CS2100 Computer Organisation

Lab #9: Using Logisim I

(Week 12: 1 – 5 November 2021)

[This document is available on LumiNUS and module website http://www.comp.nus.edu.sg/~cs2100]

Remember to bring this along to your lab!

Name: _____ Student No: _____

Lab Group: _____

Objective:

You will learn to use **logisim** to analyse a simple circuit and create a 4-bit parallel adder.

Preparation (before the lab):

1. Download **logisim** from the following website:

   http://www.cburch.com/logisim/download.html

Logisim

a graphical tool for designing and simulating logic circuits

Download
Documentation
Release History
Q & A
Comments
Links

[de] Deutsch
[el] Ελληνικά
[en] English
[pt] Português
[ru] Русский

SOURCEFORGE

Getting Logisim

Logisim should run on any platform supporting Java, version 5 or later.

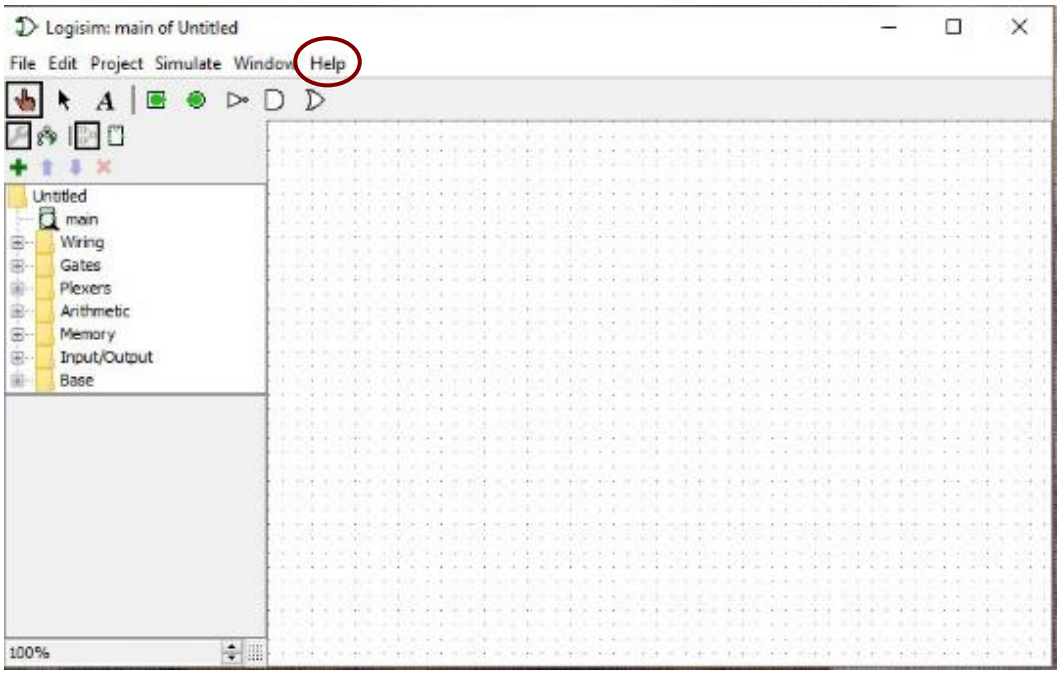
1. Logisim requires Java 5 or later. If you do not already have it on your computer, Java is available from `java.sun.com`.
2. Download Logisim from Logisim's SourceForge.net page. You will three choices of which release to download.
   - A `.jar` file - runs on any platform, though not necessarily conveniently.
   - A MacOS `.tar.gz` file
   - A Windows `.exe` file

   If you use MacOS or Windows, I would recommend using the release specific to your platform.
3. To execute the program:
   - *With the generic `.jar` file:* On Windows and MacOS systems, you will likely be able to start Logisim by double-clicking the JAR file. If that doesn't work, or if you use Linux or Solaris, you can type "`java -jar logisim-xx.jar`" at the command line.
   - *With the MacOS X version:* Once the downloaded `.tar.gz` version is uncompressed (this will likely happen automatically), just double-click the Logisim icon to start. You may want to place the icon into the Applications folder.
   - *With the Windows version:* Just double-click the Logisim icon. You may want to create a shortcut on the desktop and/or in the Start menu to make starting Logisim easier.

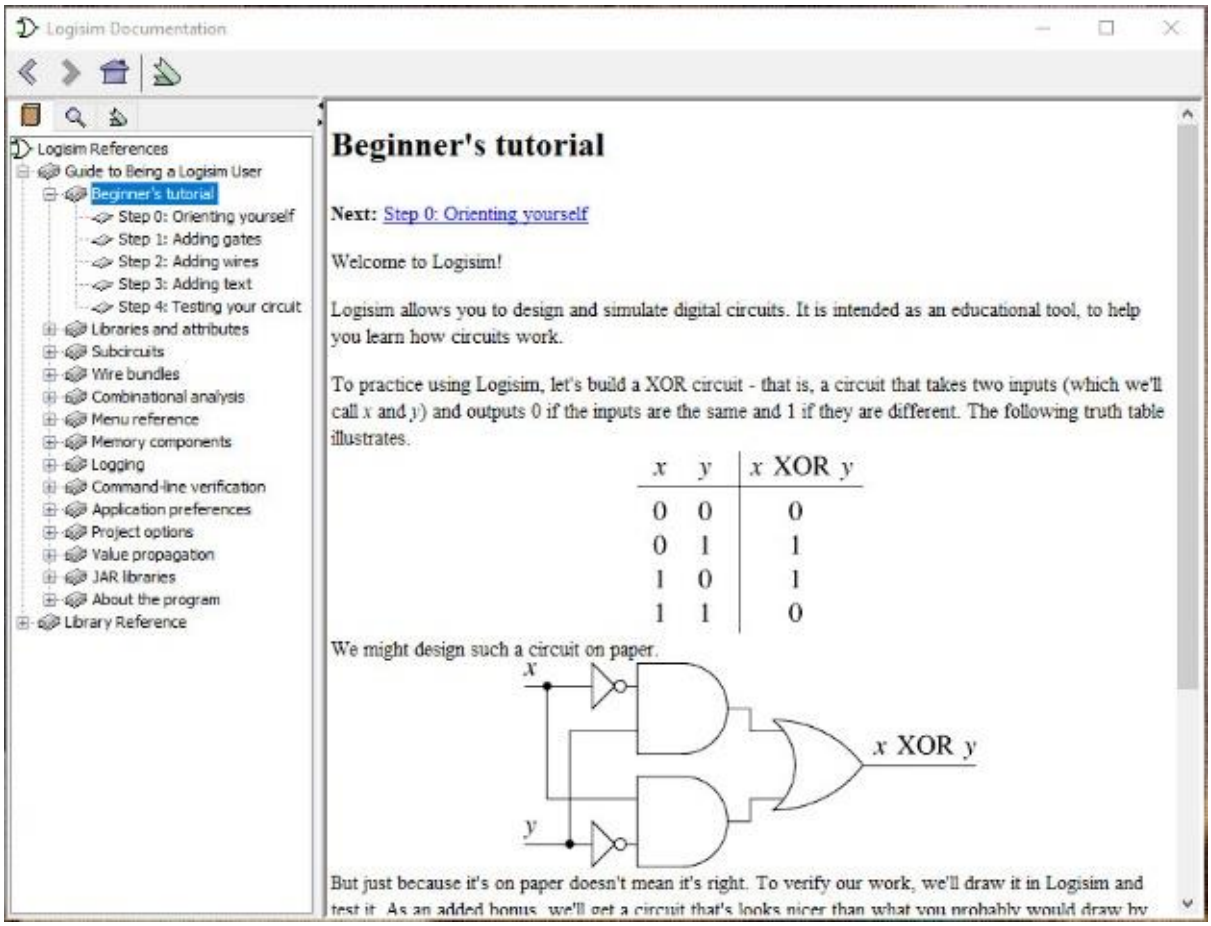
If you find Logisim useful, please send me a comment!

Copyright and authorship

2. Run **logisim** and you will see this screen:



3. Click on "Help" → "Tutorial" and read "Beginner's tutorial". Familiarise yourself with the basic working of the software. Go through the 5 steps in the tutorial and create some simple circuits yourself.



Procedure:

1. Download the file **lab9.circ** from LumiNUS Files or the CS2100 website.

2. Open **lab9.circ** in Logisim. Select the "Poke" tool  and then click on the inputs *X*, *Y* and *Z* to toggle their values, and observe the changes in the outputs.

3. What is the name of the circuit?

   Answer: _____

4. The circuit has two outputs *S* and *C*, but they are not labelled. Add the labels correctly. Show your labTA.

5. Click "Project" → "Analyze Circuit". Click on "Table", and fill in the table below with what you have observed. (If you find that the outputs do not appear in the same column-order as in the table below, you can change the order by clicking on "Outputs".)

X	***Y***	***Z***	***C***	***S***
0	0	0		
0	0	1		
0	1	0		
0	1	1		
1	0	0		
1	0	1		
1	1	0		
1	1	1		

6. Still at "Project" → "Analyze Circuit". Click on "Minimized". Below the K-map of an output you should see the simplified SOP expressions for that output. Write down the simplified SOP expressions for the two outputs *S* and *C*.

   S = _____

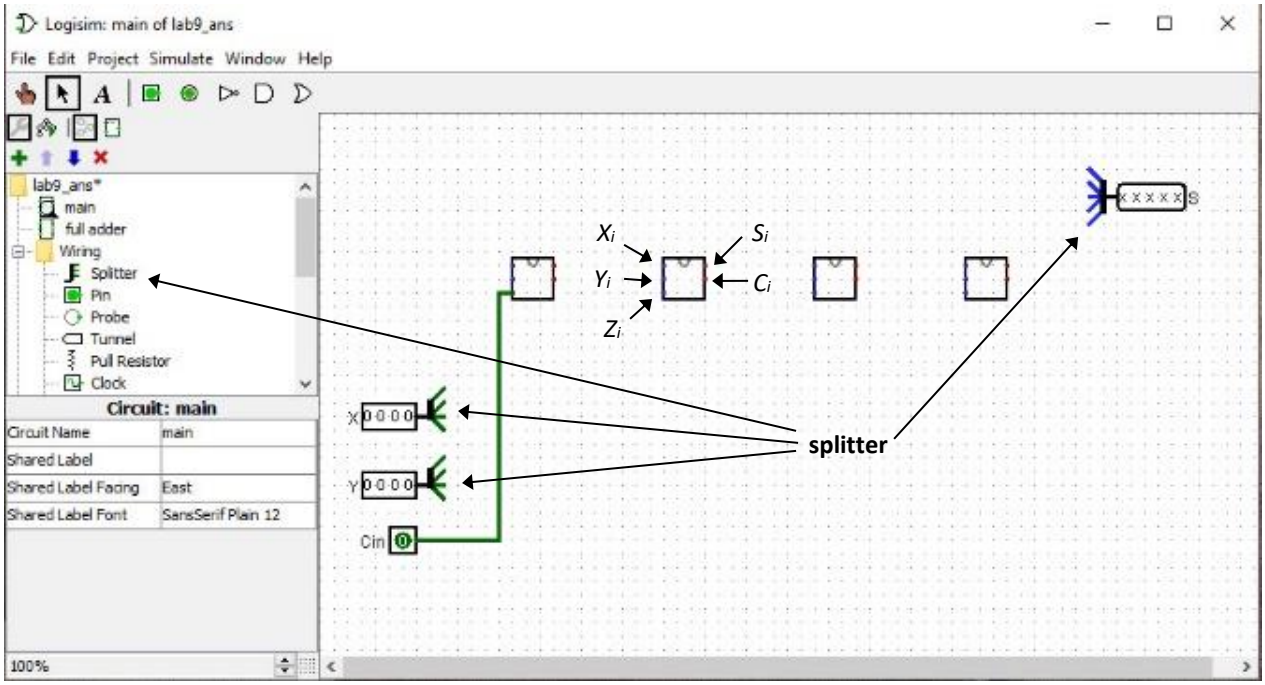
   C = _____

7. Currently, the circuit you have is in the "main" circuit. Now, click on "Project" → "Add circuit...". A pop-up menu will appear asking for the circuit name. Enter the name with the answer you have for part 3 above. This will create a new entry with that name just below "main". Let's refer to this name as xxxx here for the subsequent parts.

8. Transfer the circuit you have in "main" (using the select button  and click and hold the left mouse button to select the whole circuit, then press **ctrl-x** to cut) and paste it into the newly created "xxxx" circuit (click on "xxxx" – making sure the magnifying glass is over it – and press **ctrl-v** to paste).

9. Go back to the "main" circuit (which should be empty now). Create a **4-bit parallel adder** here by using 4 copies of the xxxx circuit you have created earlier. A partial diagram is shown below.

   - Each xxxx is represented by a block diagram. The labels are indicated in one of the block diagrams below for your reference.
   - The 4-bit inputs *X* and *Y* are created by clicking on the input pin button  and specifying 4 data bits in the attribute table. Likewise, the 5-bit output *S* is created by clicking on the output pin button  and specifying 5 data bits in the attribute table.
   - Splitters (refer to the Logisim tutorial, "Wire bundles" → "Splitters" for more details) are used to route the different bits in the inputs and outputs.



10. <u>Show the completed 4-bit parallel adder circuit to your lab TA</u>.

Report: 5 marks
Demonstration: Part 4 (2 marks), Part 10 (8 marks)
Total: 15 marks

Your graded report will be returned to you at your next lab.