

ECE 4750 Computer Architecture, Fall 2016

Tutorial 4: Verilog Hardware Description Language

School of Electrical and Computer Engineering
Cornell University

revision: 2016-11-05-14-20

| | | |
|----------|--|-----------|
| 1 | Introduction | 3 |
| 2 | Verilog Modeling: Synthesizable vs. Non-Synthesizable RTL | 4 |
| 3 | Verilog Basics: Data Types, Operators, and Conditionals | 4 |
| 3.1 | Hello World | 4 |
| 3.2 | Logic Data Types | 5 |
| 3.3 | Shift Operators | 10 |
| 3.4 | Arithmetic Operators | 11 |
| 3.5 | Relational Operators | 12 |
| 3.6 | Concatenation Operators | 14 |
| 3.7 | Enum Data Types | 15 |
| 3.8 | Struct Data Types | 17 |
| 3.9 | Ternary Operator | 19 |
| 3.10 | If Statements | 20 |
| 3.11 | Case Statements | 21 |
| 3.12 | Casez Statements | 22 |
| 4 | Registered Incrementer | 22 |
| 4.1 | RTL Model of Registered Incrementer | 23 |
| 4.2 | Simulating a Model using iverilog | 25 |
| 4.3 | Verifying a Model with Unit Testing | 27 |
| 4.4 | Reusing a Model with Structural Composition | 29 |
| 4.5 | Parameterizing a Model with Static Elaboration | 31 |
| 5 | Sort Unit | 33 |
| 5.1 | Flat Sorter Implementation | 33 |
| 5.2 | Using Verilog Line Traces | 37 |
| 5.3 | Structural Sorter Implementation | 38 |
| 5.4 | Evaluating the Sorter Using a Simulator | 40 |

| | | |
|----------|--|-----------|
| 6 | Greatest Common Divisor: Verilog Design Example | 40 |
| 6.1 | Control/Datapath Split Implementation | 41 |
| 6.2 | Evaluating GCD Unit using a Simulator | 46 |
| 7 | TravisCI for Continuous Integration | 47 |

1. Introduction

In the lab assignments for this course, we will be using the PyMTL hardware modeling framework for functional-level modeling, verification, and simulator harnesses. Students can choose to use either PyMTL or Verilog to do their register-transfer-level (RTL) modeling. If you are planning to use PyMTL, then you do not need to complete this tutorial. If you are planning to use Verilog, you should still complete the PyMTL tutorial since we will always be using PyMTL for some aspects of the lab assignment.

This tutorial briefly reviews the basics of the Verilog hardware description language, but primarily focuses on how we can integrate Verilog RTL modeling into our PyMTL framework. Although we will be using the open-source tool Icarus Verilog (`iverilog`) for compiling some simple Verilog models into simulators, we will primarily be using the open-source tool Verilator (`verilator`) as our Verilog simulator. PyMTL has built-in support for testing Verilog simulators created using Verilator. As in the PyMTL tutorial, we will also be using GTKWave (`gtkwave`) for viewing waveforms. All tools are installed and available on `ecelinux`. This tutorial assumes that students have completed the Linux and Git tutorials.

To follow along with the tutorial, access the ECE computing resources, and type the commands without the `%` character. In addition to working through the commands in the tutorial, you should also try the more open-ended tasks marked with the ★ symbol.

Before you begin, make sure that you have **sourced the `ece4750-setup` script** or that you have added it to your `.bashrc` script, which will then source the script every time you login. Sourcing the setup script sets up the environment required for this class.

You should start by forking the tutorial repository on GitHub. Start by going to the GitHub page for the tutorial repository located here:

- <https://github.com/cornell-ece4750/ece4750-tut4-verilog>

Click on *Fork* in the upper right-hand corner. If asked where to fork this repository, choose your personal GitHub account. After a few seconds, you should have a new repository in your account:

- <https://github.com/<githubid>/ece4750-tut4-verilog>

Where `<githubid>` is your GitHub ID, not your NetID. Now access `ecelinux` and clone your copy of the tutorial repository as follows:

```
% source setup-ece4750.sh
% mkdir -p ${HOME}/ece4750
% cd ${HOME}/ece4750
% git clone https://github.com/<githubid>/ece4750-tut4-verilog.git tut4
% cd tut4/sim
% TUTROOT=${PWD}
```

NOTE: It should be possible to experiment with this tutorial even if you are not enrolled in the course and/or do not have access to the course computing resources. All of the code for the tutorial is located on GitHub. You will not use the `setup-ece4750.sh` script, and your specific environment may be different from what is assumed in this tutorial.

2. Verilog Modeling: Synthesizable vs. Non-Synthesizable RTL

Verilog is a powerful language that was originally intended for building simulators of hardware as opposed to models that could automatically be transformed into hardware (e.g., synthesized to an FPGA or ASIC). Given this, it is very easy to write Verilog code that does not actually model any kind of realistic hardware. Indeed, we actually need this feature to be able to write clean and productive assertions and line tracing. Non-synthesizable Verilog modeling is also critical when implementing test harnesses. **So students must be very diligent in actively deciding whether or not they are writing synthesizable register-transfer-level models or non-synthesizable code. Students must always keep in mind what hardware they are modeling and how they are modeling it!**

Students' design work will almost exclusively use synthesizable register-transfer-level (RTL) models. It is acceptable to include a limited amount of non-synthesizable code in students' designs for the sole purpose of debugging, assertions, or line tracing. If the student includes non-synthesizable code in their actual design (i.e., not the test harness), they must explicitly demarcate this code by wrapping it in `'ifdef SYNTHESIS` and `'endif`. This explicitly documents the code as non-synthesizable and aids automated tools in removing this code before synthesizing the design. **If at any time students are unclear about whether a specific construct is allowed in a synthesizable concurrent block, they should ask the instructors.**

Appendix A includes a table that outlines which Verilog constructs are allowed in synthesizable RTL, which constructs are allowed in synthesizable RTL with limitations, and which constructs are explicitly not allowed in synthesizable RTL. There are no limits on using the Verilog preprocessor, since the preprocessing step happens at compile time.

3. Verilog Basics: Data Types, Operators, and Conditionals

We will begin by writing some very basic code to explore Verilog data types, operators, and conditionals. We will not be modeling actual hardware yet; we are just experimenting with the language. Start by creating a build directory to work in.

```
% mkdir ${TUTROOT}/build
% cd ${TUTROOT}/build
```

3.1. Hello World

Create a new Verilog source file named `hello-world.v` with the contents shown in Figure 1 using your favorite text editor. A `module` is the fundamental hardware building block in Verilog, but for now we are simply using it to encapsulate an `initial` block. The `initial` block specifies code which should be executed "at the beginning of time" when the simulator starts. Since real hardware cannot do anything "at the beginning of time" `initial` blocks are not synthesizable and you should not

```
1 module top;
2   initial begin
3     $display( "Hello World!" );
4   end
5 endmodule
```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-hello-world.v>

Figure 1: Verilog Basics: Display Statement – The obligatory "Hello, World!" program to compiling a basic Verilog program.

use them in the synthesizable portion of your designs. However, `initial` blocks can be useful for test harnesses and when experimenting with the Verilog language. The `initial` block in Figure 1 contains a single call to the `display` system task which will output the given string to the console.

We will be using `iverilog` to compile Verilog models into simulators in the beginning of this tutorial before we turn our attention to using Verilator. You can run `iverilog` as follows to compile `hello-world.v`.

```
% cd ${TUTROOT}/build
% iverilog -g2012 -o hello-world hello-world.v
```

The `-g2012` option tells `iverilog` to accept newer SystemVerilog syntax, and the `-o` specifies the name of the simulator that `iverilog` will create. You can run this simulator as follows.

```
% cd ${TUTROOT}/build
% ./hello-world
```

As discussed in the Linux tutorial you can compile the Verilog and run the simulator in a single step.

```
% cd ${TUTROOT}/build
% iverilog -g2012 -o hello-world hello-world.v && ./hello-world
```

This makes it easy to edit the Verilog source file, quickly recompile, and test your changes by switching to your terminal, pressing the up-arrow key, and then pressing enter.

- ★ *To-Do On Your Own:* Edit the string that is displayed in this simple program, recompile, and rerun the simulator.

3.2. Logic Data Types

To understand any new modeling language we usually start by exploring the primitive data types for representing values in a model. Verilog uses a combination of the `wire` and `reg` keywords which interact in subtle and confusing ways. SystemVerilog has simplified modeling by introducing the `logic` data type. We will be exclusively using `logic` as the general-purpose, hardware-centric data type for modeling a single bit or multiple bits. Each bit can take on one of four values: 0, 1, X, Z. X is used to represent unknown values (e.g., the state of a register on reset). Z is used to represent high-impedance values. Although we will use variables with X values in this course, we will not use variables with Z values (although you may see Z values if you forget to connect an input port of a module).

Table 1 shows the operators that we will be primarily using in this course. Note that Verilog and SystemVerilog support additional operators including `*` for multiplication, `/` for division, `%` for modulus, `**` for exponent, and `==`/`!=="` for special equality checks. There may occasionally be reasons to use one of these operators in your assertion or line tracing logic. However, these operators are either not synthesizable or synthesize poorly, so students are not allowed to use these operators in the synthesizable portion of their designs.

Figure 2 shows an example program that illustrates single-bit logic types. Create a new Verilog source file named `logic-sbit.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

| Logical Operators | Reduction Operators | Relational Operators |
|----------------------|----------------------------|---------------------------|
| & bitwise AND | & reduce via AND | == equal |
| bitwise OR | ~& reduce via NAND | != not equal |
| ^ bitwise XOR | reduce via OR | > greater than |
| ^^ bitwise XNOR | ~ reduce via NOR | >= greater than or equals |
| ~ bitwise NOT | ~ reduce via XOR | < less than |
| && boolean AND | ^^ reduce via XNOR | <= less than or equals |
| boolean OR | Shift Operators | Other Operators |
| ! boolean NOT | >> shift right | { } concatenate |
| Arithmetic Operators | << shift left | {N{ } replicate N times |
| + addition | >>> arithmetic shift right | |
| - subtraction | | |

Table 1: Table of Verilog Operators – Not all Verilog operators are shown, just those operators that are acceptable for use in the synthesizable RTL portion of students’ designs.

Lines 13–16 illustrate how to write single-bit literals to express constant values. Lines 23–26 illustrate basic bitwise logical operators (&, |, ^, ~). Whenever we consider an expression in Verilog, we should always ask ourselves, “What will happen if one of the inputs is an X?” Lines 33–36 illustrate what happens if the second operand is an X for bitwise logical operators. Recall that X means “unknown”. If we OR the value 0 with an unknown value we cannot know the result. If the unknown value is 0, then the result should be 0, but if the unknown value is 1, then the result should be 1. So Verilog specifies that in this case the value of the expression is X. Notice what happens if we AND the value 0 with an unknown value. In this case, we can guarantee that for any value for the second operand the result will always be 0, so Verilog specifies the value of the expression is 0.

In addition to the basic bitwise logical operators, Verilog also defines three boolean logical operators (&&, ||, !). These operators are effectively the same as the basic logical operators (&, |, ~) when operating on single-bit logic values. The difference is really in the designer’s intent. Using &&, ||, ! suggests that the designer is implementing a boolean logic expression as opposed to doing low-level bit manipulation.

- ★ *To-Do On Your Own:* Experiment with more complicated multi-stage logic expressions by writing the boolean logic equations for a one-bit full-adder. Use the `display` system task to output the result to the console. Experiment with using X input values as inputs to these logic expressions.

```

1  module top;
2
3  // Declare single-bit logic variables.
4
5  logic a;
6  logic b;
7  logic c;
8
9  initial begin
10
11     // Single-bit literals
12
13     a = 1'b0;    $display( "1'b0  = %x ", a );
14     a = 1'b1;    $display( "1'b1  = %x ", a );
15     a = 1'bx;    $display( "1'bx  = %x ", a );
16     a = 1'bz;    $display( "1'bz  = %x ", a );
17
18     // Bitwise logical operators for doing AND, OR, XOR, and NOT
19
20     a = 1'b0;
21     b = 1'b1;
22
23     c = a & b;    $display( "0 & 1  = %x ", c );
24     c = a | b;    $display( "0 | 1  = %x ", c );
25     c = a ^ b;    $display( "0 ^ 1  = %x ", c );
26     c = ~b;       $display( "~1     = %x ", c );
27
28     // Bitwise logical operators for doing AND, OR, XOR, and NOT with X
29
30     a = 1'b0;
31     b = 1'bx;
32
33     c = a & b;    $display( "0 & x  = %x ", c );
34     c = a | b;    $display( "0 | x  = %x ", c );
35     c = a ^ b;    $display( "0 ^ x  = %x ", c );
36     c = ~b;       $display( "~x     = %x ", c );
37
38     // Boolean logical operators
39
40     a = 1'b0;
41     b = 1'b1;
42
43     c = a && b;    $display( "0 && 1 = %x ", c );
44     c = a || b;    $display( "0 || 1 = %x ", c );
45     c = !b;        $display( "!1    = %x ", c );
46
47 end
48
49 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-logic-sbit.v>

Figure 2: Verilog Basics: Single-Bit Logic and Logical Operators – Experimenting with single-bit logic variables and literals, logical bitwise operators, and logical boolean operators.

Multi-bit logic types are used for modeling bit vectors. Figure 3 shows an example program that illustrates multi-bit logic types. Create a new Verilog source file named `logic-mbit.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

Lines 5–8 declares multi-bit logic variables. The square brackets contain the index of the most-significant and the least-significant bit. In this course, you should always use zero as the index of the least significant bit. Note that to declare a four-bit logic value, we use `[3:0]` not `[4:0]`.

Lines 14–17 illustrate multi-bit literals that can be used to declare constant values. These literals have the following general syntax: `<bitwidth>'<base><number>` where `<base>` can be `b` for binary, `h` for hexadecimal, or `d` for decimal. It is legal to include underscores in the literal, which can be helpful for improving the readability of long literals.

Lines 24–28 illustrate multi-bit versions of the basic bitwise logic operators. As before, we should always ask ourselves, “What will happen if one of the inputs is an X?” Lines 35–39 illustrate what happens if two bits in the second value are Xs. Note that some bits in the result are X and some can be guaranteed to be either a 0 or 1.

Lines 45–50 illustrate the reduction operators. These operators take a multi-bit logic value and combine all of the bits into a single-bit value. For example, the OR reduction operator (`|`) will OR all of the bits together.

- ★ *To-Do On Your Own:* We will use relational operators (e.g., `==`) to compare two multi-bit logic values, but see if you can achieve the same effect with the bitwise XOR/XNOR operators and OR/NOR reduction operators.

```

1  module top;
2
3  // Declare multi-bit logic variables
4
5  logic [ 3:0] A; // 4-bit logic variable
6  logic [ 3:0] B; // 4-bit logic variable
7  logic [ 3:0] C; // 4-bit logic variable
8  logic [11:0] D; // 12-bit logic variable
9
10 initial begin
11
12 // Multi-bit literals
13
14 A = 4'b0101;          $display( "4'b0101          = %x", A );
15 D = 12'b1100_1010_0101; $display( "12'b1100_1010_0101 = %x", D );
16 D = 12'hca5;          $display( "12'hca5          = %x", D );
17 D = 12'd1058;          $display( "12'd1058          = %x", D );
18
19 // Bitwise logical operators for doing AND, OR, XOR, and NOT
20
21 A = 4'b0101;
22 B = 4'b0011;
23
24 C = A & B;    $display( "4'b0101 & 4'b0011 = %b", C );
25 C = A | B;    $display( "4'b0101 | 4'b0011 = %b", C );
26 C = A ^ B;    $display( "4'b0101 ^ 4'b0011 = %b", C );
27 C = A ~ B;    $display( "4'b0101 ~ 4'b0011 = %b", C );
28 C = ~B;       $display( "~4'b0011          = %b", C );
29
30 // Bitwise logical operators when some bits are X
31
32 A = 4'b0101;
33 B = 4'b00xx;
34
35 C = A & B;    $display( "4'b0101 & 4'b00xx = %b", C );
36 C = A | B;    $display( "4'b0101 | 4'b00xx = %b", C );
37 C = A ^ B;    $display( "4'b0101 ^ 4'b00xx = %b", C );
38 C = A ~ B;    $display( "4'b0101 ~ 4'b00xx = %b", C );
39 C = ~B;       $display( "~4'b00xx          = %b", C );
40
41 // Reduction operators
42
43 A = 4'b0101;
44
45 C = &A;        $display( "& 4'b0101 = %b", C );
46 C = ~&A;       $display( "~& 4'b0101 = %b", C );
47 C = |A;        $display( "| 4'b0101 = %b", C );
48 C = ~|A;       $display( "~| 4'b0101 = %b", C );
49 C = ^A;        $display( "^ 4'b0101 = %b", C );
50 C = ~^A;       $display( "~^ 4'b0101 = %b", C );
51
52 end
53
54 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-logic-mbit.v>

Figure 3: Verilog Basics: Multi-Bit Logic and Logical Operators – Experimenting with multi-bit logic variables and literals, bitwise logical operators, and reduction operators.

3.3. Shift Operators

Figure 4 illustrates three shift operators on multi-bit logic values. Create a new Verilog source file named `logic-shift.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

Notice how the logical shift operators (`<<`, `>>`) always shift in zeros, but the arithmetic right shift operator (`>>>`) replicates the most-significant bit. Verilog requires that the left-hand operand to the arithmetic shift operator be explicitly denoted as signed, which we have done using the `signed` system task. We recommend this approach and avoiding the use of signed data types.

★ *To-Do On Your Own:* Experiment different multi-bit logic values and shift amounts.

```

1  module top;
2
3  logic [7:0] A;
4  logic [7:0] B;
5  logic [7:0] C;
6
7  initial begin
8
9      // Fixed shift amount for logical shifts
10
11     A = 8'b1110_0101;
12
13     C = A << 1;           $display( "8'b1110_0101 << 1 = %b", C );
14     C = A << 2;           $display( "8'b1110_0101 << 2 = %b", C );
15     C = A << 3;           $display( "8'b1110_0101 << 3 = %b", C );
16
17     C = A >> 1;           $display( "8'b1110_0101 >> 1 = %b", C );
18     C = A >> 2;           $display( "8'b1110_0101 >> 2 = %b", C );
19     C = A >> 3;           $display( "8'b1110_0101 >> 3 = %b", C );
20
21     // Fixed shift amount for arithmetic shifts
22
23     A = 8'b0110_0100;
24
25     C = $signed(A) >>> 1; $display( "8'b0110_0100 >>> 1 = %b", C );
26     C = $signed(A) >>> 2; $display( "8'b0110_0100 >>> 2 = %b", C );
27     C = $signed(A) >>> 3; $display( "8'b0110_0100 >>> 3 = %b", C );
28
29     A = 8'b1110_0101;
30
31     C = $signed(A) >>> 1; $display( "8'b1110_0101 >>> 1 = %b", C );
32     C = $signed(A) >>> 2; $display( "8'b1110_0101 >>> 2 = %b", C );
33     C = $signed(A) >>> 3; $display( "8'b1110_0101 >>> 3 = %b", C );
34
35     // Variable shift amount for logical shifts
36
37     A = 8'b1110_0101;
38     B = 8'd2;
39
40     C = A << B;           $display( "8'b1110_0101 << 2 = %b", C );
41     C = A >> B;           $display( "8'b1110_0101 >> 2 = %b", C );
42
43     end
44
45 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-logic-shift.v>

Figure 4: Verilog Basics: Shift Operators – Experimenting with logical and arithmetic shift operators and fixed/variable shift amounts.

3.4. Arithmetic Operators

Figure 5 illustrates the addition and subtraction operators for multi-bit logic values. Create a new Verilog source file named `logic-arith.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

These operators treat the multi-bit logic values as unsigned integers. Although Verilog does include support for signed arithmetic, these constructs may not be synthesizable so you are required to use only unsigned arithmetic. Also recall that `*`, `/`, `%`, `**` are not allowed in the synthesizable portion of your design.

Note that carefully considering the bitwidths of the input and output variables is important. Lines 22–23 illustrate overflow and underflow. You can see that if you overflow the bitwidth of the output variable then the result will simply wrap around. Similarly, since we are using unsigned arithmetic negative numbers wrap around. This is also called modular arithmetic.

- ★ *To-Do On Your Own:* Try writing some code which does a sequence of additions resulting in overflow and then a sequence of subtractions that essentially undo the overflow. For example, try $200 + 400 + 400 - 400 - 400$. Does this expression produce the expected answer even though the intermediate values overflowed?

```

1  module top;
2
3  logic [7:0] A;
4  logic [7:0] B;
5  logic [7:0] C;
6
7  initial begin
8
9      // Basic arithmetic with no overflow or underflow
10
11     A = 8'd28;
12     B = 8'd15;
13
14     C = A + B; $display( "8'd28 + 8'd15 = %d", C );
15     C = A - B; $display( "8'd28 - 8'd15 = %d", C );
16
17     // Basic arithmetic with overflow and underflow
18
19     A = 8'd250;
20     B = 8'd15;
21
22     C = A + B; $display( "8'd250 + 8'd15 = %d", C );
23     C = B - A; $display( "8'd15 - 8'd250 = %d", C );
24
25     end
26
27 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-logic-arith.v>

Figure 5: Verilog Basics: Arithmetic Operators – Experimenting with arithmetic operators for addition and subtraction.

3.5. Relational Operators

Figure 6 illustrates the relational operators used for comparing two multi-bit logic values. Create a new Verilog source file named `logic-relop.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

Lines 28–33 illustrate what happens if some of the bits are Xs for these relational operators. Notice that we can still determine two values are not equal even if some bits are unknown. If the bits we do know are different then the unknown bits do not matter; we can guarantee that the full bit vectors are not equal. So in this example, since we know that the top-two bits in `4'b1100` and `4'b10xx` then we can guarantee that the two values are not equal even though the bottom two bits of one operand are unknown.

The `<`, `>`, `<=`, `>=` operators behave slightly differently than the `==` and `!=` operators when considering values with Xs. In this example, we should be able to guarantee that `4'b1100` is always greater than `4'b10xx` (assuming these are unsigned values), since no matter what the bottom two bits are in the second operand it cannot be greater than the first operand. However, if you run this simulation, then you will see that the result is still X. This is not a bug and is correct according to the Verilog language specification. This is a great example of how Verilog has relatively complicated and sometimes inconsistent language semantics. Originally, there was no Verilog standard. The most common Verilog simulator was the de-factor language standard. I imagine the reason there is this difference between how `==` and `<` handle X values is simply because in the very first Verilog simulators it was the most efficient solution. These kind of “simulator implementation issues” can be found throughout the Verilog standard.

Lines 40–43 illustrates signed comparisons using the `signed` system task to interpret the unsigned input operands as signed values. To simplify our designs, we do not allow students to use signed types. We should explicitly use the `signed` system task whenever we need to ensure signed comparisons.

- ★ *To-Do On Your Own:* Try composing relational operators with the boolean logic operators we learned about earlier in this section to create more complicated expressions.

```

1  module top;
2
3      // Declare multi-bit logic variables
4
5      logic      a; // 1-bit logic variable
6      logic [ 3:0] A; // 4-bit logic variable
7      logic [ 3:0] B; // 4-bit logic variable
8
9      initial begin
10
11         // Relational operators
12
13         A = 4'd15; B = 4'd09;
14
15         a = ( A == B );   $display( "(15 == 9) = %x", a );
16         a = ( A != B );   $display( "(15 != 9) = %x", a );
17         a = ( A > B );     $display( "(15 > 9) = %x", a );
18         a = ( A >= B );    $display( "(15 >= 9) = %x", a );
19         a = ( A < B );     $display( "(15 < 9) = %x", a );
20         a = ( A <= B );    $display( "(15 <= 9) = %x", a );
21
22         // Relational operators when some bits are X
23
24         A = 4'b1100; B = 4'b10xx;
25
26         a = ( A == B );   $display( "(4'b1100 == 4'b10xx) = %x", a );
27         a = ( A != B );   $display( "(4'b1100 != 4'b10xx) = %x", a );
28         a = ( A > B );     $display( "(4'b1100 > 4'b10xx) = %x", a );
29         a = ( A < B );     $display( "(4'b1100 < 4'b10xx) = %x", a );
30
31         // Signed relational operators
32
33         A = 4'b1111; // -1 in twos complement
34         B = 4'd0001; // 1 in twos complement
35
36         a = (      A >      B ); $display( "(-1 > 1) = %x", a );
37         a = (      A <      B ); $display( "(-1 < 1) = %x", a );
38         a = ( $signed(A) > $signed(B) ); $display( "(-1 > 1) = %x", a );
39         a = ( $signed(A) < $signed(B) ); $display( "(-1 < 1) = %x", a );
40
41     end
42
43 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-logic-relop.v>

Figure 6: Verilog Basics: Relational Operators – Experimenting with relational operators.

3.6. Concatenation Operators

Figure 7 illustrates the concatenation operators used for creating larger bit vectors from multiple smaller bit vectors. Create a new Verilog source file named `logic-concat.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

Lines 18–20 illustrate concatenating three four-bit logic variables and then assigning the result to a 12-bit logic variable. Lines 25–26 illustrate concatenating a four-bit logic variable with an eight-bit logic variable. The repeat operator can be used to duplicate a given logic variable multiple times when creating larger bit vectors. On Line 33, we replicate a four-bit logic value three times to create a 12-bit logic value.

- ★ *To-Do On Your Own:* Experiment with different variations of concatenation and the repeat operator to create interesting bit patterns.

```

1  module top;
2
3  logic [ 3:0] A; // 4-bit logic variable
4  logic [ 3:0] B; // 4-bit logic variable
5  logic [ 3:0] C; // 4-bit logic variable
6  logic [ 7:0] D; // 18-bit logic variable
7  logic [11:0] E; // 12-bit logic variable
8
9  initial begin
10
11     // Basic concatenation
12
13     A = 4'ha;
14     B = 4'hb;
15     C = 4'hc;
16     D = 8'hde;
17
18     E = { A, B, C };    $display( "{ 4'ha, 4'hb, 4'hc } = %x", E );
19     E = { C, A, B };    $display( "{ 4'hc, 4'ha, 4'hb } = %x", E );
20     E = { B, C, A };    $display( "{ 4'hb, 4'hc, 4'ha } = %x", E );
21
22     E = { A, D };      $display( "{ 4'ha, 8'hde } = %x", E );
23     E = { D, A };      $display( "{ 8'hde, 4'ha } = %x", E );
24
25     E = { A, 8'hf0 };  $display( "{ 4'ha, 8'hf0 } = %x", E );
26     E = { 8'hf0, A };  $display( "{ 8'hf0, 4'ha } = %x", E );
27
28     // Repeat operator
29
30     A = 4'ha;
31     B = 4'hb;
32
33     E = { 3{A} };      $display( "{ 4'ha, 4'ha, 4'ha } = %x", E );
34     E = { A, {2{B}} }; $display( "{ 4'ha, 4'hb, 4'hb } = %x", E );
35
36 end
37
38 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-logic-concat.v>

Figure 7: Verilog Basics: Concatenation Operators – Experimenting with the basic concatenation operator and the repeat operator.

3.7. Enum Data Types

The `logic` data type will be the most common data type we use in our synthesizable RTL since a `logic` variable has a direct one-to-one correspondence with a bit vector in hardware. However, there are certain cases where using a `logic` variable can be quite tedious and error prone. SystemVerilog has introduced two new kinds of user-defined types that can greatly simplify some portions of our designs. In this subsection, we introduce the `enum` type which enables declaring variables that can only take on a predefined list of labels.

Figure 8 illustrates creating and using an `enum` type for holding a state variable which can take on one of four labels. Create a new Verilog source file named `enum.v` and copy all of this code. Compile this source file and run the resulting simulator.

Lines 3–8 declare a new `enum` type named `state_t`. Note that `state_t` is not a new *variable* but is instead a new *type*. We will use the `_t` suffix to distinguish type names from variable names. Note that after the `enum` keyword we have included a *base type* of `logic [clog2(4)-1:0]`. This base type specifies how we wish variables of this new type to be stored. In this case, we are specifying that `state_t` variables should be encoded as a two-bit `logic` value. The `clog2` system task calculates the number of bits in the given argument; it is very useful when writing more parameterized code. So in this situation we just need to pass in the number of labels in the `enum` to `clog2`. SystemVerilog actually provides many different ways to create `enum` types including anonymous types, types where we do not specify the base type, or types where we explicitly define the value for each label. In this course, you should limit yourself to the exact syntax shown in this example.

Line 14 declares a new variable of type `state_t`. This is the first time we have seen a variable which has a type other than `logic`. The ability to introduce new user-defined types is one of the more powerful features of SystemVerilog. Lines 21–24 sets the state variable using the labels declared as part of the new `state_t` type. Lines 28–40 compare the value of the state variable with these same labels, and these comparisons can be used to take different action based on the current value.

There are several advantages to using an `enum` type compared to the basic `logic` type to represent a variable that can hold one of several labels including: (1) more directly capturing the designer's intent to improve code quality; (2) preventing mistakes by eliminating the possibility of defining labels with the same value or defining label values that are too large to fit in the underlying storage; and (3) preventing mistakes when assigning variables of a different type to an `enum` variable.

- ★ *To-Do On Your Own:* Create your own new `enum` type for the state variable we will use in the GCD example later in this tutorial. The new `enum` type should be called `state_t` and it should support three different labels: `STATE_IDLE`, `STATE_CALC`, `STATE_DONE`. Write some code to set and compare the value of a corresponding state variable.

```

1 // Declare enum type
2
3 typedef enum logic [ $clog2(4)-1:0 ] {
4     STATE_A,
5     STATE_B,
6     STATE_C,
7     STATE_D
8 } state_t;
9
10 module top;
11
12 // Declare variables
13
14 state_t state;
15 logic result;
16
17 initial begin
18
19 // Enum lable literals
20
21 state = STATE_A; $display( "STATE_A = %d", state );
22 state = STATE_B; $display( "STATE_B = %d", state );
23 state = STATE_C; $display( "STATE_C = %d", state );
24 state = STATE_D; $display( "STATE_D = %d", state );
25
26 // Comparisons
27
28 state = STATE_A;
29
30 result = ( state == STATE_A );
31 $display( "( STATE_A == STATE_A ) = %x", result );
32
33 result = ( state == STATE_B );
34 $display( "( STATE_A == STATE_B ) = %x", result );
35
36 result = ( state != STATE_A );
37 $display( "( STATE_A != STATE_A ) = %x", result );
38
39 result = ( state != STATE_B );
40 $display( "( STATE_A != STATE_B ) = %x", result );
41
42 end
43
44 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-enum.v>

Figure 8: Verilog Basics: Enum Data Types – Experimenting with enum data types including setting the value of an enum using a label and using the equality operator.

3.8. Struct Data Types

User-defined structures are now supported in SystemVerilog. Figure 9 illustrates creating and using a struct type for holding a variable with predefined named bit fields. Create a new Verilog source file named `struct.v` and copy all of this code. Compile this source file and run the resulting simulator.

Lines 3–7 declare a new struct type named `point_t`. Again note that `point_t` is not a new *variable* but is instead a new type. As before we use the `_t` suffix to distinguish type names from variable names. Note that after the `struct` keyword we have included the `packed` keyword which specifies that variables of this type should have an equivalent underlying logic storage. SystemVerilog also includes support for unpacked structs, but in this course, you should limit yourself to the exact syntax shown in this example. In addition to declaring the name of the new struct type, we also declare the named bit fields within the new struct type. The order of these bit fields is important; the first field will go in the most significant position of the underlying logic storage, the second field will go in the next position, and so on.

Lines 13–14 declare two new variables of type `point_t`. Line 18 declares a new logic variable with a bitwidth large enough to hold a variable of type `point_t`. We can use the `bits` system task to easily determine the total number of bits required to store a struct type. Lines 24–26 set the three fields of the `point` variable and Lines 28–30 read these three fields in order to display them. Line 34 copies one `point` variable into another `point` variable. Line 42 and 49 illustrate how to convert a `point` variable to/from a basic logic variable.

There are several advantages to using a struct type compared to the basic logic type to represent a variable with a predefined set of named bit fields including: (1) more directly capturing the designer's intent to improve code quality; (2) reducing the syntactic overhead of managing bit fields; and (3) preventing mistakes in modifying bit fields and in accessing bit fields.

- ★ *To-Do On Your Own:* Create a new struct type for holding the an RGB color pixel. The struct should include three fields named `red`, `green`, and `blue`. Each field should be eight bits. Experiment with reading and writing these named fields.

```

1 // Declare struct type
2
3 typedef struct packed { // Packed format:
4     logic [3:0] x;      // 11 8 7 4 3 0
5     logic [3:0] y;      // +----+----+----+
6     logic [3:0] z;      // | x | y | z |
7 } point_t;             // +----+----+----+
8
9 module top;
10
11 // Declare variables
12
13 point_t point_a;
14 point_t point_b;
15
16 // Declare other variables using $bits()
17
18 logic [$bits(point_t)-1:0] point_bits;
19
20 initial begin
21
22 // Reading and writing fields
23
24 point_a.x = 4'h3;
25 point_a.y = 4'h4;
26 point_a.z = 4'h5;
27
28 $display( "point_a.x = %x", point_a.x );
29 $display( "point_a.y = %x", point_a.y );
30 $display( "point_a.z = %x", point_a.z );
31
32 // Assign structs
33
34 point_b = point_a;
35
36 $display( "point_b.x = %x", point_b.x );
37 $display( "point_b.y = %x", point_b.y );
38 $display( "point_b.z = %x", point_b.z );
39
40 // Assign structs to bit vector
41
42 point_bits = point_a;
43
44 $display( "point_bits = %x", point_bits );
45
46 // Assign bit vector to struct
47
48 point_bits = { 4'd13, 4'd9, 4'd3 };
49 point_a = point_bits;
50
51 $display( "point_a.x = %x", point_a.x );
52 $display( "point_a.y = %x", point_a.y );
53 $display( "point_a.z = %x", point_a.z );
54
55 end
56
57 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-struct.v>

Figure 9: Verilog Basics: Struct Data Types – Experimenting with struct data types including read/writing fields and converting to/from logic bit vectors.

3.9. Ternary Operator

Figure 10 illustrates using the ternary operator for conditional execution. Create a new Verilog source file named `ternary.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

Lines 12–19 illustrate using the ternary operator to choose what value to assign to the logic variable `c`. We can nest multiple ternary operators to compactly create expressions with multiple conditions. Lines 23–31 illustrate using four levels of nesting to choose among four different values for assigning `c`.

Lines 35–53 illustrate how the ternary operator functions if the conditional is unknown. In lines 35–43, all bits of the conditional are unknown, while in lines 45–53 only one bit of the conditional is unknown. If you examine the output from this simulator, you will see that Verilog semantics require any bits which can be guaranteed to be either 0 or 1 to be set as such, while the remaining bits are set to X. Regardless of the condition, the upper five bits of `c` are guaranteed to be 00001.

Note that the four ternary operators cover all possible combinations of the two-bit input, so the final value (i.e., 8'h0e) will never be used. In other words, if the conditionals contain unknowns this does *not* mean the condition evaluates to false. This is very different from the `if` statements described in the next subsection.

Aside: For some reason, many students insist on writing code like this:

```
a = ( cond_a ) ? 1'b1 : 1'b0;
b = ( cond_b ) ? 1'b0 : 1'b1;
```

This obfuscates the code and is not necessary. We are using a ternary operator to simply choose between 0 or 1. You should just assign the result of the conditional expression to `a` and `b` like this:

```
a = ( cond_a );
b = !( cond_b );
```

- ★ *To-Do On Your Own:* Experiment with different uses of the ternary operator.

```

1  module top;
2
3  logic [7:0] a;
4  logic [7:0] b;
5  logic [7:0] c;
6  logic [1:0] sel;
7
8  initial begin
9
10     // ternary statement
11
12     a = 8'd30;
13     b = 8'd16;
14
15     c = ( a < b ) ? 15 : 14;
16     $display( "c = %d", c );
17
18     c = ( b < a ) ? 15 : 14;
19     $display( "c = %d", c );
20
21     // nested ternary statement
22
23     sel = 2'b01;
24
25     c = ( sel == 2'b00 ) ? 8'h0a
26         : ( sel == 2'b01 ) ? 8'h0b
27         : ( sel == 2'b10 ) ? 8'h0c
28         : ( sel == 2'b11 ) ? 8'h0d
29         : 8'h0e;
30
31     $display( "sel = 1, c = %b", c );
32
33     // nested ternary statement w/ X
34
35     sel = 2'bx;
36
37     c = ( sel == 2'b00 ) ? 8'h0a
38         : ( sel == 2'b01 ) ? 8'h0b
39         : ( sel == 2'b10 ) ? 8'h0c
40         : ( sel == 2'b11 ) ? 8'h0d
41         : 8'h0e;
42
43     $display( "sel = x, c = %b", c );
44
45     sel = 2'bx0;
46
47     c = ( sel == 2'b00 ) ? 8'h0a
48         : ( sel == 2'b01 ) ? 8'h0b
49         : ( sel == 2'b10 ) ? 8'h0c
50         : ( sel == 2'b11 ) ? 8'h0d
51         : 8'h0e;
52
53     $display( "sel = x, c = %b", c );
54
55     end
56
57 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-ternary.v>

Figure 10: Verilog Basics: Ternary Operator – Experimenting with the ternary operator including nested statements and what happens if the conditional includes an unknown.

3.10. If Statements

Figure 11 illustrates using if statements. Create a new Verilog source file named `if.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

The if statement resembles similar constructs in many other programming languages. Lines 11–20 illustrate basic if statements and lines 24–33 illustrate if/else statements.

There are some subtle issues involved in how an if statement handles X values in the conditional. Lines 37–46 illustrate this issue. The `sel` value in this example is a single-bit X. What would we expect the value of `a` to be after this if statement? Since the conditional is unknown, we might expect any variables that are written from within the if statement to also be unknown. In other words, we might expect the value of `a` to be X after this if statement. If you run this example code, you will see that the value of `a` is actually `8'h0b`. This means that an X value in the conditional for an if statement is not treated as unknown but is instead simply treated as if the conditional evaluated to false! This issue is called *X optimism* since unknowns are essentially optimistically turned into known values. X optimism can cause subtle simulation/synthesis mismatch issues. If you are interested, there are several resources on the public course webpage that go into much more detail. For this course, we don't need to worry too much about X optimism since we are not actually synthesizing our designs.

- ★ *To-Do On Your Own:* Experiment with different if statements including nested if statements. Experiment with what happens when the conditional is unknown.

```

1  module top;
2
3  logic [7:0] a;
4  logic [7:0] b;
5  logic      sel;
6
7  initial begin
8
9      // if statement
10
11     a = 8'd30;
12     b = 8'd16;
13
14     if ( a == b ) begin
15         $display( "30 == 16" );
16     end
17
18     if ( a != b ) begin
19         $display( "30 != 16" );
20     end
21
22     // if else statement
23
24     sel = 1'b1;
25
26     if ( sel == 1'b0 ) begin
27         a = 8'h0a;
28     end
29     else begin
30         a = 8'h0b;
31     end
32
33     $display( "sel = 1, a = %x ", a );
34
35     // if else statement w/ X
36
37     sel = 1'bx;
38
39     if ( sel == 1'b0 ) begin
40         a = 8'h0a;
41     end
42     else begin
43         a = 8'h0b;
44     end
45
46     $display( "sel = x, a = %x ", a );
47
48 end
49
50 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-if.v>

Figure 11: Verilog Basics: If Statements – Experimenting with if statements including what happens if the conditional includes an unknown.

3.11. Case Statements

Figure 12 illustrates using case statements. Create a new Verilog source file named `case.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

The case statement resembles similar constructs in many other programming languages. Lines 12–22 illustrate a basic case statement where a two-bit `sel` variable is used to choose one of four case items.

There are similar issues as with the `if` statement in terms of how case statements handle X values in the conditional. In lines 26–36, the `sel` variable is set to all Xs. We might expect since the input to the case statement is unknown the output should also be unknown. However, if we look at the value of `a` after executing this case statement it will be `8'h0e`. In other words, if there is an X in the input to the case statement, then the case statement will fall through to the default case. In order to avoid *X optimism*, we recommend students always include a `default` case that sets all of the output variables to Xs.

Notice that it is valid syntax to use X values in the case items, as shown on lines 48–49. These will actually match Xs in the input condition, which is almost certainly not what you want. This does not model any kind of real hardware; we cannot check for Xs in hardware since in real hardware an unknown must be known (i.e., all Xs will either be a 0 or a 1 in real hardware). Given this, you should never use Xs in the case items for a case statement.

- ★ *To-Do On Your Own:* Experiment with a larger case statement for a `sel` variable with three instead of two bits.

```

1  module top;
2
3  // Declaring Variables
4
5  logic [1:0] sel;
6  logic [7:0] a;
7
8  initial begin
9
10 // case statement
11
12     sel = 2'b01;
13
14     case ( sel )
15         2'b00 : a = 8'h0a;
16         2'b01 : a = 8'h0b;
17         2'b10 : a = 8'h0c;
18         2'b11 : a = 8'h0d;
19         default : a = 8'h0e;
20     endcase
21
22     $display( "sel = 01, a = %x", a );
23
24 // case statement w/ X
25
26     sel = 2'bxx;
27
28     case ( sel )
29         2'b00 : a = 8'h0a;
30         2'b01 : a = 8'h0b;
31         2'b10 : a = 8'h0c;
32         2'b11 : a = 8'h0d;
33         default : a = 8'h0e;
34     endcase
35
36     $display( "sel = xx, a = %x", a );
37
38 // Do not use x's in the case
39 // selection items
40
41     sel = 2'bx0;
42
43     case ( sel )
44         2'b00 : a = 8'h0a;
45         2'b01 : a = 8'h0b;
46         2'b10 : a = 8'h0c;
47         2'b11 : a = 8'h0d;
48         2'bx0 : a = 8'h0e;
49         2'bx1 : a = 8'h0f;
50         default : a = 8'h00;
51     endcase
52
53     $display( "sel = x0, a = %x", a );
54
55 end
56
57 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-case.v>

Figure 12: Verilog Basics: Case Statements – Experimenting with case statements including what happens if the selection expression and/or the case expressions includes an unknown.

3.12. Casez Statements

Figure 13 illustrates using casez statements. Create a new Verilog source file named `casez.v` and copy some or all of this code. Compile this source file and run the resulting simulator.

The casez statement is very different from what you might find in other programming languages. The casez statement is a powerful construct that can enable very concise hardware models, but must be used carefully. A casez statement enables a designer to do “wildcard” matching on the input variable. Lines 10–23 illustrate using a casez statement to implement a “leading-one detector”. This kind of logic outputs the bit position of the least-significant one in the input variable. We can use ? characters in the case items as wildcards that will match either a 0 or 1 in the input variable. So both `4'b0100` and `4'b1100` will match the fourth case item. Implementing similar functionality using a case statement would require 16 items. Besides being more verbose, using a case statement also opens up additional opportunities for errors.

A casez statement behaves similarly to a case statement when there are Xs in the input. Lines 27–40 illustrate a situation where two of the bits in the input variable are unknown. This will match the default case and the output will be Xs.

Aside: Verilog includes a casex statement which you should never use. The reasoning is rather subtle, but to be safe stick to using casez statement if you need wildcard matching (and *only* if you need wildcard matching).

- ★ *To-Do On Your Own:* Experiment with a larger casez statement to implement a leading-one detector for an input variable with eight instead of four bits. How many case items would we need if we used a case statement to implement the same functionality?

```

1  module top;
2
3  logic [3:0] a;
4  logic [7:0] b;
5
6  initial begin
7
8      // casez statement
9
10     a = 4'b0100;
11
12     casez ( a )
13
14         4'b0000 : b = 8'd0;
15         4'b???1  : b = 8'd1;
16         4'b???10 : b = 8'd2;
17         4'b?100  : b = 8'd3;
18         4'b1000  : b = 8'd4;
19
20         default : b = 8'hxx;
21     endcase
22
23     $display( "a = 4'b0100, b = %x", b );
24
25     // casez statement w/ Xs
26
27     a = 4'b01xx;
28
29     casez ( a )
30
31         4'b0000 : b = 8'd0;
32         4'b???1  : b = 8'd1;
33         4'b???10 : b = 8'd2;
34         4'b?100  : b = 8'd3;
35         4'b1000  : b = 8'd4;
36
37         default : b = 8'hxx;
38     endcase
39
40     $display( "a = 4'b01xx, b = %x", b );
41
42 end
43
44 endmodule

```

Code at <https://github.com/cbatten/x/blob/master/ex-basics-casez.v>

Figure 13: Verilog Basics: Casez Statements – Experimenting with casez statements to illustrate their use as priority selectors with wildcards.

4. Registered Incrementer

In this section, we will create our very first Verilog hardware model and learn how to test this module using waveforms, ad-hoc testing, and a simple unit testing framework. As with PyMTL RTL design, it is good design practice to usually draw some kind of diagram of the hardware we wish to model before starting to develop the corresponding Verilog model. This diagram might be a block-level

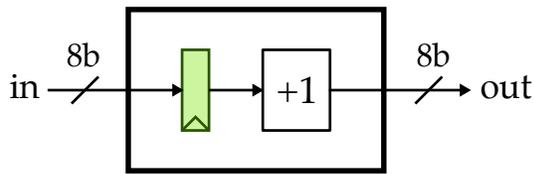


Figure 14: Block Diagram for Registered Incrementer – An eight-bit registered incrementer with an eight-bit input port, an eight-bit output port, and an (implicit) clock input.

diagram, a datapath diagram, a finite-state-machine diagram, or even a control signal table; the more we can structure our Verilog code to match this diagram the more confident we can be that our model actually models what we think it does. In this section, we wish to model the eight-bit registered incrementer shown in Figure 14.

4.1. RTL Model of Registered Incrementer

Figure 16 shows the Verilog code which corresponds to the diagram in Figure 14. Every Verilog file should begin with a header comment as shown on lines 1–9 in Figure 16. The header comment should identify the primary module in the file, and include a brief description of what the module does. Reserve discussion of the actual implementation for later in the file. In general, you should attempt to keep lines in your Verilog source code to less than 74 characters. This will make your code easier to read, enable printing on standard sized paper, and facilitate viewing two source files side-by-side on a single monitor. Note that the code in Figure 16 is artificially narrow so we can display two code listings side-by-side. Lines 11–12 create an “include guard” using the Verilog pre-processor. An include guard ensures that even if we include this Verilog file multiple times the modules within the file will only be declared once. Without include guards, the Verilog compiler will likely complain that the same module has been declared multiple times. Make sure that you have the corresponding end of the include guard at the bottom of your Verilog source file as shown on line 43.

Unlike Python, Verilog does not have a clean way to manage namespaces for macros and module names. This means that if you use the same macro or module name in two different files it will create a namespace collision which can potentially be very difficult to debug. We will follow very specific naming conventions to eliminate any possibility of a namespace collision. Our convention will to use the subdirectory path as a prefix for all Verilog macro and module names. Since the registered incrementer is in the directory `tut4_verilog/regincr`, we will use `TUT4_VERILOG_REGINCR_` as a prefix for all macro names and `tut4_verilog_regincr_` as a prefix for all module names. You can see this prefix being used for the macros on lines 11–12 and for the module name on line 14. To reiterate, *Verilog macro and module name must use the subdirectory path as a prefix*. While a bit tedious, this is essential to avoiding namespace collisions.

As always, we begin by identifying the module’s interface which in this case will include an eight-bit input port, eight-bit output port, and a clock input. Lines 15–20 in Figure 16 illustrate how we represent this interface using Verilog. A common mistake is to forget the semicolon (;) on line 20. A couple of comments about the coding conventions that we will be using in this course. All module names should always include the subproject name as a prefix (e.g., `ex_regincr_`). The portion of the name after this prefix should usually use `CamelCaseNaming`; each word begins with a capital letter without any underscores (e.g., `RegIncr`). Port names (as well as variable and module instance names) should use `underscore_naming`; all lowercase with underscores to separate words. As shown on lines 16–19, ports should be listed one per line with a two space initial indentation. The bitwidth specifiers and port names should all line up vertically. As shown on lines 15 and 20, the opening and closing parenthesis should be on their own separate lines. Carefully group ports to help the reader understand how these ports are related. Use port names (as well as variable and module instance

```

1 //=====
2 // Registered Incrementer
3 //=====
4 // This is a simple example of a module
5 // for a registered incrementer which
6 // combines a positive edge triggered
7 // register with a combinational +1
8 // incrementer. We use flat register-
9 // transfer-level modeling.
10
11 `ifndef TUT4_VERILOG_REGINCR_REG_INCR_V
12 `define TUT4_VERILOG_REGINCR_REG_INCR_V
13
14 module tut4_verilog_regincr_RegIncr
15 (
16     input logic    clk,
17     input logic    reset,
18     input logic [7:0] in,
19     output logic [7:0] out
20 );
21
22     // Sequential logic
23
24     logic [7:0] reg_out;
25     always_ff @( posedge clk ) begin
26         if ( reset )
27             reg_out <= '0;
28         else
29             reg_out <= in;
30     end
31
32     // Combinational logic
33
34     logic [7:0] temp_wire;
35     always_comb begin
36         temp_wire = reg_out + 1;
37     end
38
39     assign out = temp_wire;
40
41 endmodule
42
43 `endif /* TUT4_VERILOG_REGINCR_REG_INCR_V */

```

Figure 15: Registered Incrementer – An eight-bit registered +1 incrementer corresponding to the diagram in Figure 14.

```

1 //=====
2 // Registered Incrementer
3 //=====
4 // This is a simple example of a module
5 // for a registered incrementer which
6 // combines a positive edge triggered
7 // register with a combinational +1
8 // incrementer. We use flat register-
9 // transfer-level modeling.
10
11 `ifndef TUT4_VERILOG_REGINCR_REG_INCR_V
12 `define TUT4_VERILOG_REGINCR_REG_INCR_V
13
14 module tut4_verilog_regincr_RegIncr
15 (
16     input    clk,
17     input    reset,
18     input [7:0] in,
19     output [7:0] out
20 );
21
22     // Sequential logic
23
24     reg [7:0] reg_out;
25     always @( posedge clk ) begin
26         if ( reset )
27             reg_out <= 0;
28         else
29             reg_out <= in;
30     end
31
32     // Combinational logic
33
34     reg [7:0] temp_wire;
35     always @(*) begin
36         temp_wire = reg_out + 1;
37     end
38
39     assign out = temp_wire;
40
41 endmodule
42
43 `endif /* TUT4_VERILOG_REGINCR_REG_INCR_V */

```

Figure 16: Registered Incrementer – An eight-bit registered +1 incrementer using Verilog-2001 constructs.

names) that are descriptive; prefer longer descriptive names (e.g., `write_en`) over shorter confusing names (e.g., `wen`).

Lines 22–39 model the internal behavior of the module. We usually prefer using two spaces for each level of indentation; larger indentation can quickly result in significantly wasted horizontal space. *You should always use spaces and never insert any real tab characters into your source code.* You must limit yourself to synthesizable RTL for modeling your design. We will exclusively use two kinds of `always` blocks: `always_ff` concurrent blocks to model sequential logic (analogous to PyMTL `@s.tick_rtl` concurrent blocks) and `always_comb` concurrent blocks to model combinational logic (analogous to PyMTL `@s.combinational` concurrent blocks). We require students to clearly distinguishing between the portions of your code that are meant to model sequential logic from those portions meant to model combinational logic. This simple guideline can save significant frustration by making it

easy to see subtle errors. For example, by convention we should only use non-blocking assignments in sequential logic (e.g., the `<=` operator on line 27) and we should only use blocking assignments in combinational logic (e.g., the `=` operator on line 36). We use the variable `reg_out` to hold the intermediate value between the register and the incrementer, and we use the variable `temp_wire` to hold the intermediate value between the incrementer and the output port. `reg_out` is modeling a register while `temp_wire` is modeling a wire. Notice that both of these variables use the `logic` data type; what makes one model a register while the other models a wire is how these variables are used. The sequential concurrent block update to `reg_out` means it models a register. The combinational concurrent block update to `temp_wire` means it models a wire.

The register incrementer illustrates the two fundamental ways to model combinational logic. We have used an `always_comb` concurrent block to model the actual incrementer logic and a continuous assignment statement (i.e., `assign`) to model connecting the temporary wire to the output port. We could just have easily written logic as part of the `assign` statement. For example, we could have used `assign out = reg_out + 1` and skipped the `always_comb` concurrent block. In general, we prefer continuous assignment statements over `always @(*)` concurrent blocks to model combinational logic, since it is easier to model less-realistic hardware using `always_comb` concurrent blocks. There is usually a more direct one-to-one mapping from continuous assignment statements to real hardware. However, there are many cases where it is significantly more convenient to use `always_comb` concurrent blocks or just not possible to use continuous assignment statements. Students will need to use their judgment to determine the most elegant way to represent the hardware they are modeling while still ensuring there is a clear mapping from the model to the target hardware.

Figure 15 illustrates a few new SystemVerilog constructs. Figure 16 illustrates the exact same registered incrementer implemented using the older Verilog-2001 hardware description language. Verilog-2001 uses `reg` and `wire` to specify variables instead of `logic`. All ports are of type `wire` by default. Determining when to use `reg` and `wire` is subtle and error prone. Note that `reg` is a misnomer; it does *not* model a register! On line 34, we must declare `temp_wire` to be of type `reg` even though it is modeling a wire. Verilog-2001 requires using `reg` for any variable written by an `always` concurrent block. Verilog-2001 uses a generic `always` block for both sequential and combinational concurrent blocks. While the `always @(*)` syntax is an improvement over the need in Verilog-1995 to explicitly define sensitivity lists, `always_ff` and `always_comb` more directly capture designer intent and allow automated tools to catch common errors. For example, a Verilog simulator can catch errors where a designer accidentally uses a non-blocking assignment in an `always_comb` concurrent block, or where a designer accidentally writes the same `logic` variable from two different `always_comb` concurrent blocks. SystemVerilog is growing in popularity and increasingly becoming the de facto replacement for Verilog-2001, so it is worthwhile to carefully adopt new SystemVerilog features that can improve designer productivity.

Edit the Verilog source file named `RegIncr.v` in the `tut4_verilog/regincr` subdirectory using your favorite text editor. Add the combinational logic shown on lines 34–39 in Figure 16 which models the incrementer logic. We will be using `iverilog` to simulate this registered incrementer module, and `iverilog` does not currently support `always_ff` and `always_comb`, which is why we are using the Verilog-2001 construct for now.

4.2. Simulating a Model using `iverilog`

Now that we have developed a new hardware module, we can test its functionality using a simulation harness. Figure 17 shows an ad-hoc test using non-synthesizable Verilog. Note that we must explicitly include any Verilog files which contain modules that we want to use; Line 5 includes the Verilog source file that contains the registered incrementer. Lines 11–12 setup a clock with a period

```

1 //=====
2 // RegIncr Ad-Hoc Testing
3 //=====
4
5 `include "../tut4_verilog/regincr/RegIncr.v"
6
7 module top;
8
9     // Clocking
10
11     logic clk = 1;
12     always #5 clk = ~clk;
13
14     // Instaniate the design under test
15
16     logic      reset = 1;
17     logic [7:0] in;
18     logic [7:0] out;
19
20     tut4_verilog_regincr_RegIncr reg_incr
21     (
22         .clk   (clk),
23         .reset (reset),
24         .in    (in),
25         .out   (out)
26     );
27
28     // Verify functionality
29
30     initial begin
31
32         // Dump waveforms
33
34         $dumpfile("regincr-iverilog-sim.vcd");
35         $dumpvars;
36
37         // Reset
38
39         #11;
40         reset = 1'b0;
41
42         // Test cases
43
44         in = 8'h00;
45         #10;
46         if ( out != 8'h01 ) begin
47             $display( "ERROR: out, expected = %x, actual = %x", 8'h01, out );
48             $finish;
49         end
50
51         in = 8'h13;
52         #10;
53         if ( out != 8'h14 ) begin
54             $display( "ERROR: out, expected = %x, actual = %x", 8'h14, out );
55             $finish;
56         end
57
58         in = 8'h27;
59         #10;
60         if ( out != 8'h28 ) begin
61             $display( "ERROR: out, expected = %x, actual = %x", 8'h28, out );
62             $finish;
63         end
64
65         $display( "*** PASSED ***" );
66         $finish;
67     end
68
69 endmodule

```

Figure 17: Simulator for Registered Incrementer²⁶ – A Verilog simulator for the eight-bit registered incrementer in Figure 16.

of 10 time steps. Notice that we are assigning an initial value to the `clk` net on line 11 and then modifying this net every five timesteps; setting initial values such as this is not synthesizable and should only be used in test harnesses. If you need to set an initial value in your design, you should use properly constructed reset logic.

Lines 19–24 instantiate the device under test. Notice that we use `underscore_naming` for the module instance name (e.g., `reg_incr`). You should almost always use named port binding (as opposed to positional port binding) to connect nets to the ports in a module instance. Lines 21–23 illustrate the correct coding convention with one port binding per line and the ports/nets vertically aligned. As shown on lines 20 and 24 the opening and closing parenthesis should be on their own separate lines. Although this may seem verbose, this coding style can significantly reduce errors by making it much easier to quickly visualize how ports are connected.

Lines 28–62 illustrate an `initial` block which executes at the very beginning of the simulation. `initial` blocks are not synthesizable and should only be used in test harnesses. Lines 32–33 instruct the simulator to dump waveforms for all nets. Line 35 is a delay statement that essentially waits for one timestep. Delay statements are not synthesizable and should only be used in test harnesses. Lines 39–44 implement a test by setting the inputs of the device under test, waiting for 10 time steps, and then checking that the output is as expected. If there is an error, we display an error message and stop the simulation. We include two more tests, and if we make it to the bottom of the `initial` block then the test has passed.

Edit the Verilog simulation harness named `regincr-iverilog-sim.v` in the `tut4_verilog/regincr` subdirectory using your favorite text editor. Add the code on lines 20–26 in Figure 17 to instantiate the registered incrementer model. Then use `iverilog` to compile this simulator and run the simulation as follows:

```
% cd ${TUTROOT}/build
% iverilog -g2012 -o regincr-iverilog-sim ../tut4_verilog/regincr/regincr-iverilog-sim.v
% ./regincr-iverilog-sim
```

If everything goes as expected, then the ad-hoc test should display `*** PASSED ***`.

- ★ *To-Do On Your Own:* Edit the register incrementer so that it now increments by +2 instead of +1. Use an assign statement instead of the `always @(*)` concurrent block to do the incrementer logic. Recompile, rerun the ad-hoc test, and verify that the tests no longer pass. Modify the ad-hoc test so that it includes the correct reference outputs for a +2 incrementer, recompile, rerun the ad-hoc test, and verify that the test now pass. When you are finished, edit the register incrementer so that it again increments by +1.

4.3. Verifying a Model with Unit Testing

Writing test and simulation harnesses in Verilog is very tedious. There are some industry standard verification frameworks based on SystemVerilog, such as the Open Verification Methodology (OVM) and the Universal Verification Methodology (UVM), but these frameworks are very heavyweight and are not supported by open-source tools. In this course, we will be using PyMTL for FL modeling, but also to productively write test and simulation harnesses for our Verilog RTL models. PyMTL includes support for *Verilog import* by writing a special PyMTL wrapper model. Once we have created this wrapper model, we can use all of the sophisticated techniques we learned in the PyMTL tutorial for writing test and simulation harnesses.

```

1  =====
2  # RegIncr
3  =====
4
5  from pymtl import *
6
7  class RegIncr( VerilogModel ):
8
9      # Verilog module setup
10
11     vprefix = "tut4_verilog_regincr"
12
13     # Constructor
14
15     def __init__( s ):
16
17         # Port-based interface
18
19         s.in_ = InPort ( Bits(8) )
20         s.out = OutPort ( Bits(8) )
21
22         # Verilog ports
23
24         s.set_ports({
25             'clk'   : s.clk,
26             'reset' : s.reset,
27             'in'    : s.in_,
28             'out'   : s.out,
29         })
30
31     # Line Tracing
32
33     def line_trace( s ):
34         return "{} ( ) {}".format( s.in_, s.out )

```

Figure 18: Registered Incrementer Wrapper – PyMTL wrapper for the Verilog module shown in Figure 14.

Figure 18 illustrates such a PyMTL wrapper. Note that on line 7, we inherit from `VerilogModel` *not* `Model`. This is how we tell PyMTL that this is a special wrapper model. One line 11, we specify the `vprefix` which is the prefix discussed above for avoiding namespace collisions. The `vprefix` must match the prefix used for the name of the Verilog model we are wrapping. The interface on lines 19–20 should be identical to what we would use with a standard PyMTL implementation. On lines 24–29, we must specify how the Verilog ports correspond to the PyMTL ports. The Verilog ports are listed on the left, and the PyMTL ports are listed on the right. Notice that you must explicitly include the `clk` and `reset` signals. Finally, you can include a line tracing function in the wrapper, but keep in mind that you can only access the ports of the wrapper for line tracing. We will see later in this tutorial how we can use line tracing within our Verilog modules.

Once we have a PyMTL wrapper, we can use the exact same test harness we used in the PyMTL tutorial. The following commands will run the `RegIncr_test.py` test script.

```

% cd ${TUTROOT}/build
% py.test ../tut4_verilog/regincr/RegIncr_test.py -sv
% py.test ../tut4_verilog/regincr/RegIncr_extra_test.py -sv

```

Keep in mind that PyMTL uses the open-source `verilator` simulator instead of `iverilog` for simulating Verilog modules. The framework will output the specific `verilog` command being used to

create the simulator. You can use all of the `py.test` features we learned in the PyMTL tutorial including using the `-s` option to output line tracing and the `--dump-vcd` option to output VCD.

```
% cd ${TUTROOT}/build
% py.test ../tut4_verilog/regincr/RegIncr_test.py -s --dump-vcd
% gtkwave tut4_verilog/regincr/RegIncr_test.test_basic.verilator1.vcd
```

Note that the actual name of the VCD file will be a little different compared to when we use PyMTL RTL models. Also note that PyMTL will actually create *two* different VCD files. You always want to open the one with the `verilator1.vcd` suffix. In the lab assignments, we will mostly provide you with the appropriate top-level PyMTL wrappers, but you may need to write your own wrappers for new child models you want to test.

As we learned in the PyMTL tutorial, when testing an entire directory, we can use an iterative process to “zoom” in on a failing test case. We will start by running all tests in the directory to see an overview of which tests are passing and which tests are failing. We then explicitly run a single test script with the `-v` command line option to see which specific test cases are failing. Finally, we will use the `-k` or `-x` command line options with `--tb`, `-s`, and/or `--dump-vcd` command line option to generate error output, line traces, and/or waveforms for the failing test case. Here is an example of this three-step process to “zoom” in on a failing test case:

```
% cd ${TUTROOT}/build
% py.test ../tut4_verilog/regincr
% py.test ../tut4_verilog/regincr/RegIncr2stage_test.py -v
% py.test ../tut4_verilog/regincr/RegIncr2stage_test.py -v -x --tb=short
```

- ★ *To-Do On Your Own:* Edit the register incremter so that it now increments by +2 instead of +1. Recompile, rerun the unit test, and verify that the tests no longer pass. Modify the unit test so that it includes the correct reference outputs for a +2 incremter, recompile, rerun the unit test, and verify that the test now pass. When you are finished, edit the register incremter so that it again increments by +1.

4.4. Reusing a Model with Structural Composition

As in PyMTL, we can use modularity and hierarchy to structurally compose small, simple models into large, complex models. Figure 19 shows a two-stage registered incremter that uses structural composition to instantiate and connect two instances of a single-stage registered incremter. Figure 20 shows the corresponding Verilog module. Line 11 uses a `'include` to include the child model that we will be reusing. Notice how we must use the full path (from the root of the project) to the Verilog file we want to include.

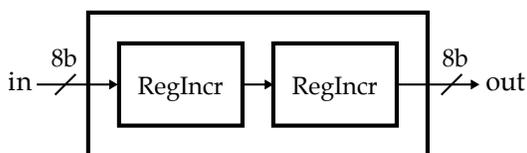


Figure 19: Block Diagram for Two-Stage Registered Incrementer – An eight-bit two-stage registered incremter that reuses the registered incremter in Figure 14 through structural composition.

```

1 //=====
2 // RegIncr2stage
3 //=====
4 // Two-stage registered incrementer that uses structural composition to
5 // instantiate and connect two instances of the single-stage registered
6 // incrementer.
7
8 `ifndef TUT4_VERILOG_REGINCR_REG_INCR_2STAGE_V
9 `define TUT4_VERILOG_REGINCR_REG_INCR_2STAGE_V
10
11 `include "tut4_verilog/regincr/RegIncr.v"
12
13 module tut4_verilog_regincr_RegIncr2stage
14 (
15     input logic    clk,
16     input logic    reset,
17     input logic [7:0] in,
18     output logic [7:0] out
19 );
20
21 // First stage
22
23 logic [7:0] reg_incr_0_out;
24
25 tut4_verilog_regincr_RegIncr reg_incr_0
26 (
27     .clk    (clk),
28     .reset  (reset),
29     .in     (in),
30     .out    (reg_incr_0_out)
31 );
32
33 // Second stage
34
35 tut4_verilog_regincr_RegIncr reg_incr_1
36 (
37     .clk    (clk),
38     .reset  (reset),
39     .in     (reg_incr_0_out),
40     .out    (out)
41 );
42
43 endmodule
44
45 `endif /* TUT4_VERILOG_REGINCR_REG_INCR_2STAGE_V */

```

Figure 20: Two-Stage Registered Incrementer – An eight-bit two-stage registered incrementer corresponding to Figure 19. This model is implemented using structural composition to instantiate and connect two instances of the single-stage register incrementer.

Lines 25–31 instantiate the first registered incrementer and lines 35–41 instantiate the second registered incrementer. As mentioned above, we should almost always use named port binding to connect nets to the ports in a module instance. Lines 27–30 illustrate the correct coding convention with one port binding per line and the ports/nets vertically aligned. As shown on lines 26 and 31 the opening and closing parenthesis should be on their own separate lines. We usually declare signals that will be connected to output ports immediately before instantiating the module.

We need to write a new PyMTL wrapper for our two-stage registered incrementer, although it will be essentially the same as the wrapper shown in Figure 18 except with a different class name. This illustrates a key point: the PyMTL wrapper simply captures the Verilog *interface* and is largely unconcerned with the implementation.

```

1  module vc_Incrementer
2  #(
3      parameter p_nbits    = 1,
4      parameter p_inc_value = 1
5  )(
6      input  logic [p_nbits-1:0] in,
7      output logic [p_nbits-1:0] out
8  );
9
10     assign out = in + p_inc_value;
11
12 endmodule

```

Figure 21: Parameterized Incrementer from vc – A combinational incrementer from vc that is parameterized by both the port bitwidth and the incrementer amount.

Edit the Verilog source file named `RegIncr2stage.v`. Add lines 33-41 from Figure 20 to instantiate and connect the second stage of the two-stage registered incrementer. Then reuse the test harness from the PyMTL tutorial as follows:

```

% cd ${TUTROOT}/build
% py.test ../tut4_verilog/regincr/RegIncr2stage_test.py -sv

```

- ★ *To-Do On Your Own:* Create a three-stage registered incrementer similar in spirit to the two-stage registered incrementer in Figure 19. Verify your design by writing a test script that uses test vectors.

4.5. Parameterizing a Model with Static Elaboration

As we learned in the PyMTL tutorial, To facilitate model reuse and productive design-space exploration, we often want to implement parameterized models. A common example is to parameterize models by the bitwidth of various input and output ports. The registered incrementer in Figure 16 is designed for only for only eight-bit input values, but we may want to reuse this model in a different context with four-bit input values or 16-bit input values. We can use Verilog *parameters* to parameterize the port bitwidth for the registered incrementer shown in Figure 16; we would replace references to constant 7 with a reference to `nbits-1`. Now we can specify the port bitwidth for our register incrementer when we construct the model. We have included a library of parameterized Verilog RTL models in the `vc` subdirectory. Figure 21 shows a combinational incrementer from `vc` that is parameterized by both the port bitwidth and the incrementer amount. The parameters are specified using the special syntax shown on lines 2–5. By convention, we use a `p_` prefix when naming parameters.

Unfortunately, writing highly parameterized models in Verilog can be very tedious or even impossible, which is one key motivation for the PyMTL framework. Having said this, Verilog-2001 does provide `generate` statements which are meant for static elaboration. Recall that static elaboration happens at compile time, not runtime. We can use static elaboration to *generate* hardware which is fundamentally different from *modeling* hardware. Figure 22 illustrates using `generate` statements to create a multi-stage registered incrementer that is parameterized by the number of stages. The number of stages is specified using the the `p_nstages` parameter shown on line 13. We create a array of signals to hold the intermediate values between stages (line 25), and then we use a `generate` for loop to instantiate and connect the stages. Using `generate` statements is one of the more advanced parts of Verilog, so we will not go into more detail within this tutorial. We can reuse the test harness from the PyMTL tutorial as follows:

```

% cd ${TUTROOT}/build
% py.test ../tut4_verilog/regincr/RegIncrNstage_test.py -sv

```

```

1 //=====
2 // RegIncrNstage
3 //=====
4 // Registered incrementer that is parameterized by the number of stages.
5
6 `ifndef TUT4_VERILOG_REGINCR_REG_INCR_NSTAGE_V
7 `define TUT4_VERILOG_REGINCR_REG_INCR_NSTAGE_V
8
9 `include "tut4_verilog/regincr/RegIncr.v"
10
11 module tut4_verilog_regincr_RegIncrNstage
12 #(
13     parameter p_nstages = 2
14 )
15     input logic      clk,
16     input logic      reset,
17     input logic [7:0] in,
18     output logic [7:0] out
19 );
20
21     // This defines an _array_ of signals. There are p_nstages+1 signals
22     // and each signal is 8 bits wide. We will use this array of signals to
23     // hold the output of each registered incrementer stage.
24
25     logic [7:0] reg_incr_out [p_nstages+1];
26
27     // Connect the input port of the module to the first signal in the
28     // reg_incr_out signal array.
29
30     assign reg_incr_out[0] = in;
31
32     // Instantiate the registered incrementers and make the connections
33     // between them using a generate block.
34
35     genvar i;
36     generate
37     for ( i = 0; i < p_nstages; i = i + 1 ) begin: gen
38
39         tut4_verilog_regincr_RegIncr reg_incr
40         (
41             .clk      (clk),
42             .reset    (reset),
43             .in       (reg_incr_out[i]),
44             .out      (reg_incr_out[i+1])
45         );
46
47     end
48     endgenerate
49
50     // Connect the last signal in the reg_incr_out signal array to the
51     // output port of the module.
52
53     assign out = reg_incr_out[p_nstages];
54
55 endmodule
56
57 `endif /* TUT4_VERILOG_REGINCR_REG_INCR_NSTAGE_V */

```

Figure 22: N-Stage Registered Incrementer – A parameterized registered incrementer where the number of stages is specified using a Verilog parameter.

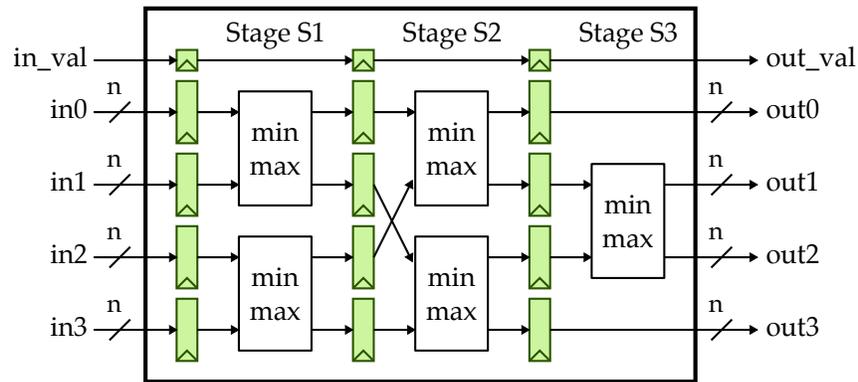


Figure 23: Block Diagram for Four-Element Sorter – An n -bit pipelined four-element sorter which arranges the four elements in ascending order.

5. Sort Unit

The previous section introduces the key Verilog concepts and primitives that we will use to implement more complex RTL models including: declaring a port-based module interface; declaring internal state and wires using logic variables; declaring `always @(posedge clk)` concurrent blocks to model logic that executes on every rising clock edge; declaring `always (*)` concurrent blocks to model combinational logic that executes one or more times within a clock cycle; and creating PyMTL wrappers.

In this section, we will explore how we can implement the same sorting unit previously seen in the PyMTL tutorial except using the Verilog hardware description language. As a reminder, the simple pipelined four-element sorter is shown in Figure 23. Each min/max unit compares its inputs and sends the smaller value to the top output port and the larger value to the bottom output. This specific implementation is pipelined into three stages, such that the critical path should be through a single min/max unit. Input and output valid signals indicate when the input and output elements are valid. Most of the code for this section is provided for you in the `tut4_verilog/sort` subdirectory.

5.1. Flat Sorter Implementation

We will begin by exploring a flat implementation of the sorter that does not instantiate any additional child modules. This implementation is provided for you in the file named `SortUnitFlatRTL.v`. Figure 24 illustrates the Verilog code that describes the interface for the sorter. Notice how we have parameterized the interface by the bitwidth of each element. Lines 2–4 declare a parameter named `p_nbits` and give it a default value of one bit. We use this parameter when declaring the bitwidth of the input and output ports, and we will also use this parameter in the implementation.

Figure 25 shows the first pipeline stage of the flat implementation of the sorter. Notice how we use the parameter `p_nbits` to declare various internal registers and wires. We cleanly separate the sequential logic from the combinational logic. We use comments and explicit suffixes to make it clear what pipeline stage we are modeling.

The corresponding wrapper is in `SortUnitFlatRTL.py` and is shown in Figure 26. There are two important differences from the PyMTL wrapper for the registered incrementer. On line 12, we set `vlinetrace` to be `True`. This tells the PyMTL framework that the Verilog module includes its own

```

1 module tut4_verilog_sort_SorterFlat
2 #(
3     parameter p_nbits = 1
4 )
5     input logic          clk,
6     input logic         reset,
7
8     input logic         in_val,
9     input logic [p_nbits-1:0] in0,
10    input logic [p_nbits-1:0] in1,
11    input logic [p_nbits-1:0] in2,
12    input logic [p_nbits-1:0] in3,
13
14    output logic         out_val,
15    output logic [p_nbits-1:0] out0,
16    output logic [p_nbits-1:0] out1,
17    output logic [p_nbits-1:0] out2,
18    output logic [p_nbits-1:0] out3
19 );

```

Figure 24: Interface for the Four-Element Sorter – The interface corresponds to the diagram in Figure 23 and is parameterized by the bitwidth of each element.

line tracing code. On lines 28–30, we can specify how the PyMTL parameters (e.g., `nbits`) correspond to the Verilog parameters (e.g., `p_nbits`). We can run the unit tests for this module as follows:

```

% cd ${TUTROOT}/build
% py.test ../tut4_verilog/sort/SortUnitFlatRTL_test.py

```

The design should pass all of the tests.

- ★ *To-Do On Your Own:* The sorter currently sorts the four input numbers from smallest to largest. Change to the sorter implementation so it sorts the numbers from largest to smallest. Recompile and rerun the unit test and verify that the tests are no longer passing. Modify the tests so that they correctly capture the new expected behavior. Make a copy of the sorter implementation file so you can put things back to the way they were when you are finished.

```

1  //-----
2  // Stage S0->S1 pipeline registers
3  //-----
4
5  logic          val_S1;
6  logic [p_nbits-1:0] elm0_S1;
7  logic [p_nbits-1:0] elm1_S1;
8  logic [p_nbits-1:0] elm2_S1;
9  logic [p_nbits-1:0] elm3_S1;
10
11 always_ff @( posedge clk ) begin
12     val_S1 <= (reset) ? 0 : in_val;
13     elm0_S1 <= in0;
14     elm1_S1 <= in1;
15     elm2_S1 <= in2;
16     elm3_S1 <= in3;
17 end
18
19 //-----
20 // Stage S1 combinational logic
21 //-----
22 // Note that we explicitly catch the case where the elements contain
23 // X's and propagate X's appropriately. We would not need to do this if
24 // we used a continuous assignment statement with a ternary conditional
25 // operator.
26
27 logic [p_nbits-1:0] elm0_next_S1;
28 logic [p_nbits-1:0] elm1_next_S1;
29 logic [p_nbits-1:0] elm2_next_S1;
30 logic [p_nbits-1:0] elm3_next_S1;
31
32 always_comb begin
33
34     // Sort elms 0 and 1
35
36     if ( elm0_S1 <= elm1_S1 ) begin
37         elm0_next_S1 = elm0_S1;
38         elm1_next_S1 = elm1_S1;
39     end
40     else if ( elm0_S1 > elm1_S1 ) begin
41         elm0_next_S1 = elm1_S1;
42         elm1_next_S1 = elm0_S1;
43     end
44     else begin
45         elm0_next_S1 = 'x;
46         elm1_next_S1 = 'x;
47     end
48
49     // Sort elms 2 and 3
50
51     if ( elm2_S1 <= elm3_S1 ) begin
52         elm2_next_S1 = elm2_S1;
53         elm3_next_S1 = elm3_S1;
54     end
55     else if ( elm2_S1 > elm3_S1 ) begin
56         elm2_next_S1 = elm3_S1;
57         elm3_next_S1 = elm2_S1;
58     end
59     else begin
60         elm2_next_S1 = 'x;
61         elm3_next_S1 = 'x;
62     end
63
64 end

```

Figure 25: First Stage of the Flat Sorter Implementation – First pipeline stage of the sorter using a flat implementation corresponding to the diagram in Figure 23.

```

1  =====
2  # SortUnitFlatRTL
3  =====
4
5  from pymtl import *
6
7  class SortUnitFlatRTL( VerilogModel ):
8
9      # Verilog module setup
10
11     vprefix = "tut4_verilog_sort"
12     vlinetrace = True
13
14     # Constructor
15
16     def __init__( s, nbits=8 ):
17
18         # Interface
19
20         s.in_val = InPort (1)
21         s.in_    = [ InPort (nbits) for _ in range(4) ]
22
23         s.out_val = OutPort(1)
24         s.out     = [ OutPort (nbits) for _ in range(4) ]
25
26         # Verilog parameters
27
28         s.set_params({
29             'p_nbits' : nbits,
30         })
31
32         # Verilog ports
33
34         s.set_ports({
35             'clk'      : s.clk,
36             'reset'    : s.reset,
37
38             'in_val'   : s.in_val,
39             'in0'      : s.in_[0],
40             'in1'      : s.in_[1],
41             'in2'      : s.in_[2],
42             'in3'      : s.in_[3],
43
44             'out_val'  : s.out_val,
45             'out0'     : s.out[0],
46             'out1'     : s.out[1],
47             'out2'     : s.out[2],
48             'out3'     : s.out[3],
49         })

```

Figure 26: Sort Unit Wrapper – PyMTL wrapper for the Verilog RTL implementation of the flat sort unit.

5.2. Using Verilog Line Traces

We learned about line tracing in the PyMTL tutorial. We have implemented a small library that enables implementing line tracing code directly within your Verilog modules. While not as elegant as PyMTL line tracing, Verilog line tracing will still be an essentially way to visualize our designs. You can use the the `-s` command-line option to see a line trace of the four-element sorter.

```
% cd ${TUTROOT}/build
% py.test ../tut4_verilog/sort/SortUnitFlatRTL_test.py -k test_basic -s
```

Figure 27 shows a portion of a representative line trace with some additional annotation. The first column indicates the current cycle. There are fixed-width columns showing the inputs and outputs of the module along with the state in the S1, S2, and S3 pipeline registers at the beginning of the cycle. Notice that we use spaces when the data is invalid which improves readability and makes it easy to see when the hardware is actually doing useful work. If you compare this line trace to the line trace generated in the PyMTL tutorial, they should look identical. Both PyMTL and Verilog enable the same kind of cycle-accurate modeling. Regardless of whether we are using PyMTL or Verilog, line traces are a powerful way to visualize your design and debug both correctness and performance issues.

| 1 | cycle | input ports | stage S1 | stage S2 | stage S3 | output ports |
|---|-------|---------------|---------------|---------------|---------------|---------------|
| 2 | | | | | | |
| 3 | 2: | | | | | |
| 4 | 3: | {04,02,03,01} | | | | |
| 5 | 4: | | {04,02,03,01} | | | |
| 6 | 5: | | | {02,04,01,03} | | |
| 7 | 6: | | | | {01,03,02,04} | {01,02,03,04} |
| 8 | 7: | | | | | |

Figure 27: Line Trace Output for Sort Unit RTL Model – This line trace is for the `test_basic` test case and is annotated to show what each column corresponds to in the model. Each line corresponds to one (and only one!) cycle, and the fixed-width columns correspond to either the state at the beginning of the corresponding cycle or the output of combinational logic during that cycle. If the valid bit is not set, then the corresponding list of values is not shown.

```
1  `ifndef SYNTHESIS
2
3  logic [ `VC_TRACE_NBITS-1:0 ] str;
4  `VC_TRACE_BEGIN
5  begin
6
7      // Inputs
8
9      $sformat( str, "%x,%x,%x,%x", in0, in1, in2, in3 );
10     vc_trace.append_val_str( trace_str, in_val, str );
11     vc_trace.append_str( trace_str, "|" );
12
13     ...
14
15 end
16 `VC_TRACE_END
17
18 `endif /* SYNTHESIS */
```

Figure 28: Example of Line Tracing Code – This code generates the first fixed-width column for the line trace shown in Figure 27

As mentioned above, we provide a verilog component (VC) library with various useful Verilog modules, including one module in `vc/trace.v` which makes it easier to create line traces in your own Verilog modules. Figure 28 shows a small snippet of code that is used in the sorter implementation to trace the input ports. Notice how we have wrapped the line tracing code in `'ifndef SYNTHESIS` and `'endif` to clearly indicate that this code is not synthesizable even though it is included in our design. Line 3 declares a temporary string variable that we will use when converting nets into strings. Lines 4–16 use helper macros to declare a task called `trace` which takes a special argument called `trace_str` that holds the current line trace string. The job of this task is to append trace information to the `trace_str` that describes the current state and operation of this module. The `vc_trace.append_str` and `vc_trace.append_val_str` helper tasks add strings to the line trace. You can also build line traces hierarchically by explicitly calling the `trace` task on a child module. Although we will provide significant line tracing code in each lab harness, you are also strongly encouraged to augment this code with your own line tracing.

- ★ *To-Do On Your Own:* Modify the line tracing code to show the pipeline stage label (`in`, `S1`, `S2`, `S3`, `out`) before each stage. After your modifications, the line trace might look something like this:

```
24: in:{05,07,06,08}|S1:{a5,a3,a2,a7}|S2:{03,04,01,02}|S3:          |out:
```

5.3. Structural Sorter Implementation

The flat implementation in `SortUnitFlatRTL.v` is complex and monolithic and it fails to really exploit the structure inherent in the sorter. Just as in the PyMTL tutorial, we can use modularity and hierarchy to divide complicated designs into smaller more manageable units; these smaller units are easier to design and can be tested independently before integrating them into larger, more complicated designs. We have started a structural implementation in `SortUnitStructRTL.v`; the structural implementation will have an identical interface and behavior as the flat implementation in `SortUnitFlatRTL.v`.

Figure 29 shows the first pipeline stage of the structural implementation of the sorter. Our design instantiates three kinds of modules: `vc_ResetReg`, `vc_Reg`, and `tut4_verilog_sort_MinMaxUnit`. The register modules are provided in the VC library. Notice how we still use the parameter `p_nbits` to declare various internal variables, but in addition, we use this parameter when instantiating parameterized sub-modules. For example, the `vc_Reg` module is parameterized, and this allows us to easily create pipeline registers of any bitwidth. Even though we are using a structural implementation strategy, we still cleanly separate the sequential logic from the combinational logic. We still use comments and explicit suffixes to make it clear what pipeline stage we are modeling.

```

1  //-----
2  // Stage S0->S1 pipeline registers
3  //-----
4
5  logic val_S1;
6
7  vc_ResetReg#(1) val_S0S1
8  (
9    .clk    (clk),
10   .reset  (reset),
11   .d      (in_val),
12   .q      (val_S1)
13  );
14
15  // This is probably the only place where it might be acceptable to use
16  // positional port binding since (a) it is so common and (b) there are
17  // very few ports to bind.
18
19  logic [p_nbits-1:0] elm0_S1;
20  logic [p_nbits-1:0] elm1_S1;
21  logic [p_nbits-1:0] elm2_S1;
22  logic [p_nbits-1:0] elm3_S1;
23
24  vc_Reg#(p_nbits) elm0_S0S1( clk, elm0_S1, in0 );
25  vc_Reg#(p_nbits) elm1_S0S1( clk, elm1_S1, in1 );
26  vc_Reg#(p_nbits) elm2_S0S1( clk, elm2_S1, in2 );
27  vc_Reg#(p_nbits) elm3_S0S1( clk, elm3_S1, in3 );
28
29  //-----
30  // Stage S1 combinational logic
31  //-----
32
33  logic [p_nbits-1:0] mmuA_out_min_S1;
34  logic [p_nbits-1:0] mmuA_out_max_S1;
35
36  tut4_verilog_sort_MinMaxUnit#(p_nbits) mmuA_S1
37  (
38    .in0    (elm0_S1),
39    .in1    (elm1_S1),
40    .out_min (mmuA_out_min_S1),
41    .out_max (mmuA_out_max_S1)
42  );
43
44  logic [p_nbits-1:0] mmuB_out_min_S1;
45  logic [p_nbits-1:0] mmuB_out_max_S1;
46
47  tut4_verilog_sort_MinMaxUnit#(p_nbits) mmuB_S1
48  (
49    .in0    (elm2_S1),
50    .in1    (elm3_S1),
51    .out_min (mmuB_out_min_S1),
52    .out_max (mmuB_out_max_S1)
53  );

```

Figure 29: First Stage of the Structural Sorter Implementation – First pipeline stage of the sorter using a structural implementation corresponding to the diagram in Figure 23.

- ★ *To-Do On Your Own:* The structural implementation is incomplete because the actual implementation of the min/max unit in `MinMaxUnit.v` is not finished. You should go ahead and implement the min/max unit, and then *as always you should write a unit test to verify the functionality of your MinMax unit!* Add some line tracing for the min/max unit. You should have enough experience based on the previous sections to be able to create a unit test from scratch and run it using `py.test`. You should name the new test script `MinMaxUnit_test.py`. You can use the registered incremter model as an example for both implementing the min/max unit and for writing the corresponding test script. Once your min/max unit is complete and tested, then test the structural sorter implementation like this:

```
% cd ${TUTROOT}/build
% py.test ../tut4_verilog/sort/SortUnitStructRTL_test.py -v
% py.test ../tut4_verilog/sort/SortUnitStructRTL_test.py -k test_basic -s
```

The line trace for the sort unit structural RTL model should be the same as in Figure 27, since these are really just two different implementations of the sort unit RTL.

5.4. Evaluating the Sorter Using a Simulator

Just like we can use PyMTL test harnesses to verify Verilog modules, we can also use PyMTL simulation harnesses for evaluation. Since our PyMTL wrapper has the exact same interface as a PyMTL RTL implementation, the exact same simulator from the PyMTL tutorial will work without change.

```
% cd ${TUTROOT}/build
% ../tut4_verilog/sort/sort-sim --stats --impl rtl-flat
% ../tut4_verilog/sort/sort-sim --stats --impl rtl-struct
```

6. Greatest Common Divisor: Verilog Design Example

In this section, we will apply what we have learned in the previous section to study a more complicated hardware unit that calculates the greatest common divisor (GCD) of two input operands. Our design will be exactly the same as what we experimented with in the PyMTL tutorial, except now we will be using the Verilog hardware description language. The code for this section is provided for you in the `tut4_verilog/gcd` subdirectory. The previous examples placed the unit test scripts in the same subdirectory as the models these tests were testing. As we start to explore much larger and more complicated designs, it can be useful to keep all of the unit tests together in a separate test subdirectory. You can see in this example, that all of the unit tests for the GCD unit are placed in the `tut4_verilog/gcd/test` subdirectory.

As a reminder, Figure 30 illustrates the interface for our module. The GCD unit will take two n-bits operands and produce an n-bit result. For flow-control we use the same latency-insensitive `val/rdy` interface that we learned about in the PyMTL tutorial. Our implementation will use Euclid's algorithm to iteratively calculate the GCD. Figure 31 illustrates this algorithm as an executable Python function.

- ★ *To-Do On Your Own:* Experiment with the algorithm using a Python interpreter. Try calculating the GCD for different input values. Add additional debugging output to track what the algorithm is doing each iteration.

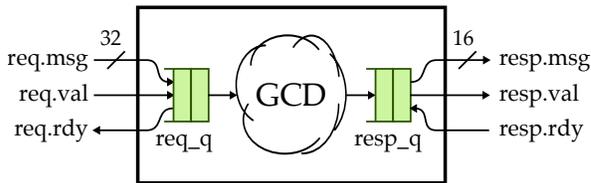


Figure 30: Functional-Level Implementation of GCD Unit – Input and output use latency-insensitive val/rdy interfaces. The input message includes two 16-bit operands; output message is an 16-bit result. Clock and reset signals are not shown.

```

1 def gcd( a, b ):
2     while True:
3         if a < b:
4             a,b = b,a
5         elif b != 0:
6             a = a - b
7         else:
8             return a

```

Figure 31: Euclid’s GCD Algorithm – Iteratively subtract the smaller value from the larger value until one of them is zero, at which time the GCD is the non-zero value. This is executable Python code.

6.1. Control/Datapath Split Implementation

As discussed in the PyMTL tutorial, we will usually divide more complicated designs into two parts: the *datapath* and the *control unit*. The datapath contains the arithmetic operators, muxes, and registers that work on the data, while the control unit is responsible for controlling these components to achieve the desired functionality. The control unit sends *control signals* to the datapath and the datapath sends *status signals* back to the control unit. Figure 32 illustrates the datapath for the GCD unit and Figure 33 illustrates the corresponding finite-state-machine (FSM) control unit. The Verilog code for the datapath, control unit, and the top-level module which composes the datapath and control unit is in `GcdUnitRTL.v`.

Take a look at the datapath interface which is also shown in Figure 35. We clearly identify the data signals, control signals, and status signals. Figure 35 also shows the first two datapath components, but take a look in `GcdUnitRTL.v` to see the entire datapath. Notice how we use a very structural implementation that *exactly* matches the datapath diagram in Figure 32. We leverage several modules from the VC library (e.g., `vc_Mux2`, `vc_ZeroComparator`, `vc_Subtractor`). You should use a similar structural approach when building your own datapaths for this course. For a net that moves data from left to right in the datapath diagram, we usually declare a dedicated wire right before the module instance (e.g., `a_mux_out` and `a_reg_out`). For a net that moves data from right to left in the datapath diagram, we need to declare a dedicated wire right before it is used as an input (e.g., `b_reg_out` and `b_sub_out`).

Take a look at the control unit and notice the stylized way we write FSMs. An FSM-based control unit should have three parts: a sequential concurrent block for the state, a combinational concurrent block for the state transitions, and a combinational concurrent block for the state outputs. We often use case statements to compactly represent the state transitions and outputs. Figure 36 shows the portion of the FSM responsible for setting the output signals. We use a task to set all of the control signals in a single line; as long as the task does not include non-synthesizable constructs (e.g., delay statements or system tasks) the task itself should be synthesizable. Essentially, we have created a “control signal table” in our Verilog code which exactly matches what we might draw on a piece of paper. There is one row for each state or Mealy transition and one column for each control signal. These compact control signal tables simplify coding complicated FSMs (or indeed other kinds of control logic) and can enable a designer to quickly catch bugs (e.g., are the enable signals always set to either zero or one?).

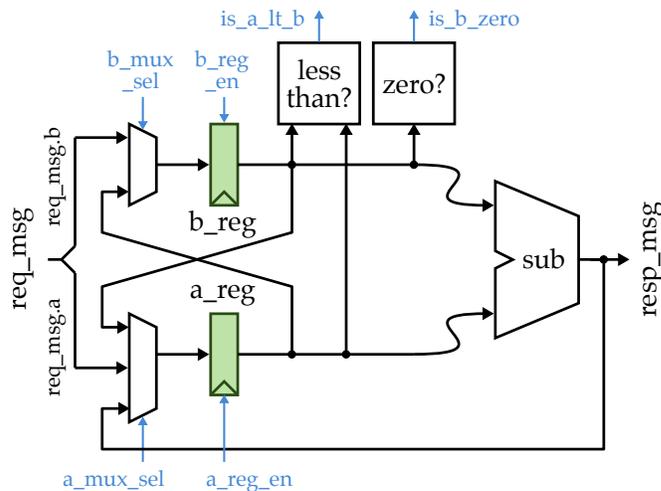


Figure 32: Datapath Diagram for GCD – Datapath includes two state registers and required muxing and arithmetic units to iteratively implement Euclid’s algorithm.

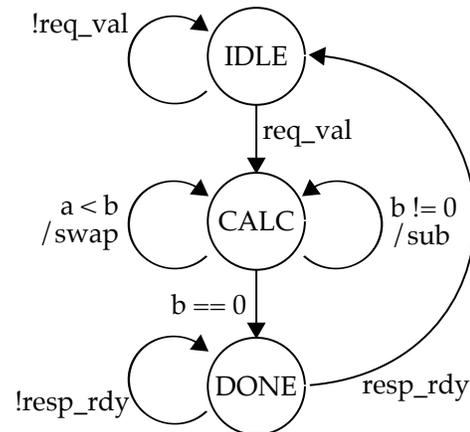


Figure 33: FSM Diagram for GCD – A hybrid Moore/Mealy FSM for controlling the datapath in Figure 32. Mealy transitions in the calc state determine whether to swap or subtract.

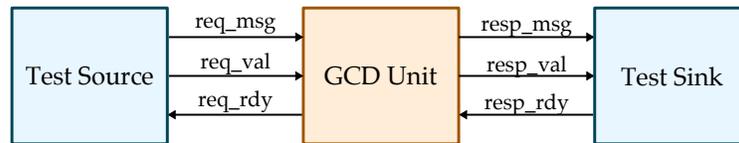


Figure 34: Verifying GCD Using Test Sources and Sinks – Parameterized test sources send a stream of messages over a val/rdy interface, and parameterized test sinks receive a stream of messages over a val/rdy interface and compare each message to a previously specified reference message. Clock and reset signals are not shown.

Figure 37 shows a portion of the top-level module that connects the datapath and control unit together. Lines X and Y use the new implicit connection operator (`.*`) provided by SystemVerilog. Using the implicit connection operator during module instantiation means to connect every port to a signal with the same name. So these two lines take care of connecting all of the control and status signals. This is a powerful way to write more compact code which avoids connectivity bugs, especially when connecting the datapath and control unit.

We can use the exact same source/sink testing that we used in the PyMTL tutorial to test our Verilog implementation of the GCD unit. As a reminder, Figure 34 illustrates the overall connectivity in the test harness. The test source includes the ability to randomly delay messages going into the DUT and the test sink includes the ability to randomly apply back-pressure to the DUT. By using various combinations of these random delays we can more robustly ensure that our flow-control logic is working correctly. We can reuse the same test harnesses from the PyMTL tutorial as follows:

```
% cd ${TUTROOT}/build
% py.test ../tut4_verilog/gcd/test/GcdUnitRTL_test.py -v
% py.test ../tut4_verilog/gcd/test/GcdUnitRTL_test.py -sv -k basic_0x0
```

```

1  module tut4_verilog_gcd_GcdUnitDpath
2  (
3      input logic      clk,
4      input logic      reset,
5
6      // Data signals
7
8      input logic [31:0] req_msg,
9      output logic [15:0] resp_msg,
10
11     // Control signals
12
13     input logic      a_reg_en, // Enable for A register
14     input logic      b_reg_en, // Enable for B register
15     input logic [1:0] a_mux_sel, // Sel for mux in front of A reg
16     input logic      b_mux_sel, // sel for mux in front of B reg
17
18     // Status signals
19
20     output logic      is_b_zero, // Output of zero comparator
21     output logic      is_a_lt_b // Output of less-than comparator
22 );
23
24     localparam c_nbits = 16;
25
26     // Split out the a and b operands
27
28     logic [c_nbits-1:0] req_msg_a = req_msg[31:16];
29     logic [c_nbits-1:0] req_msg_b = req_msg[15:0 ];
30
31     // A Mux
32
33     logic [c_nbits-1:0] b_reg_out;
34     logic [c_nbits-1:0] sub_out;
35     logic [c_nbits-1:0] a_mux_out;
36
37     vc_Mux3#(c_nbits) a_mux
38     (
39         .sel    (a_mux_sel),
40         .in0    (req_msg_a),
41         .in1    (b_reg_out),
42         .in2    (sub_out),
43         .out    (a_mux_out)
44     );
45
46     // A register
47
48     logic [c_nbits-1:0] a_reg_out;
49
50     vc_EnReg#(c_nbits) a_reg
51     (
52         .clk    (clk),
53         .reset  (reset),
54         .en     (a_reg_en),
55         .d      (a_mux_out),
56         .q      (a_reg_out)
57     );

```

Figure 35: Portion of GCD Datapath Unit – We use struct types to encapsulate both control and status signals and we use a preprocessor macro from the GCD message struct to determine how to size the datapath components.

```

1  //-----
2  // State Outputs
3  //-----
4
5  localparam a_x   = 2'dx;
6  localparam a_ld  = 2'd0;
7  localparam a_b   = 2'd1;
8  localparam a_sub = 2'd2;
9
10 localparam b_x   = 1'dx;
11 localparam b_ld  = 1'd0;
12 localparam b_a   = 1'd1;
13
14 task cs
15 (
16     input logic    cs_req_rdy,
17     input logic    cs_resp_val,
18     input logic [1:0] cs_a_mux_sel,
19     input logic    cs_a_reg_en,
20     input logic    cs_b_mux_sel,
21     input logic    cs_b_reg_en
22 );
23 begin
24     req_rdy   = cs_req_rdy;
25     resp_val  = cs_resp_val;
26     a_reg_en  = cs_a_reg_en;
27     b_reg_en  = cs_b_reg_en;
28     a_mux_sel = cs_a_mux_sel;
29     b_mux_sel = cs_b_mux_sel;
30 end
31 endtask
32
33 // Labels for Mealy transitions
34
35 logic do_swap;
36 logic do_sub;
37
38 assign do_swap = is_a_lt_b;
39 assign do_sub  = !is_b_zero;
40
41 // Set outputs using a control signal "table"
42
43 always_comb begin
44
45     set_cs( 0, 0, a_x, 0, b_x, 0 );
46     case ( state_reg )
47         //
48         //          req resp a mux  a  b mux b
49         //          rdy val  sel   en sel  en
50         STATE_IDLE: cs( 1, 0,  a_ld, 1, b_ld, 1 );
51         STATE_CALC: if ( do_swap ) cs( 0, 0,  a_b, 1, b_a, 1 );
52                     else if ( do_sub ) cs( 0, 0,  a_sub, 1, b_x, 0 );
53         STATE_DONE: cs( 0, 1,  a_x, 0, b_x, 0 );
54         default    cs('x, 'x,  a_x, 'x, b_x, 'x );
55     endcase
56
57 end

```

Figure 36: Portion of GCD FSM-based Control Unit for State Outputs – We can use a task to create a “control signal table” with one row per state or Mealy transition and one column per control signal. Local parameters can help compactly encode various control signal values.

```

1  module tut4_verilog_gcd_GcdUnitRTL
2  (
3      input  logic          clk,
4      input  logic          reset,
5
6      input  logic          req_val,
7      output logic          req_rdy,
8      input  logic [31:0]   req_msg,
9
10     output logic          resp_val,
11     input  logic          resp_rdy,
12     output logic [15:0]   resp_msg
13 );
14
15     //-----
16     // Connect Control Unit and Datapath
17     //-----
18
19     // Control signals
20
21     logic          a_reg_en;
22     logic          b_reg_en;
23     logic [1:0]    a_mux_sel;
24     logic          b_mux_sel;
25
26     // Data signals
27
28     logic          is_b_zero;
29     logic          is_a_lt_b;
30
31     // Control unit
32
33     tut4_verilog_gcd_GcdUnitCtrl ctrl
34     (
35         .*
36     );
37
38     // Datapath
39
40     tut4_verilog_gcd_GcdUnitDpath dpath
41     (
42         .*
43     );
44
45 endmodule

```

Figure 37: Portion of GCD Top-Level Module – We use the new implicit connection operator (`.*`) to automatically connect all of the control and status signals to both the control unit and datapath.

Figure 38 illustrates a portion of the line trace for the randomized testing. We use the line trace to show the state of the A and B registers at the beginning of each cycle and use specific characters to indicate which state we are in (i.e., I = idle, Cs = calc with swap, C- = calc with subtract, D = done). We can see that the test source sends a new message into the GCD unit on cycle 296. The GCD unit is in the idle state and transitions into the calc state. It does five subtractions and a final swap before transitioning into the done state on cycle 304. The result is valid but the test sink is not ready, so the GCD unit waits in the done state until cycle 310 when it is able to send the result to the test sink. On cycle 311 the GCD unit accepts a new input message to work on. This is a great example of how an effective line trace can enable you to quickly visualize how a design is actually working.

```

1 cycle src      A    B    Areg Breg ST out    sink
2
3 296: 002d00e1 > 00e1:002d(0002 0000 I )    >
4 297: #        > #        (00e1 002d C-)    >
5 298: #        > #        (00b4 002d C-)    >
6 299: #        > #        (0087 002d C-)    >
7 300: #        > #        (005a 002d C-)    >
8 301: #        > #        (002d 002d C-)    > .
9 302: #        > #        (0000 002d Cs)    >
10 303: #        > #        (002d 0000 C )    >
11 304: #        > #        (002d 0000 D )#    > #
12 305: #        > #        (002d 0000 D )#    > #
13 306: #        > #        (002d 0000 D )#    > #
14 307: #        > #        (002d 0000 D )#    > #
15 308: #        > #        (002d 0000 D )#    > #
16 309: #        > #        (002d 0000 D )#    > #
17 310: #        > #        (002d 0000 D )002d > 002d
18 311: 002200cc > 00cc:0022(002d 0000 I )    >
19 312: #        > #        (00cc 0022 C-)    >
20 313: #        > #        (00aa 0022 C-)    >
21 314: #        > #        (0088 0022 C-)    >

```

Figure 38: Line Trace for RTL Implementation of GCD – State of A and B registers at the beginning of the cycle is shown, along with the current state of the FSM. I = idle, Cs = calc with swap, C- = calc with subtract, D = done.

- ★ *To-Do On Your Own:* Optimize the GCD implementation to improve the performance on the given input datasets.

A first optimization would be to transition into the done state if either *a* or *b* are zero. If *a* is zero and *b* is greater than zero, we will swap *a* and *b* and then end the calculation on the next cycle anyways. You will need to carefully modify the datapath and control so that the response can come from either the *a* or *b* register.

A second optimization would be to avoid the bubbles caused by the IDLE and DONE states. First, add an edge from the CALC state directly back to the IDLE state when the calculation is complete and the response interface is ready. You will need to carefully manage the response valid bit. Second, add an edge from the CALC state back to the CALC state when the calculation is complete, the response interface is ready, and the request interface is valid. These optimizations should eliminate any bubbles and improve the performance of back-to-back GCD transactions.

A third optimization would be to perform a swap and subtraction in the same cycle. This will require modifying both the datapath and the control unit, but should have a significant impact on the overall performance.

6.2. Evaluating GCD Unit using a Simulator

As with the previous section, we have provided a simulator for evaluating the performance of the GCD implementation. In this case, we are focusing on a single implementation with two different input datasets. You can run the simulators and look at the average number of cycles to compute a GCD for each input dataset like this:

```

% cd ${TUTROOT}/build
% ../tut4_verilog/gcd/gcd-sim --stats --impl cl --input random
% ../tut4_verilog/gcd/gcd-sim --stats --impl rtl --input random

```

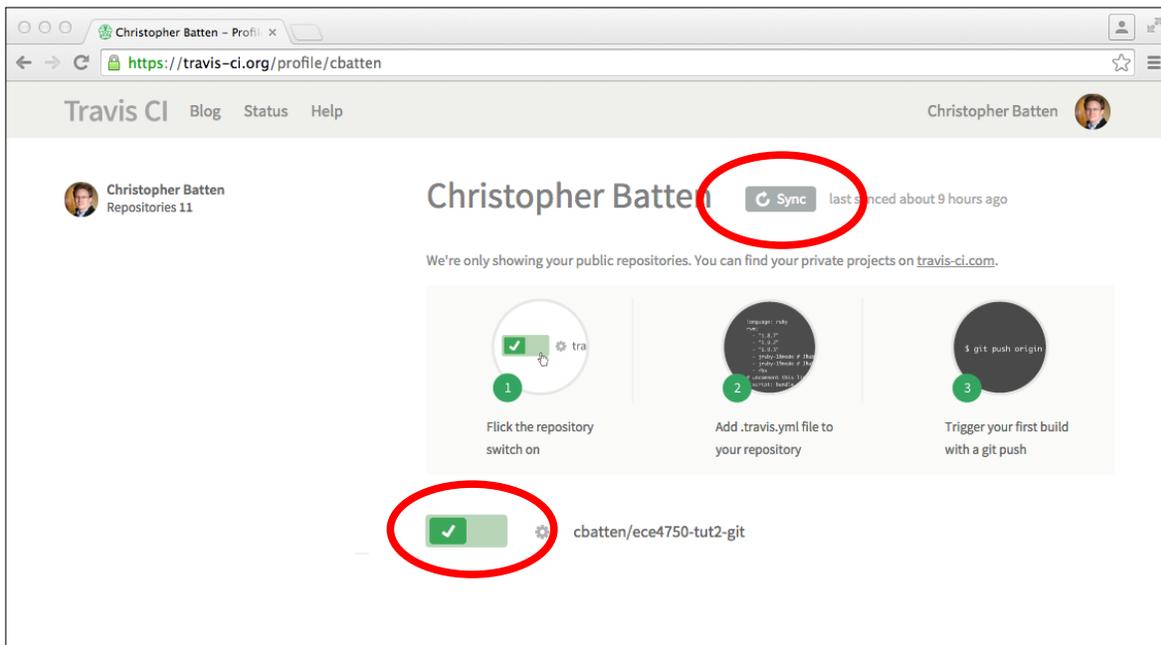


Figure 39: TravisCI Settings Page

7. TravisCI for Continuous Integration

As discussed in the Git tutorial, TravisCI is an online continuous integration service that is tightly coupled to GitHub. TravisCI will automatically run all tests for a students' lab assignment every time the students push their code to GitHub. We will be using the results reported by TravisCI to evaluate the code functionality of the lab assignments. In this section, we do a small experiment to illustrate how TravisCI works for PyMTL projects.

The first step is to enable TravisCI for the remote repository in GitHub. Log into TravisCI using your GitHub ID and password:

- `https://travis-ci.org/profile`

Once you have signed in, you should go to your TravisCI profile and find the list of your public GitHub repositories. You may need to click *Sync* to ensure that TravisCI has the most recent view of your public repositories on GitHub. Turn on TravisCI with the little “switch” next to the repository we have been using in this tutorial (`<githubid>/ece4750-tut4-verilog`). Figure 39 shows what the TravisCI settings page should look like and the corresponding “switch”. After enabling TravisCI for the `<githubid>/ece4750-tut4-verilog` repository, you should be able to go to the TravisCI page for this repository:

- `https://travis-ci.org/<githubid>/ece4750-tut4-verilog`

TravisCI will report that there are no builds for this repository yet. Go ahead and commit all of the work you have done in this tutorial, then push your local commits to the remote repository on GitHub. If you revisit the TravisCI page for this repository, you should see TravisCI starting to build and run all of your tests. Figure ?? shows what the TravisCI build log will look like for a brand new fork of the tutorial repository. Study the TravisCI log output to verify that TravisCI is: (1) installing

```

1 Using worker: worker-linux-docker-b5b6c65c.prod.travis-ci.org:travis-linux-3
2
3 Build system information system_info
65
66 $ git clone --depth=50 --branch=master git://github.com/cbatten/ece4750-tut4-verilog.git git.checkout 0.07s
67
68 This job is running on container-based infrastructure, which does not allow use of 'sudo', setuid and setgid
69 executables.
70 If you require sudo, add 'sudo: required' to your .travis.yml
71 See http://docs.travis-ci.com/user/workers/container-based-infrastructure/ for details.
72
73 $ source ~/.virtualenv/python2.7/bin/activate 0.00s
74 $ python --version
75 Python 2.7.9
76 $ pip --version
77 pip 6.0.7 from /home/travis/virtualenv/python2.7.9/lib/python2.7/site-packages (python 2.7)
78
79 $ wget --quiet http://brg.csl.cornell.edu/artifacts/verilator-travis-3.876.tar.gz install.1 8.03s
80 $ tar xzf verilator-travis-3.876.tar.gz install.2 1.62s
81 $ export VERILATOR_ROOT=${PWD}/verilator-3.876 install.3 0.00s
82 $ export PATH=${VERILATOR_ROOT}/bin:$PATH install.4 0.00s
83 $ export PYMTL_VERILATOR_INCLUDE_DIR=${VERILATOR_ROOT}/include install.5 0.00s
84 $ verilator --version install.6 0.10s
85 $ pip -q install git+https://github.com/cornell-brg/pymtl.git#ece4750 install.7 8.65s
86 $ mkdir -p sim/build before_script.1 0.01s
87 $ cd sim/build before_script.2 0.00s
88 $ py.test .. 57.75s
89
90 ===== test session starts =====
91 platform linux2 -- Python 2.7.9 -- py-1.4.26 -- pytest-2.6.4
92 plugins: xdist, xdist, xdist
93 collected 96 items
94
95 ../tut4_verilog/gcd/GcdUnitCL_test.py .....
96 ../tut4_verilog/gcd/GcdUnitFL_test.py .....
97 ../tut4_verilog/gcd/GcdUnitMsg_test.py ...
98 ../tut4_verilog/gcd/GcdUnitRTL_test.py .....
99 ../tut4_verilog/gcd/gcd_sim_test.py .....
100 ../tut4_verilog/regincr/RegIncr2stage_test.py FFFF
101 ../tut4_verilog/regincr/RegIncrNstage_test.py FFFFFFFFFFFFFFFF
102 ../tut4_verilog/regincr/RegIncr_extra_test.py FFFF
103 ../tut4_verilog/regincr/RegIncr_test.py F
104 ../tut4_verilog/sort/SortUnitCL_test.py .....
105 ../tut4_verilog/sort/SortUnitFL_test.py ....
106 ../tut4_verilog/sort/SortUnitFlatRTL_test.py .....
107 ../tut4_verilog/sort/SortUnitStructRTL_test.py FFFFF
108 ../tut4_verilog/sort/sort_sim_test.py .....
109
110 ===== 28 failed, 68 passed in 57.45 seconds =====
111
112 The command "py.test .." exited with 1.
113
114 Done. Your build exited with 1.

```

Figure 40: TravisCI Log

Verilator; (2) installing PyMTL; (3) creating a build directory; and (4) running all of your unit tests. Confirm that if all of the tests pass on ecelinux then they also pass on TravisCI.

Acknowledgments

This tutorial was developed for ECE 4750 Computer Architecture course at Cornell University by Christopher Batten.

Appendix A: Constructs Allowed in Synthesizable RTL

| Always Allowed in Synthesizable RTL | Allowed in Synthesizable RTL With Limitations | Explicitly Not Allowed in Synthesizable RTL |
|-------------------------------------|---|---|
| logic | always ¹ | wire, reg ¹³ |
| logic [N-1:0] | enum ² | integer, real, time, realtime |
| & ^ ~ ~ (bitwise) | struct ³ | signed ¹⁴ |
| && ! | casez, endcase ⁴ | ==, != |
| & ~& ~ ^ ^~ (reduction) | task, endtask ⁵ | * / % ** ¹⁵ |
| + - | function, endfunction ⁵ | #N (delay statements) |
| >> << >>> | = (blocking assignment) ⁶ | inout ¹⁶ |
| == != > <= < <= | <= (non-blocking assignment) ⁷ | initial |
| {} | typedef ⁸ | variable initialization ¹⁷ |
| {N{}} (repeat) | packed ⁹ | negedge ¹⁸ |
| ?: | \$clog2() ¹⁰ | casex, endcase |
| always_ff, always_comb | \$bits() ¹⁰ | for, while, repeat, forever ¹⁹ |
| if else | \$signed() ¹¹ | fork, join |
| case, endcase | read-modify-write signal ¹² | deassign, force, release |
| begin, end | | specify, endspecify |
| module, endmodule | | nmos, pmos, cmos |
| input, output | | rnmos, rpmos, rcmos |
| assign | | tran, tranif0, tranif1 |
| parameter | | rtran, rtranif0, rtranif1 |
| localparam | | supply0, supply1 |
| genvar | | strong0, strong1 |
| generate, endgenerate | | weak0, weak1 |
| generate for | | primitive, endprimitive |
| generate if else | | defparam |
| generate case | | unnamed port connections ²⁰ |
| named port connections | | unnamed parameter passing ²¹ |
| named parameter passing | | all other keywords |
| | | all other system tasks |

1 Students should prefer using `always_ff` and `always_comb` instead of `always`. If students insist on using `always`, then it can only be used in one of the following two constructs: `always @(posedge clk)` for sequential logic, and `always @(*)` for combinational logic. Students are not allowed to trigger sequential blocks off of the negative edge of the clock or create asynchronous resets, nor use explicit sensitivity lists.

2 `enum` can only be used with an explicit base type of `logic` and explicitly setting the bitwidth using the following syntax: `typedef enum logic [$clog2(N)-1:0] { ... } type_t;` where `N` is the number of labels in the `enum`. Anonymous enums are not allowed.

3 `struct` can only be used with the `packed` qualifier (i.e., `unpacked` structs are not allowed) using the following syntax: `typedef struct packed { ... } type_t;` Anonymous structs are not allowed.

4 `casez` can only be used in very specific situations to compactly implement priority encoder style hardware structures.

5 `task` and `function` blocks must themselves contain only synthesizable RTL.

6 Blocking assignments should only be used in `always_comb` blocks and are explicitly not allowed in `always_ff` blocks.

- 7 Non-blocking assignments should only be used in `always_ff` blocks and are explicitly not allowed in `always_comb` blocks.
- 8 `typedef` should only be used in conjunction with `enum` and `struct`.
- 9 `packed` should only be used in conjunction with `struct`.
- 10 The input to `$clog2/$bits` must be a static-elaboration-time constant. The input to `$clog2/$bits` cannot be a signal (i.e., a wire or a port). In other words, `$clog2/$bits` can only be used for static elaboration and cannot be used to model actual hardware.
- 11 `$signed()` can only be used around the operands to `>>>`, `>`, `>=`, `<`, `<=` to ensure that these operators perform the signed equivalents.
- 12 Reading a signal, performing some arithmetic on the corresponding value, and then writing this value back to the same signal (i.e., read-modify-write) is not allowed within an `always_comb` concurrent block. This is a combinational loop and does not model valid hardware. Read-modify-write is allowed in an `always_ff` concurrent block with a non-blocking assignment, although we urge students to consider separating the sequential and combinational logic. Students can use an `always_comb` concurrent block to read the signal, perform some arithmetic on the corresponding value, and then write a temporary wire; and use an `always_ff` concurrent block to flop the temporary wire into the destination signal.
- 13 `wire` and `reg` are perfectly valid, synthesizable constructs, but `logic` is much cleaner. So we would like students to avoid using `wire` and `reg`.
- 14 `signed` types can sometimes be synthesized, but we do not allow this construct in the course.
- 15 Multipliers can be synthesized and small multipliers can even be synthesized efficiently, but we do not allow this construct in the course. If you need to multiply or divide by a power of two, then you can use the left and right shift operators.
- 16 Ports with `inout` can be used to create tri-state buses, but tools often have trouble synthesizing hardware from these kinds of models.
- 17 Variable initialization means assigning an initial value to a `logic` variable when you declare the variable. This is not synthesizable; it is not modeling real hardware. If you need to set some state to an initial condition, you must explicitly use the `reset` signal.
- 18 Triggering a sequential block off of the `negedge` of a signal is certainly synthesizable, but we will be exclusively using a positive-edge-triggered flip-flop-based design style.
- 19 If you would like to generate hardware using loops, then you should use `generate` blocks.
- 20 In very specific, rare cases unnamed port connections might make sense, usually when there are just one or two ports and there purpose is obvious from the context.
- 21 In very specific, rare cases unnamed parameter passing might make sense, usually when there are just one or two parameters and their purpose is obvious from the context.