

# ELC 3338 Project Book

Steve Potter

September 17, 2020

# Contents

<b>1</b>	<b>Program Counter Register</b>	<b>3</b>
1.1	Program Counter Register . . . . .	3
1.2	Testbench . . . . .	6
1.3	Your Assignment . . . . .	7
<b>2</b>	<b>Program Counter Incrementer and Mux</b>	<b>9</b>
2.1	Adder . . . . .	9
2.2	Adder Test Bench . . . . .	10
2.3	Input Selection via Mux . . . . .	10
2.4	Your Assignment . . . . .	12
<b>3</b>	<b>Fetch Stage</b>	<b>14</b>
3.1	Instruction Memory Module . . . . .	14
3.2	Fetch Stage . . . . .	15
3.3	Your Assignment . . . . .	16
<b>4</b>	<b>Beginning to Decode</b>	<b>18</b>
4.1	Instruction Decode . . . . .	18
4.2	Register File . . . . .	20
4.3	Your Assignment . . . . .	22

# Lab 1

## Program Counter Register

During the course of this semester, we will build a 64-bit computer. To do this, we will make a synthesizable machine in Verilog, a common hardware description language (HDL).

A computer runs a program by executing individual instructions in sequential order. The instructions are stored in memory and are accessed by their memory address. During each clock cycle, an instruction is fetched from memory and executed on the processor. The memory address of the next instruction to fetch is stored in a register called the Program Counter (PC). During Lab 1, we will build and test the Program Counter register. In Lab 2, we build an incrementer (to count to the next instruction) and a mux (to select between the incremented count or a new starting value).

### 1.1 Program Counter Register

In order to make the Program Counter, we are going to make a Verilog module that explains how to build a register (a D flip-flop). Let me unpack the previous sentence:

1. Verilog is a Hardware Description Language (HDL).
2. We write Verilog code to tell Vivado how we want our register module to behave.
3. Vivado reads our Verilog code and synthesizes a realizable digital hardware design that meets the behavior that we specified. Thank you Vivado!
4. Vivado also simulates the behavior of the hardware, allowing you to test your design without building/programming hardware.

Consider the Verilog code in Listing 1.1. It is made up of three sections:

1. Header - this code includes a file name definitions.vh which contains information necessary for the program to run.

2. Port list (also known as interface) - specifies the signals coming into or going out of the module. In this case, there are three inputs and one output.
3. Body (also known as implementation) - describes the functionality of the module.

Listing 1.1: Verilog code to make a register.

```
'include "definitions.vh"

module register(
    input  clk ,
    input  reset ,
    input  ['WORD-1:0] D,
    output reg ['WORD-1:0] Q
);
always @(posedge( clk ) ,posedge( reset )) begin
    if ( reset==1'b1 )
        Q<=WORD'b0;
    else
        Q <= D;
    end
endmodule
```

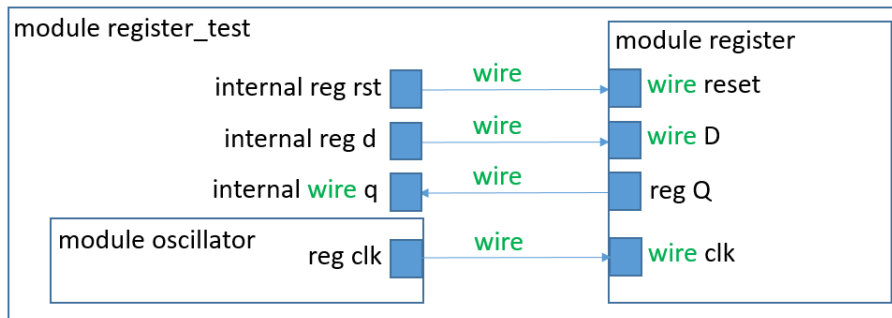
The first part is the header. We will use this same header each time. It tells the Verilog compiler to get all the data from a file called definitions.vh. The extension vh is a Verilog header. We use this to specify common pieces of data we will use across our design, so that all the components we build will be consistent. By putting them in one file, we make it easier to maintain, and prevent mistakes that can happen easily by having multiple copies of these basic pieces of data. For our first lab, one item that we will be using from definitions.vh is WORD (set to 64), which is the size (in bits) of the memory addresses in our computer. Note that if we build things based around WORD, rather than the number 64, we can just change the value of WORD in the file and get a computer with a different size with a couple key strokes.

The second part is the port list or interface. In this area we specify what signals are coming in (input), going out (output), or could go either direction (inout). Ports can be defined as either "wire" or "reg". This can be confusing to some students. Think of it this way:

#### 1. Wire

- (a) A wire is just a conductor that connects one component or module to another.
- (b) The value on a wire can only be changed by using combinational logic (as opposed to sequential logic).

Figure 1.1: Module Diagram.



- Internal regs are set in the 'initial' block or an 'always' block

- (c) It has no memory, meaning that the value on the wire is driven by the results of combinational logic at that particular moment.
- (d) Module inputs are always wires.
- (e) Module outputs can be wires or regs.

## 2. Reg

- (a) A reg more closely resembles a variable in software programming languages.
- (b) A value of a reg can only be set by using sequential logic.
- (c) A reg has memory, meaning that the value of the reg will remain the same until a sequential logic element updates it.
- (d) You can directly set a reg to a value using a procedural assignment.
- (e) Regs can be used internally in a module (neither input nor output), or they can be used as module outputs. They cannot be used as module inputs.

If you don't specify anything for the port type, you will get a wire - it is the default. In our case we have four signals: three inputs (always wires), and one output that is a register. The first two inputs are single-bit wires. One is the clock, which specifies the timing, and the other is reset, which clears the contents of the register (makes them zero). The final input is the value we want to store in memory, and I have called it D, following the convention of digital logic. D has multiple bits that are numbered from WORD-1 down to 0. Thus the leftmost bit is 63 in this case, and the rightmost bit is 0<sup>1</sup>. The output Q

<sup>1</sup>If you want to be technical this is called little endian, since the little end (the least significant or unit bit) is going into the first memory location (bit 0). If you reversed the order by putting the 0 first and the WORD-1 last it would be big endian, since the big end (most significant bit) would go in the lowest addressed bit.

(also the digital logic conventional name) is a register (it will hold its value) and should also be of size `WORD` and follow the same bit order as the input `D`.

To help clarify this, please examine Figure 1.1, which shows the interconnection of the modules in this lab.

The final section is the body or implementation. It is composed of a single thread of code, that will keep running (hence always). It will run one time every time there is a positive edge (0 to 1 transition) for either the clock or reset. Reset has higher priority, so if reset is asserted the register is cleared (`Q` is set to zero), otherwise the value of `D` is stored in `Q`. That is it. A nice, simple module. Please note that the provided register module is fully operational. You do not need to modify it. This file is provided for you in `ARM-Lab/code/1_fetch/register.v`.

## 1.2 Testbench

We now want to test our register module using System Verilog. System Verilog is very similar to Verilog, but it adds the ability to verify that we get the results that we are expecting. To test the register module, we need to tell the simulator to build a copy (instantiate) of the register module, and then we will need to supply the inputs and evaluate the outputs to verify that the module works correctly. Evaluate the testbench, `ARM-Lab/code/1_fetch/register_test.sv`.

When evaluating this testbench module (`register_test`), notice that there are no ports. A testbench is providing all the signals to simulate the inputs to the unit under test (UUT) and thus does not need ports. This is how Verilog finds a top level simulation module - there are no ports. The clock signal will be driven by a module named `oscillator`, which will give us a square wave with period `CYCLE`, which is another constant defined in our `definitions.vh` file. The code thus makes an oscillator and a register, then runs the 'initial' section (it runs once at the start then never again).

This initial section of the testbench follows a relatively simple pattern:

1. The inputs to the register module are set to particular values.
2. The system delays for some amount of time. For instance, a one cycle delay is inserted with `#CYCLE`.
3. The value of `er` (expected result) is set to the output value that you expect to get from the register module.
4. The value and size of `er` are compared with the value and size of `ar` (actual result). The actual result is the output of your register module. I provide (in "`verification_functions.sv`", included at the top of the testbench) the `verify` function and a few other functions that allow us to easily verify the behavior of our system. Each time the '`verify`' function is called, it reports and keeps track of whether the test passed (`ar == er`) or failed (`ar != er`).
5. At the end of the testbench, the `final_result` function is called to report the results of the test. This function will show the number of passing and failing test cases.

## 1.3 Your Assignment

You are to:

1. Evaluate the provided testbench. It is not an exhaustive testbench, but it tests a number of cases that commonly occur in our system. Note that you should not change the input values or timing of the testbench, nor should you add additional test cases to the testbench.
2. Create an Expected Results Table for your testbench. The idea behind the Expected Results Table is that you should identify how you think the system should operate. If you don't know how it should work, you will not know whether your simulation results are correct. The Expected Results Table should have a row for each signal in your simulation results (and the row order should match between your Expected Results Table and Simulation Results). The table should also have a column for each test case in the testbench. These test cases are the points in time that correspond to the 'verify' function calls in the testbench.

I have provided an Expected Results Table for Lab 1 in ARM-Lab/tables. It includes everything you need, except that the row for `er_q` is not filled in. Please fill in this row with the expected value of `q` for each test case. Note that you don't need to show the `clk` signal in the Expected Results Table.

3. The provided testbench does not set `er_q` to the correct value, therefore causing your test to fail. Your job now is to take the values from your expected results table and enter these values in the testbench as `er_q` (expected result of `q`).
4. Run a behavioral simulation. Evaluate the timing diagram and verify that it matches the Expected Results Table. Also evaluate the printouts in the Tcl Console window in Vivado. These printouts will indicate the number of passes and fails that occurred in the test. If you chose the correct `er` values and all tests pass, then your module is verified to work properly.
5. Produce a landscape mode PDF called `Lab1_lastname.pdf` that includes (in this order):
  - (a) Your name and the lab number.
  - (b) A snip (using the Snipping Tool) of your Expected Results Table.
  - (c) A snip of the Simulation Results (the black window that shows each signal). Make sure to show all values in decimal form and don't cut off the signal names on the left.
  - (d) The test results copied and pasted from the Tcl Console. The results should show the entire log from `BEGIN TEST RESULTS` to `END TEST RESULTS`.

- (e) I have included a sample, ARM-Lab/reports/Lab1\_Potter.pdf to give you an idea of what I'm looking for.
6. Upload Lab1\_lastname.pdf file to Canvas.
  7. Zip up your ARM-Lab directory and submit it on Canvas as well. Please make sure that you give me the ARM-Lab directory rather than the ARM-Project directory. I do not want the project files in the ARM-Project directory. Before you submit your zip file, make sure that it contains directories like code, manual, tables, etc. I will extract your zip file and run your code against my correct testbench to verify that your code and testbench work correctly. Since I give you working register.v code in this lab, this is pretty easy. In future labs, you must create your own module code.



## Lab 2

# Program Counter Incrementer and Mux

As mentioned in the last lab, the program counter is a register that is one word in length. It holds the address in memory of the next instruction to be fetched and executed. There are several ways that the program counter is updated:

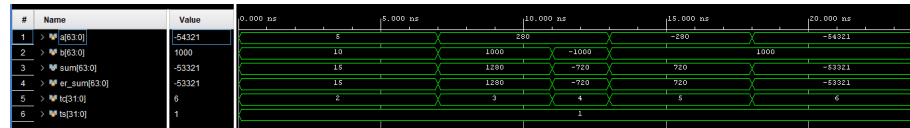
1. If the program does not branch (via an if statement, while loop, etc), then the program counter should advance to the next address (by adding 4 to the current PC) each clock cycle.
2. If the conditions of a conditional branch are met, then the program counter should be updated with the branch destination address.
3. If an unconditional branch or jump occurs, then the program counter should be updated with the branch destination address.
4. If an interrupt or error occurs, then the program counter should be updated with the interrupt or error handler address.

The instructions will be fetched in sequential order the majority of the time.

### 2.1 Adder

We will build a program counter incrementer by making a simple adder. Later in our computer we will need another adder, so we will re-use this code. Therefore, the adder code should be stored in `ARM-Lab/code/0_common` because it will be used in multiple stages. When used as the program counter, we will pass it a 4 because each instruction is 32-bits long (even though it is a 64-bit computer) and we want to increment to the next instruction in memory. Most machines are byte addressable, because one ASCII character (a `char` in `c/c++`) is a byte. For a machine with 32-bit instructions like we are using, that would mean that each instruction would be 4 bytes later in memory ( $32/8 = 4$  bytes). Therefore,

Figure 2.1: Adder Test Simulation Results



we will be adding 4 to the program counter each time we want to increment the program counter.

An adder is very simple in Verilog. There are two inputs (the two numbers to be added) and one output (the result). All the ports are size ‘WORD because they hold 64-bit integers.

In this lab you will make your own adder module. Your adder module should be called 'adder' and should have inputs of `a_in` and `b_in`. The output should be `add_out`. HINT: this should be very easy. Verilog is a Hardware Description Language, so use Verilog to describe what you want to do. Don't make it complicated. The adder code should be stored in `ARM-Lab/code/0_common/adder.v`. You will need to create this file.

## 2.2 Adder Test Bench

I have provided an incomplete test bench for the adder. While I put most of the infrastructure in place, you will need to fill in the following details. I labeled these spots with comments that start with TODO so that you can see where you need to do your work.

1. Create an instance of the adder module
2. Fill in the details of each test case so that your simulation results and test log match the results shown in Figure 2.1 and Figure 2.2. Pay careful attention to detail to make sure your results are identical to these figures. This includes making sure that the timing matches.

### 2.3 Input Selection via Mux

We will also need to be able to choose between normal advancing (sequential stepping) and branching (loops, if statements, etc.). We will use a multiplexor (mux) to do this. A mux is a simple device that connects one of the inputs to the output based on how the control bit is set. If the control bit is 0 then input a is connected to the output, and if the selector is 1 then input b is connected to the output. The heading and port list is provided for you in Listing 2.1. You just need to fill in the body to make it operate like a mux.

Figure 2.2: Adder Test Log

```

***** BEGIN TEST RESULTS *****

Test Case 1
Inputs: a = 5 | b = 10
+++ Step 1: Pass: |sum| time = 7 ns | er = 15 | ar = 15 | er_bits = 64 | ar_bits = 64 +++

Test Case 2
Inputs: a = 280 | b = 1000
+++ Step 1: Pass: |sum| time = 11 ns | er = 1280 | ar = 1280 | er_bits = 64 | ar_bits = 64 +++

Test Case 3
Inputs: a = 280 | b = -1000
+++ Step 1: Pass: |sum| time = 13 ns | er = -720 | ar = -720 | er_bits = 64 | ar_bits = 64 +++

Test Case 4
Inputs: a = -280 | b = 1000
+++ Step 1: Pass: |sum| time = 18 ns | er = 720 | ar = 720 | er_bits = 64 | ar_bits = 64 +++

Test Case 5
Inputs: a = -54321 | b = 1000
+++ Step 1: Pass: |sum| time = 24 ns | er = -53321 | ar = -53321 | er_bits = 64 | ar_bits = 64 +++

Pass Count = 5
Fail Count = 0

***** END TEST RESULTS *****

```

One interesting addition in this block of code is the addition of a size parameter. Parameters allow you to pass a value into a module, making the module more flexible and reusable. In the mux code, the parameter is specified by  `#(parameter SIZE=8)`. Parameters are constants, so they cannot be explicitly changed within the module. Rather, they are specified when an instance of this module is created. In the provided `mux_test.sv`, the mux module is instantiated by:  `mux #(‘WORD) UUT_64`. Note that ‘WORD is a macro (from `definitions.vh`) that defines another name for the value 64. So the test bench passes in a parameter value of 64, and the mux module uses 64 anywhere the term SIZE is used. If no parameter is specified when the instance is created, then the parameter will be assigned the default value. `SIZE = 8` defines the default value for the SIZE parameter in this module.

Note parameters are constants and cannot be changed later in the module. In this lab, we are using parameters to set the number of wires that compose the inputs and output. In our lab project, we will need some muxes to switch entire words (64 bits) like this one, but later we will also need to switch register addresses (5 bits). Rather than write two muxes, we will make one and then use the parameter to change the size when they are declared. The mux code should be stored in `ARM-Lab/code/0_common/mux.v`.

The provided testbench instantiates both a 64 bit mux and a 5-bit mux so that two different mux sizes can be tested, ensuring that the mux works now and will work when we use it later in the semester.

Listing 2.1: Verilog code to make a mux.

```
'include "definitions.vh"

module mux#(
    parameter SIZE=8)(
    input  [SIZE-1:0] a_in ,
    input  [SIZE-1:0] b_in ,
    input  control ,
    output [SIZE-1:0] mux_out
    );

    // TODO: Add body of mux here

endmodule
```

## 2.4 Your Assignment

You are to:

1. Create an adder module.
2. Update adder\_test.sv as described above.
3. Fill in the body of the mux module in mux.v.
4. Use the provided mux\_test.sv to verify that the mux works properly. Note that you cannot/should not make any changes to the test bench. The correct results are already in the test bench.
5. Produce a landscape mode PDF called Lab2\_lastname.pdf that includes (in this order):
  - (a) Your name and the lab number.
  - (b) A snip of the Simulation Results for the adder\_test. Make sure to show all values in decimal form and don't cut off the signal names on the left.
  - (c) The adder\_test results copied and pasted from the Tcl Console. The results should show the entire log from BEGIN TEST RESULTS to END TEST RESULTS.
  - (d) A snip of the Simulation Results for the mux\_test. Make sure to show all values in decimal form and don't cut off the signal names on the left.
  - (e) The mux\_test results copied and pasted from the Tcl Console. The results should show the entire log from BEGIN TEST RESULTS to END TEST RESULTS.

6. Upload Lab2\_lastname.pdf file to Canvas.
7. Zip up your ARM-Lab directory and submit it on Canvas as well. Please make sure that you give me the ARM-Lab directory rather than the ARM-Project directory. I do not want the project files in the ARM-Project directory. Before you submit your zip file, make sure that it contains directories like code, manual, tables, etc. I will extract your zip file and run your code against my correct testbench to verify that your code and testbench work correctly.

## Lab 3

# Fetch Stage

We are ready to build our fetch stage. To do this, we will make one more module, our instruction memory. Then we will make a module to assemble all of our modules together into a working fetch stage.

### 3.1 Instruction Memory Module

Instructions are stored in memory and are accessed by using the address where they are stored. You can think of memory like a giant hotel for our data. Each piece of data is stored in a room (memory location), which we can find by its room number (memory address). To get a piece of data stored in memory (like an instruction) we need to take its address, go to that location, and grab the data. In Verilog, a bunch of memory locations that are accessed by an address is called an array. Arrays in Verilog are declared like they are in C; the data type is specified, then the name, then the array size.

To store the instructions, we will need an array of 32-bit numbers (definitions.vh defines `INSTR_LEN` as 32, please use this macro), which means the data type must be `reg['INSTR_LEN-1:0]`. After the name is specified (imem in this case), we are going to use a parameter called `SIZE` to specify how many elements the array has: `[SIZE-1:0]`. Therefore, your array will be defined as `reg['INSTR_LEN-1:0] imem [SIZE-1:0]`.

Now we need to populate this array with instructions. Rather than populating the array element by element, we will read the instruction values in from a file called `instrData.data` that I have provided in the testfiles area of the ARM-Lab repository. Note that we are just initializing the array with the values from the file. The values should only be read from the file once at the beginning of the simulation. Then we will access the `imem` array for an instruction value. To read the file and put the contents into our `imem` array, we will use `$readmemb` to read in hexadecimal values from `instrData.data`. `$readmemb` could be used if we chose to format our data file in binary rather than hexadecimal. It is very important that we use a macro in `definitions.vh` for the name and path of the in-

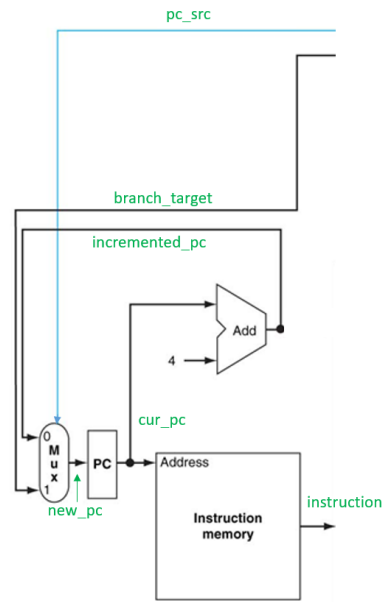
strData.data file, as this might change and it will be necessary for grading. Make sure to update the definition of IMEMFILE to point to your ARM-Lab/testfiles directory (as opposed to the potter/ARM-Lab/testfiles directory). The line of code to read the data in from the file is `$readmemh('IMEMFILE', imem);`

Once the array is populated, we can access imem and provide the instruction that corresponds to the requested address. Instructions should only be updated on the positive edge of the clock. I have provided a complete testbench for this module. I have also provided a partially complete file called instr\_mem.v in code/1.fetch directory. Complete the module code to make a working instruction memory module. Test it against instr\_mem.test.sv and verify that the module works as expected. The test will compare your instruction output to the instruction values that I provided in instrData.data.

## 3.2 Fetch Stage

Now we need to connect our modules together to make a fetch stage. The components of our instruction fetch (sometimes called ifetch or just fetch) stage are shown in Figure 3.1.

Figure 3.1: Instruction Fetch Stage.



Any wire (or reg) that comes into or goes out of the figure are input or output ports of the iFetch module. In Figure 3.1, the blue wire is a control signal and comes ultimately from the control unit, which you will build in the decode stage. Wires (or regs) that are completely contained in the figure are local to the iFetch module and are thus defined internally in the module (not an input or output). The one exception to this is the current program counter (cur\_pc). While there is no reason (at this point) that it must be output from the iFetch module, you must still make it an output so that it shows up on your simulation results, helping you to keep track of the program counter for the instruction that is currently executing. And it is required to verify functionality with the testbench.

While the input and output signals are easily identified by the diagram, you must also determine the size of each signal and whether it is a wire or reg. When you look at the figure I cut from a figure in the book, note that I labeled every wire on the diagram in green. For the sake of consistency

and debugging, it is required that you use these names.

**IMPORTANT NOTE:** Throughout your entire project, your signal names should follow the convention of the Freescale Semiconductor Verilog guide, which states that signal names should be all lower case, with words separated by an underscore.

Once you have figured out all your connecting signals (wires and regs), you should identify the components you are going to use. We have already created the modules, so now we just need to tell Verilog to instantiate them in the iFetch module and connect them together. Create the iFetch module in a new file called iFetch.v in the code/1\_fetch directory.

I have provided a complete testbench called iFetch\_test.sv. Use this testbench to determine if your fetch stage is working properly. As we progress through this lab project, you will learn how critical timing is. Please look at the cur\_pc value and the instruction value and verify that the instruction that was fetched is the correct instruction, according to instrData.data and the current program counter. Note that no instruction should be fetched in the first 5ns, as this is a half clock cycle and does not have a rising edge.

I have included a file called delay.v in code/0\_common. It includes a module that inputs a clock signal and outputs a clock signal that is delayed by some number of ns. This will be useful when resolving timing issues. Throughout this entire lab project, the delay module should only be instantiated in the top-level module (testbench). If you need the output of the delay function in a lower-level module, add a port to the lower-level module and pass it through that port.

### 3.3 Your Assignment

You are to:

1. Complete the instruction memory module.
2. Verify the module by running against the provided testbench.
3. Create the iFetch stage.
4. Verify the module by running against the provided testbench.
5. Produce a landscape mode PDF called Lab3\_lastname.pdf that includes (in this order):
  - (a) Your name and the lab number.
  - (b) A snip of the Simulation Results for the instr\_mem\_test. Please show all values in decimal except for the instructions. Please show instructions in hex.
  - (c) The instr\_mem\_test results copied and pasted from the Tcl Console. The results should show the entire log from BEGIN TEST RESULTS to END TEST RESULTS.



- (d) A snip of the Simulation Results for the iFetch\_test. Please show all values in decimal except for the instructions. Please show instructions in hex.
  - (e) The iFetch\_test results copied and pasted from the Tcl Console. The results should show the entire log from BEGIN TEST RESULTS to END TEST RESULTS.
6. Upload Lab3\_lastname.pdf file to Canvas.
  7. Zip up your ARM-Lab directory and submit it on Canvas as well. Please make sure that you give me the ARM-Lab directory rather than the ARM-Project directory. I do not want the project files in the ARM-Project directory. Before you submit your zip file, make sure that it contains directories like code, manual, tables, etc. I will extract your zip file and run your code against my correct testbench to verify that your code and testbench work correctly.

## Lab 4

# Beginning to Decode

### 4.1 Instruction Decode

The next stage in the datapath is the iDecode stage. The iDecode stage evaluates the binary instructions (an output of the iFetch stage) and determines what needs to be done. There are many aspects to the iDecode stage, and some get fairly complex. But today we will begin the process of decoding an instruction by decomposing the instructions into the key parts of R-Type and D-Type instructions:

1. opcode
2. address (used only in D-Type instructions)
3. rm\_num (used only in R-Type instructions)
4. rn\_num
5. rd\_num (though the book uses Rt for D-type instructions, we will use Rd for the last operand of D-type instructions)

To do this, you will create a new module called `instr_parse` in the `2.decode` directory. This module will simply read an input and assign appropriate output values. These outputs should be assigned using continuous assignments. The input is a 32-bit instruction. Outputs are listed for you above. Although R-type and D-type instructions have different operands, you can treat them the same for now. For instance, you can still assign an Address field on an R-type instruction, and you can still assign an Rm field on a D-type instruction. When we create the Control Module in a future lab, the control signals will drive what fields of the instruction are used and what fields are ignored. Notice how, because of the commonality of instruction format, Opcode, Rn, and Rd are all universal across these instruction types. Please remember to use the style specified in the previous lab, where all items are lower case with underscores separating them. For instance, for Rd, you should use the signal name `rd_num`. Appending num

Figure 4.1: Instruction Parse Test Output

```

***** BEGIN TEST RESULTS *****

Test Case 1
LDUR X9, [X22, #240]
+++ Step 1: Pass: |rn_num| time = 2 ns | er = 22 | ar = 22 | er_bits = 5 | ar_bits = 5 +++
+++ Step 2: Pass: |rd_num| time = 2 ns | er = 9 | ar = 9 | er_bits = 5 | ar_bits = 5 +++
+++ Step 3: Pass: |address| time = 2 ns | er = 240 | ar = 240 | er_bits = 9 | ar_bits = 9 +++
+++ Step 4: Pass: |opcode| time = 2 ns | er = 7c2 | ar = 7c2 | er_bits = 11 | ar_bits = 11 +++

Test Case 2
ADD X10, X21, X9
+++ Step 1: Pass: |rm_num| time = 12 ns | er = 9 | ar = 9 | er_bits = 5 | ar_bits = 5 +++
+++ Step 2: Pass: |rn_num| time = 12 ns | er = 21 | ar = 21 | er_bits = 5 | ar_bits = 5 +++
+++ Step 3: Pass: |rd_num| time = 12 ns | er = 10 | ar = 10 | er_bits = 5 | ar_bits = 5 +++
+++ Step 4: Pass: |opcode| time = 12 ns | er = 458 | ar = 458 | er_bits = 11 | ar_bits = 11 +++

Test Case 3
STUR X10, [X23, #64]
+++ Step 1: Pass: |rn_num| time = 22 ns | er = 23 | ar = 23 | er_bits = 5 | ar_bits = 5 +++
+++ Step 2: Pass: |rd_num| time = 22 ns | er = 10 | ar = 10 | er_bits = 5 | ar_bits = 5 +++
+++ Step 3: Pass: |address| time = 22 ns | er = 64 | ar = 64 | er_bits = 9 | ar_bits = 9 +++
+++ Step 4: Pass: |opcode| time = 22 ns | er = 7c0 | ar = 7c0 | er_bits = 11 | ar_bits = 11 +++

Pass Count = 12
Fail Count = 0

***** END TEST RESULTS *****

```

on the end of the name indicates that this is the register number, not the value from the register.

To test this module, you will need to finish `instr_parse_test.sv`. I have provided some starter code for the testbench as well as detailed comments that describe what you need to do. The testbench will feed the module with instructions. The instructions are specified in the testbench and are very similar (yet slightly different) to the instructions that were encoded in the lecture on Machine Code. You will need to update the testbench by:

1. Creating signals to be used in the testbench
2. Setting the instruction to the correct value per the instruction listed in the comments
3. Add code to verify that each output of your `instruction_parse` module is correct

The final output of your testbench should match the output shown in Figure 4.1.

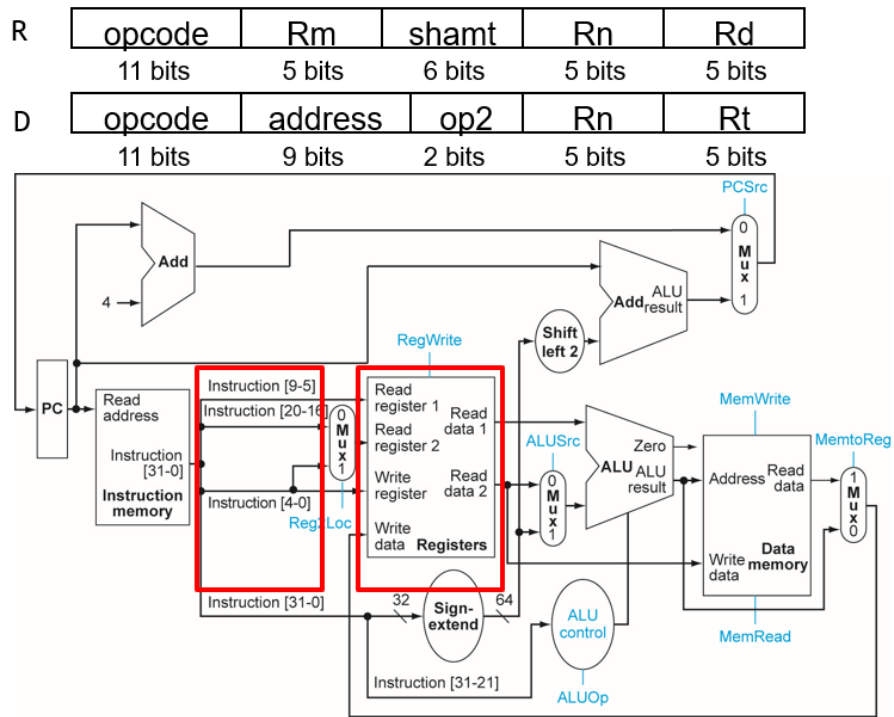
## 4.2 Register File

Next, we will create the register file. The register file is a piece of memory in the processor that holds the 32 register values that are used by most instructions (X0-X31). You will create a new module called regfile (in a new file, regfile.v in 2.decode). The regfile module should retrieve data from the registers on the rising edge of read\_clk as well as write to the registers on the rising edge of write\_clk when the regWrite flag is set. Two different clocks are used here because the regfile will be read at a different time than it is written to. The regfile should use a verilog reg array, similar to the array used in instruction memory. Since we don't currently have the ability to do loads and stores (since we don't have data memory yet), the values for the registers should be stored in a datafile, regData.data and copied into the array during the initial block, just like we did with the instr\_mem.v file. regData.data is provided for you. The regfile module will have a lot of similarities to the instr\_mem module, so I recommend reusing concepts and code from the instr\_mem module.

Inputs to the module should include a signal called read\_clk and a signal called write\_clk as well as all inputs shown on the Register file in Figure 4.2. Don't forget reg\_write. This is a control signal that determines whether data should be written to the register. Some instruction write to registers, others do not. The outputs should be the outputs of the Register file in Figure 4.2. Use names such as read\_register1, read\_data2, etc.

I have provided the majority of the testbench for this module, regfile\_test.sv. It provides input values and verifies that the outputs match expected behavior. Notice how the testbench utilizes the delay module to create different clocks for read\_clk and write\_clk. The testbench is designed to test a variety of scenarios and to verify the timing aspects of this module. Your only job on the testbench is to fill in the expected result (er) values in the testbench. They are currently populated with X.

Figure 4.2: Instruction Parse and Regfile Diagrams



### 4.3 Your Assignment

You are to:

1. Create an `instruction_parse` module as described above.
2. Update `instr_parse_test` and verify the functionality of the `instruction_parse` module. For this testbench, please use hex for instructions and opcodes and unsigned decimal for all other signals.
3. Create a `regfile` module.
4. Update the `regfile_test` module as described above and verify the functionality of the `regfile` module. For this testbench, please use signed decimal for all signals.
5. Produce a landscape mode PDF called `Lab4_lastname.pdf` that includes (in this order):
  - (a) Your name and the lab number.
  - (b) A snip of the Simulation Results for the `instr_parse_test`.
  - (c) The `instr_parse_test` results copied and pasted from the Tcl Console. The results should show the entire log from BEGIN TEST RESULTS to END TEST RESULTS.
  - (d) A snip of the Simulation Results for the `regfile_test`.
  - (e) The `regfile_test` results copied and pasted from the Tcl Console. The results should show the entire log from BEGIN TEST RESULTS to END TEST RESULTS.
6. Upload `Lab4_lastname.pdf` file to Canvas.
7. Zip up your ARM-Lab directory and submit it on Canvas as well. Please make sure that you give me the ARM-Lab directory rather than the ARM-Project directory. I do not want the project files in the ARM-Project directory. Before you submit your zip file, make sure that it contains directories like `code`, `manual`, `tables`, etc. I will extract your zip file and run your code against my correct testbench to verify that your code and testbench work correctly.