

Chapter 4

Schematics and Symbols

This chapter will deal with the creation of schematic drawings and symbol generation. The tool used is called *Virtuoso Schematic Composer* which is a trademark of **Cadence Design Systems, Inc.**

More on this topic can be found in the on-line manuals *Virtuoso Schematic Composer User Guide* and the *Virtuoso Schematic Composer Tutorial*.

4.1 Creating a Schematic Drawing

The library, in which to store the cell, has to be created first. The procedure for this is described in section 3.4.2. The cell can then be created by *CIW: File > New > Cellview*. The view should be set to *schematic*. It is important that *schematic* is spelled correctly. Although Cadence can build a schematic drawing with any viewname, the hierarchy will not work properly.

4.1.1 Opening an Existing Schematic

A schematic drawing that already exists can be opened for editing by selecting the library, cell, and view in the *Library Manager* and then using the middle button on the mouse to bring forward the pop-up menu with the *Open* command.

The command *CIW: File > Open* can also be used.

4.1.2 Adding Instances

A schematic is comprised of components, input and output terminals and their interconnections. Figure 4.1 shows a memory cell consisting of two inverters, two

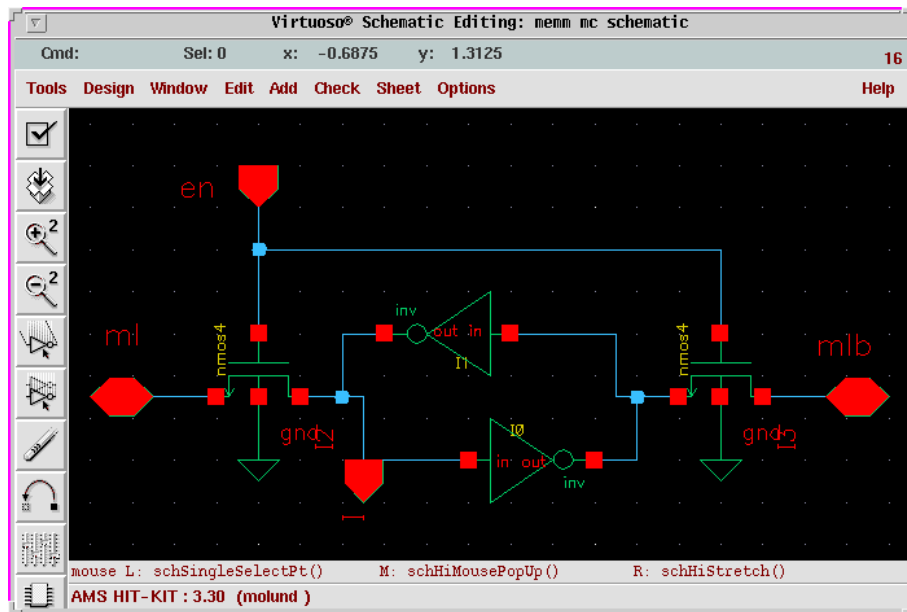


Figure 4.1: The schematic view of a memory cell.

transistors, some terminals, and a global ground connection. Components are included from a reference library as instances in the design. Bigger structures like logic gates or amplifiers can also be instantiated.

1. Select *Add > Instance* and the form **Add Instance**, fig. 4.2, will appear.
2. The fields **Libray**, **Cell**, and **View** has to be filled out to designate the cell that is to be instantiated. By clicking the **Browse** button a browser is started from which the wanted cell can be selected. The view for cells that are to be instantiated in a schematic should be *symbol*.

When the cell has been selected the form will extend to show if there are any parameters to the cell in question, width and length of a transistor for instance.

In the field **Names** a name of the instance can be given or Cadence will supply one. The orientaion can also be changed by the buttons **Rotate**, **Upsidedown**, and **Sideways**.

3. When the form is filled out an outline of the selected cell is connected to the cursor when it is moved back into the design area. If the cursor cannot be moved freely try to switch off **Gravity** in *Options > Editor*.

Place the instance by clicking left when the symbols is at the correct location in the schematic. A new instance of the same type and size appears and can be placed in the same manner. If another cell is wanted select it in the browser.

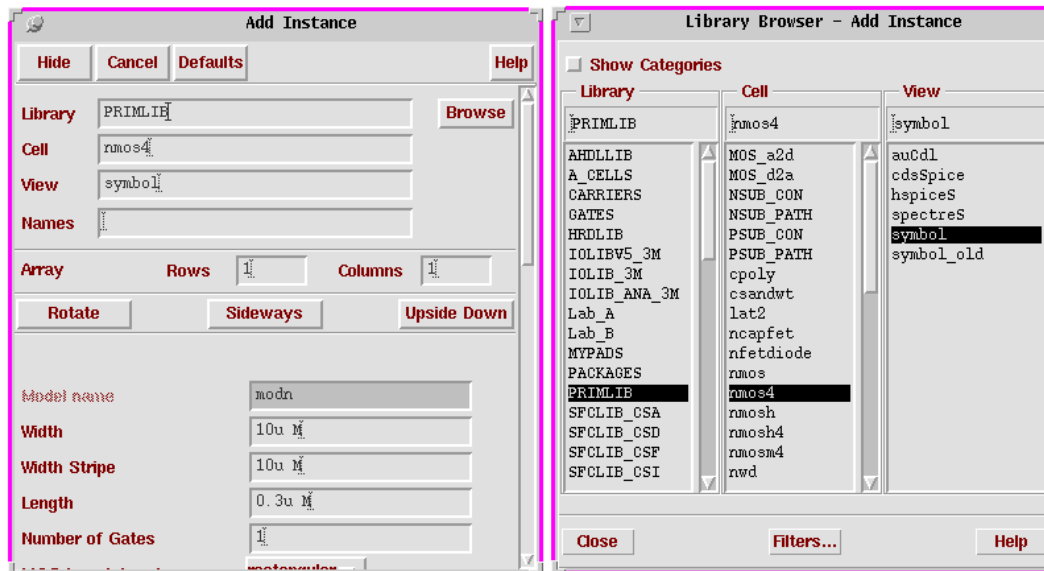


Figure 4.2: Adding an instance

4. When all instances are placed click on **Cancel** in the form or press **Esc** to abort the **Add Instance** command.

4.1.3 Modifying Instances

After they are placed, instances can be modified in several ways. Their parameters can be altered, they can be moved and rotated, or even the type can be changed. It is also possible to delete them.

Changing parameters

The parameters, properties, can be modified by first selecting the object and then executing *Edit > Properties > Objects (q)*. In the form any parameter can be changed, it is also possible to change the type of the cell. Press **OK** when finished.

Moving instances

Instances can be moved around or their orientation can be changed by using the command *Edit > Move (M)* and then selecting the object to be moved in the working area. This will cause the instance to be disconnected from the schematic and it will follow the cursor like it did when it was first placed. By pressing **F3** a small form appears from which the orientation can be changed.

The related command *Edit > Stretch (m)* is used to move an instance without disconnecting it from the schematic.

Deleting instances

A selected instance can be deleted by *Edit > Delete* or by pressing the Delete button on the keyboard.

Deleted instances can be brought back with *Edit > Undo (u)*.

4.1.4 Pin Connections

In Cadence *Pins* are used as the primary input and output terminals. They are needed so that the system can connect all signals through the hierarchy and for simulation.

Figure 4.3: The Add Pin form.

A pin is created and placed from the **Add Pin** form which is activated by *Add > Pin (p)*. The names should be entered in the field **Pin Names**. It is possible to enter several names at the same time as in figure 4.3, separate the individual names with a space. **Direction** should be set correctly.

The first pin in the list will be connected to the cursor so that it can be placed, then it will disappear from the list and the next one becomes active.

4.1.5 Wires

Wires are used to connect the terminals of the various instances to each other.

1. Execute *Add > Wire (narrow)* (**w**) in the schematic window.
2. The starting point is selected by clicking on it or by pressing the **s** key when the little yellow diamond shaped marker appears over the terminal.
3. By clicking left on a point it is possible to change direction of the wire.
4. The connection to the end point are made in the same way as at the start, clicking left on the terminal or using **s**.
5. The wire can be ended in “thin air” by clicking twice at the same coordinates.

4.1.6 Verification and Saving

When the schematic is finished it should be checked for errors. The automatic check program will detect errors like floating wires or terminals, short circuits, etc. If a symbol view of the cell also exists, any discrepancies between the schematic and the symbol will be reported.

Use *Design > Check and Save* (**X**) to start the checker. If any errors are found a small dialog box will appear, reporting what went wrong, otherwise the cell will be saved back into the library. Any errors can be examined by *Check > Find Markers* (**g**) which will bring up a form listing the errors and what caused them.

4.1.7 Creating Symbols

All schematics that are to be used as instances by another schematic must have a corresponding *symbol* view. The two inverters in figure 4.1 are symbols of a home-made inverter schematic. Fig. 4.4 shows the windows activated for the symbol generation.

1. *Design > Create Cellview > From Cellview* will bring up the first form, **Cellview From Cellview**. The field **To View Name** should read **symbol**.
2. An **OK** will open the second form **Symbol Generation Options** in which the location of the pins, on the symbol, can be changed.
3. When this form is **OKed**, the symbol editor will start up where further editing can be done if the default rectangular box is not sufficient. The green structures in the view is what will be shown when the symbol is used in a schematic and the red rectangle should contain everything with the pins on the border.
4. Before the symbol editor is closed down the view should be saved!

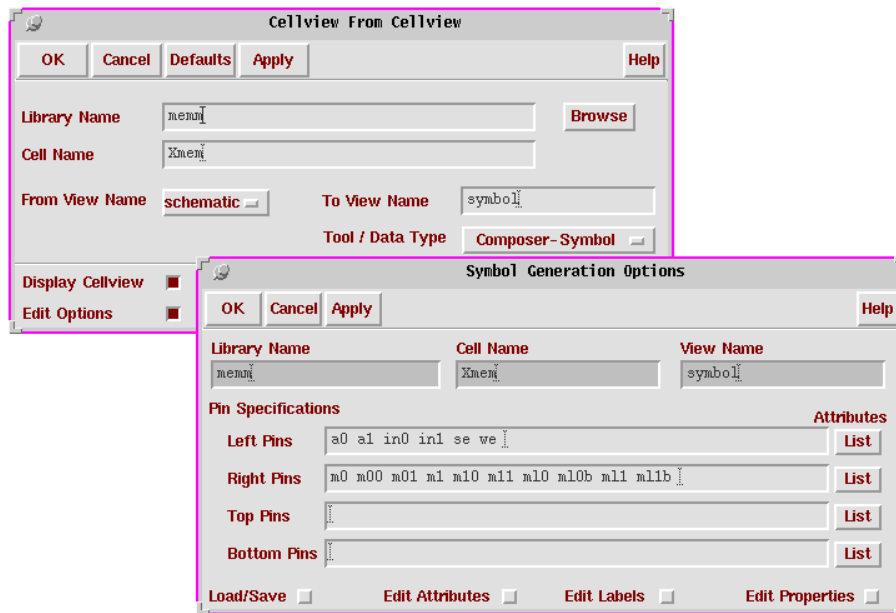


Figure 4.4: Generation of a Symbol View

4.2 Plotting a Schematic

From the departmental machines the following procedure can be used to generate a plot of a schematic.

From a schematic window, with the cell to be plotted showing, the job is launched by *Design > Plot > Submit* and the form **Submit Plot** will open. In this the button **Plot Options** gives a form where the printer name can be changed to a suitable one. The names with a trailing **bw** will convert the view to black and white before sending it to the printer, this works better for some layouts.

Since the efd-computers can not send to the departmental printers and plotters the jobs has to be written to a file instead. In the **Plot Options** form the button **Send Plot Only To File** can be selected and a file name filled in. This will produce a postscript file which can later be sent to a printer or included in a laboratory report.