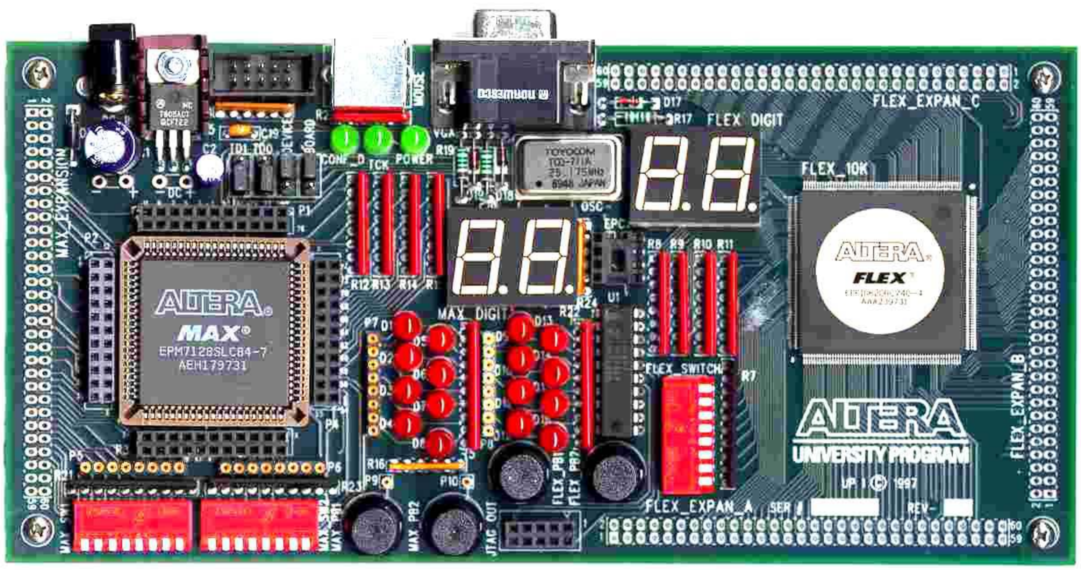
MAX PLUS II TUTORIAL



PASCAL LYONNAIS
WINTER 2003

Acknowledgement

This document is a modified part of lab manual and tutorial contained in the following documents:

- Altera MAX PLUS+II Tutorial available on the web from Altera official website.
- Dueck, Robert K., *Digital Design with CPLD Applications and VHDL*, Delmar Thompson Learning.
- James O. Hamblem and Michael D. Furman, *Rapid prototyping of digital systems, A tutorial Approach*, second edition, Kluwer Academic Publisher.

DOCUMENTATION: HOW TO / FAQ with Altera MAX+PLUS II

You can also search the web with your favourite search engine (Google.com recommended).

Q: How can I get training on MAX+PLUS II software?

There is many ways to get trained with MAX+PLUS II. You will find below some documents and links that provided tutorial. Again the Web is full of tutorial.

- This basic tutorial.
- Check the MAX+PLUS II Software help menu and files.
- Altera MAX+PLUS II Getting Started: http://www.altera.com/literature/manual/81_gs.pdf
- Altera MAX+PLUS II Tutorial : http://www.altera.com/literature/manual/81_gs3.pdf
- Altera Feature Textbooks: http://www.altera.com/education/univ/unv-textbooks.html

Q: Where can I find more Altera literature?

- Altera University program: http://www.altera.com/education/univ/unv-students.html
- Altera Data and complete literature: http://www.altera.com/literature/lit-index.html
- University Program Design laboratory Package: http://www.altera.com/literature/univ/upds.pdf
- University Program Altera FAQ:
  http://www.altera.com/education/univ/unv-faq.html
  http://www.altera.com/education/univ/unv-kits.html

  http://www.altera.com/education/univ/unv-index.html and click on **laboratory kits.**
  http://www.altera.com/literature/univ/upds.pdf

  http://www.altera.com/literature/ds/m7000.pdf

Q: How can I install Altera MAX+PLUS II on my PC at home and get a license?

Follow the instruction at:
http://www.altera.com/education/univ/unv-software.html

MAX+PLUS Student Edition Software at:

MAX+PLUS II Student edition software:

https://www.altera.com/support/software/download/altera_design/mp2_student/dnl-student.jsp

After installation, students can register to obtain an authorization code:

http://www.altera.com/support/licensing/lic-university.html

Altera UP-1/2 board documentation:

http://www.altera.com/education/univ/unv-kits.html

http://www.altera.com/education/univ/unv-index.html and click on **laboratory kits.**
http://www.altera.com/literature/univ/upds.pdf

http://www.altera.com/literature/ds/m7000.pdf

There is a lot of documentation on the web, just do a search to find out more… But don't forget to acknowledge any contributions.

Introduction to MAX+PLUS II Software Design

Objectives

This laboratory experiment is intended:

- To initiate the students who are not familiar with the Altera MAX+PLUS II Software Design.
- To act as a review for the more advance students.

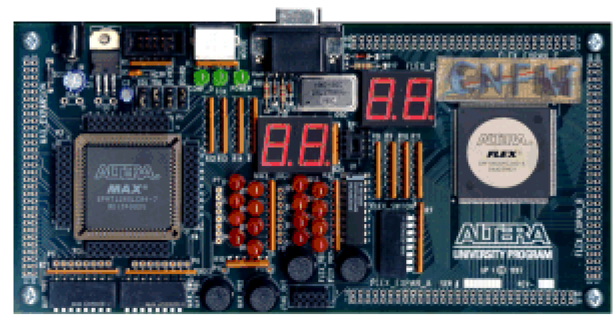


Figure 1 Altera UP1/2 board

On completion of this tutorial, the student will be able to:

- Understand the basic of the Altera environment.
- Design a simple logic circuit using the Graphic editor.
- Compile, simulate, debug, and test their design.
- Download and run their design on the Altera UP1/2 board.

PreLab

1. Read the Altera UP1/2 board documentation and visit the Altera website to familiar yourself with the Altera UP-1/2 board.

Laboratory

In this tutorial, we will implement a simple circuit as shown below with AND, NAND and NOR functions to provide an introduction to the Altera MAX+PLUS II tools.

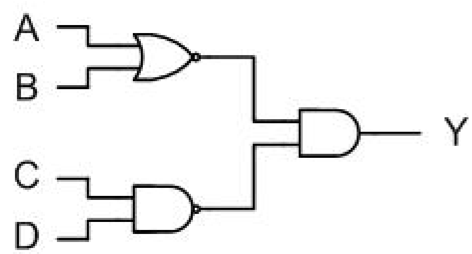


Figure 2 A simple circuit with AND, NAND and NOR gates

PART I

A. Design using the Graphic Editor

1. Start **MAXPLUS** software. Choose **File, New**, in the new window select **Graphic Editor** (*.gdf file) and click **OK** to create a blank schematic worksheet.
2. From the **File** menu, click **Save as**, and save the file in a new folder (eg., ..\my file\tutorial.gdf) and click **OK**. If the new folder was not created, just type the complete path in the **File Name** box.
3. In the **File** menu, select **Project**, then **Set Project to Current File**.
4. Then select **Assign**, **Device.** In the **Device Family,** select the **MAX7000S** device family. Uncheck **Show Only Fastest Speed Grades**. Under **Devices,** select **EPM7128SLC84-7** and click **OK.**
5. If a dialog box open up and recommends to turning on the "Maintain Current Synthesis…", just click **No**.

B. Creating the schematic

1. **Right click** in the center of the worksheet, and then choose **Enter Symbol**. In the **Symbol Libraries** box, double click on the **..\prim** library.
2. Scroll down in the **Symbol Files** box and double click on **nor2**. The symbol should appear in the center of the **Graphic Editor**.
3. Repeat step 1 and 2 and select a **nand2** symbol.
4. Repeat step 1 and 2 again and select an **and2** symbol.

C. Assigning Output and Input pin

1. From the toolbar, select **Symbol → Enter Symbol** and click **Ok**.
2. In the **Symbol Libraries** box, double click on the **..\prim** library.
3. Scroll down in the **Symbol Files** box and double click on **Output**. The symbol should appear in the center of the Graphic Editor.
4. Repeat step 1 and 2 and select an **Input** symbol.
5. With the right mouse button copy and paste three more **Input** symbol.

D. Connecting the Symbol

1. Go to the end of a symbol with the mouse and when the cross-symbol cursor appears drag the wire to the point it connect, see diagram below for the connection.
2. Repeat the previous step for all connection.
3. If a wire is not properly run, just selected (wire turns red) and hit delete to remove it.
4. If you have problem running the wired from one point completely to another, try running half way from both devices.
5. The mouse can also be used to move a wire to the desired position.
6. Now, your diagram should look like the one below.

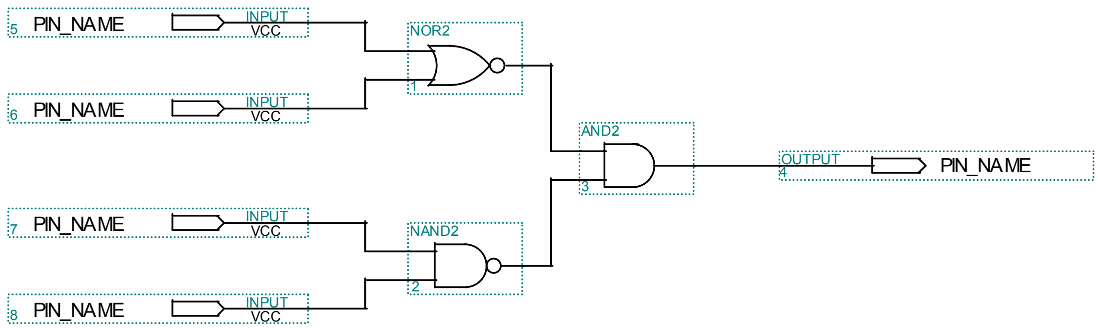


Figure 3 Schematic with input and output wired

E. Editing Pin Names

1. **Right click** on an **INPUT** Symbol and select **Edit Pin Name**.
2. Name the pin as shown below.
3. Double clicking on the pin names will have the same results.
4. Repeat for all pins.

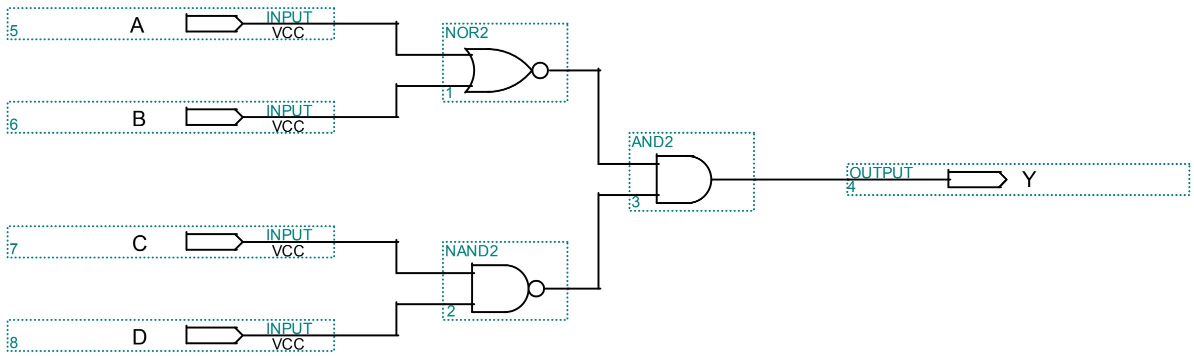


Figure 4 Schematic Ports Name

F. Assign PIN Numbers

1. Right click on the input symbol **A** and select **Assign→Pin/Location/Chip**.
2. Make sure you the right symbol name is in the **Node Name** box.
3. In **Chip Resource**, select **Pin** and enter the **Pin** number, see table below.
4. In the **Pin Type** box, select the required type, see table below.
5. Select the **Assign Device** box and make sure that the device is **EPM7128SLC84-7,** click **OK**.
6. Click **OK** in the **Pin/Location/Chip** box and repeat the previous step for all **Inputs/Outputs.** See table below for all pins numbers and types.
7. Do not forget to **save** your file.

Pin Name	Pin Number	Pin Type	Location On the board (use only one)	
A	12	Input	P2 #12	MAX_EXPAN #22
B	16	Input	P2 #16	MAX_EXPAN #24
C	18	Input	P2 #18	MAX_EXPAN #26
D	21	Input	P2 #21	MAX_EXPAN #28
Y	25	Output	P2 # 25	MAX_EXPAN #30

Table 1 Device Pin connections and name

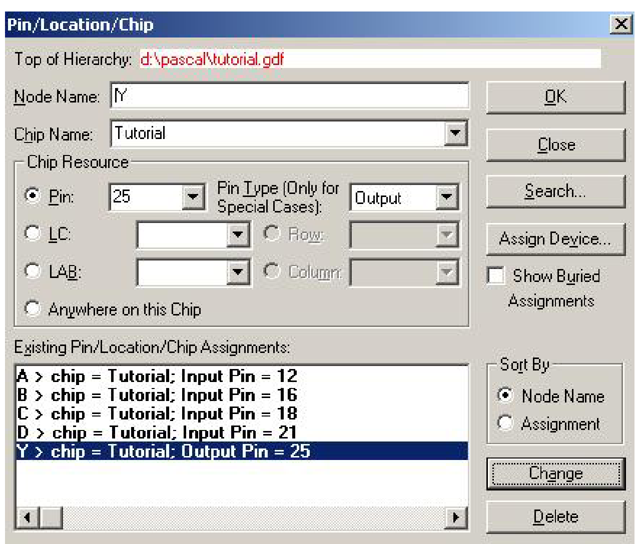


Figure 5 Pin/Location/Chip window

G. Compiling your project

1. Select **File →Project→ Save and Compile.**
2. A similar window should appear.

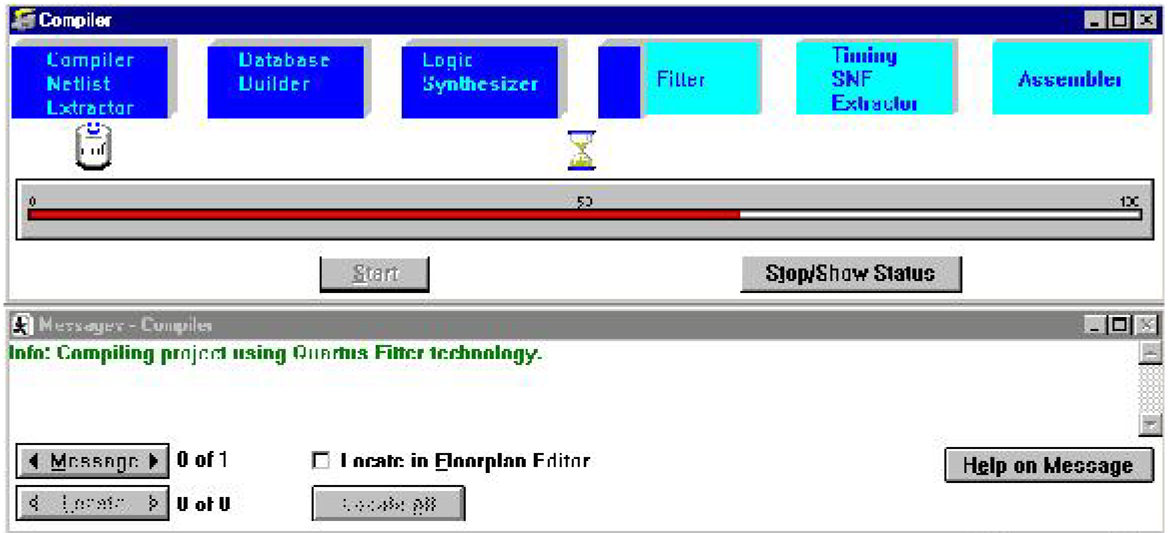


Figure 6 Compiling Window

3. The project should compile with **0 errors**. If any errors appear verify if you have performed the entire steps correctly.

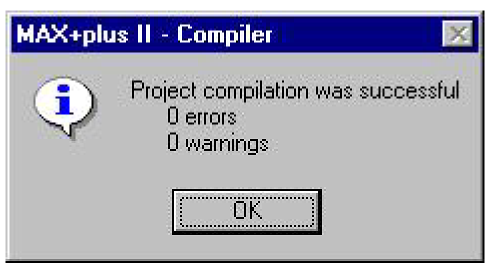


Figure 7 0 error window

4. Close the compiler window.

PART II

A. Simulating your project

1. Select **File →New→ Waveform Editor File** and click **OK**.
2. From the toolbar, select **Node→Enter Nodes from SNF** (Simulator Netlist File).
3. Click on **List**.
4. Select (highlight) A, B, C, D and Y from **Available Nodes & Groups**.
5. Click on => to have all A, B, C, D, and Y in the **Selected Node & Groups**.
6. Click **OK**.
7. You can drag the Node to have an appropriate order.

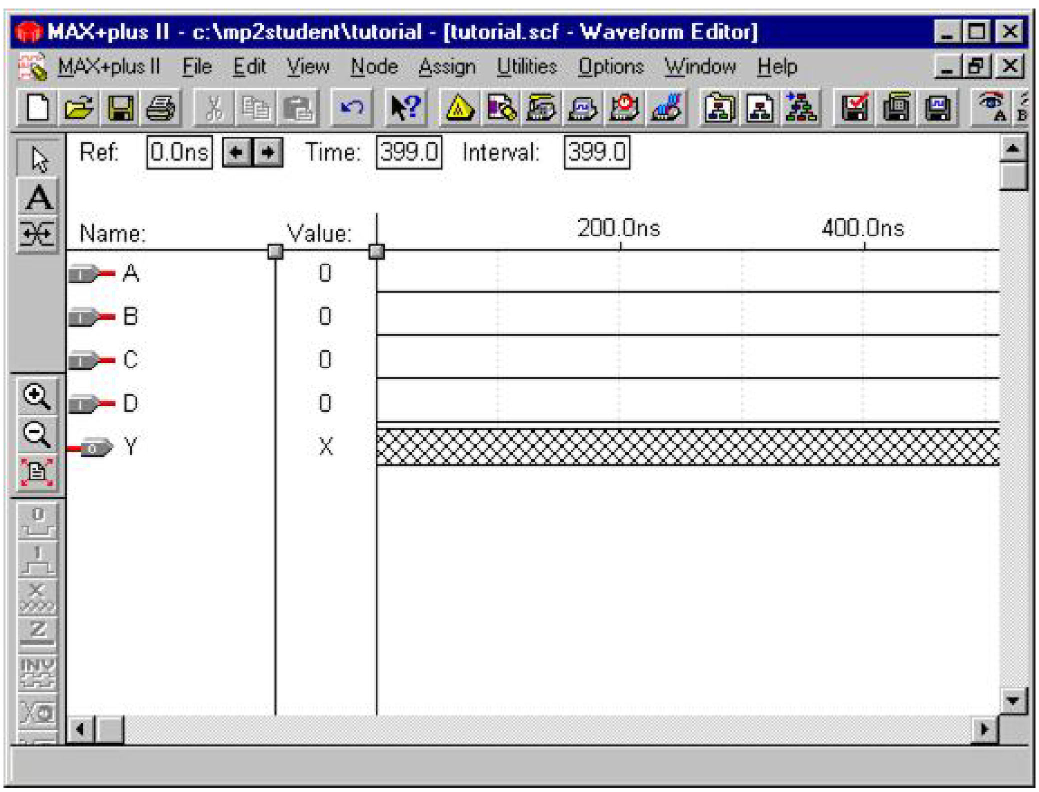
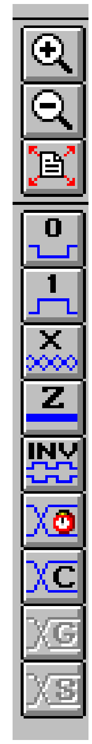


Figure 8 Waveform Editor Window

8. Right click on **A** and select **Overwrite→Count Value** and click **OK**.
9. Select **View→Time Range**, set **From and To** range to **0.0ns** and **500.0ns**.
10. Right click on **B** and select **Overwrite→Count Value**, change **Multiplied By** to **2** and click **OK**
11. With the Mouse left button, click and drag the mouse from 100.0ns to 300.0ns for Node C, This interval would then be highlighted.
12. Go to waveform manipulation buttons and select 1 for the desire interval.
13. Repeat step 11 and 12 for Node D and referred below to reproduce the same interval as the example.
14. Go to **File** menu and select **Save**.
15. Save option would automatically select filename to be the same as the project name, click **OK**.
16. Go to **MAX+PLUS II** menu and select **Simulator**.
17. Click **Start** when the simulator dialog box appears.
18. Once simulation is done, the finish dialog box would appear. Click **OK**.
19. Click **Open SCF** to see the simulation result.
20. Simulate different scenario and explain the simulation result.



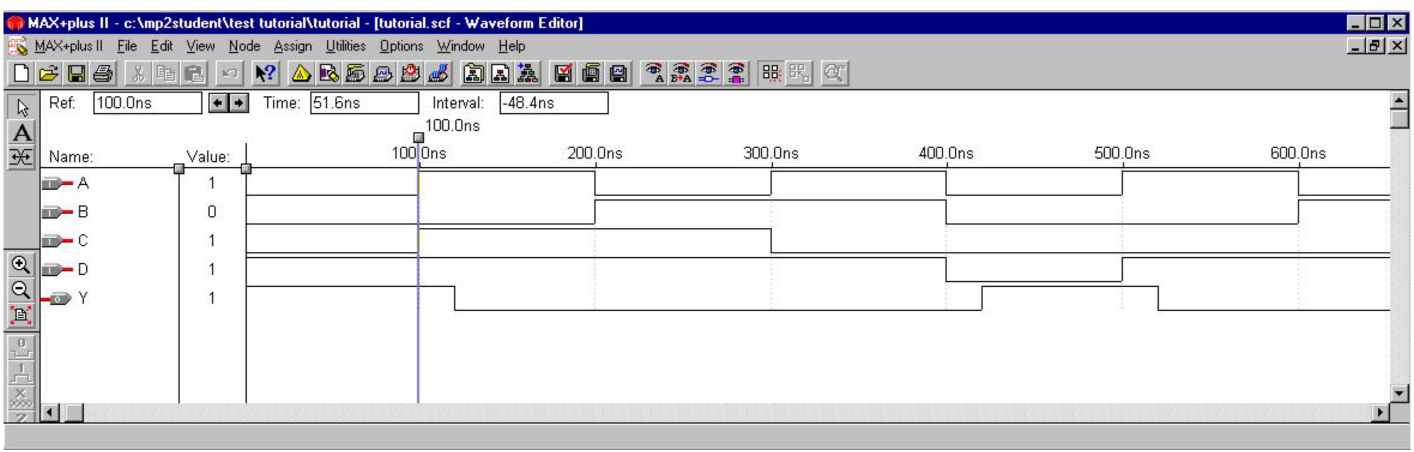


Figure 9 Simulation example

PART III

A. Downloading your project to the UP1/2 board

1. Make sure the **Byteblaster** cable is attach to the board and to the parallel port on the PC.
2. Verify that the board is properly powered using a **9V DC transformer** and is attach to the **DC_IN** located on the corner on the UP1/2 board.
3. Verify the jumper setting, see appendix for more information.
4. Select **MAX+PLUS II → Programmer**.
5. From **Option→ Hardware Setup**, in the **Hardware Type** box, select **Byteblaster(MV)** and click **OK.**
6. Select **JTAG→ Multi-Device JTAG chain Setup**.
7. Click **Select Programming File** and select **tutorial.pof** file and click **OK** (sof files are for FLEX devices).
8. Click **Add** in the **Multi-Device JTAG Chain Setup** window.
9. Click on **Detect JTAG Chain Info** button. You should get a confirmation hardware check window. If not, verify if you have performed the entire step correctly.
10. Click **OK** to exit the **JTAG Setup** window.
11. In the **Programmer** window, click on **Program.**
12. Click **OK** when the **Configuration complete** box appears.

PART VI

A. Connecting wire to the UP1/2 board

1. Connect the output of the circuit to the LED. Locate the output hole and connect a wire to a LED (see table 1).
2. Connect the input of the circuit to the DIP switch. Locate the inputs holes and connect four wires to one of the two DIP switches (see table 1).
3. Verify your circuit according to the simulation using the DIP switch as input and the LED as output. Remember that the LED illuminate when the input is 0.

PART V

Repeat the previous step with the following circuit.

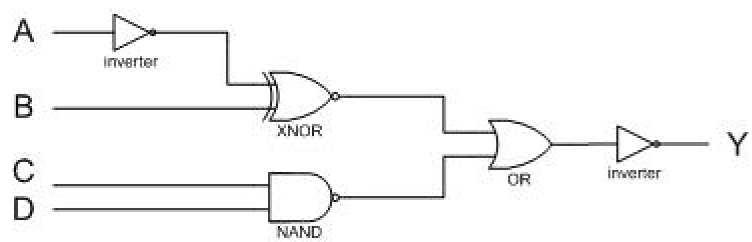


Figure 10 Schematic of new circuit

Now that you are comfortable, do the following:

1. Verify the truth table of the circuit.
2. Verify you circuit according to your simulation.