

ELEC 301

Lab 2: Cadence Basic

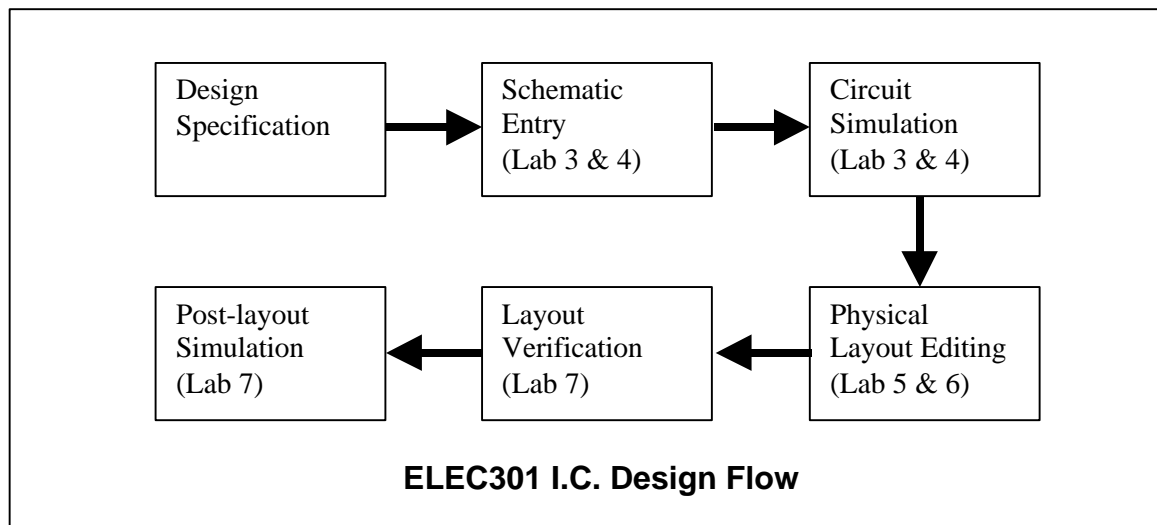
Revision: 2.1

Last modified: Aug. 98

Introduction

In this class, you will be introduced to the Cadence suit of IC design tools. These tools are a very powerful set of tools. They are very popular in industry and can do everything from schematic entry, circuit simulation and layout all the way to design a complete microprocessor. Consider that the Power PC group and Pentium group both use a subset of the tools that we will use in this class. Obviously, these tools are very powerful, but because of that they also have a pretty high learning curve.

In the next lab, the basics of Cadence Design Framework and schematic capture of a simple inverter in transistor level will be covered. Notice that unlike most of the other design courses, we will not be working with discrete components (i.e. TTL parts, transistors, LEDs, etc.) in this class. Instead, we will design the actual chips themselves, thus giving us greater flexibility.



Starting Cadence

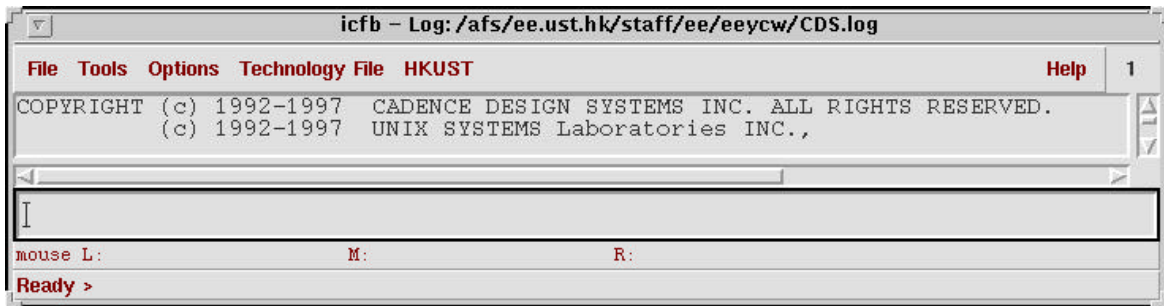
Step 1. Change directory to your Cadence working directory by typing

```
cd ~/cadence
```

1.1 Start Cadence Design by typing

```
i cfb &
```

- This step will take few seconds to execute. After that, the main Cadence window, the Command Interpreter Window (CIW) appears.



- The title of the window will be “icfb – Log: /afs/ee.ust.hk/.../CDS.log”. This “CDS.log” file will store all the Cadence commands you use during this session. It is also very useful to help you figure out why some problems occur.
- You should always watch the messages in the CIW window very carefully during the lab session and report any errors or warnings to the lab instructor.

1.2 Quit Cadence by clicking the *left mouse button* on the menu **File -> Exit ...** of the CIW. Click on “**Yes**” to answer the confirmation box to finally exit Cadence.

IMPORTANTCE: Never attempt to quit Cadence by other means. It will leave the Cadence program running even you logout.

1.3 To logout of the workstation, hold down the *right mouse button* on the screen background (an area where no windows exist), drag the mouse to the “**Exit...**” command and click on “**Exit**”. You are now logged off the system.

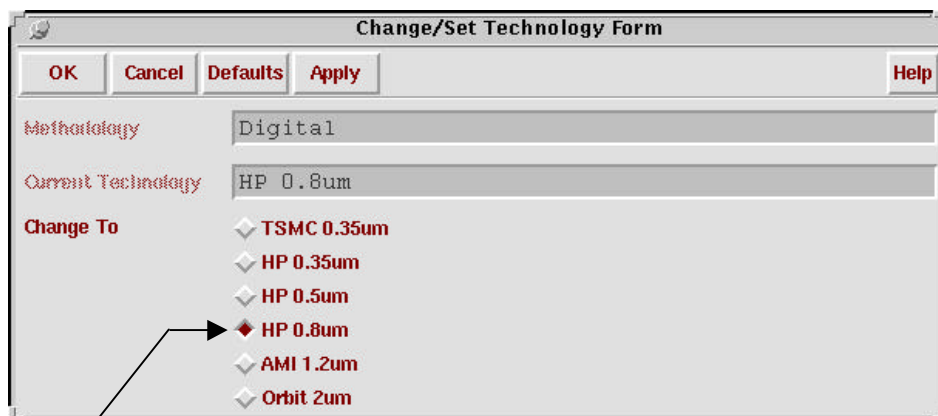
Creating Cadence Library

You will now create your own design *library*. You can think of libraries as being analogous to directories in a file system. Like grouping related files in a directory, we will group all our related design data in a Cadence library. For example, you may want to create a new library for your project to put all related designs in it.

Step 2. Start Cadence again in your Cadence working directory.

2.1 In CIW, click on **HKUST -> MOSIS -> Change/Set Default Technology ...**

- A window entitled *Change/Set Technology Form* will appear as below



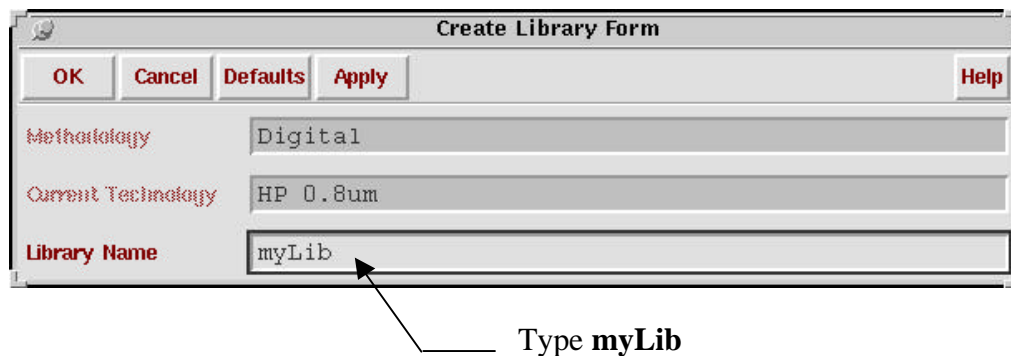
Click on this button

- 2.2 Then, click the button next to **HP 0.8um** to change the technology to HP 0.8um process and click on the **OK** button. This brings up another window entitled *Information*. Click on the **Close** button. Quit and restart Cadence.

The technology defines the design rules, physical layer names, colors, SPICE simulation parameters, etc. The choice of technology depends on the process that your designs are using. In this course, you will be using a MOSIS 0.8um process and all the information regarding that process is stored in a technology library.

- 2.3 In CIW, click on **HKUST -> MOSIS -> Create Library ...**

This brings up another window entitled *Create Library Form*. Click on the *Library Name* field (not on the words themselves, but rather in the box opposite to it). The box should be highlighted now with a black border. Type in the library name, *myLib*. Also, notice that, next to the *Current Technology* field, it should display "HP 0.8um" that you have just selected.



- 2.4 Click on **OK** in the *Create Library Form*.

- The *Create Library Form* will disappear. Look for the following message in the CIW window:

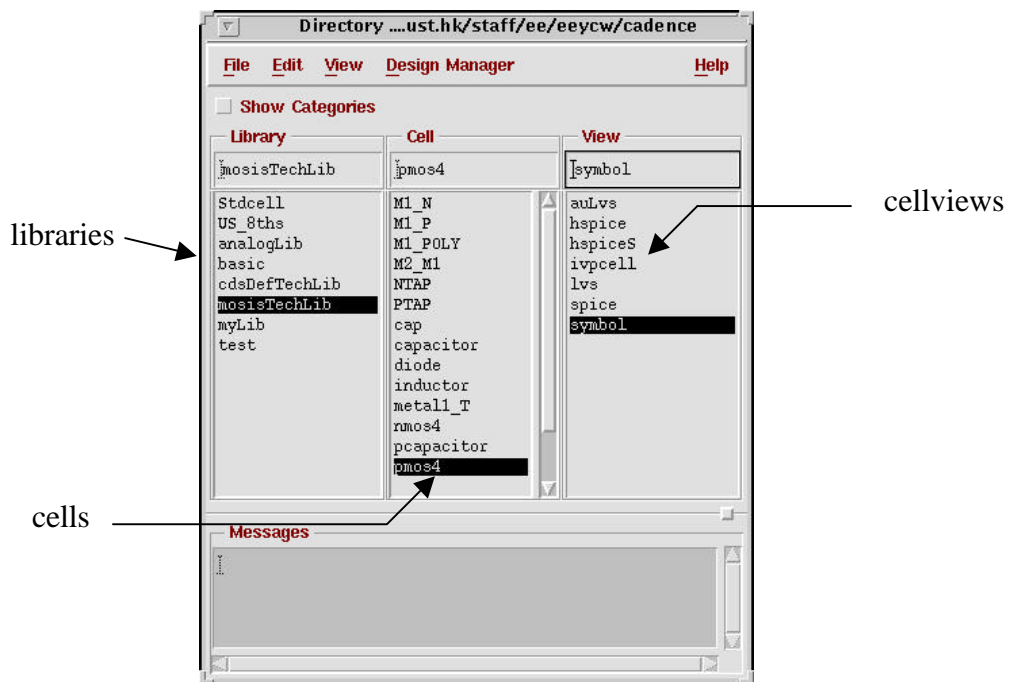
```
Info: Creating library...
Info: Library created.
Info: Click on View->Refresh in Library Manager for refresh library list
```

- If there are any warnings or errors in the CIW, show them to the lab instructor to ask for remedial action.

Using Library Manager

Cadence uses the *Library Manager* to do all of its file management tasks such as opening a file, creating a new file, deleting files, copying files, etc.

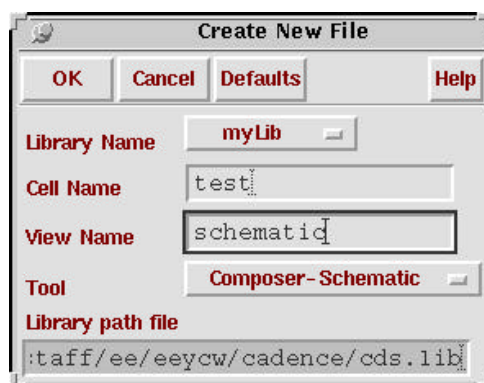
Step 3. In the CIW, click on **Tools -> Library Manager ...**. The library manager will appear as shown below:



Step 4. Create new design.

4.1 In CIW, click **File -> New -> Cellview ...**

The *Create New File* form will appear. Choose the *Library Name* and *Tool* to be **myLib** and **Composer-Schematic** respectively. Also, enter the *Cell Name* and *View Name* as shown below.



4.2 Click on **OK**.

- A blank schematic window will appear. You will learn about schematic entry in Lab 3. You should also notice that the newly created cellview appears in the *Library Manager*.

4.3 In the schematic window, click on **Window -> Close**.

4.4 Create one more design for exercise.

- Repeat 4.1 except *View Name* and *Tool* are changed to *layout* and **Virtuoso**.

4.5 We will discuss the layout editor in Lab 5.

- Close the layout window with the same procedure as you just done before.
- You have one *Cell* called *test* and 2 *Views* associate with it under *myLib*.

Step 5. Open existing designs using *Library Manager*.

5.1 In the *Library Manager*, expand the design by clicking on *myLib*, *test* and *schematic* under *Library*, *Cell* and *View* entries in sequence.

5.2 Hold down the middle mouse button at the expanded *schematic*. A sub-menu will appear.

5.3 Drag the mouse pointer to **Open...** and release the mouse button.

- The schematic shows up.

5.4 Try to open *test – layout* and other cellviews under different directories. Notice that for designs not owned by you, you should drag the mouse pointer to **Open (Read-Only)** instead of **Open...**

Step 6. Copy design.

6.1 Hold down the middle mouse button at the cellview *myLib - test – schematic*.

6.2 Drag the mouse pointer to **Copy...** and then release the mouse button.

- A *Copy View* form will appear.

6.3 Fill in the form as shown below.

The image shows a 'Copy View' dialog box with the following fields and options:

- From:**
 - Library: myLib
 - Cell: test
 - View: schematic
- To:**
 - Library: myLib
 - Cell: test1
 - View: schematic1
- Options:**
 - Copy Hierarchical
 - Skip Libraries (Stdcell basic test ar)
 - Copy All Views
 - Views To Copy: schematic
 - Update Instances
- Buttons: OK, Apply, Cancel, Help

6.4 Click on **OK**.

- You will see that there is a cell called *test1* with cellview *schematic* in the *Library Manager*.

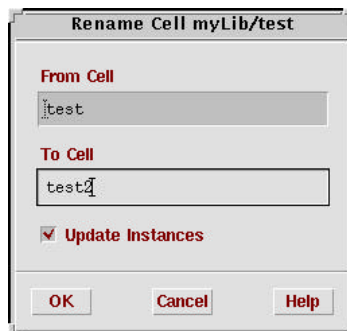
Step 7. Rename design.

7.1 Hold down the middle mouse button at the cell *myLib - test*.

7.2 Drag the mouse pointer to **Rename...** and then release the mouse button.

- A *Rename Cell myLib/test* form shows up.

7.3 Fill in the form as shown below.



The image shows a dialog box titled "Rename Cell myLib/test". It has two text input fields: "From Cell" with the text "test" and "To Cell" with the text "test2". Below these fields is a checked checkbox labeled "Update Instances". At the bottom of the dialog are three buttons: "OK", "Cancel", and "Help".

7.4 Click on **OK**.

- You will see that the original cell *test* is renamed to *test2* in the *Library Manager*.

Step 8. Delete design.

8.1 Hold down the middle mouse button at the cellview *myLib - test2 - layout*.

8.2 Drag the mouse pointer to **Delete...** and then release the mouse button.

8.3 Click on **OK** on the *Delete Cellviews* form. The *Delete Confirmation* box will pop up. Click on **Yes**.

- You will notice that the cellview *layout* of *test2* disappears in the *Library Manager*.

IMPORTANT NOTES

- Never start two “icfb” simultaneously. It will make you Cadence data corrupted and can’t be recovered.

End of Lab