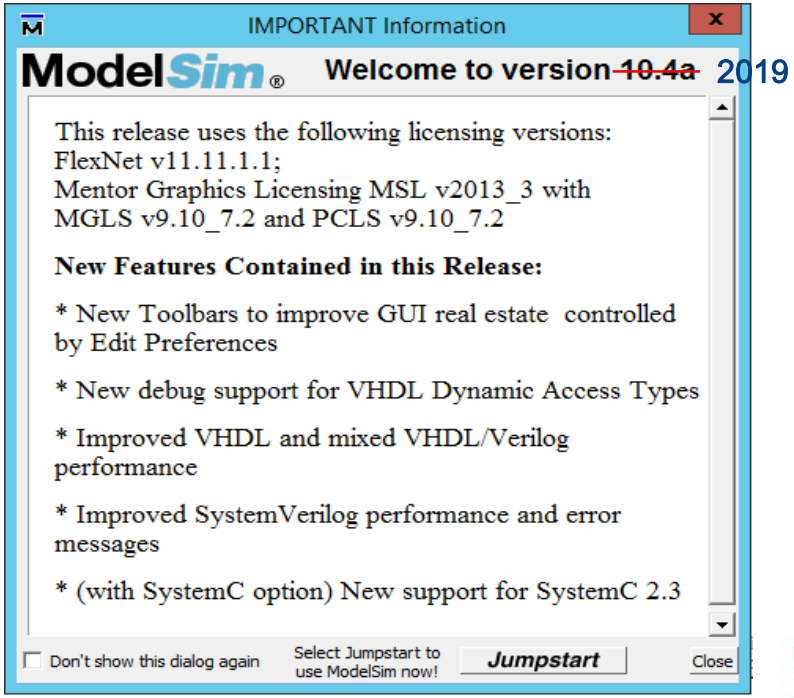
MICROELECTRONIC SYSTEMS DESIGN RESEARCH GROUP

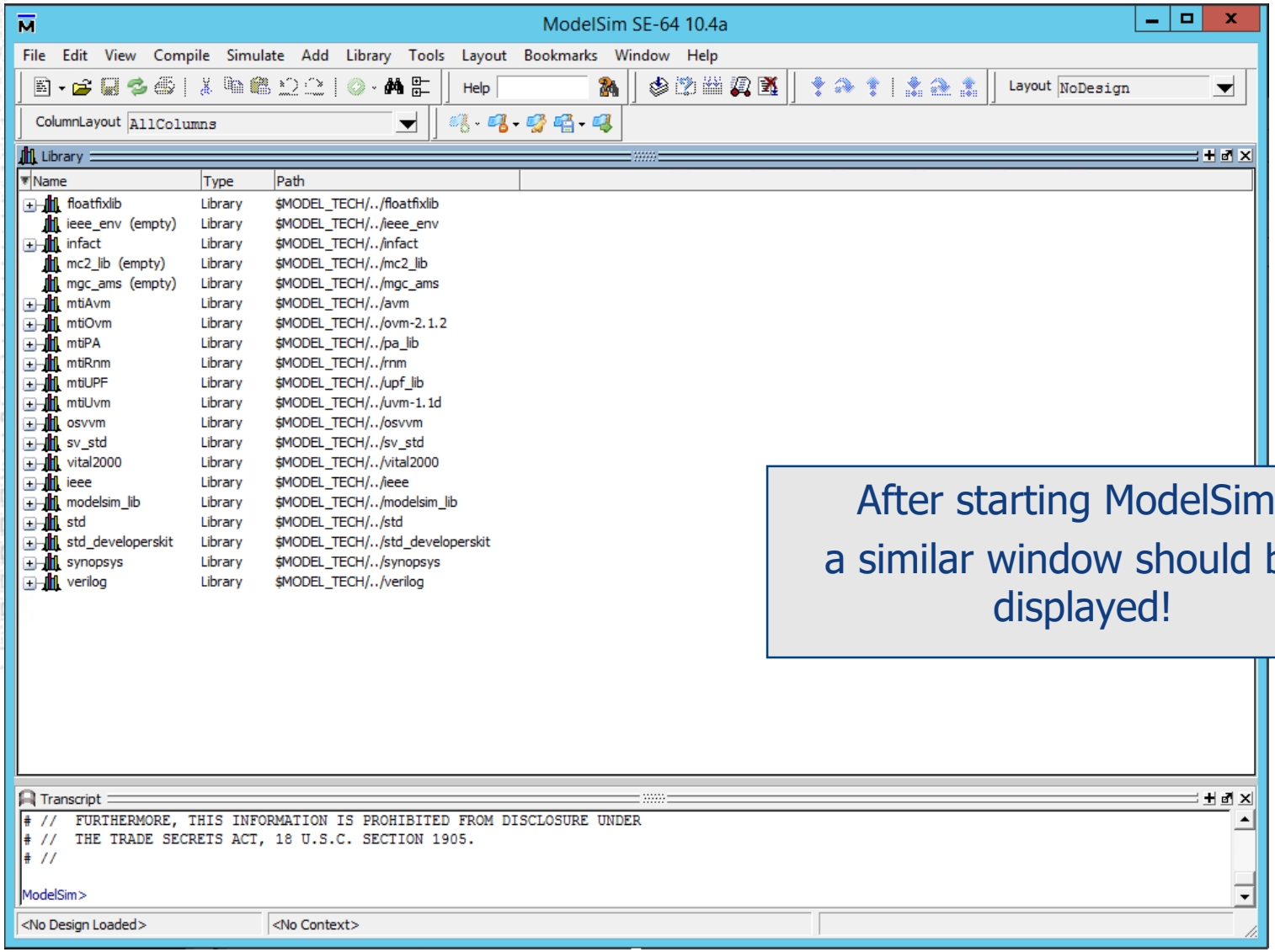
ModelSim Tutorial

Christian Weis, Uwe Wasenmüller, Hans Peter Goldhammer



TECHNISCHE UNIVERSITÄT KAISERSLAUTERN

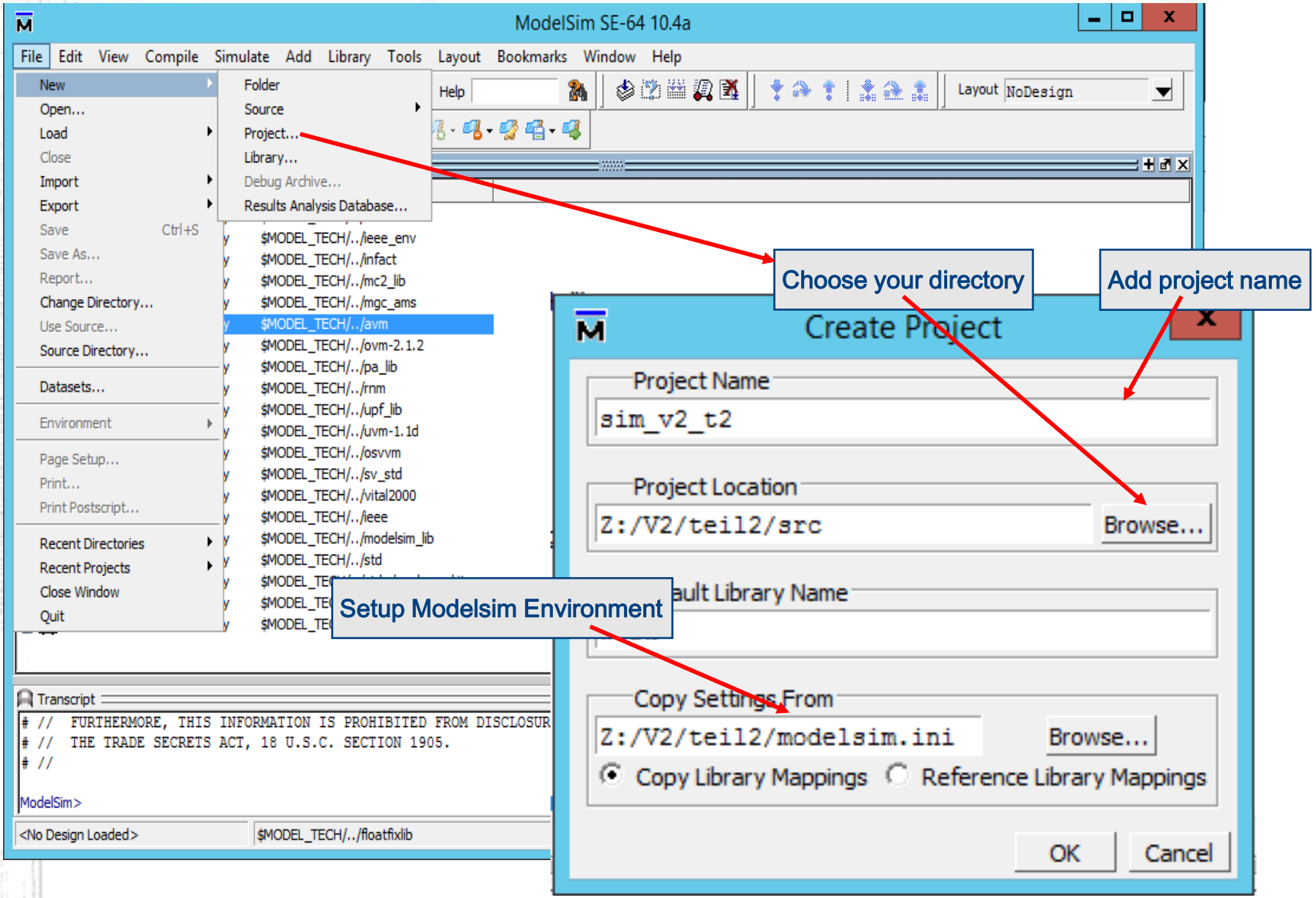
Getting Started



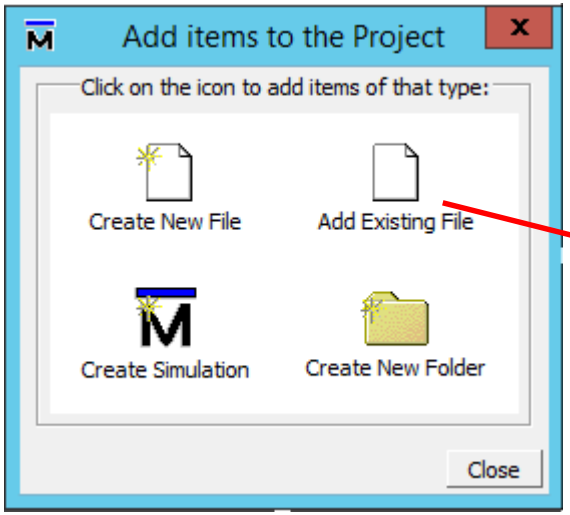
After starting ModelSim

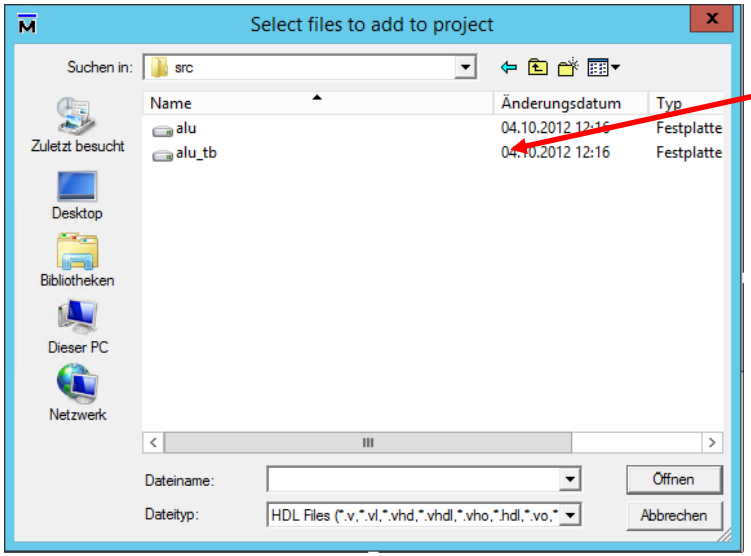
a similar window should be displayed!

Creating a New Project (Example Exercise 2)

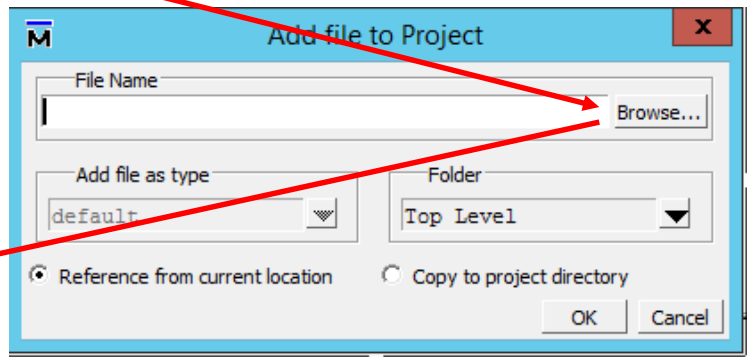


Adding Files to a Project





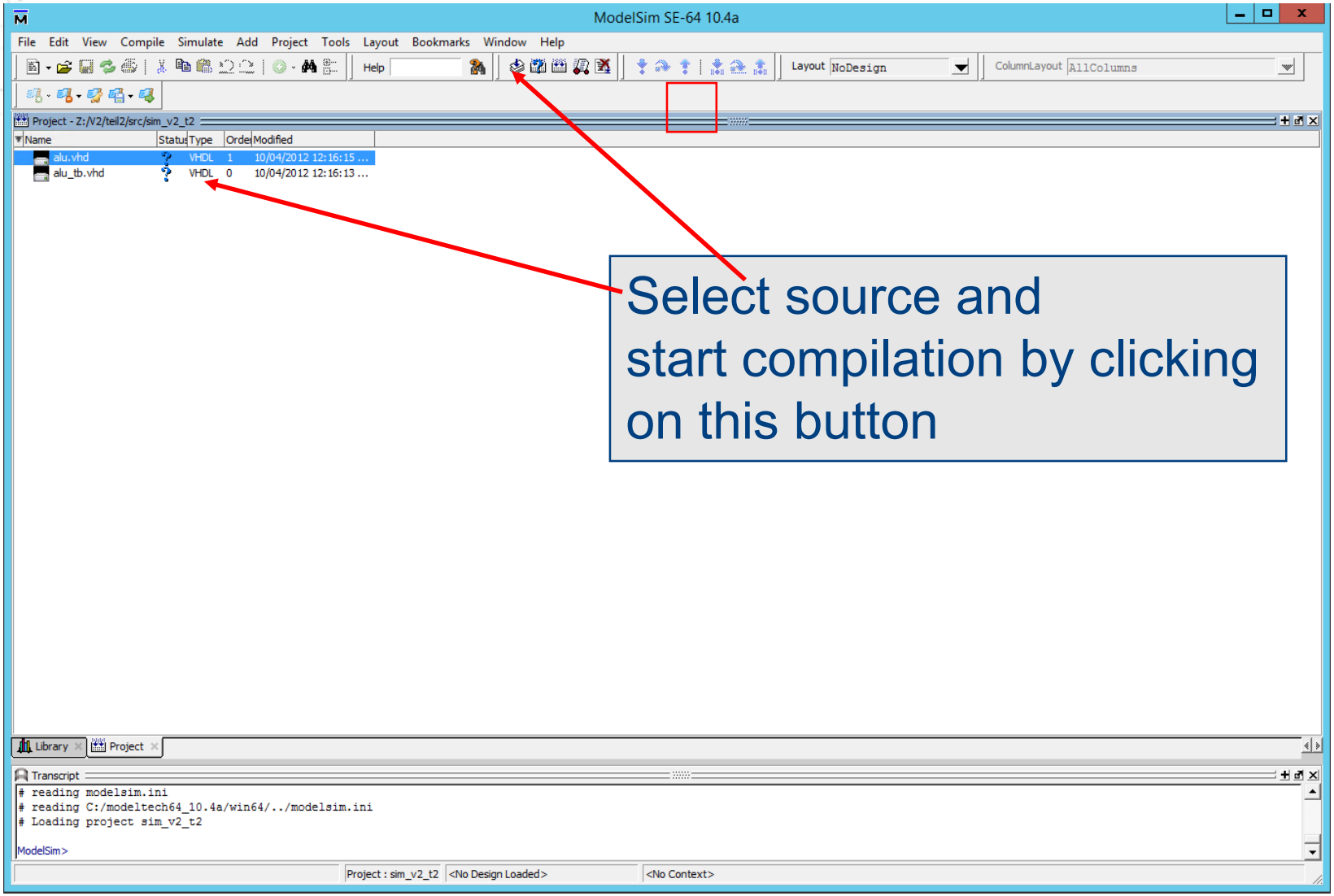
1.
Adding files by pressing the "Browse …" button - multiple selections possible



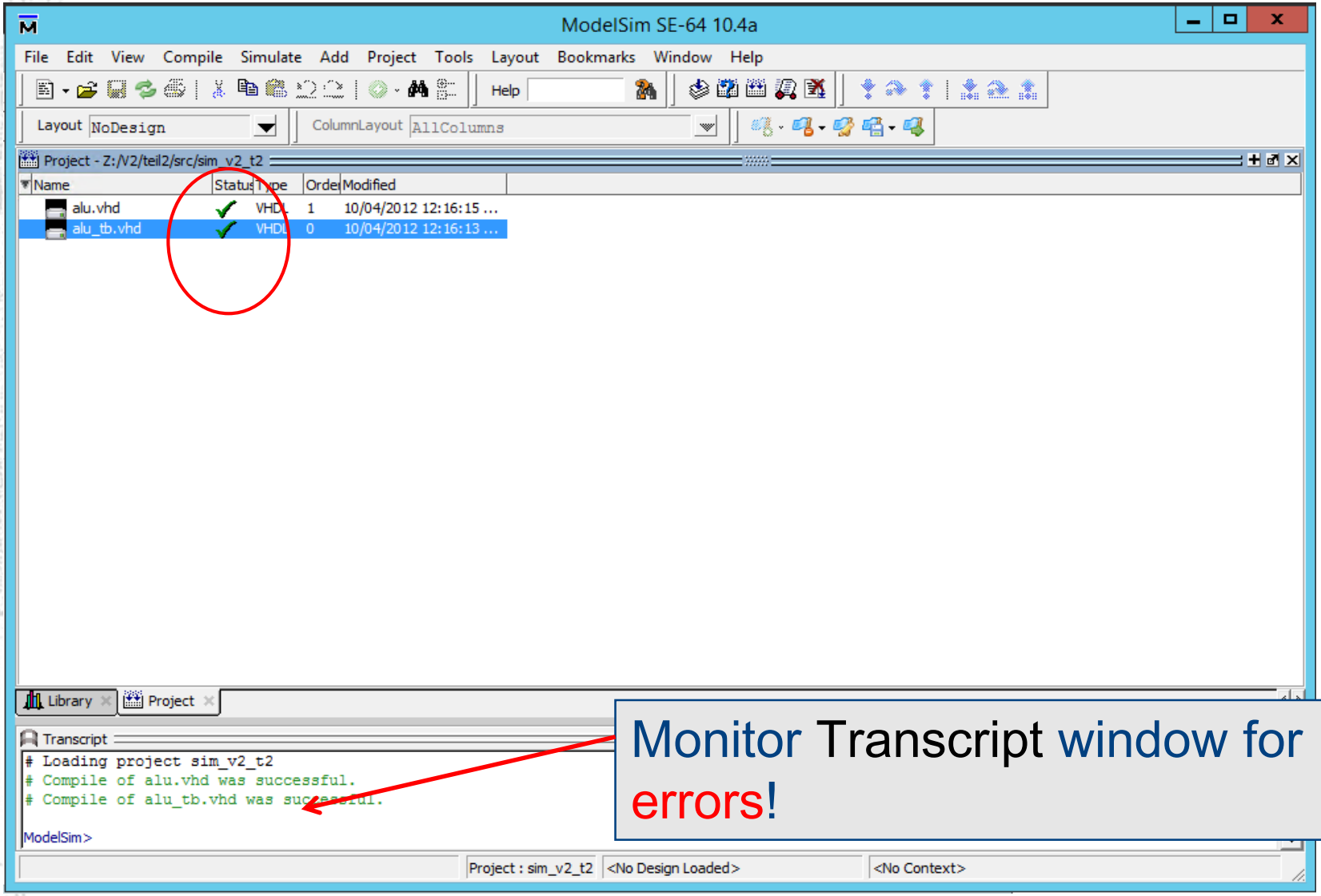
2.
Take care to add **all** required files to the project

3.
Later adding of files by
Menu: Project => Add to Project

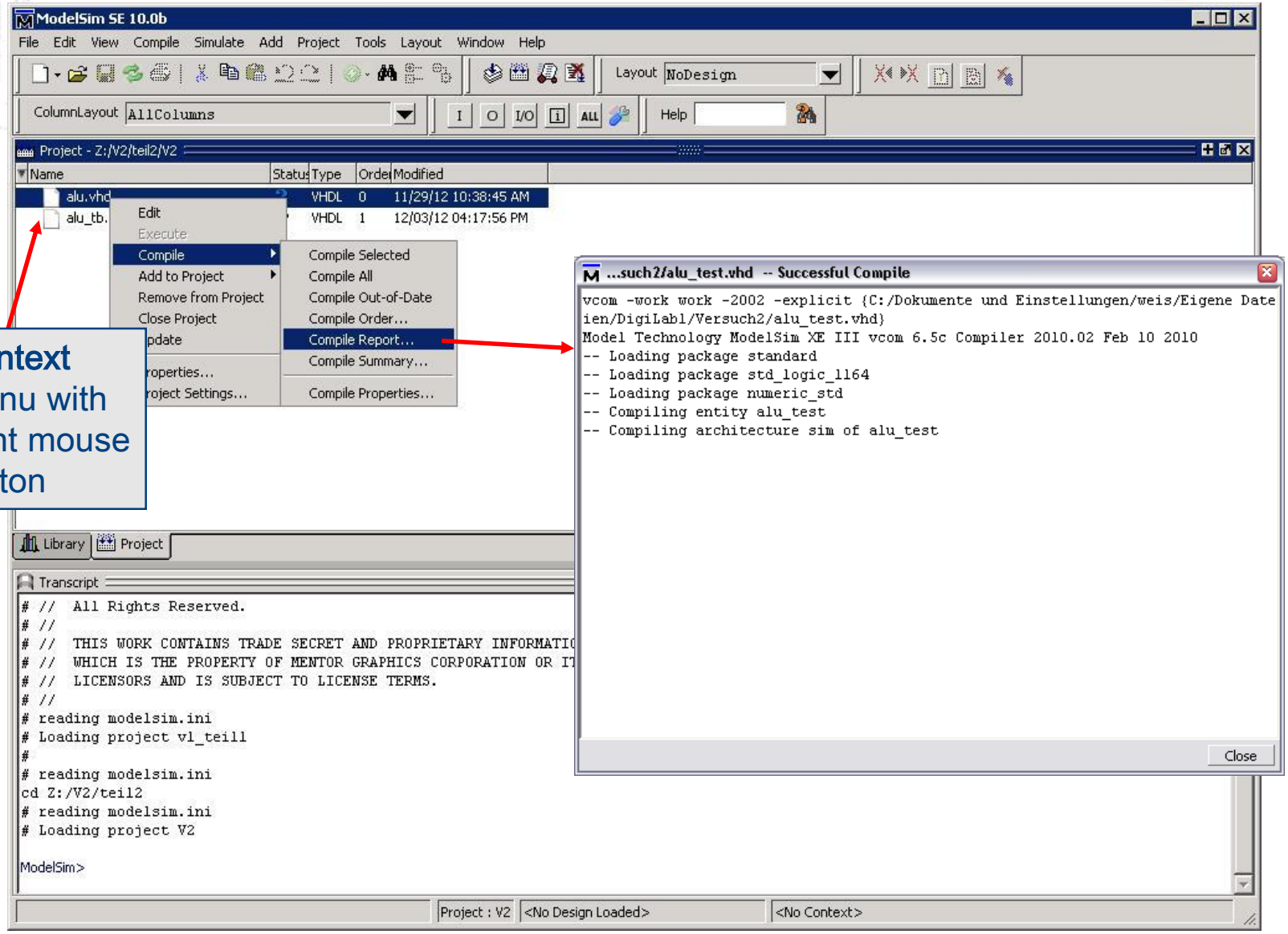
Compilation



After Compilation



After Compile – Detailed Report

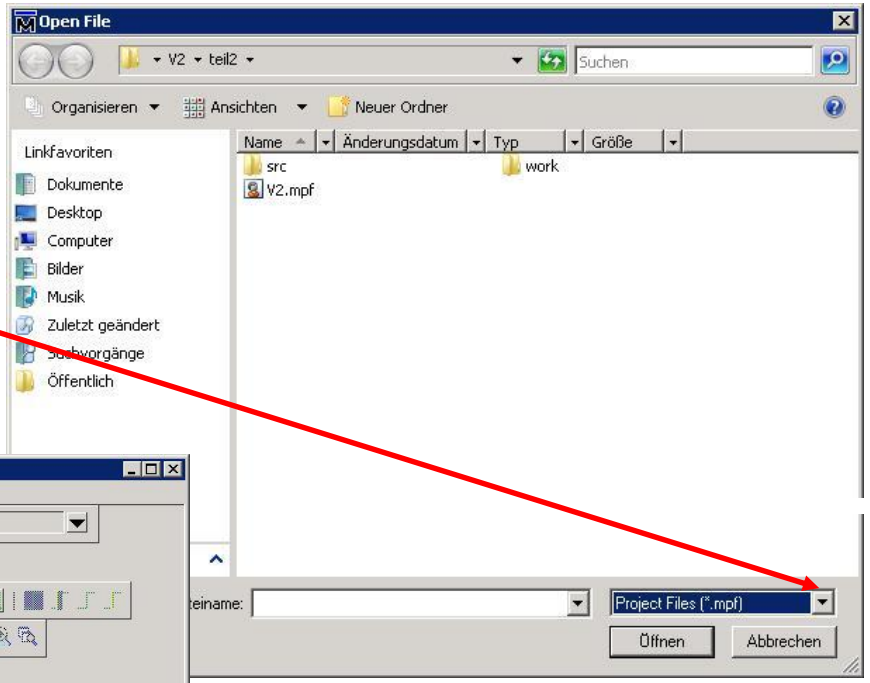


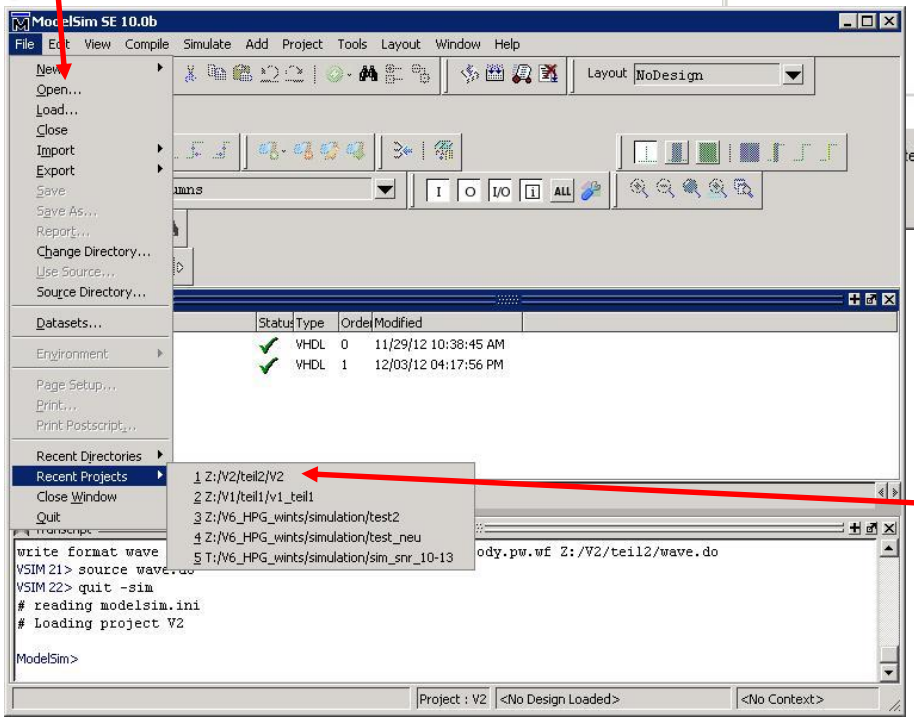
Context menu with right mouse button

Open Existing Projects

Existing projects can be opened with
Menu: File => open

Change file-type to „mpf“





Last projects in ModelSim can be reopened with
Menu: File => Recent Projects

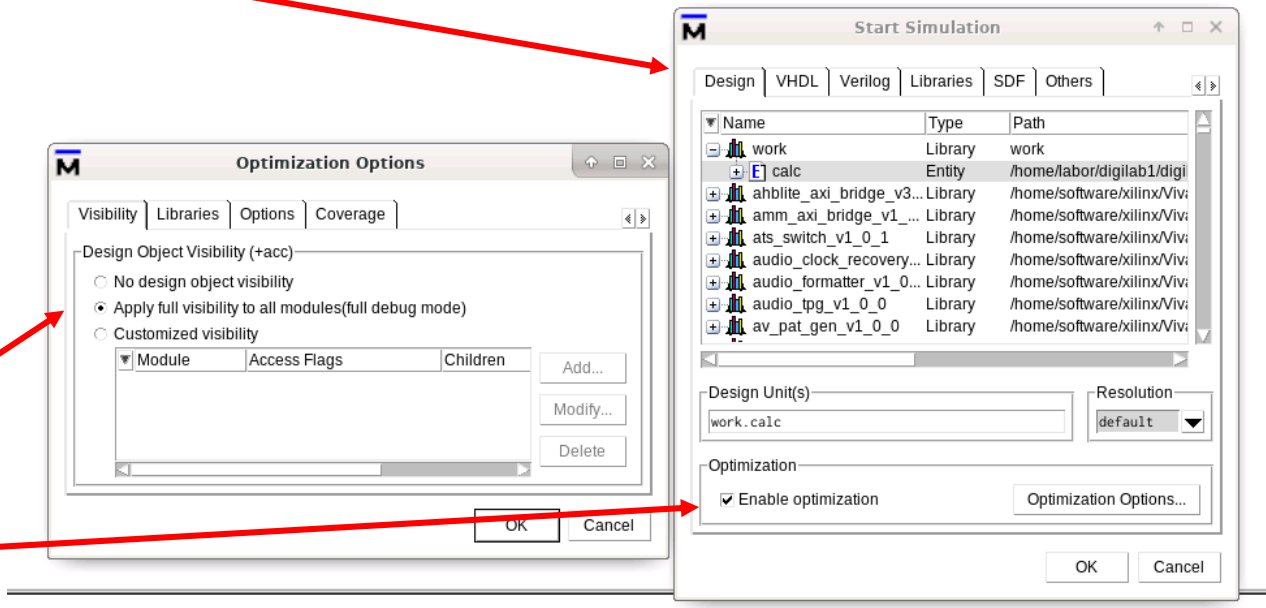
Start Simulation

After creation of a new project (and successful compilation) or opening of an existing project, you can start a simulation

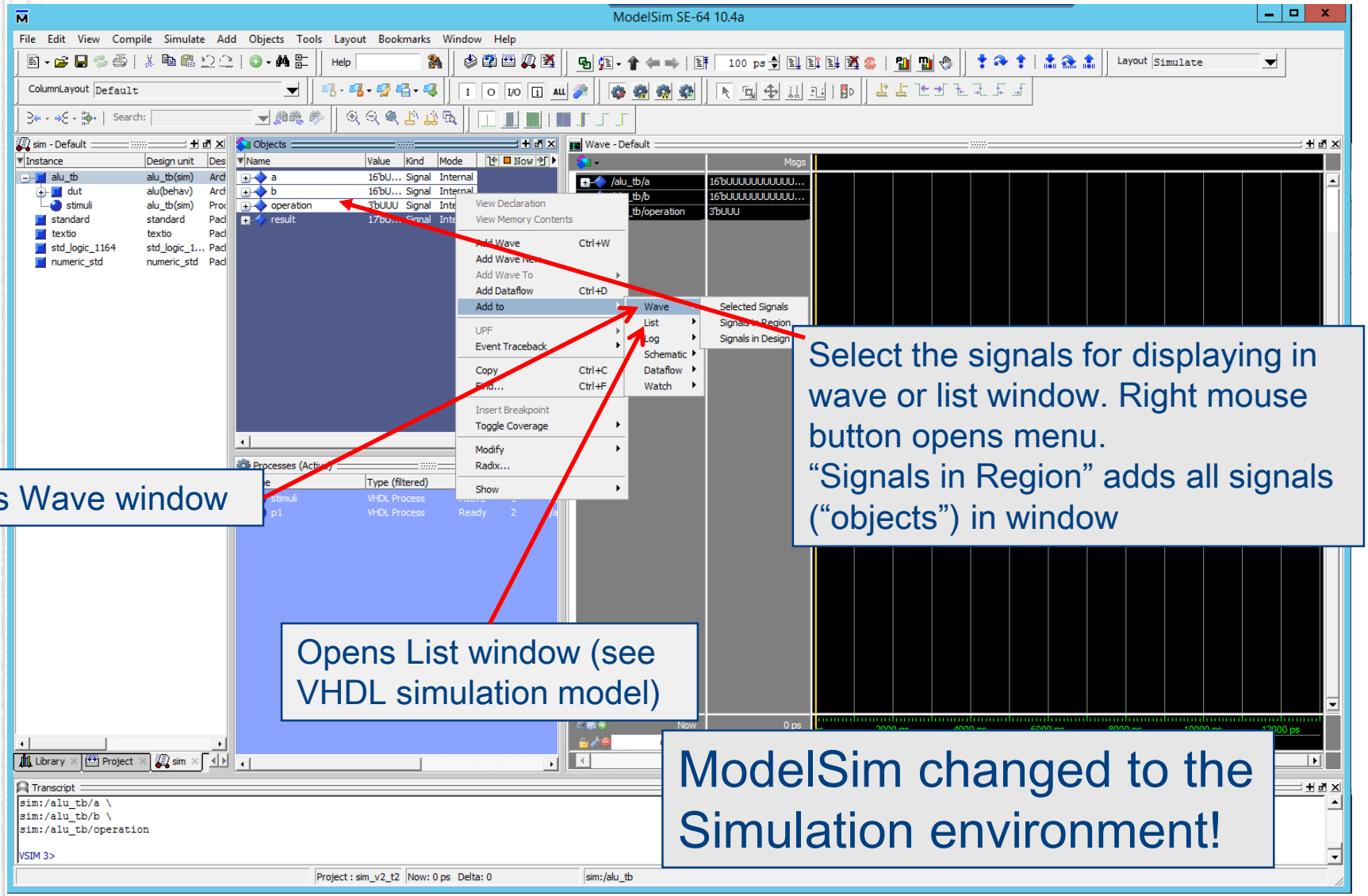
Menu: Simulate => Start Simulation…

Select the desired model or test bench (example alu_tb) from the **work** library

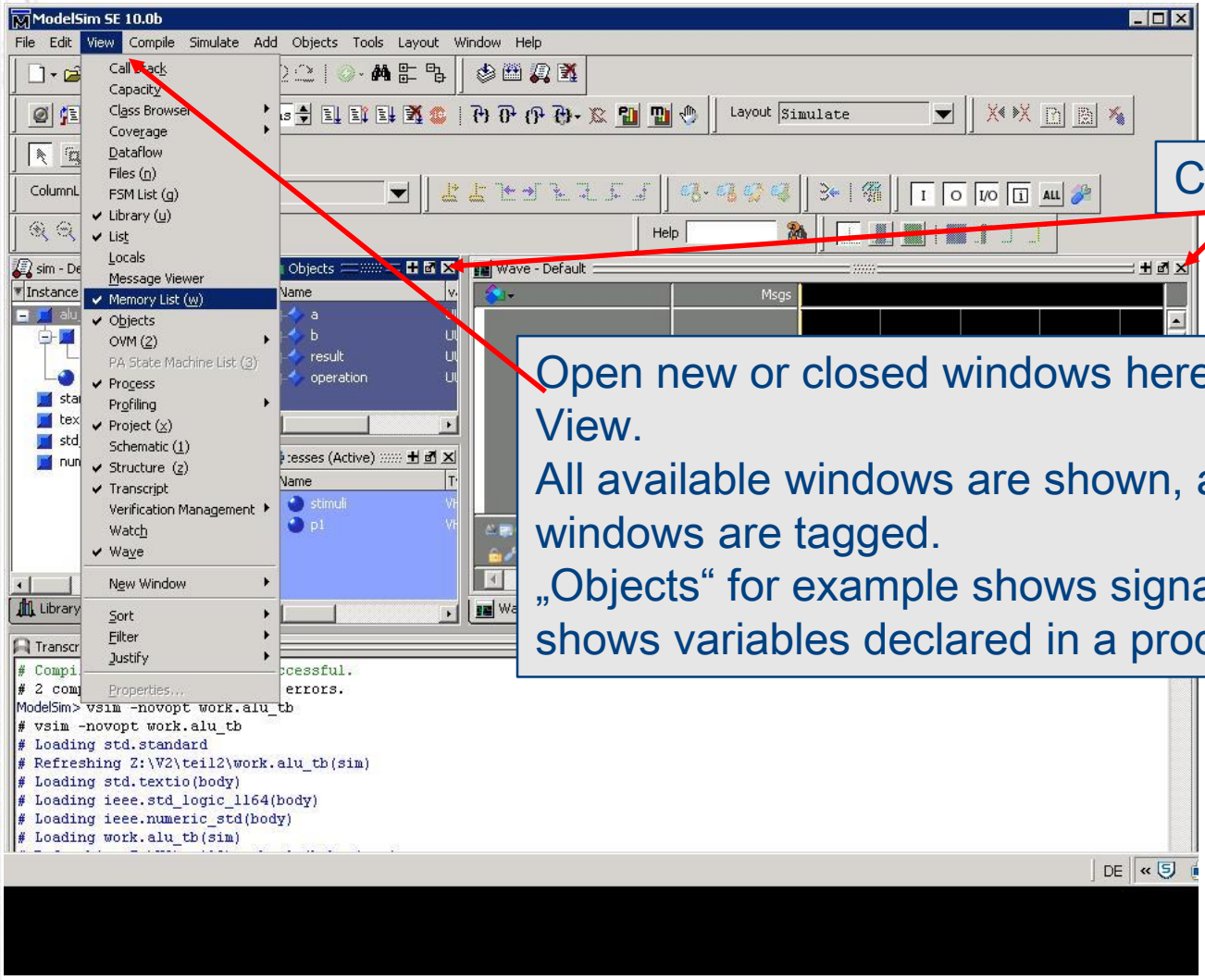
Enable optimization and press button OK to start simulation (otherwise not all signals can be displayed)
And apply full visibility



View Signals in Simulation



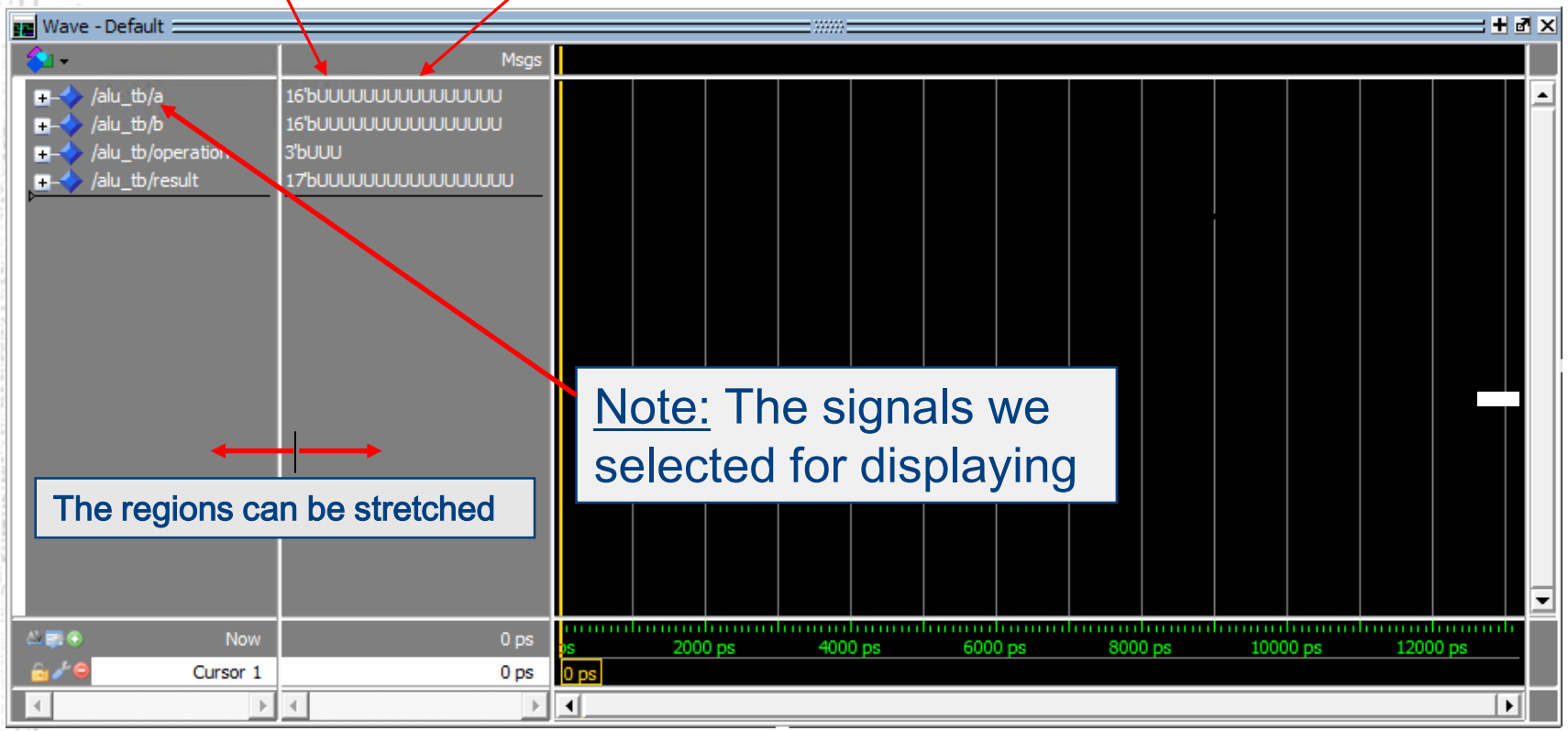
Adding Windows



Close window

Open new or closed windows here in menu View.
All available windows are shown, active windows are tagged.
„Objects“ for example shows signals, „Locals“ shows variables declared in a process.

Viewing the Signals (Waveform)

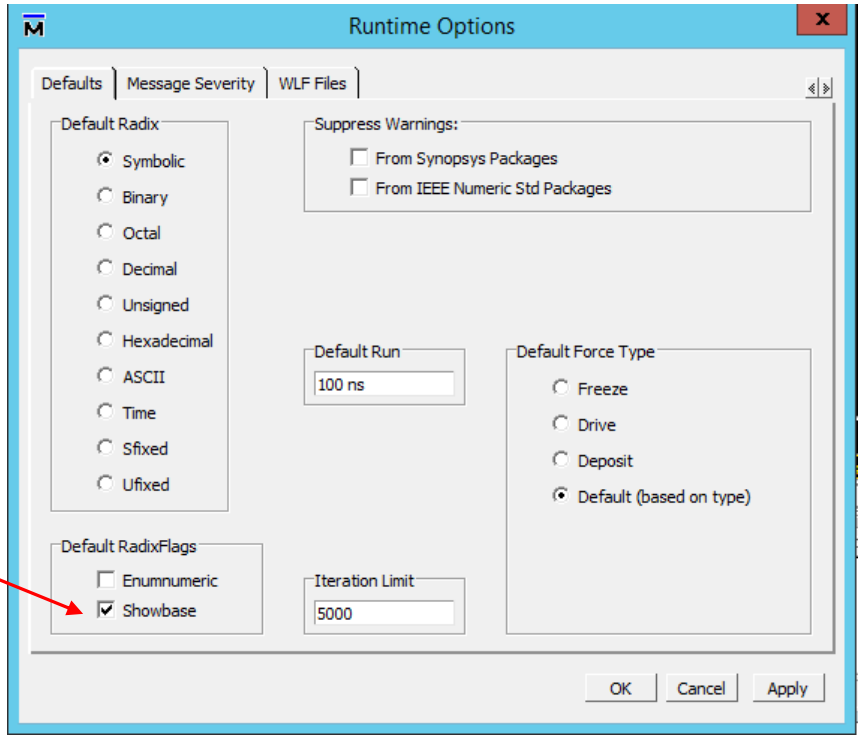


ModelSim Wave window

The signals have standard initialization value (here: U).

Running the Simulation

Menu: Simulate => Runtime Options…



Radix is shown in Wave window

After setting the default run time to 100ns; possible commands in Transcript window:

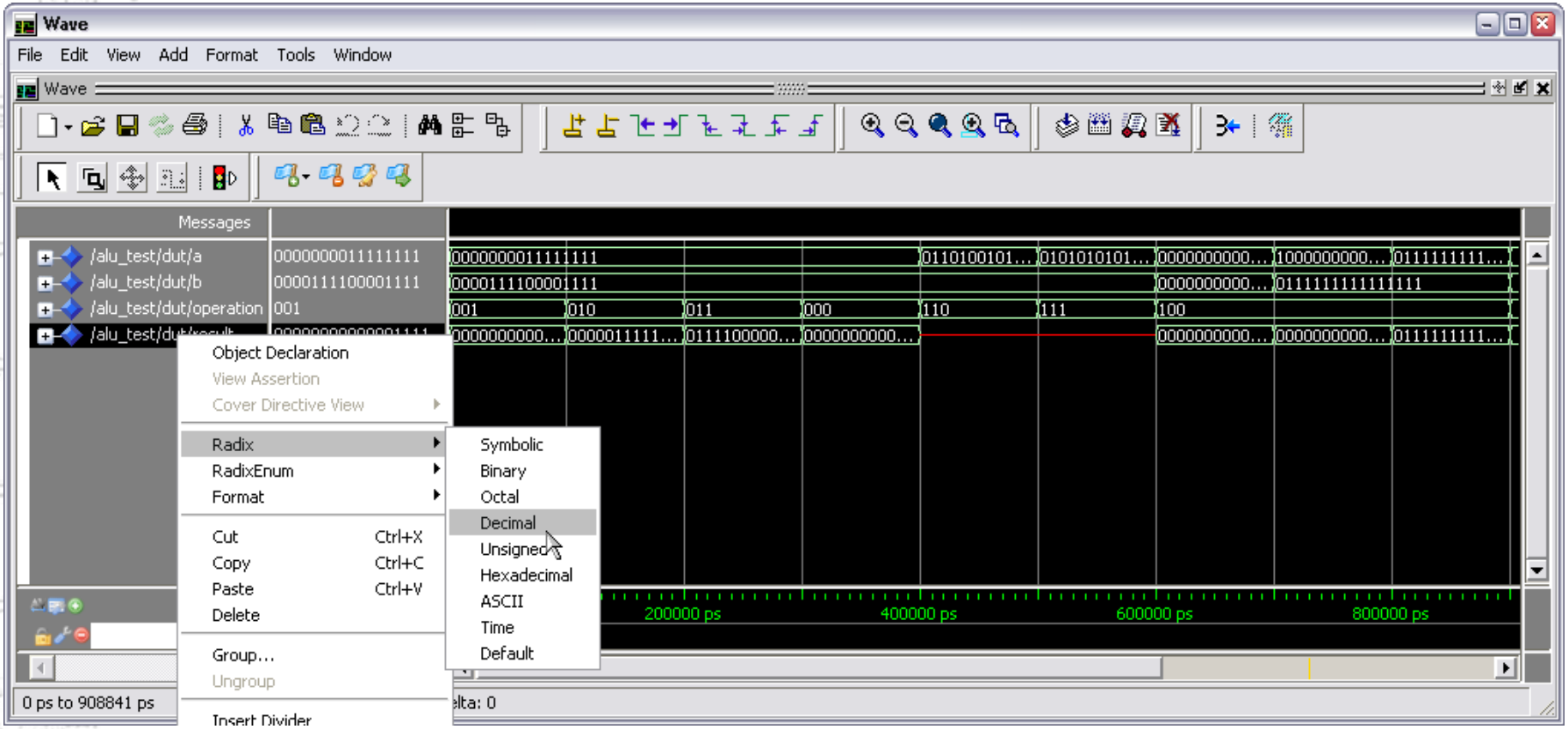
VSIM 6> run (100 ns simulation time)

VSIM 7> run 1000 ns

Waveform Viewing (1)

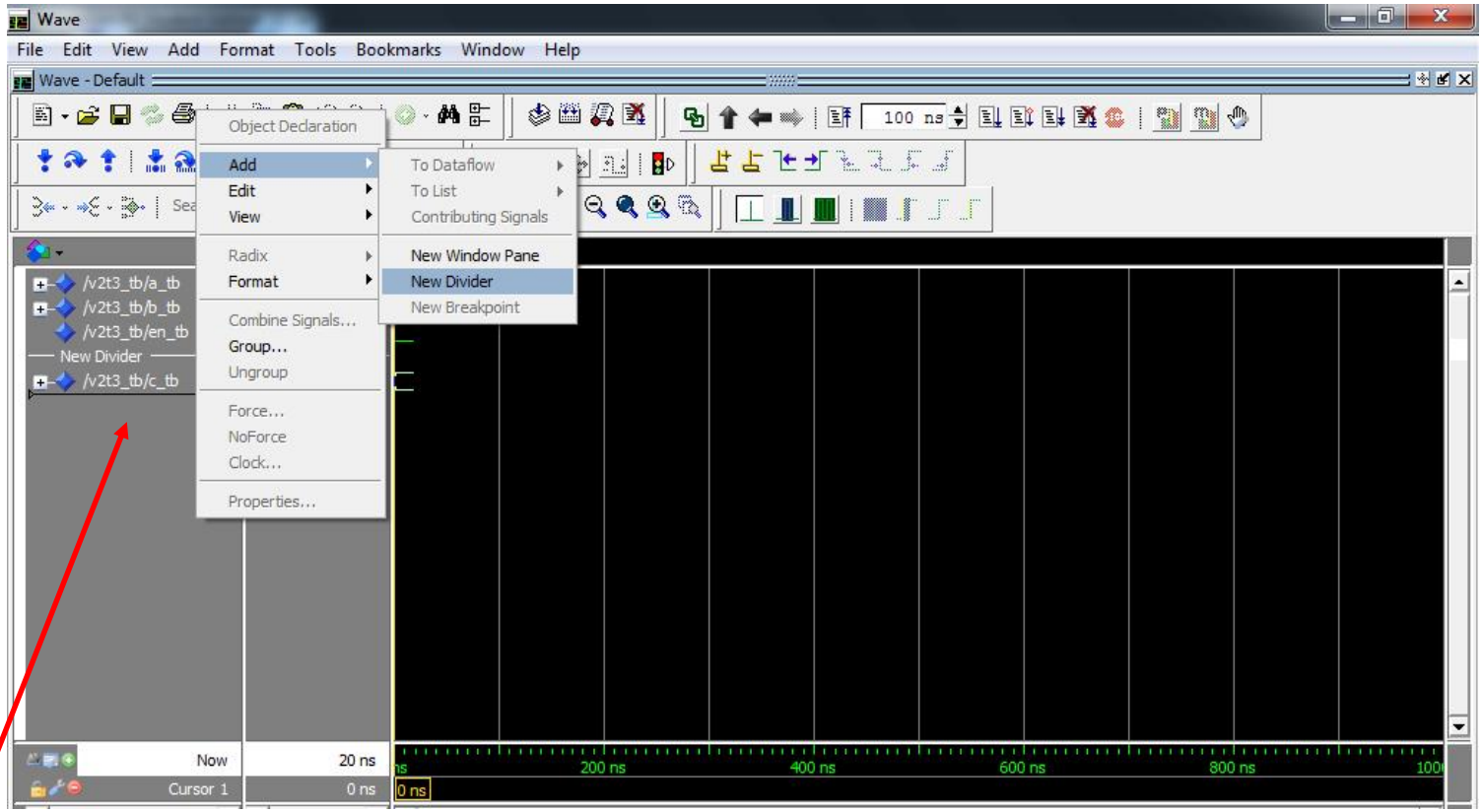
After running the simulation the signals now change their values in the waveform display – they are currently in symbolic format => for changing the **radix** press the right mouse button on the signal and select

- **Decimal** for a signed representation
- **Unsigned** for an unsigned representation



Waveform Viewing (2)

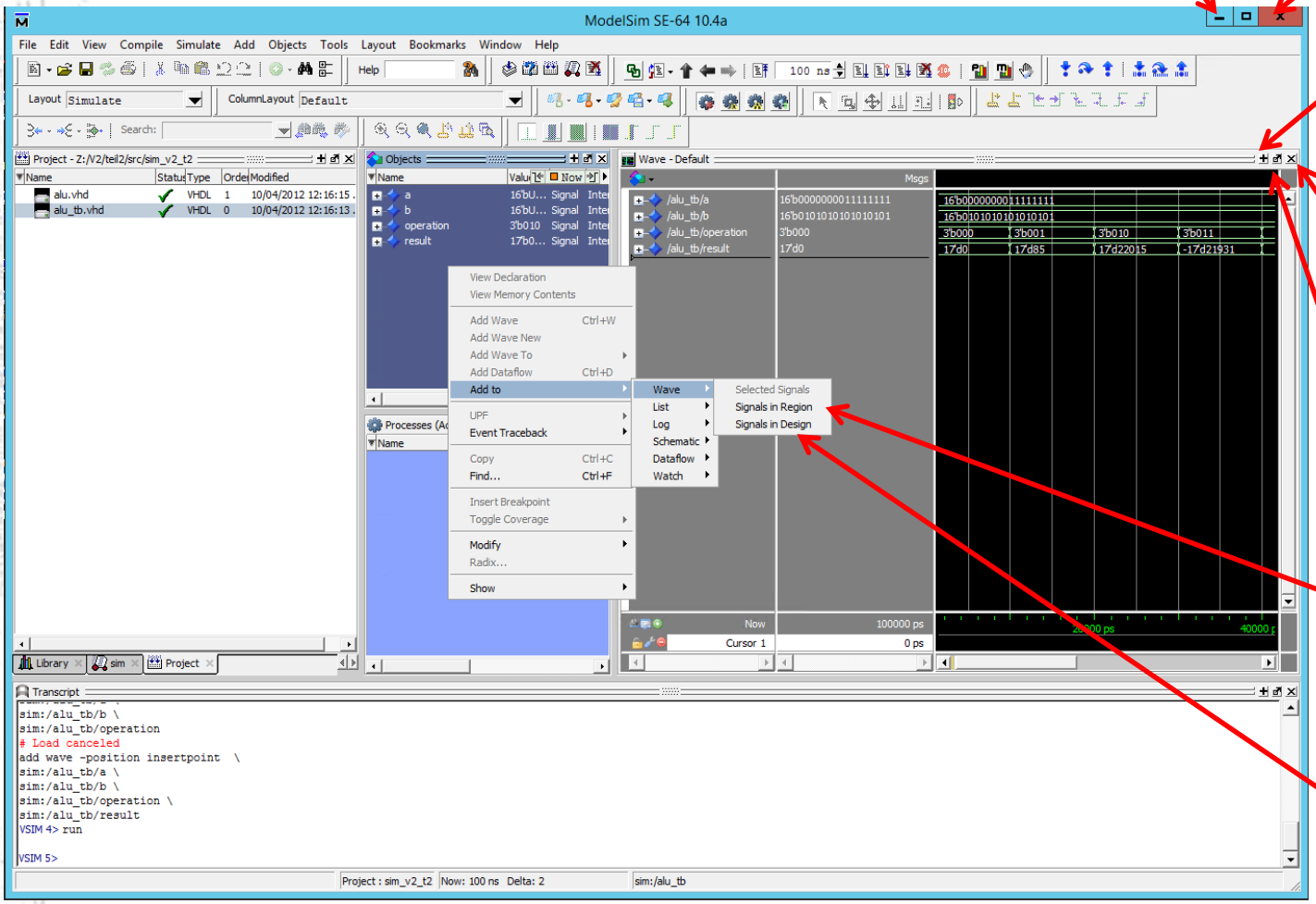
Adding dividers in wave-windows and grouping signals helps understanding simulation results and locating design problems:



You can rearrange signal order by moving signal names with pressed left mouse button to requested position

Waveform Viewing (3)

(Sub)-Window manipulation and signal setup



Iconize Window

Maximize Window

Enlarge Window

Close Window

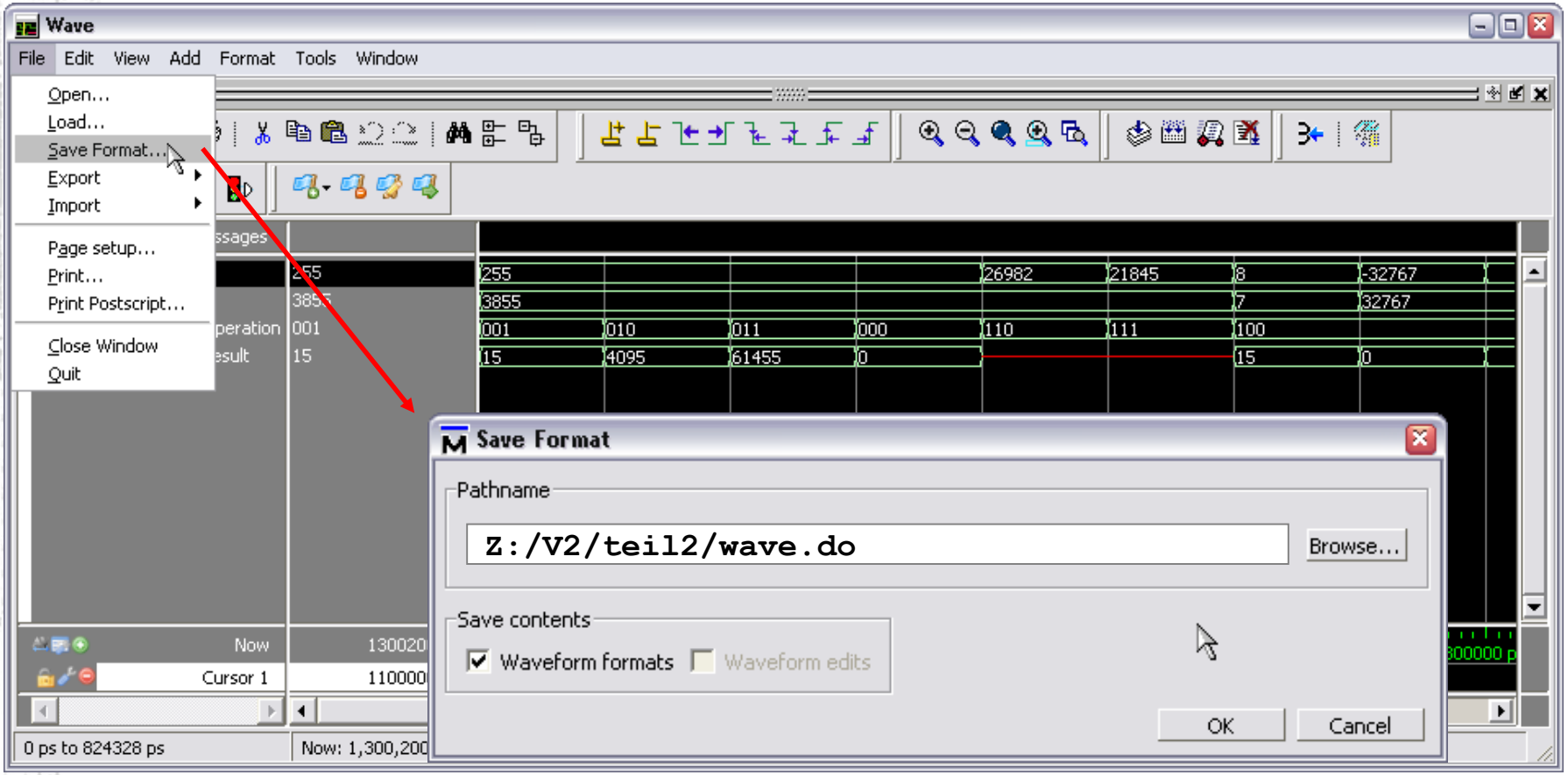
Undock Window

View all signals of object

View all signals incl. lower hierarchies

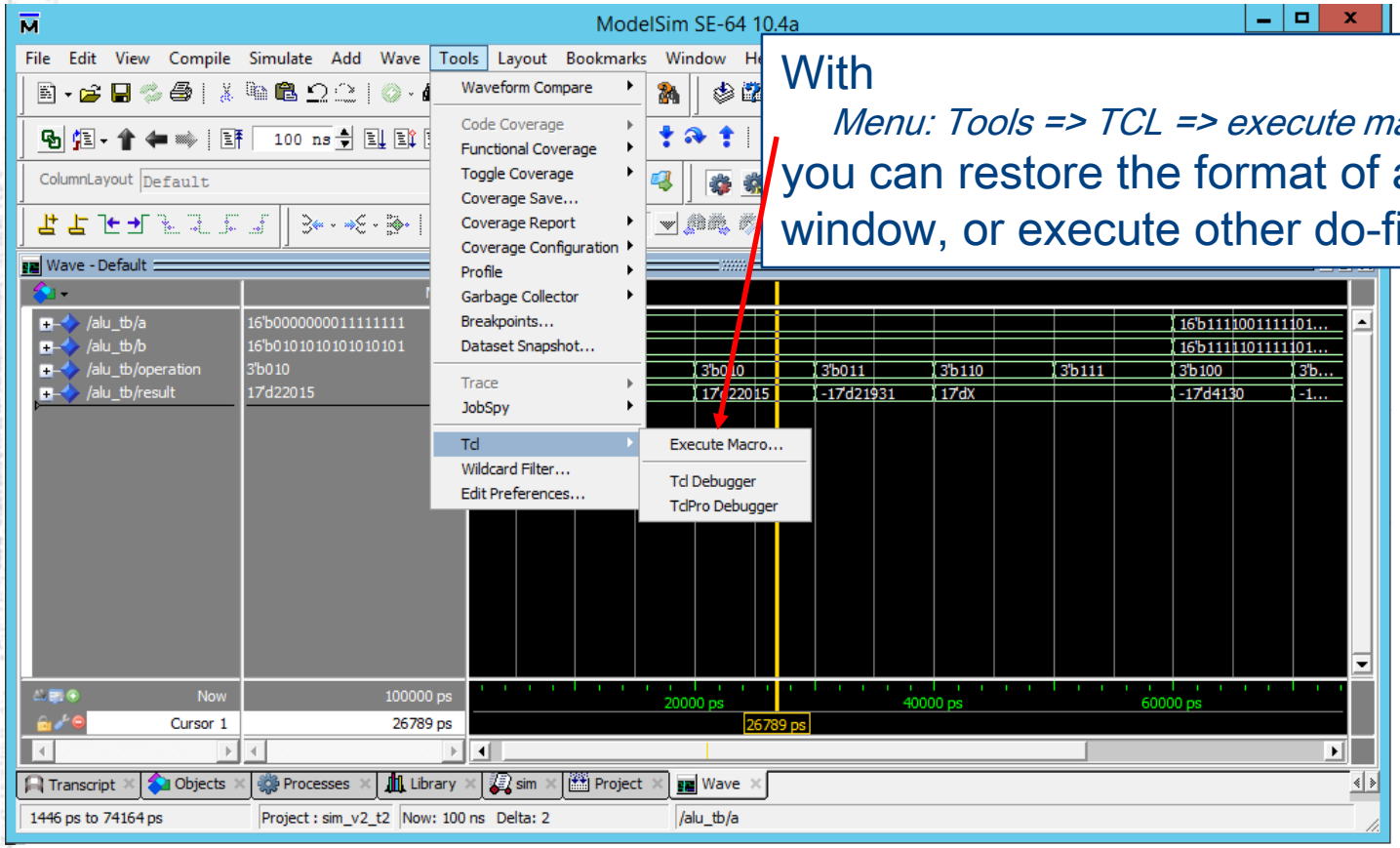
Saving Signal Setup (wave.do)

The setup of the wave window can be saved: **This saves a lot of time for further setups!**



Using Signal Setup (wave.do)

The stored setup of the wave window can be used: **This saves a lot of time**



With

Menu: Tools => TCL => execute macro => wave.do

you can restore the format of a wave-window, or execute other do-files.

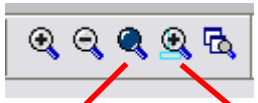
The “wave.do”-file can also be loaded by command in Transcript window before running the simulation e.g.:

VSIM 8> source wave.do

Waveform Viewing (4)

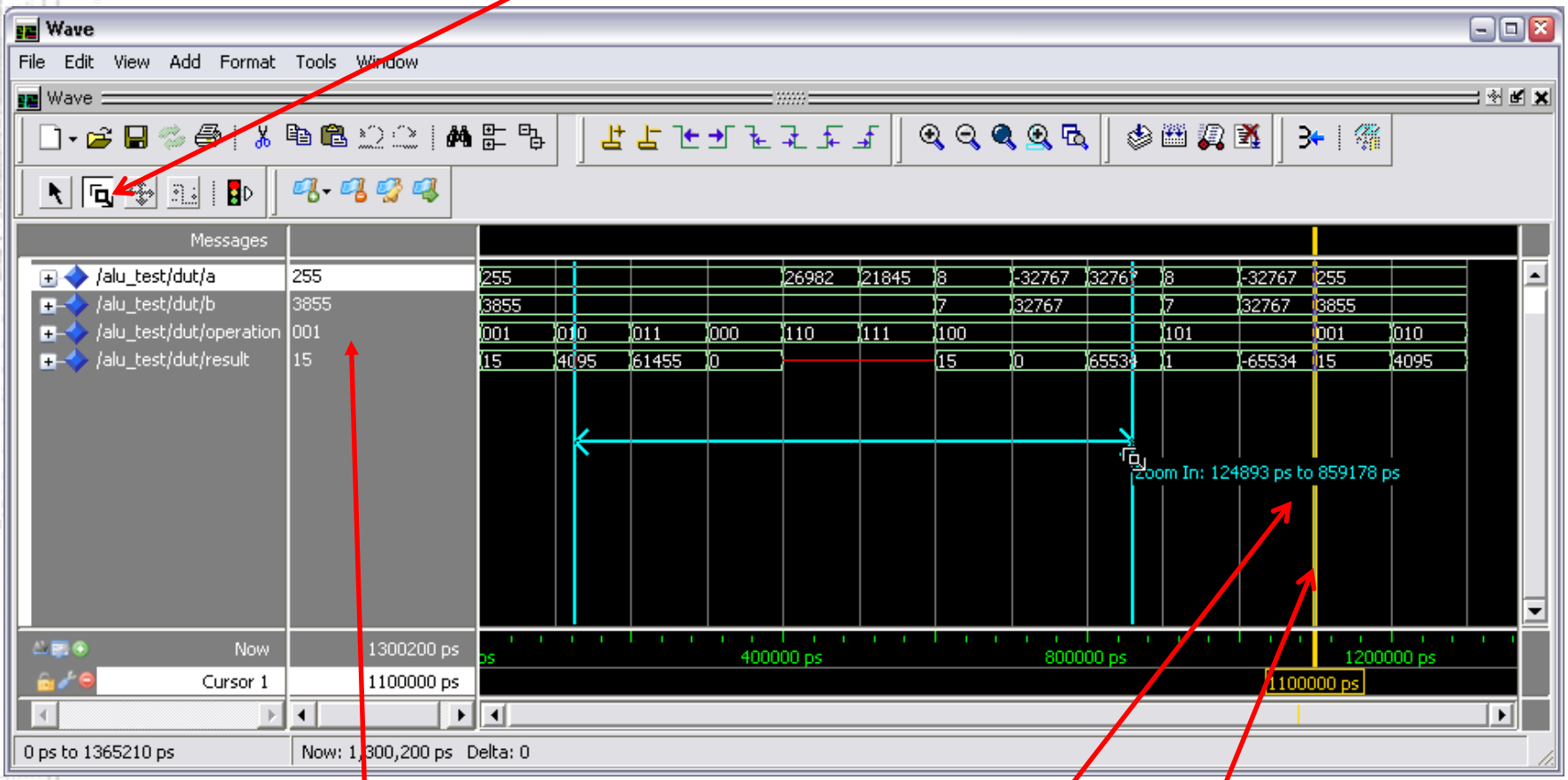
Zooming:

Standard zooms:



Select mode **Zoom** mode

Full view Zoom to cursor



Signal values at time of active cursor position

Cursor

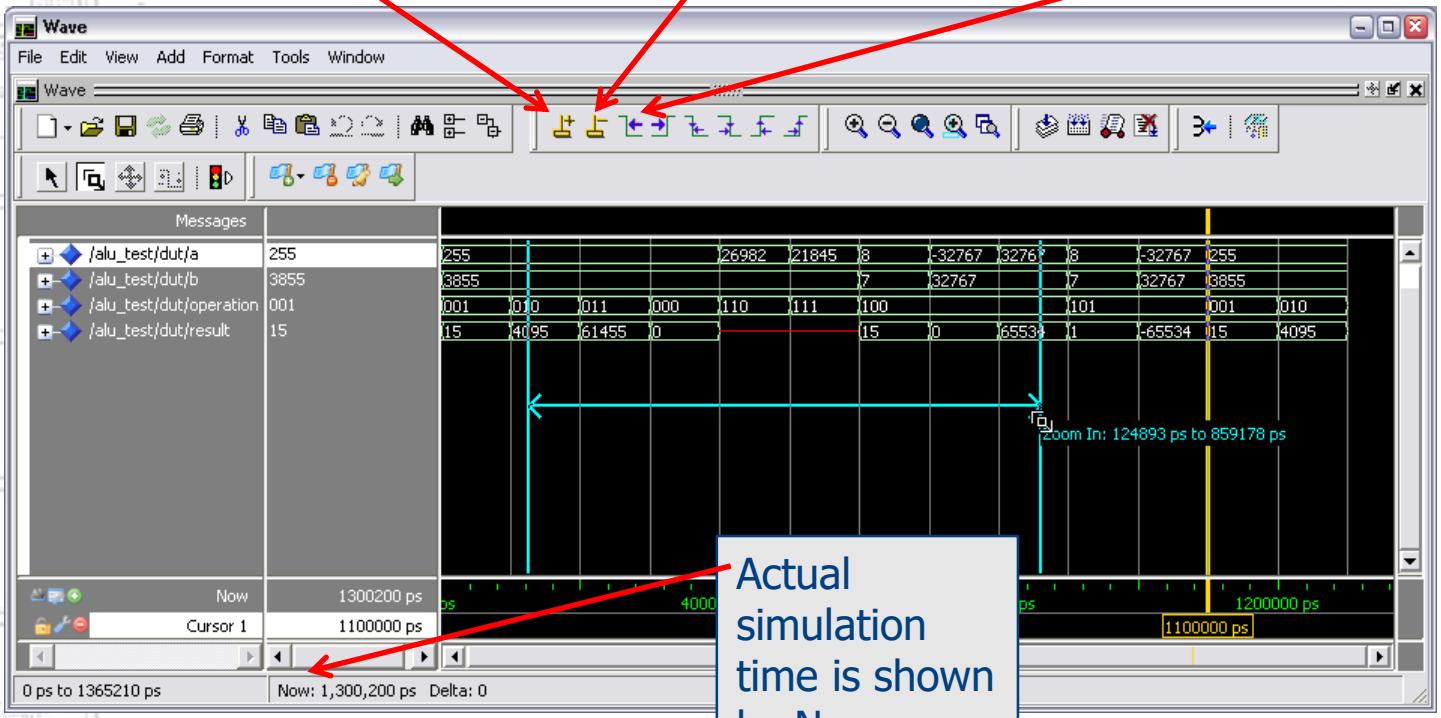
Waveform Viewing (5)

Further Waveform checking

Additional Cursor
Time difference between cursors is always shown

Remove active cursor

Move active Cursor to next/last event of selected signal



Actual simulation time is shown by Now:

Debugging

Restarting the simulation (if VHDL code is changed please re-compile before) by command in Transcript window:

VSIM 6> restart

VSIM 7> run 1000 ns

[ ] means optional parameter; without time parameter means at current simulation time

Single forces:

VSIM 2> force <object_name> <value> [<relative time of assignment>]

e.g. force b 10#5 0 ns e.g. force b 10#6

VSIM 3> run 100 ns

10# means following number (here 6) uses base 10

Single step – enables debug mode with an arrow at the source code (VDHL)

VSIM 8> step <number> (if no number entered: default is 1)

Breakpoints can be set with the left mouse button, if the source code (VHDL) is opened in ModelSim:

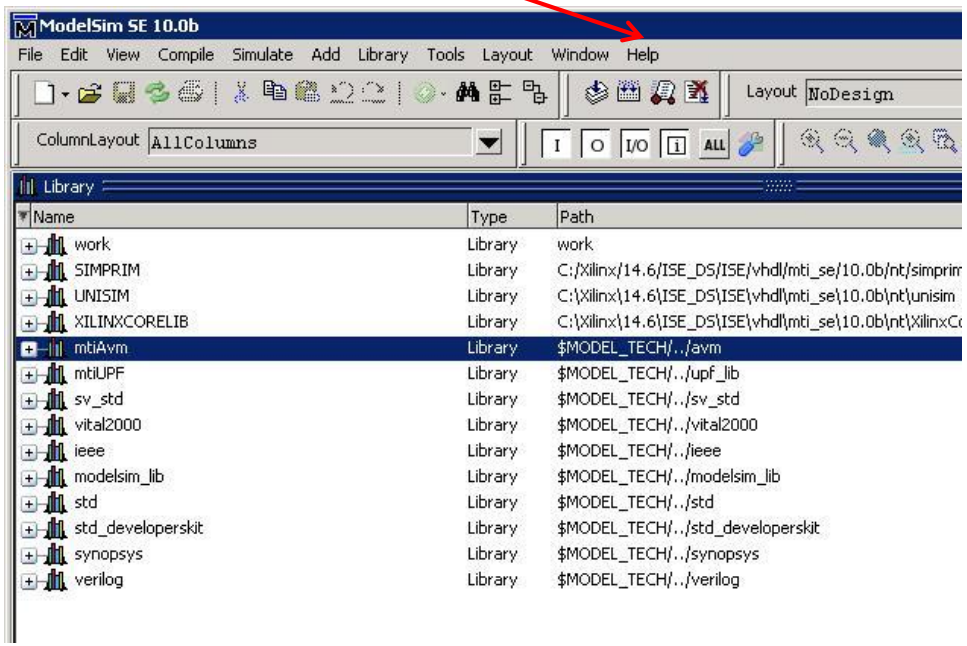
Useful Hints (1)

Starting ModelSim for the first time, it opens empty. Later always the last project is reopened and the last directory is active, even when you want to start with new things.

Start with each new exercise a new project.

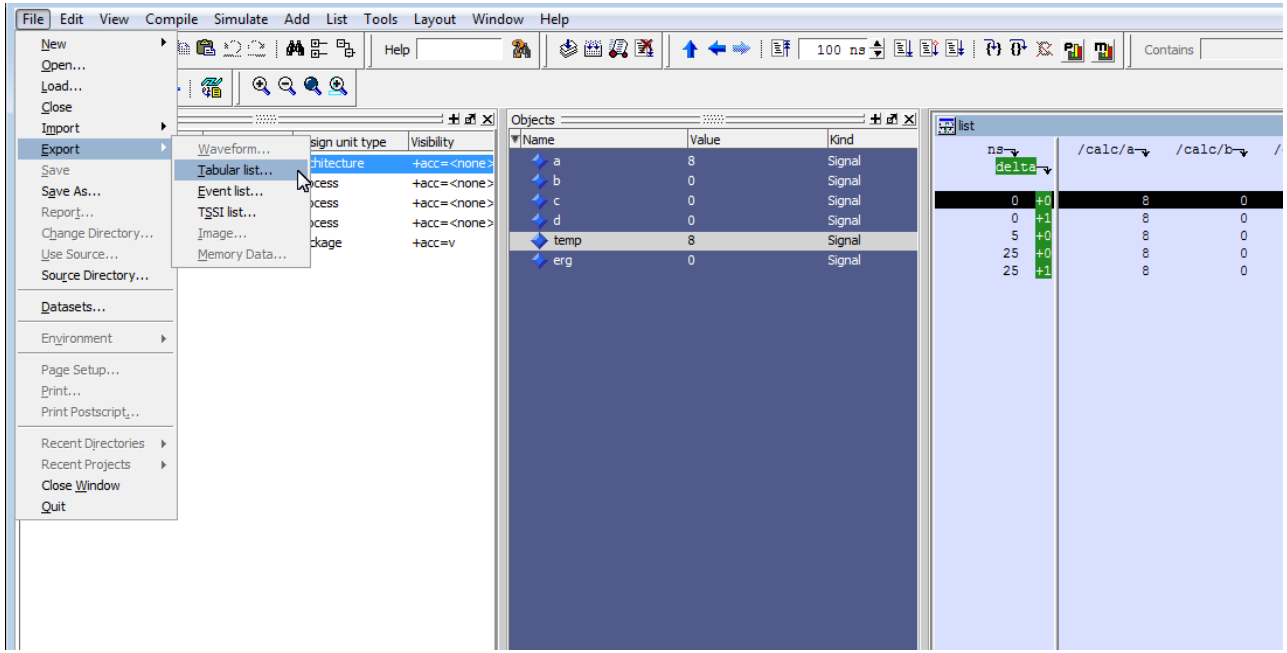
Compiling everything into one work-library causes confusion.

Help (user guide, tutorials ...) is available with
Menu: Help => Documentation PDF-Bookcase



Useful Hints (2)

You can export simulation results of the list window into text files.
Activate the list-window, then *Menu: File => Export => Tabular list*
This is useful to document your simulation results for testation:



After this command a new window will open, where you can specify the write-list-filename (extension ".lst").