Mentor Graphic's ModelSim-SE tool: Simplified Usage flow

1. Start ModelSim with one of the fallowing:
for Linux at the shell prompt:
vsim
for windows-your option-from a windows shortcut icon, from the start menu, or from a DOS prompt: Modelsim.exe

2. Create a Project
select file > New > project from the ModelSim Main window

3. **Add VHDL/Verilog file to Project.**
Add file > [browse to directory] > [select files]

4. Edit/Compilation

Compile the Files into new library by selecting the **compile** button on the tool bar (Prompt: vcom filename)
Complete the compilation by selecting file from the file list and clicking **compile**.
Select **Done** when you are finished.
You can compile multiple files in one session from the file list. Individually select and compile the files in the order required by your design.

In order to edit the VHDL file, double click on the file name

Warnings:
?: no compiled version file
X: try to compile it, but failed
√: compiled and ready for simulation

5. Load the design unit. Select the **Load Design** button from the tool bar (Prompt: vsim file name)

The Load design dialog box lets you select the library and top-level design unit to simulate. You can also select the resolution limit for this simulation.

6. Select the **entity** and choose **Load** to accept these settings

7. Select **view> All** from the main window menu to open all ModelSim windows (prompt: view*)

8. From the signal window menu, select **view >List>Signals in Region**.
This command displays the top-level signals in the list window. (prompt: add list/filename/*)

9. Add top-level signals to the wave window by selecting **view>wave>signals** in Region from the signals window menu.
(prompt: add wave/filename/*)

Running the simulation:
Start the simulation by applying stimulus to the clock input

1. Click in the main window and enter the fallowing command at the VSIM prompt:

force clk 1 50, 0 100 –repeat 100
(Menu: signals>Edit>clock)

Modelsim interprets this **force** command as follows:

- force clk to the value 1 at 50 ns after the current time
- then to 0 at 100 ns after the current time
- repeat this cycle every 100 ns

2. Now you will exercise two different **Run** functions from the toolbar buttons on either main window or wave window. Select the **Run** button first. When the run is complete, select **Run All**.

Run: This causes the simulation to run and then stop after 100ns
(prompt: run 100) (Menu: Run>Run 100 ns)
Run-all: This causes the simulator simulator run forever. To stop the run, go on to the next step.
(prompt: run -all) (Menu: Run>Run -All)

3. Select the **Break** button on either **Main or Wave** toolbar to interrupt the run. The simulator will stop running as soon as it gets to an acceptable stopping point.

4. Select the **Continue Run** button to resume the run that you interrupted. ModelSim will hit the breakpoint, as shown by an arrow in the source window and by a Break message in the Main window. (prompt: run -continue) (Menu: Run > Continue)

5. Click the **Step** button to single-step through the simulation. Notice that the values change in the variables window. You can keep clicking **Step** if you wish. (prompt: run -step) (prompt: step)

6. When you're done, quit the simulator by entering the command:
quit -force

This command exits ModelSim without asking for confirmation
For more commands pls see the quick ref guide enclosed with this

Recommended reading:
Modelsim-SE Tutorial
Modelsim-SE User's manual.
Contact: S.D Thimmappa sdt@ee.iitb.ac.in