
Installation Tutorial: Installing ANSYS, Inc. Products on One Machine and a License Server on Another Machine

Introduction

This tutorial walks you through installing ANSYS, Inc. products on one Windows machine and installing the ANSYS, Inc. License Manager on a different Windows machine that will act as a license server. This tutorial assumes that no ANSYS, Inc. products or licensing have previously been installed on these machines or, if products have been installed, all components, including licensing, have been completely removed.

If this scenario does not suit your needs, please see the other tutorials available on the Download Center at <https://www1.ansys.com/customer/>.

You can follow this tutorial while actually installing the products and setting up the license server. To do so, simply make the selections that are pertinent to you or insert your specific information where noted.

Before You Begin

We recommend that you have the following information available before you begin this tutorial:

- An account on the ANSYS Customer Portal. If you do not have an account, you may register at <https://www1.ansys.com/customer/> to receive your own account.
- Your license file from ANSYS, Inc., saved to a temporary directory.
- Open port numbers for both the FLEXlm and ANSYS Licensing Interconnect on your license server machine. Defaults are 1055 and 2325, respectively. To verify that these port numbers are available, open a command line on the server machine and enter the following command:

```
netstat -a -p tcp
```

You will see a list of active ports. If 1055 and 2325 are listed, they are already in use and cannot be used for ANSYS, Inc. licensing. In this case, you will need to specify different port numbers where indicated later in this tutorial.

- The name of the machine that will act as a license server.

You should also verify that you are running on a supported platform. ANSYS, Inc. supports both 32- and 64-bit systems running Windows XP, Vista, and Windows 7. For detailed information on which products are supported on which platforms, please see the [ANSYS, Inc. Windows Installation Guide](#), which is available for download when you download ANSYS, Inc. products or as a printed manual if you receive installation media.

Verify that you have sufficient disk space to download, uncompress, and install the products you will be installing. Approximate disk space requirements for each product are shown in the [ANSYS, Inc. Windows Installation Guide](#).

If you have any problems with or questions about the installation process please log a Service Request on the ANSYS Customer Portal. A Systems Support Specialist will respond to assist you.

This tutorial is divided into three sets of instructions:

- Product Download Instructions: This set of instructions describes the download and extraction process.
- Product Installation with Client Licensing: This set of instructions describes the product installation, including the client licensing portion.
- License Manager Installation: This set of instructions describes the license manager installation.

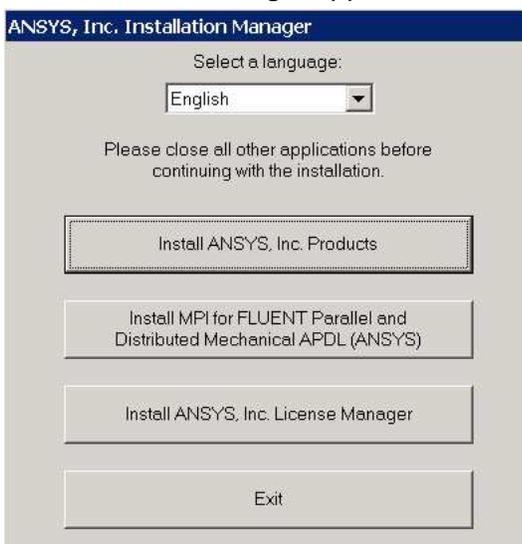
For this tutorial, the product and the license manager will be installed on separate Windows machines. *You must complete both the client licensing portion and the license manager installation in order to run ANSYS, Inc. products.*

Product Download Instructions

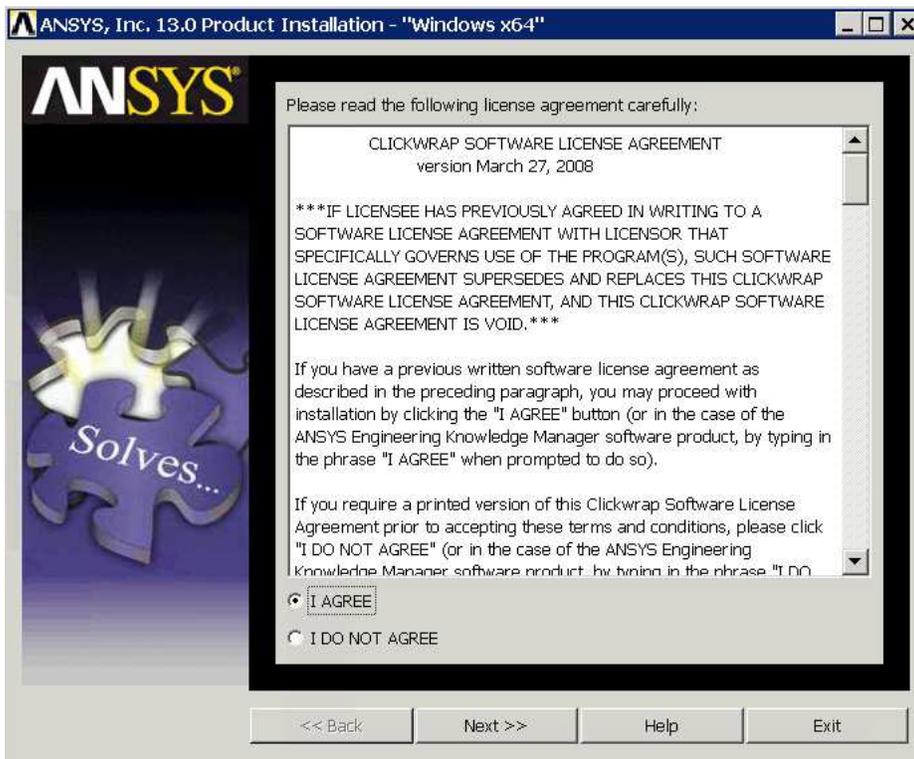
1. From the Customer Portal, click on **Download Software**.
2. The ANSYS Download Center Wizard page displays an overview of the download process. Review the overview and click **Next**.
3. For the Download Type, choose **Current Release**. Click **Next Step**.
4. Choose the hardware platform for which you want to download installation packages. You can select only one platform at a time. You will need to repeat the download procedure for each platform that you want to download. Click **Next Step**.
5. Choose the products you wish to download. Products are listed by product names that correspond to the licensing product names. You can choose to list products grouped by product group or alphabetically. All products that you currently have licensed are highlighted and pre-selected for your convenience. After you have selected all products that you wish to download, click **Next Step**.
6. The license manager, product, and documentation packages that you've selected to download are listed. Click on each link provided to begin the downloads.
7. After the downloads have completed, uncompress each package using standard uncompression utilities for your specific platform. We strongly recommend that you extract the files into a new, temporary directory.
8. Begin the product installation as described in the next section.

Product Installation with Client Licensing

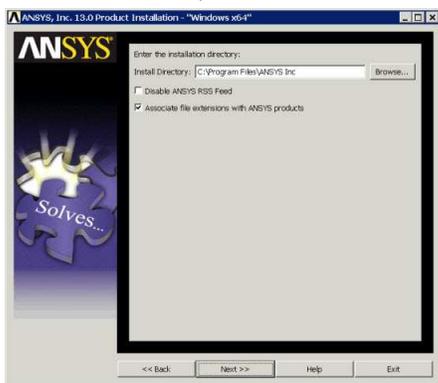
1. Navigate to the directory where you extracted the files. Double-click the `setup.exe` file. The ANSYS, Inc. Installation Manager appears.



2. Select the language you want to use. English is the default.
3. Click the **Install ANSYS, Inc. Products** button.
4. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, click **I Agree**. Click **Next**.



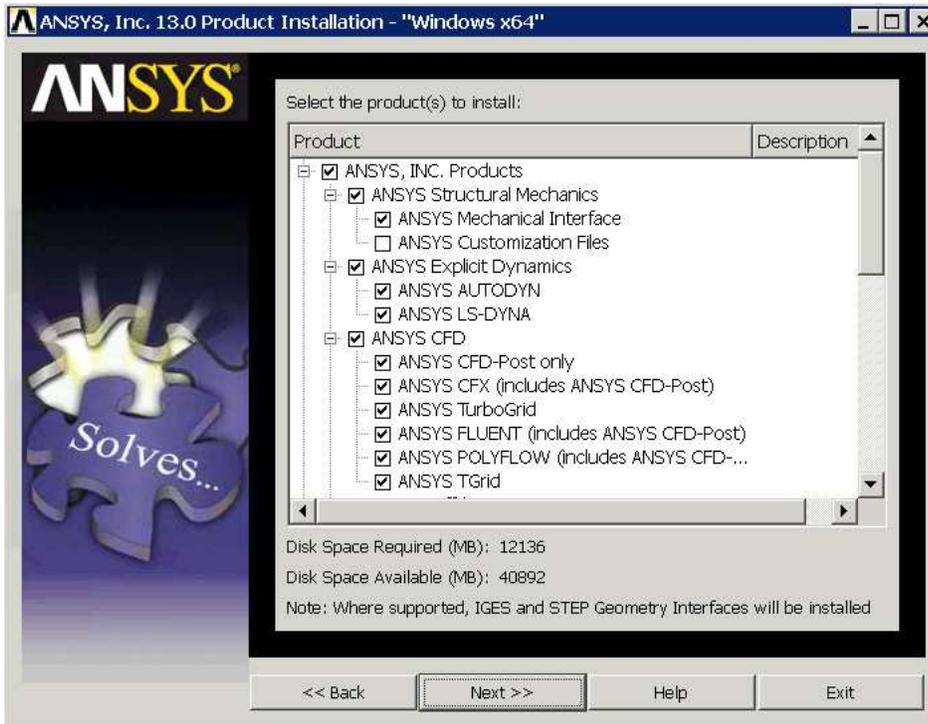
5. The directory where you wish to install the ANSYS, Inc. products is shown in the **Install Directory** field. You can install the products into any directory you wish, but you must have write permissions to the directory you choose. The default is C:\Program Files\ANSYS Inc. We recommend using the default directory.



Leave the **Associate file extensions with ANSYS products** checkbox marked. This allows you to double-click files from Windows Explorer to launch the various products.

Click **Next**.

6. All products available in the installation package(s) you downloaded are listed. You can select or deselect any combination of products. ANSYS Workbench is automatically installed with most ANSYS, Inc. products; there is no individual product selection for ANSYS Workbench.



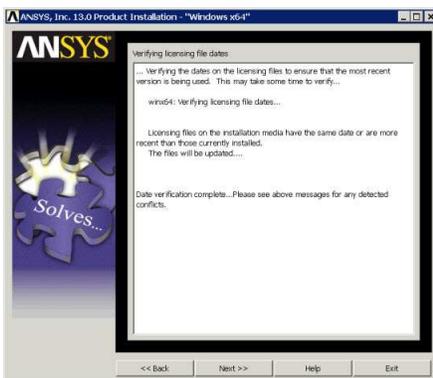
If you select Pro/ENGINEER and/or NX, you will have additional installation steps. This tutorial assumes you are not installing these CAD packages. If you do want to install these products, please see the section *Installing ANSYS, Inc. Products* in the [ANSYS, Inc. Windows Installation Guide](#) for more information on the additional steps required for Pro/ENGINEER and NX.

You will also see an estimate of the disk space required to install all of the selected components, and the disk space you have available. The actual amount of disk space required may be less, but if you choose to run the installation with insufficient disk space available, we strongly recommend that you review the log files at the end of the installation to verify that all products were installed correctly. Installation log files are written to the installation directory.

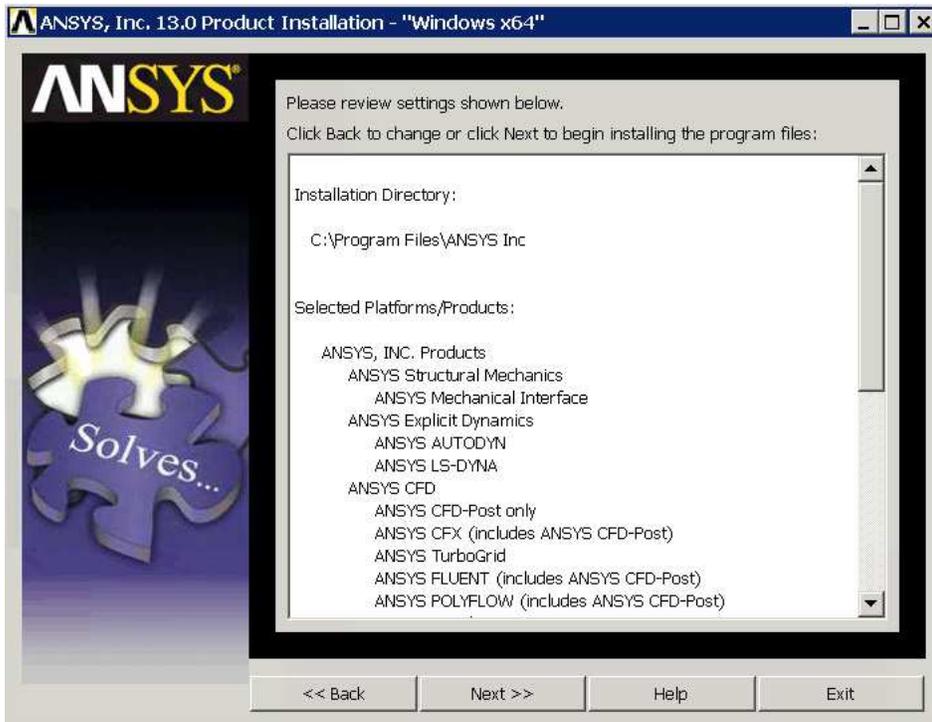
Select the products you want to install and click **Next**.

7. The dates on the licensing files being installed are compared to any that may already exist on your machine. The more recent set of files will be used. This tutorial assumes that this is the first time you are installing ANSYS, Inc. products on this machine; therefore, no licensing files will already exist and the files included with the installation package will be used.

Click **Next**.

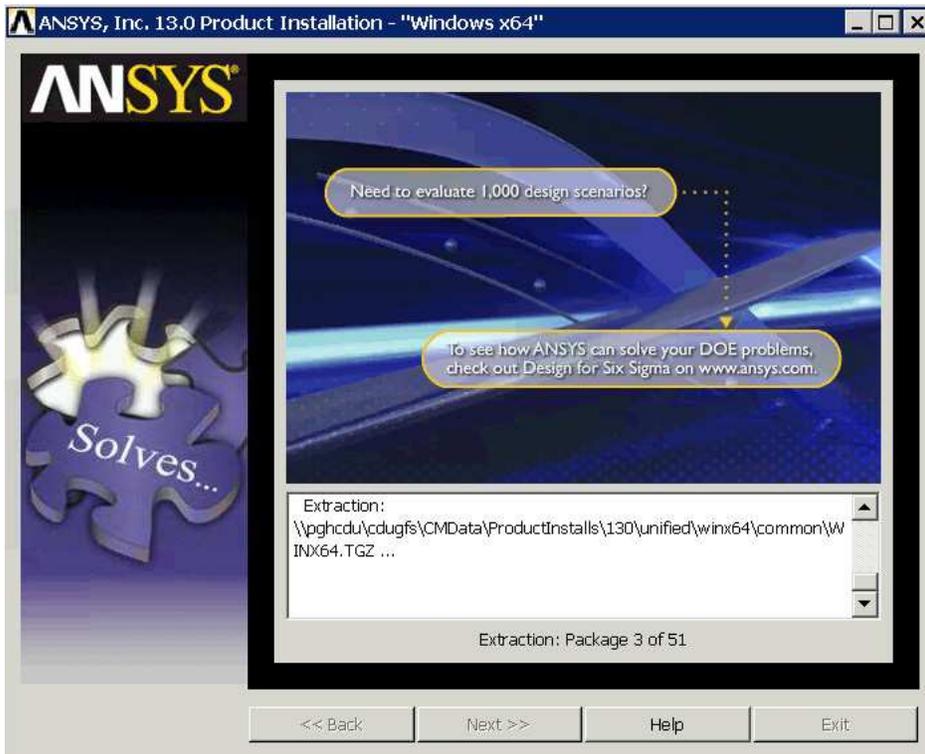


8. A summary screen appears that lists everything you have chosen to install. Review this list carefully to verify that it is correct. If it is not, click **Back** to go to earlier screens to correct the product selections.



When you have verified the product selections, click **Next**. When you click **Next** on this screen, the installation will begin.

9. As each product package is uncompressed and installed, it will display in the status window. Depending on the number of products you have selected, the installation time required could be lengthy. You will not be able to interrupt the installation process. Please be patient while the installation completes.



10. After all products are installed and configured, the client licensing needs to be run. Click **Next** to begin the client licensing configuration. A Licensing Client Installation Configuration window appears.

Licensing settings are recorded in a configuration file named `ansyslmd.ini`. This tutorial assumes that licensing has not previously been installed on this machine and therefore the `ansyslmd.ini` file does not exist. A new screen, **Specify the License Server Machine - Add Server Machine Specification** appears.

You will need to specify the port numbers to be used by the ANSYS licensing. Defaults are provided and will work in most cases. You may need to check with your IT department to confirm that the default port numbers are valid or to get different port numbers if necessary.

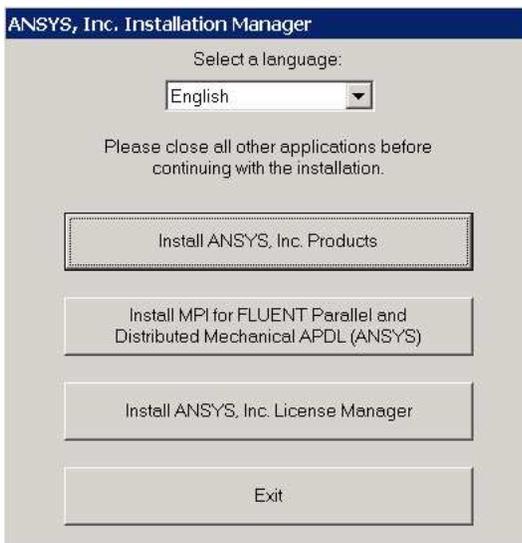
Then specify the hostname of the machine acting as a license server. Type in that server machine name and click **OK**.

11. When the client licensing installation has completed, click **Exit**.
12. The product installation window reappears with a message noting that the installation is complete. Click **Next**.
13. On the next screen, you can complete the survey or you can click **Finish** immediately to skip the survey and complete the product installation.

License Manager Installation

Because you will be using a network server, you must install and configure the ANSYS, Inc. License Manager on the server machine. The License Manager controls access to the ANSYS, Inc. products you have purchased.

1. On the server machine, navigate to the directory where the installation packages reside. Double-click the `setup.exe` file. The ANSYS, Inc. Installation Manager appears. Click **Install ANSYS, Inc. License Manager**.



2. You will see a warning stating that if the license manager is currently running, it will be shut down. This tutorial assumes that you have not previously installed the products or the licensing on this machine, and are not already running the ANSYS, Inc. License Manager on this machine. You may safely ignore this message and click **OK**.
3. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, click **I Agree**. Click **Next**.

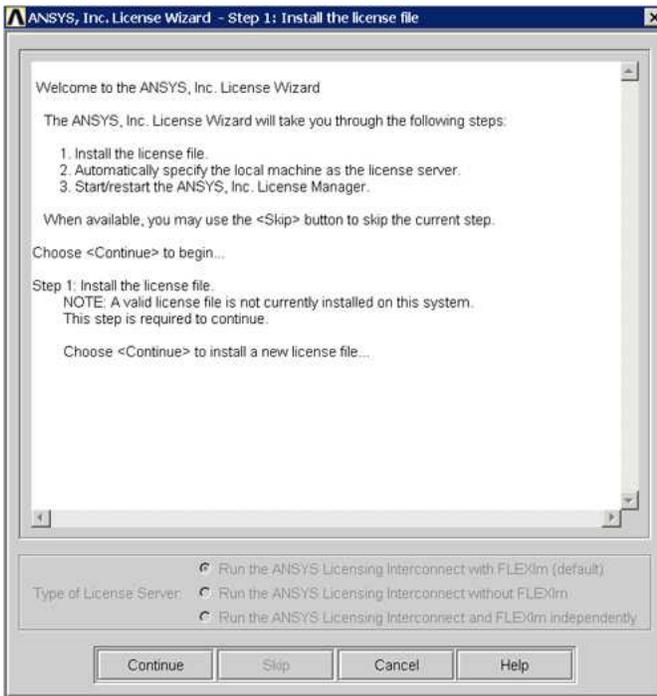
4. The directory where the license manager will be installed is shown in the **Install Directory** field. You cannot specify an installation directory on Windows; the license manager must be installed into C:\Program Files\ANSYS Inc. Click **Next**.
5. The ANSYS, Inc. License Manager is selected as the only product available to install. As with the product installation, the required and available disk space numbers are shown. Click **Next**.



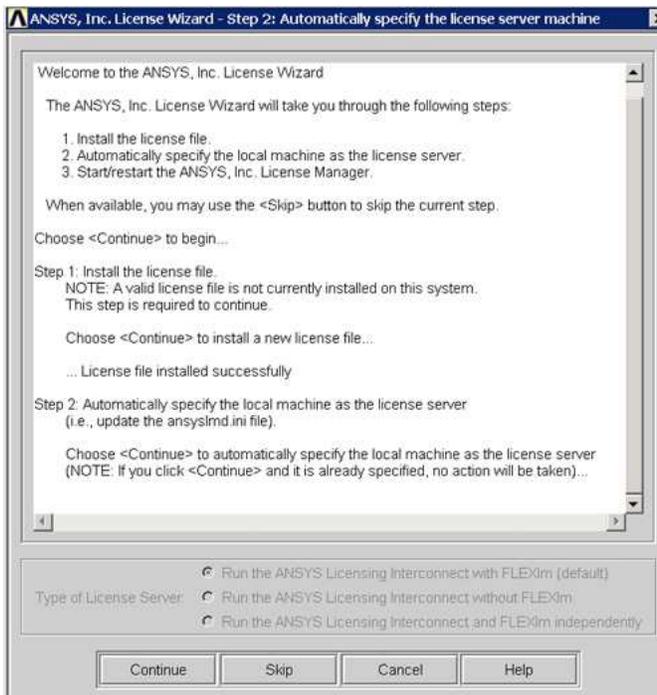
6. The licensing file dates are compared to ensure that the most recent version of the licensing files are being used. This tutorial assumes that this is the first time you are installing ANSYS, Inc. products on this machine; therefore, no licensing files will already exist and the files included with the installation package will be used. Click **Next**.
7. A summary screen appears that lists the products to be installed. Because this is a license manager installation, the ANSYS, Inc. License Manager is the only product listed.

Click **Next**. The license manager installation begins.

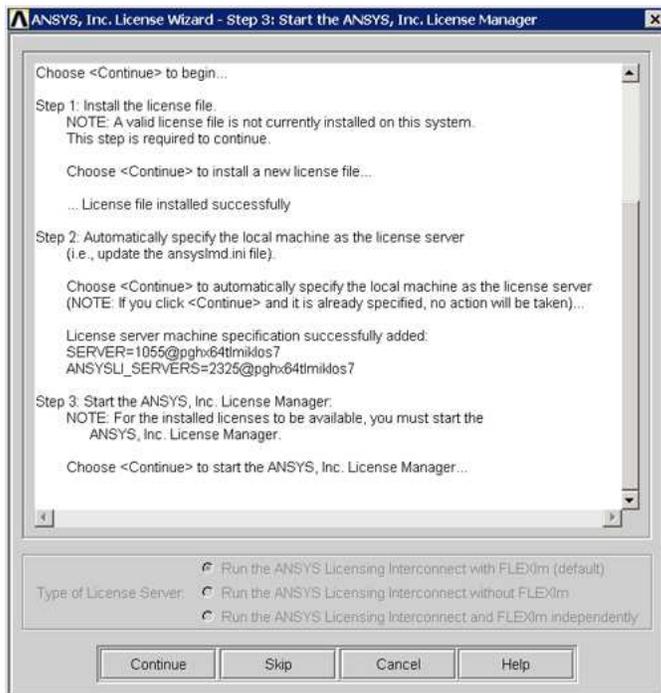
8. When the license manager installation completes, click **Next** to begin the license manager configuration.
9. The **License Server Installation Configuration** window appears, and several configuration steps will be completed. When the ANSYS, Inc. License Wizard appears, verify that the Type of License Server selected at the bottom of the screen is **Run the ANSYS Licensing Interconnect with FLEXlm (default)**. Click **Continue**.



10. The first step is to install a license file. Click **Continue**. You will then need to specify the location of the license file sent to you by ANSYS, Inc. Use the **Browse** button on the file locator dialog box to navigate to the location of the license file.
11. Click **Continue** to configure the local machine as the license server (the default action).



12. Click **Continue** to start the ANSYS, Inc. License Manager.



13. Click **Exit** to close the License Wizard.
14. Click **Exit** to close the **Licensing Server Installation Configuration** screen.
15. Click **Finish** to close the License Manager Installation screen.
16. Click **Exit** to close the ANSYS, Inc. Installation Manager.
17. From the machine on which the products were installed, choose **Start> Programs> ANSYS 13.0> <product>** to launch the ANSYS, Inc. products that you have installed.

