

# CADTOOLS TUTORIALS

School of Electrical Engineering and Computer Science  
Washington State University, Pullman, WA-99163.

---

This tutorial is an introduction to Cadence tool for circuit design and simulations. It deals with the schematic entry of the circuits and their simulations using Cadence tools

- Composer for schematic capture
- Analog Artist Environment for design simulations.

Composer is the tool for schematic entry of the circuit designed. Once the schematic is ready, the simulations can be done using the Analog Artist Environment.

### Logging onto the SUN THIN CLIENTS or any LINUX MACHINES

#### Creating a working directory

Once logged in successfully, open a terminal window. To do this

Either:

1. Right click on the desktop and select the **Open Terminal** option.

OR

2. From the **Menu** click on **Applications** select **Accessories** and then select **Terminal** (Applications->Accessories->Terminal).

At the prompt within the terminal window: type **mkdir Cadwork** to create a working directory named Cadwork. Then enter the directory by typing **cd Cadwork** at the prompt.

Now you are in the directory Cadwork. Within the Cadwork directory create two other directories: CMOS065 and NCSU. Use the "mkdir" command to make these two subdirectories. All the work done will be stored in these subdirectories depending on whether you are using the CMP or NCSU Design Kits.

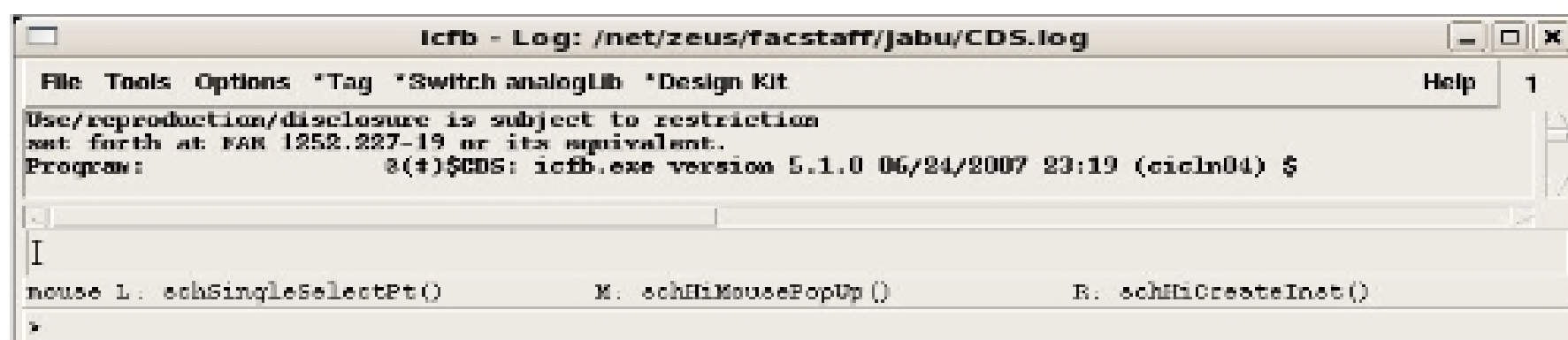
#### Starting Cadence

Depending on the design Kit you are using you will need to change directory into the corresponding design kit: for an example "cd CMOS065" for the CMP 65 nm Design Kit or "cd NCSU" for the NCSU design kit. We will use the CMP process as an example from here on and the procedure will be the same for the NCSU design kit.

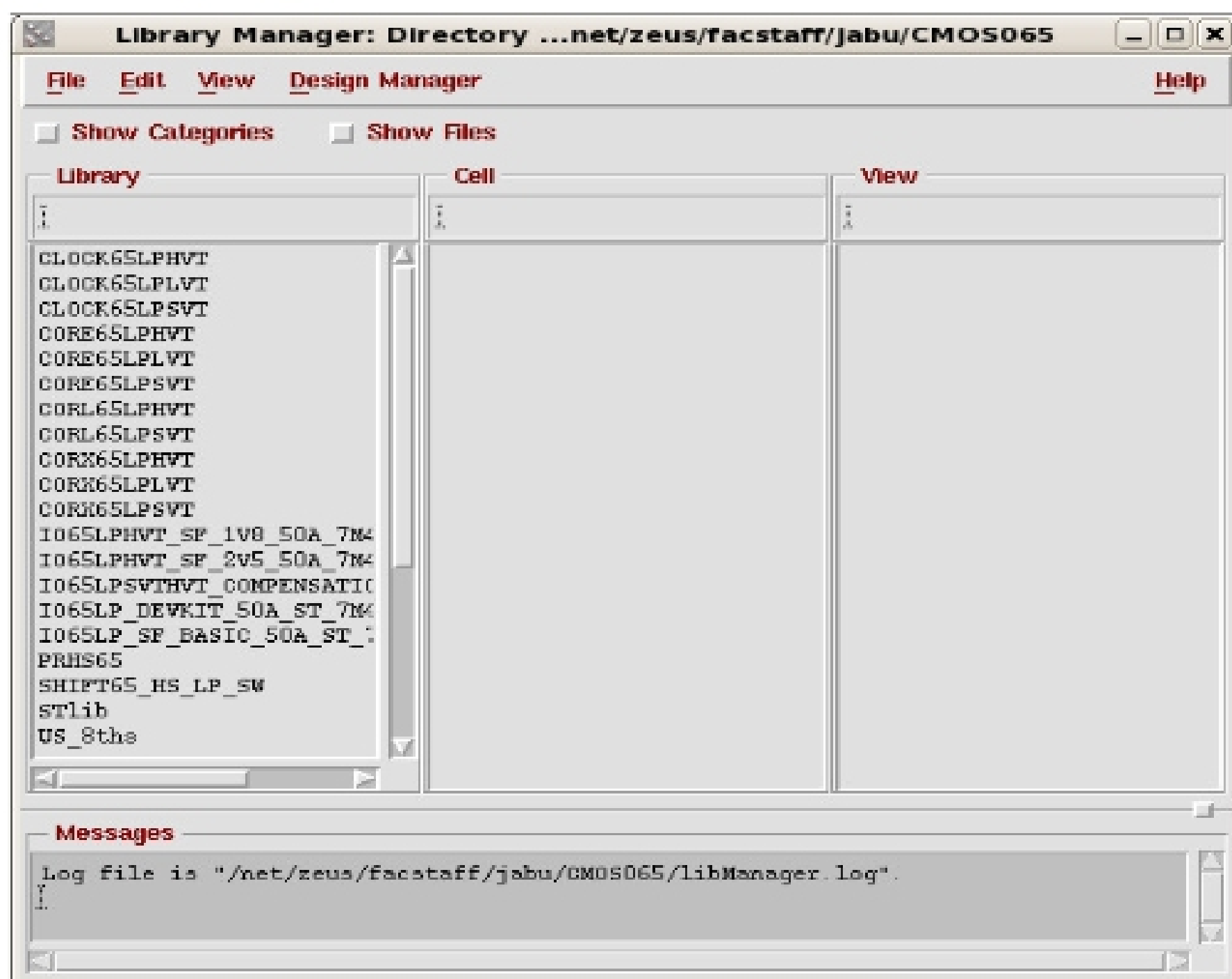
Invoking Cadence for the first time: change directory to CMOS065 and issue the command: **/net/ictools/bin/cmos65** (this copies essential files and scripts into your user space within the CMOS065 subdirectory which will now serve to hold all your work related to the CMP 65 nm design kit). The command will launch cadence and display the **command interpreter window** (CIW) shown below. From here you are able to create your libraries (covered later).

Edit your **.cshrc** file to include the line: **setenv PATH "\${PATH}:/net/ictools/bin"**

The next time you login to use cadence you must be able to issue the command "cmos65" from the prompt but from the **~/Cadwork/CMOS065/** directory.



Open the Library manager window, by clicking on the Tools menu and selecting Library Manager (Tools->Library Manager). This launches the window shown below.



You will also get a "What's New" window which you can read and then close or minimize.

To get online manuals on the 65 nm design kit type "cmos65doc" at the prompt from within ~/Cadwork/CMOS065 and you will get a window that lists the available manuals. To get Help on Cadence click the Help Tab and a browser will be launched from which you can search the manuals for the topic you want to learn about.