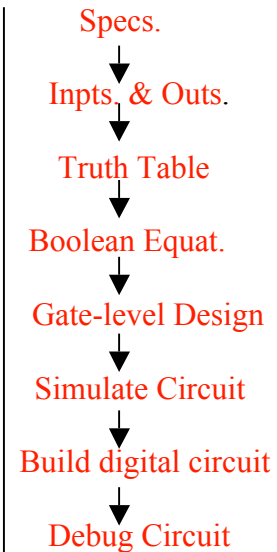


Lab 13: Programmable Logic Design Techniques I

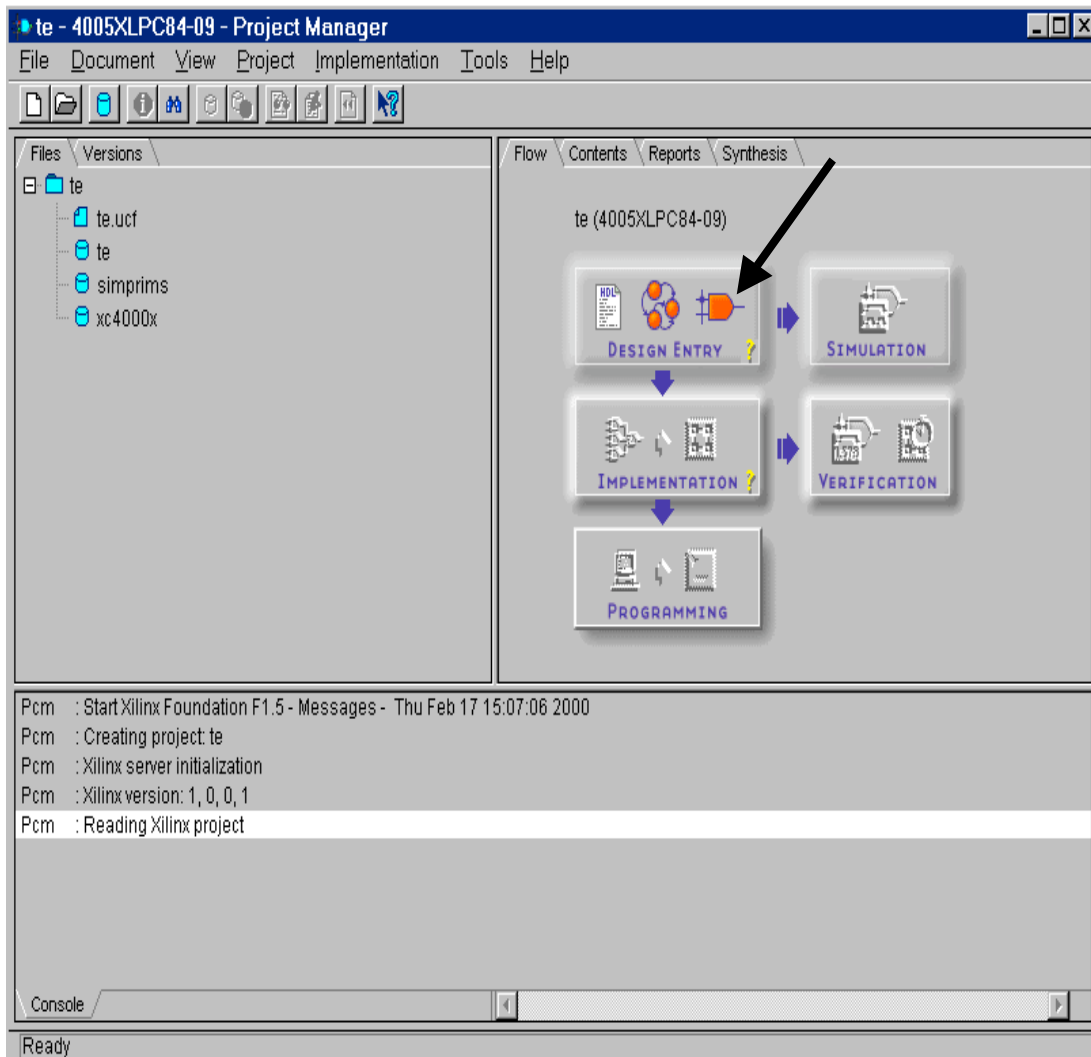
The design of digital circuits is a multi-step process. It starts with **specifications** describing what the circuit must do. Defining what a circuit receives as **inputs** and the **outputs** it generates is the next step. Once these are known the designer has to create a **truth table**, which lists what values the outputs will have for each possible combination of input values. Once the truth table is written down, the designer has to derive **Boolean equations** that describe how each binary output can be computed from the binary inputs using logical operations such as AND, OR, NOT etc. Next, the Boolean equations are transformed into a **gate-level** circuit schematic drawing. Each AND, OR etc. operation in the Boolean equation is replaced with a corresponding AND gate, OR gate etc. in the schematics. Inputs and outputs of these gates are wired to let the passage of binary results between logical operations. Before building the circuit it is a good thing to make sure that all previous steps have been completed correctly. It is done by manual or computer **simulation**. If successful, a physical circuit **is build**. Real inputs are applied to the circuit and **possible bugs**, if any, are fixed.



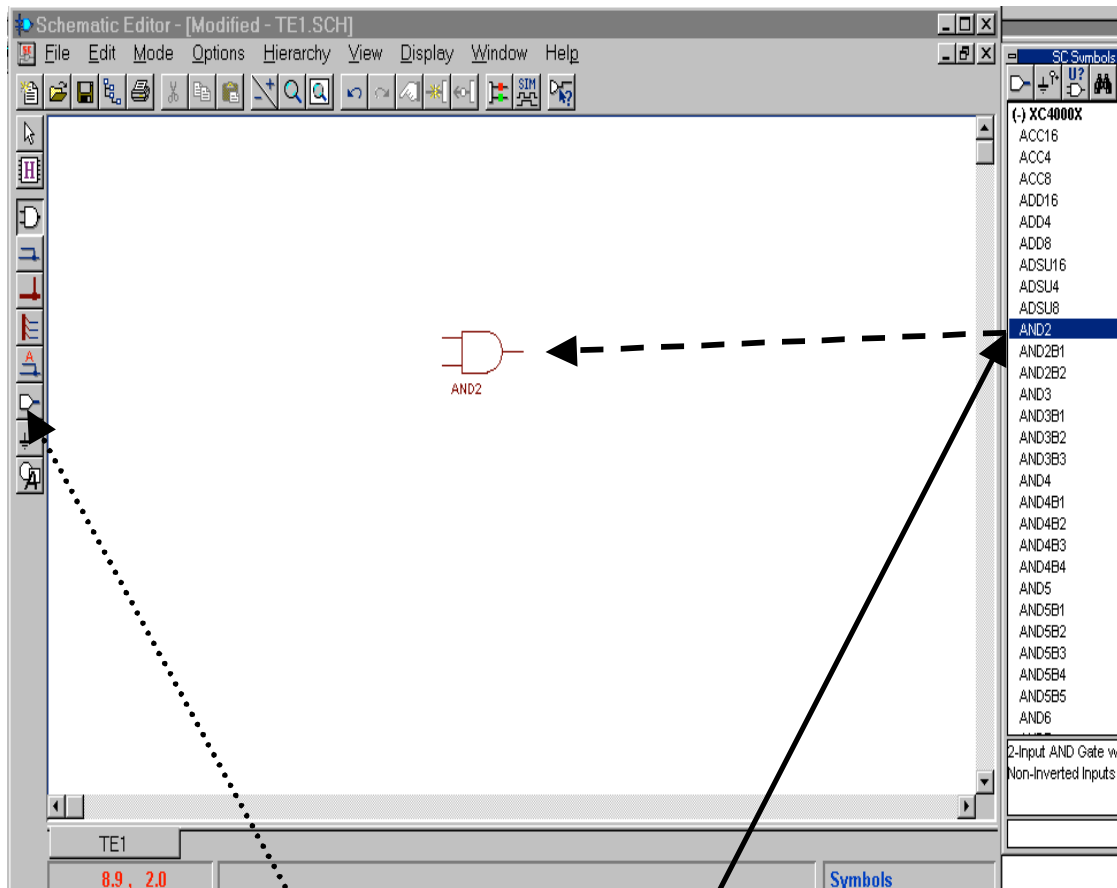
You already built and tested some simple digital circuits using transistor-transistor logic (TTL) devices, and a breadboard with the connections were made using wires. Please, recall Lab 3 and Lab 4 for more details. As you have probably realized, this method of building circuits is not quite convenient: not every type of TTL may be immediately available; wires are often plugged into the wrong place and a lengthy check must be made to find the error; once the circuit works it has to be taken apart to make room for the next circuit; and last but not the least relatively complex digital circuits of more than 10-20 gates are practically impossible to be built and tested. These problems may be eased if a different approach is followed and more elaborated tools employed. The design of digital circuits may still begin by describing the truth table. The details of the logic circuit needed to realize the truth table are, however, worked out by a logic-synthesis program and not by hand. The operation of the "virtual" circuit built is checked using a simulation program. If the circuit simulates correctly, the gates and wires are mapped into a Field Programmable Gate Array (FPGA) using specialized place & route programs. FPGA contains logic gates and the means for interconnecting them within a single Integrated Circuit (IC). The programmed FPGA can be used independently or placed into a larger circuit where it will perform its functions. In this Lab you will be using the XILINX Foundation Series 2.1i software tools to create and test logic designs that can be downloaded into the XC4003E FPGA. *To be successful in this Lab you should review Labs 3 & 4, Chapters 11 & 12 of DH and the Guides to the XILINX software and hardware posted on the PHY440 WWW site.*

Problem 1. Create a digital circuit of a single AND gate and verify its truth table.

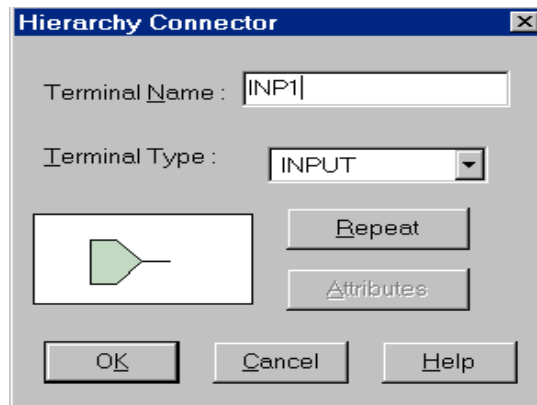
To begin, click on the "XILINX" icon. This will bring up the **Project Manager** Window. Select the **File --> New project** menu item. Then, enter the project name (on your choice), project directory (keep the default one), type of design flow (Foundation Series 2.1i; Schematic), chip family(XC4003E), chip part number(4003EPC84) and device speed (default). Click **OK** to return to the Project Manager Window.



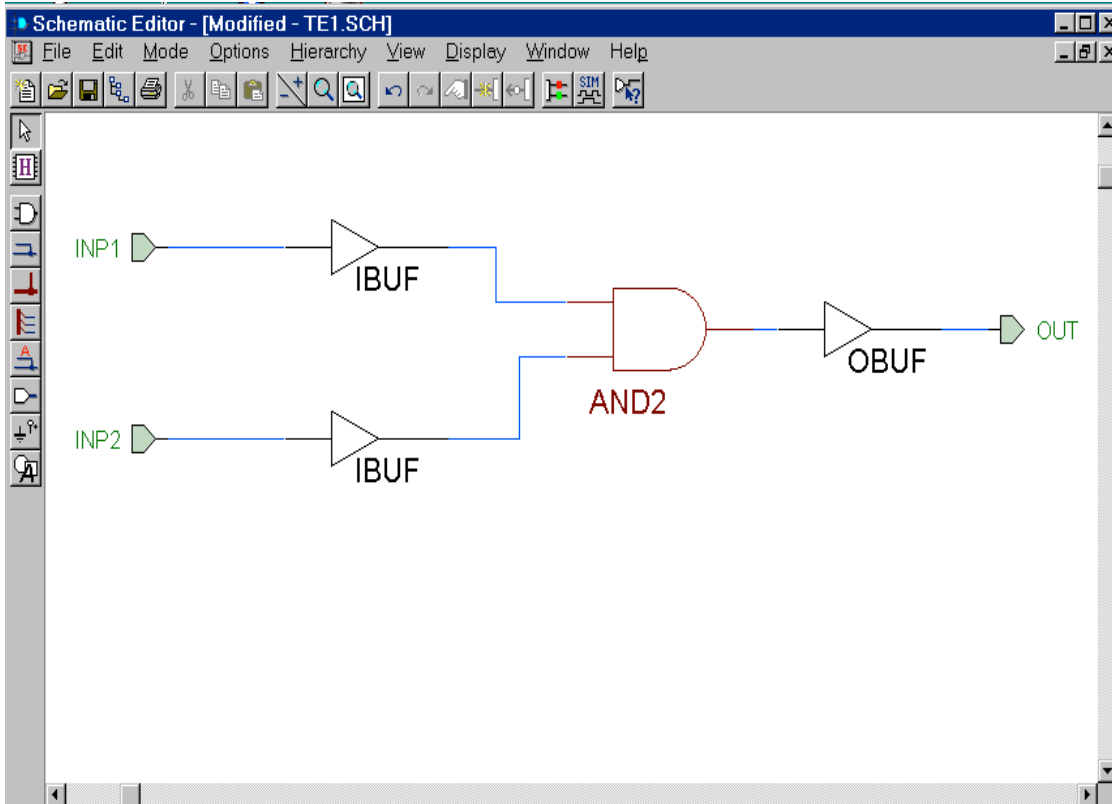
In the **Project Manager** window click on the **Schematic Editor** button (see the screenshot above) and a schematic editor window will appear (see below).



Select the **Mode** --> **Symbols** menu item and the **SC** window will appear with a list of all type of gates/components we can use. Scroll the list of components in the **SC** window, click on **AND2** (two-input AND gate), move the cursor into the drawing area and drop the **AND2** gate there. We need to get our inputs and outputs into the circuit as well. To do this click on the **Inputs** button. A dialog window (see below) will appear in which you have to type the name and the type of each input and output.



While we have entered the inputs and output terminals, we still need to add buffers between the terminals and the logic gate. The buffers indicate that the signals attached to them will actually enter and exit the FPGA chip via its I/O pins. To add input and output buffers we select **IBUF** and **OBUF** symbols, respectively, from the **SC symbol** window and drop them in the drawing area. The next step is to connect(wire) the input/outputs to the AND gate. Select the **Mode --> Draw Wires** menu and do the wiring. A line will appear connecting the inputs/outputs to the gate as shown below.

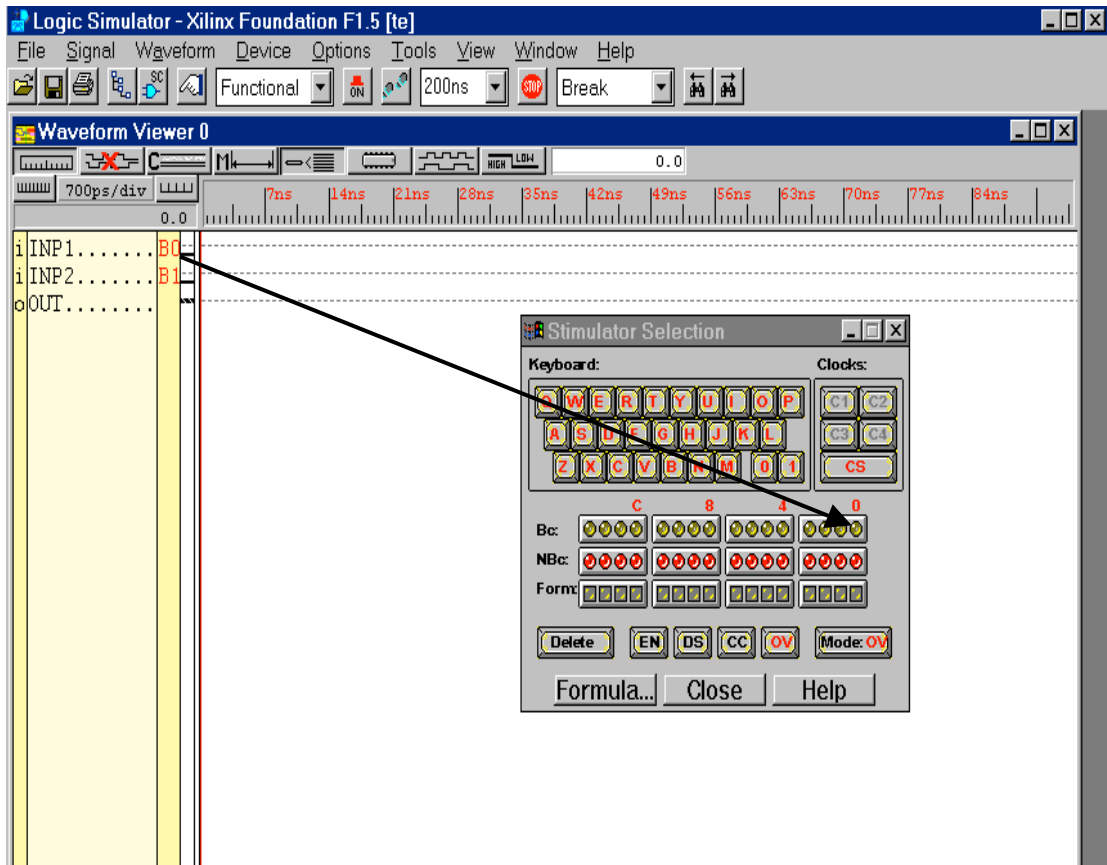


Now that the schematics is done, we need to check it for errors. First, select **Options --> Create Netlist**. This will activate a program that examines the schematic drawing and generates a machine-readable netlist which describes what type of gates are used (only one in our case) and how they are connected. Once it is done, select **Options --> Integrity test** to initiate an error check. The check should indicate there are no errors. Then save the schematics using the **File --> Save** menu item. Also, the netlist created must be exported in a format that other XILINX tools understand. First click on the **Options --> Export Netlist** and then on the **Open** button. Finally, select **File --> Exit** to close the **Schematic editor** and go back to the **Project Manager** window.

Now it is time to use the functional simulator to see if what we have entered is working correctly. In the present case it is to check the truth table for the AND gate. Start the functional **Simulator** by clicking the corresponding button in the **Project Manager** window. This brings up the **Logic Simulator** window. The first thing to do is add the inputs and outputs of the logic circuit to the **Waveform Viewer** so we can see


what is happening as the circuit is simulated. Do this by selecting the **Signal --> Add Signals..** menu item. The **Component Selection for Waveform Viewer** window will appear. Click on the one of the inputs to highlight it and then click on the **Add** button. Repeat it for the rest terminals. Then click on the **Close** button.

Now the inputs and the output are displayed but nothing happens since the two inputs are set to logic 0. We need to apply a stimulus to the circuit, so we select the **Signal --> Add Stimulators** menu item. This brings up the **Stimulator Selection** window. We are interested in the 16-bit counter labeled **Bc** (see below).

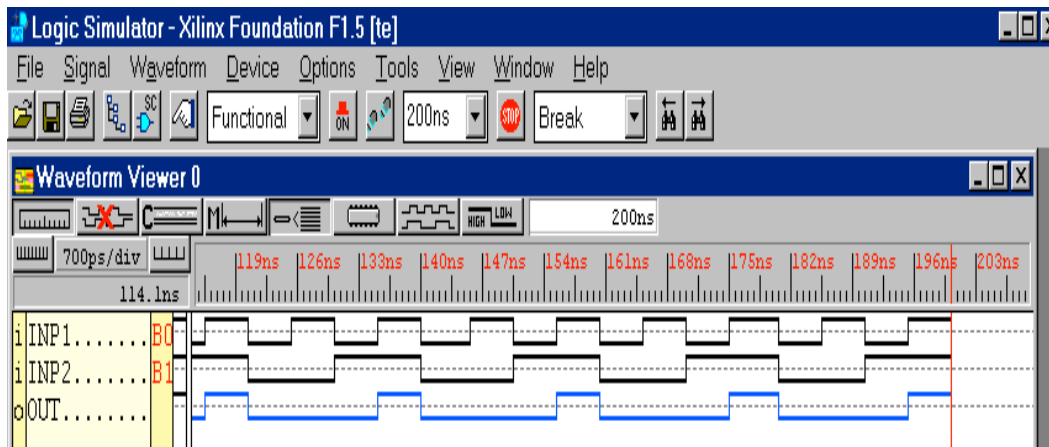


During a simulation the right-most 4 bits of this counter will go through the following sequence: 0000, 0001, 0010, 0011, 0100, 0101...etc. We can test the response of our circuit (a single AND gate) to every possible combination of inputs by attaching the two inputs of our logic circuit to two of these bits, let say B0 and B1. We do this by clicking on the name of the input in the **Waveform viewer** and then clicking on the corresponding bit-circle in the **Bc** section of the **Stimulator Selection** window. Once the two inputs are attached to the counter bits, we click **Close** to leave the **Stimulator Selection** window. Next we set up the parameters that control the speed of the simulation. **Select Options --> Preferences** and set the frequency to 50 MHz. Then click **OK**.



Now we can run the simulations. Click on the  button to initiate the functional simulation.

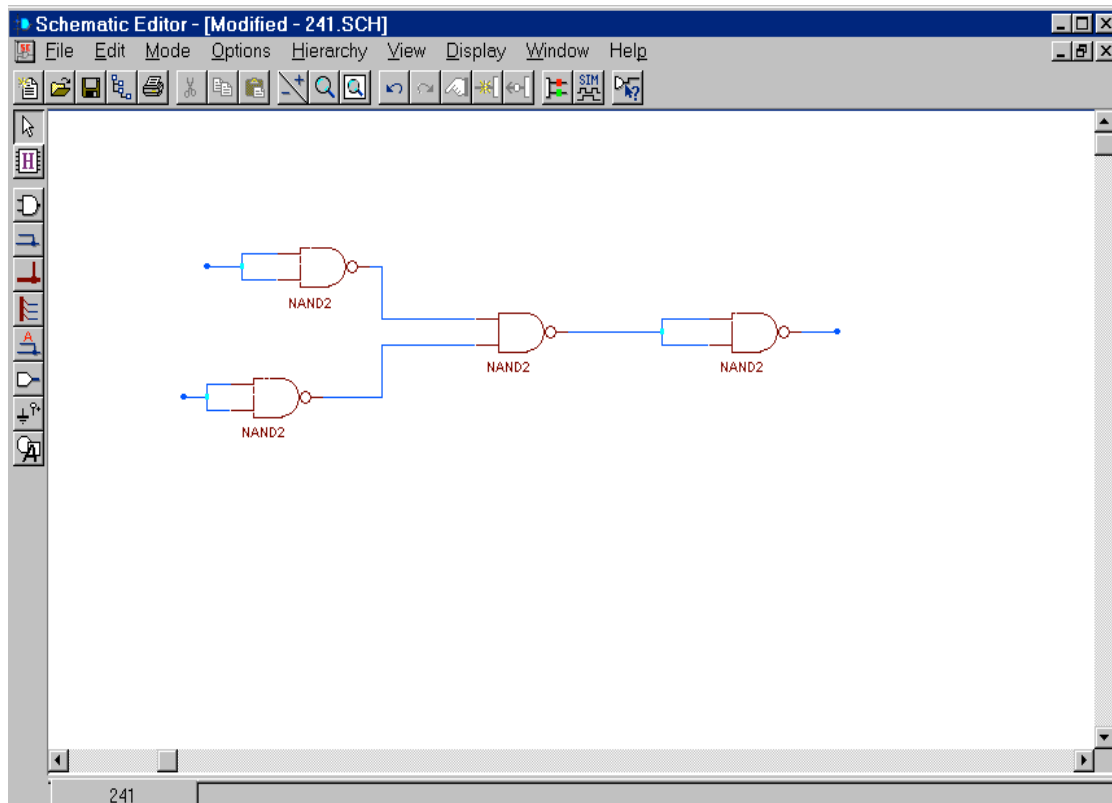
Within seconds, the results of the simulation appear as shown below.



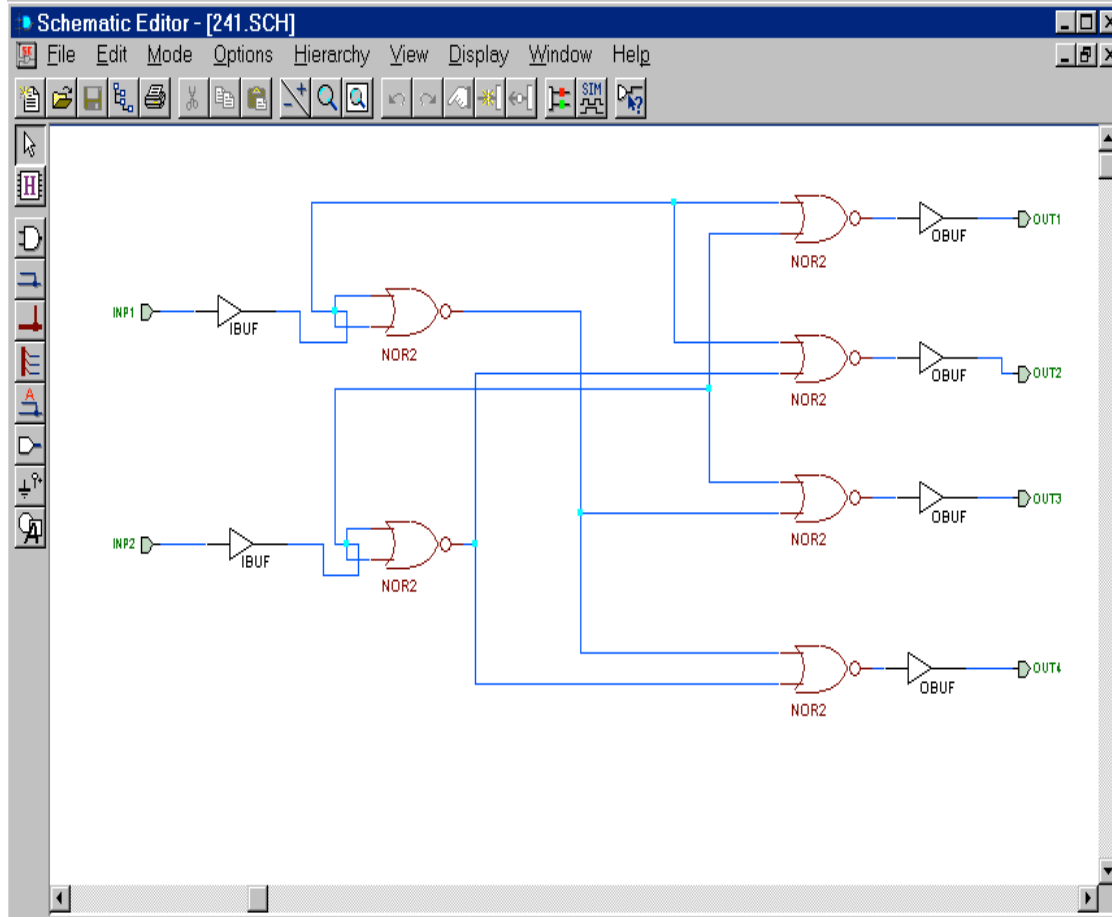
Note that the output goes high only when the two inputs are high. This exactly matches the truth table for the AND gate.

Following the procedure described above verify the truth tables for OR and NAND gates.

Problem 2. Design the circuit given below and show (by simulation) that 4 NAND gates can make a single NOR gate (recall Experiment 8, Lab 3).

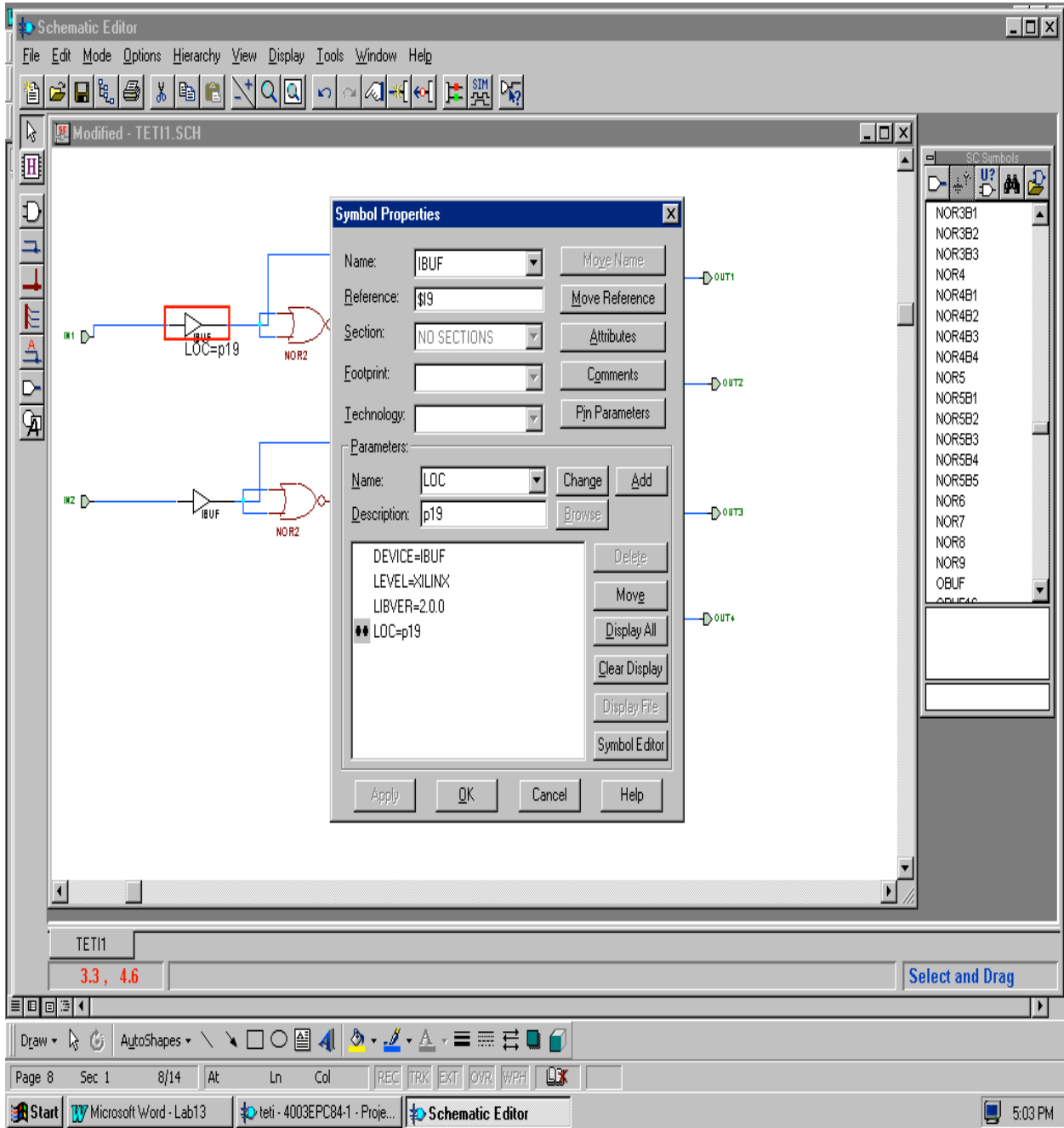


Problem 3. Design a 2-to-4 Decoder using 6 NOR gates (Recall Experiment 4, Lab 4). Compile the logic design and download it to the Xilinx FPGA (XC4003) demo board. Test the Decoder using one of the 7-segment LED on the board.

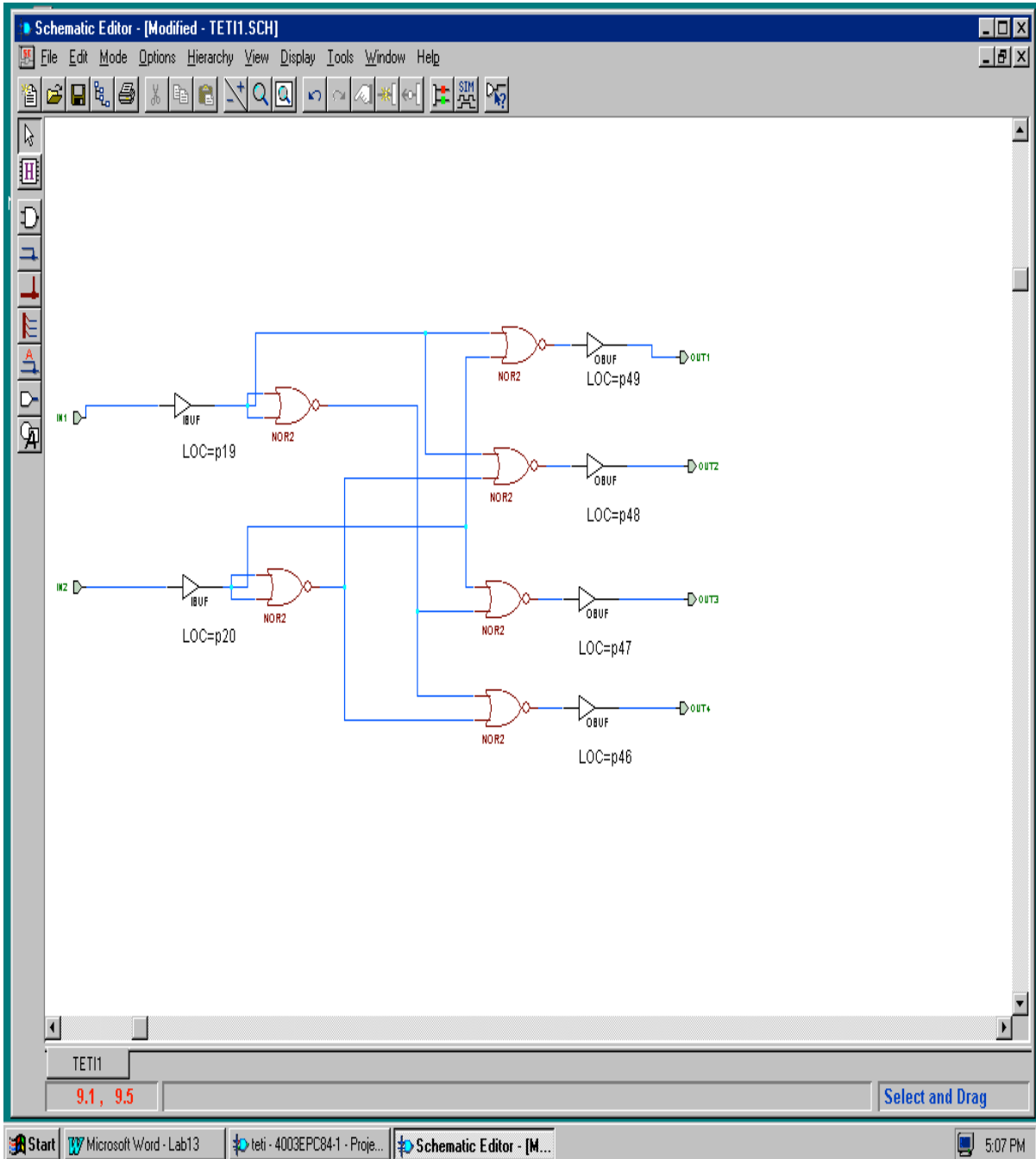


Suppose you have designed and simulated the circuit (see above) successfully. Now we can proceed with compiling the design into a bit-stream that can be loaded into the FPGA chip and tested. The test is done by applying signals to certain pins of the FPGA that correspond to the inputs of the circuit. We also have to hook an LED to the pins that carry the outputs so we can visually observe if the circuit is operating correctly.

First, we have to assign the two inputs of the circuit to two pins of the XC4003 FPGA which can be driven by a corresponding switch (labeled SW3) incorporated in the board. The pins we can use are # 19, 20 and 23 to 28. To do it, double-click on any IBUF to set its attributes. The **Symbol Properties** window will appear (see below). Click in the **Name** box and type **LOC**. Click in the **Description** box and type **p19**. Click the **Add** button. Click **OK**. Let us assign the INP1 and INP2 inputs (the buffers !) to pins 19 and 20, respectively.

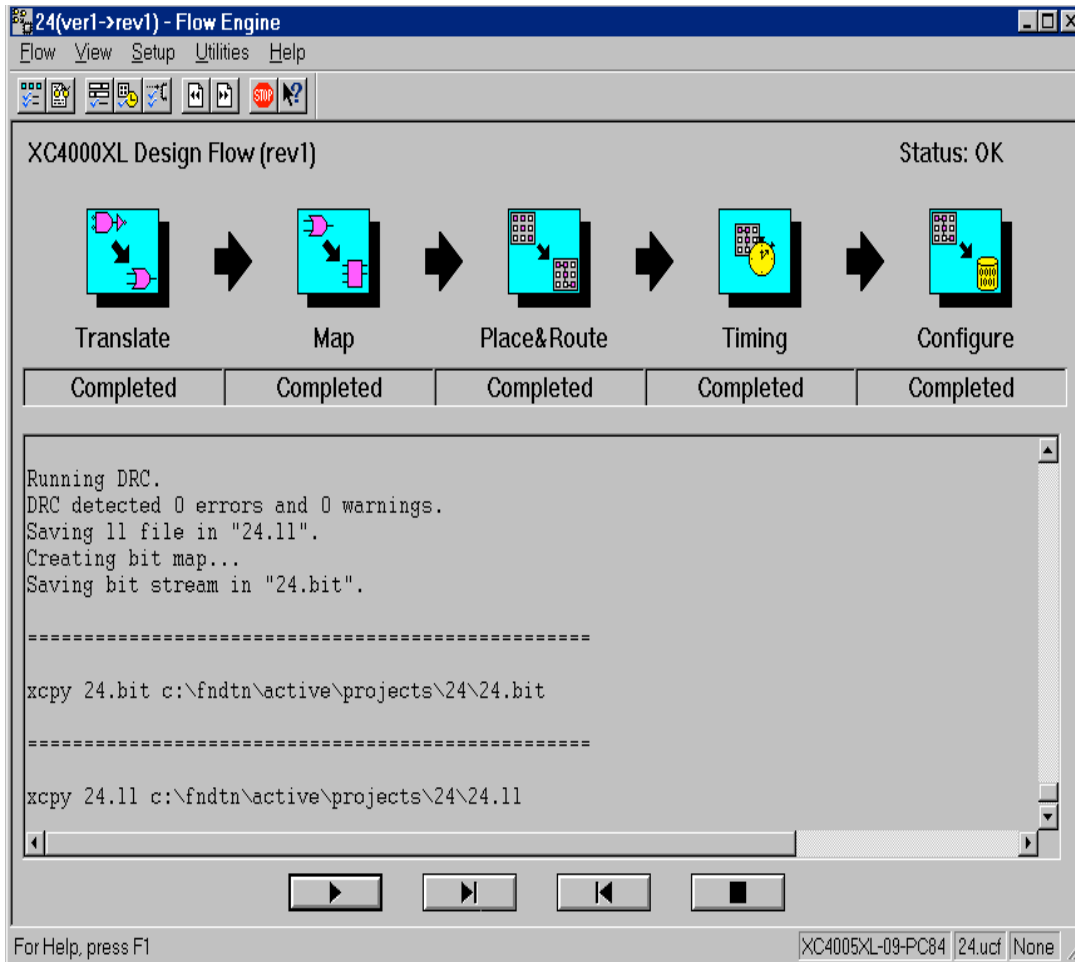


Let assign the OUT1, OUT2, OUT3 and OUT4 outputs (the buffers !) to pins 49, 48,47 and 46. Pins 49, 48, 47, 46, 45,50 and 51 drive the 7-segment LED (labeled U8) on the board. Pin 41 drives the decimal point on U8. So by associating the output of the logic circuit with pins 49 to 46 we could observe the results coming out on the LED. Hopefully you ended up with a circuit like the following one:



Once all inputs and outputs are assigned to pins do not forget to create and export a *netlist* ! Save your design and **Exit** back to the **Project Manager**.

Now it is time to compile your design. Do it by selecting the **Implementation menu** item. When it appears Click on **Run**. The **Flow engine** (see below) will show up. The compilation goes through the following steps: (they do not require any actions of yours)



Translate: Converts the *netlist* to an appropriate internal format.

Map: Optimizes the logic circuit.

Place&Route: The gates in the *netlist* are assigned to particular programmable switch matrices in the FPGA.

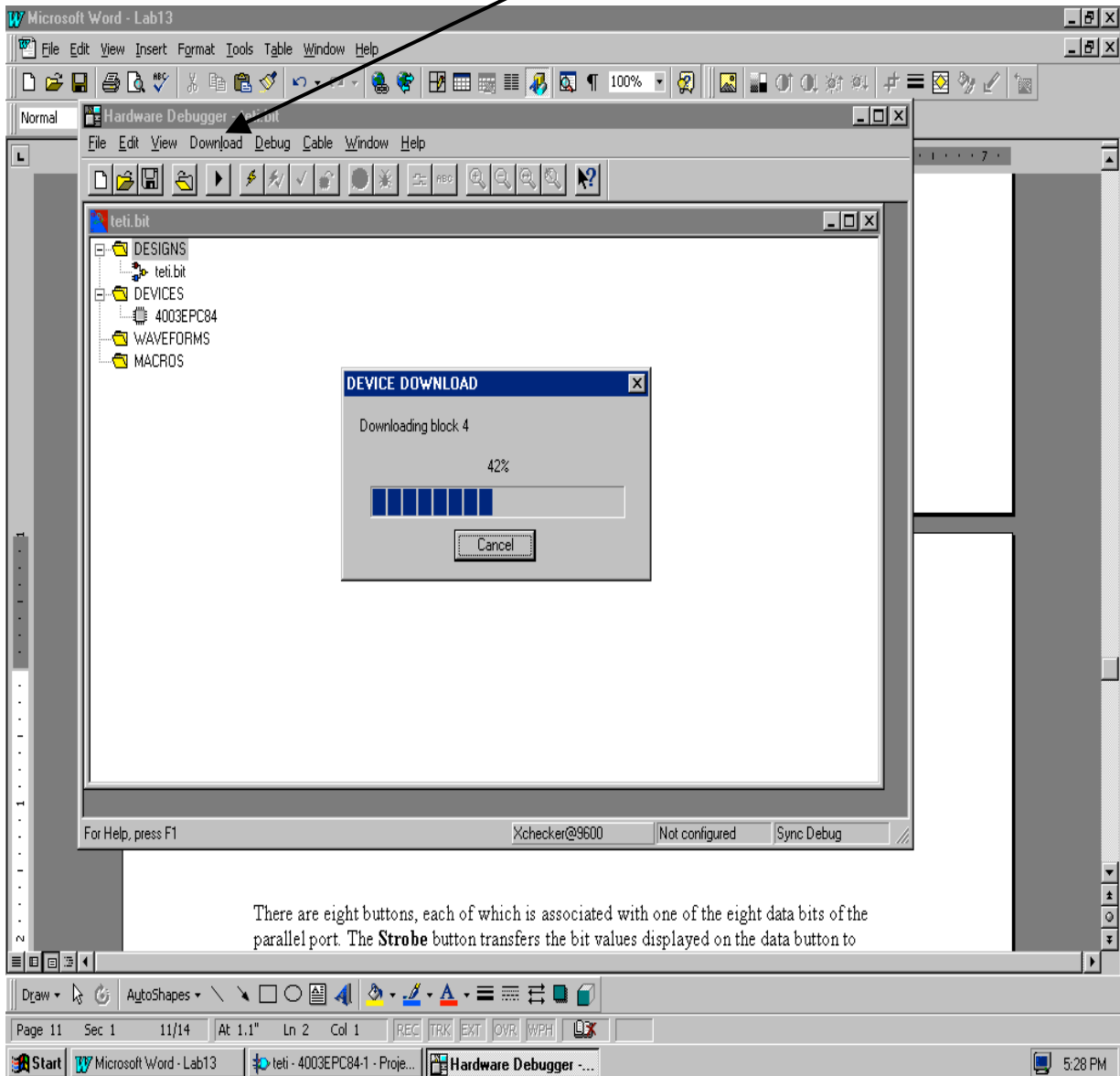
Timing: Computes the delay of the signal while it propagates through the circuit.

Configure: Stream of bits is generated which can be downloaded into the FPGA chip to configure it to carry out the logic functions described in the schematic file.

The next step is to download the bit-stream file to the demo board and program the FPGA. Go back to the **Project Manager** window and click on the **Programming** button. Select the **Hardware debugger** program. It is a graphical interface that allows you to download a design to a FPGA, verify the downloaded configuration, and display the

internal states of the programmed FPGA. The bit-stream file is downloaded to the board through a serial port cable (XChecker cable).

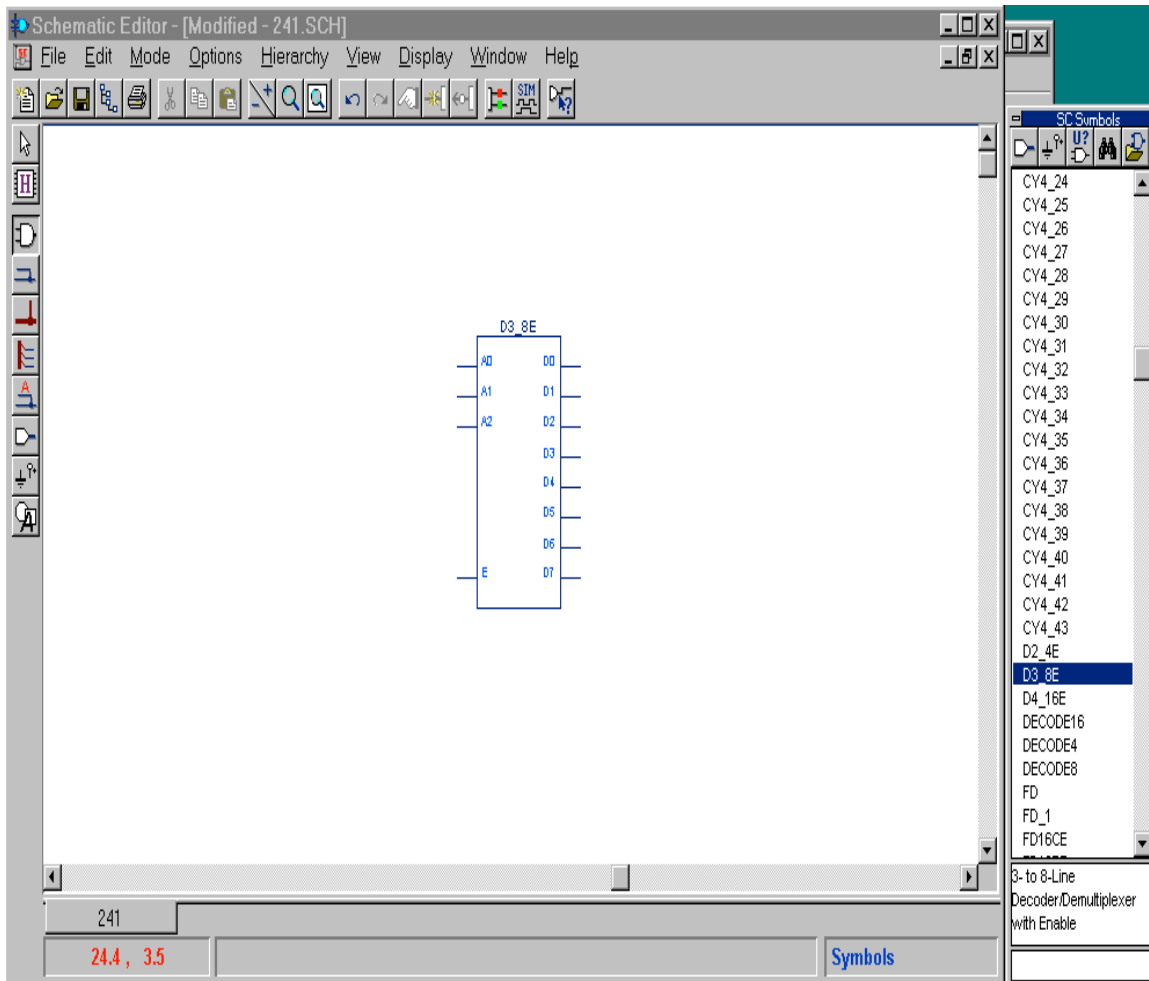
Once the **Hardware debugger** starts select **Download/Download design** menu item.



When the downloading is finished, **Exit** the program.

Exercise the functions of your design by providing logic inputs to the FPGA just programmed. Use the SW3 switch(es) on the board to set the associated input pin(s) of FPGA to a logic “1” or “0”. Close (“1”)/open (“0”) the corresponding switch(es) and examine the response of LED. Does your circuit function properly? Please, note that each LED segment is turned on by driving the corresponding FPGA output pin “LOW” with a logic “0”.

Problem 4. Design a 3-to-8 Decoder. (Recall Experiment 5, Lab 4). Compile the logic design and download it to the demo board. Test the decoder using the 7-segment LED on the board. (Hint: use the D3_8E component available in the SC library)



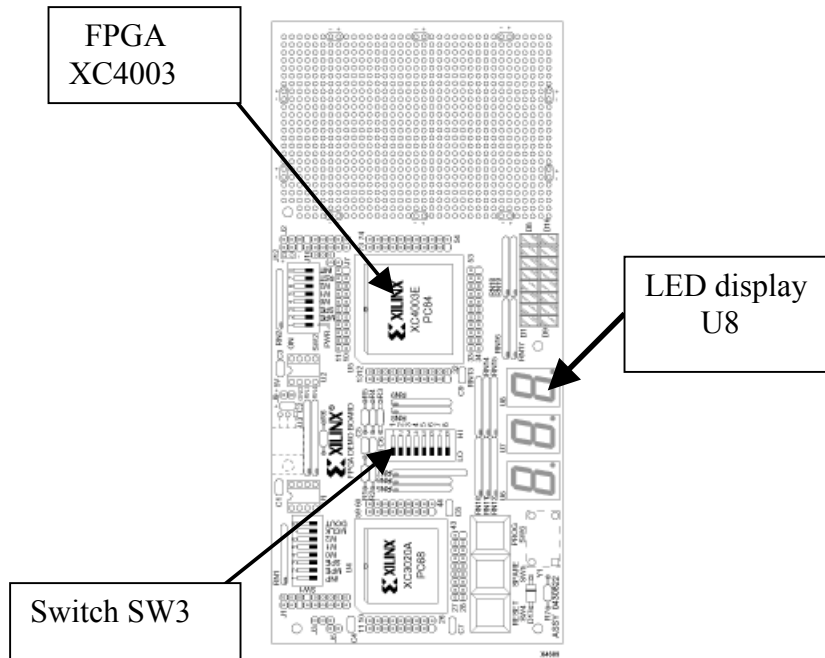


Figure 3-2 FPGA Demonstration Board

