

Verilog simulation with Modelsim

Opritoiu Flavius
flavius.opritoiu@cs.upt.ro

September 18, 2023

Verilog simulation with the `modelsim` environment

Objectives:

- ▶ Compile Verilog design with Modelsim
- ▶ Simulate a Verilog design using the Modelsim environment

Windows/Linux `modelsim` installers are available [▶ here](#).

Note: an Intel account is needed for downloading the installers.

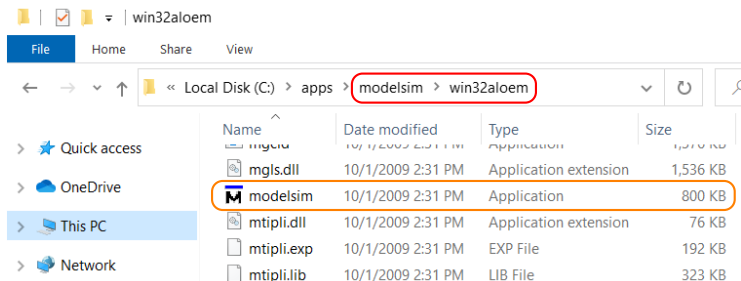
In these guides, the "Modelsim Altera Starter Edition 6.5b" version is used. This edition does not require a license.

Note: the Modelsim environment includes its own source file editor.

Portable modelsim installation for Microsoft Windows

A portable version of the modelsim environment, running on Microsoft Windows OS-es, can be downloaded from [here](#).

After extracting the archive, the Modelsim environment can be started by running "modelsim\win32aloem\modelsim.exe".



Docker-based `modelsim` instalation for macOS

A Docker image of the `modelsim` environment was built to be used on macOS computers. The procedure outlined in the following slides was tested on macOS Catalina, version 10.15.7.

Step 1: Install Docker Desktop for macOS as described [▶ here](#).

Step 2: Load the Modelsim Docker image:

- i. Start Docker Desktop for macOS
- ii. Download the archived image from [▶ here](#)
- iii. Open the Terminal app and, assuming the archive was saved in `~/Downloads`, run the following commands:

```
cd ~/Downloads
tar xJf ./msim_dking.tar.xz
docker load -i ./modelsim_v6.5.tar
```
- iv. (optional) Files `./msim_dking.tar.xz` and `./modelsim_v6.5.tar` can be deleted

Docker-based `modelsim` installation for macOS (contd.)

Step 3: Install and configure XQuartz:

- i. Install XQuartz as described [▶ here](#).

Note: accept installation's final request to log out

- ii. Start the XQuartz app. In the newly opened window (titled "xterm") go to `XQuartz menu` `Preferences` and in the Security tab make sure "Allow connections from network clients" option is enabled
- iii. Restart the computer

Step 4: Download the `start_modelsim_macos.command` script.

Note: Script can be placed anywhere (e.g. on Desktop, for convenience)

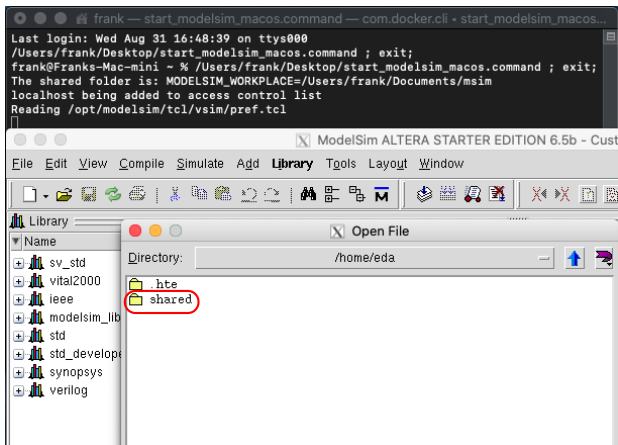
Step 5: Configure the folder to be shared with the Modelsim environment: open the script in a text editor (e.g. TextEdit) and, in line 3, change variable `MODELSIM_WORKPLACE` to indicate to a folder to be shared.

Important: The shared folder is accessible in the Modelsim environment at the `/home/eda/shared` path.

Docker-based modelsim installation for macOS (contd.)

For starting the Modelsim environment on macOS:

- i. Launch Docker Desktop for macOS
- ii. Launch the `start_modelsim_macos.command` script
- iii. The shared content is accessible through **File** **Open**, by selecting the "shared" folder



Portable modelsim installation for Ubuntu

A portable version of the Modelsim environment, running on Ubuntu OS, can be downloaded from [▶ here](#). This environment was tested on 64-bit Ubuntu 20.04 LTS and 22.04 LTS OS-es.

For 64-bit Ubuntu OS-es, the package's required dependencies can be installed with the following commands:

```
sudo dpkg --add-architecture i386
sudo apt-get update
sudo apt-get install libc6:i386 libncurses5:i386
sudo apt-get install libx11-6:i386 libxtst6:i386
sudo apt-get install libstdc++6:i386
```

The above commands are included in the "INSTALL" file from the root folder of environment's archive.

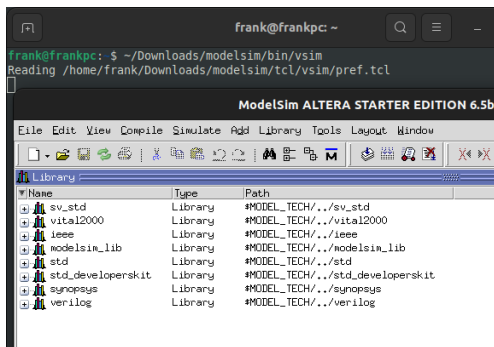
Portable modelsim installation for Ubuntu (contd.)

Assuming the archive was saved in `~/Downloads`, extract it with:

```
cd ~/Downloads && tar xJf ./modelsim.tar.xz
```

The Modelsim environment is started with:

```
~/Downloads/modelsim/bin/vsim
```



It might be possible to adapt this portable installation to other Linux distributions. Otherwise, a Docker-based Linux installation, presented in the next slides, could be used.

Docker-based `modelsim` installation for Linux

A Docker image of the `modelsim` environment was built to be used on Linux computers. The procedure outlined in the following slides was tested on 64-bit Ubuntu 22.04 LTS.


Step 1: Install Docker Engine for your distribution: [▶ link](#).

Step 2: Load the Modelsim Docker image:

- i. Download the archived image from [▶ here](#)
- ii. Open a terminal and, assuming the archive was saved in `~/Downloads`, run the following commands:

```
cd ~/Downloads
tar xJf ./msim_dking.tar.xz
sudo docker load -i ./modelsim_v6.5.tar
```
- iii. (optional) Files `./msim_dking.tar.xz` and `./modelsim_v6.5.tar` can be deleted

Docker-based modelsim installation for Linux (contd.)

Step 3: Download the `start_modelsim_linux.sh` . Assuming the script was saved in `~/Downloads`, mark it as executable:

```
chmod +x ~/Downloads/start_modelsim_linux.sh
```

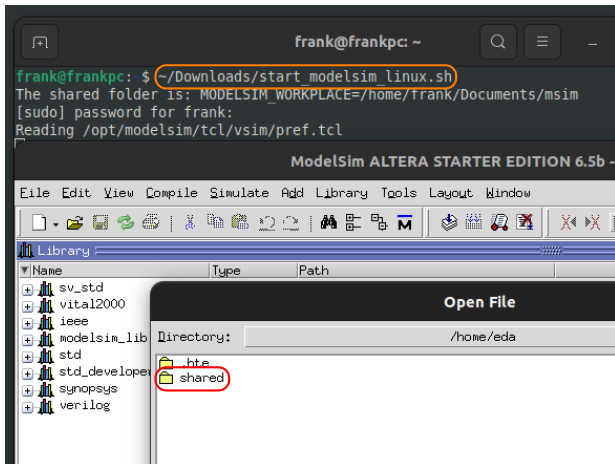
Step 4: Configure the folder to be shared with the Modelsim environment: open the script in a text editor (e.g. gedit) and, in line 3, change variable `MODELSIM_WORKPLACE` to indicate to a folder to be shared.

Important: The shared folder is accessible in the Modelsim environment at the `/home/eda/shared` path.

Docker-based modelsim installation for Linux (contd.)

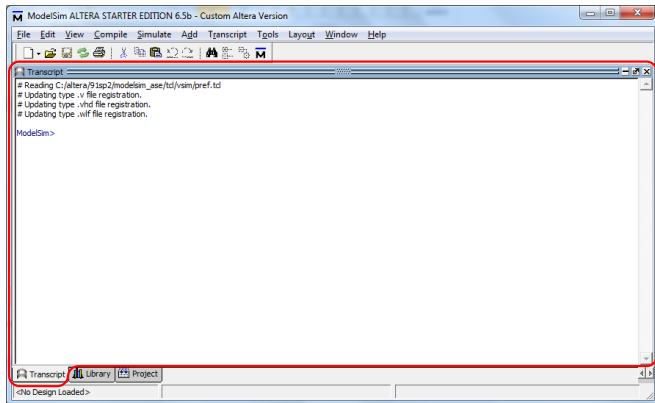
For starting the Modelsim environment on Linux:

- i. Run the `start_modelsim_linux.sh` script from terminal
- ii. The shared content is accessible through **File** > **Open**, by selecting the "shared" folder



The modelsim environment

The Transcript window is marked out in the image bellow:



All commands will be issued in the Transcript window

Prepare a modelsim project

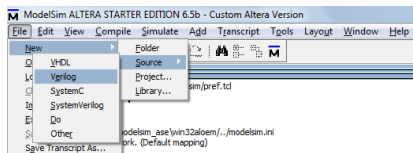
Select or create a folder to hold your project files. In this example, folder "d:\student\msim" will be used as the project's folder.

Change the Modelsim's working directory by issuing the following command in the Transcript window:

```
cd d:/student/msim
```

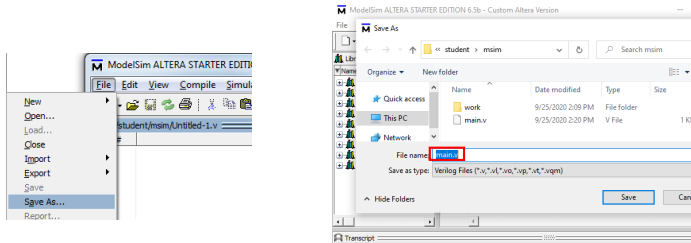
Writing Verilog code using modelsim environment

Use menu **File** **>** **New** **>** **Source** **>** **Verilog** for creating a new file.



Save the file using the **File** **>** **Save As...** menu before writing any code and name it *main.v* for this project.

Important: Don't forget to append extension ".v" to the filename: *main.v*



Verilog example code in modelsim

Add the following Verilog code to the file *main.v*.

```
1 module main;  
2     initial  
3         $display(" Hello ,_world!");  
4 endmodule
```

For compiling and simulating the Verilog module, download the *run_helloworld.txt* script available [▶ here](#).

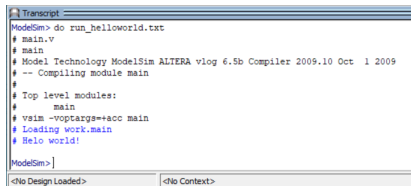
Compile and simulate in modelsim

Issue the following command in the Transcript window:

```
do run_helloworld.txt
```

The *run_helloworld.txt* script will compile the *main.v* source file and, afterwards, launch the simulation of *main* module.

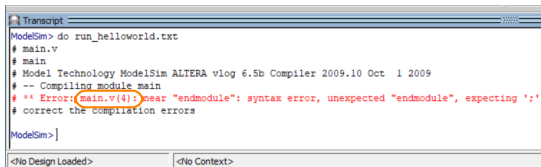
If no error occurred during compilation and simulation, the result in the Transcript window can be seen in the figure bellow:



```
Transcript
ModelSim> do run_helloworld.txt
# main.v
# main
# Model Technology ModelSim ALTERA vlog 6.5b Compiler 2009.10 Oct 1 2009
# -- Compiling module main
#
# Top level modules:
#   main
# vsim -voptargs==+acc main
# Loading work.main
# Helo world!
ModelSim>
<No Design Loaded>    <No Context>
```


modelsim compilation errors

The figure below presents the Transcript window after a compilation error occurred. Highlighted is the filename and an indication of the line containing the incorrect code



```
Transcript
ModelSim> do run_helloworld.txt
# main.v
# main
# Model Technology ModelSim ALTERA vlog 6.5b Compiler 2009.10 Oct 1 2009
# -- Compiling module main
# ** Error: main.v(4): near "endmodule": syntax error, unexpected "endmodule", expecting ';'
# correct the compilation errors

ModelSim> ]
<No Design Loaded> <No Context>
```

After correcting the implementation errors, the same command will be launched again from the Transcript window.