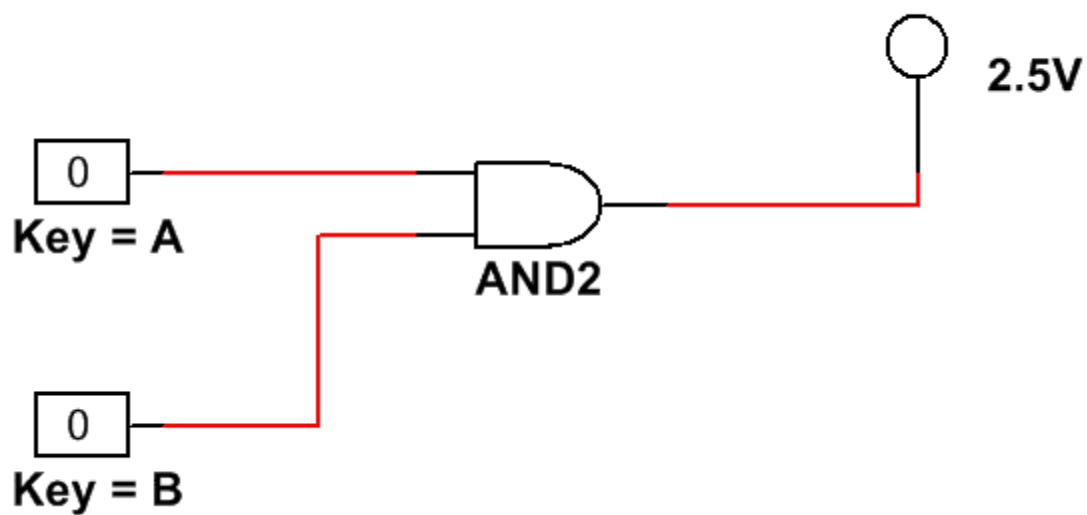


Lab Manual:

Digital Electronics

Using the Digilent Digital Electronics Board for NI ELVIS III



Lab 1: Multisim Circuit Simulation

© 2018 National Instruments

All rights reserved. Neither this resource, nor any portion of it, may be copied or reproduced in any form or by any means without written permission of the publisher.

National Instruments respects the intellectual property of others, and we ask our readers to do the same. This resource is protected by copyright and other intellectual property laws. Where the software referred to in this resource may be used to reproduce software or other materials belonging to others, you should use such software only to reproduce materials that you may reproduce in accordance with the terms of any applicable license or other legal restriction.

LabVIEW and National Instruments are trademarks of National Instruments.

All other trademarks or product names are the property of their respective owners.

Additional Disclaimers: The reader assumes all risk of use of this resource and of all information, theories, and programs contained or described in it. This resource may contain technical inaccuracies, typographical errors, other errors and omissions, and out-of-date information. Neither the author nor the publisher assumes any responsibility or liability for any errors or omissions of any kind, to update any information, or for any infringement of any patent or other intellectual property right.

Neither the author nor the publisher makes any warranties of any kind, including without limitation any warranty as to the sufficiency of the resource or of any information, theories, or programs contained or described in it, and any warranty that use of any information, theories, or programs contained or described in the resource will not infringe any patent or other intellectual property right. THIS RESOURCE IS PROVIDED "AS IS." ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING, BUT NOT LIMITED TO, ANY AND ALL IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF INTELLECTUAL PROPERTY RIGHTS, ARE DISCLAIMED.

No right or license is granted by publisher or author under any patent or other intellectual property right, expressly, or by implication or estoppel.

IN NO EVENT SHALL THE PUBLISHER OR THE AUTHOR BE LIABLE FOR ANY DIRECT, INDIRECT, SPECIAL, INCIDENTAL, COVER, ECONOMIC, OR CONSEQUENTIAL DAMAGES ARISING OUT OF THIS RESOURCE OR ANY INFORMATION, THEORIES, OR PROGRAMS CONTAINED OR DESCRIBED IN IT, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGES, AND EVEN IF CAUSED OR CONTRIBUTED TO BY THE NEGLIGENCE OF THE PUBLISHER, THE AUTHOR, OR OTHERS. Applicable law may not allow the exclusion or limitation of incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Lab 1: Multisim Circuit Simulation

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.

Additionally, you will learn to create digital circuits and deploy them to the *NI Digital Systems Development Board*. In a later lesson, students will explore more about *Programmable Logic Devices (PLDs)* and a specific type of PLD called a Filed-Programmable Gate Array, or FPGA. FPGAs are an interesting technology that allow circuit designers to build custom circuits on a reconfigurable piece of hardware. When a circuit design needs to change, the designer can change the circuit in hardware and then simply redeploy it to the FPGA instead of soldering a new circuit.

In this lab series, you will use the FPGA on the Digital Electronics Board to deploy the circuits that you design in Multisim onto the board, allowing you to implement custom circuits which use the peripherals on the board.

Learning Objectives

In this lab, students will:

1. Open pre-configured circuits and PLD designs in Multisim.
2. Configure and build circuits and PLD designs in Multisim.
3. Simulate simple digital circuits.
4. Deploy PLD designs to the FPGA.

Required Tools and Technology

Platform: NI ELVIS III

- ✓ View User Manual:
<http://www.ni.com/en-us/support/model.ni-elvis-iii.html>
- ✓ View Tutorials:
https://www.youtube.com/playlist?list=PLvcPluVaUMIWm8ziaSxv0gwtshBA2dh_M

Hardware: Digilent Digital Electronics Board for NI ELVIS III

- ✓ View NI Digital Electronics Board Manual:
<http://www.ni.com/pdf/manuals/376627b.pdf>

Software: NI Multisim 14.0.1 Education Version or newer

- ✓ Install Multisim:
http://www.ni.com/gate/gb/GB_ACADEMICEVALMULTISIM_US
- ✓ View Help:
<http://www.ni.com/multisim/technical-resources/>

Software: NI LabVIEW FPGA Vivado 2014.4

- ✓ Install: <http://www.ni.com/download/labview-fpga-module-2015-sp1/5920/en/>

Note: Digilent Driver (The installer above automatically downloads the installer below onto your computer)

- ✓ Navigate to:
C:\NIFPGA\programs\Vivado2014_4\data\xicom\cable_drivers\nt64\digilent
- ✓ Install: install_digilent.exe

Expected Deliverables

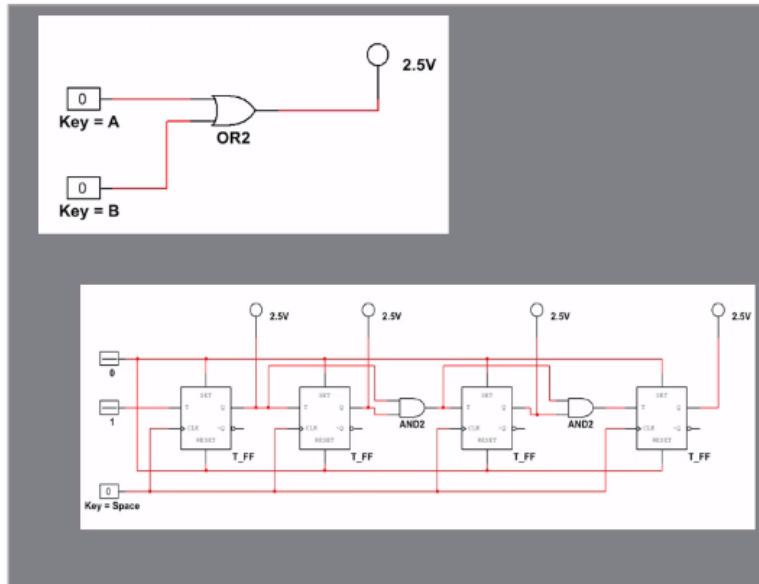
In this lab, you will collect the following deliverables:

- Analysis of gate behavior
- Comments on the inputs of LEDs
- Conclusion questions

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

1.1 Theory and Background

What is NI Multisim?



Virtual Environment

Build and Test Circuits

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

Circuit

Open the circuit file: https://cf-ts.mythinkscape.com/ckeditor/AND_Gate_2.ms14 .

- 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.

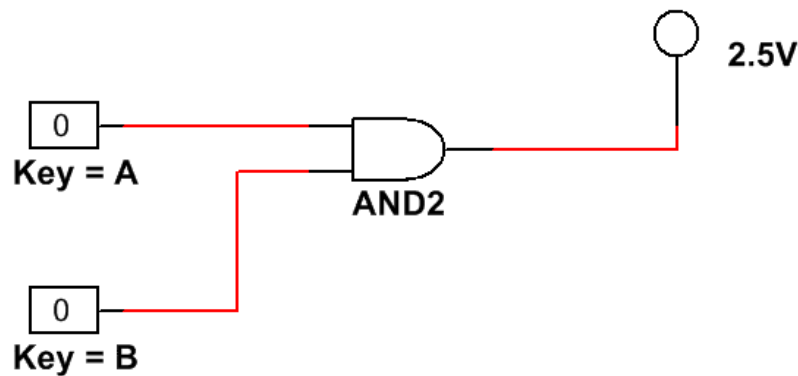


Figure 1-2 AND gate circuit

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:

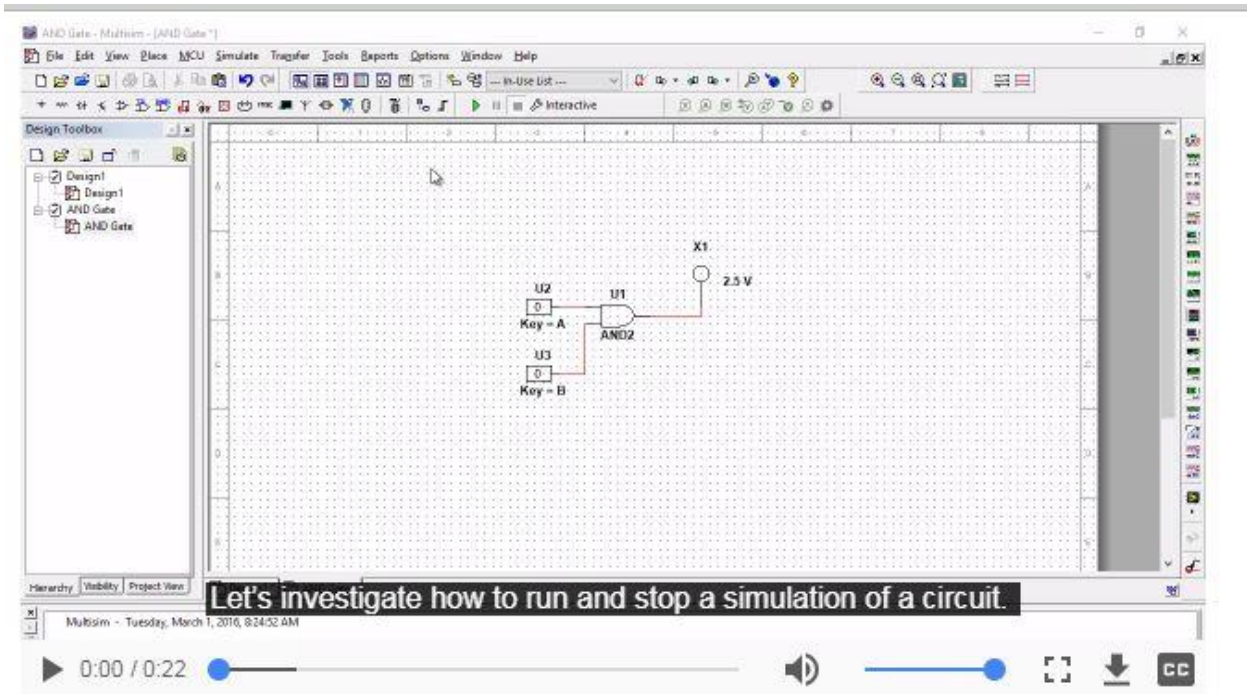


Figure 1-3 Video Tutorial: https://cf-ts.mythinkscape.com/video/002_Run_and_Stop.mp4

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

Video tutorial on how to change input values:

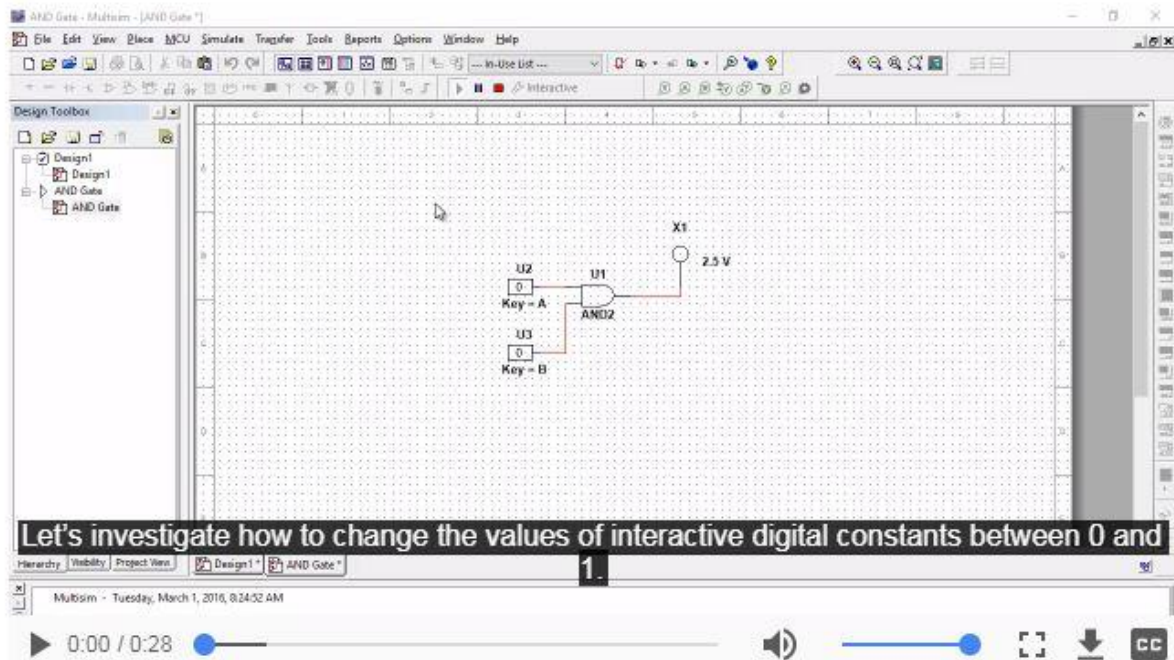


Figure 1-4 Video Tutorial: https://cf-ts.mythinkscape.com/003_Changing_the_Input_Values_2.mp4



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.

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

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.

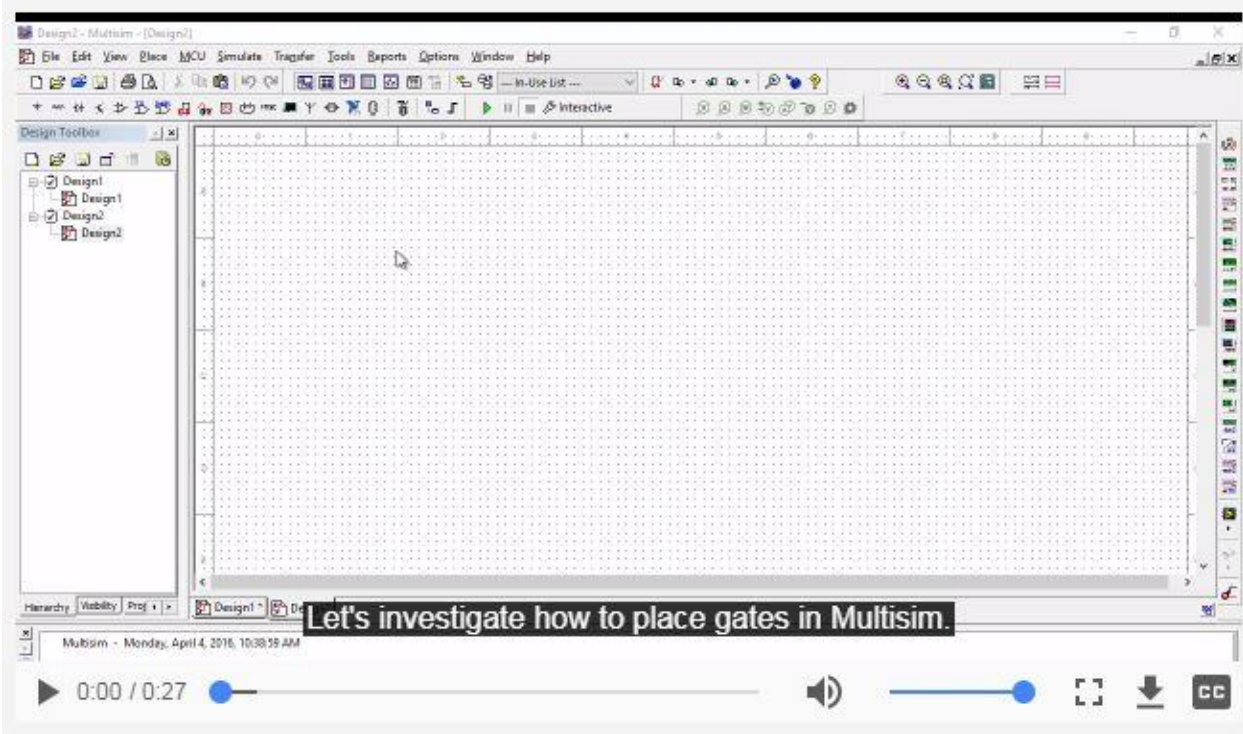


Figure 1-6 Placing gates in Multisim: https://cf-ts.mythinkscape.com/052_Placing_an_OR_Gate_2.mp4

Placing Circuit Components

- 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.

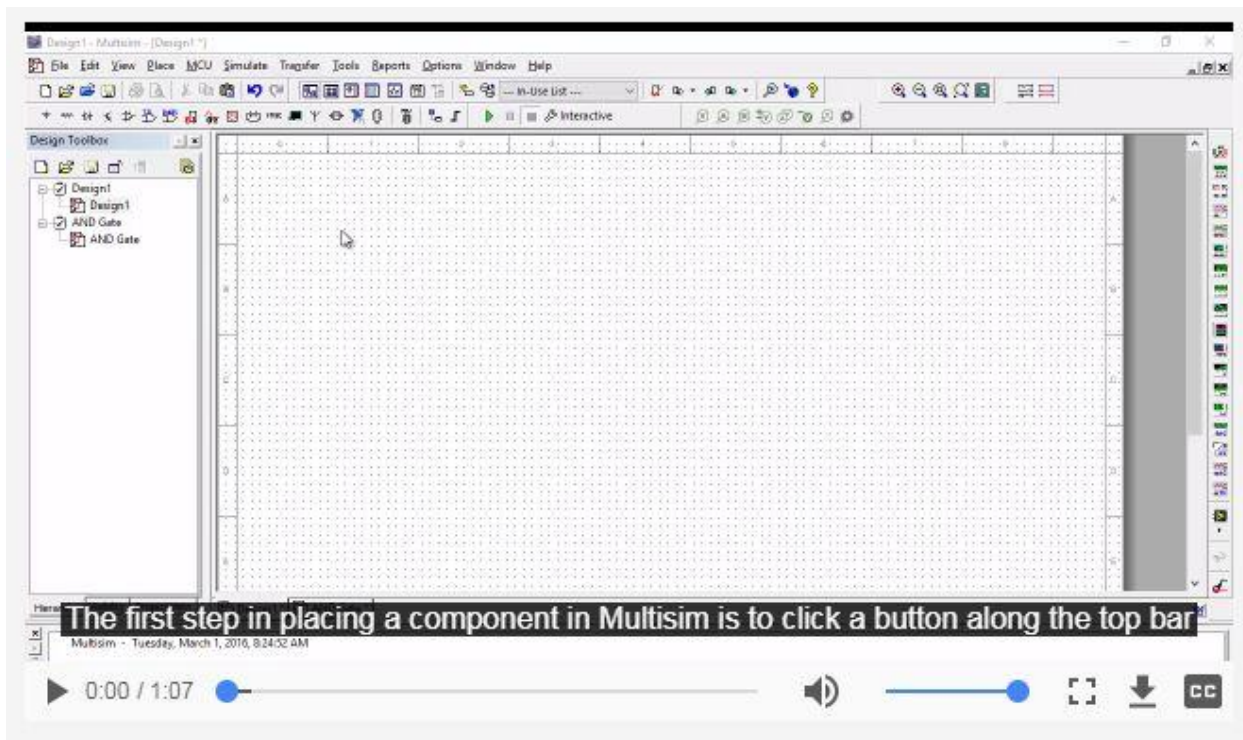


Figure 1-7 Placing components in Multisim:

<https://cfts.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.

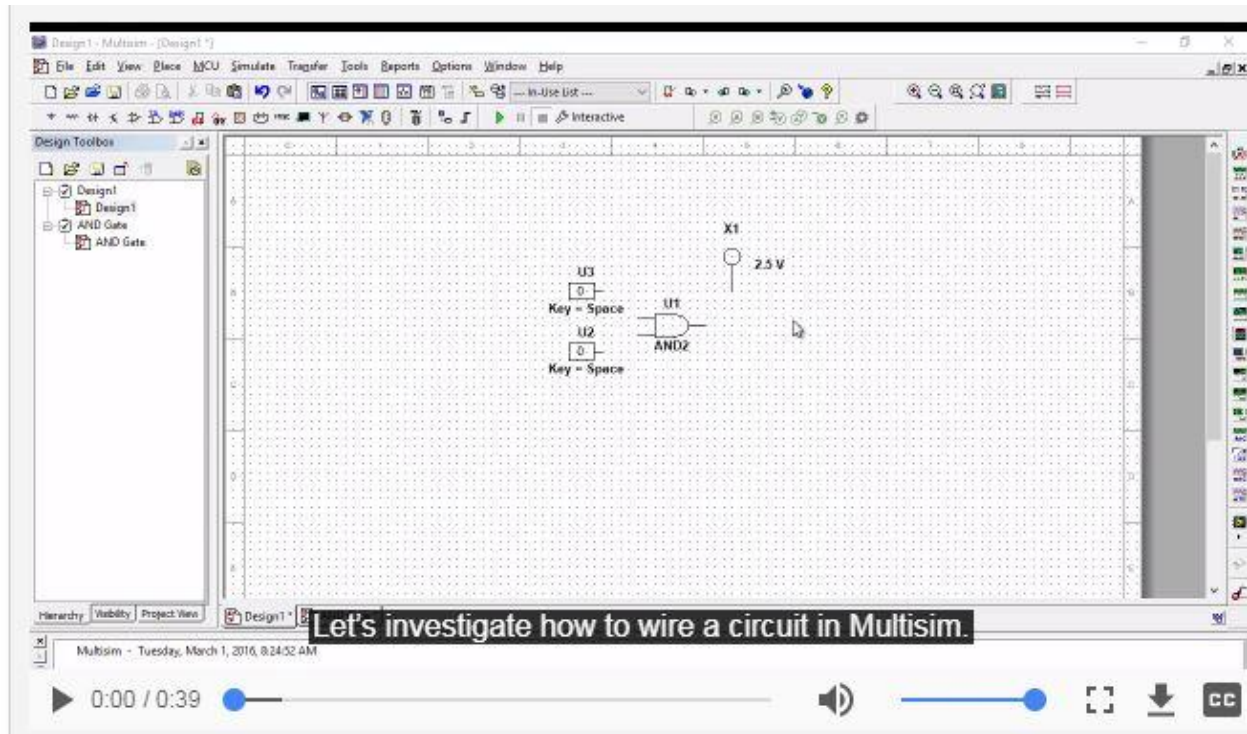


Figure 1-8 Wiring a circuit in Multisim: https://cfts.mythinkscape.com/007_Wiring_a_Circuit_3.mp4

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.

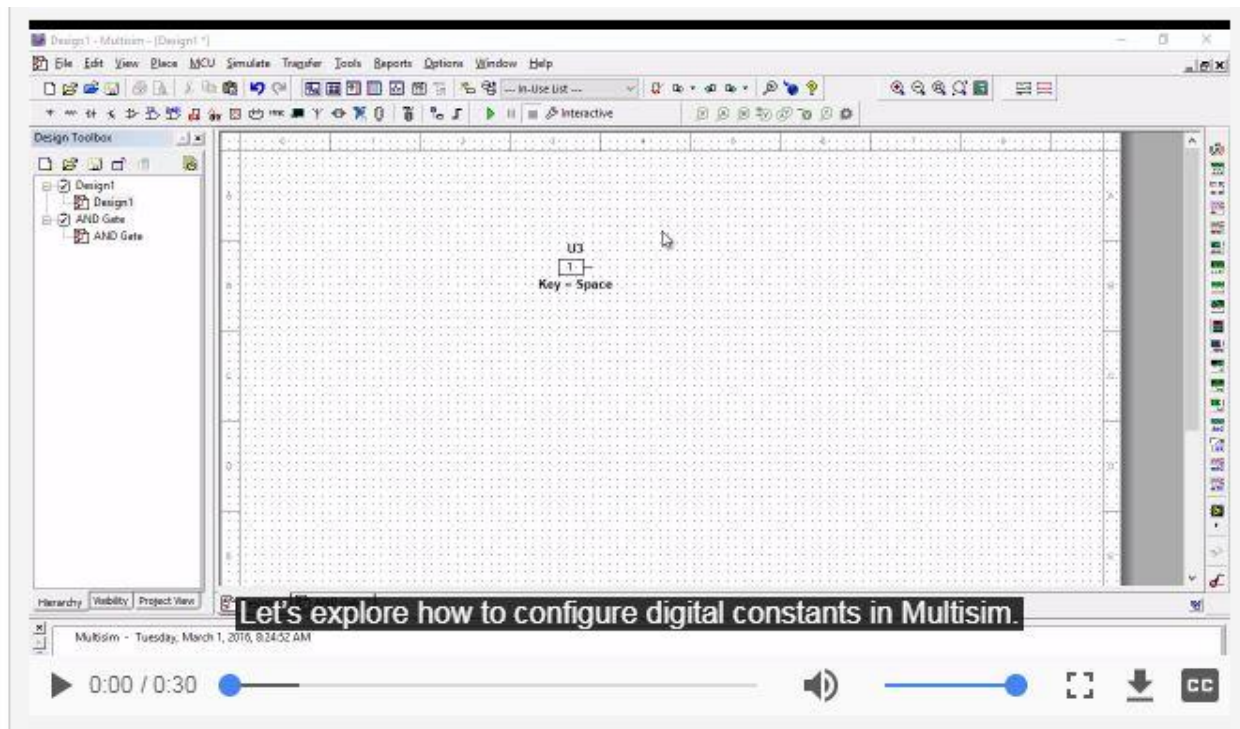


Figure 1-9 Configuring digital constants in Multisim:

https://cfts.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?

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



Figure 1-11 Stop button

1.4 Exercise: Building Circuits on the Digital Electronics Board

DE Circuit

As we saw in the previous section, it's quite easy to build and simulate digital circuits in Multisim. In this section, we're going to do something similar, but this time we'll deploy a **PLD** circuit to the **FPGA** on the Digital Electronics Board.

- Open the circuit file https://cf-ts.mythinkscape.com/OR_gate_-_PLD.ms14 .
- Select **File>>Open**, and then browse to the location where the files for this lab has been saved.

This circuit reads the values from **Switch 0** and **Switch 1** on the Digital Electronics Board, inputs them to an OR gate, and sends the output to the **LED 0** on the Digital Electronics Board.

Deploying a Circuit to the Digital Electronics Board

To run this circuit, we need to deploy it to the board

- Select **Transfer>>Export** to PLD
- In the window that appears, leave the default settings, as seen, and click **Next**.

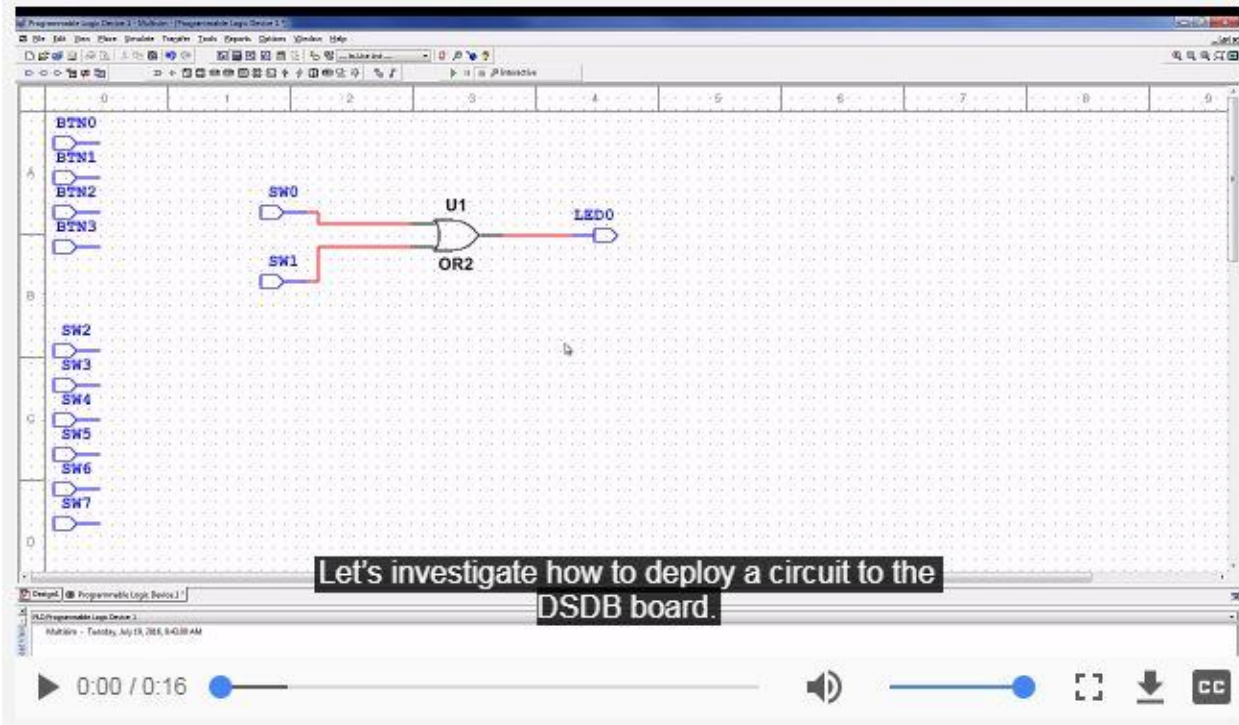


Figure 1-12 Deploying a circuit to the Digital Electronics Board:

https://cf-ts.mythinkscape.com/video/009_Part_1_Deploying_a_Circuit_to_the_DSDB_Board_1.mp4

Configure the second step.

- Choose the **Xilinx Vivado Design Suite** tool.
- Select a location for the life.
- Select the **XC7Z020**.
- Click the **Refresh** button to see if your device is detected.
- Click the **Finish** button to deploy to the FPGA

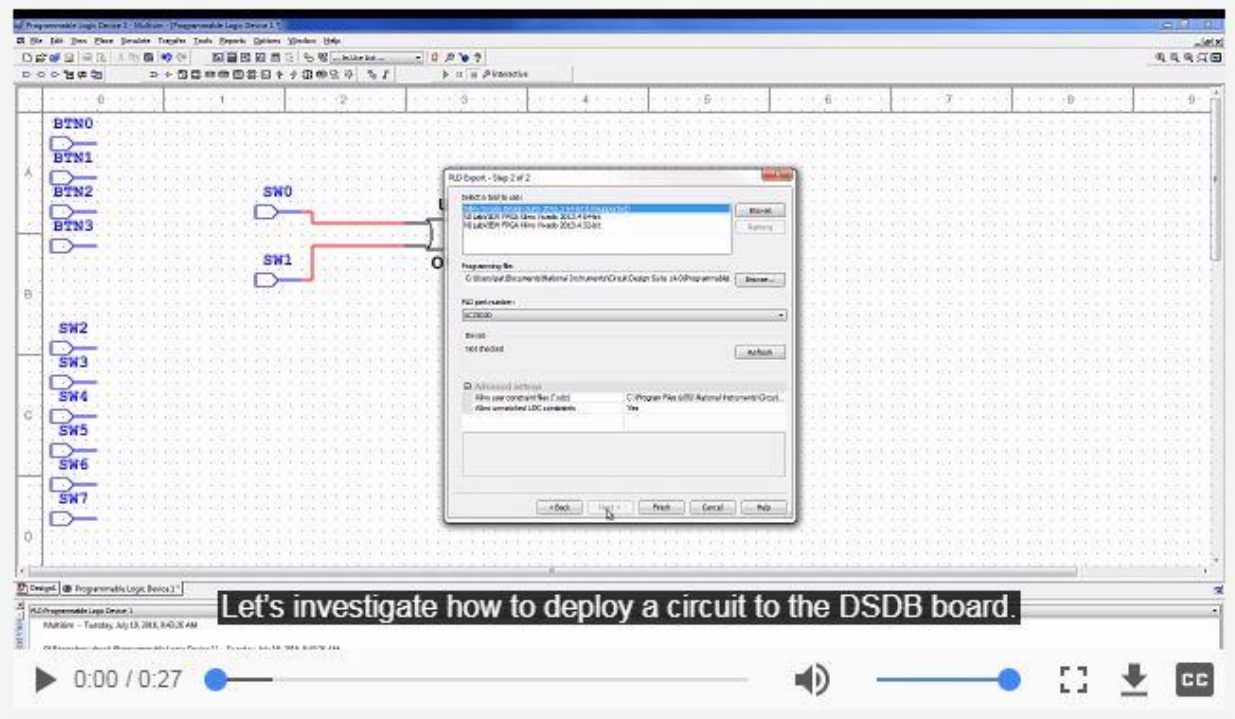


Figure 1-13 Deploying a circuit to the Digital Electronics Board:

[https://cf-ts.mythinkspace.com/video/009 Part 2 Deploying a Circuit to the DSDB Board 1.mp4](https://cf-ts.mythinkspace.com/video/009%20Part%20Deploying%20a%20Circuit%20to%20the%20DSDB%20Board%201.mp4)

Testing an OR Gate on the Digital Electronics Board

Note: A number of steps will run to compile and deploy the code. This can take a little while.

- When the code is finished deploying, change the value of Switch 0 on the Digital Electronics Board.

1-9 Does the LED light up?

- Yes
- No

- Change the Switch 0 to the off position, then turn on Switch 1.

1-10 Does the LED light up?

- A. Yes
- B. No

- Change Switch 0 to the on position, so that both switches are on.

1-11 Does the LED light up?

- A. Yes
- B. No

1-12 How would you describe this gate?

1.5 Exercise: Creating a New PLD Design

Create a new PLD design.

- Select **File>>New**
- In the window that appears, select the **PLD design** and click **Create**.

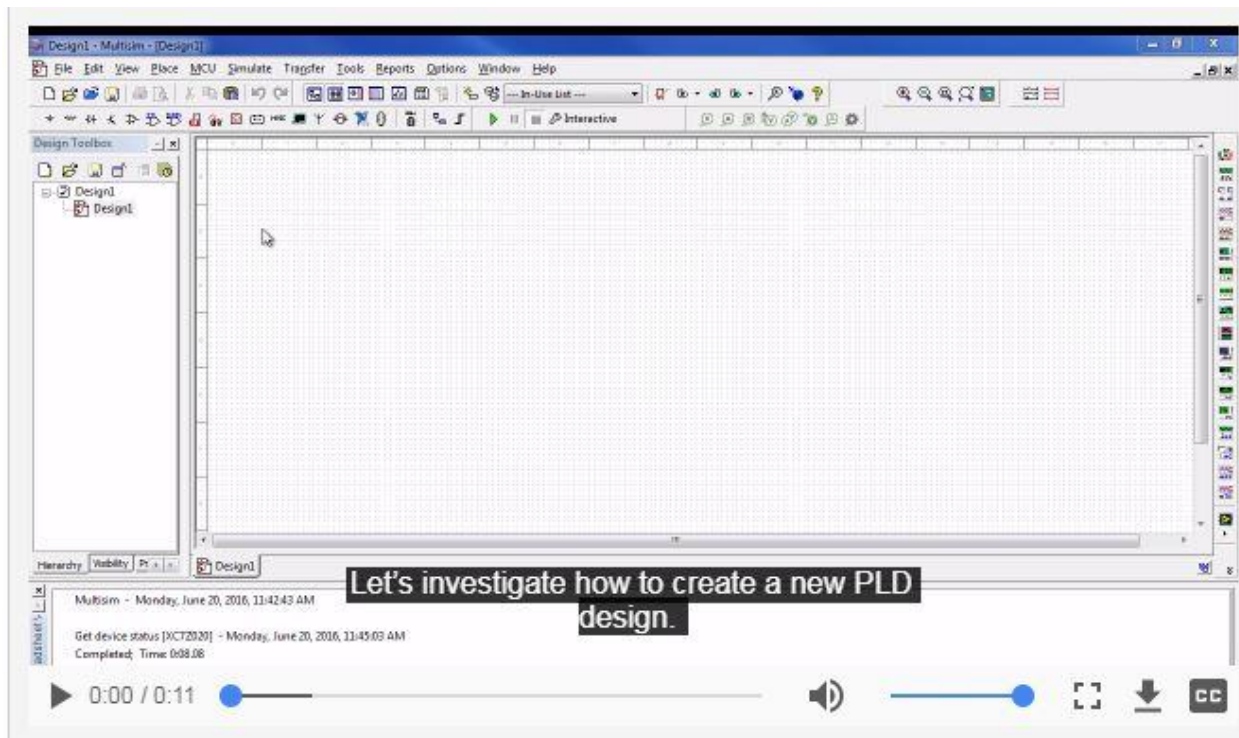


Figure 1-14 Creating a new PLD design:

https://cf-ts.mythinkscape.com/video/010_Part_1_Creating_a_New_PLD_Design.mp4

- Use the standard configuration for the NI Digital Electronics Board.
- In the second step, name the PLD Design '**AND Gate**', and click **Next**.
- In the third step, leave the default settings and click **Finish**.

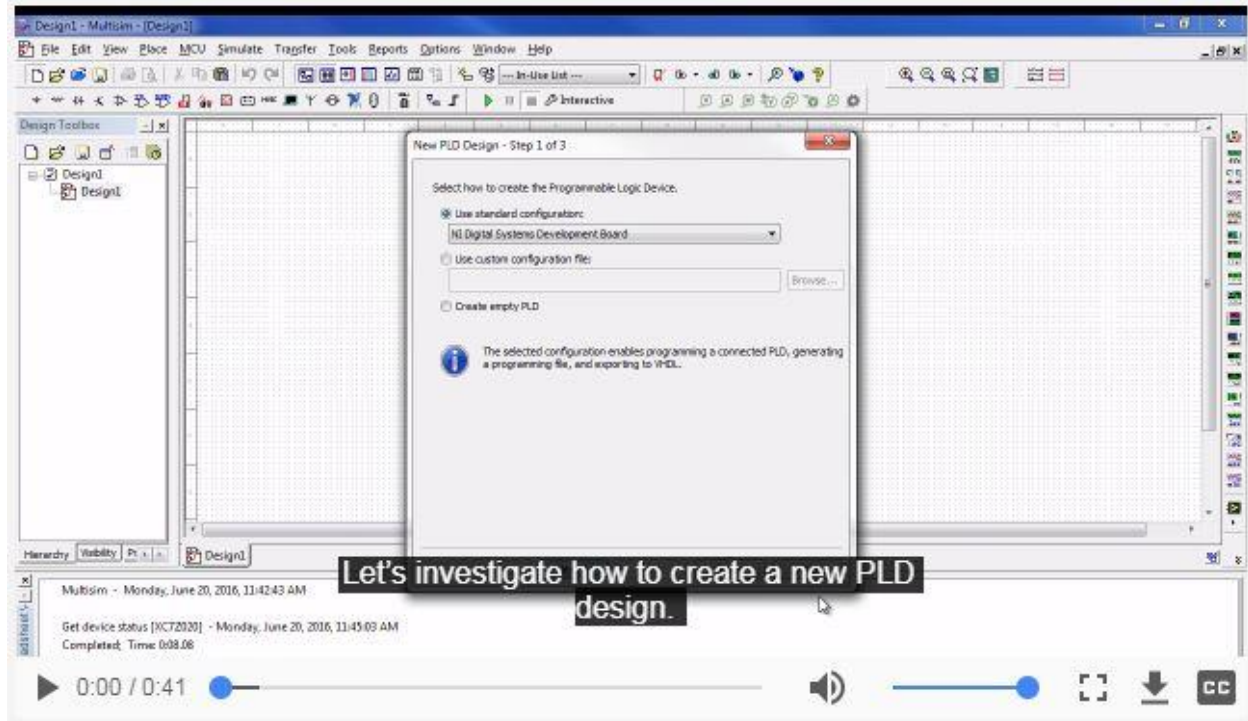


Figure 1-15 Creating a new PLD design:

https://cf-ts.mythinkscape.com/video/010_Part_2_Creating_a_New_PLD_Design_vs2.mp4

1.6 Building and Testing an OR Gate on the Digital Electronics Board

Using the switches, LEDs, and logic gates, create the following circuit.

- When you're done, deploy it to the FPGA, and observe its behavior.

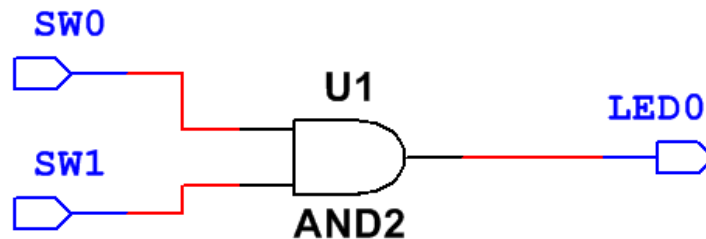


Figure 1-16 OR gate circuit

- Comment on the combination of inputs that turns the LED on and the general behaviour of the gate here:

1.7 Conclusion

1-13 Was it faster to simulate a circuit in Multisim or run a circuit on the FPGA?

1-14 When you might you simulate a circuit in Multisim and when might you run it on the FPGA?

1-15 The NI Multisim environment is used to:

- A. Explore the function of only digital circuits
- B. Create virtual circuits without needing all of the physical components
- C. Test whether a component (such as gates) is defective
- D. All of the above

1-16 What is the main advantage of creating a circuit in Programmable Logic Devices (PLDs) ?

- A. They can be deployed to the Digital Electronics Board
- B. They have more components than Multisim
- C. They convert digital circuits to analog ones
- D. They do not require a USB connection to the NI ELVIS III

1-17 What is the role of the interactive digital constant in the circuit?

- A. Lights up when current passes through it
- B. Exports the circuits from Multisim to Digital Electronics Board
- C. Inverse the direction of current flow
- D. Allows the user to provide a signal that can be set high or low

1-18 In Multisim, we can change the value of which component?

- A. AND Gate
- B. OR Gate
- C. Interactive digital constant
- D. Digital probe

1-19 Which of the following is a benefit of creating circuits with FPGAs?

- A. They're more power efficient
- B. They allow access to intermediate nodes of the circuit for measurement
- C. They're reconfigurable
- D. None of the above

Answer Key – Check Your Understanding Questions Only



Check Your Understanding

1-1 A