# CSCI 150 - Lab 1

Due: Wednesday, Week3
Submit to: Moodle

Note: this Lab1 aims to be started from the Lab class and to be finished within the class as much as possible. Show the instructor your done or partially done lab by the end of the class.

## Objectives:

- Start up **LogicWorks 5**
- Get to know different windows in the LogicWorks 5
- Build circuits in the design/schematic window
- Test circuits
- Simulate circuits
- Save and print circuits
- Submit lab files

## Lab activities:

1. Login into your Columbia College account
2. Right click on Start button, open Windows Explore
3. Crete a new folder called csci150 under your OneDrive
4. Create a sub folder called Lab1_yourName under csci150
5. Start up LogicWorks 5
6. Read the following lab tutorial and practice with LogicWorks 5
7. Save your practice to the file called Lab1.ctt under your lab1 folder.
8. Create a Truth table for the Boolean Algebra
9. Simulate and verify the waveforms against the truth table

10. Save a screenshot and the truth table in a word document as Lab1_report.doc and convert it into Lab1_report.pdf
11. Make a zip file as Lab1_yourName.zip containing the Lab1.ctt file and Lab1_report.pdf

Notes:

- While working on campus, always store all your files on your OneDrive. That way your files will be accessible to you from any computer at Columbia College.
- Unless you install a copy of LogicWorks 5 on your PC / Laptop, you need to work from the college computer to access LogicWorks 5 and save all your files to your OneDrive.

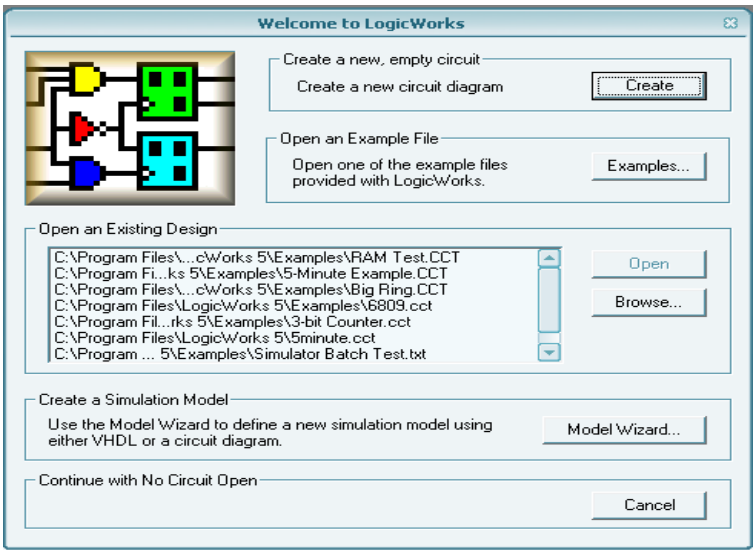
## Introduction

LogicWorks is an interactive circuit design tool intended for teaching and learning digital logic. It is a program that we can use for designing and simulating circuits. This lab will introduce you the basic features of LogicWorks and familiarize you with the program by stepping you through the construction of a simple circuit.

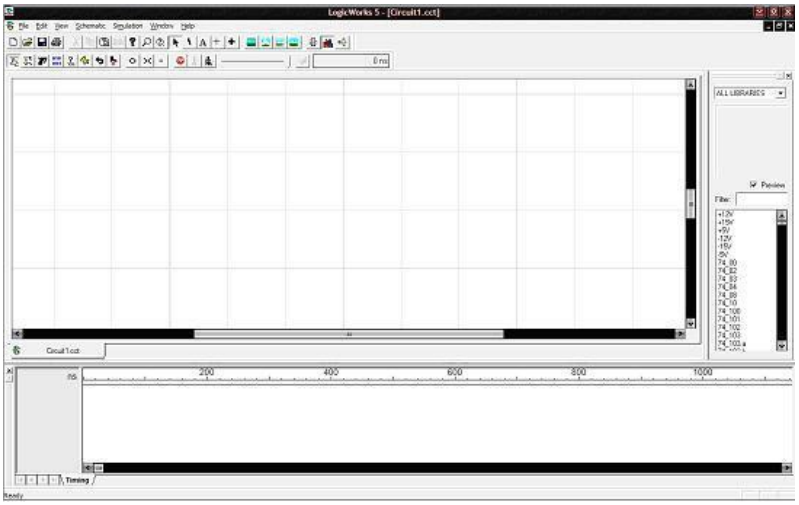
## 1. Start up LogicWorks 5

You start LogicWorks by clicking on the icon 

on the desk top or selecting it from the Microsoft Windows Start menu. Once the tool is open, you will see the "Welcome to the LogicWorks" screen.



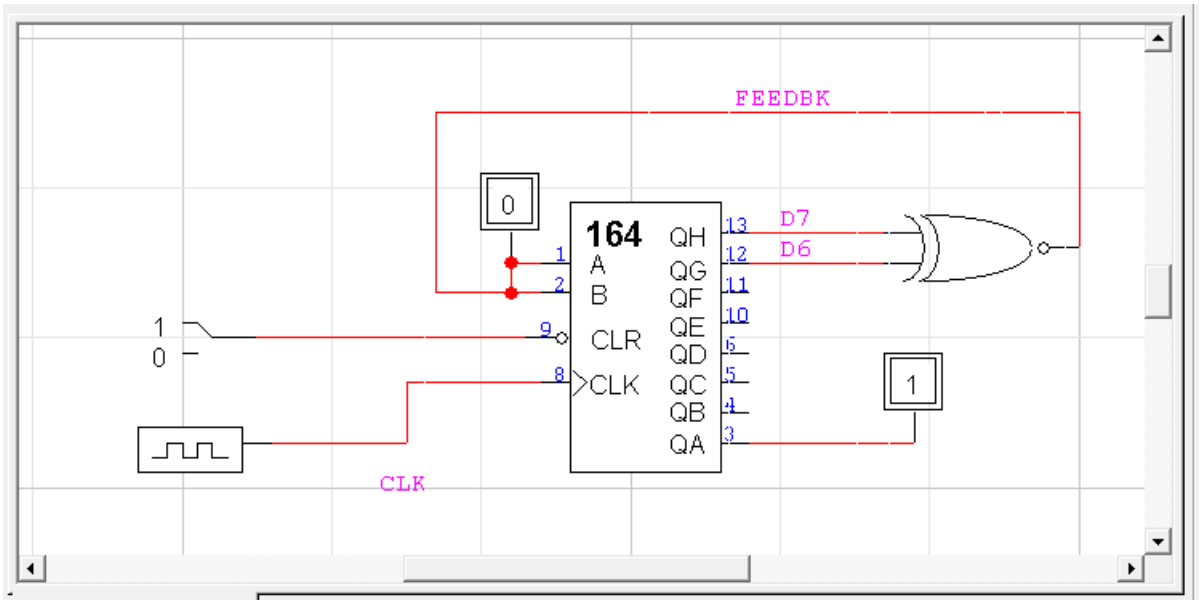

Select "Create a new circuit diagram" by clicking on the "Create" button. Then the following screen is displayed.



There are three primary window components in LogicWorks. They are Circuit Window, Timing Window, and the Parts Palette. Now let's look at each of them.

## 2. Circuit Window

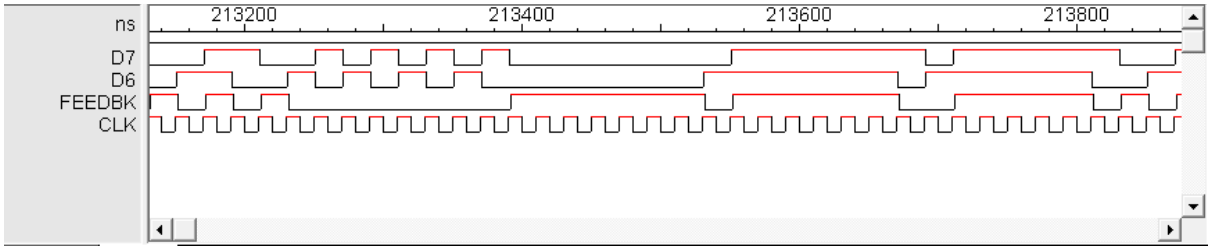
The circuit window is where you build your circuits. For example, you can select different parts from the Parts Palette and place the device in the circuit Window to build your circuits. The following is a screen capture of the circuit window with a sample circuit already drawn.




You can select any part of the circuit by left clicking on it, once selected, a right click can bring up a pop menu that displays the options for manipulating the selected component.

## 3. Timing Window

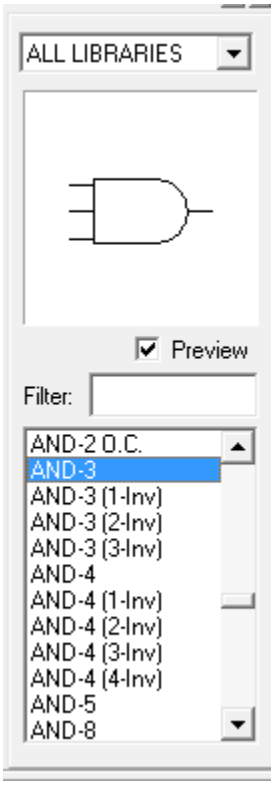
When a circuit is simulated, the Timing diagram window can be displayed to show signal values versus time. Only one Timing window can be displayed at any given time, and it gives waveforms generated by the current design. Closing the Timing window does not close the circuit design file. The following picture shows the corresponding timing window of the above example circuit.




Like the Circuit window, the intervals of the timing diagram can be selected by using click and drag the mouse over the desired interval. Once a section of the diagram is selected, right click brings up a pop menu with options that can be used to manipulate the Timing diagram.

## 4. Parts Palette

Parts Palette is another very important part of LogicWorks interface. By default, it is located to the right of the Circuit Window.




When you click on a device, it shows in the preview area. This is illustrated by the above picture. Parts Palette contains a group of parts libraries. You can choose a library and select a device by double clicking on the device name, then place it in the Circuit Window for building circuits.

## 5. Create a Circuit

In this section, you will create and test a circuit that implements a Boolean equation, for example:

$$Z = AB + BC + A\overline{C}$$

**This requires the following components:**

1. Three AND gates with two inputs (AND-2)
2. Two OR gates with two inputs (OR-2)
3. A single NOT gate commonly called an inverter
4. Some wire to connect the gates
5. Three switches to provide a way to modify the input values for testing
6. A binary probe to show the results of the circuit

**The first three components can be found in the "Simulation Gates.clf" library:**

Click on the pull-down menu on the *Parts Window*
Click on the "Simulation Gates.clf" library.

"Simulation Gates.clf" should now be displayed in the pull-down menu window of the *Parts Window* as shown in Fig. 1. The next portion of the *Parts Window* displays a list of the logic gates contained in this library. The scroll bar can be used to view them all.

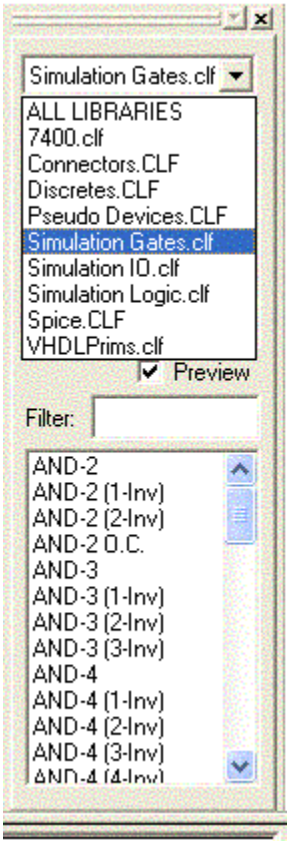


Fig. 1

**Put the desired devices on the schematic (The *Circuit Design Window*):**

### *AND Gates:*

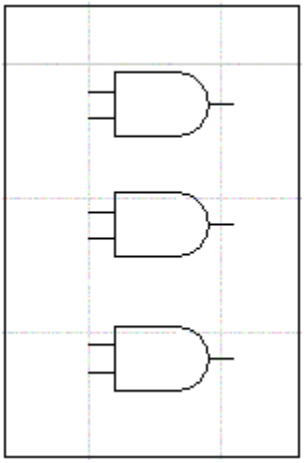


Fig. 2

Find the AND-2 gate in the *Parts Window* and double click on it.
Move the mouse pointer to the *Circuit Design Window* and place three of these gates where you want them by clicking once for each gate. See Fig. 2.
***Note:***Spreading the gates apart a little makes it easier to connect parts to them later.
Hit *Esc/Space* so that no more AND gates are selected.

### *OR Gates:*

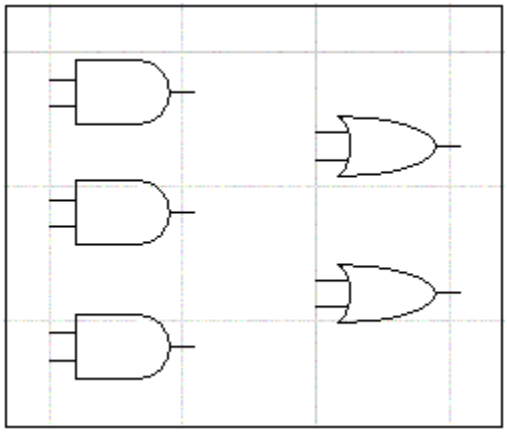


Fig. 3

Find the OR-2 gate in the *Parts Window* and double click on it.
Move the mouse pointer to the *Circuit Design Window* and place two of these gates to the right of the AND gates. See Fig. 3
Hit *Esc/Space* again so no more OR gates are selected.

### *NOT Gates (Inverters):*

Find the NOT gate in the *Parts Window* and double click on it. Move the mouse pointer to the *Circuit Design Window* and place one NOT gate to the left of the bottom AND gate. See Fig. 4 Remember to hit *Esc/Space* so the NOT gate is no longer selected.

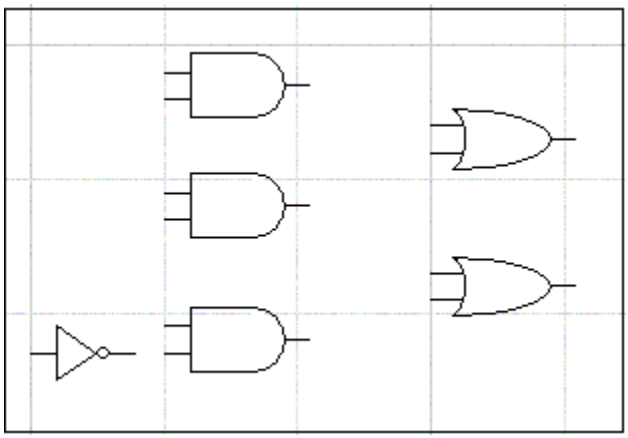


Fig. 4

## 6. Moving and deleting existing gates:

By single clicking on an item on the schematic (The *Circuit Design Window*)

the device will be selected and highlighted. While the device is selected,

the *Delete* key will remove the item from the circuit window.

To move a device, point at it and hold down the left mouse button,

then move the mouse to the desired location. Release the mouse button.

## 7. Adding switches:

- Click on the pull-down menu in the *Parts Window*.
- Click on the "**Simulation IO.clf**" library.
- Double click on the binary switch entry and place three of them on the circuit window on the left of your design.

- Double click on the Binary Probe entry and place it on the circuit window on the right of your design

**Note:** This library contains primitive Input and Output (I/O) devices such as "Binary Switch", "Binary Probe", etc.

## 8. Connecting the Devices

## 9. Adding wires:

- Place the cursor on the right edge of the switch and hold down the left mouse button.
- Drag the mouse a half-inch or so to the right and release the button.
- A red wire should now be attached to the switch, ending in the middle of nowhere. See Fig. 6.
- Repeat this for each of the three switches.

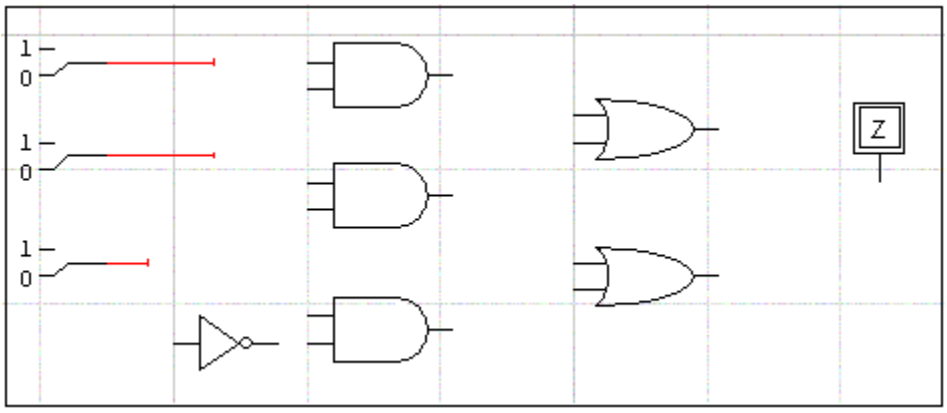



Fig. 6

Recall that we are trying to implement Z = AB + BC + AC'. To clarify your design and show its relation to the equation above, we will label the wires A, B, C, and Z.

## 10. Labeling wires:

- Right-click on the wire you want to name.
- Select Name from the box that appears, as in Fig. 7.
- Type the name of the wire in the text box. Be sure to check Visible as in Fig. 8

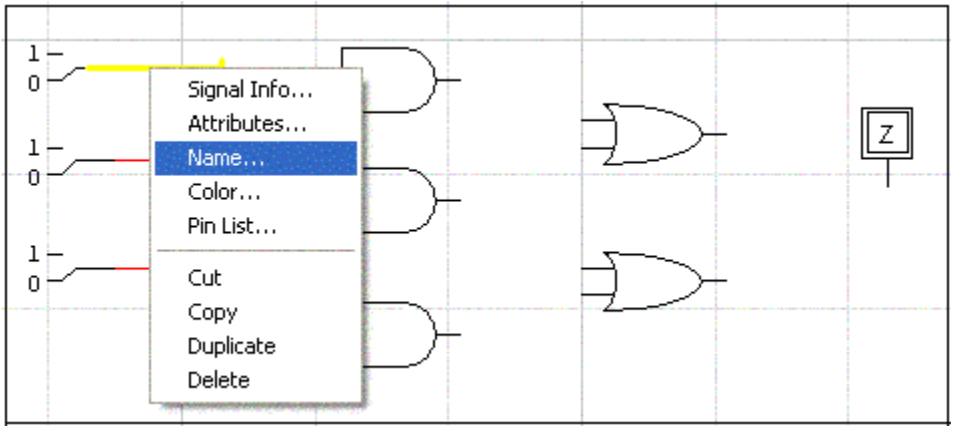



Fig. 7

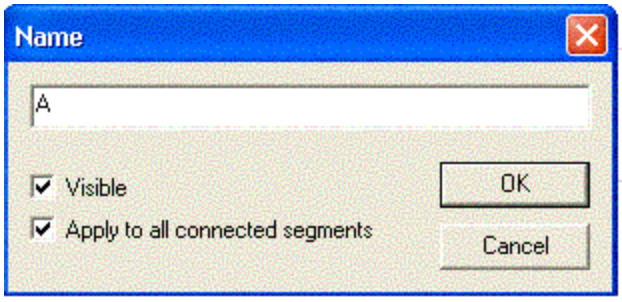



Fig. 8

***Note:*** This process can also be done by clicking on the button **A** on the top tool bar.  The mouse pointer then looks like a little pencil. Point this pencil at one of the wires (remember only the red parts are wires, if you click on a black part, you are labeling a pin) and click on it. A text box appears, and you can type a name for the wire. Hit *Esc/Space* to exit the name process.

When you named these wires, you should have noticed that in The Timing Window the names appeared. This will later show you the values of these three variables.

## 11. Connect the devices together with wires:

- Place the pointer on the end of the wire extending from switch A and hold down the left button, then drag the mouse to an input of the first AND gate.
- You have now connected the switch for input A to the first input of the AND gate.
- Connect the wire from switch B to the other input of the first AND gate in a similar manner.
- Click on the first input of the second AND gate.

- Drag a wire to any point of the wire extending from switch B. A dot will appear on the intersection if the wires were successfully connected.
- Connect the C switch wire to the input of the inverter.
- Connect the output of this inverter to one of the inputs of the last AND gate and connect the other AND gate input to switch A.
- Your design should now resemble the circuit window in Fig. 9.

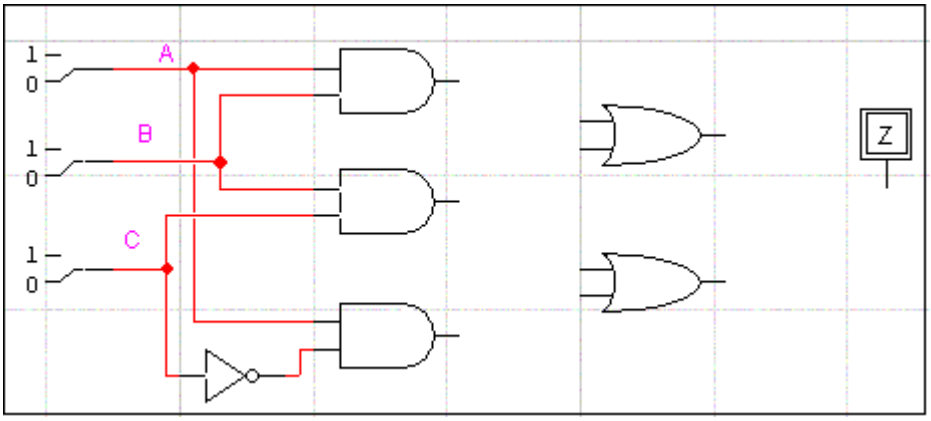



Fig. 9

***Note:*** The wire will go around one corner for you, but you may have to stop in the middle of the circuit window with one wire and then restart to go in another direction. Also, an intersection without a dot is simply two wires crossing without making a connection.

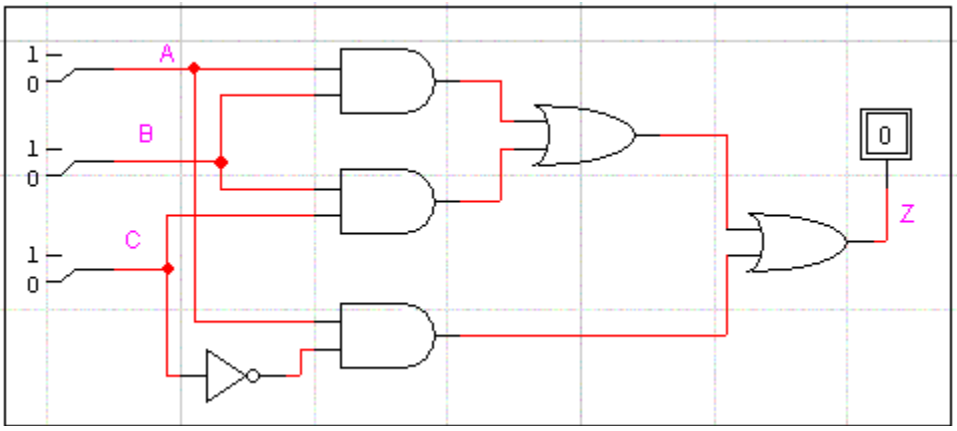



Fig. 10.

- Connect the outputs of two of the AND gates to the inputs of one of the OR gates.
- Finally we connect the output of the remaining AND gate and the output of the first OR gate to the input of the last OR gate.
- The Binary Probe should now be connected to the output of the final OR gate and the wire should be labeled Z. See Fig. 10. Notice that the output Z showed up in the *Timing Window*

## 12. Simulation Controls

Here is the Simulator Toolbar:


- Position your mouse pointer on each botton, you will find the function of each of them. You may try it on your own.

**For example,**

1. Clicking on the <> or >< button can adjust the range of the time values to suit the display data.
2. Clicking on the Reset button can make the simulation restart at time 0.
3. Clicking on the Run button can start the simulation.

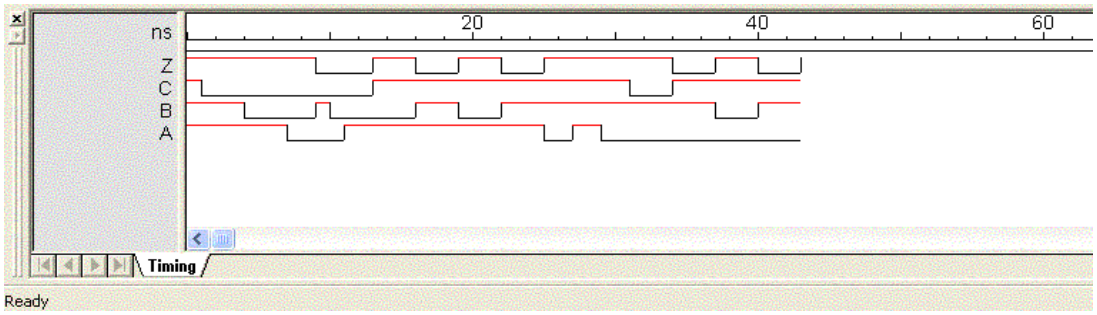
- You may have noticed that in the *Timing Window* that there are small lines drawn like underscores for inputs A, B, and C. These are low because the switches are all set to the zero value.

## 13. Testing your Circuit:

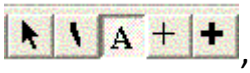
- Click on the handle of switch A to point to the value 1.

> Notice that the A line in the *Timing Window* went up to a high level, indicating the value 1. Also notice that the Z output went high. This is because A is high and C' is high and according to the equation Z should be high with these input values.

- Change the values of A, B, and C and observe the results in the Timing Window.
- Click on the button **<>** in the Simulator toobar. This will stretch the time out, making the waveforms easier to see.



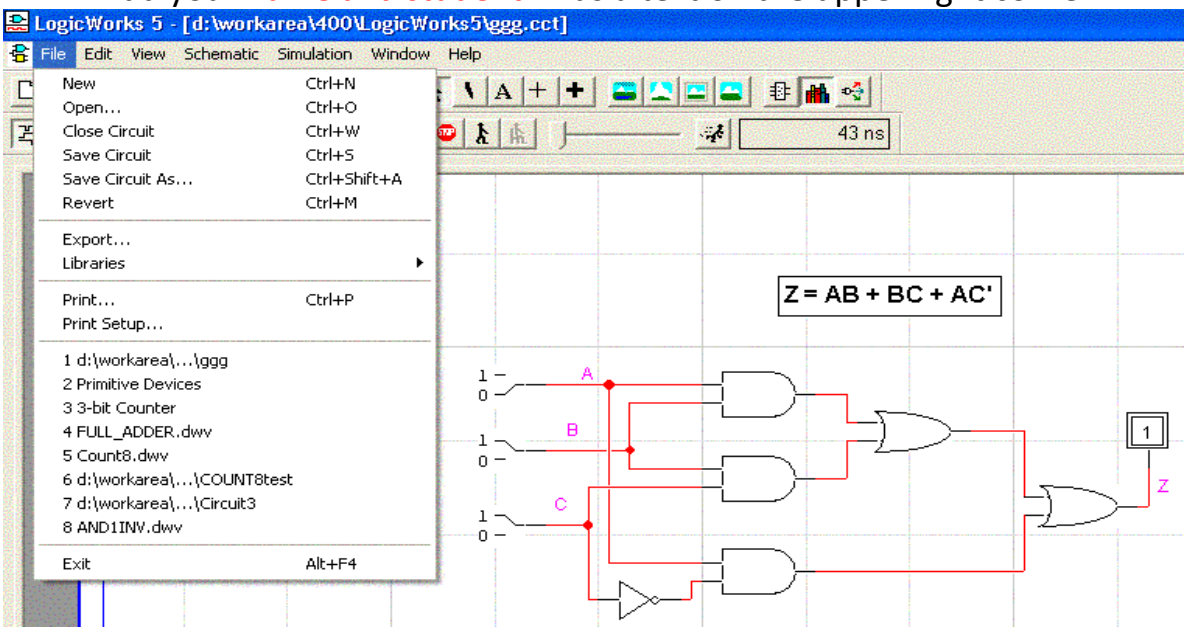

## 14. Inserting Text in the Circuit Window:

- To insert the text in the circuits window, click on the button **A** in the design toolbar  , then follow the following steps.

> 1. Position your pencil (cursor) at the desired location.
> 2. Type in the text
> 3. Change cursor to the pointer by clicking on the pointer in the toolbar or press the space bar or escape key.
> 4. Click on the text entered, then right click.

5. Select "Text Style" and then select the preferred options and then, press ok.

- Add the Equation of the circuit as a text in the circuit design window.
- Add your name and student ID as a text on the upper right corner




## 15. Simulating the circuit with the Truth Table

- Create a Truth Table of the given Boolean algebra equation
- Click on the input binary switches, to simulate and verify the output against the Truth Table

## 16. Saving your work:

1) To save your work, click the File menu and then select the proper option. The following picture illustrates the idea.
2) Save Circuit as Lab1.cct
3) Press "Alt + Prt Sc" to make a screenshot of circuit window and timing window, and paste it to a word document as Lab1.doc
4) Add the truth table into Lab1.doc
5) Convert Lab1.doc into a PDF file as Lab1.pdf

**Notes:** Save your work frequently whenever you made some progress, to avoid from losing your work.

## 17. Submit your work:

- Make sure you have created the following zip file containing two files:
  - Lab1_yourName.zip
    - Lab1.ctt (file)
    - Lab1_report.pdf (file)
- Submit your Lab1_yourName.zip file to Moodle