SynthWorks℠

VHDL Training Solutions for hardware design and test.

Active-HDL™ Quick Reference

1. Steps to Running Active-HDL

Start the Simulator
Creating a Workspace
Adding Files to a Design.
Compile a Design
Load the Testbench
Display Waveforms
Run a Simulation

2. Start the Simulator

Double-click on the Active-HDL Icon and either
1) create a new one using dialog (directions follow)
2) open an existing workspace,
3) or close dialog and run a script (done in testbench class)

3. Creating a Workspace

If not continuing from the previous step, use the menu command, "File > New > Workspace" to start the dialog.

1st box: specify design name, working directory, and check the "Add New Design in Workspace" box.
2nd box: Check "Add Existing Resource Files" & press next.
3rd box: Browse to your VHDL source directory, select files, and press next.
4th box: Press Next.
5th box: Specify Design name and press next.
6th box: Press Finish.

This step adds files to a workspace and compiles them the first time. To add and compile additional files, see below.

4. Adding Files to Workspace

Do when updating design or adding testbenches. See also "Activating A Design".

Menu: Design > Add Files to Design

5. Compile a Design

First Compile or after adding files
Menu: Design > Compiles All with File Reorder

All other compiles:
Command: comp
Menu: Design > Compile All

6. Load the Testbench

Note make sure to compile all first.

Command: asim TbMemIO

Menu **Two Steps**
Step 1: Set the testbench to be the top of the design. To do this, in the Design Browser (left side window), select the "Files" tab and then in the library, select the testbench. Right-click on the testbench and select "Set as Top-Level"
Step 2: Load the Testbench. Select the menu command, "Simulation > Initialize Simulation".

7. Display Waveforms

Do before running simulation.

Command: add wave /*

Design Browser, Structure Tab: **One Step**
Right-click on the design under test in the "structure" tab of the design browser and select "Add to Waveform".

8. Run and Stop a Simulation

8.1 Run for Specified Time (most common)

Command: run 5 ms
Menu: Simulation > Run For
Specify Run Time in Box

8.2 Run Until

Menu: Simulation > Run Until
Enter amount of time in pop-up window

8.3 Stopping

Break Button: Red Circle with X through it.

8.4 Restarting a Simulation

Command: restart
Menu: Simulation > Restart Simulation

8.5 Run All

Menu: Simulation > Run

9. Waveform Viewer Details

9.1 Detaching the Waveform Viewer

Detach waveform viewer by pressing the ">>" symbol in the upper right corner of the simulation window.

9.2 Zooming

Zoom In, Out, and Fit is available with "View > Zoom >" menu. They are also available from the button bar using the Magnifying Glass icons.

9.3 Selecting Signals

Select one signal by clicking the left mouse button on its name.
To select multiple signals, either (just like windows)
Select one signal and then press shift-left mouse button to select a range
Select one signal and then add more signals by pressing ctrl-left mouse button.

9.4 Moving Signals

Select signal(s) and drag to new location in wave window.

9.5 Specifying Signal Radix (Hex, Unsigned, …)

The radix of the selected signal (or signals) can be specified using the "right click > Properties" menu command.

10. Running Script Files

Menu: Tools > Execute Macro

Note only *.do files are visible by default and you will have to use the dialog box to make either *.tcl files visible.

Since Aldec has a project style compile, the main use of a script like this is to initialize a project the first time (such as when a new member joins a project or the project is un-archived).

11. Manual Compile Order

Used for special circumstances.
Menu: Design > Compilation Order …

12. Activating a Design

When using multiple libraries, to add a file to a library, the library must be the active library.

Activate a library by right-clicking on the library (LIB_MEMIO) in the Design Browser and select "Set as Active Design".

After making the library active, files can be added and then the library recompiled (required when adding files).

13. Simulator Time Resolution

In Aldec, the simulator resolution is set to Auto by default. To change the simulator resolution, use the "Design > Settings" menu command, select the "Simulation", and set the simulator resolution in the box (currently with Auto in it).

14. Transitioning to the Next Lab

Working on the same design:
Add files to the current project as described in "Adding Files".

Starting a new design:
Use steps described in "Creating a Workspace".

SynthWorks Design Inc.
VHDL Hardware Synthesis and Verification Training
11898 SW 128th Ave. Tigard OR 97223 1-(503)-590-4787
http://www.SynthWorks.com jim@synthworks.com