ANSYS Student

Frequently Asked Questions

This FAQ provides answers to general questions about ANSYS Student, as well as resolutions to common error messages that you may encounter. Use your browser's "Find" feature to search for keywords in this FAQ page.

Q1. Can I submit technical support requests, or otherwise obtain technical support for ANSYS Student?

We do not provide technical support for this product beyond this FAQ page, the product documentation included with the software, and the Support Resources webpage.

Q2. Are there any training resources available for ANSYS Student?

The self-guided support material available on the Student Resources includes some getting started and "how to" videos.

Q3. Why do I have to run the installer using Run as Administrator on Microsoft Windows 7 & 8?

Microsoft Windows 7 and 8 have higher internal security controls than previous versions of Windows. These controls are known to interfere with the installation and license setup of ANSYS products. Using **Run as Administrator** when you install the software avoids these problems.

Q4. Workbench starts, but why don't I see any (or just four default) tools available under Analysis or Component Systems?

This usually means that Workbench is unable to locate the license file. Try the following actions to find the file:

• Confirm that the license file has been installed according to the Installation Instructions. If you installed ANSYS Student on Windows 7 or 8, ensure that you installed it using **Run as Administrator.**

- Ensure ANSYSLMD_LICENSE_FILE variable points to the actual license file, not just the directory where they've saved it.

- Rename the %APPDATA%\Ansys\v160 folder to %APPDATA%\Ansys\v160.old and restart Workbench.

- Run C:\Program Files\Ansys Inc\v160\commonfiles\tools\win64\ProductConfig.exe, choose Install Required Prerequisites and Configure Products.
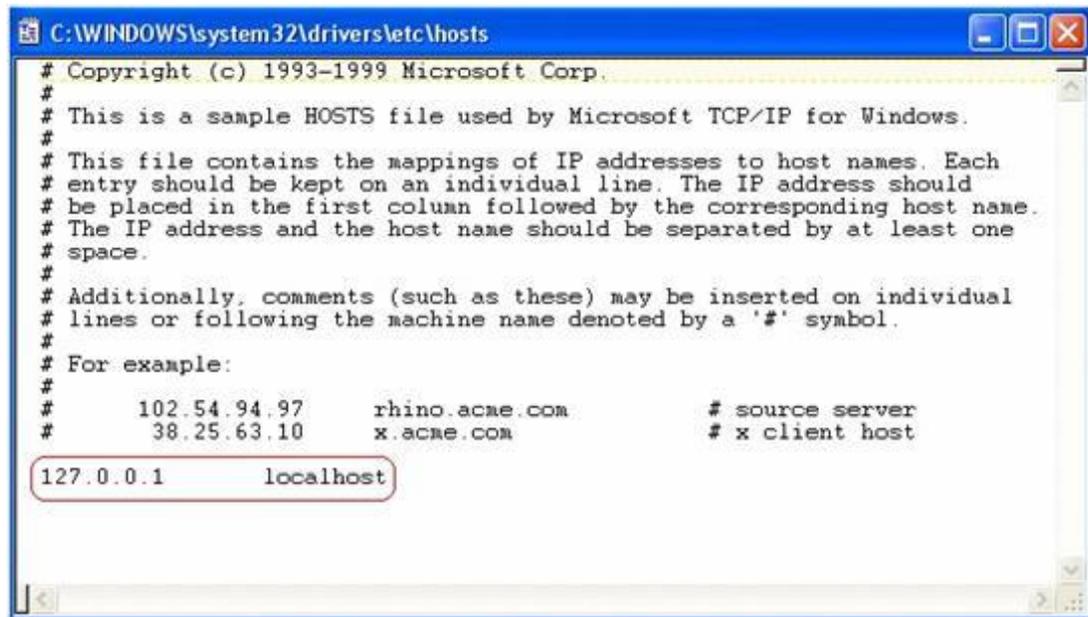

```
C:\WINDOWS\system32\drivers\etc\hosts
# Copyright (c) 1993-1999 Microsoft Corp.
#
# This is a sample HOSTS file used by Microsoft TCP/IP for Windows.
#
# This file contains the mappings of IP addresses to host names. Each
# entry should be kept on an individual line. The IP address should
# be placed in the first column followed by the corresponding host name.
# The IP address and the host name should be separated by at least one
# space.
#
# Additionally, comments (such as these) may be inserted on individual
# lines or following the machine name denoted by a '#' symbol.
#
# For example:
#
#      102.54.94.97     rhino.acme.com          # source server
#       38.25.63.10     x.acme.com              # x client host

127.0.0.1       localhost
```

- Disable IPV6 as described in Question 6.

Q5. Why did my ANSYS Student license stop working?

The main reason for this is that your license has expired. The ANSYS Student license is valid for 6 months (if you are within 30 days of expiration, ANSYS Student warns you upon startup that you need to renew your license). You can easily renew your license by downloading the most current license file from the ANSYS Student download page and modifying your ANSYSLMD_LICENSE_FILE variable to point to the new license file.

Q6. What do I do if I receive one of the following error messages: "AnsysWBModule has stopped working"; "ANSYSWBU.exe Encountered a problem and needs to close.";or "ANSYSWBU.exe Encountered a problem. A diagnostic file has been written…"?

Try the following to resolve the errors:

1. Rename the %APPDATA%\Ansys\v160 folder to %APPDATA%\Ansys\v160.old and restart Workbench.

2. Update your video card driver, as described in Question #9.

3. Run C:\Program Files\Ansys Inc\v160\commonfiles\tools\win64\ProductConfig. exe, choose Install Required Prerequisites and Configure Products.

Q7. Why did I get the following error message: "Local server exited or could not read local server port ANSYSLI_LOCAL_PORT"?

The TEMP variable that defines the directory location for Window temporary files is set to a network drive or a location that is not writable. In the Windows Control Panel, choose System > Advanced System Settings > Environment Variables and reset the System variable TEMP to C:\Temp.

ANSYS®

Q8. Why am I having one of the the following problems when I use ANSYS DesignModeler?

- DesignModeler takes several minutes to open.
- When DesignModeler opens, the dialog: ANSYS Workbench - Select desired length units does not display.
- In Tools > Options panel > DesignModeler > Units, the panel shows two identical preference settings labeled Length Unit.

These symptoms are most likely to occur if Windows Defender is installed and running. There have been reports that Microsoft Windows Defender removes the localhost setting from the C:\Windows\system32\drivers\etc\hosts file. See the instructions in FAQ Question 5.

Q9. When trying to open a simulation, I received an error message: "Mechanical editor is unable to load". However, the project page and DesignModeler opened without any problems.

This issue is usually resolved by updating your graphics driver. Here are the steps to follow:

1. First locate your video card and driver information:
   a. Open the Windows Control Panel
   b. Navigate to Display > Settings > Advanced.
   c. Select Adapter (this will tell you what video card you are using).
   d. Select Properties > Driver (this will tell you what driver version you are using).

2. After you have your video card and driver information, go to the vendor's web site and download a newer driver.

3. Remove the existing driver:
   a. Open up the Windows Control Panel.
   b. Select Add/Remove Programs or Programs and Features (Microsoft Windows 7 and 8).
   c. Uninstall the NVIDIA or ATI Drivers.
   d. Reboot.
   e. Install the new driver you downloaded.
   f. Reboot again.

NOTE: It is very important that you follow these steps exactly, especially when removing the existing driver!

Q10. ANSYS Inc. has recently announced the availability of Release 18.0. When is the next release available?

The Student Product download is typically made available within 1 month of each commercial release.

Q11. Why did I get the following Mechanical Wizard error, similar to this Release 14.5 screenshot?
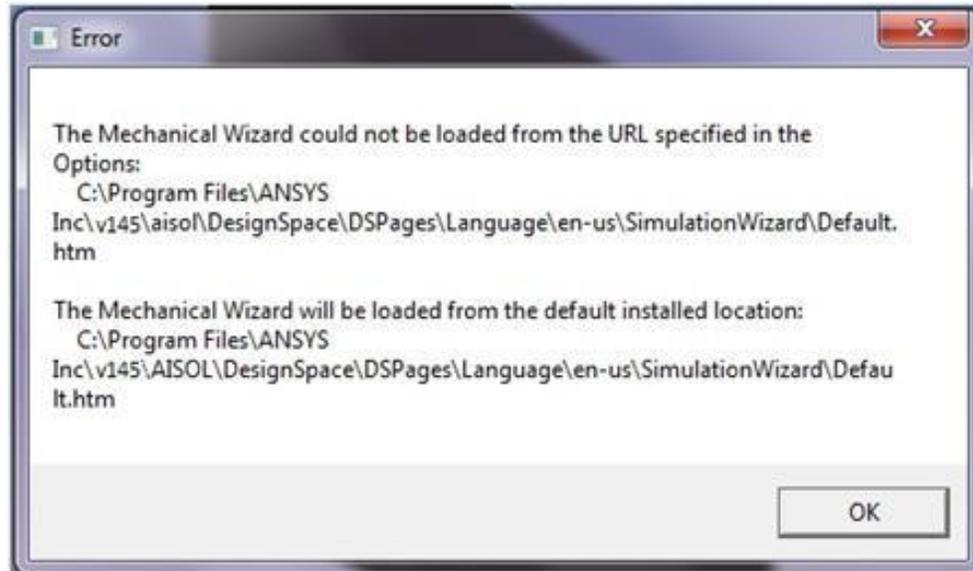
This could be related to your security settings. Please take a look at Control Panel > Internet Options > Security. Adjust your security settings to allow ActiveX components to run in Internet Explorer.

Q12. Regarding high performance computing (HPC), what does support for up to 16 cores (SMP) mean?

Sixteen core Shared Memory Parallel (SMP) allows you to make use of up to 16 cores (any multi-core processor combinations up to 16 cores) when solving simulations on your laptop or local machine where you have the product installed. All of the SMP processor cores share the memory of a single machine, as opposed to distributed memory parallel (DMP) that uses multiple cores located in multiple machines. The number of SMP cores used is specified in the solver settings. Note that the ANSYS Student product license does not support distributed solutions (DANSYS) or MPI.

Q13. I have already installed an earlier release of ANSYS Student. Since a more recent release is now available, can I upgrade to this recent release?

Yes! Your existing license file will work, but you will need to download and install the latest ANSYS Student media. Please follow these steps:

1. Uninstall the earlier release ANSYS Academic Student product. Make sure that you do not remove the ANSYS License Manager.

2. Go to to the ANSYS Student download page and download the latest installation media.

3. Extract the media and run the setup.exe file to install the latest release product. If you are installing on a Windows 7 or 8 system, be sure to right-click Run as Administrator. (See FAQ Question 3 for more information.)

4. Follow the rest of the installation instructions on the [Product Download page](#).

Q14. When I have multiple ANSYS Workbench windows open, why do I receive an error message similar to the following (note the capability name could be different): "Capability CAD Interface - Parasolid does not exist in the ANSYS licensing pool."?

This problem occurs when you have changed the License Preferences setting from shared to not shared. The solution is to open License Preferences and select the shared setting. Alternatively you can rename %appdata%\ansys\v162 to v162.old and restart ANSYS Workbench (you may need to change the "v162" in the folder & filename if you are using a different version).

Q15. Why did I receive the following error when trying to open Mechanical via a Model or Setup cell in ANSYS Workbench: "The shared license is not available to perform the Geometry Attach operation."?

There are two possible solutions to resolve this:

1. ANSYS DesignModeler is open with the geometry at the same time you are trying to open the geometry in Mechanical via the Model or Setup cell in ANSYS Work bench. You should make sure to close ANSYS DesignModeler before opening the Model or Setup cell.

2. You have changed the License Preferences setting from "Shared" to "Not shared". The solution is to open License Preferences and select the shared setting. Alterna tively you can rename %appdata%\ansys\v162 to v162.old and restart ANSYS Workbench.

Q16. I accidentally installed the ANSYS License Manager, how do I uninstall?

Follow these steps to uninstall every trace of the ANSYS License Manager:

1. All commands must be run in an administrator command prompt window. Go to the Windows start menu, type cmd and then right-click on the cmd.exe and select "Run as administrator."

2. Stop the ANSYS, Inc. License Manager from the ANSYS License Management Center….Click on "View Status/Start/Stop License Manager" and click "STOP"

3. Uninstall the ANSYS, Inc. License Manager service. You must use the following command:

"C:\Program Files\ANSYS Inc\Shared Files\Licensing\winx64\ansysli_server" –k uninstall

4. Uninstall the ANSYS Licensing Tomcat service. Use the following commands:
   sc stop "ANSYSLicensingTomcat"
   sc delete "ANSYSLicensingTomcat"

5. Remove the ANSYS, Inc. License Manager folder from the Start menu.

6. Remove the ANSYSLIC_DIR and the ANSYSLIC_SYSDIR environment variables, if set.

7. Install the free product again following our installation instructions carefully.

Q17. After successfully installing ANSYS Student, selecting the "Help" menu results in nothing happening. How do I access the Help System?

For some systems the Help Viewer doesn't launch when selected from within ANSYS Student. The workaround is to go to Start Menu > All Programs > ANSYS 18.0 > Help > ANSYS Help 18.0.