



## Installation Steps for Release 17.2 and Prior Releases of Student Products

### Before installing:

- Ensure that your computer meets the technical requirements necessary to run the software. Please see the [product data sheet](#) for details.
- Ensure that you have Administrator privileges for the computer you are installing this product on.
- Uninstall any existing ANSYS, Inc. products present on your computer. The ANSYS Student installation will overwrite any existing installations.
- Make sure that you have your license key available. This is a separate download covered in the "Free Download" section.
- Please review our Installation Guide and [FAQs](#).

For installing student products, see the [installation videos](#) or follow the installation steps given below in part 1, part 2, and part 3.

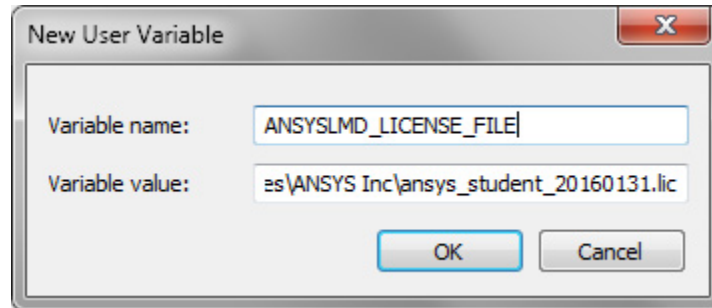
### Installation Steps - Part 1 - ANSYS Student Product Installation:

- Extract (Unzip) the downloaded installation files.
- Run setup.exe from the extracted files. Right-click on setup.exe and select **Run as Administrator**.
- Click the **Install ANSYS, Inc. Products** text.
- Read and accept the click wrap to continue.
- Click the Next button to accept the default values throughout the installation.
- **Important:** Skip over the "Enter License Server Specification" step. **DO NOT enter any port information or otherwise install the ANSYS License Manager.**
- After the installation is completed, click **Next** and a message marked **\*\*\*Important\*\*\*** displays indicating that the installation is complete.
- Check the survey box at the bottom if you wish to submit the survey.
- Click the **Exit** button.
- You are now back to the Release Installation home page. **Be sure to NOT select any other options as these aren't supported by the ANSYS Student license. Do not select License Manager Installation.**
- Click the **Exit** Button to close the installer.
- The ANSYS Student software is now installed.
- Next, follow the steps in **Part 2** below.

### Installation Steps - Part 2 - ANSYS License File Installation:

1. Download the license file - ansys\_student\_<yyyymmdd>.lic - to your computer. The <yyyymmdd> portion of the filename indicates the expiration date of the license file.
2. Extract the downloaded license file and note the location where you save it to:
  - We recommend moving the license file to C:\Program Files\ANSYS Inc\Shared Files\Licensing, but you can also put the license file on your desktop.
  - Do not put the license file in a **Temp** directory
  - Do not save this license file using the MS Word format.

3. Set the **ANSYSLMD\_LICENSE\_FILE** environment variable to the location of the ansys\_student\_.lic file, including the filename. For example:  
C:\Program Files\ANSYS Inc\Shared Files\Licensing\ansys\_student\_20160131.lic
  - o Follow these steps to set an environment variable in MS Windows:
    - Open the **Control Panel**
    - Select **System and Security**
    - Select **System**
    - Select **Advanced system settings**
    - Click the **Environment Variables** button
    - Select the **New** button under System variables



- o Note that there will be several environment variables with very similar names already present. Be careful to use this EXACT environment variable name.
4. Next, follow the steps in Part 3 below.

### **Installation Steps - Part 3 - Running the ANSYS Student product:**

After completing Parts 1 and 2 of the installation, we recommend rebooting your machine. Then the best way to run ANSYS Student is from the start menu.

1. Go to the **Start Menu** and either select **Workbench [version #]** OR select **All Programs**, then select the **ANSYS [version #]** folder, and select **Workbench [version #]**.
2. The ANSYS Student Workbench project page will launch.
3. Congratulations you are now ready to start using ANSYS Student!

If you run into any problems following the above instructions, then be sure to review our [Frequently Asked Questions](#).