

ZedBoard Tutorial

EEL 4720/5721 – Reconfigurable Computing

Introduction:

In this lab, you will be learning how to create a custom peripheral in the programmable logic on the ZedBoard and how to communicate with that peripheral from software running on the ARM processor.

UPDATE: You will be working in groups of 2 on this project. Unless you are an EDGE student or have special permission from the instructor, you must work in a group of 2. Note that those in any overflow sections are *not* EDGE students, and must work in a group of 2. Please only submit once per group.

Part 1 – Base project and peripheral tutorials

- 1) Read the `base_project.pdf` and `peripheral_tutorial.pdf` link on the lab website. Step through these example yourself until you have the resulting bitfile from the peripheral tutorial. This will introduce you to the basic procedure for how to use Vivado and the Zynq processor.

Note that some of the steps are no longer needed and have changed, so I advise you to watch the lecture where I repeat the process to see what steps I skip.

IMPORTANT: Make sure to select the ZedBoard when creating the project.

The example tutorial uses a Microzed instead of the ZedBoard. If you follow that part of the instructions, your design won't work on our FPGAs.

If you have trouble getting the ZedBoard to show up in Vivado, see the workarounds on the lab website.

IMPORTANT: You do not need to use the specified multiplier entity from the base project tutorial. Create your own multiplier any way you want instead.

- 2) Find the generated bitfile in:

`project_path\project_name.runs\impl_1`

Note that `project_path` is the path of your Vivado project. `Project_name` is the name of that project. The bitfile will be called `block_diagram_name_wrapper.bit`, which is likely `design_1_wrapper.bit` if you accepted the defaults values when creating your block diagram.

Rename the bitfile to `part1.bit`. Place it in the same directory as the provided C++ code for `part1`.

- 3) Log onto `reconfig.hcs.ufl.edu` using an ssh client (e.g. putty) using your Gatorlink account information.
- 4) From the provided shell, run the following command:
`/usr/bin/zedboard/setup/setup_account.sh`
- 5) **Log off the server and then log back on.**
- 6) Use an scp program (e.g. WinSCP) to transfer the provided code with the bitfile into your account on `reconfig.hcs.ufl.edu`.

ZedBoard Tutorial

EEL 4720/5721 – Reconfigurable Computing

- 7) From reconfig.hcs.ufl.edu, compile the C++ code using “make”. Make sure there are no errors or warnings.
- 8) Run your code on a ZedBoard using:

```
zed_schedule.py ./zed_app part1.bit
```

zed_schedule is a scheduling script that connects to the board server and finds an available board. If a board is available, the script copies your current directory to the selected board and then runs the specified executable (e.g., ./zed_app). This application takes the name of the bitfile as a command line input, so make sure to include part1.bit.

- 9) Verify there are no errors. The output should look similar to this:

```
Starting job "./zed_app part1.bit" on board 192.168.1.102:  
Application completed successfully!
```

- 10) Take a screenshot of the software output.
- 11) Find and save the IP directory for your peripheral. This directory should have a component.xml file, in addition to various other directories (e.g., src, hdl, drivers, example_designs, etc.). Its location depends on the path of the IP repository that was specified when you created the peripheral.
- 12) Submission requirements: IP directory, screenshot of software output, and part1.bit. See submission instructions at the end of the document for details.

Part 2 – Create your own custom peripheral

In part 2, you will be using what you learned in part 1 to create a new peripheral.

- 1) Start a new project for the ZedBoard, following the same steps you did in part 1.
- 2) Create a new AXI peripheral called part2. On the Add Interfaces dialog box, **select 8 registers instead of the 4 registers you used in part 1.**
- 3) Download the provided entity peripheral_test.vhd. Instantiate this entity within the AXI peripheral code (part2_v1_0_S00_AXI.vhd) in a similar way as the multiplier in part 1. Registers 0-3 should connect to inputs 0-3. Registers 4-7 should connect to outputs 0-3. For the generic width, use a value of 32 bits to match the width of the registers.
- 4) Modify the AXI peripheral code to prevent registers 4-7 from having multiple sources by commenting out every assignment to those registers. This modification is required because in the generated code, the AXI bus writes to all the registers. We only want peripheral_test to write to registers 4-7, so we have to remove the generated code that writes to these registers. After these modifications, you are actually removing registers 4-7 and are just reusing those signals to act as wires. If you want, you could rename the signals to be more accurate, but this is not required.

- 5) Fill in the default architecture for `peripheral_test.vhd` with the following functionality. You can use either **signed** or **unsigned** for the arithmetic; the output is identical for these operations:
$$\text{out0} = \text{in0} * \text{in1} \quad (\text{ignore overflow by only using the lower width bits})$$
$$\text{out1} = \text{in0} + \text{in1}$$
$$\text{out2} = \text{in2} - \text{in3}$$
$$\text{out3} = \text{in2} \text{ xor } \text{in3}$$
- 6) Create a testbench `peripheral_test_tb.vhd` that demonstrates the correctness of `peripheral_test`. You do **not** need a simulation screenshot. It is up to you to determine the thoroughness of the testbench.
- 7) Synthesize the IP to make sure there are no errors.
- 8) Package your peripheral IP and connect it to the Zynq in the block diagram (see tutorials in part 1).
- 9) Generate a bitfile.
- 10) Find the bitfile and rename it to `part2.bit`. Copy it into the directory with the provided C++ code for `part2`.
- 11) Transfer the directory with the C++ code and the bitfile to `reconfig.hcs.ufl.edu`.
- 12) From `reconfig.hcs.ufl.edu`, compile the C++ code using “make”.
- 13) Run your code on a ZedBoard using:

```
zed_schedule.py ./zed_app part2.bit
```

At this point, there should be no errors for `out0` and `out1`. If there are errors, your peripheral is not working. IMPORTANT: `out2` and `out3` should be **incorrect** at this step.

- 14) Open `main.cpp` and find the code that transfers `in0` and `in1` to the ZedBoard. Extend the code to write `in2` and `in3` to the board. Make sure to use the correct AXI address (see the enum at the top of the file).
- 15) Find the code that reads results from `out0` and `out1`. Add the code that replicates this process for `out2` and `out3`.
- 16) Recompile using “make”.
- 17) Rerun the code on the ZedBoard. If there are still errors, there is either a problem with your peripheral or with the C++ code you added. Debug until there are no errors.

- 18) Take a screenshot of the working execution (or of the errors that you received if you don't have it working). e.g.:

```
Out0 Errors: 0
Out1 Errors: 0
Out2 Errors: 0
Out3 Errors: 0
Total Errors: 0
Application completed successfully!
```

- 19) Find and save the IP directory for part2_peripheral.

- 20) Submission requirements: IP directory, screenshot of software output, C++ code, part2.bit, peripheral_test.vhd, peripheral_test_tb.vhd, and part2_v1_0_S00_AXI.vhd. See submission instructions at the end of the document for details.

SUBMISSION INSTRUCTIONS (One submission per group)

Create a directory called lab2. Inside this directory, create a part1 and part 2 directory. Include a brief readme.txt file in the lab2 directory that specifies the members of the group, any other relevant information that the grader might need (e.g., if one of the parts doesn't work, crashes, doesn't compile, etc.). The submitted directory structure should look like this:

```
lab2/
  readme.txt
  part1/
    screenshot
    part1_1.0/          // IP core from repository
    part1.bit           // DON'T FORGET
  part2/
    screenshot
    part2_1.0/          // IP core from repository
    sw/
      main.cpp
      Board.h
      Board.cpp
      Makefile
      part2.bit         // DON'T FORGET
  hdl/
    peripheral_test.vhd
    peripheral_test_tb.vhd
    part2_v1_0_S00_AXI.vhd
```

Zip the entire lab2 directory and submit the lab2.zip file.

Extra Credit:

In `peripheral_test_tb.vhd`, create a testbench that exhaustively tests all possible input combinations for an instance of `peripheral_test` with a width of 4 bits. Because there are 4 inputs, a width of 4 corresponds to $2^4 = 16$ possible combinations. NOTE: Larger widths will take an extremely long time, so do not try this.

Common Problems:

- Do not create a bitfile for the base project tutorial. Only do it after adding a peripheral. Running a bitfile on the boards without the peripheral will lock them up and require a reboot.
- If you make a mistake in the IP core, you can't directly edit the code from within your project. This occurs because Vivado copies a read-only version of the IP core into your project. If you need to edit it, right click the core in the block diagram and select "Edit in IP Packager". This will open a separate project for the IP core. From the IP project, modify the code, re-package the IP after any changes, and then refresh the IP within your original project. This process will get very annoying, so make sure you have thoroughly tested your IP core code in simulation before packing it to use with the Zynq.

There actually is a way to edit the read-only copy of the core, but I don't recommend that yet. I'll explain how for later labs.

- Unless you make modifications to the code that are not explained in the instructions, you will receive a few synthesis warnings for your peripheral. E.g.:

```
[Synth 8-3331] design part2_v1_0 has unconnected port
s00_axi_awprot[1]
[Synth 8-3332] Sequential element
(\part2_v1_0_S00_AXI_inst/axi_awaddr_reg[0] ) is unused and
will be removed from module part2_v1_0.
```

Although for future labs I would encourage you to get rid of all warnings, these warnings will not cause any problems for this lab.