

# 10

## Introduction to Logisim

### 10.1 Objectives

- Get familiar with Logisim
- Use Logisim to model and simulate the behavior of digital logic circuits

### 10.2 Logisim Environment Layout

Logisim is an educational and user friendly tool. It mainly consists of an interactive graphical schematic editor and logic simulator. Logisim has a layout similar to most available software tools ( **Figure 1**).

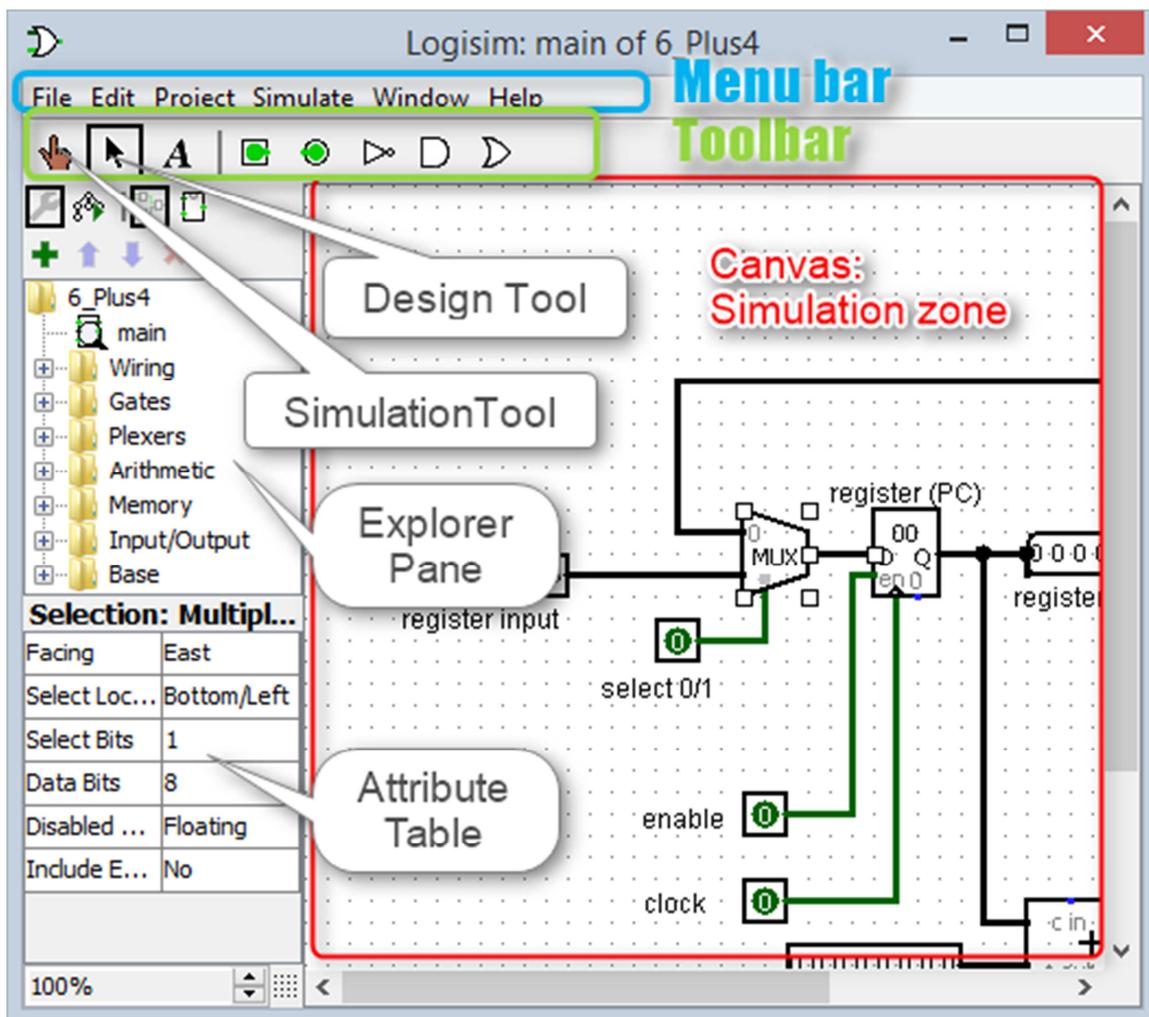


Figure 10.1: Logisim Main Window Layout

The main window consists of the following items:

- **Toolbar:** contains short cuts to several commonly used items
  - The simulation tool: shaped like a hand, is used in simulation mode to alter input pins.
  - The design tool: is used while designing the circuit.
  - The input pin: green circle surrounded by a square box, is used to send a signal through a wire. When placing the input on the canvas it initializes the input to logic 1 or 0. The number of bits can be increased in the Attribute Table.
  - The output pin: green circle surrounded by a circular shape, is used to observe the output from a gate or a block. The output pin toggles in real time as long as the simulation is enabled from the menu bar: **Simulate > Simulation Enabled**
- **Explorer Pane:** This pane contains the list of wiring, gates, multiplexers and other components that are available for digital design in Logisim.
- **Attribute Table:** Gives detailed attributes of digital design components (e.g., AND, OR, XOR gates). The attribute table allows you to alter the number of inputs/outputs that a digital component may have.
- **Canvas:** The canvas is the area for you to create your digital circuits. In this area you may simulate your circuits while designing in real time.

## 10.3 Circuit Design

Logisim operates in two modes: Design and Simulate. Logisim is first operated in edit mode while designing a logic circuit, then in simulate mode to see how the circuit would actually work.

### 10.3.1 Edit Mode

- To use the edit mode, select the **Design** tool (Figure 10.1).
- Select then a component in the library on the left. To add it in your design, simply drag and drop the desired component in the canvas:
  - To insert a logic gate, expand the folder Gates and click on the desired logic gate. Once the gate is selected, the mouse will turn into the shape of the selected gate. Place the gate in the canvas or circuit area on the right.
  - Each selected component has modifiable attributes in the attribute area. For example for an AND gate, the number of input signals, the number of bits per input or output and the size of the component can all be modified.
  - It is also possible to copy and paste of one or more components in the canvas. In this case, the components retain all the attributes previously defined.
- To connect the gates, select the arrow icon on top. Then drag from the output of one gate to the input of another gate. You can connect to any of the small blue dots on the gate symbol.
- To create an input to the circuit, select the “**Add Pin**” icon with an outline of a *square*.
- Similarly, add an output to the circuit by using the “**Add Pin**” icon with an outline of a *circle*.
- To assign a name to the input/output pin, click on the pin while the arrow icon is selected.
- You may then add the label for the pins.
- While components are added to the circuit, notice how the color of the wire changes (Table 10.1).

**Table 10.1:** Logisim Wire Color Conventions

Wire Color	Meaning
dark green	logical `0'
light green	logical `1'
blue	unknown value
gray	unconnected wire
<b>black</b>	wire has more than one bit (bus)
red	conflicting values, error
orange	incompatible bus width

### 10.3.2 Full Adder Design

In Logisim, a digital circuit may be designed using either one of two approaches:

- Drawing schematic: the user has to do the design by choosing the necessary components as per his design, based on the digital schematic.
- Analyzing the circuit, by providing the truth tables with necessary inputs and outputs and let Logisim do the design.

#### Schematic Design:

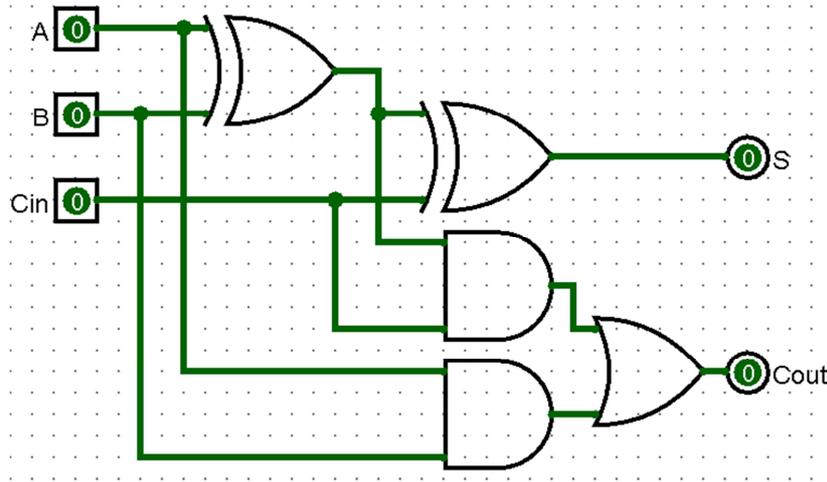
- a. Set up the truth table for a simple Full Adder. Let A B and Cin be the inputs, S and Co the outputs.

A	B	Ci	S	Co
0	0	0	0	0
0	0	1	1	0
0	1	0	1	0
0	1	1	0	1
1	0	0	1	0
1	0	1	0	1
1	1	0	0	1
1	1	1	1	1

- b. Derive the logic equations for S and Cout:

$$S = A \oplus B \oplus C_i \quad C_o = (A \wedge B) + (C_i \wedge (A \oplus B))$$

- c. Finally, design the circuit based on the above equations (Figure 10.2)



**Figure 10.2:** Full Adder Circuit Design

### **Circuit Analysis:**

From the Menu Bar select "**Window**" then "**Combinational Analysis**". You may also go to **Project** then, select "**Analyze Circuit**".

- a. Create three inputs named Cin, A, and B.
- b. Create two outputs named Cout and S.
- c. Complete the truth table for a full adder.
- d. Look at the "**Expression**" and "**Minimized**" tabs.
- e. Press the "**Build Circuit**" button and enter FA0 as the name of the new circuit.

Once your design is complete, you may proceed by testing its functionality:

- a. Use the Simulation Tool in the Toolbar.
- b. Click on an input to see its state toggle between the values 0 and 1
- c. While doing so, check the outputs that they correspond exactly to your design.

### 10.3.3 Test Mode

Select the Simulation Tool (Figure: 10.1) so you can test the circuit. You can toggle the value of an input pin by clicking on it. Note that the values of all subsequent wires change instantly.

- a. Change the values of the Ci, A, and B inputs
- b. Observe the Co and S outputs to verify the correct operation of the circuit

### **Probes:**

As you test your circuit you may also add probes to check the values on wires or buses. These values may be displayed in different formats.

- a. From the explorer pane, select Wiring then Probe.
- b. Set a direction for your probe,
- c. Then select the numbering system that you would like to use to display your data.

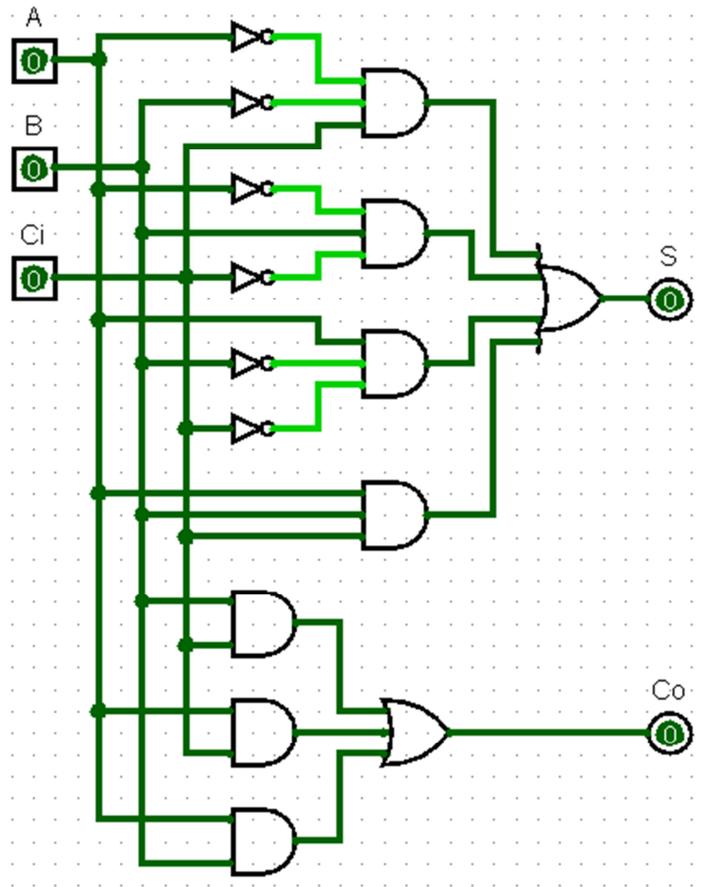


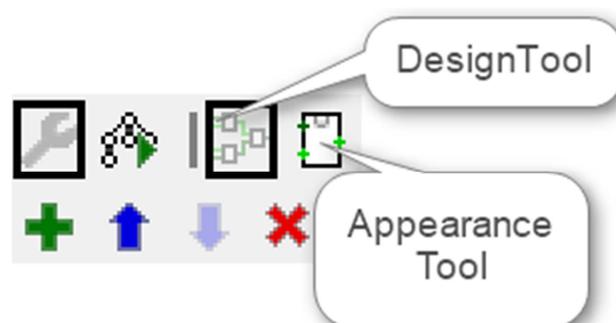
Figure 10.3: Full Adder Circuit Designed Through Analysis

## 10.4 Hierarchical Design

A complex circuit is usually broken down into smaller sub-circuits. These sub-circuits are designed independently, tested, arranged into libraries, and then used to build the big circuit.

### 10.4.1 Circuit Appearance

Later on, you may use your circuit as a **sub-circuit** in another design. For that purpose, you may want to alter the locations of the inputs and outputs or the appearance of the whole circuit as a module. To do so, while your design is on the editor, click the Appearance Tool under the toolbar (Figure 10.4). This will show your circuit as a block with the inputs and outputs; once you click any of the input/output pins, on the bottom right corner your circuit will appear showing the location of the pin you selected.



## Figure 10.4: Design and Appearance Tools

You may then modify the following properties of your circuit appearance: pin locations, general appearance, and color.

### 10.4.2 Using Sub-Circuits

One of the good features of Logisim is that you can use a circuit you have already built as a building block in another, more complex circuit.

You can create a new sub-circuit with (Project > Add Circuit)

- Save all your sub-circuits in one folder that you will make your own library.
- On the menu bar select load Project > Load Library > Logism-Library then select your folder and the module you would like to add.
- Your circuit will appear as a new element in the explorer pane. Select your module then drag and drop.
- You may then add pins for input and output, or connect your module to your circuit using wires.

Within the newly created full adder FA0 circuit designed above (Figures 10.2 or 10.3), change the orientation of the Ci input so that it is facing south. Change the orientation of the Co output so that it is facing north (Figure 10.5). Your circuit is ready to be used as a module or sub-circuit in a more complex digital system design.

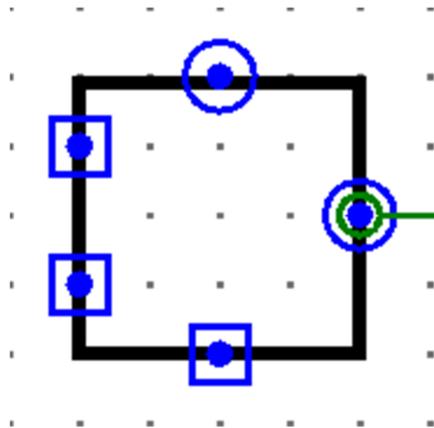


Figure 10.5: Full Adder Block

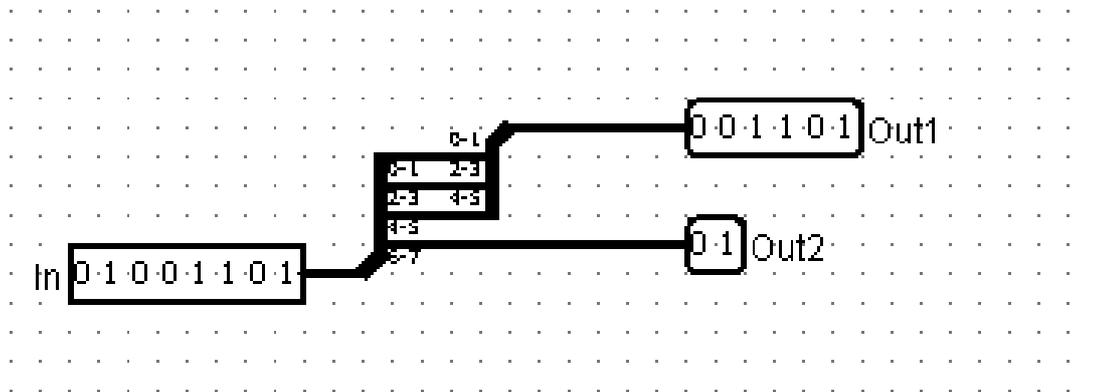
### 10.4.3 Splitters and Tunnels:

These are components mainly used to simplify wiring. Splitters are used to group or separate bits in a bus, and tunnels are used to avoid lengthy connections between components in the design.

#### Splitters:

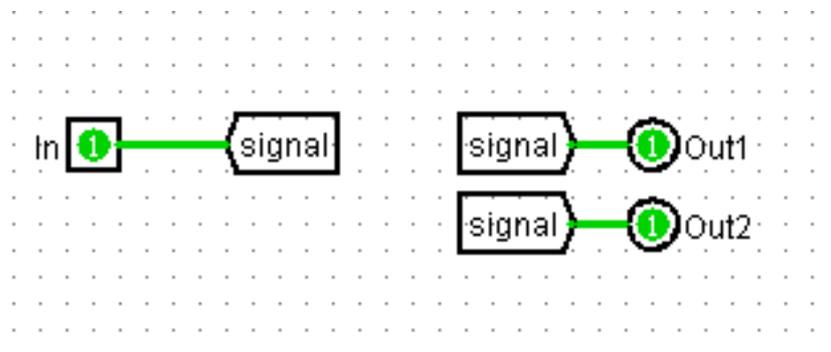
Wire splitters split a multi-bit wire into smaller wires (an 8-bit wire into 2 x 4-bit wires, for example). They can also combine multiple wires into a single wire with a wider bit width.

- To use a splitter from the menu select (Wiring > Splitter)
- Set the fan out and the bit width in properties.
  - "Fan Out" controls how many teeth the wire splitter has.
  - "Bit Width In" controls the actual size of the data the splitter is splitting.
- If you have more data bits than outputs, some of the fan outs would carry more than one data bit.



### Tunnels:

Tunnels are you to transmit signals without using wires.



- From the menu, select (**Wiring > Tunnel**)
- Tunnels are mainly used to avoid excessive wiring, but can be used to connect control and clock signals.

## 10.5 In-Lab Design

### 10.5.1 A 32-Bit Adder Design:

- Using the full adder already designed as a block, we intend to design an 8-bit ripple carry adder.
  - a. Add three splitters to the circuit. Each splitter should have an input width of 8 bits and a fan out of 8.

- b. Attach two east-facing splitters to the 8-bit inputs A and B.
  - c. Attach a west-facing splitter to the 8-bit output S.
  - d. Create eight instances of the FA circuit.
  - e. Connect the S outputs of the eight FA instances to the splitter for the 8-bit S output.
  - f. Connect the carry inputs and outputs of the eight FA instances so that carries will propagate appropriately from the  $C_i$  input, through the 1-bit adders, to the  $C_o$  output.
  - g. Connect the A inputs of the eight FA instances to the splitter for the A input.
  - h. Connect the B inputs of the eight FA instances to the splitter for the B input.
  - i. Test your circuit
- Finally save your design as 1 module or sub-circuit
  - Using the above adder as a block, design a 32 bit adder then test it.

### 10.5.2 A 2-Bit Multiplier Design:

Using the Combinational Analysis option in Logisim, design a 2 bit multiplier. First you need to set up the truth table of your multiplier by choosing appropriate names for your inputs and outputs. Derive the logical equations for all your outputs. Use the Design option to let Logisim design the multiplier. Test your circuit and make it as one complete module. Compare your design to the multiplier provided in Logisim.

### 10.5.3 Bonus Question:

Make use of your multiplier to design a 4 bit multiplier, test it on compare to the built-in multiplier provided in Logisim.