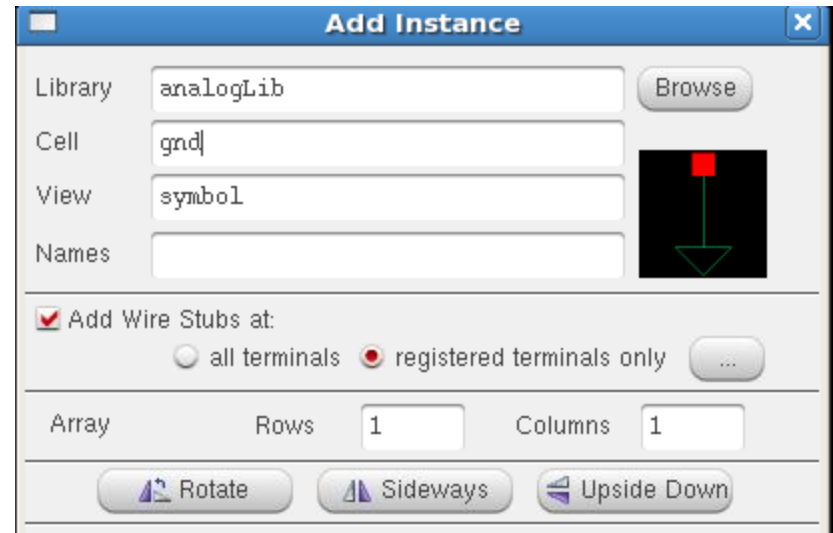
Add resistors, inductor & capacitor

- **AnalogLib->res**
  - resistance = 1 for series
  - resistance = 1k for load

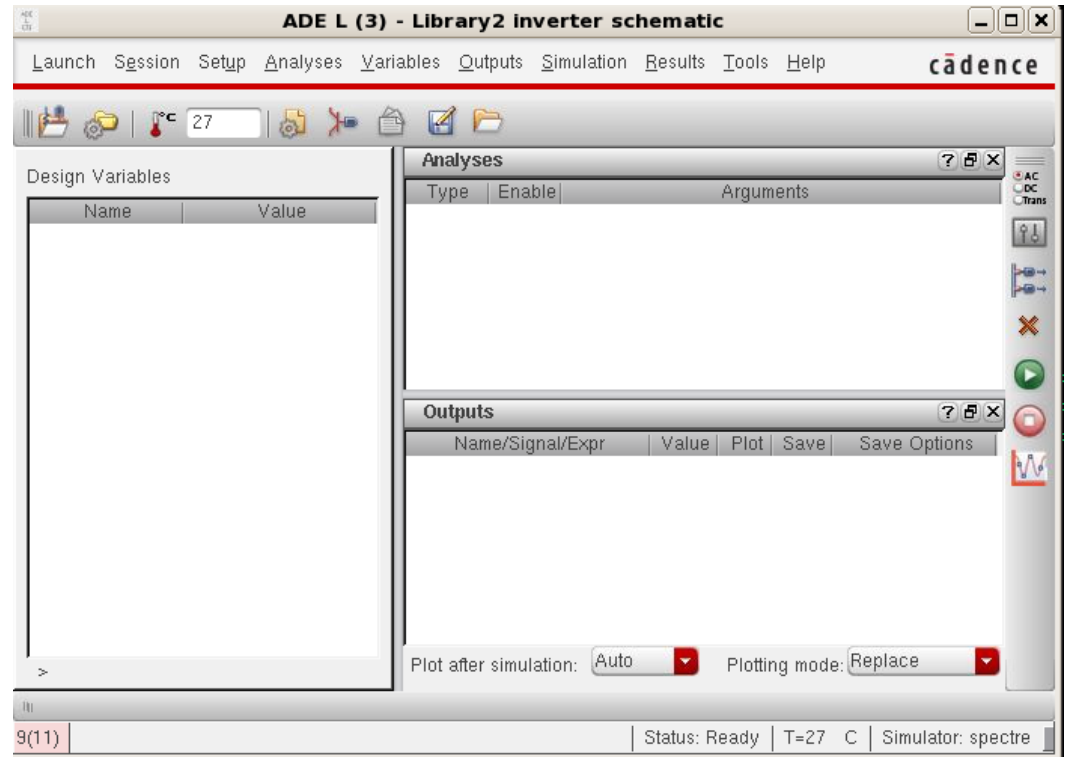- **AnalogLib->cap**
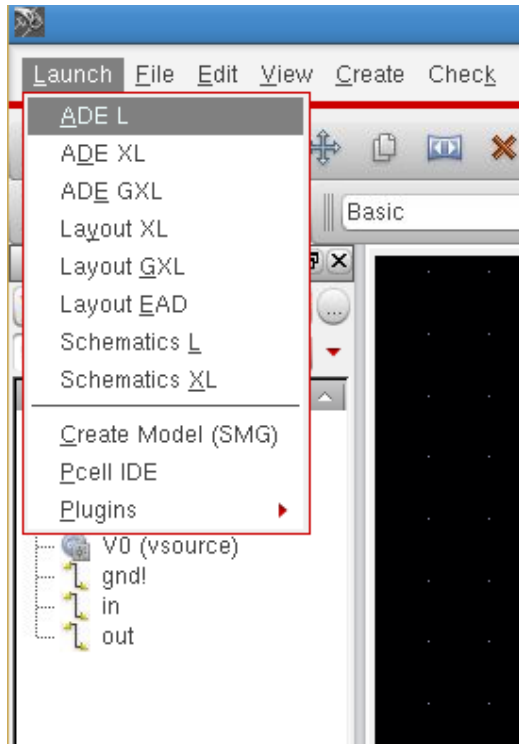 - capacitance = 1n

Connect up

- **Use Create -> Wire(narrow)**
  - this gives a connect function – wire up the circuit

- **Add wire names**
  - Create -> Wire Name
  - Label the input vin and the output vout
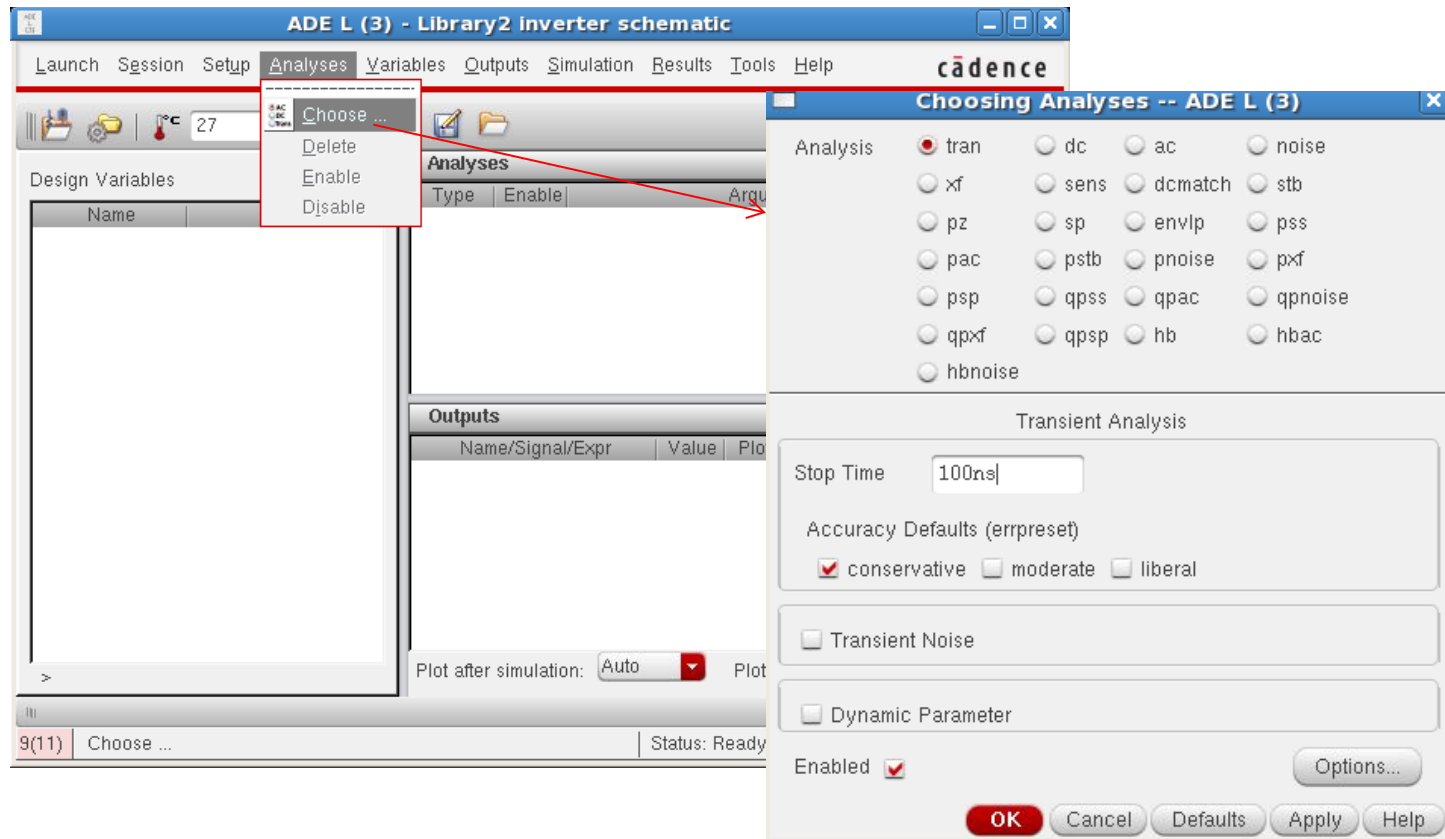
- **Check and Save**

Simulate

- **Click on Launch – ADE L**

Simulate

- **Click on Analyses Tab and choose tran analysis and set Stop Time: 100ns as shown below**
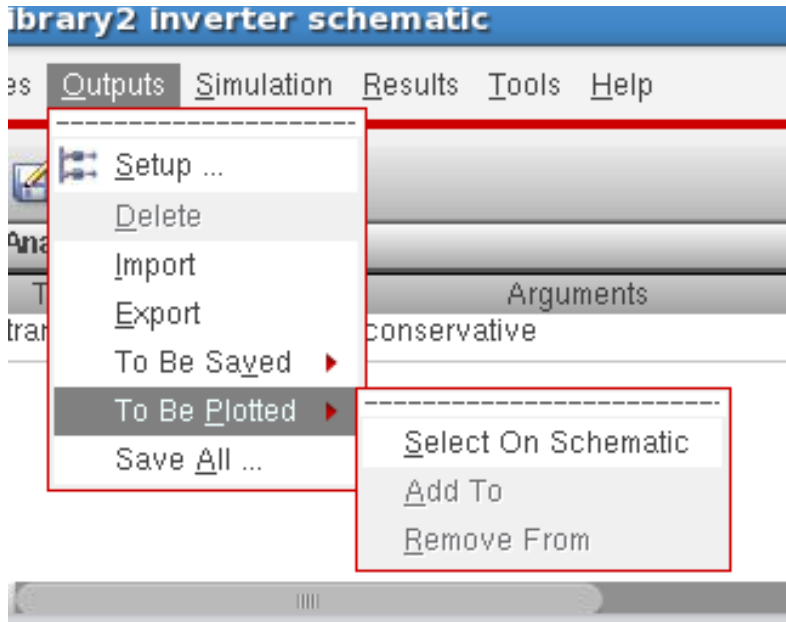
Simulate

■ **Select output to be plotted by clicking on:**
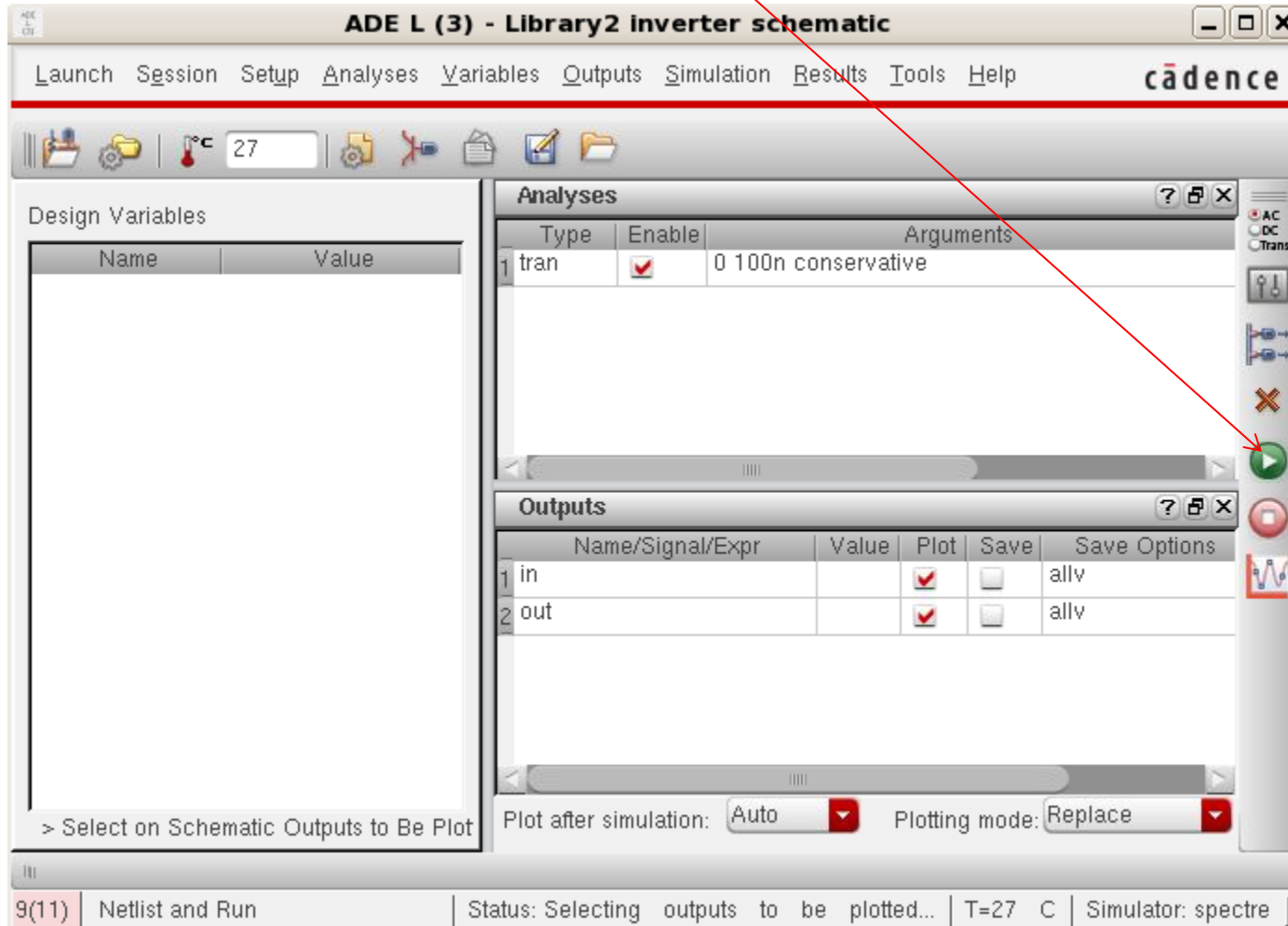
Outputs Tab - To be Plotted – Select on Schematic

Then choose the wires: in and out in the schematic window
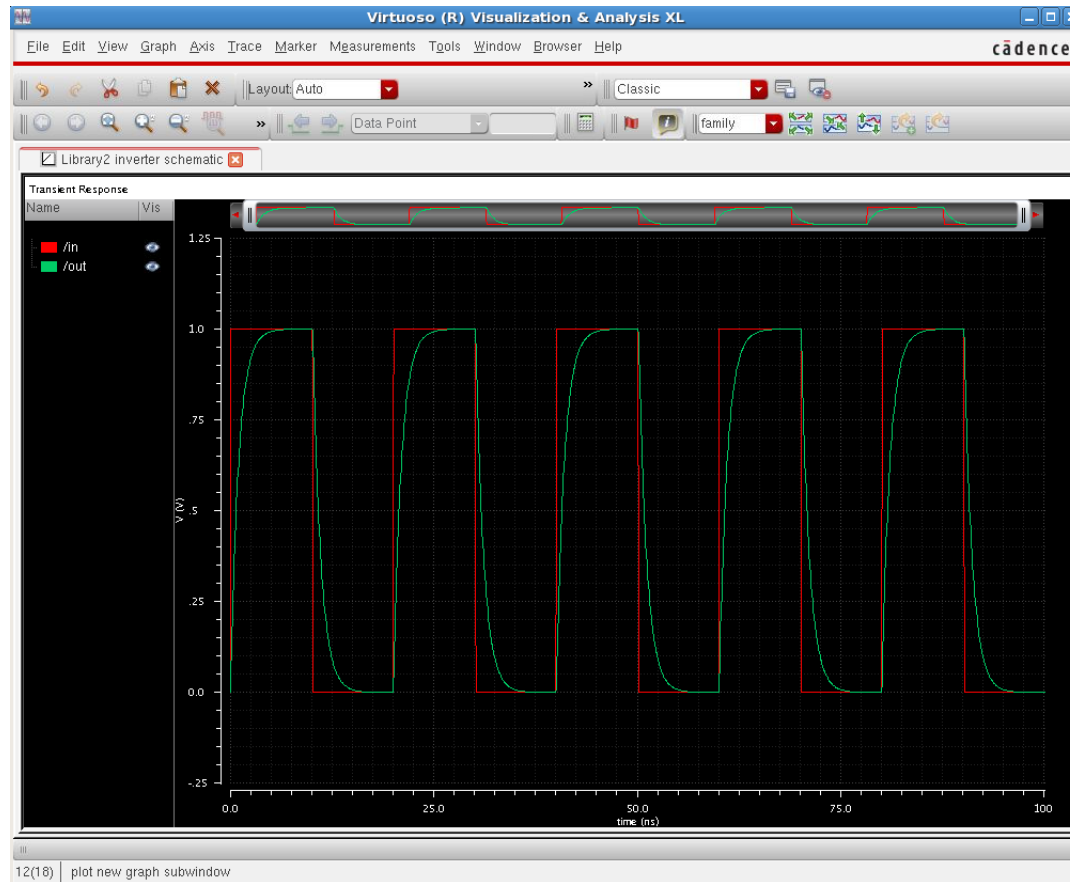
Simulate

- **Than click Netlist and Run Tap to simulate**

Simulate

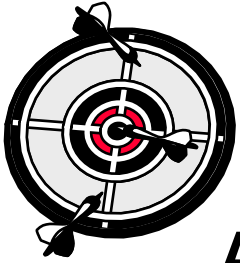- **You should be able to see the simulation results plotted**

- **You can now close the simulation window and the schematic view of your cell**

re-opening the schematic

- **From the library manager, find your schematic cell view**

- **Right Click on the selected cell view**

- **Choose the "Open" option**

- **Composer will open your schematic for editing**

Learning Outcomes

After completing this unit, you should be able to:

1. **Set up a directory to run Cadence**

2. **Install the design files for the AMS 0.35um PDK**

3. **Run the Cadence Design Manager (ICFB)**

4. **Create a Cadence Project**

5. **Create a simple Cadence Cell View (Schematic)**

6. **Link to an external library in Cadence**

7. **Use the external blocks in your design**

How does Cadence Know about Libraries?

- **Cadence uses a file called cds.lib to establish the libraries**

- **It looks first in the current working directory, then your home directory for this**

- **It also has a default cds.lib if none exists**

- **This is an ASCII file that has the library name and actual location**

- **environmental variables are often used in these path definitions to make them general and portable**

Which cds.lib file?

- **It is good practice to set up your working directories with custom design kit settings AND libraries**

- **Keeping projects separate makes administration and management of files MUCH easier.**

- **Therefore, each project directory should have its own cds.lib**

How can I reference someone else's design?

- **Given that the definition of a library is done using a name and pathname, can we use this to reference other libraries not in our design space?**
  - Yes, but remember that the technology MUST be compatible
  - This is how the design kits work – we simply reference the libraries in the design kit location
  - You may also not have edit rights – perhaps read only

- **You need to know, therefore:**
  - The library NAME
  - The library LOCATION

An example

- **There is an examples library which has the name library_lab1**

- **It is in the location /home/bh9/lab1/library_lab1**

- **How can we link to that existing library?**

- **First load up the library manager and select :**
  - Edit->Library Path

Edit Library Path

- **Type in the library name you need**

- **Then type in the absolute path name to the directory, including the library name itself**

- **Close the editor and save the cds.lib file**

New Library….

- **Now the new library is available in the library manager**

Summary

- **You can now create and edit your own libraries**

- **You can see how to link and share other directories into your Cadence library manager**

- **In the library_lab1 library you will find a basic RC circuit**