

Chapter 5

Simulation of a Schematic Design

The topic of this chapter is on how to verify a schematic design by functional simulation. The tool used for this are called *Affirma Analog Artist* (trademark of **Cadence Design Systems, Inc**).

The chapter starts by describing how to move from the schematic to the simulation environment, how to perform the simulation, and finally how to view and analyze the results.

More information about simulation and the tools to display the results can be found in the manuals *Affirma Analog Artist Circuit Design Environment User Guide*, *Analog Waveform User Guide*, and *Waveform Calculator User Guide* from the on-line set.

5.1 Preparation of the Schematic

Before the schematic drawing can be simulated some preparation is needed. First, the schematic should pass a check (sec. 4.1.6), to verify that all the terminals are connected and no short circuits exists.

Second a test-bench has to be created. A schematic should only consist of the components that is to exist on the final layout of the design. Therefore, a test-bench is created to contain the other components necessary for simulation. A test-bench, which also is a schematic view, for an inverter is shown in fig. 5.1. From the left it consists of a power source (**vdc**) which sets a voltage difference of 3 volts between the global vdd and gnd symbols **on all levels of the hierarchy**. A signal source (**vpulse**) is used to generate input to the inverter, which is included as an instance. Finally the output is loaded down with a capacitance.

The supply and power sources are picked from the library **analogLib** and the most

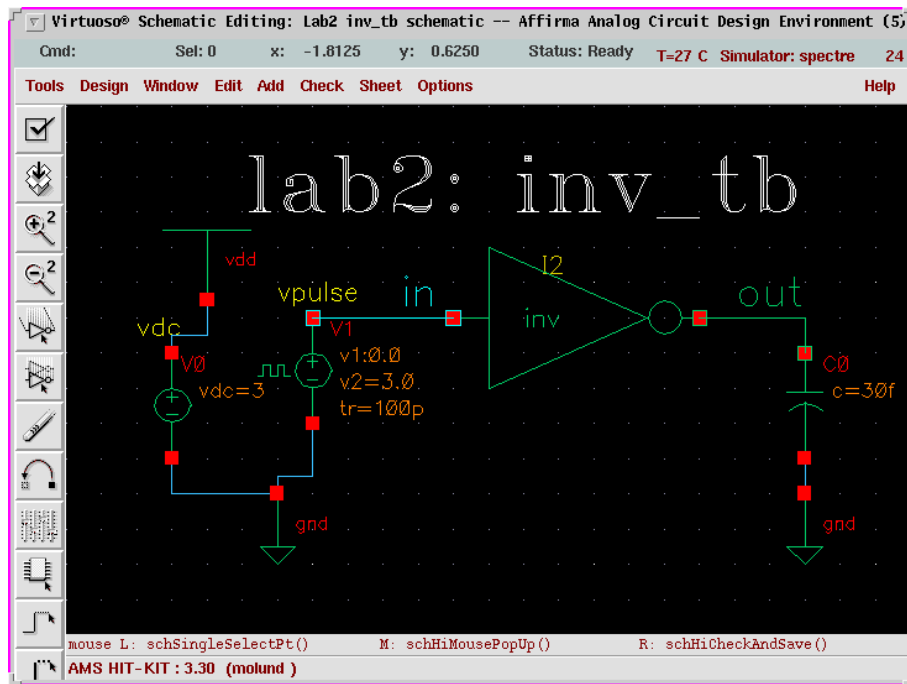


Figure 5.1: A test-bench for an inverter.

common are:

vdd, gnd: Global power connections between all levels in the hierarchy.

vdc: Power source. Parameter for voltage.

vpulse: Pulsed signal. Settings possible for voltage, delay, rise- and fall time, etc.

cap: Gives a capacitance of selected value.

5.2 Starting the Simulation Environment

The environment is started directly from the schematic editor by the command *Tools > Analog Environment* which will cause the controlling window, **Affirma Analog Circuit Design Environment**, of the simulator to materialize.

The main simulation window contains four fields with information regarding the current simulation, circuit designation, a list of the type of analyses chosen, the variables - if any, and nodes selected to be saved.

As shown in figure 5.2 the cell that is to be simulated is already filled in and the default simulator can be seen in the status banner. If it isn't **spectre** it can be

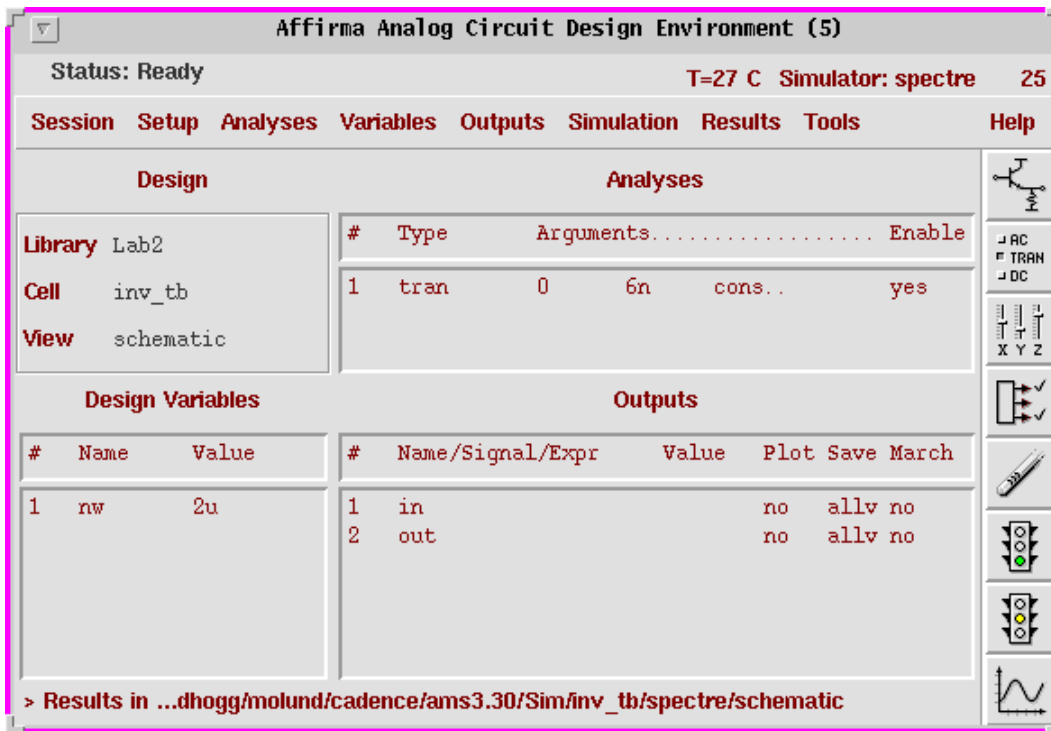


Figure 5.2: The simulator main window.

changed in the *Setup* menu. In this menu there are commands for changing the environment, what is to be simulated, where the model files are located, etc.

Temporary and result files, needed during the simulation, are stored in the unix directory `./Sim`, which is in the users own file system. During large circuit simulations this can grow tremendously in size and cause quota problems. To avoid that problem this directory can be changed at **Project Directory**, in the form gotten by *Setup* > *Simulator/Directory/Host*, to `/tmp/<user>/sim` for instance. This library can later be removed with no permanent loss since all data therein will be recreated during the next simulation.

In the *Analyses* menu are commands to select the kind of simulation to be performed, AC, DC sweep, transient, noise analysis, etc.

From *Variables*, some user defined variables for component properties can be handled. More in section 5.2.6.

The node currents and voltages to be analyzed are controlled by commands in the *Outputs* menu. Data can be observed interactively during the simulation or saved to disk for further study.

The most important commands in *Simulation* are *Netlist and Run* and *Stop*. Some

specification of initial conditions can also be made here.

Results controls all data management after the simulation. The menu is used to select what node currents or voltages are to be displayed in a waveform window.

From the *Tools* menu it is possible to start a calculator to perform further analysis of the simulation data. Here a *Parametric Analysis* can also be started. This means that a circuit's behaviour for a certain parameter is analyzed by sweeping the parameter over a defined interval of values. This is described in section 5.2.7.

5.2.1 Choosing Analysis Type

The analysis type is chosen by *Analyses > Choose*. In the resulting form the kind of simulation to be performed can be selected along with several options.

5.2.2 Node Selection and Management

There are two ways of presenting simulation data. The first is to plot the data in a waveform window continuously during the simulation. This way is useful for short simulations. The drawback is that the signals are only shown in the waveform window, it is not possible to do any further analysis of the data.

For a more complex circuit it is more convenient to save the simulation data to disk to be viewed after the simulation. Then it is also possible to perform operations on the curves shown.

Selecting Nodes

The command *Outputs > To Be Saved > Select On Schematic* in the simulator window activates selection mode. Thereafter nodes are selected by using the left mouse button.

Voltage levels are selected by clicking on the wire connections.

Currents are selected by choosing the nodes, i.e. the red boxes.

Selected wires are highlighted, nodes circled, and their names will be listed in the window.

By using *Design > Hierarchy > Descend Edit* in the schematic window it is possible to traverse the hierarchy and select nodes not visible from the top level.

In order to get the nodes automatically plotted after the simulation *Outputs > To Be Plotted > Select On Schematic* can be used. The last alternative is to have the

nodes plotted during the simulation which is activated by *Outputs > To Be Marched > Select On Schematic*.

Modifying Selected Nodes

The command *Outputs > Setup* gives a form in which the selected set of nodes can be altered. The selections of *Plot*, *Save*, or *Marched* can be changed. Nodes can be deleted and it is possible to define operations to be done on the node values.

Saving the Configuration

All the configuration done in the window can be saved and later loaded back into the simulator. This is controlled by *Session > Save State*. In the form the parts that is to be saved are selected and given a name to store it under. The files will end up in the directory **.artist_states** in the users home directory.

5.2.3 Starting a Simulation

The simulation run is started by *Simulation > Netlist and Run* or by clicking on the green traffic light among the icons. The simulator uses its own log window to report on events and errors.

A running simulation can be halted by *Simulation > Stop*. It is not possible to restart a halted simulation, it must be run from the start each time.

5.2.4 The Waveform Window

After the simulation, the signals for which **Plot** was chosen will be visible in a waveform window, like the one in fig. 5.3. Other nodes are selected by *Results > Direct Plot > Transient Signal* - if it was a transient simulation - and selecting them from the schematic, as instructed.

When the window appears all curves are plotted together. It is possible to separate them by *Axes > To Strip, ... > To Composite* brings them back together. The curveforms can be moved among each other by clicking and dragging on them.

It is possible to zoom among the curves presented and the cursor tracks the closest curve with the values visible in the status banner.

By default the window is recreated after each simulation. In order to add a curve to an existing window without causing a new one to appear the option **Overlay Plots**, in the form given by *Results > Printing/Plotting Options*, must be selected.

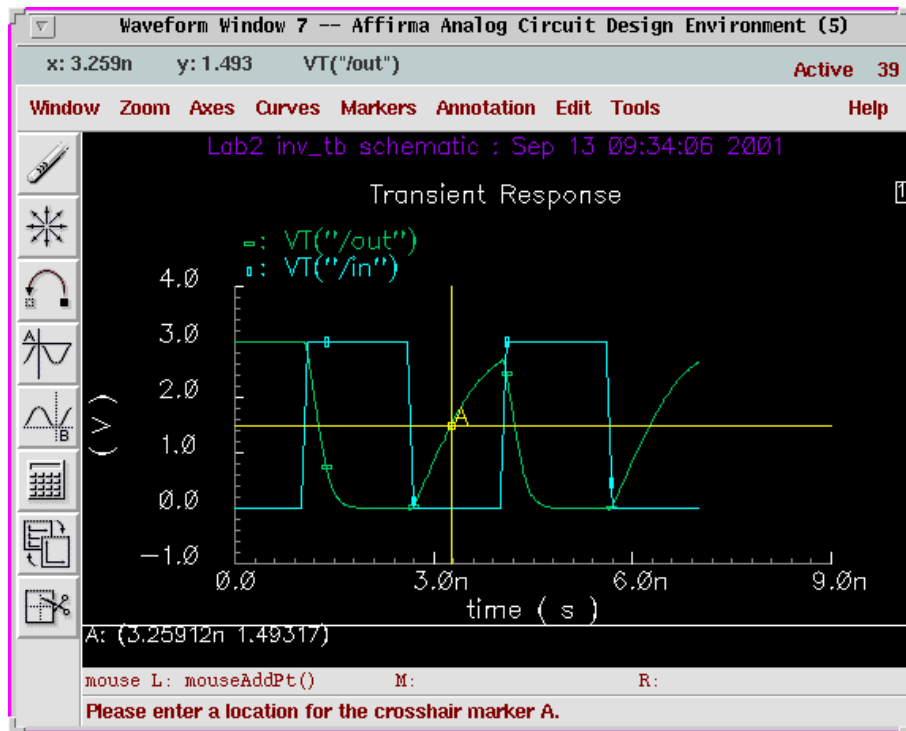


Figure 5.3: The Waveform Window.

To improve documentation text strings can be added to the waveform window. It is very useful for comments etc. *Annotation > Edit* gives a form that handles this. After entering suitable text, size, and colour a click on the **Add** button will enable the string to be placed in the window. The title and subtitle of the window can also be modified from the *Annotation* menu.

To get a plot of the waveform window there is a form activated by *Window > Hardcopy* which looks much like the one from the schematic window.

5.2.5 DC Operating Points

The DC operating points for transistors are often interesting to view. Given that a DC simulation has been run they can be viewed by *Results > Print > DC Operating Points* and selecting the transistor to be presented. The result will be presented in a new window from which the results can be saved.

5.2.6 Using Variables

By replacing component property parameters with variables, instead of specific values, it is possible to directly change the values from the simulator. Otherwise the schematic would have to be modified and saved before the new parameters are visible to the simulator.

In the properties form (*Edit > Properties > Objects (q)*) a variable is written in place of a value for a parameter. “**nwidth M**” for the width of the n-transistor and “**2.2*nwidth M**” for the width of the p-transistor in an inverter design for example. By setting a value on the **nwidth** variable from the simulator it is possible to quickly change the strength of the inverter.

The variables are read into the simulator from the form **Edit Design Variables** started by *Variables > Edit*. The button **Copy From** will include all variables from the schematic into the simulator. After editing the variables the simulator can be started directly from the variables form by clicking **Apply & Run Simulation**.

5.2.7 Parametric Analysis

By a parametric analysis several simulations are run after each other while a component parameter is varied through an interval. The procedure is related to the analysis with variables but the process of setting a new value on the variable has been automated.

The component parameter must be a variable for a parametric analysis.

From *Tools > Parametric Analysis* the **Parametric Analysis** form is started. The name of the variable to be swept are to be filled in at **Variable Name** and in the lower field the limits of the interval are entered.

As an example the value of the load capacitor in fig. 5.1 can be a variable that is then swept through several values. The result is presented as a set of curves in the waveform window as in fig. 5.4.

5.2.8 Shutting Down the Simulation Environment

When all simulations and analyses are run the environment is closed down by *Session > Quit* from the main simulation window. This will close down all associated windows in an orderly manner.

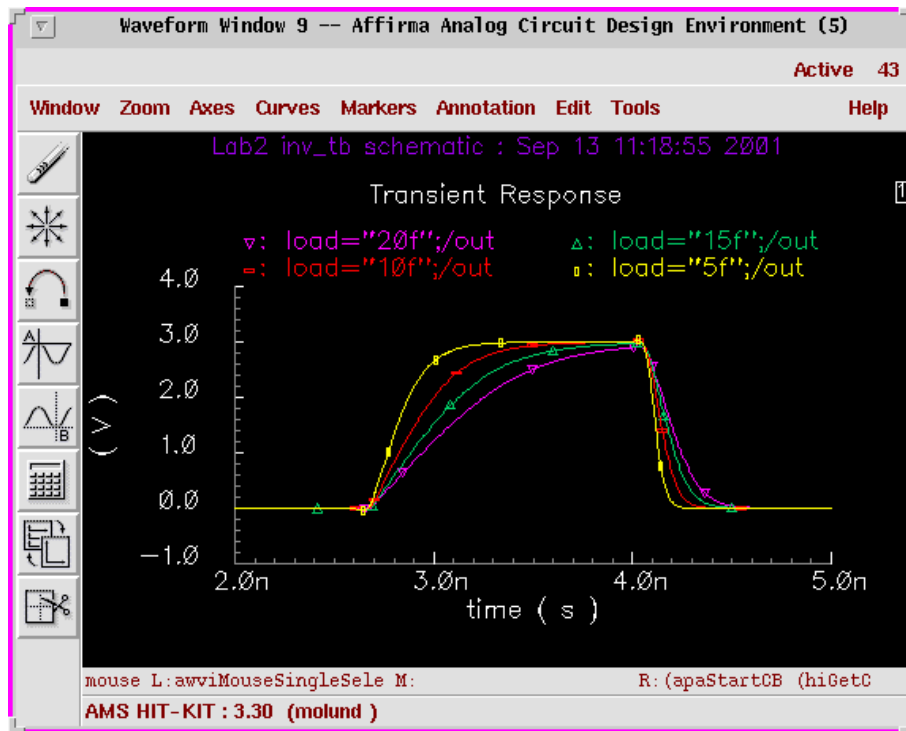


Figure 5.4: Results of a Parametric Analysis.

5.3 The Waveform Calculator

The *Waveform Calculator* is a tool for performing advanced calculations on the simulation data. Node values can be entered from the schematic view or from the waveform window. The calculator is started by *Tools > Calculator*.

The calculator works with *Reverse Polish Notation (RPN)* which means that the operands are entered first followed by the operation to be performed on them. This mode can be changed by *Options > Set Algebraic*.

5.3.1 Entering Data

There are several buttons for entering simulation data to the calculator depending on the simulation type that generated them.

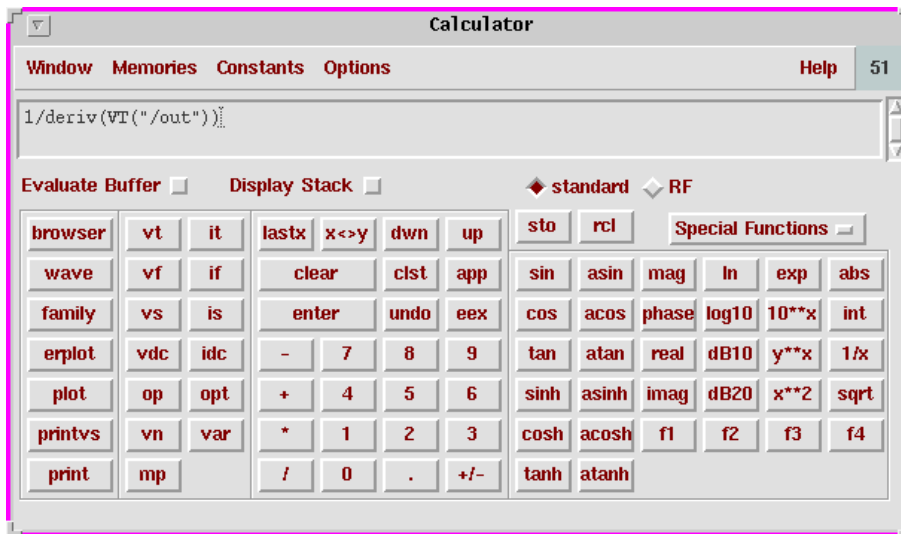


Figure 5.5: The Waveform Calculator.

vt	transient voltage	it	transient current
vf	frequency voltage	if	frequency current
vs	sweep voltage	is	sweep current
vdc	DC voltage	op	DC operating point
vn	noise voltage	opt	transient op. point
var	design variable	mp	model parameter

The data are entered by first clicking on the appropriate button in the calculator followed by selecting the wire (voltage) or node (current) of interest in the **schematic** editor window. For example, in figure 5.1 **vt** followed by selecting the output wire labeled “out” would cause **VT("/out")** to appear in the calculator window.

By using the button **wave** it is possible to enter a curveform from the waveform window.

5.3.2 Waveform Analysis

After loading a curve into the calculator several operations can be executed on it by the buttons on the calculator, inverted, squared, multiplied by a factor, etc. It is also possible to add or subtract several curves. The result of the expression shown can be viewed by pressing **Plot** which will cause the resulting curve to be drawn in the waveform window. **Print** is used for operations resulting in a value rather than a curve.

Special Functions

There is a **cyclic field** named **Special Functions** in the calculator window. This contain several functions of interest when analyzing a circuit, derivating, integrating, rise- and falltime, slewrate, etc. Figure 5.6 shows the windows involved when calculating the rise time of a signal.

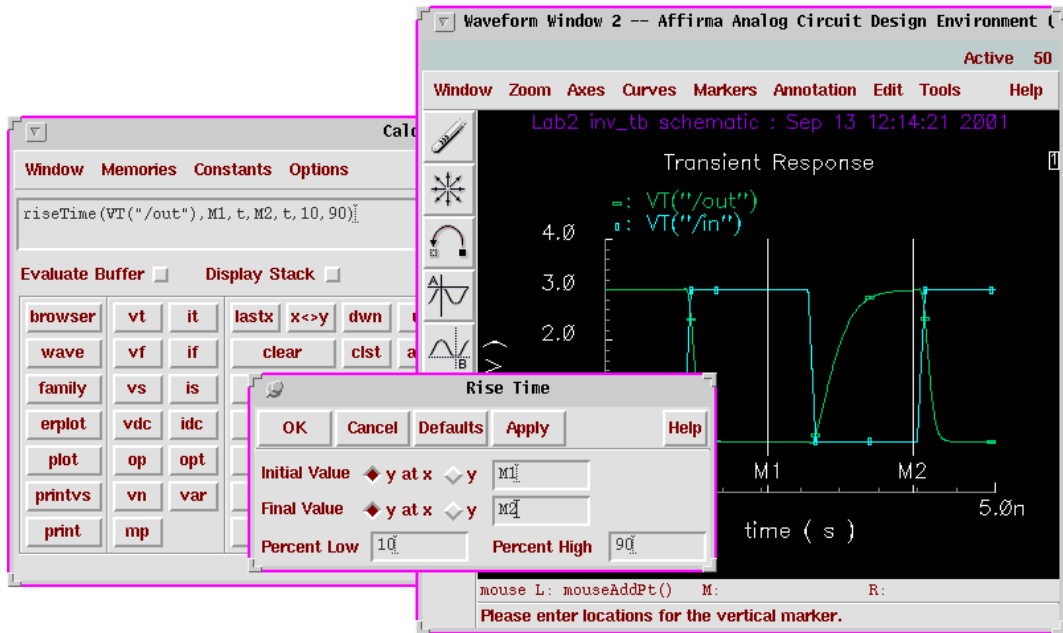


Figure 5.6: Calculation of rise time of an output.

The function **riseTime**, like many others, needs an interval in which to do its calculations. These intervals are easiest defined by setting **Markers** in the waveform window. By *Markers > Vertical Marker* and **Add Graphically** it is possible to define two points in time (**M1** and **M2**) that encompass the interesting rising flank. In the **Rise Time** form this markers can then be used. By clicking **Apply** the calculator window changes to that of fig. 5.6. Finally **Print** will cause the value to appear in a window.

A very useful command in the **Markers** form is **Display Intercept Data** which will give a text window with information of existing markers.