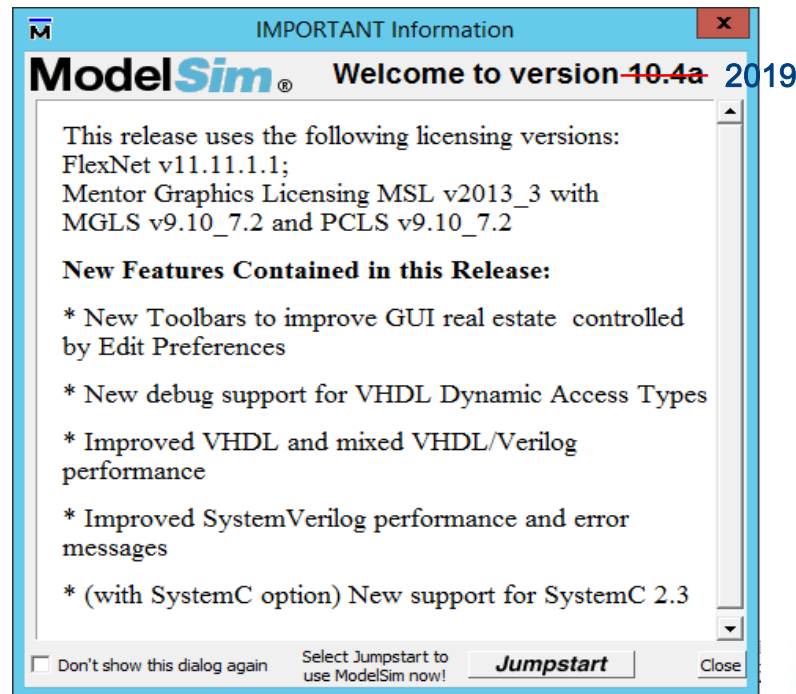
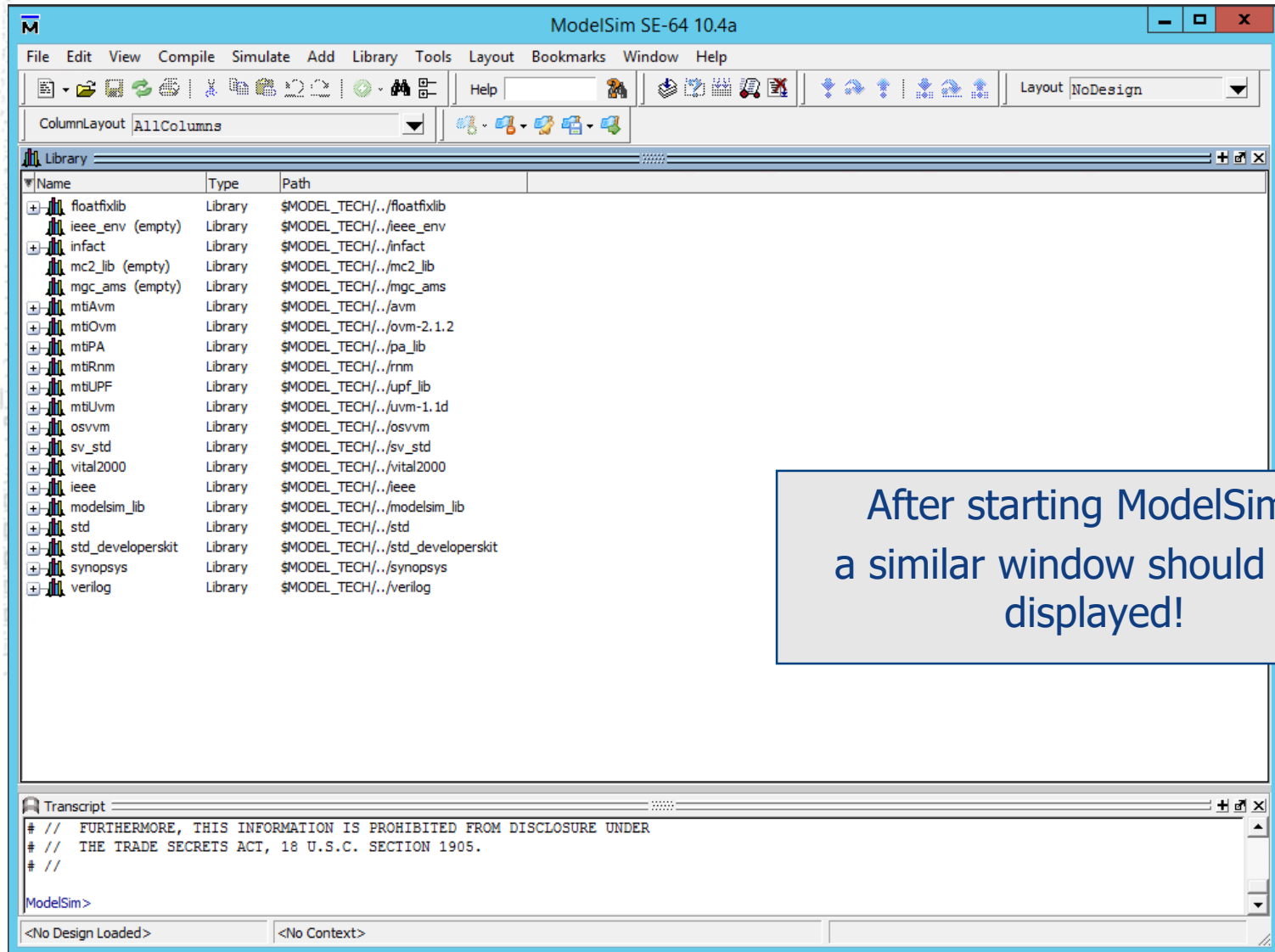


ModelSim Tutorial

Christian Weis, Uwe Wasenmüller, Hans Peter Goldhammer



Getting Started



After starting ModelSim
a similar window should be
displayed!

Creating a New Project (Example Exercise 2)

The screenshot shows the ModelSim SE-64 10.4a interface. The 'File' menu is open, and the 'Project...' option is selected. A red arrow points from 'Project...' to the 'Create Project' dialog box. In the 'Create Project' dialog, the 'Project Name' field contains 'sim_v2_t2'. A red arrow points from the 'Add project name' callout to this field. The 'Project Location' field contains 'Z:/V2/teil2/src', and a red arrow points from the 'Choose your directory' callout to the 'Browse...' button next to it. The 'Copy Settings From' field contains 'Z:/V2/teil2/modelsim.ini', and a red arrow points from the 'Setup Modelsim Environment' callout to the 'Browse...' button next to it. The 'Copy Library Mappings' radio button is selected. The 'OK' and 'Cancel' buttons are at the bottom right of the dialog.

File Edit View Compile Simulate Add Library Tools Layout Bookmarks Window Help

New Open... Load Close Import Export Save Save As... Report... Change Directory... Use Source... Source Directory... Datasets... Environment Page Setup... Print... Print Postscript... Recent Directories Recent Projects Close Window Quit

Folder Source Project... Library... Debug Archive... Results Analysis Database...

\$MODEL_TECH/./ieee_env
\$MODEL_TECH/./infact
\$MODEL_TECH/./mc2_lib
\$MODEL_TECH/./mgc_ams
\$MODEL_TECH/./avm
\$MODEL_TECH/./ovm-2.1.2
\$MODEL_TECH/./pa_lib
\$MODEL_TECH/./rnm
\$MODEL_TECH/./upf_lib
\$MODEL_TECH/./uvvm-1.1d
\$MODEL_TECH/./osvvm
\$MODEL_TECH/./sv_std
\$MODEL_TECH/./vital2000
\$MODEL_TECH/./ieee
\$MODEL_TECH/./modelsim_lib
\$MODEL_TECH/./std
\$MODEL_TECH/./std
\$MODEL_TECH/./std
\$MODEL_TECH/./std

Create Project

Project Name
sim_v2_t2

Project Location
Z:/V2/teil2/src Browse...

Default Library Name

Copy Settings From
Z:/V2/teil2/modelsim.ini Browse...

☒ Copy Library Mappings ☐ Reference Library Mappings

OK Cancel

Setup Modelsim Environment

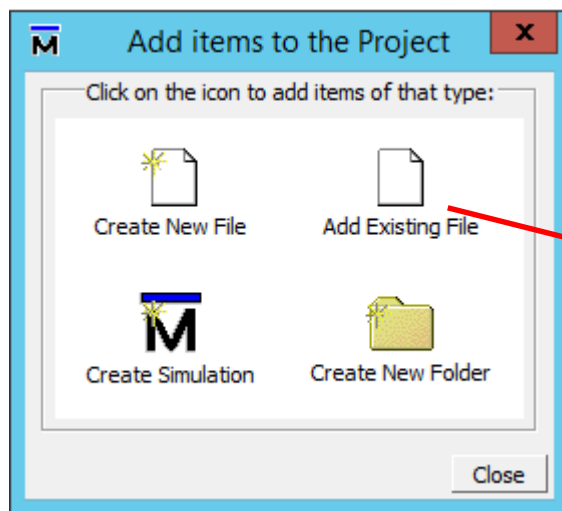
Transcript

```
# // FURTHERMORE, THIS INFORMATION IS PROHIBITED FROM DISCLOSURE
# // THE TRADE SECRETS ACT, 18 U.S.C. SECTION 1905.
# //
```

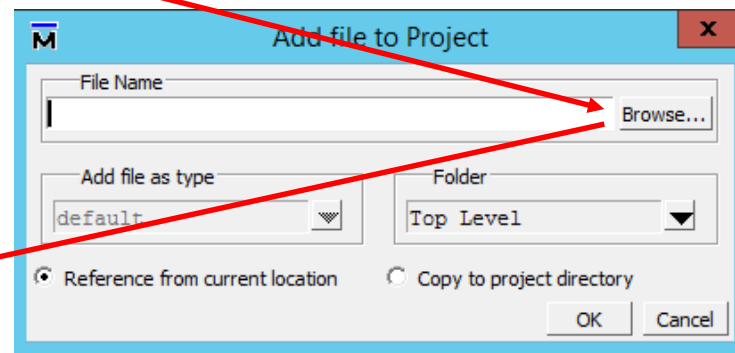
ModelSim>

<No Design Loaded> \$MODEL_TECH/./floatfixlib

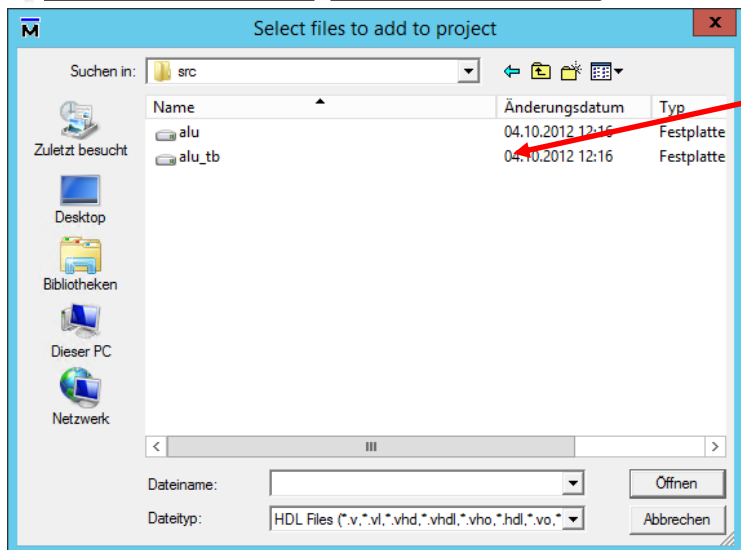
Adding Files to a Project



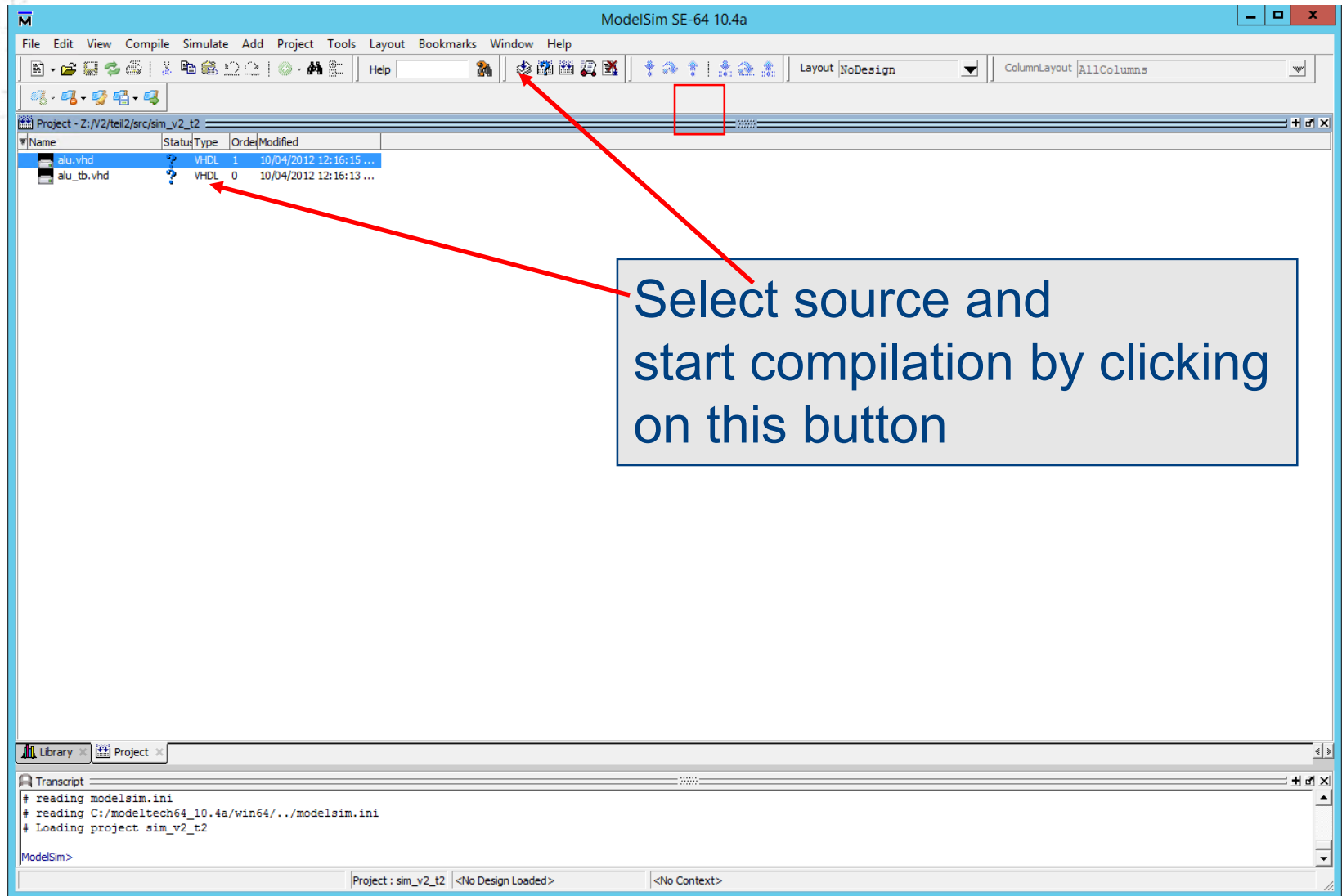
1.
Adding files by pressing the
“Browse ...” button -
multiple selections possible



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



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



After Compilation

The screenshot shows the ModelSim SE-64 10.4a interface. The top menu bar includes File, Edit, View, Compile, Simulate, Add, Project, Tools, Layout, Bookmarks, Window, and Help. Below the menu bar is a toolbar with various icons. The main window displays a project named 'Project - Z:/V2/teil2/src/sim_v2_t2'. A table lists the project files:

Name	Status	Type	Order	Modified
alu.vhd	✓	VHD	1	10/04/2012 12:16:15 ...
alu_tb.vhd	✓	VHD	0	10/04/2012 12:16:13 ...

A red circle highlights the 'Status' column for both files, indicating successful compilation. The bottom window shows the 'Transcript' window with the following text:

```
# Loading project sim_v2_t2
# Compile of alu.vhd was successful.
# Compile of alu_tb.vhd was successful.
ModelSim>
```

A red arrow points from the 'Transcript' window to a callout box that says 'Monitor Transcript window for errors!'.

After Compile – Detailed Report

Context
menu with
right mouse
button

The screenshot shows the ModelSim SE 10.0b interface. A context menu is open for the file 'alu_vhd' in the Project window. The menu options include Edit, Execute, Compile, Add to Project, Remove from Project, Close Project, Update, Properties..., and Project Settings.... The 'Compile' option is selected, and a sub-menu is visible with options: Compile Selected, Compile All, Compile Out-of-Date, Compile Order..., Compile Report..., Compile Summary..., and Compile Properties.... A red arrow points from the 'Compile Report...' option to a dialog box titled '...such2/alu_test.vhd -- Successful Compile'. The dialog box contains the following text:

```
vcom -work work -2002 -explicit {C:/Dokumente und Einstellungen/weis/Eigene Date
ien/DigiLab1/Versuch2/alu_test.vhd}
Model Technology ModelSim XE III vcom 6.5c Compiler 2010.02 Feb 10 2010
-- Loading package standard
-- Loading package std_logic_1164
-- Loading package numeric_std
-- Compiling entity alu_test
-- Compiling architecture sim of alu_test
```

The Project window shows a table with the following data:

Name	Status	Type	Order	Modified
alu_vhd		VHDL	0	11/29/12 10:38:45 AM
alu_tb		VHDL	1	12/03/12 04:17:56 PM

The Transcript window shows the following text:

```
# // All Rights Reserved.
# //
# // THIS WORK CONTAINS TRADE SECRET AND PROPRIETARY INFORMATION
# // WHICH IS THE PROPERTY OF MENTOR GRAPHICS CORPORATION OR ITS
# // LICENSORS AND IS SUBJECT TO LICENSE TERMS.
# //
# reading modelsim.ini
# Loading project vl_teil1
#
# reading modelsim.ini
cd Z:/V2/teil2
# reading modelsim.ini
# Loading project V2
ModelSim>
```

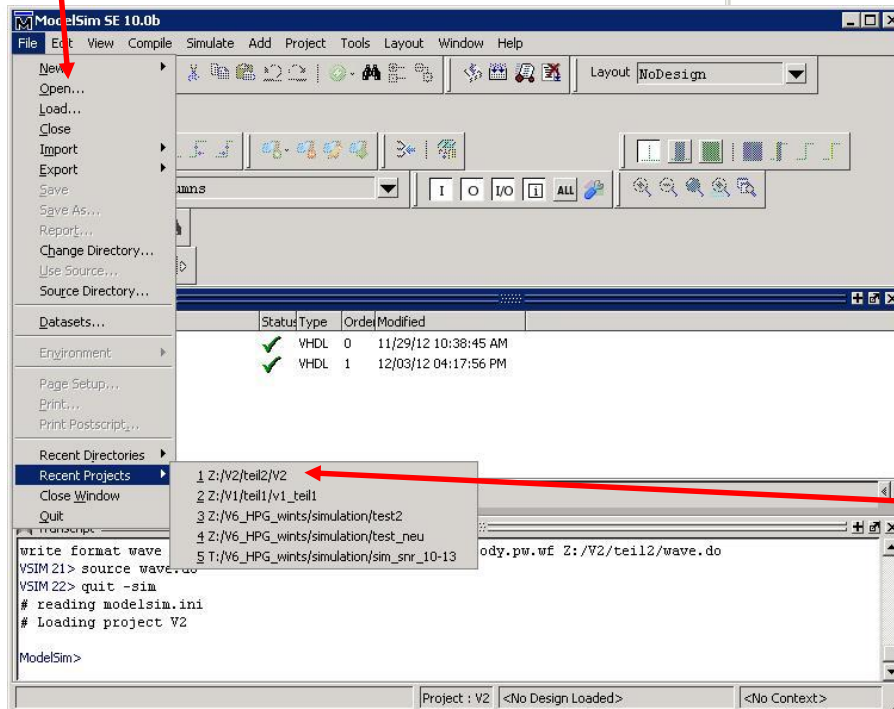
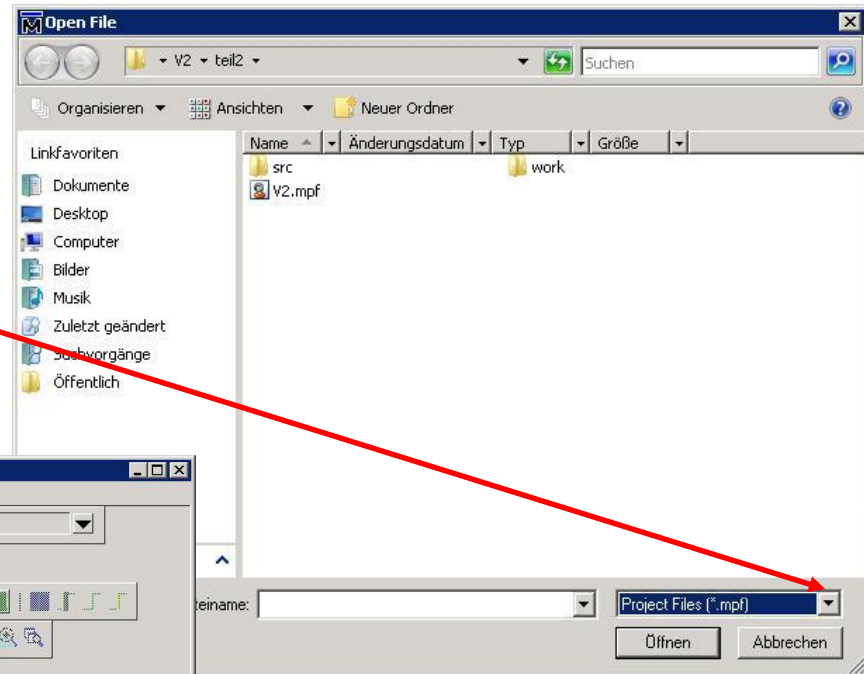
The status bar at the bottom shows 'Project : V2', '<No Design Loaded>', and '<No Context>'.

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

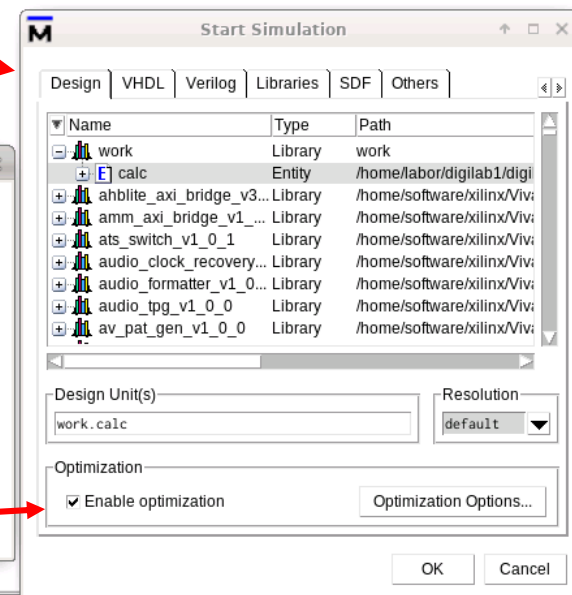
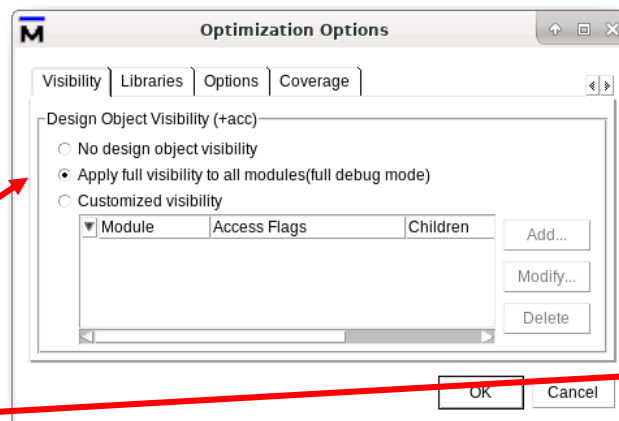
Start Simulation

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

The screenshot shows the ModelSim SE-64 10.4a interface. The main window displays the simulation results for a VHDL process. The 'Objects' pane on the left lists the design units and their instances. The 'Wave' pane on the right shows the signal waveforms. The 'List' pane in the center displays the signals in the simulation model.

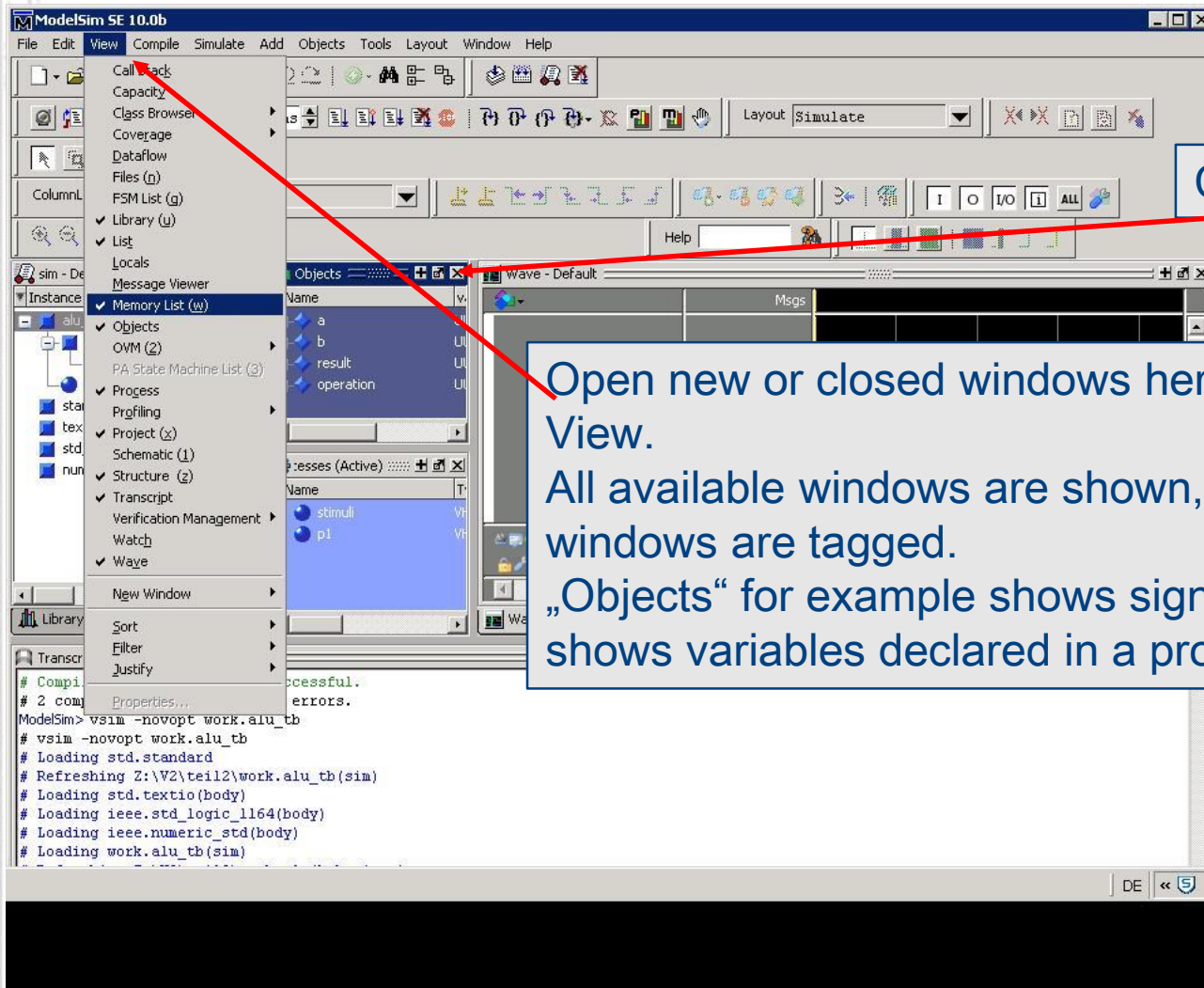
Opens Wave window

Opens List window (see VHDL simulation model)

Select the signals for displaying in wave or list window. Right mouse button opens menu. "Signals in Region" adds all signals ("objects") in window

ModelSim changed to the Simulation environment!

Adding Windows



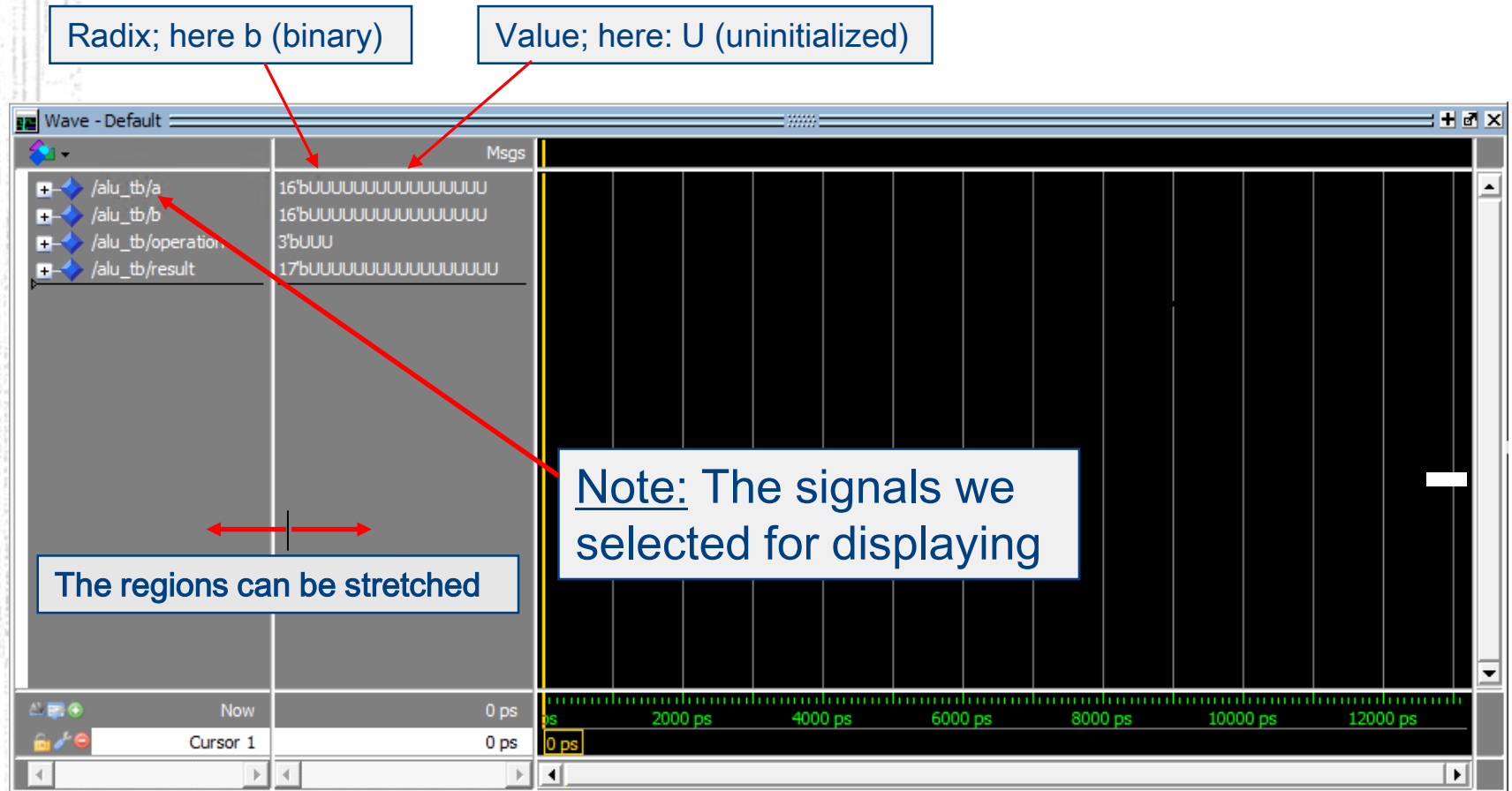
Close window

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)

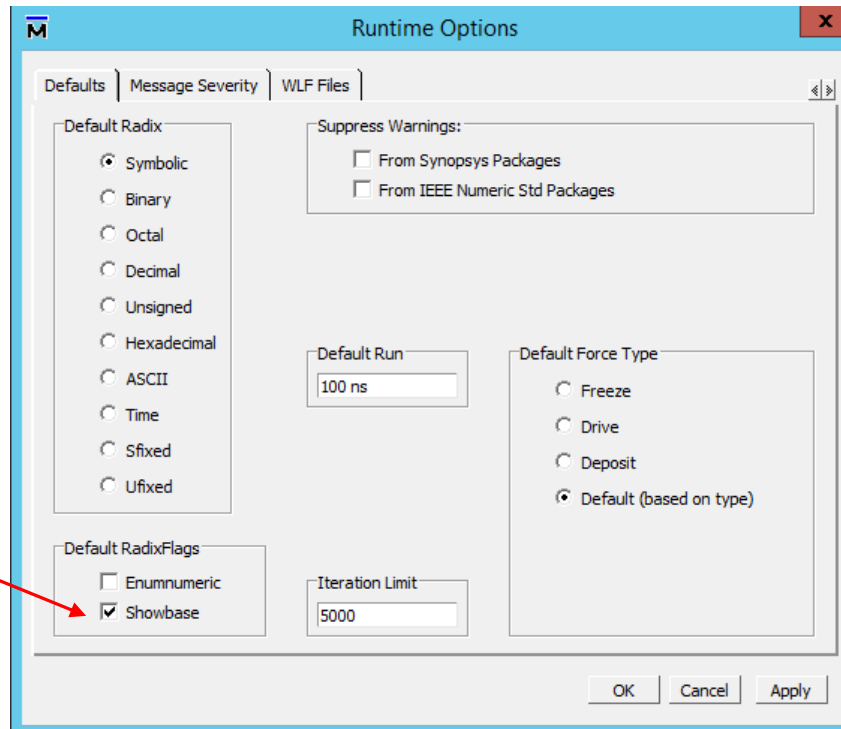


ModelSim Wave window

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:

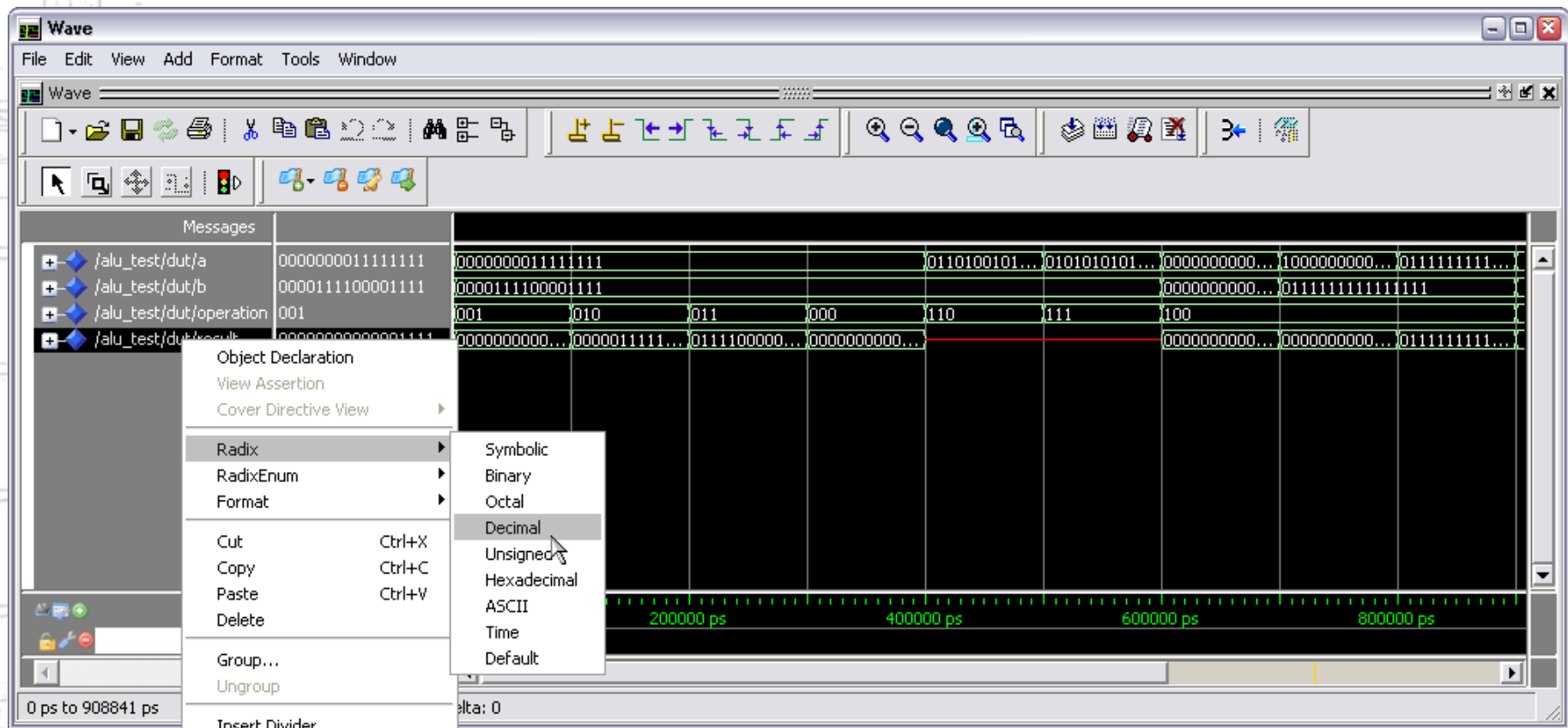
VSIM 6> run (100 ns simulation time)

VSIM 7> run 1000 ns

Waveform Viewing (1)

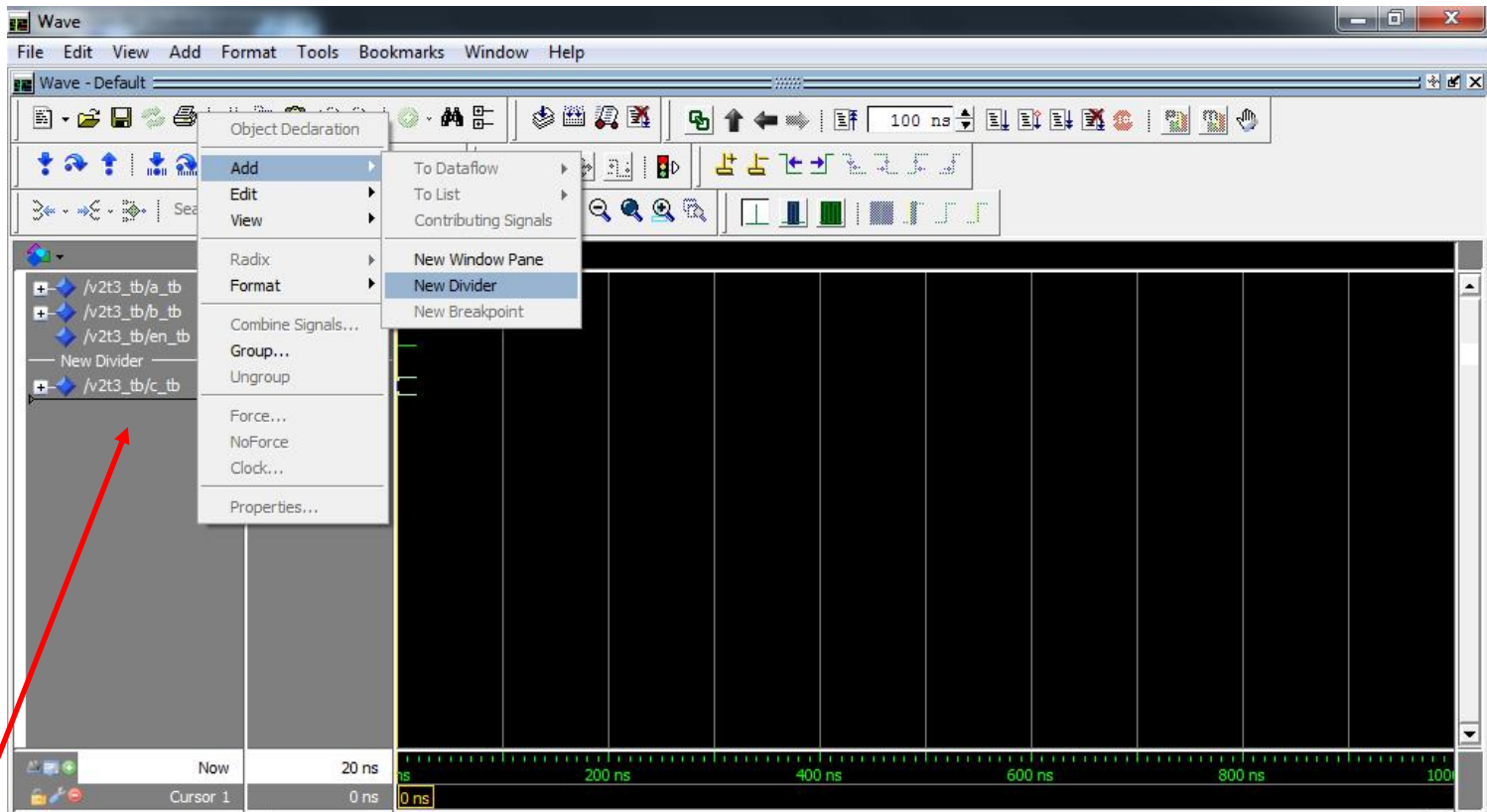
After running the simulation the signals now change their values in the waveform display – they are currently in symbolic format => for changing the radix press the right mouse button on the signal and select

- **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

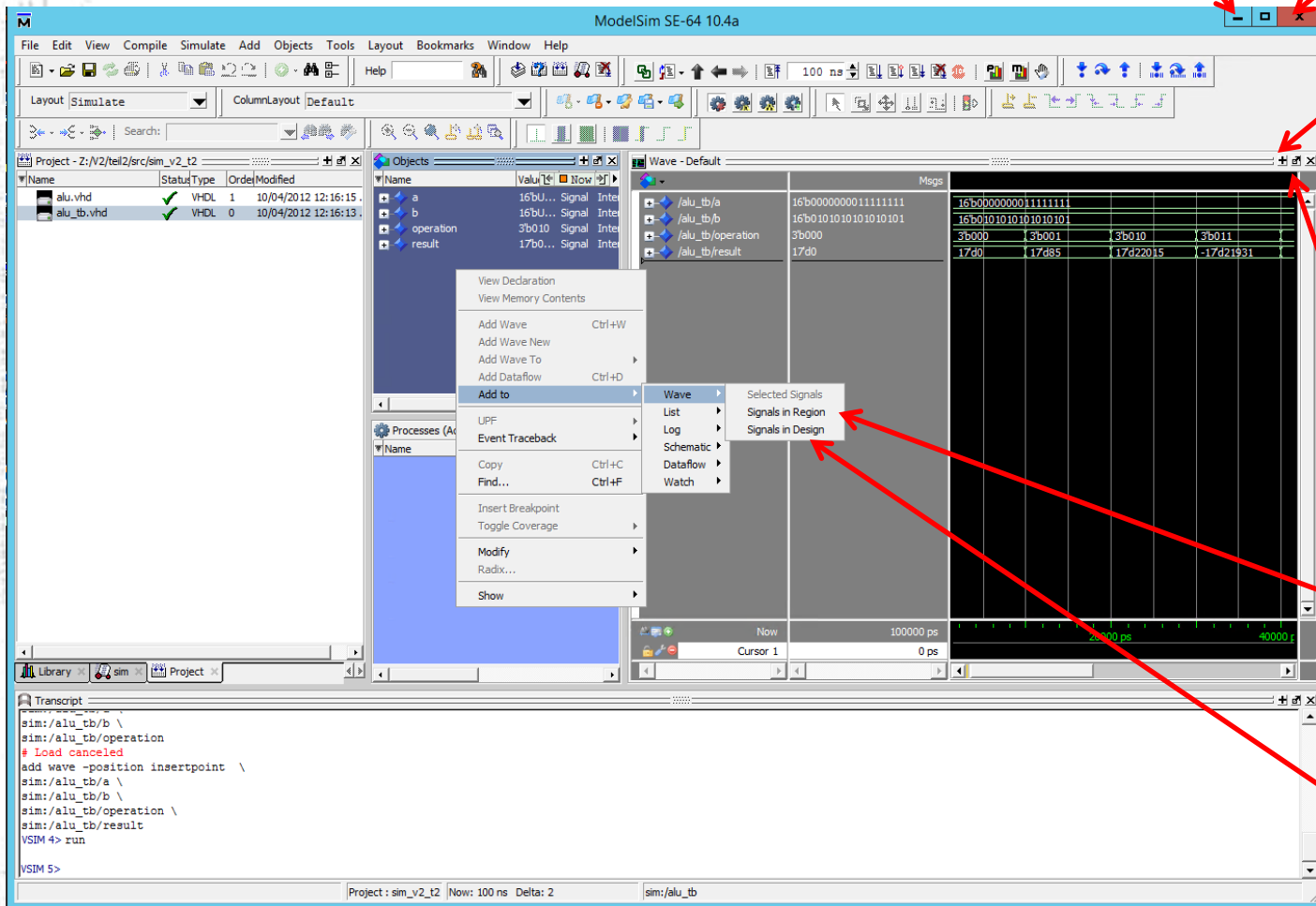
Enlarge
Window

Close
Window

Undock
Window

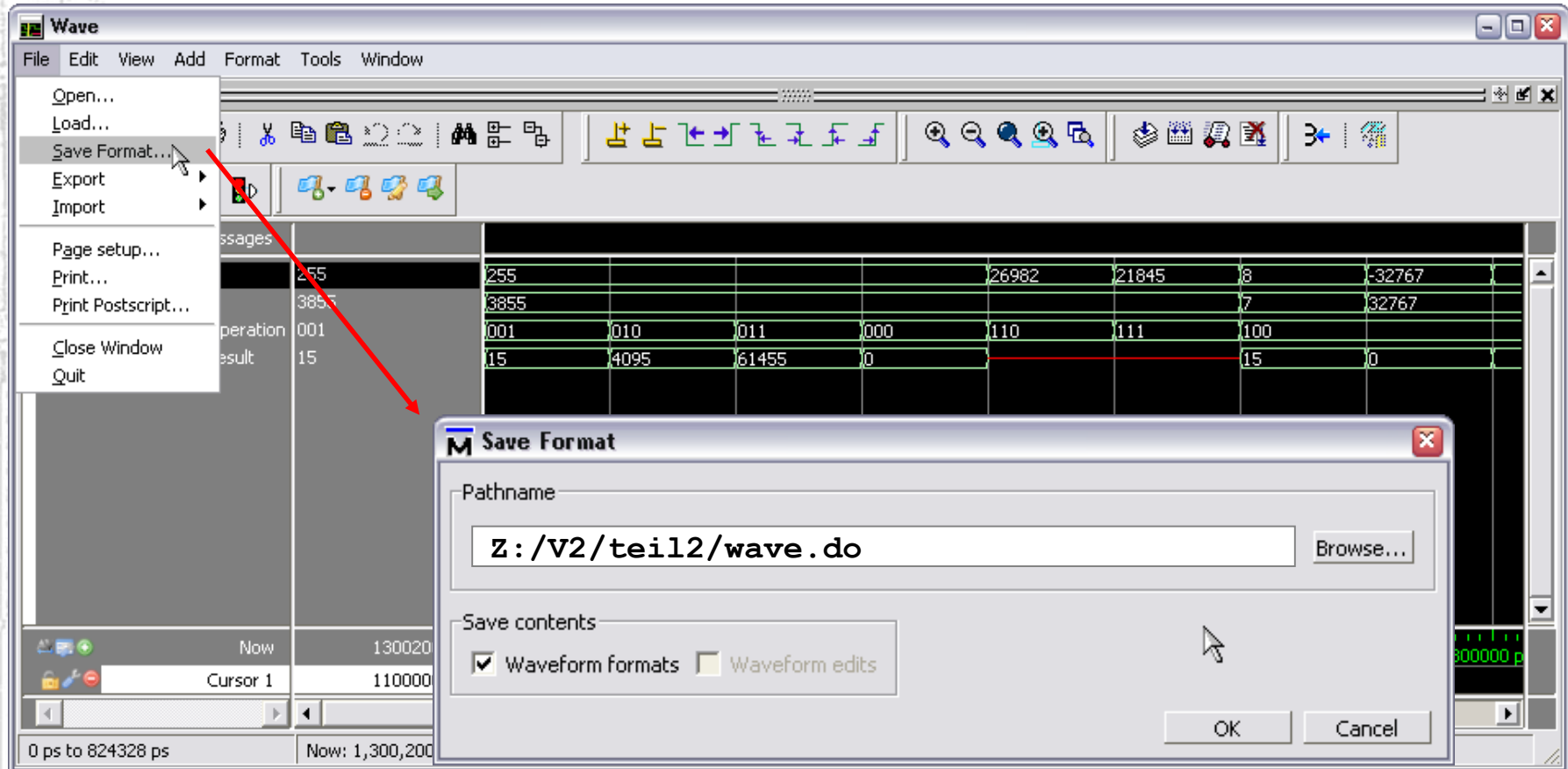
View all
signals
of object

View all
signals incl.
lower
hierarchies



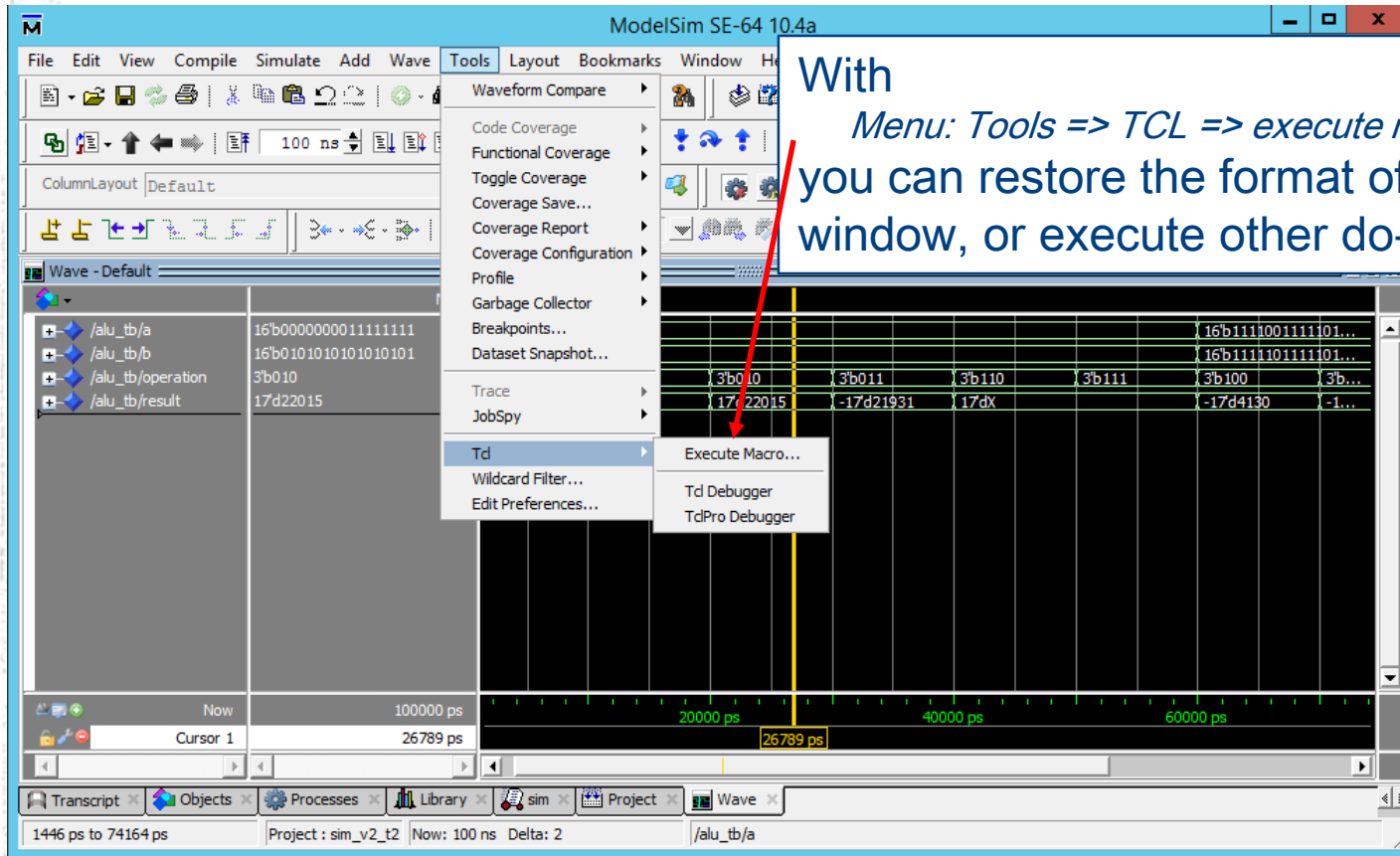
Saving Signal Setup (wave.do)

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)

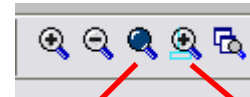
Zooming:



Select mode

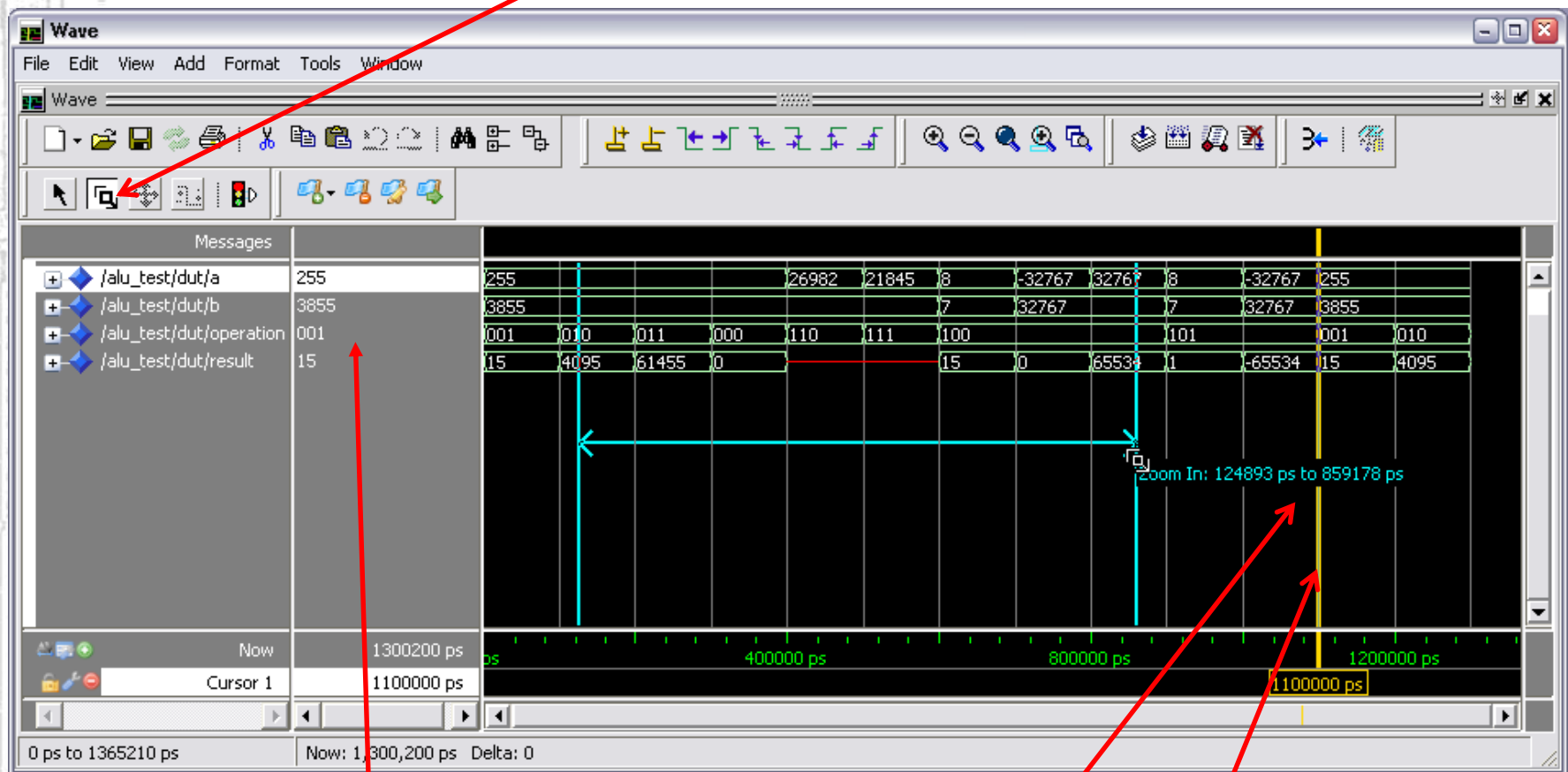
Zoom mode

Standard zooms:



Full view

Zoom to cursor



Signal values at time of active cursor position

Cursor

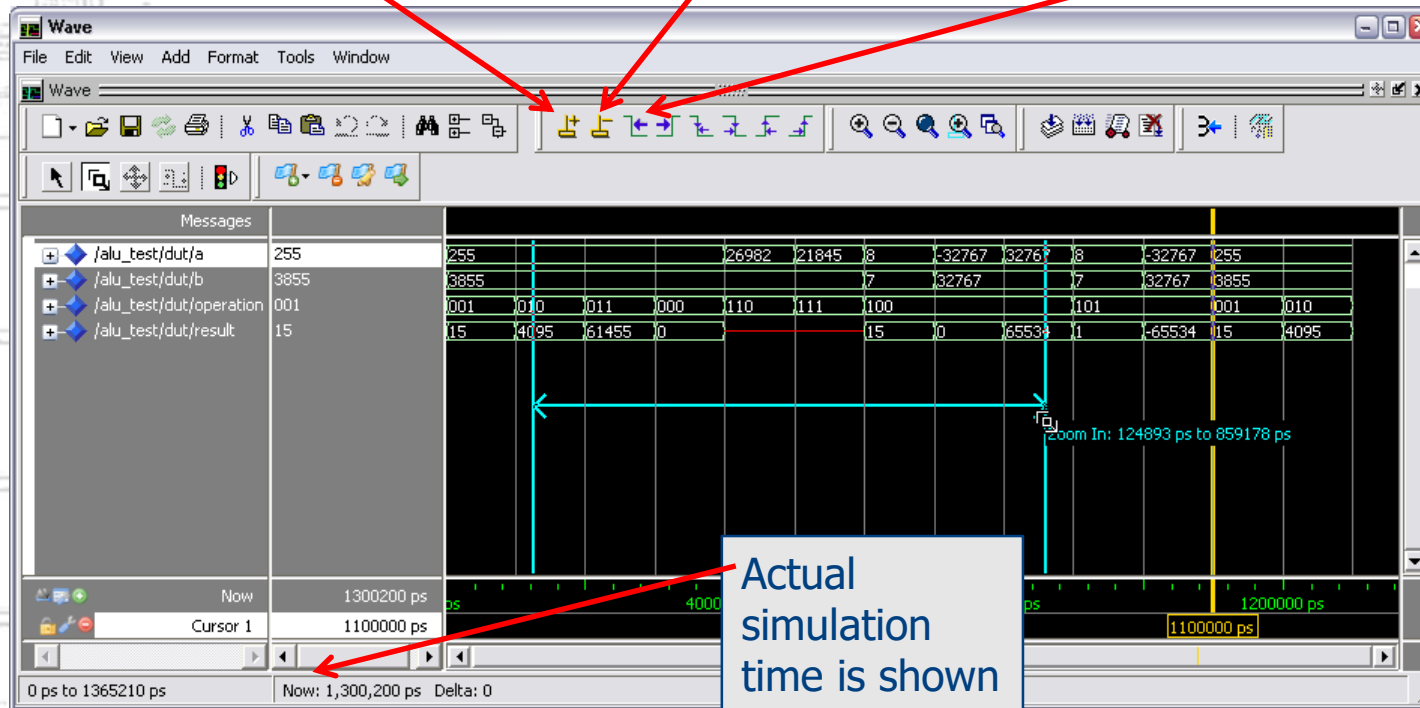
Further Waveform checking

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 (VHDL)

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:



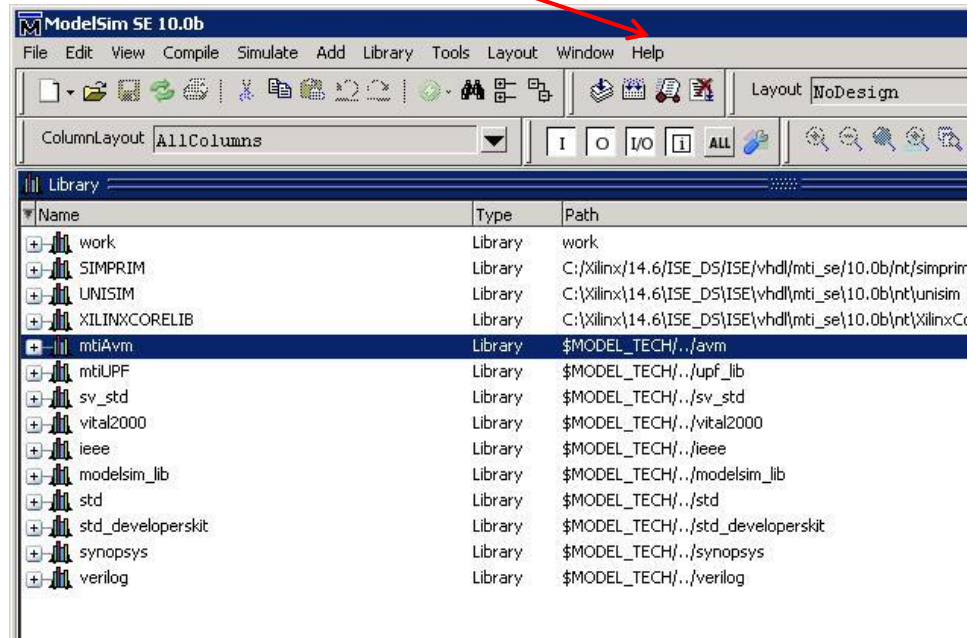
Useful Hints (1)

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.

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

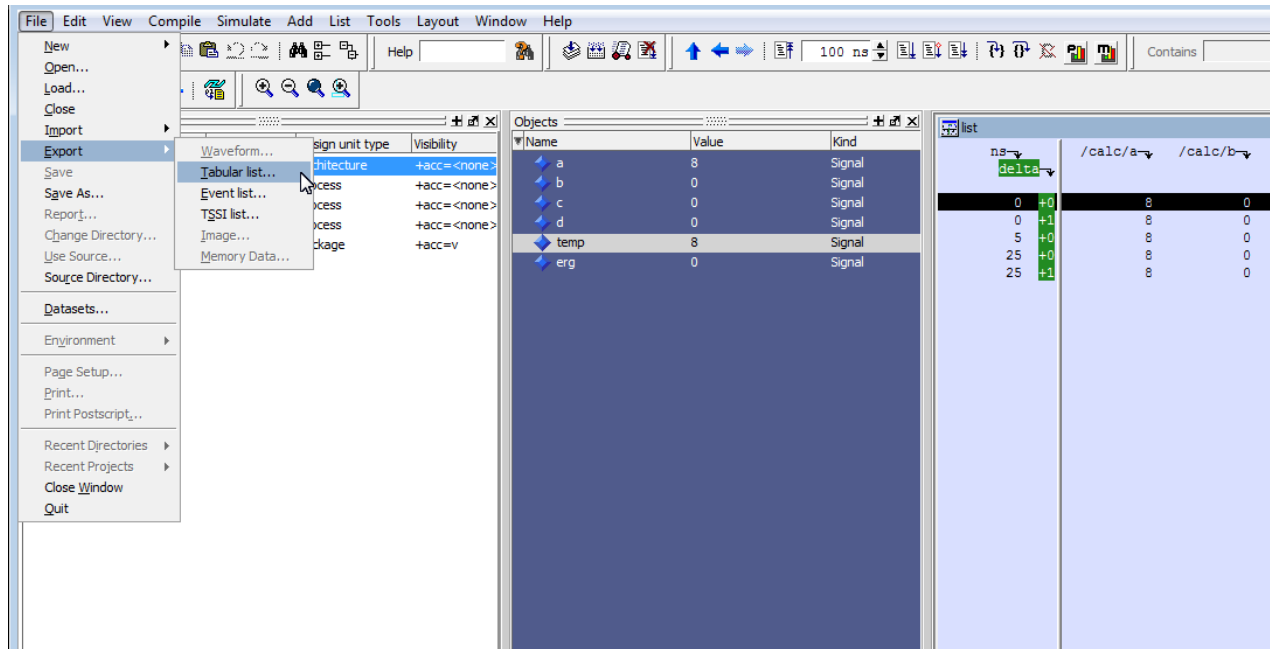
Start with each new exercise a new project.

Compiling everything into one work-library causes confusion.



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”).