

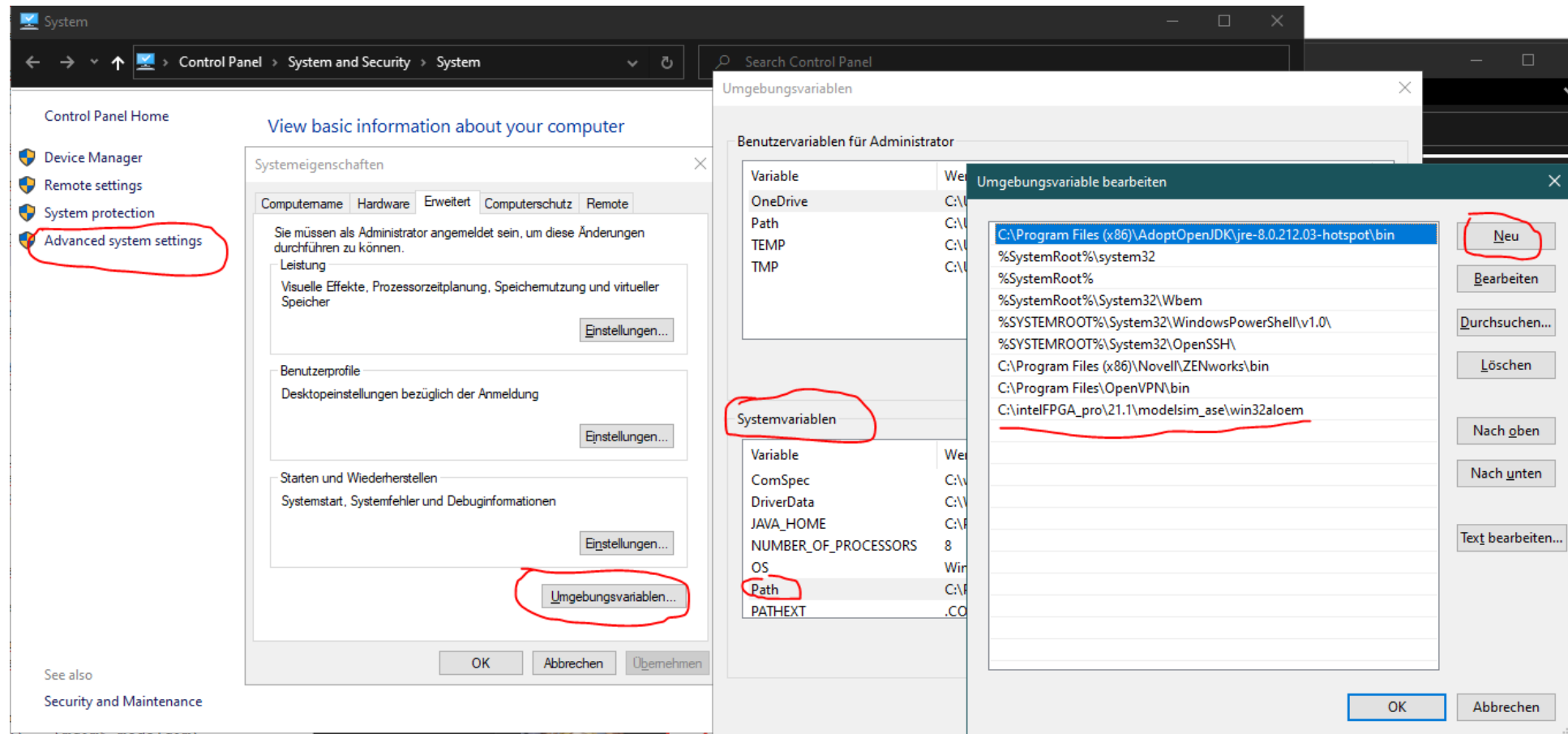
ModelSim Environment

ModelSim Environment

- You can download ModelSim Intel FPGA edition from
<https://cdrdv2.intel.com/v1/dl/downloadStart/660921/660958?filename=ModelSimSetup-20.1.1.720-windows.exe>
<https://cdrdv2.intel.com/v1/dl/getContent/661030/661059?filename=ModelSimSetup-20.1.0.711-linux.run>
- After the download, install “ModelSimSetup-20.1.1.720-windows.exe” (for Windows) or “ModelSimSetup-20.1.1.720-linux.run” (for Linux)
- Installation on Linux requires running these commands in your directory:
 - > `chmod +x ModelSimSetup-20.1.1.720-linux.run`
 - > `./ModelSimSetup-20.1.1.720-linux.run`

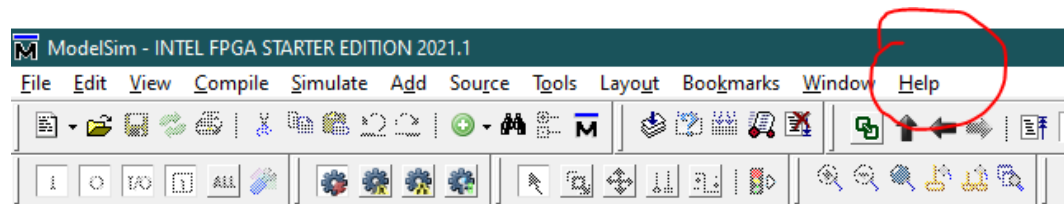
ModelSim Environment

- Make sure that ModelSim executable directory is added to your DOS path (an option may be given to you during the installation)



ModelSim Environment

- Once you have installed the program, you can start ModelSim graphical interface and access all documents under Help
 - You are strongly advised to "at least" read the Tutorial and Manuals under Help/PDF Documentation



- You can create a project or directly use ModelSim via "do" files (scripts) as given in the example implementations provided after the lab
- For each of your homeworks/projects create the following directory structure:
 - proj_dir** → **vlog** (directory for your Verilog-HDL sources)
 - **msim** (directory for your ModelSim files)

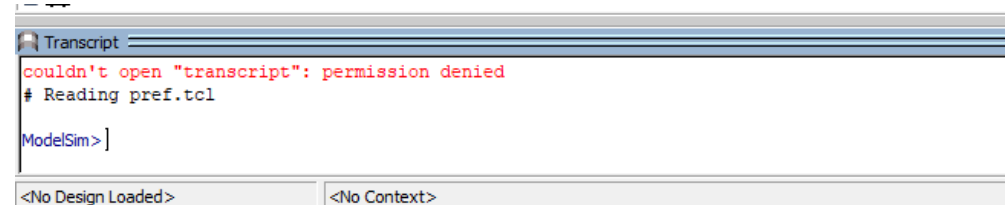
ModelSim Environment

Under **vlog**, place your Verilog-HDL sources. Under **msim**, copy the sim.do and wave.do files given with the examples. Normally, if you name your testbench top module (in the **<design>_tb.v** file) as test, you can use the sim.do file as is. Otherwise, edit the sim.do file and correct the name of the testbench top module to the name you picked. You will probably need to edit wave.do file anyway, since this is where you specify the input, output and internal nets you want to see on the waveform viewer

```
1 vlib work
2 vmap work work
3 vlog -work work ../vlog/*.v
4 vsim -voptargs="+acc" -t ns work.test
5 do wave.do
6 run -all
```

→ example "sim.do" file

Either start ModelSim by clicking on the Windows icon and then from ModelSim command editor (located at the lower part of the GUI) change to your **msim** directory (e.g., `cd C:/Users/username/Desktop/tutorial/low_latency/msim`)
Note that you can enter most Unix-type commands from the ModelSim command editor

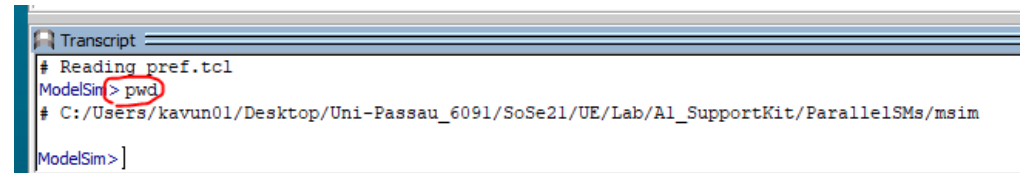


Or open a DOS command window. Change to your msim directory (`cd c:\...\msim`) and start ModelSim by entering the "modelsim" command from the command window (e.g. `c:\...\msim> modelsim`)

```
C:\Users\kavun01\Desktop\Uni-Passau_6091\SoSe21\UE\Lab\A1_SupportKit\ParallelSMs>modelsim
```

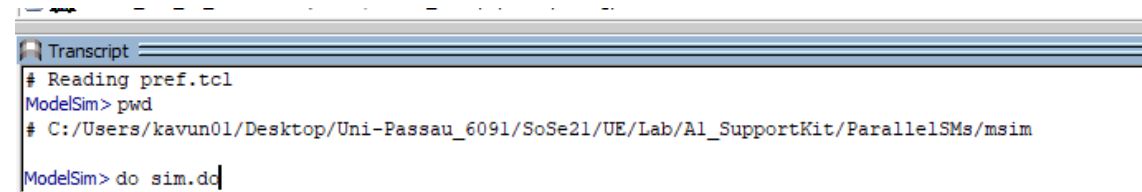
ModelSim Environment

Make sure you are in the correct directory, by typing "pwd" in the ModelSim command editor



```
Transcript
# Reading pref.tcl
ModelSim> pwd
# C:/Users/kavun01/Desktop/Uni-Passau_6091/SoSe21/UE/Lab/A1_SupportKit/ParallelSMs/msim
ModelSim>
```

From the ModelSim command editor, type "do sim.do". This will compile your source files, run simulation, open waveform viewer (you can pop the window out later if it does not happen directly, it will stay popped out after first run: option named "Dock/Undock") and plot all requested waveforms

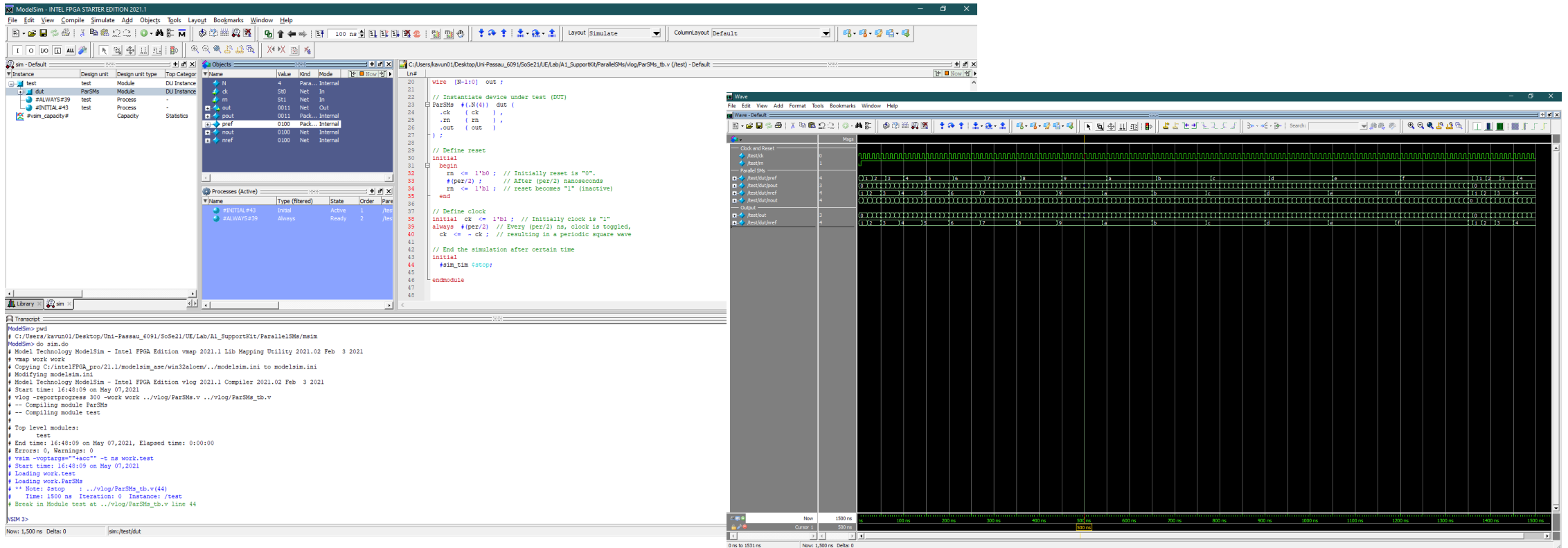


```
Transcript
# Reading pref.tcl
ModelSim> pwd
# C:/Users/kavun01/Desktop/Uni-Passau_6091/SoSe21/UE/Lab/A1_SupportKit/ParallelSMs/msim
ModelSim> do sim.do
```

In case of an execution error, carefully inspect the error/warning messages. Correct your HDL files. You can work on a separate code editor program (Notepad++, etc.) for convenience. However, ModelSim may ask you to confirm that the source file(s) is/are updated, in case any source file was open in the simulation tool window (i.e., this would probably happen if you modify the testbench file as it opens in ModelSim tool by default when you start the program). You should confirm these changes. After that, execute "do sim.do" again

ModelSim Environment

Other options are also available in ModelSim simulation tool, such as adding waves to your current simulation using the tool (i.e., modifying the wave.do file is not the only option for this) and running your simulation from scratch or step-by-step using the corresponding buttons in the tool. You can read the tutorial to learn these



If you want to switch to another directory to run a different design simulation, you need to quit current one by typing "quit -sim". You can then change to the corresponding **msim** directory as explained above