Lab 1

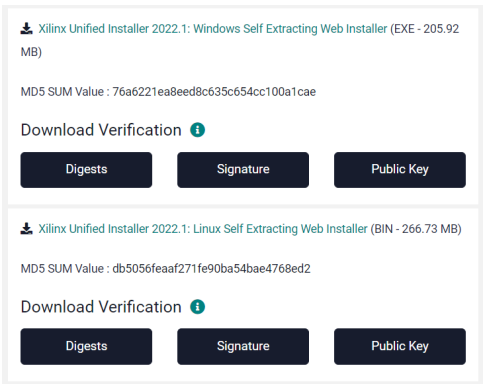
`Objective: To get started with Vivado.`

1 Getting Vivado

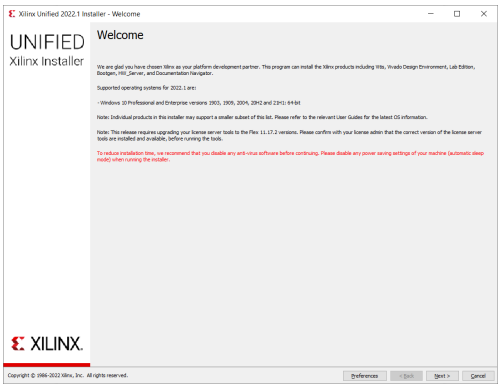
The main goal of this lab is to get yourself familiar with the software needed for this course. In this course, you will be using the Vivado Design Suite from Xilinx to design, simulate, and implement your hardware design. Vivado is a production-ready industrial software. As such, while it is extremely powerful and allows you to control every aspect of your hardware design, it can also be challenging to work with at times. Even it is a full feature software, Vivado software is available free for basic uses that is adequate for our course.

You may therefore either choose to install Vivado locally on your own computer, or to access the software remotely on our course server at `tux-2.eee.hku.hk`. Note that `tux-2` can be very busy, especially on the night before homework due. It is therefore recommended that you prepare to run the software on your own computer.

1.1 Install Vivado locally. Before you start, make sure you are going to install Vivado on Windows or Linux. For OSX, you can install a Linux virtual machine and run the Linux version of Vivado on the VM. Ask question on Piazza if you need help. The detailed version of the supported operating systems can be found in Xilinx document UG973.



(a) Download installer



(b) Run installer

Figure 1: Install Vivado

Download **Xilinx Unified Installer 2022.1** from Xilinx Download Center. Since it is a large download, we will also host a locally hosted copy of the installer. See link from Piazza. Regardless of where you download the installer, you should select one of the web installers based on your system. You will need to register a free account before you download on the Xilinx site.

Run the installer, then,

1. Click **Next**.

2. Verify your account, select **Download and Install Now** and click **Next**.
3. Select **Vivado** and click **Next**.
4. Select **Vivado ML Standard** and click **Next**.
5. Keep the default selections and click **Next**.
6. Agree and click **Next**.
7. Select installation directory and click **Next**.
8. Click **install** and wait for the installation to complete.

1.2 Connect to `tux-2` remotely. The server `tux-2` has been installed with the X2Go server for remote desktop login. To log in to a remote desktop on `tux-2`, you need to install X2Go client on your computer. You can follow the instruction in X2Go installation guide.

Open the X2Go client and create a new session with the following configurations:

- Session name: tux-2 (or whatever you like)
- Host: `tux-2.eee.hku.hk`
- Login: (your EEE account username)
- SSH Port: 22
- Session type: LXDE

Click **OK** and log in to the session with your EEE account password. You will see the desktop:

Figure 2: X2Go session

Your connection might be blocked by the firewall if you are not inside the HKU network. You can use HKU VPN when you are not on campus.

2 Running Vivado

In this section, you will be launching Vivado and running a simple simulation with the given source file. Before you start, you need to download the source file tarball from:

`http://www.eee.hku.hk/~elec3342//fa22/labs/elec3342lab1.tar.gz`

On Windows, you can untar it with 7-Zip. On Linux, you can untar it by running:

```
tar xzf elec3342lab1.tar.gz
```

2.1 Launch Vivado. On Windows, you should find some new shortcuts created during installation. Just double-click the Vivado icon to launch it.

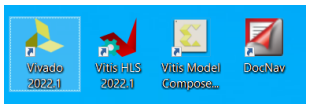


Figure 3: Xilinx shortcuts

On Linux, you need to launch Vivado from a terminal if the shortcuts don't show up. In this case,

1. Open a terminal on the desktop.
2. Set up the environment by running:

   ```
   source <install_path>/Vivado/2022.1/settings64.sh
   ```

   Replace `<install_path>` with the path you selected during installation. Or, if you are using `tux-2`, simply run:

   ```
   source ~elec3342/elec3342.bashrc
   ```

3. Launch Vivado from the directory `elec3342lab1`:

   ```
   cd elec3342lab1
   vivado
   ```

You should see the Vivado welcome page after launching:

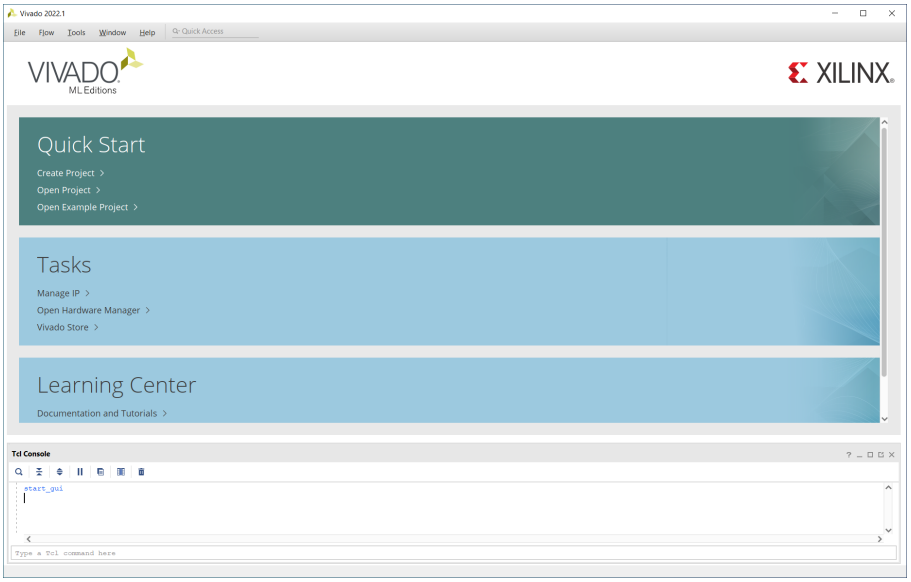


Figure 4: Vivado welcome page

2.2 Create project. Click **Create Project** under **Quick Start**. A wizard will show up to guide you through the process of creating a project. Then,

1. Click **Next**.
2. Modify project name as **lab1**. Select the directory `elec3342lab1` as project location and uncheck **Create project subdirectory**. Click **Next**.
3. Select **RTL project**. Make sure the two options under it are unchecked. Click **Next**.
4. Click **Add Files** and add `src/or_gate.vhd`. Uncheck **Copy sources into project**. Select **VHDL** as target language. Click **Next**.
5. There is no constraint to add. Just click **Next**.
6. Select **Basys3** under **Boards** tab and click **Next**.
7. Click **Finish**.

You should see the main page of Vivado now. You will be working on this page most of the time.

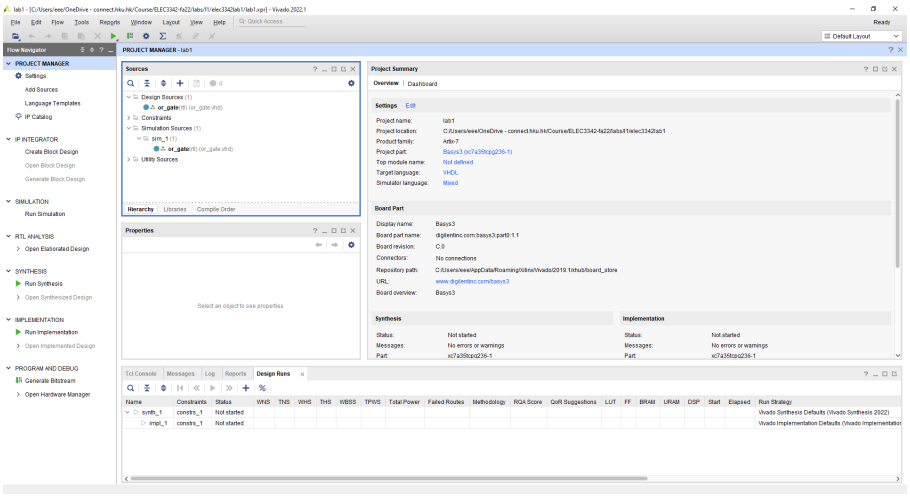


Figure 5: Vivado main page

2.3 Run simulation. Now the project is created. You can run the simulation and check the waveform.

Before you start the simulation, you need to add a simulation source to the project. Click the + button in **Sources** window. Then,

1. Select **Add or create simulation sources** and click **Next**.
2. Click **Add Files** and add `src/or_gate_tb.vhd`. Make sure **Copy sources into project** is unchecked. Click **Finish**.

Your **Sources** window should look like this:

Figure 6: Vivado **Sources** window

Now find **Run Simulation** in **Flow Navigator** on left-hand side.

Click on **Run Simulation** and select **Run Behavioral Simulation** to run the simulation. Wait until the blue cursor is pointing at `finish;` in text editor, meaning that the simulation is finished. Then click the **Untitled 1** tab and you will see the waveform.

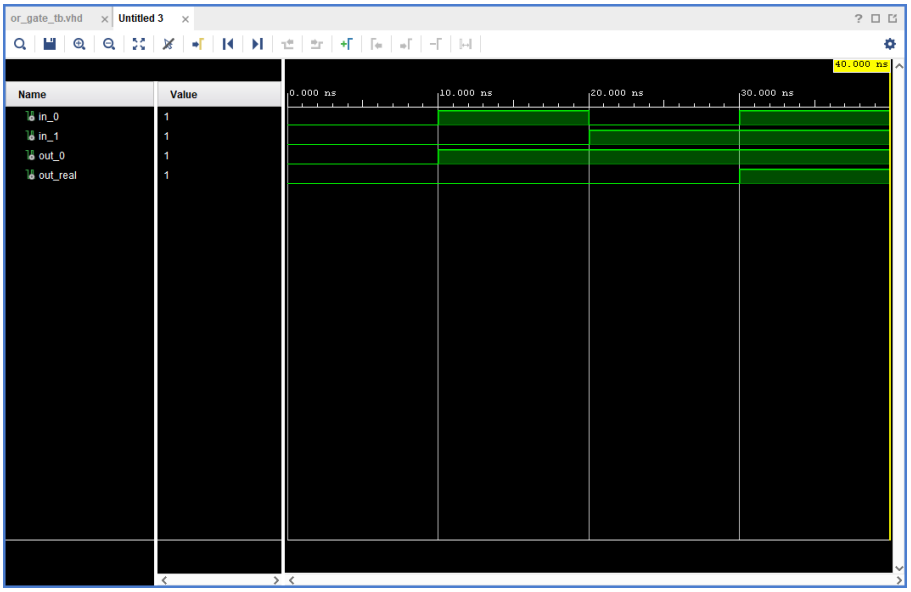


Figure 7: Simulation waveform

Find the **Scope** tab and right-click on the very top item. Then select **Log to Wave Database** → **Objects in Scope and below**. The waveform will be saved.

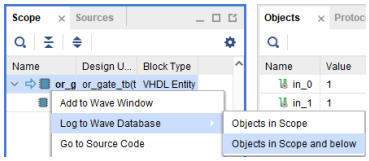


Figure 8: Save the waveform

Now exit the Vivado and check your `elec3342lab1` directory. You will find some new files and directories:

1. A file named `lab1.xpr`. This is the entrance of the whole project. You can simply double-click this file on Windows or run `vivado lab1.xpr` in the project directory on Linux to re-open this project.
2. Some directories named like `lab1.*`. The intermediate files generated by the project are saved in these directories.

2.4 Checkoff 1

Go to `lab1.sim/sim_1/behav/xsim` and find the waveform file `or_gate_tb_behav.wdb` and two log files `elaborate.log` and `simulate.log`. Compress them to a `zip` file:

```
<your-zip-file>
├─ or_gate_tb_behav.wdb
├─ elaborate.log
└─ simulate.log
```

Submit it in Moodle.