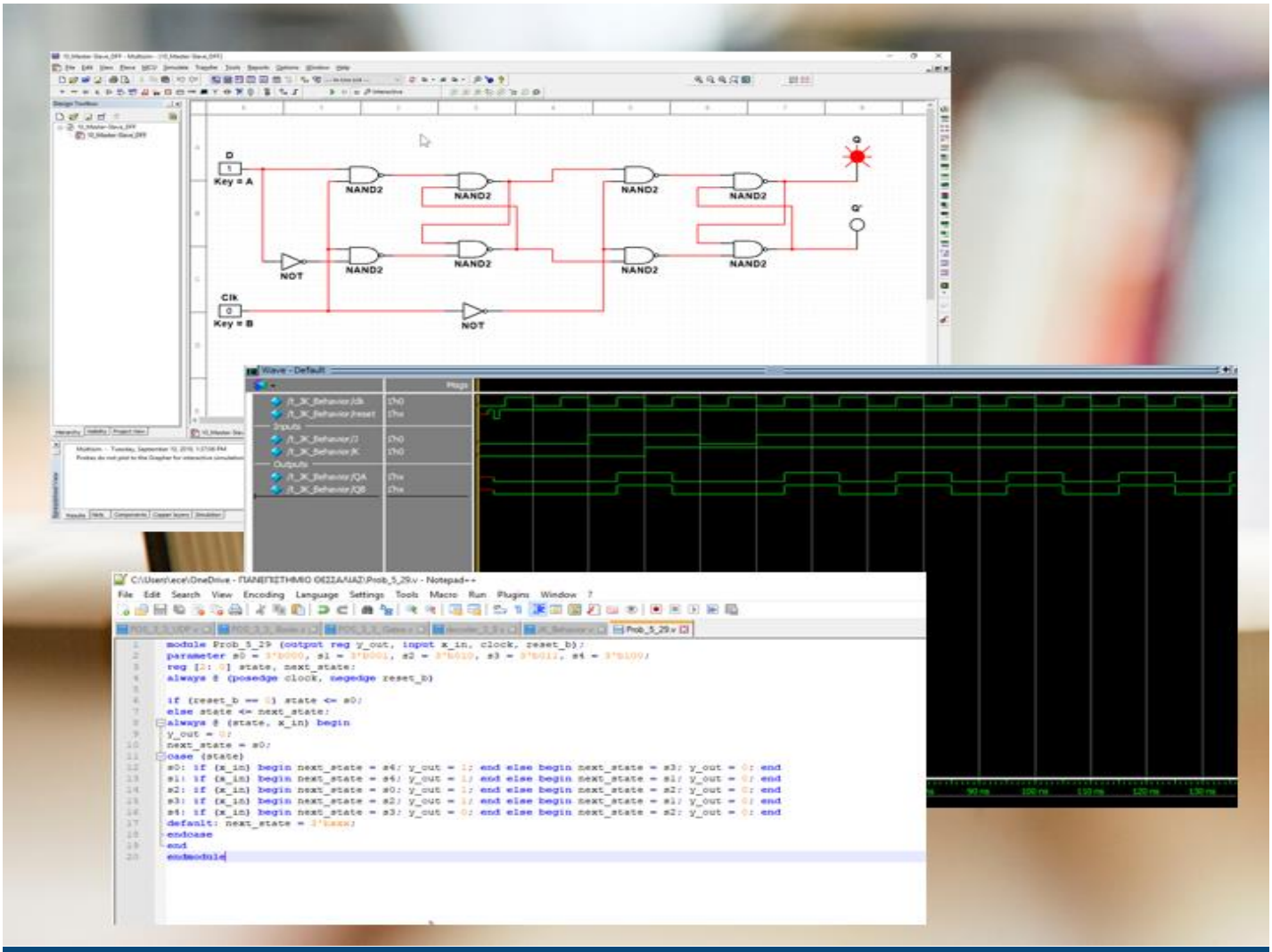
ΠΑΝΕΠΙΣΤΗΜΙΟ ΘΕΣΣΑΛΙΑΣ

ΤΜΗΜΑ ΗΛΕΚΤΡΟΛΟΓΩΝ ΜΗΧΑΝΙΚΩΝ & ΜΗΧΑΝΙΚΩΝ ΥΠΟΛΟΓΙΣΤΩΝ



ECE119 Ψηφιακή Σχεδίαση

Εργαστηριακές ασκήσεις, Multisim - Verilog

Lab 1: Multisim Circuit Simulation and Basic Gates

Καραμπερόπουλος Δημήτρης

Σεπτέμβριος 2026

Lab 1

Required Tools and Technology 1
Lab 1: Multisim Circuit Simulation and Basic Gates 2
Learning Objectives 2
Expected Deliverables 2
1.1 Theory and Background 3
1.2 Simulate: Circuits in Multisim (AND Gate) 4
1.3 Exercise: Creating a New Multisim Circuit (OR Gate) 8
1.4 Exercise: Βασικά ολοκληρωμένα κυκλώματα. 13

Required Tools and Technology

Software: NI Multisim 14.0 or newer

- ✓ **Install Multisim:** http://www.ni.com/gate/gb/GB_ACADEMICEVALMULTISIM/US
- ✓ **View Help:** http://www.ni.com/multisim/technical-resources/

Lab 1: Multisim Circuit Simulation and Basic Gates

In this lab, you will learn to use simulation software to simulate the behavior of digital circuits. Using the *Multisim* environment, you will be able to simulate basic and advanced circuits to learn fundamental concepts of digital electronics without needing physical components such as wires, probes, and chips. This offers many advantages to the student, such as being able to experiment with circuits and components before needing to build the circuit.

Learning Objectives

In this lab, students will:

1. Open pre-configured circuits in Multisim.
2. Configure and build circuits in Multisim.
3. Simulate simple digital circuits.

Expected Deliverables

In this lab, you will collect the following deliverables:

- Analysis of gate behavior
- Comments on the inputs of LEDs
- Conclusion questions
- Truth Tables of the gates
- Saved circuit with Multisim
- Exercise with gates in Multisim

Your instructor may expect you to complete a lab report. Refer to your instructor for specific requirements or templates.

1.1 Theory and Background

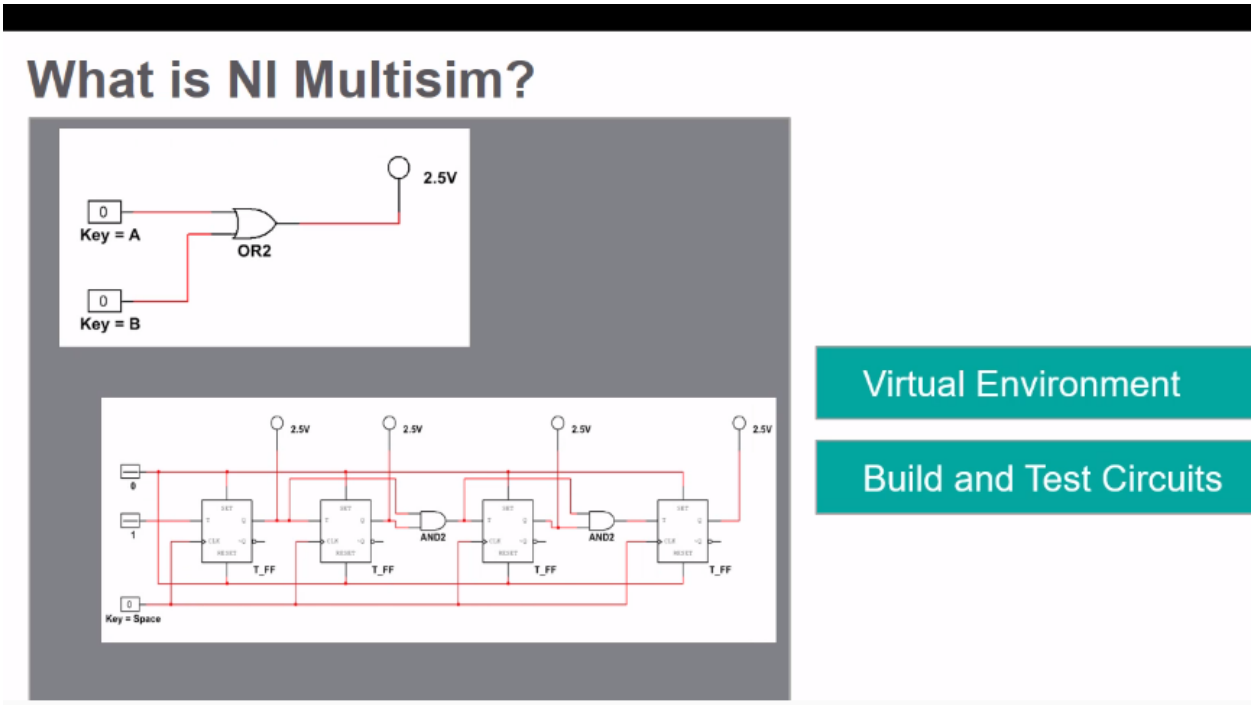


Figure 1-1 Video Screenshot. View the video here: https://youtu.be/hZL5O4Z8v7Q

Video Summary

- NI Multisim will be used to build circuits and observe their behavior
- Multisim is a virtual environment which allows you to build and test circuits without needing physical components
- Circuits that have been built and tested can be deployed to hardware to be observed

1.2 Simulate: Circuits in Multisim (AND Gate)

Circuit

Open the circuit file: https://cf-ts.mythinkscape.com/ckeditor/AND_Gate_2.ms14
(You can also find it on Eclass)

- Note: the file name is **AND_Gate.ms14**
- Select **File>>Open,** and then browse to the location where the files for this lab have been saved.
- This circuit takes two digital inputs, wires them into an AND gate, and wires the output to a probe. Using Multisim, we can explore the behavior of this circuit.

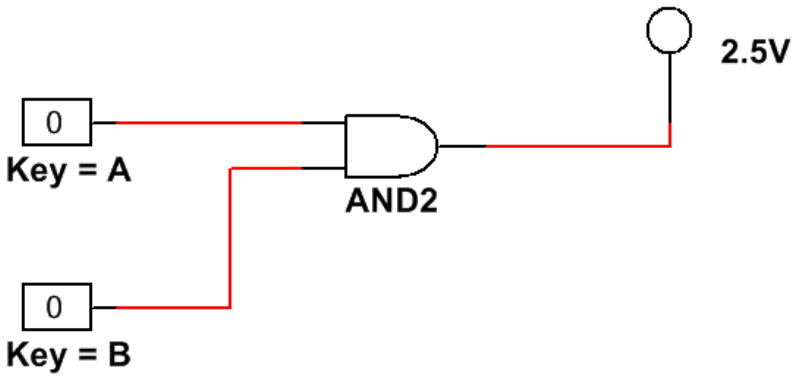


Figure 1-2 AND gate circuit

Testing an AND Gate

- Click the **Run** button to begin simulating the circuit.

Video tutorial on how to run and stop a simulation of a circuit:

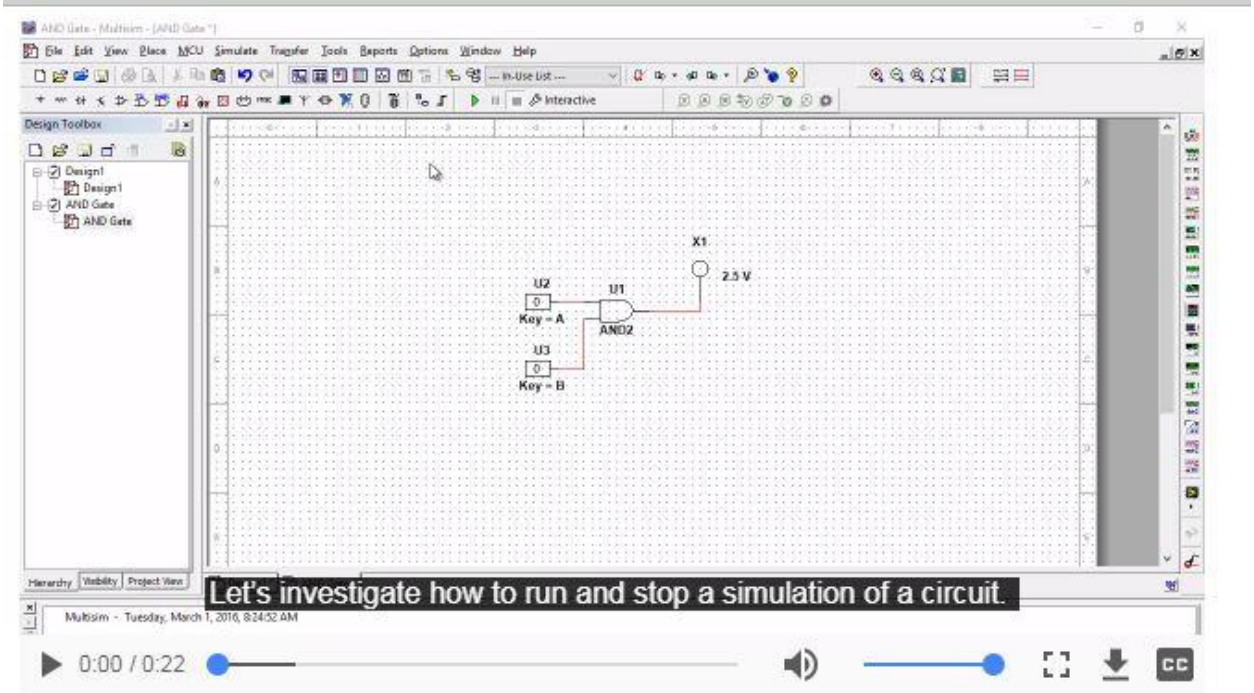


Figure 1-3 Video Tutorial: https://cf-ts.mythinkscape.com/video/002_Run_and_Stop.mp4

- Press the '**A**' key on the keyboard to change the value of that input to **1.**

Video tutorial on how to change input values:

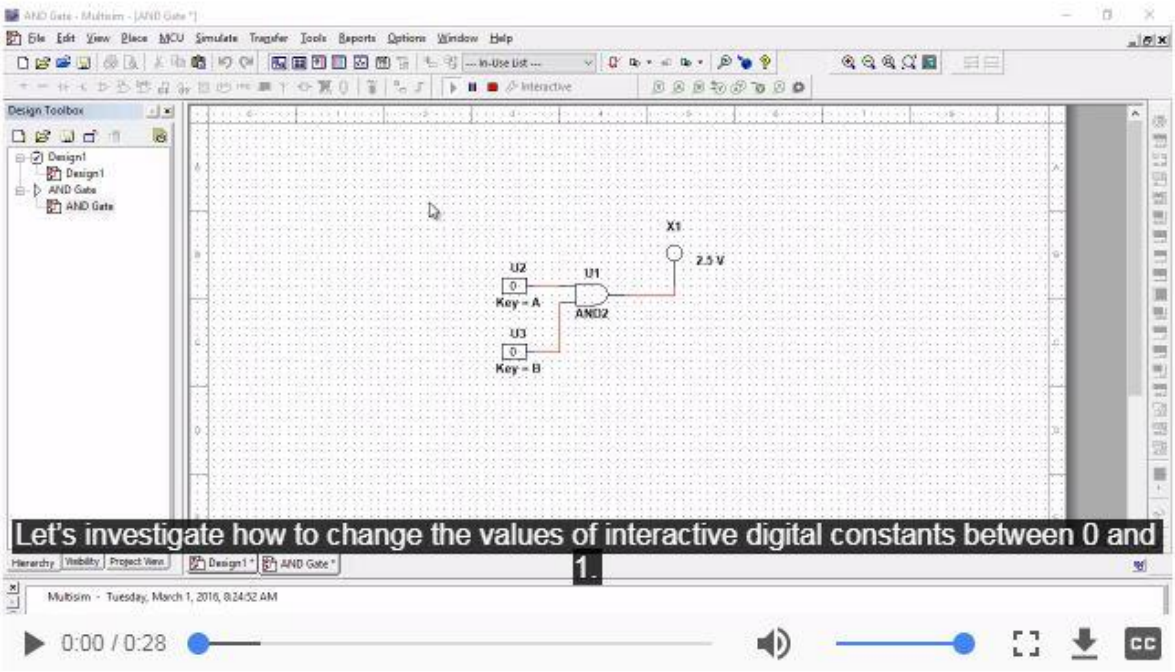


Figure 1-4 Video Tutorial: https://cf-ts.mythinkscape.com/003__Changing_the_Input_Values_2.mp4

Check Your Understanding

Note: The following questions are meant to help you self-assess your understanding so far. You can view the answer key for all "Check your Understanding" questions at the end of the lab.

- Notice that: U2 = **1,** U3 = **0**.

1-1 Does the probe turn on?

A. Yes
B. No

AND Gate Testing

- Press the '**A**' key again to change the top input back to **0**.

- Press the '**B**' key to change the second input to **1**.

1-2 Does the probe turn on?

A. Yes
B. No

- Press the '**A**' key, so that both inputs are equal to **1**.

1-3 Does the probe turn on?

A. Yes
B. No

1-4 How would you describe the behavior of this gate?

_____

_____

_____

_____

- When you're done, stop the simulation by clicking the **Stop** button.



Figure 1-5 Stop button

1.3 Exercise: Creating a New Multisim Circuit (OR Gate)

Create a Circuit

Now create a similar circuit for yourself, following the instructions below.

- Select **File>> New**
- In the menu that appears, select **Blank** and click **Create**.

Placing an OR Gate

- Click the **Place Misc Digital** button along the top bar.
- In the window that appears, select **TIL** from the Family and scroll down to find **OR2** in the Component box.
- Click **OK** and place the component in the middle of the diagram.

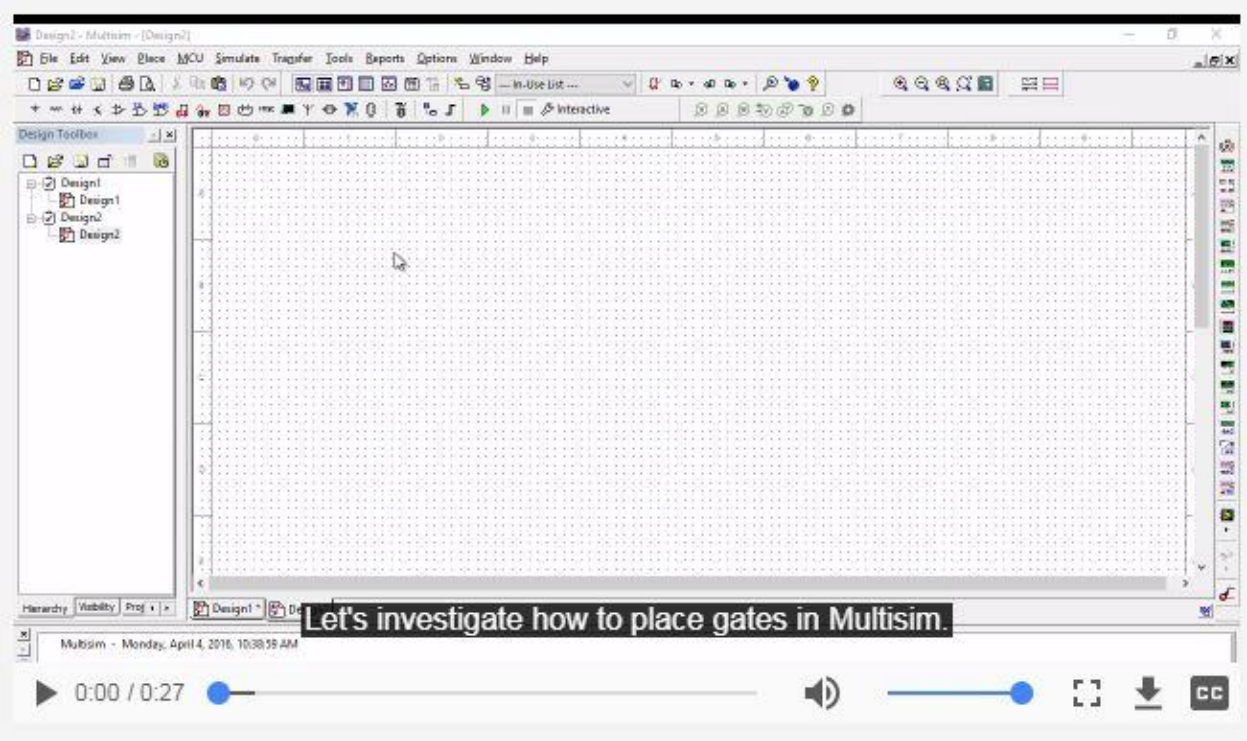


Figure 1-6 Placing gates in Multisim: https://cf-ts.mythinkscape.com/052_Placing_an_OR_Gate_2.mp4

- The window will temporarily go away and then come back. When it comes back, select **Sources** from the Group drop-down, then select the **DIGITAL_SOURCES** family, and the **INTERACTIVE_DIGITAL_CONSTANT** component.
- Place two **INTERACTIVE_DIGITAL_CONSTANT** to the left of the OR gate.
- Select Indicators in the Group drop-down, then select the **PROBE** family, and the **PROBE_DIG_RED** component.
- Place this to the right of the OR gate.
- Click the **Close** button when the window reappears.

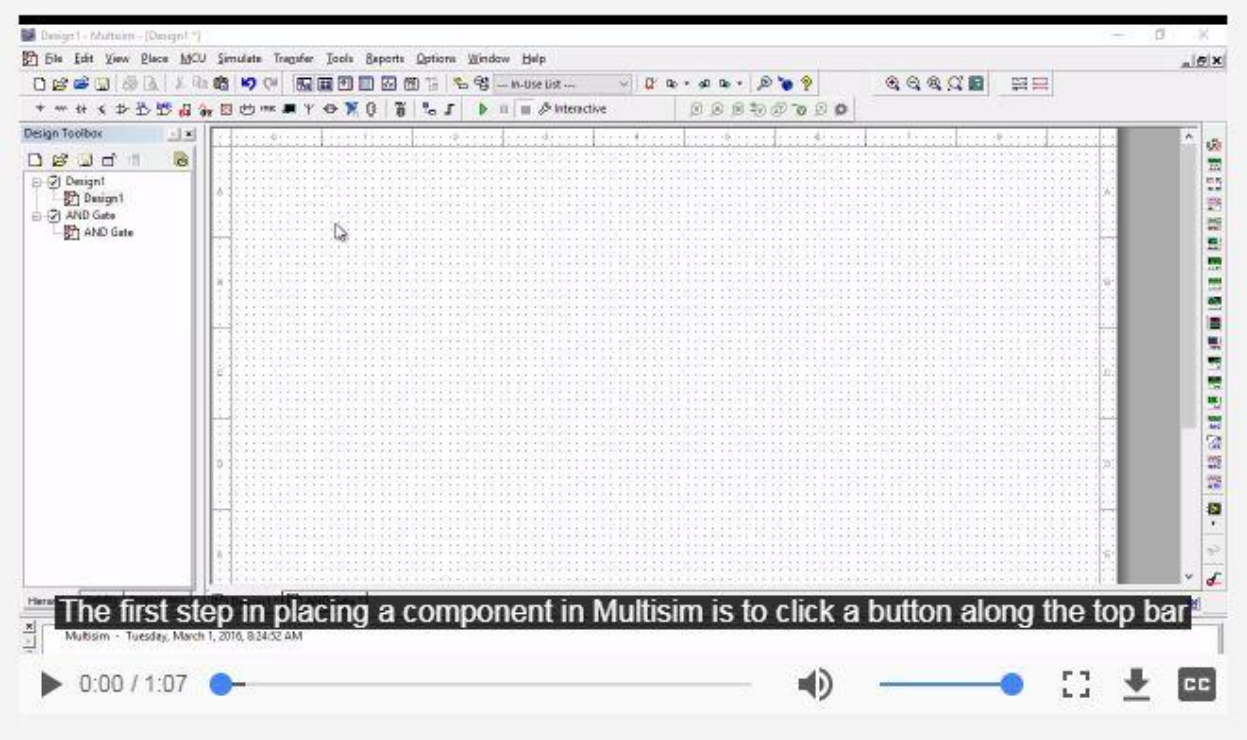


Figure 1-7 Placing components in Multisim:
https://cf-ts.mythinkscape.com/006_Placing_a_Gate_and_Circuit_Components_General_2.mp4

Wiring a Circuit

Note: To wire the components together, click the end of the wire for one component and then for another component. This will create a wire between them.

- Wire the two Digital Constants into the OR gate and then wire the OR gate into the Probe.

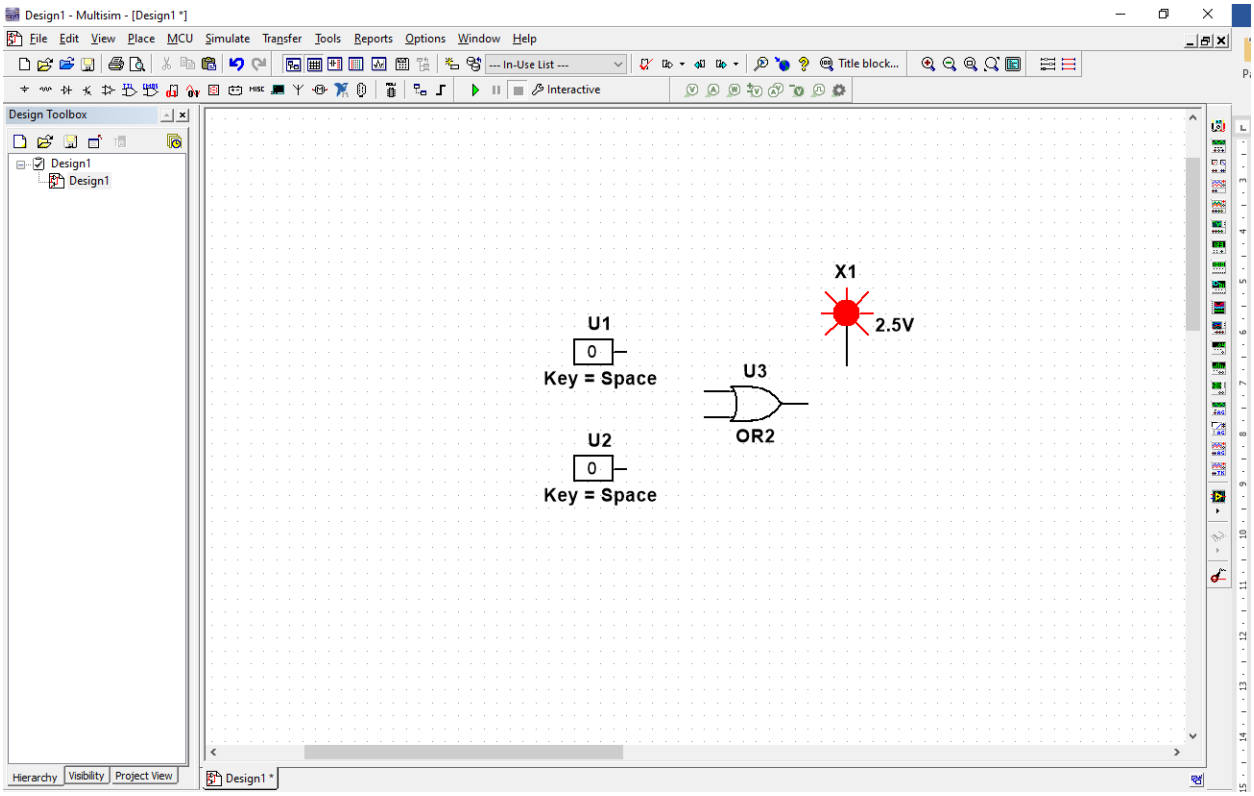


Figure 1-8 Wiring a circuit in Multisim: https://cf-ts.mythinkscape.com/007_Wiring_a_Circuit_3.mp4

Configuring Digital Constants

Notice that the two constants both use the Space key to change their value. To configure components in the diagram, double-click them to change their settings.

- Double-click the top **Digital Constant.**
- In the window that appears, select '**A**' from the Key for toggle dropdown.
- Change the second constant to toggle with the '**B**' key.

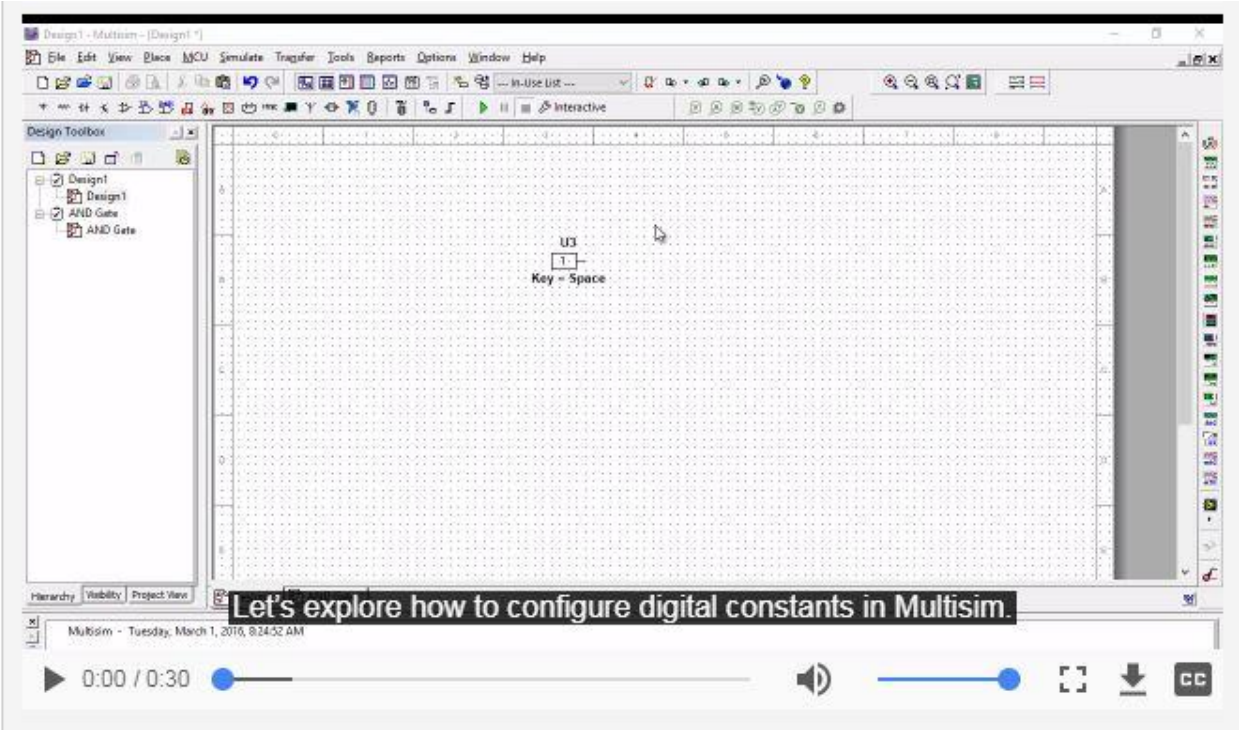


Figure 1-9 Configuring digital constants in Multisim:
https://cf-ts.mythinkscape.com/008_Configuring_Digital_Constants_2.mp4

Testing an OR Gate

- Click the **Run** button to begin simulating the circuit.



Figure 1-10 Run button

- Press the '**A**' key on the keyboard to change the value of that input to **1**.

1-5 Does the probe turn on?

A. Yes
B. No

- Press the '**A**' key again to change the top input back to **0**.
- Press the '**B**' key to change the second input to **1**.

1-6 Does the probe turn on?

A. Yes
B. No

- Press the '**A**' key, so that both inputs are equal to **1**.

1-7 Does the probe turn on?

A. Yes
B. No

1-8 How would you describe the behavior of this gate?

_____

_____

_____

_____

- When you're done, stop the simulation by clicking the **Stop** button.



Figure 1-11 Stop button

1.4 Exercise: Βασικά ολοκληρωμένα κυκλώματα.

Χρησιμοποιείστε δύο εισόδους για όλες τις πύλες που θα χρειαστείτε. Συνδέστε την μία από τις εισόδους με *ένα ρολόι παραγωγής τετραγωνικών παλμών* (Clock) το οποίο παράγει τετραγωνικούς παλμούς συχνότητας *1 Hz* και την άλλη είσοδο με έναν διακόπτη (INTERACTIVE_DIGITAL_CONSTANT) ικανό να δώσει τις τιμές 1 και 0.

Με χρήση *λαμπτήρων/λυχνιών* (PROBE) παρατηρήστε τον παλμό και την έξοδο κάθε πύλης.

Γράψτε τους πίνακες αληθείας για την περίπτωση που η μία είσοδος είναι συνδεδεμένη στον παλμό και η άλλη στο λογικό ‘0’ ή στο λογικό ‘1’.

Οι πύλες που καλείστε να αναγνωρίσετε και να εξετάσετε είναι οι πύλες 7408 και 7432. Θα τις βρείτε: Master Database / TTL / 74LS / 74LS08N και αντίστοιχα 74LS32N.

Το clock θα το βρείτε: Master Database / Sources / DIGITAL_SOURCES / DIGITAL_CLOCK

Η προσομοίωση να γίνει με το Multisim.



- Multisim: Όνομα αρχείου “**1_Basics.ms14**”.
Προσθήκη στο zip file με όνομα ”Lab01_ονοματεπώνυμο_ΑΜ.zip”