



Step 1: Multisim Installation Guide

Updated: 6/20/2017

Procedure

It is imperative that you consult your IT team before you install any software or attempt to change any settings on your computer. Administrative rights are based on a profile created by your System Administrator. Many variations are possible, and these “rights” may not allow you to install this software properly.

Note: This procedure is the first step in a four-step process. This step takes about 60 minutes to complete.

Prepare Your System

1. Disable the User Account Controls (set to **Never notify**).
2. Disable your anti-virus software.
3. Disable the pop-up blocker in the browser you are using.

Note: You can re-enable these settings after the software is installed and operational.

4. Ensure you have permissions to install USB drivers for any Android, Robotics, or other devices that will connect to the computer via USB.

*To install Multisim for Classroom Installation, use the following serial number: **M82X51585***

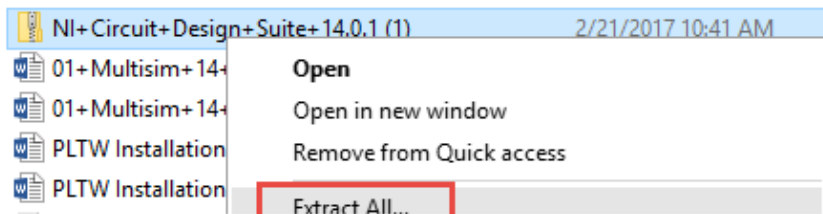
*To install Multisim for Student home installation use the following serial number: **H11X42001***

Note: This procedure is the first in a four-step process. This step takes about 40 minutes to complete.

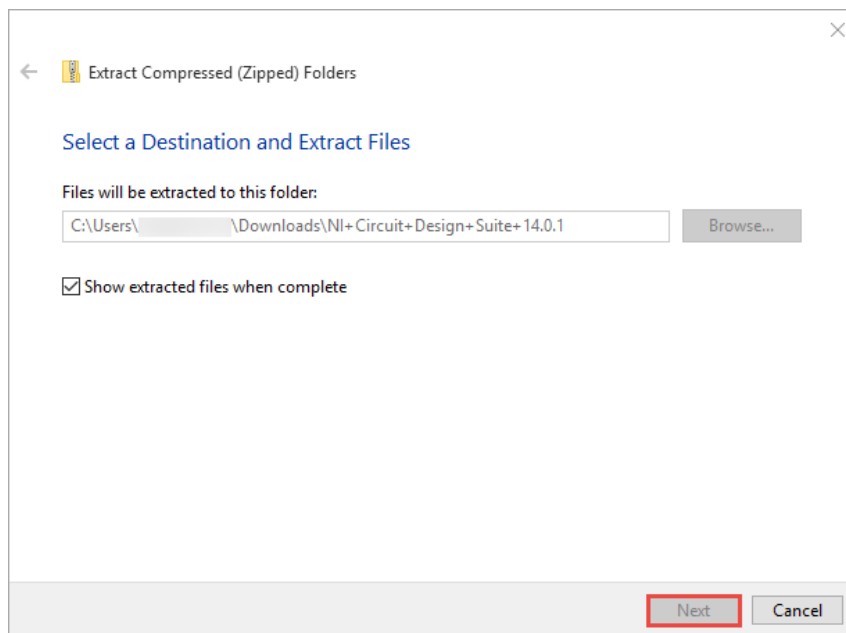
Step 1 – Multisim 14.1

Part I: Install Multisim

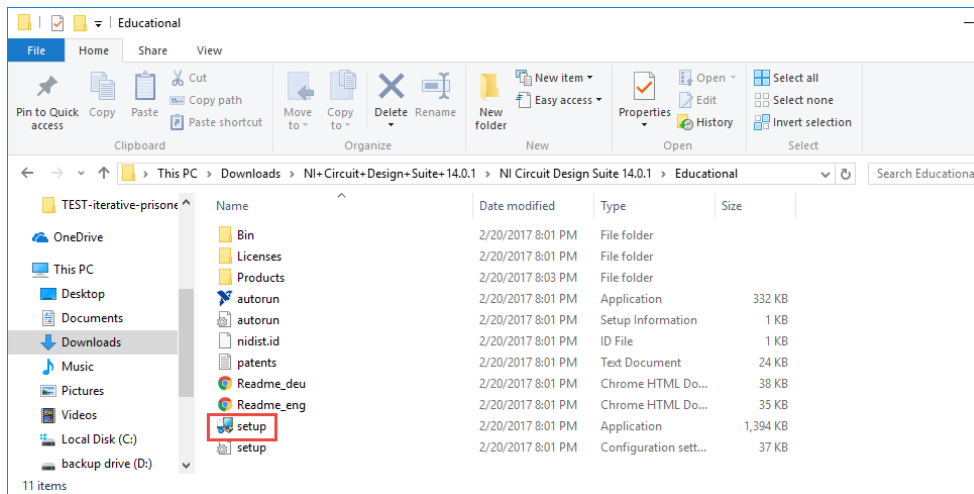
1. Select the following link to download the installer: <https://s3.amazonaws.com/support-downloads.pltw.org/Installers/Digital+Electronics/NI+Circuit+Design+Suite+Education.zip>
2. Go to the downloaded file and extract the .zip file.



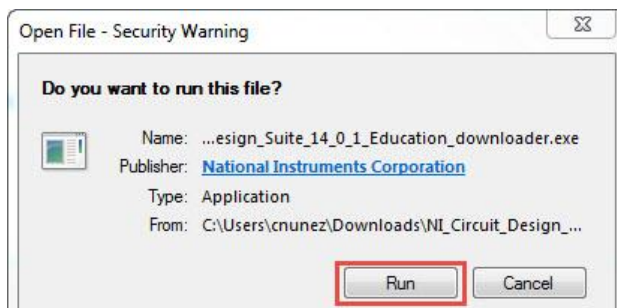
3. Choose a location for the installer to extract and begin extraction. Once the extraction is complete, select the **next** button.



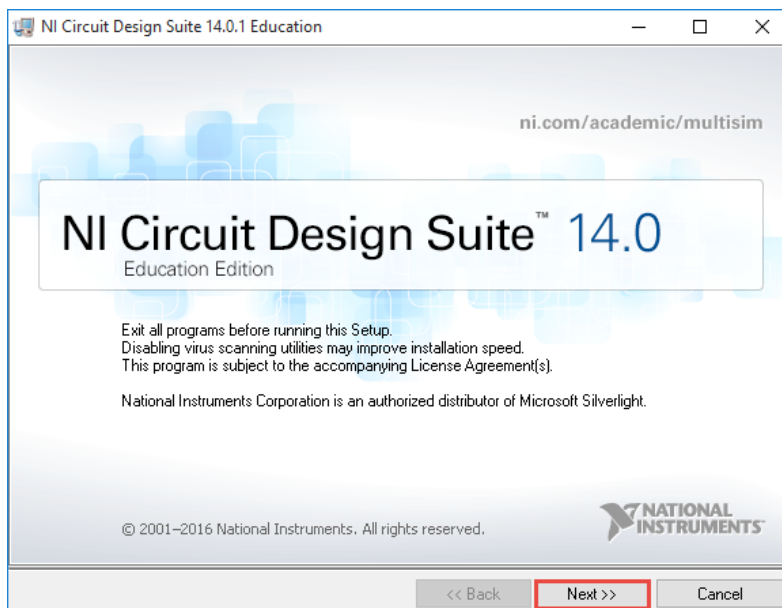
4. Open the downloaded folder and click through the folders until you can select **setup**.



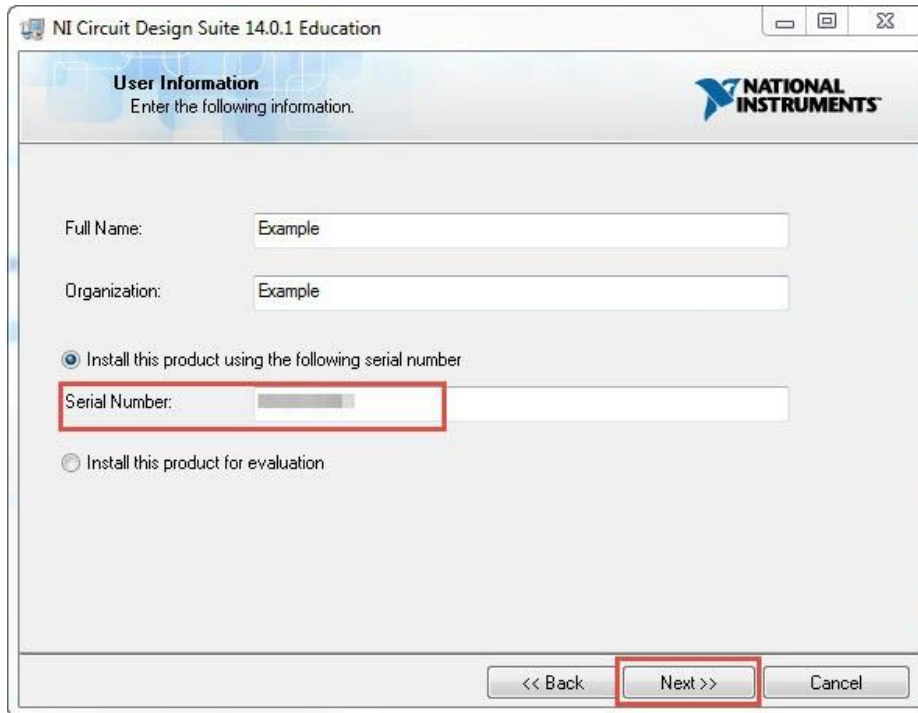
5. If prompted to allow the software to make changes to your computer, select **Run**.



6. Select **Next** on the installer.

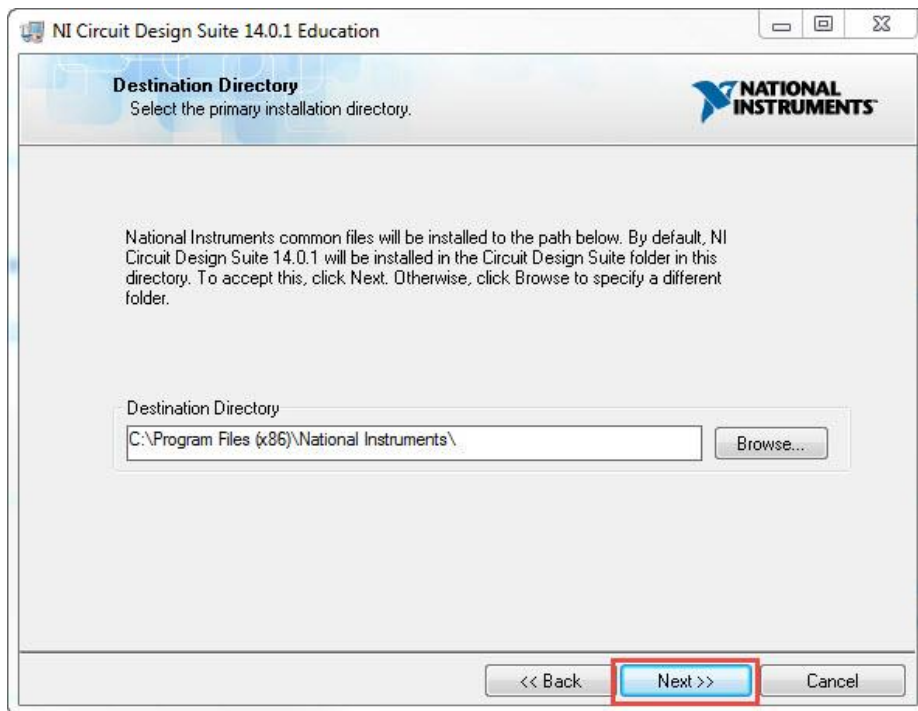


7. Enter your **Name, Organization, and the Serial Number** from page 1 (**M82X51585**) and select **Next**.



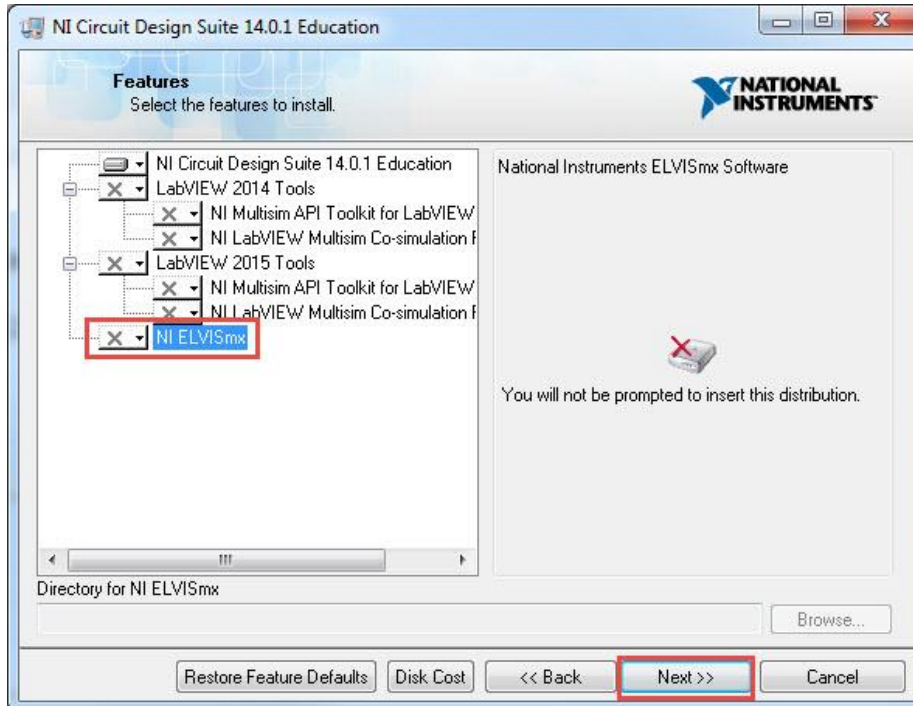
The dialog box is titled "NI Circuit Design Suite 14.0.1 Education" and "User Information". It prompts the user to "Enter the following information." The fields include "Full Name:" with the example text "Example", "Organization:" with the example text "Example", and "Serial Number:" with a red box around the input field. There are two radio buttons: "Install this product using the following serial number" (selected) and "Install this product for evaluation". At the bottom, there are three buttons: "<< Back", "Next >>" (highlighted with a red box), and "Cancel".

8. To keep the default directory, select **Next**.

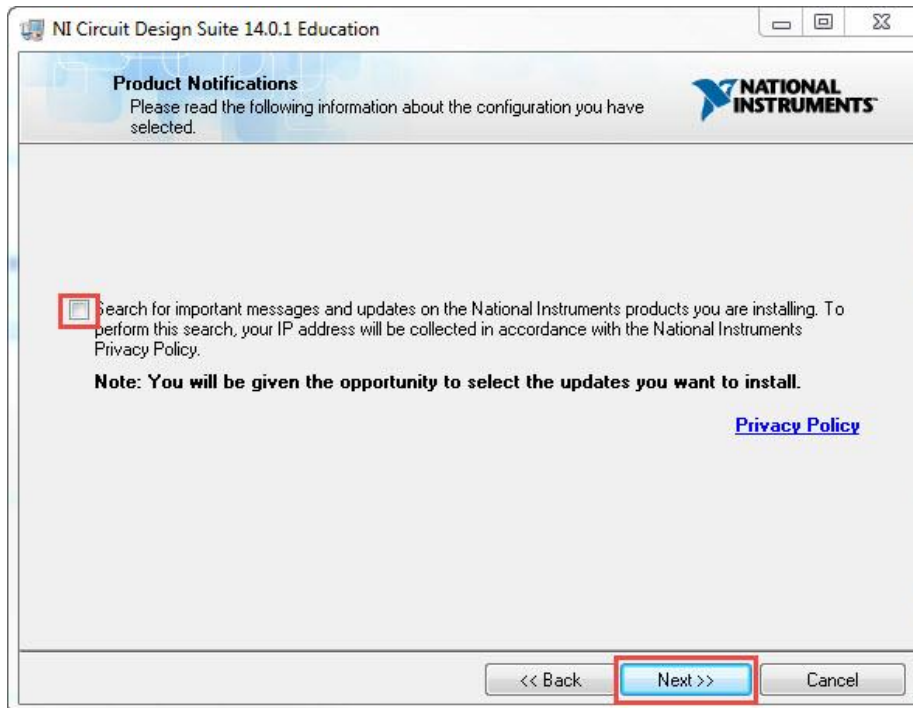


The dialog box is titled "NI Circuit Design Suite 14.0.1 Education" and "Destination Directory". It prompts the user to "Select the primary installation directory." The text states: "National Instruments common files will be installed to the path below. By default, NI Circuit Design Suite 14.0.1 will be installed in the Circuit Design Suite folder in this directory. To accept this, click Next. Otherwise, click Browse to specify a different folder." The "Destination Directory" field shows "C:\Program Files (x86)\National Instruments\" with a "Browse..." button next to it. At the bottom, there are three buttons: "<< Back", "Next >>" (highlighted with a red box), and "Cancel".

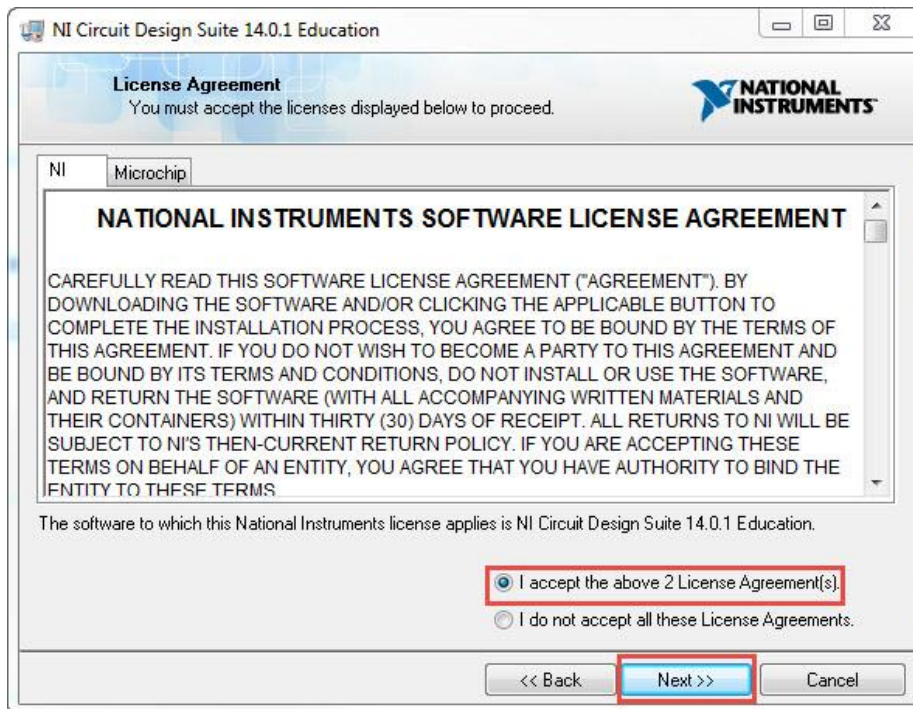
9. In the Features window, select **Do not install NI ELVISmx**, then select **Next**.



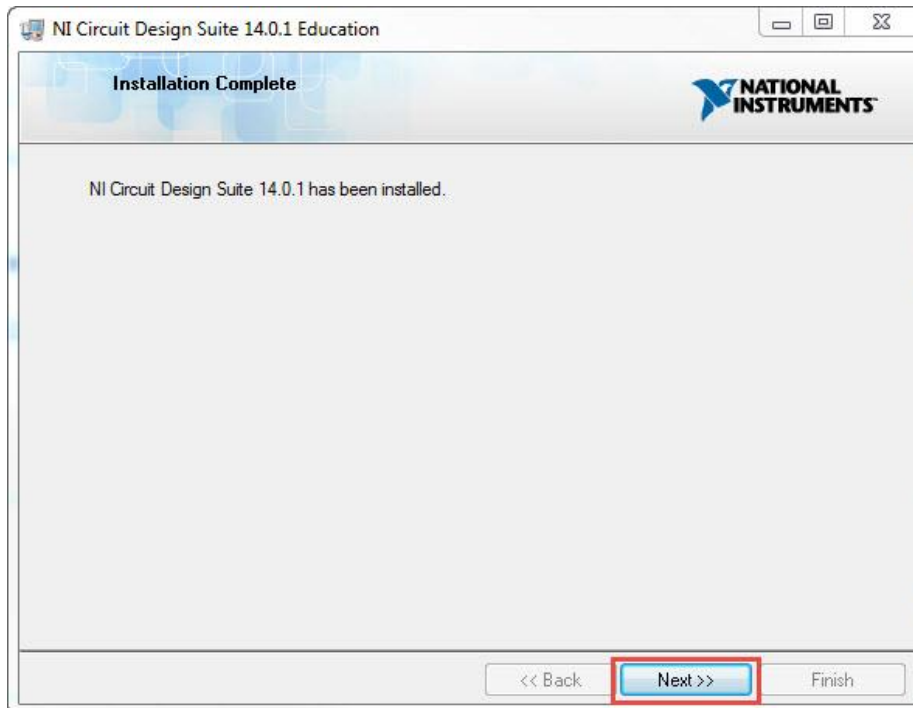
10. Clear the check box and select **Next**.



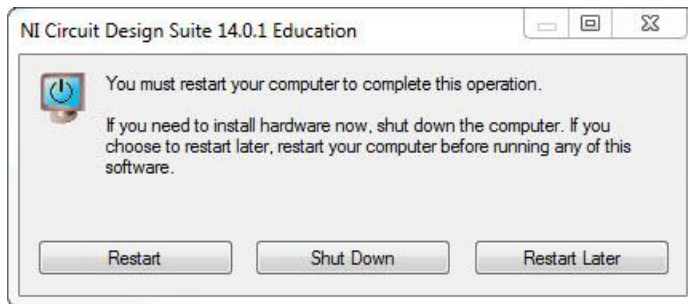
11. For the License Agreement, select **I accept** and **Next**.



12. When the installation is complete, select **Next**.

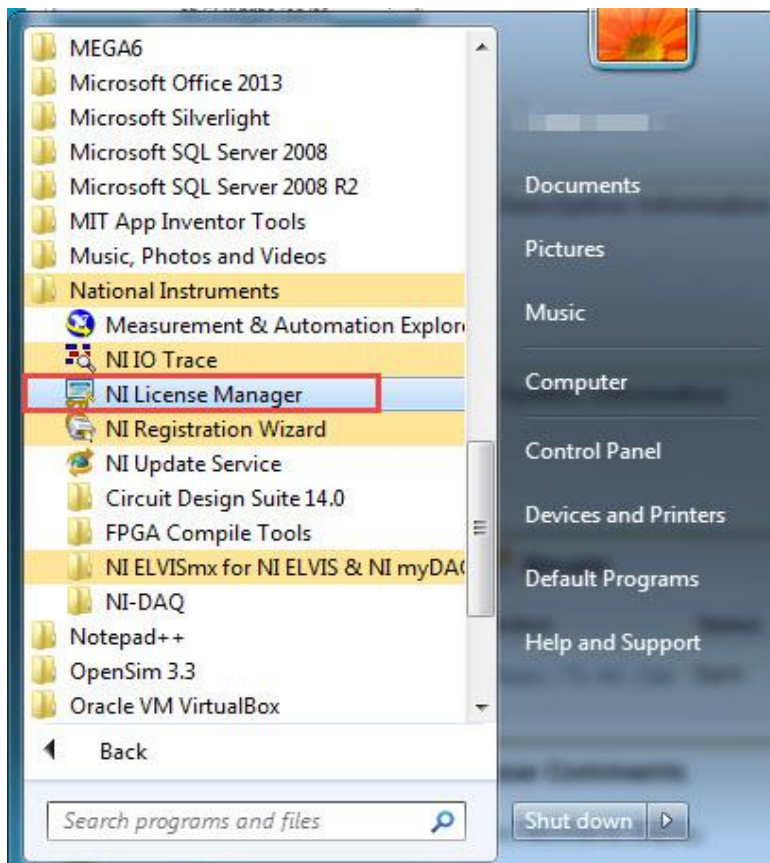


13. Select **Restart** or **Restart Later**.

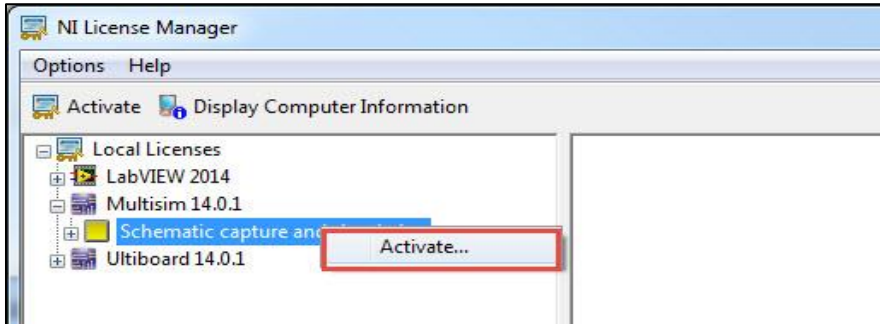


Part 2: **Activate Multisim**

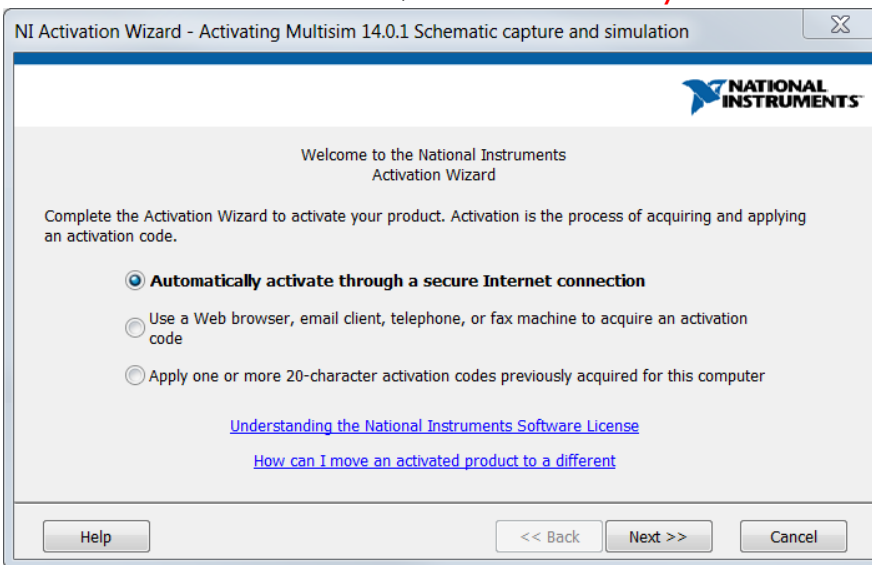
14. Select **Start > All Programs > National Instruments > NI License Manager**.



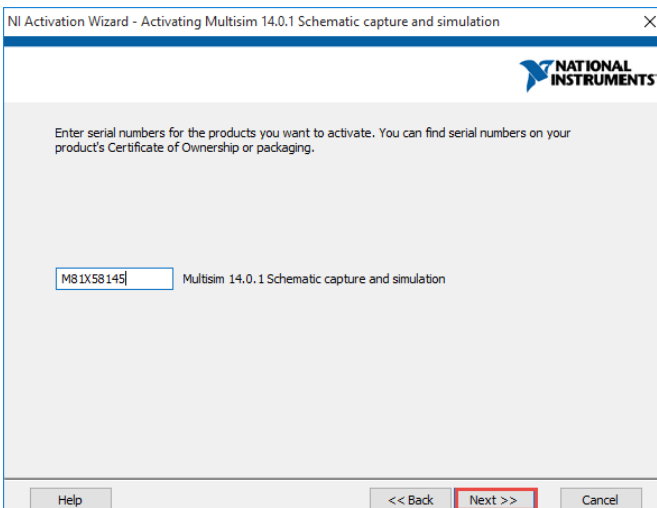
15. Under Multisim 14.0.1, right-click **Schematic capture and simulation** and select **Activate**.



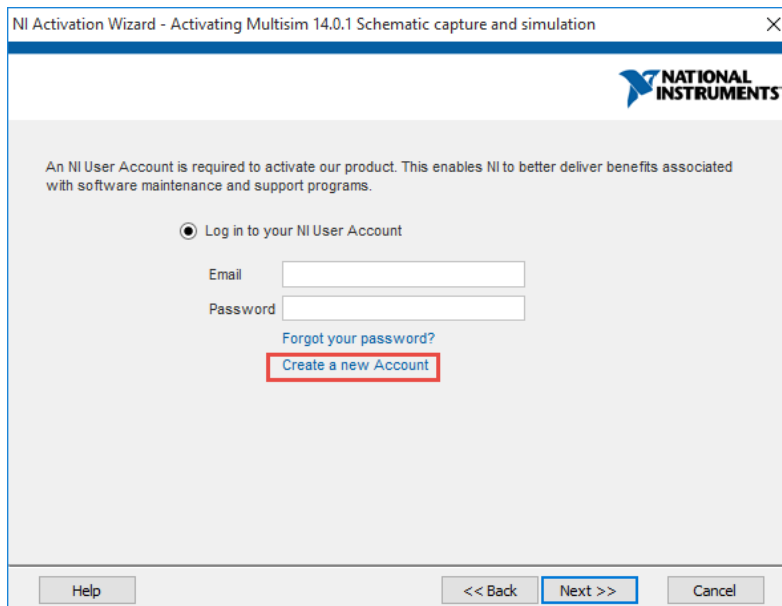
16. On the Activation Wizard box, select **Automatically activate** and then select **Next**.



17. Input the serial number from page 1 and select **Next**.



18. Select **Create a new Account**.



NI Activation Wizard - Activating Multisim 14.0.1 Schematic capture and simulation

NATIONAL INSTRUMENTS

An NI User Account is required to activate our product. This enables NI to better deliver benefits associated with software maintenance and support programs.

☒ Log in to your NI User Account

Email

Password

[Forgot your password?](#)

Create a new Account

Help << Back Next >> Cancel

19. Fill out the information requested on the NI website and select **Create Account**.

Create an NI User Account

Already have an account? [Log In >](#)

First Name

Last Name

Company

Company Address

City

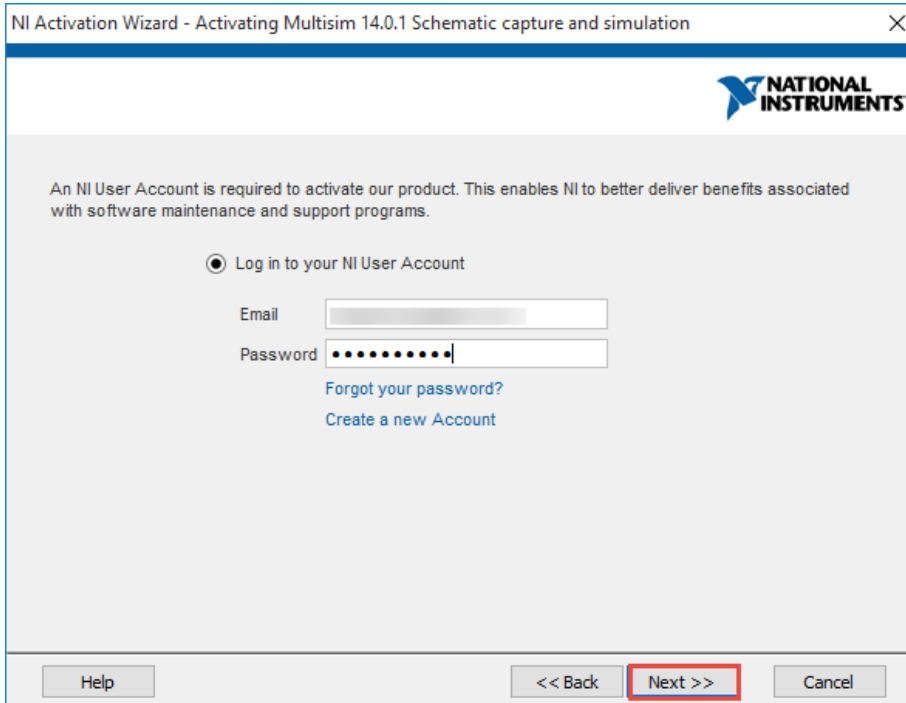
State Postal Code

Email Address

Password

CREATE ACCOUNT

20. After your account is successfully created, you can return to the activation wizard and input your credentials. Then click **Next** to proceed.



NI Activation Wizard - Activating Multisim 14.0.1 Schematic capture and simulation

NATIONAL INSTRUMENTS

An NI User Account is required to activate our product. This enables NI to better deliver benefits associated with software maintenance and support programs.

☒ Log in to your NI User Account

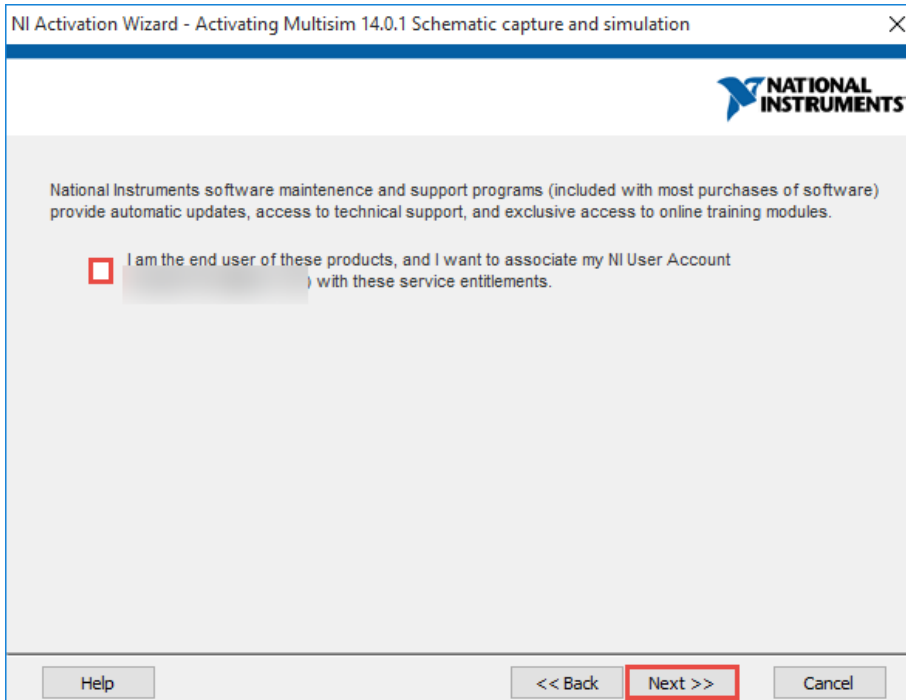
Email

Password

[Forgot your password?](#)

[Create a new Account](#)

21. Deselect the check box, unless you would like to receive automatic updates and other information by email, and select **Next**.



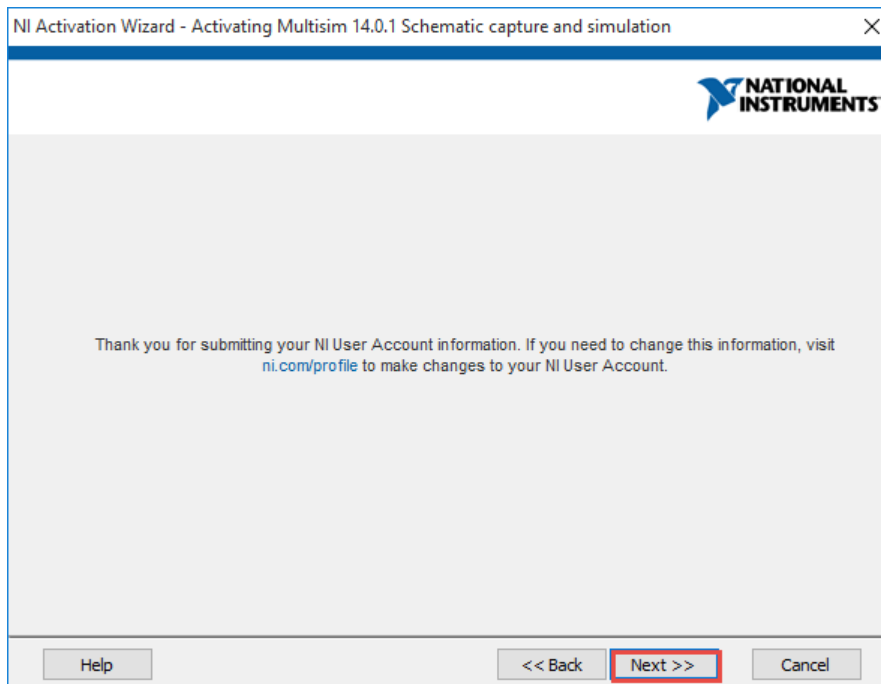
NI Activation Wizard - Activating Multisim 14.0.1 Schematic capture and simulation

NATIONAL INSTRUMENTS

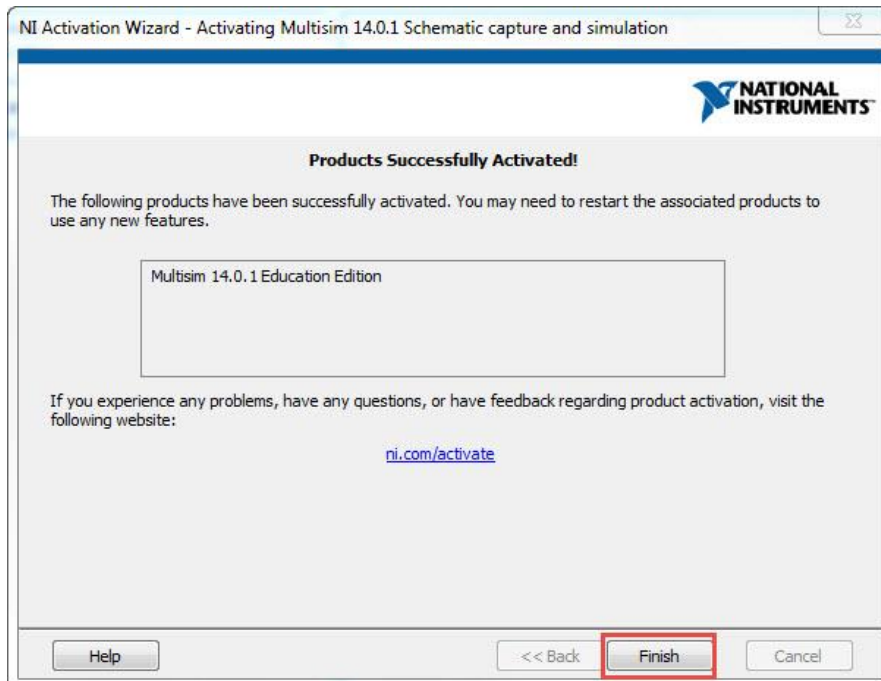
National Instruments software maintenance and support programs (included with most purchases of software) provide automatic updates, access to technical support, and exclusive access to online training modules.

☐ I am the end user of these products, and I want to associate my NI User Account with these service entitlements.

22. Select **Next**



23. When activation is complete, select **Finish**.



24. Continue with the instructions for Step 2 in the LabVIEW/Xilinx 14.7 Tools Installation Guide.