7.43 Release the middle mouse button

The object you selected will now be deleted.

Alternativaly use the bind key ‘***Delete***’ to delete the selected objects.

Note: Any object selected will be deleted, so make sure that ***only*** the object you want to delete is selected.

The ***Undo*** command can be used to reverse the action of the last command.

7.44 Click on ***Edit->Undo*** on the schematic menu banner.

This command has the bind key ***u***.

The ***Move*** command can be used to move selected objects.

7.45 ***Select*** the object you want to move.

7.46 Click on ***Edit->Move*** on the schematic menu banner.

The Move form appears. Hide the form by clicking on ***Hide***.

7.47 Click at the reference point for the move.

7.48 Click to place the object at its destination point.

7.49 Press ***Esc*** to cancel the Move command.

You may also use the bind key ***M*** for this command.

Copying objects.

7.50 First ***select*** the objects you want to copy.

7.51 Move the cursor onto the schematic background, outside the design.

7.52 Hold down the middle mouse button to pop up the schematic menu.

7.53 Slide the mouse down to ***Copy***

7.54 Release the middle mouse button

The Copy form appears. Hide the form by clicking on ***Hide***.

7.55 Click at the reference point for the copy.

7.56 Click to place the object at its destination point.

You may also use the bind key ***c*** for this command.

Experiment with these Graphical editing facilities until you are familiar with them. Once you have finished close the window by clicking on ***Window->Close*** on the schematic menu banner.

At this point a prompt appears in the CIW, asking you to Enter the first corner of the box you wish to enlarge.

7.26 Click on one corner of the area to enlarge.

7.27 Move the mouse to the opposite corner, as you do a bounding box will move with the mouse, then click to zoom in.

Now a zoomed in portion of the schematic appears.

Note: There is also a bind key which will let you zoom in. This key is the letter *z*.

The ***fit*** command is used to redisplay the whole design.

7.28 Move the cursor onto the schematic background, outside the design.

7.29 Hold down the middle mouse button to pop up the schematic menu.

7.30 Slide the mouse down to ***Fit***.

7.31 Release the middle mouse button.

The design is now redisplayed so that the whole design is shown. This command also has a bind key. Press the key ***f*** to fit the design.

Selecting and ***Deselecting*** objects.

7.32 To select objects click the left mouse button over the object.

7.33 To deselect objects move the cursor onto the schematic background and click the left mouse button.

7.34 To select all objects use the bind key ***Control-a***.

7.35 To deselect all objects use the bind key ***Control-d***.

7.36 To select objects by area use the bind key ***A***,***shift-a***

7.37 To deselect objects by area use the bind key ***D***,***shift-d***.

7.38 To select multiple objects hold the ***Shift*** key down and click the left mouse button over the required objects.

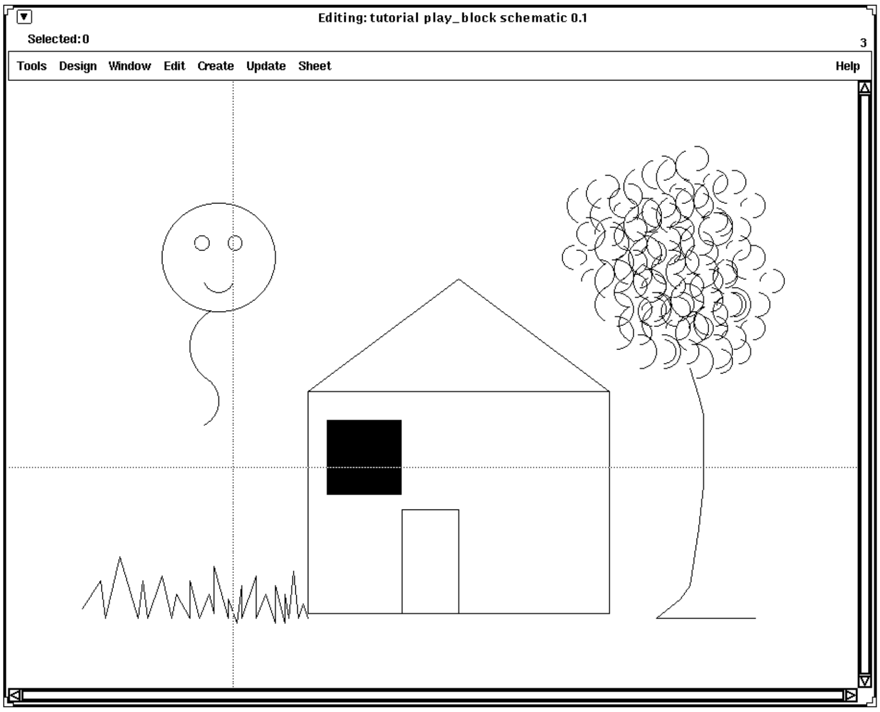
Deleting objects.

7.39 First ***select*** the objects you want to delete.

7.40 Move the cursor onto the schematic background, outside the design.

7.41 Hold down the middle mouse button to pop up the schematic menu.

7.42 Slide the mouse down to ***Delete***



The ***Pan*** command is used to move the viewing area of the window without changing the scale of the design.

7.18 Click on Window->Pan on the schematic menu banner.

7.19 The CIW displays the message

'Enter the reference point for your pan:'.

7.20 Click on the schematic window.

7.21 Move the mouse to the destination point and click.

The window will now be redisplayed with the new viewing area.

The ***Zoom*** commands increase and decrease the size at which the design is displayed.

7.22 Move the cursor onto the schematic background, outside the design.

7.23 Hold down the middle mouse button to pop up the schematic menu.

7.24 Slide the mouse down to ***Zoom in***.

7.25 Release the middle mouse button.

Wait while the library is created.You now have to copy the ***play_block*** schematic from the reference library to the ***tutorial*** library.

7.6 Click on ***Design Manager->Library Browser*** on the CIW to open the Library Browser.

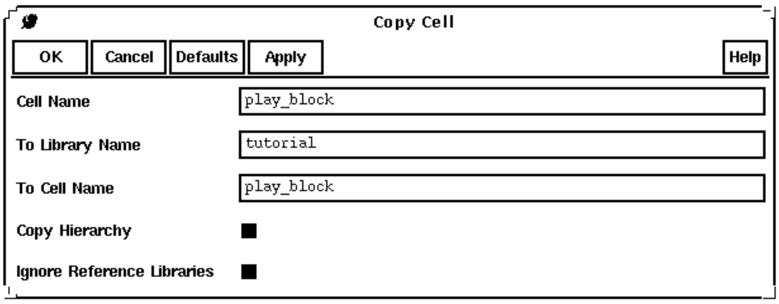
7.7 Click on the ***HOUSE*** library to show its contents.

7.8 Hold down the middle mouse button over the ***play_block*** schematic to reveal the menu.

7.9 Slide the mouse to the ***Copy*** option.

7.10 Release the mouse.

The Copy Cell form appears.



7.11 Fill in the ***Library Name*** with the library ***tutorial***.

7.12 Type in the ***Cell Name***.

7.13 Click on ***OK***.

The Cell ***play_block*** is now present in your own library - it can now be edited.

7.14 Click on the ***tutorial*** library to show its contents.

7.15 Click on ***play_block*** to show its cellviews.

7.16 Move the mouse over the cellview ***schematic*** and hold down the middle mouse button to reveal the menu.

7.17 Move the mouse to ***edit*** and release the button.

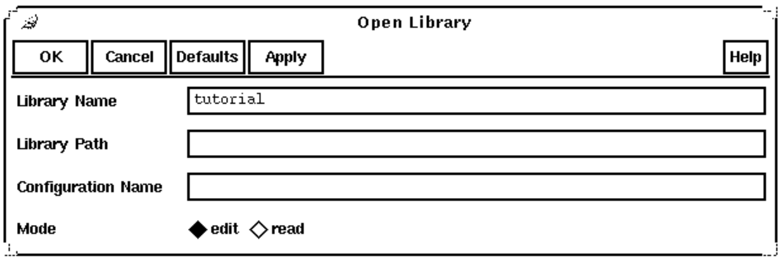
The schematic now appears ready for you to edit.

7.0 Graphical Editing Facilities

You will now be introduced to a few Graphical Editing Facilities such as Delete, Zoom in, Pan. This will be done with the aid of a schematic in the HOUSE library called ***play_block***. You will first need to create a library in which to place the schematic and then copy the schematic in the reference library to the library you have just created.

7.1 Click on ***Open->Library*** on the CIW menu banner.

7.2 Fill the ***Library Name*** with ***tutorial.***



7.3 Click on ***OK***.

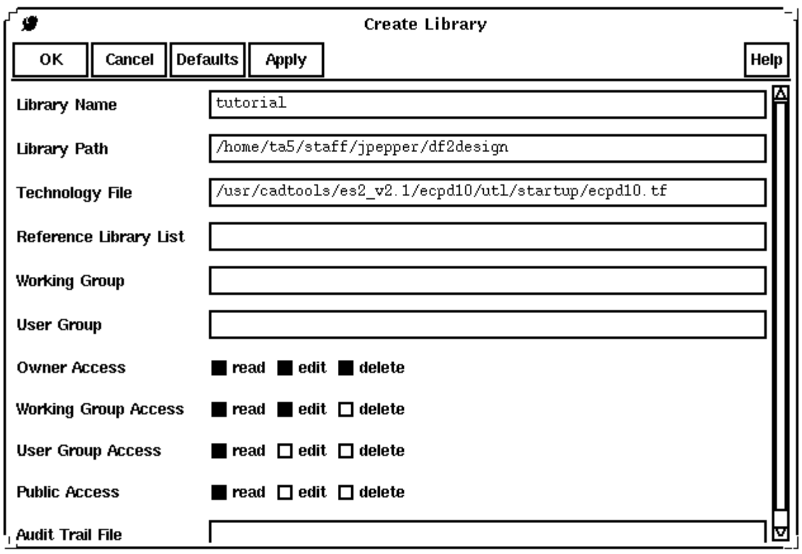
A dialog box appears with the following message:

'Library tutorial was not found

Do you want to create it?'

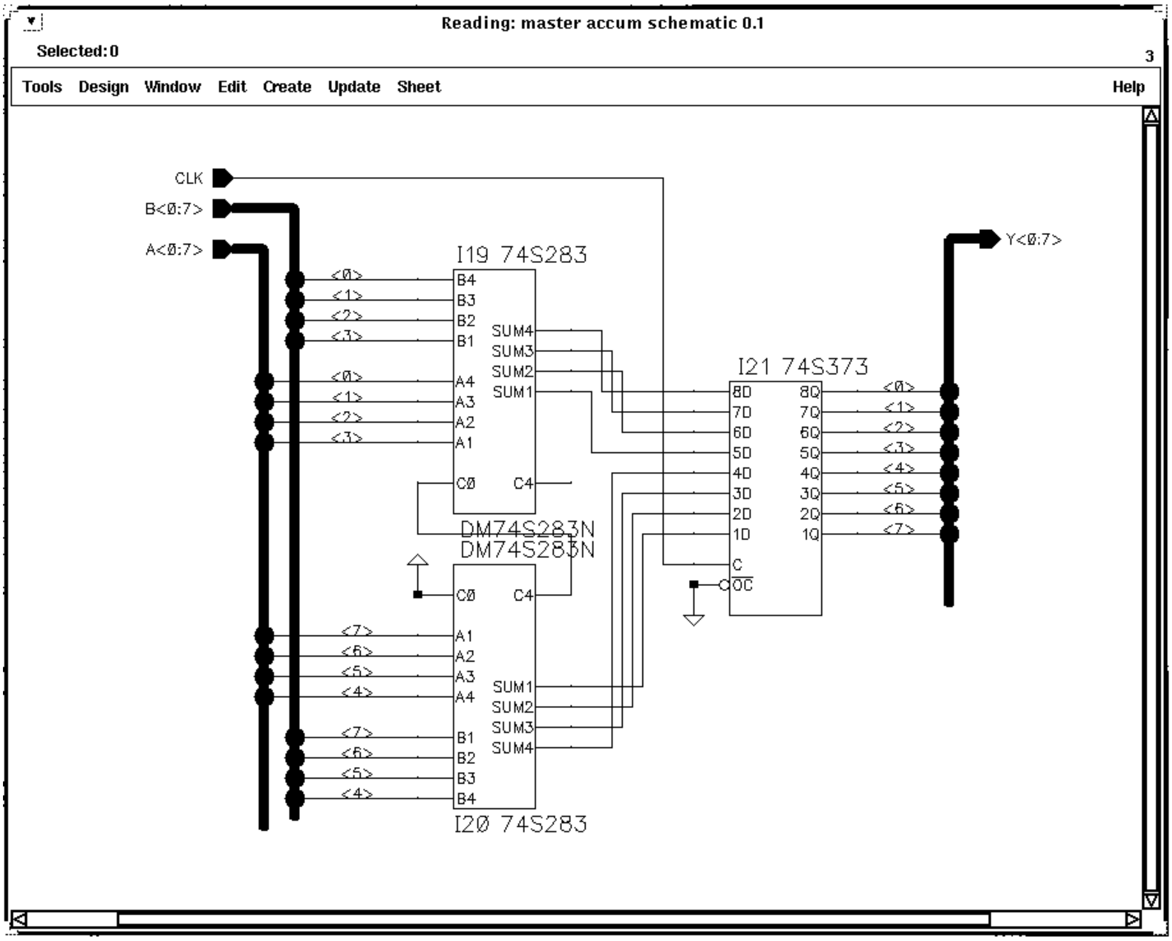
7.4 Click on ***OK*** in the dialog box.

7.5 Another form appears - accept the values entered by clicking on ***OK.***



6.9 Click on ***OK*** on the second Descend form, as schematic is the only option.

The schematic for the accumulator now appears. Use zoom to look at the schematic in more detail.



6.10 To return to the ***top_level*** schematic move the cursor onto the schematic background, outside the accumulator design.

6.11 Hold down the middle mouse button to reveal the pop up menu.

6.12 Slide to ***Return***.

6.13 Release the mouse button.

The top_level schematic now reappears.

6.14 Press ***f*** to select the ***Fit*** command. The fit command fits the schematic to the window displaying the whole schematic.

6.15 Click on ***Window->Close*** from the schematic banner menu to close the schematic window.

6.0 Browsing the Schematic Hierarchy

The ***top_level*** schematic is the top schematic in a hierarchy. In this section you will learn how to descend the hierarchy to see the schematics for the symbols and also learn how to zoom in and out to view areas of the schematic.

6.1 Move the cursor onto the schematic background, outside the design.

6.2 Hold down the middle mouse button to pop up the schematic menu.

6.3 Slide the mouse down to ***Zoom in***.

6.4 Release the middle mouse button.

At this point a prompt appears in the CIW, asking you to Enter the first corner of the box you wish to enlarge.

6.5 Click on one corner of the area to enlarge.

6.6 Move the mouse to the opposite corner - as you do a bounding box will move with the mouse, then click to zoom in.

Now a zoomed in portion of the schematic appears.

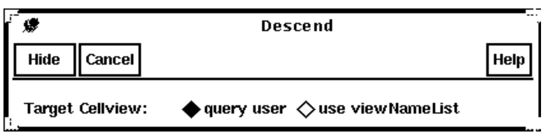
Note: There is also a bind key which will let you zoom in. This key is the letter *z*. By pressing the key you miss out 6.1 to 6.4 and just do 6.5 and 6.6. To redisplay the whole design you may also use the ***fit*** command. This has a bind key ***f.*** Press f to redisplay the whole design.

Now follow the steps below to descend the hierarchy and to view the schematic of the accumulator in the design.

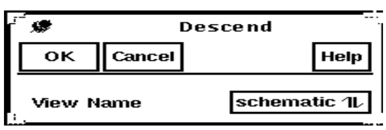
6.7 Click on ***Design->Hierarchy->Descend Read*** from the schematic menu banner.

A message now appears in the CIW asking you to Point at the instance to descend into.

The first descend form also appears. Ignore this form for now (It lets you choose how the software decides which cellview to descend into).



6.8 Move the cursor onto the ***accum*** instance and click. Now the second descend form appears. This form lets you choose which cellview to descend into.



5.0 Displaying the top_level Schematic

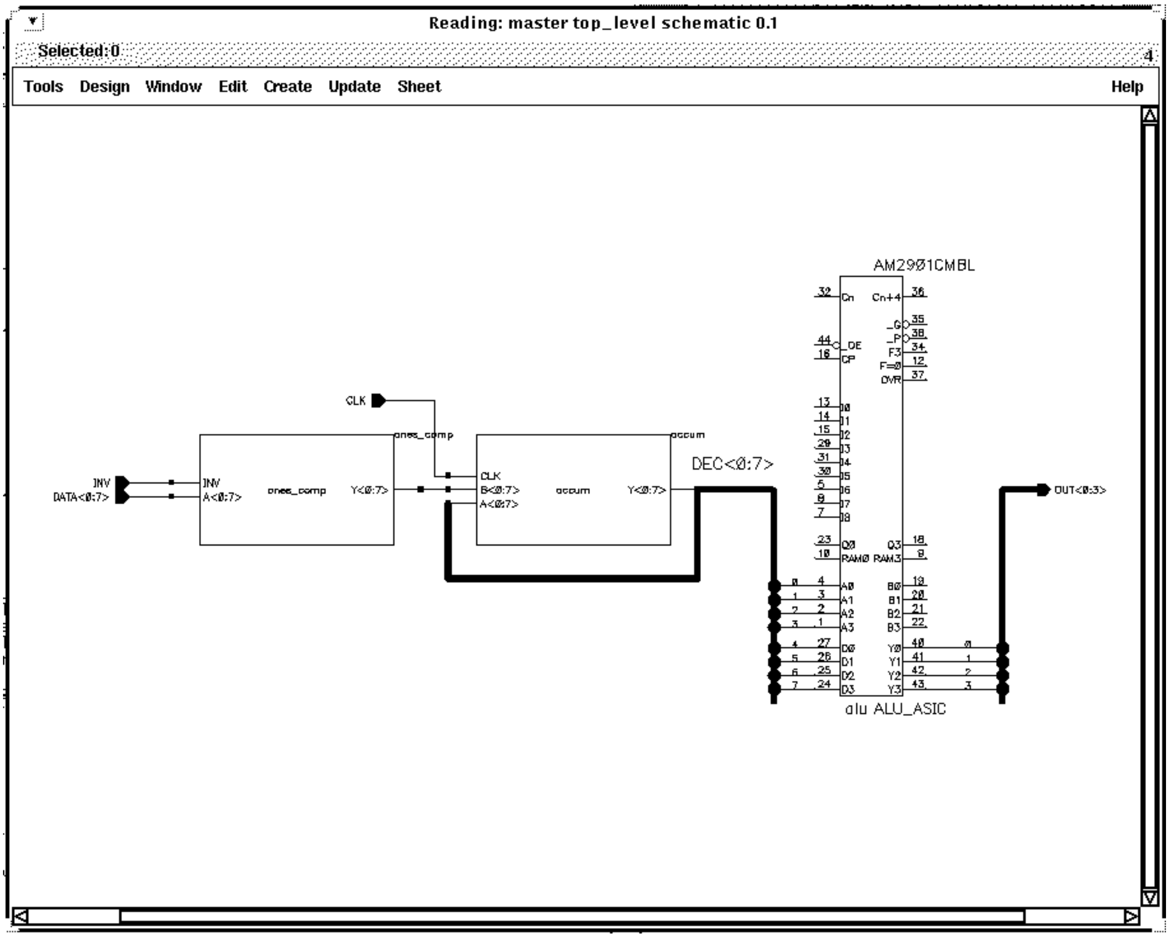
You can open a cellview in two ways. The first is from the CIW and the second is from the Library Browser.

5.1 From the Library Browser move the cursor onto version ***0.1*** of the top_level schematic.

5.2 Hold down the middle mouse button to pop up the version_level menu.

5.3 Slide the mouse down to the ***Read*** option (The master library is read only).

5.4 Release the mouse button.

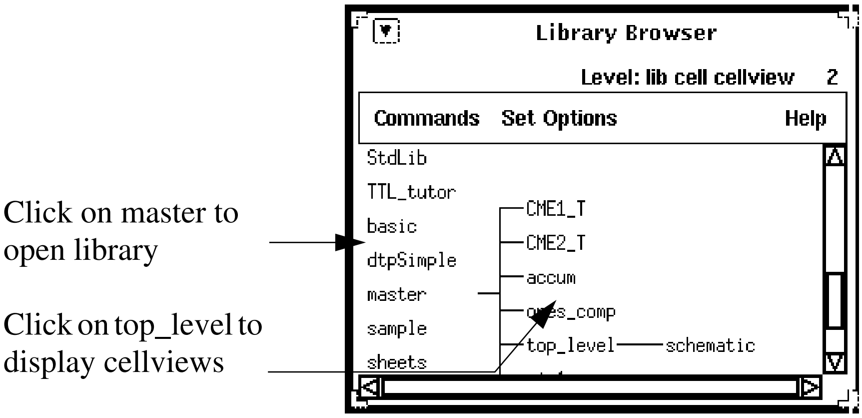


The schematic window opensshowing the ***top_level*** schematic. The name of the schematic appears in the top of the schematic window. The names of the pull down menus appear in the Menu Banner.

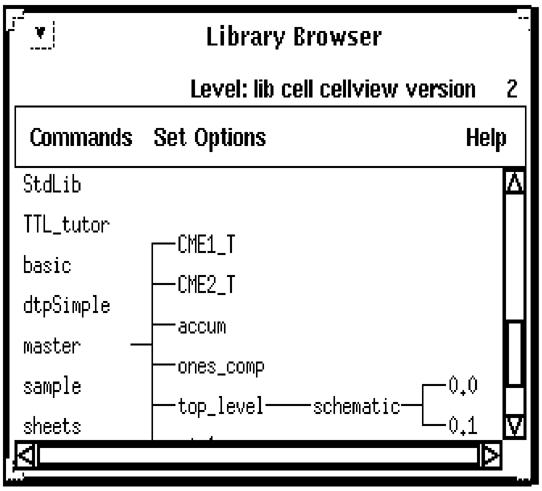
4.2 You may now display the contents of these different libraries. Click on ***master*** on the Library Browser to open the ***master*** library.You can now see the individual cells in the master library. A cell represents a particular unit of design.

4.3 Then click on ***top_level*** to display the cellviews for the ***top_level*** design.

A cellview is a specific representation of a cell, for example a schematic or symbol.



4.4 If you now click on ***schematic***, the Browser will list the versions that the schematic has. You can use the versions to save the design at appropriate milestones.



3.0 Exiting the Cadence Software

You can exit Cadence at any time, regardless of where you are in your work. If you have any work that is not saved, then it will prompt you to save it before you quit.

3.1 Click on ***Open*** from the CIW banner menu to display the Open menu.

3.2 Click on ***Quit*** to exit the software.

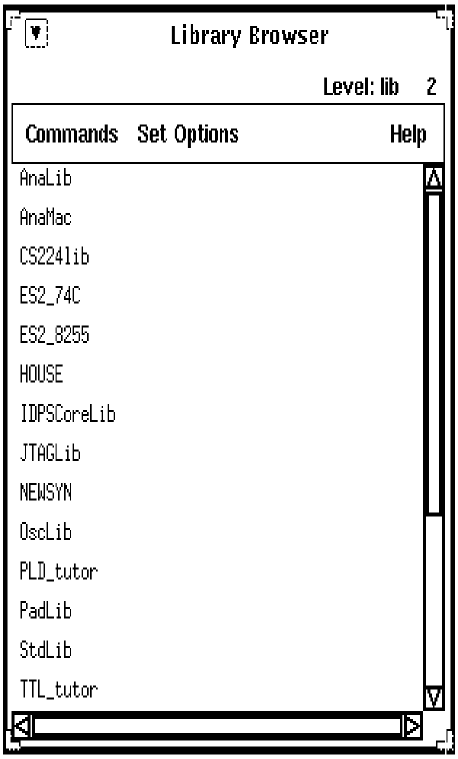
3.3 Click on ***OK*** in the dialog box to confirm that you want to exit.

4.0 Opening the Library Browser

The Library Browser displays design libraries and their contents.You may also access many of the commands for working on the designs.

4.1 In the CIW, click on ***Design Manager->Library Browser*** to open the Library Browser.

Note: Notice the three dots (...) following ***Library Browser*** on the menu. Any command that is followed by three dots displays a form as part of the command.



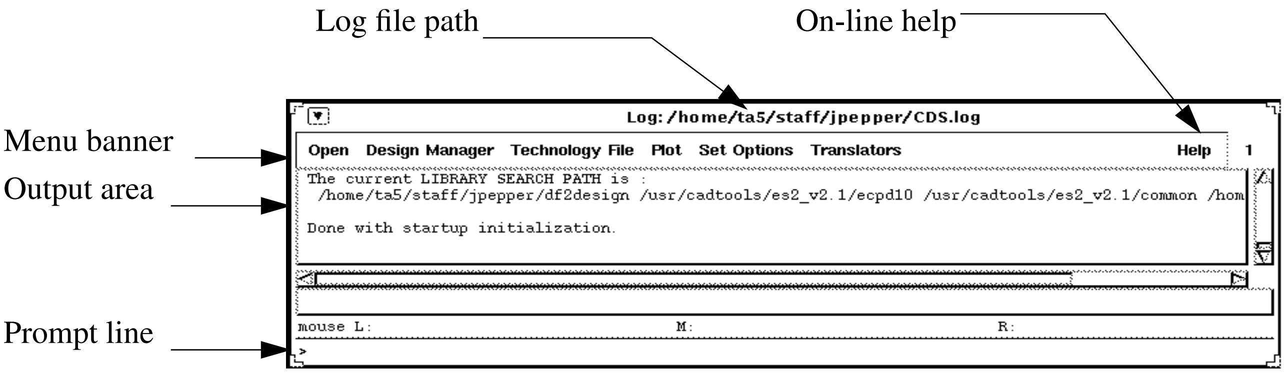
2.0 Starting the Cadence Software

You can start the cadence software by typing ***cdsgo*** in a window.

The software might take a few minutes to load. Once it has finished loading, the Command Interpreter Window (CIW) appears with the message:

‘Done with startup initialization’

The CIW is the main control window for the Cadence software.The following figure points out the parts of the CIW:



Log file path: This displays the path to the log file that records the current session.

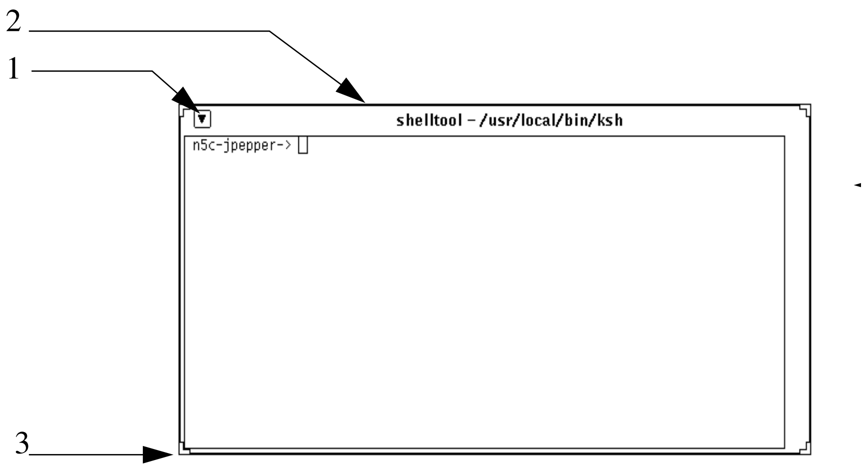
Menu Banner: Used to pull down command menus from the menu banner across the top of the CIW.

Output area: This displays a running history of the commands you execute and their results.

Prompt line: This prompts you for the next step in carrying out a command.

On-line help: Click on help in the menu banner to view the DF2 manuals and local documents. You will have to wait for FrameMaker to load when you call ‘Help’.

The following diagram illustrates the main features of the OLWM windows.

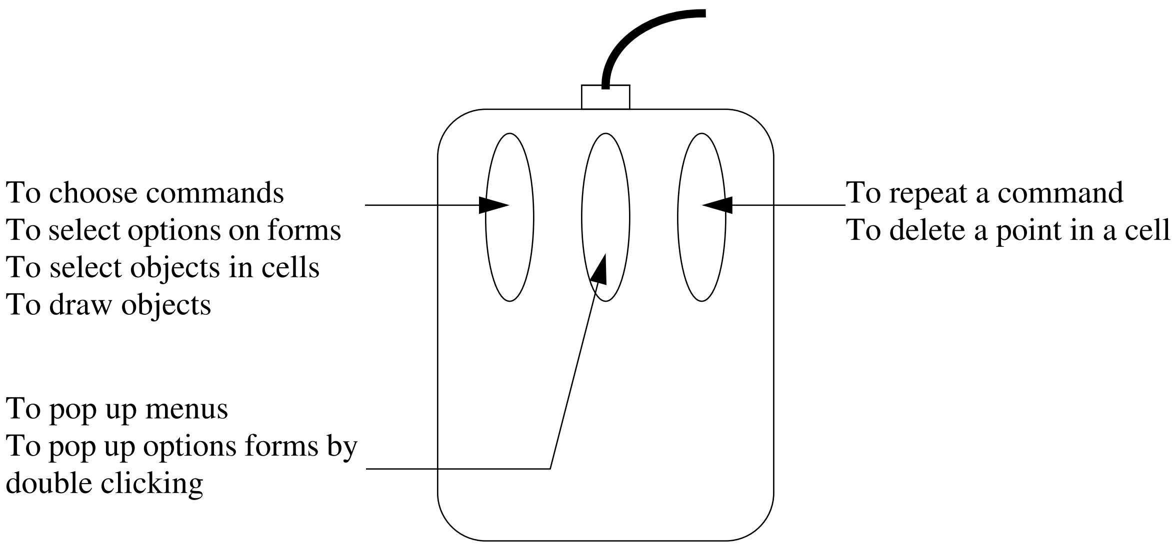


1: Place the cursor on here and hold the right button down to reveal a menu with options such as ***Quit*** and ***Move.*** Or click the left mouse button to Iconify the window.

2: Place the cursor inside the title bar and hold the left mouse button down to move the window.

3: Move the cursor to touch a corner, then hold down the left mouse button to resize the window.

1.4 Mouse Functions.



Getting Started

1.0 Introduction

1.1 Documentation

This document gives an introduction to the Cadence Design Framework II CAD system (referred to hereafter as DF2). It describes the commands and menu systems and how to access designs and libraries. It is one of a set of documents prepared for the use of students in the Department of Computer Science of the University of Manchester. The use of specific tools, design entry, simulation, chip layout etc. is described in other documents. Those currently available are:

- DF2 Schematic Design Entry.
- Register Level Design using Verilog Functional modelling and the Verilog_XL simulator.
- Gate Level Design using library substitution and HDL synthesis.
- Chip layout.

A set of manuals is provided with DF2; hardcopy versions can be viewed on request, but they can be viewed on-screen as FrameMaker documents as described below in section 2.

1.2 Before Starting

You should ensure that you have the correct paths set and initialisation files. Please refer to the Laboratory noticeboard or to $CS224/README.doc

1.3 Windows

DF2 is designed to be run with the Motif Window Management (MWM) system or OpenLook Window Management (OLWM) system. All the documentation shows the use of the (OLWM) system.

To use Open-Look:

1.3.1 When logging on, choose ***Openwindows - o***.

1.3.2 Then choose ***Xnews Server - n***.