



# Installation and Licensing Documentation

---



## Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

---

---

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---



## ANSYS, Inc. Licensing Guide

---



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 13.0  
November 2010  
002910

ANSYS, Inc. is  
certified to ISO  
9001:2008.

---

## Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superceded by a newer version of this guide.

## Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---

# Table of Contents

Preface .....	vii
<b>1. Introduction .....</b>	<b>1</b>
1.1. Explanation of Licensing Terms .....	2
1.1.1. The FLEXlm License Server Manager (lmgrd) .....	2
1.1.2. The Vendor Daemon (ansyslmd) .....	2
1.1.3. ANSYS Licensing Interconnect (ansysli) .....	3
1.1.4. The License File .....	3
1.1.4.1. License Files Format .....	3
1.1.4.1.1. SERVER Lines .....	4
1.1.4.1.2. VENDOR Lines .....	4
1.1.4.1.3. INCREMENT Lines .....	5
1.1.4.1.4. Sample License Files .....	6
1.1.4.1.5. Recognizing an ANSYS, Inc. License File .....	6
1.1.5. The Application Programs .....	7
1.1.6. The License Server Machines .....	7
1.1.6.1. Selecting License Server Machines .....	7
1.1.6.2. Redundant Server Options .....	8
<b>2. Installing the ANSYS License Manager .....</b>	<b>11</b>
2.1. Communications Requirements .....	12
2.1.1. Configuring TCP/IP .....	12
2.1.1.1. Determining Whether TCP/IP Is Installed on a Microsoft Windows System .....	12
2.1.2. Changing the Default ANSYS Licensing Interconnect and FLEXlm Port Numbers .....	13
2.2. Installing the License Manager .....	13
2.2.1. License Manager Installation Instructions - Windows .....	14
2.2.2. License Manager Installation Instructions - UNIX/Linux .....	15
2.2.3. Silent License Manager Installation Instructions .....	16
2.2.4. Advanced Licensing Configuration Options .....	17
2.3. Post-Installation Instructions .....	21
2.3.1. Start the ANSYS License Manager at System Boot Time .....	21
2.3.2. Configuring the License Server Machine(s) .....	23
2.3.3. Modify License Manager Startup Options .....	24
2.3.4. Create a Group .....	24
2.3.4.1. Defining Group Restrictions for the Licensing Interconnect .....	24
2.3.5. Specify User Privileges .....	25
2.3.6. Specifying the License Server and License Files .....	25
2.3.7. Specifying Firewall Settings .....	26
2.3.8. Setting Up Redundant (Triad) Servers .....	26
<b>3. License Administration Using ANSLIC_ADMIN .....</b>	<b>29</b>
3.1. Using the ANSLIC_ADMIN Utility .....	29
3.1.1. Register License Server Machine Information .....	31
3.1.2. Display the License Server Machine Hostid .....	31
3.1.3. Run the License Wizard .....	31
3.1.4. Install the License File .....	32
3.1.5. Start the ANSYS, Inc. License Manager .....	33
3.1.6. Stop the ANSYS, Inc. License Manager .....	34
3.1.7. Reread the License Manager Settings .....	34
3.1.8. Specify the License Server Machine .....	35
3.1.8.1. Sample Scenario .....	37
3.1.9. Remove a Client License .....	37
3.1.10. Set License Preferences for <i>User</i> .....	38

3.1.11. Run the ANSYS Borrow Utility .....	39
3.1.11.1. Setting up License Borrowing .....	39
3.1.11.2. Running the Borrowing Utility .....	40
3.1.12. Set Site Preferences .....	41
3.1.12.1. Edit the FLEXlm Options File .....	41
3.1.12.1.1. The Options File Format .....	42
3.1.12.1.2. Sample Options File .....	44
3.1.12.2. Specify Product Order .....	45
3.1.12.3. Modify Startup Options .....	45
3.1.12.4. Specify License Servers to Cache .....	46
3.1.13. View Status/Diagnostic Options .....	47
3.1.13.1. Display the License Status .....	47
3.1.13.2. Display Queued Licenses .....	48
3.1.13.3. Display the Customer Number .....	49
3.1.13.4. View the ANSYS Licensing Interconnect Debug Log File .....	49
3.1.13.5. View the ANSYS FLEXlm License File .....	49
3.1.13.6. View the ANSYS FLEXlm Debug Log File .....	49
3.1.13.7. Gather Diagnostic Information .....	49
3.1.14. Uninstall the License Manager .....	50
<b>4. End-User Settings .....</b>	<b>51</b>
4.1. Client Environment Variable Settings .....	51
4.1.1. License Files Settings Precedence .....	51
4.2. Establishing User Licensing Preferences .....	52
4.2.1. ANSYS Workbench Licensing Methods .....	52
4.2.2. HPC Licensing .....	55
4.2.2.1. Specifying HPC License Order .....	56
4.3. Setting Up License Queuing .....	56
4.4. Using License Borrowing .....	57
<b>5. Troubleshooting .....</b>	<b>59</b>
5.1. Getting Additional License Debug Information .....	59
5.2. Gathering Diagnostic Information .....	60
5.3. Problem Situations .....	60
5.3.1. License Manager Will Not Start .....	60
5.3.2. License Manager Will Not Stop .....	61
5.3.3. License Manager Will Not Stop in a Three-Server Environment .....	61
5.3.4. The Application Does Not Show the Correct License(s) .....	62
5.3.5. I Do Not See an HPC Product Category in the Specify Product Order or the Set License Preferences for <i>User</i> Dialogs .....	62
5.3.6. FLEXlm Log File Shows Unexpected Messages When the License Manager Is Stopped .....	62
5.3.7. Unable to Check Out Licenses .....	63
5.3.8. Jobs Abort When License Manager Goes Down in Three-Server Environment .....	63
5.3.9. Licensing Log File Not Created .....	64
5.3.10. Queuing Does Not Work .....	64
5.3.11. The FLEXlm Utility Imutil Does Not Shut Down License Manager .....	64
5.3.12. The FLEXlm Utility Imcksum May Give Misleading Information .....	65
5.3.13. The Mechanical APDL Launcher is Excessively Slow to Start .....	65
5.3.14. Mechanical APDL Launcher is Not Using ANSYS130_PRODUCT Environment Variable Cor- rectly .....	65
5.3.15. Cannot Run a Product Listed in the Mechanical APDL Product Launcher .....	65
5.3.16. Remove a Client Option Does Not Work .....	65
5.3.17. No Licensing Interconnect or FLEXlm Path Available from Display the License Status Op- tion .....	66

5.3.18. Cannot Enter Data in Text Fields .....	66
5.3.19. Cannot See the Entire Run License Wizard Dialog Box .....	66
5.3.20. Licenses Remain Checked Out After Job Completes .....	66
5.4. Licensing Installation Process Errors .....	67
5.5. Licensing-Related Mechanical APDL Launcher Error Messages .....	67
5.6. Licensing Error Messages .....	68
5.7. ANSYS License Borrowing Errors .....	73
5.8. FLEXlm License Log File Errors .....	74
<b>6. Product Variable Table .....</b>	<b>77</b>
Glossary .....	85
Index .....	91

## List of Tables

1.1. Identifying License Files .....	7
2.1. Configuring TCP/IP .....	12
2.2. License Manager Automatic Startup Instructions .....	21
2.3. Removing License Manager Automatic Startup Instructions .....	23
6.1. Product/Feature Names for Licensed Products .....	77





# Preface

---

This document contains information for running the ANSYS, Inc. License Manager with all ANSYS, Inc. products. However, some information may pertain only to specific products/licensing levels, such as Mechanical APDL (ANSYS) or ANSYS FLUENT.

## Important Notice

If you wish to run multiple releases of ANSYS, Inc. software, you **MUST** install them chronologically (i.e., Release 12.1 followed by Release 13.0). If you install an earlier release after installing Release 13.0, you will encounter licensing issues that may prevent you from running any products/releases. If you need to install an earlier release after you have already installed Release 13.0, you **MUST** uninstall Release 13.0, then re-install the releases in order.

## Supported Hardware Platforms

This document details information about licensing ANSYS, Inc. products on the hardware platforms listed below. Not all products support all platforms listed below. Please refer to the installation documentation for platform specifics, including the platforms on which your specific product runs. The name in parentheses indicates the directory name for each platform (referred to as *<platform>* throughout this document).

- HP-UX Itanium 64(hpia64)
- IBM AIX 64 (aix64)
- Sun SPARC 64 (solus64)
- Sun Solaris x64 (solx64)
- Linux 32 (lin32)
- Linux Itanium 64 (linia64)
- Linux x64 (linx64)
- Windows x64 (winx64)
- Windows 32 (win32)

For specific operating system requirements, refer to the installation guide for the product and platform you are running.

## Intended Audience

The *ANSYS Licensing Guide* is intended for the person responsible for licensing administration of all ANSYS, Inc. products at a site. This person is typically the system administrator. End users of the product may also find this information useful, especially client options.

## Applicability

This guide applies to all products licensed with the ANSYS License Manager. See [Product Variable Table \(p. 77\)](#) for a list of products.

## Summary of New and Changed Features

Listed below is a summary of those licensing items that are either new or have been changed since the last release. For a list of all major new and changed features of any product, see the *Release Notes* document for that product.

- In order to run ANSYS Release 13.0 products, you must upgrade to the Release 13.0 license manager. The Release 13.0 License Manager will continue to support ANSYS licensing from prior ANSYS releases.
- At ANSYS Release 13.0, the license manager daemons (`lmgrd` and `ansyslmd`) have been upgraded to FLEXlm 11.8 (FLEXnet 11.8). We strongly recommend that you upgrade to this version of the license manager, regardless of whether you are upgrading to ANSYS Release 13.0. This version of the license manager supports our current licenses as well as provides support for FLEXlm Tamper Resistant Licensing (TRL) licenses. When you receive a license that contains TRL, you must be using this version of the license manager or you will not be able to run ANSYS, Inc. products.
- ANSYS, Inc. no longer requires you to choose between commercial and academic licenses when setting your license preferences, giving customers with both academic and commercial licenses greater flexibility in managing their licenses.
- The ANSYS, Inc. License Manager can now be installed silently using the `-silent` command line option. See *Silent License Manager Installation Instructions* (p. 16) for detailed information on running a silent license manager installation.
- The ANSYS Licensing Interconnect now supports the use of IP addresses in the FLEXlm options file for those settings that allow their use, such as EXCLUDE and INCLUDE.
- The **ANSLIC\_ADMIN** utility now includes a queued license tracking capability. Use the **Display Queued Licenses** option under **View Status/Diagnostic Options** to see a list of capabilities that are queued and awaiting availability, and the applicable licenses that are being used.
- We have enhanced many licensing messages to include more detailed information to assist you in resolving errors. We have also added more diagnostic information, such as enhancements to the **ANSLIC\_ADMIN's Gather Diagnostic Information** option under **View Status/Diagnostic Options**.

## Conventions Used in This Document

Computer prompts and responses and user input are printed using this font:

```
/ansys_inc/shared_files/licensing/lic_admin/anslic_admin
```

Lengthy user input lines that exceed the width of the page are listed in multiple lines, with the second and subsequent lines indented:

```
exec_path <drive>:\Program Files\Ansys Inc\ansys130\ac4  
         \bin\pro\<platform>\ac4pro130.exe
```

Wild card arguments and variables are *italicized*.

Commands appear in **bold face**.

---

### Note

Note paragraphs are introduced by the text *Note*. A note contains information that supplements the main topic being discussed.

**Caution**

Paragraphs that begin with the word “Caution” in bold, with the rest of the text in normal type, warn you about actions or situations that potentially may cause problems or unexpected behavior or results in ANSYS, Inc. products.

**Warning**

Paragraphs that begin with the word “Warning” in bold warn you about actions or situations that can shut down ANSYS, Inc. products, damage files, etc.



---

## Chapter 1: Introduction

---

ANSYS, Inc. uses the FLEXlm license manager (also called the FLEXnet License Manager) for all of its licensed products. FLEXlm is best known for its ability to allow software licenses to be available (or float) anywhere on a network. Floating licensing benefits both users and license administrators. Users can make more efficient use of fewer licenses by sharing them on the network. License administrators can control who uses the licensed application and the machine(s) where the licenses will be available. The format of the traffic between the ANSYS product(s) and the license manager is machine independent, allowing for heterogeneous networks. The license server machine and the computer running an application can be different hardware platforms or even different operating systems (Windows and UNIX/Linux, for example).

The actual communication between the ANSYS applications and `lmgrd` occurs through an intermediary process call the ANSYS Licensing Interconnect. The Licensing Interconnect is nearly transparent; you should not see any noticeable difference in your day-to-day operation of ANSYS products.

To get the full set of files necessary to run as a *server*, you will need to run the license server installation. The general licensing process is explained here; for detailed installation instructions, see [Installing the ANSYS License Manager \(p. 11\)](#). All files necessary to run as a *client* are automatically installed during the product installation.

### The Licensing Process

The licensing process for ANSYS, Inc. products is as follows:

1. Select the license server machine(s). See [Selecting License Server Machines \(p. 7\)](#) for guidelines.
2. Install and configure TCP/IP. See [Communications Requirements \(p. 12\)](#) for information on configuring TCP/IP.
3. Install the software. See [Installing the ANSYS License Manager \(p. 11\)](#), or see the appropriate installation manual for your product and platforms for detailed installation instructions.
4. Register the license server machine(s). See [Step 1 \(p. 23\)](#).
5. After you receive your license file, run the License Wizard. See [Run the License Wizard \(p. 31\)](#) for information on installing your license file.
6. Set up the licensing environment. See [Post-Installation Instructions \(p. 21\)](#).

### Compatibility with Other FLEXlm-Licensed Software

Because of FLEXlm's popularity, you may have FLEXlm licenses from more than one vendor.

The ANSYS License Manager uses FLEXlm 11.8. If you do have other FLEXlm-licensed products running FLEXlm 11.8 or higher, you can run them in conjunction with ANSYS products with no intervention or special configuration requirements. However, we do not support combining ANSYS, Inc. license files with other license files using one `lmgrd` license manager.

## Links to FLEXlm

For more information on using FLEXlm, please visit the FLEXlm web site at <http://www.flexerasoftware.com/>. You can refer to the *FLEXnet Licensing End User Guide* for more detailed information on all FLEXlm features. You can access this document from the **ANSLIC\_ADMIN Help** menu option **View the FLEXnet Licensing End User Guide**.

### 1.1. Explanation of Licensing Terms

The main components of our licensing are:

- The ANSYS, Inc. license manager, including:
  - Licensing Interconnect (`ansysli`) -- Also includes `ansysli_monitor`, which ensures that the license server is functioning correctly and attempts to correct the situation if the license server is not running or is unresponsive.
  - FLEXlm license server manager (`lmgrd`)
  - Vendor daemon (`ansyslmd`)
- License files
- Application program (e.g., Mechanical APDL (ANSYS) or ANSYS FLUENT)
- License server machines

These components are explained in more detail in the following sections.

The ANSYS, Inc. License Manager monitors what products are being run, who is running them, and from what computer system. It grants or denies permission to run products. When an ANSYS product begins, it requests permission to execute from a server. The server checks the pool of available license tasks and grants the request only if the required tasks are available. For each request that is granted, the required license tasks are removed from the pool. As each ANSYS product execution ends, these license tasks are returned to the pool of available tasks.

#### 1.1.1. The FLEXlm License Server Manager (lmgrd)

`lmgrd` is one of the FLEXlm components of the ANSYS, Inc. License Manager. Its primary purpose is to start and maintain the vendor daemon (`ansyslmd`). It also refers application checkout requests to the vendor daemon (`ansyslmd`).

`lmgrd` must be running on the license server machine to run ANSYS, Inc. products.

#### 1.1.2. The Vendor Daemon (ansyslmd)

Licenses are granted by running processes. Each vendor who has a FLEXlm-licensed product on the network has one process, called the vendor daemon. The vendor daemon keeps track of how many licenses are checked out, and who has them. If the vendor daemon terminates for any reason, all users for that product lose their licenses. The ANSYS vendor daemon is `ansyslmd`.

Client programs, including `ansysli`, communicate with `ansyslmd`, usually through TCP/IP network communications. The vendor daemon (`ansyslmd`) is started by `lmgrd`. The vendor daemon `ansyslmd` must be running on the license server machine to run ANSYS, Inc. products.

### 1.1.3. ANSYS Licensing Interconnect (ansysli)

The ANSYS Licensing Interconnect (`ansysli`) is an intermediary process that communicates with the FLEXlm component of the license manager to authenticate and process all license requests. In a typical configuration, the ANSYS Licensing Interconnect starts the FLEXlm component `lmgrd`, which then starts `ansyslmd`. We do offer advanced licensing configuration options for sites with well-established procedures; please see [Advanced Licensing Configuration Options \(p. 17\)](#) for more details.

With the Licensing Interconnect, your license file and license options file are still applicable and in effect. Using an intermediary process allows us to seamlessly integrate our full range of product offerings to continually offer you access to the latest products with minimal disruption to your licensing environment. It also allows us a platform on which to enhance important licensing features.

The Licensing Interconnect must be running on the license server machine to run ANSYS, Inc. products.

### 1.1.4. The License File

Licensing data is stored in a text file called the license file. The license file is created by ANSYS, Inc. and is installed by the license administrator. It contains information about the server machines and vendor daemon, and at least one line of data (called INCREMENT lines) for each licensed product. Each INCREMENT line contains a license key based on the data in that line, the hostids specified in the SERVER line(s), and other vendor-specific data.

The default and recommended location for the ANSYS license file (`license.dat`) is in the licensing directory. End users can override this location by setting the environment variable **ANSYSLMD\_LICENSE\_FILE** to point elsewhere. License files need to reside only on license server machines.

#### 1.1.4.1. License Files Format

License files usually begin with a SERVER line (or three SERVER lines for redundant triad servers) followed by a VENDOR line, followed by one or more INCREMENT lines.

You can modify only these data items in the license file:

- System host names on the SERVER line(s)
- Port numbers on the SERVER line(s)
- Vendor daemon file paths on the VENDOR line(s)
- Options file paths on the VENDOR line(s)
- Optional port numbers on the VENDOR line(s) (for firewall support only)

Long lines normally use the "\" line-continuation character to break up long lines. A space character must precede the line-continuation character.

---

#### Note

Everything else is used to compute the license key and should be entered exactly as supplied. All data in the license file is case sensitive, unless otherwise indicated.

### 1.1.4.1.1. SERVER Lines

The SERVER line specifies the hostname and hostid of the license server machine and the `lmgrd` port number. Normally a license file has one SERVER line. Three SERVER lines mean that you are using redundant servers. License administrators do not have the option of deleting SERVER lines from a license file because the hostids from all of the SERVER lines are computed into the license keys on every INCREMENT line.

The format of the SERVER line is:

```
SERVER host hostid 1055
```

where:

*host* is the license server machine host name or IP address; a string returned by the UNIX/Linux `hostname` or `uname -n` command. On Windows systems, `ipconfig /all` returns the host name. This can also be an IP address (nnn.nnn.nnn.nnn format).

*hostid* is usually the string returned by the **Display the License Server Hostid** option of the **ANSLIC\_ADMIN** utility.

*1055* is the FLEXlm TCP port number to use. ANSYS' default FLEXlm port number is 1055. This can be changed if port 1055 is already in use on your system.

Example:

```
SERVER enterprise 0122345 1055
```

---

#### Note

If you change the FLEXlm port number, then you must also change the FLEXlm port number that is specified in the `ansyslmd.ini` file (see [Specify the License Server Machine \(p. 35\)](#)) or in the **ANSYSLMD\_LICENSE\_FILE** environment variable on all client machines to match the port number specified in the SERVER line.

### 1.1.4.1.2. VENDOR Lines

The VENDOR line specifies the vendor daemon's name and path. `lmgrd` uses this line to start the vendor daemon, and the vendor daemon reads it to find its options file. The format of the VENDOR line is shown below.

```
VENDOR ansyslmd [vendor_daemon_path]
[[options=]options_file_path] [[port=]port]
```

where:

*ansyslmd* is the name of the ANSYS vendor daemon.

*vendor\_daemon\_path* is the path to the executable for this daemon. This path is optional. ANSYS, Inc. does not supply this field because `lmgrd` will look for the vendor daemon `ansyslmd` executable in the directory where `lmgrd` is located and all ANSYS products install both of these daemons into the same directory.



## Note

If you do supply this path and the path includes spaces, enclose the entire directory path in double quotes, as in the following example:

```
VENDOR ansyslmd "C:\Program Files\Ansys Inc\Shared Files\Licensing\win32"
```

*options\_file\_path* is the full path to the end-user options file for this daemon. FLEXlm does not require an options file. The options file need not be specified on this line. As long as the options file *ansyslmd.opt* is located in the same directory as the license file (the licensing directory), the vendor daemon will automatically find and use it. The **Edit the FLEXlm Options File** option of the **ANSLIC\_ADMIN** utility will put the options file in the correct directory location.

If the directory path includes spaces, enclose the entire directory path in double quotes, as in the following example:

```
VENDOR ansyslmd options="C:\Program Files\Ansys Inc\Shared Files\Licensing\ansyslmd.opt"
```

*port* is the vendor daemon port number. Note: This is for firewall support only and is otherwise not recommended. In the following example, ##### would be replaced with the port number you choose:

```
VENDOR ansyslmd options=/ansys_inc/shared_files/licensing/ansyslmd.opt port=#####
```

### 1.1.4.1.3. INCREMENT Lines

An INCREMENT line describes the license to use a product. The syntax of the INCREMENT line is:

```
INCREMENT feature ansyslmd feat_version exp_date #lic key  
[VENDOR_STRING="vendor_str"] [ISSUED="..."] [START="..."]
```

where:

*feature* is the name representing the product/capability being licensed.

*ansyslmd* is the name of the ANSYS vendor daemon; also found in the VENDOR line. The specified daemon serves this feature.

*feat\_version* is the latest build date of this feature that is supported by this license. For paid-up customers, this is usually set to the expiration date of the maintenance agreement. The value of 9999.9999 is used when this field is not applicable.

*exp\_date* is the expiration date of license, e.g., 7-may-2010.

*#lic* is the number of concurrent licenses for this feature.

*key* is the encryption key for this INCREMENT line.

Additional fields may follow. See the *FLEXnet Licensing End User Guide* (accessible from the **ANSLIC\_ADMIN** utility) for more information. Note that ANSYS, Inc. may not support all options described in the *FLEXnet Licensing End User Guide*.

**VENDOR\_STRING Keywords** ANSYS, Inc. has created two VENDOR\_STRING keywords:

- timezones

- customer

The `timezones` keyword specifies the time zone in which the client needs to be running. The absence of the `timezones` keyword indicates that a client can run in any time zone. The format is **`timezones:X[,Y,Z,...]`**, where X is a numeric value from 0-23, beginning with GMT. Values move to the east in one hour increments. For example, 0 represents GMT, and 19 represents EST. Multiple time zones are separated by commas (,): **`timezones:17,18,19`**. This keyword cannot be modified by the user.

The `customer` keyword specifies the customer number. The format is **`customer:<cust_num>`**, where `cust_num` is the customer's number. This keyword cannot be modified by the user.

#### 1.1.4.1.4. Sample License Files

A sample license file is shown here. This file is for 15 ANSYS Mechanical tasks and 12 ANSYS CFD tasks.

```
SERVER gagh 690daec6 1055
VENDOR ansyslmd
INCREMENT ansys ansyslmd 9999.9999 30-sep-2011 15 8C59A481BA50 \
  VENDOR_STRING=customer:00012345 ISSUED=10-sep-2011 \
  START=10-sep-2010
INCREMENT acfd ansyslmd 9999.9999 30-sep-2010 12 47A354CA1291 \
  VENDOR_STRING=customer:00012345 ISSUED=10-sep-2011 \
  START=10-sep-2010
```

where:

- *gagh* is the hostname of the license server
- *690daec6* is the hostid
- *1055* is the FLEXlm port number
- *ansyslmd* is the vendor daemon
- *ansys* is the feature representing ANSYS Mechanical and *acfd* is the feature representing ANSYS CFD.
- *9999.9999* indicates that the maintenance agreement is not applicable. Otherwise, this the highest supported build date for the product. For Mechanical APDL (ANSYS) only, you can view the build date by running Mechanical APDL (ANSYS) with the **-v** command option.
- *30-sep-2011* is the expiration date
- *15* and *12* are the number of tasks for ANSYS Mechanical and ANSYS CFD, respectively
- *8C59A481BA50* and *47A354CA1291* are encryption keys for ANSYS Mechanical and ANSYS CFD, respectively
- *customer:00012345* is the customer number
- *ISSUED=10-sep-2010* is the date the license was created
- *START=10-sep-2010* is the start date

#### 1.1.4.1.5. Recognizing an ANSYS, Inc. License File

If you receive a license file and are not sure if it is an ANSYS, Inc. license file, you can determine if it is by looking at the contents of the license file. If it is an ANSYS, Inc. license file, then:

- In the line beginning with the word `VENDOR`, the next field/item is `ansyslmd`.
- In the line(s) beginning with the word `INCREMENT`, the third field/item is `ansyslmd`.

ANSYS, Inc. supplies some licenses for other applications that use different license managers. These license files are not compatible with the ANSYS License Manager. The applications and how to recognize their license

files are explained below. Look for the words shown in the **VENDOR**, **DAEMON**, **INCREMENT**, and **FEATURE** lines for each application. For these license files, you must use their respective license manager tools.

**Table 1.1 Identifying License Files**

<b>Application</b>	<b>VENDOR</b>	<b>DAEMON</b>	<b>INCREMENT</b>	<b>FEATURE</b>
ANSYS CFX		CFDS	CFDS	
ANSYS ICEM CFD	ICEM_CFD			ICEM_CFD
ANSYS AUTODYN	AUTODYN			AUTODYN
ASAS, AQWA	ASASLM			ASASLM
TAS, TASPCB, PTD	thermal			thermal
FLUENT, POLYFLOW, Icepak		FluentLM	FluentLM	
ANSOFT	ansoftd		ansoftd	

### 1.1.5. The Application Programs

Application programs are software programs such as Mechanical APDL (ANSYS), CFX, FLUENT, etc. ANSYS application programs need to be able to communicate with the ANSYS License Manager.

### 1.1.6. The License Server Machines

License administration is controlled from specific computers on the network called license server machines. License server machines run the license manager, which controls access to all licenses.

The server machine or machines are designated by you—the end user. You have the option of designating one server or three servers. In a one-server network, if the server machine goes down, the licenses are no longer available for use until the server machine is back in service. In a three-server (redundant triad) network, as long as two of the three machines are still running, the licenses are still available for use.

The *master* server actually controls the license administration. If a network has only one server machine, then that server machine is automatically the master server. In a three server environment, the order of the **SERVER** lines in the license file determines which server is the master. *The order of the servers must match on all machines in a three server environment.* The first is the master, the second is the first backup, etc. If the order of the **SERVER** lines does not match on the three servers, then the servers will attempt to determine the master server; however, this attempt may not be successful. In a three-server network, if the master server is unavailable, then the first backup acts as the master.

You must make sure the order of the **SERVER** lines is consistent between redundant servers; otherwise, re-connections may fail.

#### 1.1.6.1. Selecting License Server Machines

Before running any ANSYS, Inc. software, you must select which machine(s) will be license servers, and provide the **hostid** and **hostname** of those machines to ANSYS, Inc. Use the **Register License Server Information** option of the **ANSLIC\_ADMIN** utility to capture the necessary system information and create the `licserver.info` file, which then needs to be forwarded to your ANSYS sales representative. If the ANSYS license manager software is not available for installation on a license server machine, you can download a utility to capture license server machine information from [http://www.ansys.com/services/license\\_hostid](http://www.ansys.com/services/license_hostid).

You need to select the computer systems that will act as server machines before we can supply you with the licenses that are required to activate your licensed product(s). Information about the server machines is used to generate the necessary license key(s).

Consider the following points when deciding which computer(s) will be used as server(s):

- All files used in conjunction with the license manager software must be located on a disk that is physically local to the server computer(s).
- Computers must have a high-speed, reliable Ethernet connection.
- Computers that experience extremely high levels of network traffic or processing lags due to high CPU and/or I/O usage are poor candidates for servers.
- Do not use computers that are frequently rebooted as servers.
- The license server machine must have a static IP address.
- We do not allow the use of wide area networks (WANs) for license servers (with the standard ANSYS contract).
- You cannot use DHCP for license server machines.
- If using a three-server network, we recommend that you choose three machines that are of the same platform type (that is, three Linux, three Windows machines, etc.).
- If using a three-server network, we highly recommend that all three server machines be on the same subnet in the same physical location.

If these guidelines are not followed, the ability of the ANSYS License Manager to perform consistently will be compromised.

---

### Caution

Do not change the date on the license server machine. Doing so will prohibit the ANSYS product from running. Restoring the system to its original state prior to the date change may require significant effort.

#### 1.1.6.2. Redundant Server Options

Throughout this document, we use the term “three-server network” when referring to redundant triad servers. Redundant server setup is a network configuration where multiple machines are designated as license servers. Redundancy can be achieved in two ways:

- Any number of license server machines can be running independently. The total number of licenses is split between each license server. For example, if you are licensed for 20 tasks of a certain product, and you have two license server machines, each license server machine will serve ten licenses. In this example, if one of these machines fails, only ten licenses will be available.
- Three different machines can be selected to work together in tandem, where two of the three must be running at all times. These three license server machines work from a single set of licenses. This option is not recommended.

We recommend the first option if your site requires redundancy. Be sure to review [Selecting License Server Machines \(p. 7\)](#) for guidelines and special considerations when choosing license servers. Single license server networks are usually sufficient for most sites.

**Note**

If you are running redundant servers, you should have the license file (as well as the entire licensing directory) installed locally on each license server. If you do not, you lose all the advantages of having redundant servers, since the file server holding these files becomes a single point of failure.



---

## Chapter 2: Installing the ANSYS License Manager

---

The ANSYS, Inc. License Manager installation includes both a server and a client component. You must install the license manager software on your server machine(s) at this release in order to be able to run ANSYS, Inc. products. The updates at this release include the new ANSYS Licensing Interconnect, the latest daemons/services, and the latest **ANSLIC\_ADMIN** updates. If you do not install the license manager at this release, your ANSYS, Inc. products will not run. You will find detailed instructions for installing the license manager on your server machines later in this chapter. References in this chapter to licensing installation refer to installing all necessary licensing components on a machine designated as a license server machine.

Once the updated License Manager is installed on your server machine(s) and you have installed the product(s), the client component will run successfully. The client component is installed automatically during a product installation; users running as a client will not need to perform any additional installation steps to successfully run the product(s).

**UNIX/Linux Server Installation Notes:** If you are installing the License Manager on the same machine where the product will be run, use the same top level directory path for both the product installation and the License Manager installation. We strongly recommend that you install the licensing files relative to the product installation directory to avoid manually editing the product run scripts.

A typical product installation on a UNIX/Linux machine allows you to install the product anywhere on the system and then creates a symbolic link from the installation directory to `/ansys_inc`. If you did not set the symbolic link during the product installation, or if you installed the licensing files somewhere other than relative to the installation directory, replace all references to `/ansys_inc` in this guide with the name of the installation directory you used. Any reference throughout this manual to the licensing directory on UNIX/Linux platforms means `/ansys_inc/shared_files/licensing`.

**Windows License Server Installation Notes:** The licensing installation on a Windows machine installs all necessary licensing files into the `\Program Files\ANSYS Inc\Shared Files` directory, located on the same drive as the operating system, regardless of where your product installation resides. You cannot change this location. You must have administrative privileges on your machine to install the licensing files. The license manager components are shared across all ANSYS, Inc. products and need to reside in a fixed location.

Any reference throughout this manual to the licensing directory on Windows platforms means `\Program Files\Ansys Inc\Shared Files\Licensing`, located on the same drive as the operating system.

**Windows/UNIX/Linux License Client Installation Notes:** The license client files are automatically installed into the `\Shared Files\Licensing` (Windows) or the `/shared_files/licensing` (UNIX/Linux) subdirectory of the product installation at the end of the product installation.

## 2.1. Communications Requirements

### 2.1.1. Configuring TCP/IP

TCP/IP needs to be configured and started for any ANSYS, Inc. product and the license manager to be able to run. You should consult your network administrator for assistance with this configuration. The TCP/IP protocol must be installed on any machine on which you want to run an ANSYS product.

TCP/IP is supplied as part of the UNIX/Linux operating system. *Table 2.1: Configuring TCP/IP* (p. 12) specifies the system utility used to configure TCP/IP on the various hardware platforms. You should consult your network administrator for assistance with this configuration.

---

#### Note

Linux systems require an Ethernet card.

**Table 2.1 Configuring TCP/IP**

Hardware Platform	TCP/IP Configuration Utility
HP	<b>sam</b>
IBM	<b>smit tcpip</b>
Sun (including x64)	System Management Console ( <code>/usr/sadm/bin/smc</code> )
Linux (Red Hat Enterprise) (32- and 64-bit)	<b>system-config-network</b>
Linux (SUSE Enterprise) (32- and 64-bit)	<b>yast</b>

For Windows systems, the TCP/IP protocol is included as part of the operating system and is typically installed by default. If you do need to install TCP/IP, remember that it must be bound to a network adapter.

On machines that are connected to an internal network, TCP/IP must be bound to a network card such as an Ethernet adapter. The vast majority of systems using TCP/IP will fall into this category.

On machines that connect to the Internet or corporate intranet through a modem, TCP/IP can be bound to a dial-up connection.

#### 2.1.1.1. Determining Whether TCP/IP Is Installed on a Microsoft Windows System

To determine if TCP/IP is installed on your system, open the Control Panel. Continue as follows based upon the machine's operating system:

- Windows XP -- Open Network Connections, highlight Local Area Connection and right-mouse click. Select Properties. Internet Protocol (TCP/IP) should be listed.
- Windows 7/Vista -- Select **Control Panel> Network and Sharing Center**. For Local Area Connection, click **View Status**. Click **Properties**. Internet Protocol (TCP/IP) should be listed.

If TCP/IP is installed, you must determine whether it is bound to a network adapter card or a dial-up connection. A network card or a Dial-Up Adapter will be shown under **Connect Using:**.



**Caution**

If your computer is connected to a network, it is *highly recommended* that you contact your Information Technology Department before installing or modifying TCP/IP on your machine.

## 2.1.2. Changing the Default ANSYS Licensing Interconnect and FLEXlm Port Numbers

A port number specifies the communications channel by which two or more processes can communicate. ANSYS uses 2325 as the default port number for the ANSYS Licensing Interconnect and 1055 as the default port number for the FLEXlm component of the license manager. `ansyslmd` also uses a port designated by the operating system, unless one is manually specified in the license file on the VENDOR line. If you encounter a conflict in port numbers, you can change the default by modifying all of the following files:

**ANSYS Licensing Interconnect Port Number** The ANSYS Licensing Interconnect port number is defined in the `ansyslmd.ini` file. You can change this file by selecting **Specify the License Server Machine** on the **ANSLIC\_ADMIN** utility. Enter the new port number in the **ANSYS Licensing Interconnect Port Number** field. The Licensing Interconnect port number may also be specified via the **ANSYSLI\_SERVERS** environment variable, if set.

**FLEXlm Port Number** To change the default FLEXlm port number, you need to change the following files:

- On the license server machine(s): the port number listed on the SERVER line in the license file (`license.dat`).
- On the client machine(s): the port number listed in the `ansyslmd.ini` file. Use the **Specify the License Server Machine** option of the **ANSLIC\_ADMIN** utility to change the `ansyslmd.ini` file. The FLEXlm port number may also be specified via the **ANSYSLMD\_LICENSE\_FILE** environment variable, if set.

**ansyslmd Port Number** The `ansyslmd` daemon uses a port designated by the operating system, unless one is manually specified in the license file on the VENDOR line. See [VENDOR Lines \(p. 4\)](#) for information on specifying this port number on the VENDOR line. You should need to specify this port number manually only if using a firewall.

For information on firewall settings, see [Specifying Firewall Settings \(p. 26\)](#).

## 2.2. Installing the License Manager

You must have administrator privileges to install the ANSYS, Inc. License Manager on Windows systems. On UNIX/Linux systems, you can install using the `-noroot` option, but you may encounter permission problems during the installation. To proceed with the installation, the current license manager must be shut down if it's running. The installation process will shut down the license manager; however, you should verify that no users will be affected while you run the installation.

The licensing installation process will install all necessary files, including the newest **ANSLIC\_ADMIN** files, as well as set the **ANSYSLIC\_DIR** environment variable in the product run scripts.

You cannot perform a platform license server installation (i.e., installing files for a different machine type than you're running on) on Windows systems.

New licenses for existing customers may not be supplied with the ANSYS Release 13.0 package. They will be supplied when the current licenses expire or when TECS expires.

Run the license manager installation on all machines that will act as license servers.

If you encounter a problem during the licensing installation process that results in a failure or abort, or if you are concerned that the licensing installation did not complete correctly, try running the **Complete Unfinished Licensing Installation Configuration** option from the **ANSLIC\_ADMIN** utility's **Tools** menu.

### 2.2.1. License Manager Installation Instructions - Windows

Follow the instructions below to install the ANSYS License Manager on Windows systems that will act as license servers. Client licensing is installed automatically when the product is installed; you do not have to take any further steps to run as a client if you have installed a product.

1. Run `setup.exe`. Select **Install ANSYS, Inc. License Manager**.

If you downloaded only the license manager installation, run `setupLM.exe`.

2. Select a language.
3. You will be notified that the license manager, if running, will be shut down. Click **OK**.
4. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
5. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. License Manager. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. Click **Next** to continue.
6. The mount directory (location where the installation is located) is specified. You should accept the default. The installation directory is set to `<OS_Drive>\Program Files\ANSYS Inc\Shared Files`. You cannot change this directory. Click **Next** to continue.
7. Select the component you want to install. The amount of disk space required and the disk space available appear at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. However, if you choose to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully.

Click **Next** to continue.

8. A licensing file date verification summary appears. If no conflicts are found between the installation files and the files already on your machine from a previous installation, the message **Date verification complete** appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.
9. A summary of the selected installation data appears. Information shown includes platform, installation directory, and product. Review the information carefully, and if correct, click **Next** to continue the installation.

The ANSYS License Manager is now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation.

10. The **Licensing Server Installation Configuration** box appears. As the license manager is configured, progress messages appear in the box.

11. The License Wizard will be launched. This wizard walks you through the process of installing or updating a license file, specifying the license server(s) (which updates the `ansyslmd.ini` file), and starting the license manager. The wizard will prompt you for the necessary information at each step. During this process, the license manager will be shut down if it is running. Be aware that this can impact any users currently running using the license manager.

Click **Continue** on the License Wizard to begin, and follow the instructions on the screen.

If you are not performing a default License Server installation or require more information on Types of License Servers, please refer to [Advanced Licensing Configuration Options \(p. 17\)](#).

12. When the License Wizard is complete, click **Finish** on the wizard screen and then click **Finish** again on the Licensing Installation Configuration Log screen.
13. When the license manager installation is complete, click **Finish**. A new Start Menu item named **ANSYS, Inc. License Manager** will be created automatically. It will include selections for the **ANSLIC\_ADMIN** server utility, the *ANSYS, Inc. Licensing Guide*, and the *FLEXnet Licensing End User Guide*.

---

### Note

Client machines will also include an **ANSLIC\_ADMIN-Client** option under the ANSYS Release 13.0 Start Menu item. The client-only version of **ANSLIC\_ADMIN** offers access to a limited number of **ANSLIC\_ADMIN** features, such as user configuration and status/reporting options.

## 2.2.2. License Manager Installation Instructions - UNIX/Linux

Follow the instructions below to install the ANSYS License Manager on UNIX/Linux systems that will act as license servers. Client licensing is installed automatically when the product is installed; you do not have to take any further steps to run as a client if you have installed a product.

1. Run `INSTALL.LM` to launch the license manager installation. If you downloaded the license manager installation package, this file will reside in the directory where you untarred the files. If you are running from a DVD, this file will reside in the top level of the DVD.
2. Select a language.
3. You will be notified that the license manager, if running, will be shut down. Click **OK**.
4. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
5. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. License Manager. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. Click **Next** to continue.
6. The mount directory (location where the installation is located) is specified. You should accept the default. You can also accept the default installation directory or specify an alternate path and directory name where the products are to be installed.

We recommend that you also set the symbolic link `/ansys_inc` to the directory where the ANSYS, Inc. product is installed. The `/ansys_inc` symbolic link is set by default. The symbolic link option is available only if you are installing as root. If you chose to set the symbolic link during the product installation, you should set it here as well.

Click **Next** to continue.

7. Select the component you want to install. The amount of disk space required and the disk space available appear at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. However, if you choose to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully.

Click **Next** to continue.

8. A licensing file date verification summary appears. If no conflicts are found between the installation files and the files already on your machine from a previous installation, the message **No conflicts found** appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.
9. A summary of the selected installation data appears. Information shown includes platform, installation directory, and product. Review the information carefully, and if correct, click **Next** to continue the installation.

The ANSYS License Manager is now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the license manager.

10. The **Licensing Server Installation Configuration** box appears. As the license manager is installed, progress messages appear in the box.
11. The License Wizard will be launched. This wizard walks you through the process of installing or updating a license file, specifying the license server(s) (which updates the `ansyslmd.ini` file), and starting the license manager. The wizard will prompt you for the necessary information at each step. During this process, the license manager will be shut down if it is running. Be aware that this can impact any users currently running using the license manager.

Click **Continue** on the License Wizard to begin, and follow the instructions on the screen.

If you are not performing a default License Server installation or require more information on Types of License Servers, please refer to [Advanced Licensing Configuration Options \(p. 17\)](#).

12. When the License Wizard is complete, click **Finish** on the wizard screen and then click **Finish** again on the Licensing Installation Configuration Log screen.
13. When the license manager installation is complete, click **Finish**.

### 2.2.3. Silent License Manager Installation Instructions

You can deploy an ANSYS, Inc. License Manager installation in silent mode on both Windows and UNIX/Linux systems. Client licensing is installed automatically when the product is installed; you do not have to take any further steps to run as a client if you have installed a product. To run a silent product installation, including the client licensing, please see *Silent Product and License Manager Installation* in the *ANSYS, Inc. Installation Guide* for your platform.

To silently install the ANSYS License Manager on Windows systems that will act as license servers, you must run the `setupLM.exe` with the `-silent` option:

```
setupLM.exe -silent
```

To install the ANSYS License Manager on UNIX/Linux systems that will act as license servers, you must run the `INSTALL.LM` with the `-silent` option:

```
INSTALL.LM -silent -install_dir path
```

The silent license manager installation is valid only for the default Licensing Configuration option **Run the ANSYS Licensing Interconnect with FLEXlm**.

You can use the following arguments when running a silent license manager installation:

-silent	Initiates a silent installation.
-install_dir path	Specifies the directory to which the license manager is to be installed (UNIX/Linux only). If you want to install to the default location, you can omit the -install_dir argument. The default location on UNIX/Linux is /ansys_inc if the symbolic link is set; otherwise, it will default to /usr/ansys_inc. For the license manager installation on Windows, it will always be installed to <os drive>:\Program Files\ANSYS Inc\.
-licfilepath path	Specifies the location of the license file to install. If the path is not specified or if the path is the same as the existing license file, the license file will not be installed. For the license file path on Windows, you must enclose the path in quotes if you have spaces in the pathname.

During a silent installation, any messages will be written to the licensing installation log file, `install_lic-config.log`, located in the installation directory. In rare circumstances with a silent licensing installation, the licensing installation messages may not be written to the `install_licconfig.log` (for example, if the silent licensing installation aborts); in these cases, you may find error messages in the `.ansys_install_temp_licconfig_<user>_<index>.log` file, located in %TEMP% on Windows or in /var/tmp on UNIX/Linux.

### Caution

A silent license manager installation could shut down the ANSYS, Inc. License Manager, affecting other users who are using that license server machine.

If you are running a silent client installation, you can specify license server information as well.

-licserver- info	Specifies information to be used by the client for the license server. Valid only in conjunction with a silent product installation ( <code>setup.exe</code> or <code>INSTALL</code> ). Please see the <i>ANSYS, Inc. Installation Guide</i> for your platform for details on running a silent product installation, including client licensing.
---------------------	--

## 2.2.4. Advanced Licensing Configuration Options

In addition to the default configuration, ANSYS, Inc. offers two advanced licensing configuration options when you run the License Wizard:

- **Run the ANSYS Licensing Interconnect without FLEXlm** -- Use this option if you want to run a local copy of the ANSYS Licensing Interconnect for better performance, such as if your server machine is in a remote location, or to manage the server load better if your license server machine serves many users. With this option, FLEXlm licenses will be taken from another server machine.
- **Run the ANSYS Licensing Interconnect and FLEXlm independently** -- Use this option if you want to manage your FLEXlm licenses independently of the ANSYS, Inc. tools (for example, using Flexera Software's FLEXnet Manager to manage multiple companies' product licenses). This option should be used only by experienced users with well-established licensing procedures. Use this option to run the Licensing

Interconnect on additional systems other than your license server; your site must still have the Licensing Interconnect running on the license server machine.

These options are available only for license server machines and are available only with a license manager installation or via the server **ANSLIC\_ADMIN** option **Run the License Wizard** after a license manager installation has been run on this machine.

You can select either of these options by clicking the appropriate button in the **Type of License Server** field in the License Wizard. You can also set up these types of license servers manually. Instructions for both using the wizard and configuring manually are provided in the following sections.

If you have already configured your licensing and are changing to a different configuration option, you must stop and restart the ANSYS, Inc. License Manager for these changes to take effect. Rereading the license manager settings will not be sufficient.

### ***Run the ANSYS Licensing Interconnect without FLEXlm***

To use the License Wizard to set up a license server that will run the Licensing Interconnect without FLEXlm, follow the steps below.

---

#### **Important**

When using this advanced option, the license server holding the FLEXlm licenses must also have the Licensing Interconnect installed and running.

1. Select **Run the License Wizard** from the **ANSLIC\_ADMIN** utility.
2. In the **Type of License Server** field, select **Run the ANSYS Licensing Interconnect without FLEXlm**. Click **Continue** on the message box.
3. Click **Continue** to begin the configuration process.
4. Click **Continue** to specify the local machine as the primary Licensing Interconnect.
5. If you have not yet specified license server machines, you will need to specify the server from which you will check out ANSYS, Inc. FLEXlm licenses. If you have already specified license server machines, that specification will be used. If you need to change the server specification, use the **Specify the License Server Machine** option of **ANSLIC\_ADMIN** after you have completed the wizard.

Click **Continue**.

6. You will now be asked to start or restart the license manager. Click **Continue**.
7. Click **Exit**.

You can also configure a machine manually to run the Licensing Interconnect without FLEXlm:

1. Choose **Modify Startup Options** from the **ANSLIC\_ADMIN** utility.
2. Under **FLEXlm Options**, select **Start ANSYS Licensing Interconnect without FLEXlm** and click **OK**.
3. Choose **Specify the License Server Machine** from the **ANSLIC\_ADMIN** utility. If you see a message, click **OK** on the message box.
4. In the **Specify the License Server Machine** dialog box, add this machine to be the primary Licensing Interconnect and specify the Licensing Interconnect port number but not the FLEXlm port number. If necessary, use the **Move up** button to move this machine to the beginning of the list of servers.



5. Verify that you also have a machine specified from which you will use FLEXlm licenses. This machine must appear below the primary Licensing Interconnect in the list of servers. If you do not have a machine specified, add it.

---

**Note**

For the machine holding the FLEXlm licenses, you must specify *both* the FLEXlm and the Licensing Interconnect port numbers, even though the Licensing Interconnect running on that machine will only be used as a backup.

6. Start or restart the license manager using the **Start the ANSYS, Inc. License Manager** option of the **ANSLIC\_ADMIN** utility.

The above procedures configure your local machine to be the primary Licensing Interconnect. If you do not want your machine to be the primary Licensing Interconnect (i.e., your machine is a client machine), but you do want to connect to another machine that is a Licensing Interconnect machine not running FLEXlm, follow the steps below:

1. Choose **Specify the License Server Machine** from the **ANSLIC\_ADMIN** utility. Click **OK** on the message box.
2. Add or select the machine that is to be the primary Licensing Interconnect, and use the **Move up** button to move that machine to the beginning of the list of servers. If you are adding the machine, specify only the Licensing Interconnect port number. If the machine is already listed and includes a FLEXlm port number, you will need to select **Edit Selected Server Machine** to remove the FLEXlm port number.
3. Add or verify the machine from which you will use FLEXlm licenses. This machine must appear below the primary Licensing Interconnect in the list of servers.

---

**Note**

For the machine holding the FLEXlm licenses, you must specify *both* the FLEXlm and the Licensing Interconnect port numbers, even though the Licensing Interconnect running on that machine will only be used as a backup.

## ***Run the ANSYS Licensing Interconnect and FLEXlm independently***

Use this option if you want to manage your FLEXlm licenses on a server machine independently of the ANSYS, Inc. tools (for example, using Flexera Software's FLEXnet Manager to manage multiple companies' product licenses). This option should be used only by experienced users with well-established licensing procedures. To use the License Wizard to set up a license server that will run the Licensing Interconnect independently of FLEXlm, follow the steps below. Running these processes separately requires caching the license file, meaning that you specify the location of an existing license file that contains FLEXlm licenses for ANSYS, Inc. products. Because the **ANSLIC\_ADMIN** will no longer be managing FLEXlm, you will need to use your license management tools (such as FLEXNet Manager) to start/restart FLEXlm or to reread the license file if you make changes to the license file.

## Important

If you make changes to the license file, in addition to rereading the license file in FLEXlm (using your license management tools), you must also recache the license file in the Licensing Interconnect by restarting the Licensing Interconnect or by using the **Reread the License Manager Settings** option of **ANSLIC\_ADMIN**.

To use the License Wizard to set up a license server that will run the Licensing Interconnect and FLEXlm independently, follow the steps below:

1. Select **Run the License Wizard** from the **ANSLIC\_ADMIN** utility.
2. In the **Type of License Server** field, select **Run the ANSYS Licensing Interconnect and FLEXlm independently**. Click **Continue** on the message box.
3. Click **Continue** to begin the configuration process.
4. Specify the location of an existing license file that contains FLEXlm licenses for ANSYS, Inc. products. Note that the license file will not be installed, (i.e., moved into the default ANSYS license file location). Click **Continue**.
5. Specify your local machine as the license server machine.  
  
Click **Continue**.
6. You will now be asked to start or restart the license manager. Click **Continue**.
7. Click **Exit**.

---

## Note

Starting or restarting the license manager using this procedure will start **ONLY** the Licensing Interconnect. You must start or restart FLEXlm using whatever license management tool(s) you will be using to manage FLEXlm (such as FLEXNet Manager or LMTOOLS).

You can also configure your licensing manually to run the Licensing Interconnect independently of FLEXlm:

1. Choose **Modify Startup Options** from the **ANSLIC\_ADMIN** utility.
2. Under **FLEXlm Options**, select **Start ANSYS Licensing Interconnect without FLEXlm**.
3. Select **Cache FLEXlm Licenses**.
4. Verify the license path shown in the **License File Path** field and correct it if necessary. Click **OK**.
5. If you wish to use ANSYS licenses from this machine when running ANSYS products on it, choose **Specify the License Server Machine** from the **ANSLIC\_ADMIN** utility. If this machine is not in the list of servers, add it. Specify both the Licensing Interconnect and FLEXlm port numbers.
6. Start or restart the license manager using the **Start the ANSYS, Inc. License Manager** option of the **ANSLIC\_ADMIN** utility. This step will start (or restart) only the Licensing Interconnect component of the license manager.
7. Use your license management tools (such as FLEXNet Manager) to start FLEXlm if it is not already started.



## 2.3. Post-Installation Instructions

After you have installed the ANSYS, Inc. License Manager, you may want to complete some post-installation steps in order to optimize the performance of the ANSYS, Inc. License Manager at your site.

We recommend that you set your license manager to boot at system startup. Other optional post-installation tasks are included here. You will only need to configure your license server machine and configure your client machines if those steps were not completed during the product or license server installation processes.

### 2.3.1. Start the ANSYS License Manager at System Boot Time

You can set the license manager to start automatically at system boot time. This task is optional but is recommended and should be done regardless of which type of license server configuration you use.

[Table 2.2: License Manager Automatic Startup Instructions \(p. 21\)](#) details the steps for each UNIX/Linux hardware platform that must be performed on each license server to start the license manager automatically when the system is rebooted. You should substitute your platform name (see [Supported Hardware Platforms \(p. vii\)](#)) wherever you see *<platform>*. The instructions below include steps to remove any existing `boot_ansflex`; if you have already removed the `boot_ansflex`, you can safely skip that step. On Windows systems, the license manager is set to start up automatically at system reboot.

#### Note

The procedure described in this section starts the license manager at boot time as root. It is not essential that the license manager be started by the root user; it may be run by a non-privileged user, depending on your preference. If you do not want the license manager to be started by root, see the FLEXnet Licensing End User Guide for an example of starting the license manager as a non-root user at boot time.

**Table 2.2 License Manager Automatic Startup Instructions**

Platform	Instructions
HP	<p>Remove existing license manager startup information:</p> <pre>rm -f /sbin/init.d/boot_ansflex rm -f /sbin/rc2.d/S900FLEX</pre> <p>Issue the new license manager startup instructions:</p> <pre>cp /ansys_inc/shared_files/licensing/init_ansysli /sbin/init.d chmod 555 /sbin/init.d/init_ansysli ln -s /sbin/init.d/init_ansysli /sbin/rc0.d/K103init_ansysli ln -s /sbin/init.d/init_ansysli /sbin/rc2.d/S997init_ansysli</pre>
IBM	<p>Remove existing license manager startup information:</p> <ol style="list-style-type: none"> <li>1. Use a text editor to edit <code>/etc/inittab</code> and remove any lines that reference the <code>boot_ansflex</code> script.</li> </ol> <p>Issue the new license manager startup instructions:</p> <pre>echo "ansli:2:once:/ansys_inc/shared_files/licensing/ start_ansysli &gt; /dev/console 2&gt;&amp;1" &gt;&gt; /etc/inittab</pre>

Platform	Instructions
Sun	<p>Remove existing license manager startup information:</p> <ol style="list-style-type: none"> <li>1. Locate all instances of the boot_ansflex script:  <pre>find /etc -name "*ansflex" -print</pre> </li> <li>2. Remove each file, one at a time using <b>rm</b>.</li> </ol> <p>Issue the new license manager startup instructions:</p> <pre>cp /ansys_inc/shared_files/licensing/init_ansysli /etc/init.d chmod 555 /etc/init.d/init_ansysli ln -s /etc/init.d/init_ansysli /etc/rc0.d/K03init_ansysli ln -s /etc/init.d/init_ansysli /etc/rc2.d/S97init_ansysli</pre>
Linux: Red Hat	<p>Remove existing license manager startup information. Edit the file /etc/rc.d/rc.local and delete the ANSYS boot_ansflex lines.</p> <p>Issue the new license manager startup instructions:</p> <pre>cp /ansys_inc/shared_files/licensing/init_ansysli /etc/init.d chmod 555 /etc/init.d/init_ansysli chkconfig --add init_ansysli chkconfig init_ansysli on</pre>
Linux: SUSE	<p>Remove existing license manager startup information. Edit the file /etc/rc.d/boot.local and delete the ANSYS boot_ansflex lines.</p> <p>Issue the new license manager startup instructions:</p> <pre>cp /ansys_inc/shared_files/licensing/init_ansysli /etc/init.d chmod 555 /etc/init.d/init_ansysli chkconfig --add init_ansysli chkconfig init_ansysli on</pre>

Once the procedure is in place for starting the license manager automatically at boot time, reboot the system to verify that the automatic boot procedure is working correctly.

When the system comes back up, check to see that the license manager is running by typing the appropriate **ps** command and looking for `ansyslmd` and `ansysli_server` in the resulting display under the column labeled **COMMAND**. For example:

```
ps -ef
```

Next, check the `license.log` and the `ansysli_server.log` files in the licensing directory for error messages. This file contains a history of ANSYS product activity across the network when this computer is chosen as the master license server by the licensing software. It will also contain start-up messages and possibly error messages.

## Removing the Automatic Startup Information

If you remove the license manager server installation from a machine and need to remove this capability, follow the instructions below.

**Table 2.3 Removing License Manager Automatic Startup Instructions**

Platform	Instructions
HP	Issue the following commands:  <pre> /sbin/init.d/init_ansysli stop rm /sbin/init.d/init_ansysli rm /sbin/rc0.d/K103init_ansysli rm /sbin/rc2.d/S997init_ansysli </pre>
IBM	Issue the following commands:  <pre> vi /etc/inittab (delete the line that begins with "ansli") kill -1 1 /ansys_inc/shared_files/licensing/stop_ansysli </pre>
Sun	Issue the following commands:  <pre> /etc/init.d/init_ansysli stop rm /etc/init.d/init_ansysli rm /etc/rc0.d/K03init_ansysli rm /etc/rc2.d/S97init_ansysli </pre>
Linux	Issue the following commands:  <pre> /etc/init.d/init_ansysli stop chkconfig --del init_ansysli rm /etc/init.d/init_ansysli </pre>

## 2.3.2. Configuring the License Server Machine(s)

If you did not complete the configuration tasks when you installed the license manager on your license server machines, you should complete the steps below to register your license server information and then run the license wizard to complete the configuration.

1. Register your license server information.

Run the **Register License Server Information** option of the **ANSLIC\_ADMIN** utility if you do not have a license file. This option creates a file named `licserver.info` in the home and licensing directory. The content of this file is generated by prompting you for information about your company and you (the person running the utility). This is important so that we know to which of our customers this information belongs and to whom we may speak if we have questions about the data. The option also pulls necessary information from your license server (such as computer type, operating system level, and hostid) and places it in the file. Create this file on each system that will be a license server. Return the `licserver.info` file(s) to your ANSYS sales representative so that licenses can be supplied to you.

2. Run the license wizard.

This wizard walks you through the process of installing or updating a license file, specifying the license server(s) (which updates the `ansyslmd.ini` file), and starting the license manager or rereading the license file. The wizard will prompt you for the necessary information at each step. The license file will be named `license.dat` and will be created in the licensing directory. You should use this option

when you receive a new license file as it will walk you through the necessary steps to activate the new license.

If you are running in a three-server environment, the license file must reside on all three servers and must match on all three servers. Therefore, run the license wizard on all three servers.

### 2.3.3. Modify License Manager Startup Options

This **ANSLIC\_ADMIN** utility option writes an `ansyslmd.ini` file in the licensing directory. This file contains any changes to the default values as well as your license server specifications and other settings (such as log file locations). Use this **ANSLIC\_ADMIN** utility option to change the default values.

For a detailed description of the settings that you can change with this option, see [Modify Startup Options](#) (p. 45).

### 2.3.4. Create a Group

You may want to create a group of users with specific usage capabilities, such as users with licensing administrative privileges, or users who are authorized to shut down the license manager. An `ladmin` group is the most common type of group.

To specify a list of users with access to licensing administrative options, you need to create an `ladmin` group on computers from which license administration will be performed. If you create an `ladmin` group, you must include `root` in order for `root` to continue to have access to these functions. For more details on using an `ladmin` group, see the *FLEXnet Licensing End User Guide* (accessible from the **ANSLIC\_ADMIN** utility). This option is available for UNIX/Linux platforms only.

Follow the instructions below for your hardware platform to create an `ladmin` group.

**HP, Sun, Linux** Add a line to the file `/etc/group` as follows:

```
ladmin:*:nn:root,user1,user2,user3...usern
```

where `nn` represents any unique group number and `user1, user2, user3, ..., usern` represent a list of `n` users in the group.

**IBM** Use **smit** to create an `ladmin` group and add users to this group.

#### 2.3.4.1. Defining Group Restrictions for the Licensing Interconnect

On UNIX/Linux machines only, you can also specify who can shut down the ANSYS, Inc. License Manager by using the **ANSLIC\_ADMIN** utility's **Modify Startup Options**. By default, anyone can stop the license manager. You can restrict shutdown capabilities to only the user who started it, or to a group, such as `ladmin`. If you choose group, you will need to specify the name of the group. Note that the user who started the license manager will still be able to shut it down, even if he is not part of a group with shutdown capabilities.

If you specify a group restriction, any users in that group who wish to perform that operation must have the specified group as their primary group. For example, adding the following restriction:

```
RESTRICT_SHUTDOWN=GROUP:ladmin
```

restricts the ability to shut down the license manager to only members of the `ladmin` group. Any user who is a member of the `ladmin` group and wants to shut down the license manager must have `ladmin` as his primary group:

```
machineabc{user1}: groups
other testing dev1 ladmin
```

In the above example, *user1* is a member of the groups *other*, *testing*, *dev1*, and *ladmin*, where *other* is the primary group and *testing*, *dev1*, and *ladmin* are secondary groups.

To change a group from a secondary to a primary group, issue the **newgrp** (or equivalent) command:

```
machineabc{user1}: newgrp ladmin
```

In this example, *user1* now has *ladmin* as his primary group and will be able to shut down the license manager:

```
machineabc{user1}: groups
ladmin other testing dev1
```

### 2.3.5. Specify User Privileges

Establish user privileges by editing the FLEXlm Options file. The options file allows you, the license administrator, to control various operating parameters of FLEXlm:

- Allow or deny the use of features based on user, hostname, display name, group, etc.
- Reserve licenses based on user, hostname, display name, group, etc.
- Control the amount of information logged about license usage.

By using the options file, you can be as secure or as open with licenses as you like.

The default location of the options file, *ansyslmd.opt*, is in the licensing directory. If you have a three-server system, the options file must match exactly on all three servers.

If you are using an options file, you must specify the pathname to the file on the **VENDOR** line in the license file, unless it resides in the same directory as the license file, which is the default when using the **ANSLIC\_ADMIN** utility. On UNIX/Linux systems, the **VENDOR** line would look like this for an options file named *my.opt*:

```
VENDOR ansyslmd options=/ansys_inc/shared_files/licensing/my.opt
```

On Windows systems, if the path has spaces in it, you must enclose it in quotes:

```
VENDOR ansyslmd options="c:\Program Files\Ansys Inc\Shared Files\Licensing\my.opt"
```

Use the **Edit the FLEXlm Options File** option of the **ANSLIC\_ADMIN** utility to edit the license options file. The license options files must match on all three systems if you are using redundant systems. See [Edit the FLEXlm Options File \(p. 41\)](#) for details on editing the FLEXlm Options file.

### 2.3.6. Specifying the License Server and License Files

ANSYS products need to be able to locate and communicate with the license server for your ANSYS, Inc. products to run correctly. The license server and license files should be specified correctly during the product and license manager installations. Use the **Specify the License Server Machine** option of the **ANSLIC\_ADMIN** utility only if you need to change the settings already specified.

The **Specify the License Server Machine** option of the **ANSLIC\_ADMIN** utility creates a file named *ansyslmd.ini* in the licensing directory. The *ansyslmd.ini* file contains your license server specification. For more information on specifying the license server, see [Specify the License Server Machine \(p. 35\)](#) in [License Administration Using ANSLIC\\_ADMIN \(p. 29\)](#).

On Windows server machines, use the **Server ANSLIC\_ADMIN** utility (accessed via **Start> Programs> ANSYS, Inc. License Manager> Server ANSLIC\_ADMIN Utility**) to modify the `ansyslmd.ini` file. On client machines, use the **Client ANSLIC\_ADMIN** utility (accessed via **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN Utility**). If you have both the server and the client **ANSLIC\_ADMIN** utilities on the same system, use the **Client ANSLIC\_ADMIN** to specify machine-specific (local) settings.

If you want to set your license paths to have a particular license server machine for your use only (not used by others who are running ANSYS products on this machine), you can do so by setting the **ANSYSLI\_SERVERS** and/or **ANSYSLMD\_LICENSE\_FILE** environment variables. Use the **ANSYSLI\_SERVERS** environment variable to specify the Licensing Interconnect port number. Use the **ANSYSLMD\_LICENSE\_FILE** to specify the FLEXlm port number.

If the **ANSYSLMD\_LICENSE\_FILE** environment variable is set but the **ANSYSLI\_SERVERS** environment variable is not set, the same server machines will be used to specify the Licensing Interconnect but the port number will be replaced by the Licensing Interconnect default port of 2325. When both variables are set, **ANSYSLMD\_LICENSE\_FILE** explicitly defines the FLEXlm servers while **ANSYSLI\_SERVERS** explicitly defines the Licensing Interconnect servers.

### 2.3.7. Specifying Firewall Settings

If you use a firewall at your site and if you run ANSYS products outside the firewall but access the license server within the firewall, you will need to add the Licensing Interconnect and FLEXlm port numbers to the exceptions list, as well as the PORT designated on the VENDOR line of the license file. By default, these port numbers are:

Licensing Interconnect: 2325

FLEXlm: 1055

PORT designated by PORT=#### on the VENDOR LINE in the `license.dat` file (see *VENDOR Lines* (p. 4) for information on this PORT number)

### 2.3.8. Setting Up Redundant (Triad) Servers

Redundant server setup is a network configuration where multiple machines are designated as license servers. Redundancy can be achieved by running any number of license server machines independently, or by running three different machines to work in tandem. We recommend using multiple independent license server machines to achieve redundancy; however, if your site requires the use of three license server machines working in tandem, follow the guidelines in this section to configure the servers.

1. Install the License Manager on each of the three servers as described in *Installing the License Manager* (p. 13). Be sure that each machine meets all of the prerequisites.
2. Start each of the license servers using the same license file on each machine. The license file for the three-server setup will begin with three SERVER lines:

```
SERVER myserver1 00e2463 1055
SERVER myserver2 00132460b724 1055
SERVER myserver3 001a246d4e64 1055
```

The first SERVER line is the license server you have designated as the primary server followed by the second and third servers, which act as the backup servers.

3. Always start the primary server first. If not, the second or third server will take over as the master server.

When you start each license server, you will see a message indicating that the server is starting. The primary server will not be fully started until you start the second or third server, creating a quorum. Until you have a quorum of servers started, the Status window on the **ANSLIC\_ADMIN** utility will show the Licensing Interconnect and monitor running while FLEXlm will show as not running. You could see a delay of up to five minutes in the startup of a server's FLEXlm component while the communication between all three servers is being established. If a connection cannot be established within the five minutes, the Licensing Interconnect and Licensing Interconnect Monitor will stop. Check the status box in **ANSLIC\_ADMIN** to verify that all components have stopped before you attempt to restart the ANSYS, Inc. License Manager on any server.

To stop redundant servers, use the **Stop the ANSYS, Inc. License Manager** option of the Server **ANSLIC\_ADMIN** utility. In a redundant server configuration, the Licensing Interconnect will stop all three servers when you choose to stop any one of the three servers. As with starting the license servers, if one person is stopping each of the three servers, you could see a similar delay of five or more minutes before all three are fully shut down.





---

## Chapter 3: License Administration Using ANSLIC\_ADMIN

---

This chapter explains how to use the **ANSLIC\_ADMIN** utility to perform many of the procedures necessary to administer licenses. You can use the **ANSLIC\_ADMIN** utility to perform most licensing procedures.

On Windows license server machines, you will have access to a server (full) version of the **ANSLIC\_ADMIN** utility and a client version. The option descriptions below note which options are available on the Client **ANSLIC\_ADMIN** utility as well as the Server **ANSLIC\_ADMIN** utility.

The Server **ANSLIC\_ADMIN** utility is accessed via **Start> Programs> ANSYS, Inc. License Manager> Server ANSLIC\_ADMIN Utility** and updates the `ansyslmd.ini` file in the global license directory that resides at `<os drive>\Program Files\Ansys Inc\Shared Files\Licensing`. The Server **ANSLIC\_ADMIN** offers access to all options and should be used to administer all licenses across a site that use that machine as a license server machine. Changes made using the Server **ANSLIC\_ADMIN** utility will affect all users who use that machine as a license server machine, where appropriate. Note that settings such as Specifying the License Server are always local settings and do not affect other machines, even on a license server.

From a client machine, you will have access to a limited client version of the **ANSLIC\_ADMIN** utility. The Client **ANSLIC\_ADMIN** utility is accessed via **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN Utility** and updates the `ansyslmd.ini` file in the licensing directory that resides with the product installation. The Client **ANSLIC\_ADMIN** utility should be used only to make changes to that local machine's configuration. Changes made using the Client **ANSLIC\_ADMIN** utility will affect only this machine and will not affect other users.

On UNIX/Linux machines, you can access only the full version of **ANSLIC\_ADMIN**; no client version is available.

Some options may apply to UNIX/Linux or Windows systems only and are so noted.

See [Advanced Licensing Configuration Options](#) (p. 17) for details on configuring your license server to use LMTOOLS or FLEXNet Manager to manage FLEXlm. For more information on advanced FLEXlm operations, select **Help > View FLEXnet Licensing End User Guide**.

### 3.1. Using the ANSLIC\_ADMIN Utility

To run **ANSLIC\_ADMIN** on Windows, choose **Start> Programs> ANSYS, Inc. License Manager> Server ANSLIC\_ADMIN Utility** (for the server version) or **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN Utility** (for the client version). To run the utility on UNIX/Linux, type the following:

```
/ansys_inc/shared_files/licensing/lic_admin/anslic_admin
```

You do not need system administrator privileges to run the **ANSLIC\_ADMIN** utility; however, you must have system administrator privileges or be a member of an `ladmin` group to run the **Remove a Client License** and the **Uninstall the License Manager** options. Additionally, you need to have system administrator privileges on Windows machines to use the **Start the ANSYS, Inc. License Manager** and **Stop the ANSYS, Inc. License Manager** options.

**Windows 7/Vista Administrator Privileges** On Windows 7/Vista machines, you need to always run these **ANSLIC\_ADMIN** options as an administrator, preferably with UAC turned off. If UAC is on, even as administrator, you must right-mouse click on the **ANSLIC\_ADMIN** selection from the Start menu and choose **Run**

**as Administrator.** See the Windows 7/Vista discussion in the [Platform Specifics](#) section of the *ANSYS, Inc. Windows Installation Guide* for more details on working with Windows 7/Vista and UAC.

We do not recommend running the **ANSLIC\_ADMIN** on Windows machines by double-clicking the executable directly, especially on Windows 7/Vista systems. Doing so could launch the utility with unexpected permission levels.

For server machines, status of your machine appears on the bottom left-hand side of the screen, in the **Machine Name status** area. Status of the Licensing Interconnect, the ansysli monitor, and FLEXlm are all displayed. The status will update automatically when the machine's status changes.

The right side of the utility usually displays a log of your session activities (the session log). The reporting options will also display information in this area. Use the buttons at the bottom to clear the session log or to write the log out to a file. If you contact customer support, you will need to send them this log file.

For some items, such as **Display the License Status**, the right-hand window converts to a status window showing the requested information. Use the buttons underneath this status area to return to the session log, to write the status information to the session log, or to write the status information directly to a file.

To use the utility, select an action from the list of buttons on the left.

The **ANSLIC\_ADMIN** utility consists of the following options:

- [Register License Server Machine Information](#) (p. 31)
- [Display the License Server Machine Hostid](#) (p. 31)
- [Run the License Wizard](#) (p. 31)
- [Install the License File](#) (p. 32)
- [Start the ANSYS, Inc. License Manager](#) (p. 33)
- [Stop the ANSYS, Inc. License Manager](#) (p. 34)
- [Reread the License Manager Settings](#) (p. 34)
- [Specify the License Server Machine](#) (p. 35)
- [Remove a Client License](#) (p. 37)
- [Set License Preferences for User](#) (p. 38)
- [Run the ANSYS Borrow Utility](#) (p. 39) (Windows only)
- [Set Site Preferences](#) (p. 41) -- includes the following options:
  - [Edit the FLEXlm Options File](#) (p. 41)
  - [Specify Product Order](#) (p. 45)
  - [Modify Startup Options](#) (p. 45)
  - [Specify License Servers to Cache](#) (p. 46)
- [View Status/Diagnostic Options](#) (p. 47) -- includes the following options:
  - [Display the License Status](#) (p. 47)
  - [Display Queued Licenses](#) (p. 48)
  - [Display the Customer Number](#) (p. 49)
  - [View the ANSYS Licensing Interconnect Debug Log File](#) (p. 49)
  - [View the ANSYS FLEXlm License File](#) (p. 49)

- [View the ANSYS FLEXlm Debug Log File \(p. 49\)](#)
- [Gather Diagnostic Information \(p. 49\)](#)

From the **Tools> Uninstall** menu (UNIX/Linux only), you can select **Uninstall the License Manager**. See [Uninstall the License Manager \(p. 50\)](#) for more information on uninstalling the License Manager.

From the **Tools** menu, you can **Complete Unfinished Licensing Installation Configuration**. This option will re-run the licensing installation configuration process, allowing you to complete any unfinished configuration steps. Note that the license manager will need to be shut down to proceed with this option. Use this option if your licensing installation configuration process was unable to complete (for example, the license manager was not able to be shut down).

From the **Help** menu, you can select:

- **View the ANSLIC\_ADMIN Help**
- **View the ANSYS, Inc. Licensing Guide**
- **View the FLEXnet Licensing End User Guide**
- **About ANSYS, Inc. Licensing**
- **About ANSLIC\_ADMIN**

To exit the **ANSLIC\_ADMIN** utility, choose **File>Exit**.

### 3.1.1. Register License Server Machine Information

This option is used for license server machines only.

Use this option to register your license server information with ANSYS, Inc. You will be prompted to supply your customer (agreement) number (optional), company name, your name, email address, etc. This utility also obtains information about your machine, such as hostname, hostid, hardware platform, operating system level, and current date. In addition, this utility requires you to identify the type of license server machine. In the case of a three-server system, you must supply the server information on each server machine.

The utility places the information it gathers into a file named `licserver.info`. By default, this file is written to the home directory; it will also create it in the licensing directory. You can browse to a directory other than the home directory. You must forward the file, created on each license server machine, to your ANSYS sales representative.

If you are running this utility on a Windows laptop system, you should disconnect from the docking station before running this option so that the laptop's hostid information is used and not the docking station's.

### 3.1.2. Display the License Server Machine Hostid

Use this option to obtain and display the hostid of the machine on which you are currently running. Selecting this option runs the FLEXlm utility `lmhostid`. To write this information out to a file, click the **Create File** button. By default, the file is named `ansysid.machinename.txt`, although you can change the filename.

This option is available for both the server and client versions of **ANSLIC\_ADMIN**.

### 3.1.3. Run the License Wizard

You can run the license wizard either from the **ANSLIC\_ADMIN** utility or as part of the license manager installation. The license wizard is used to configure license server machines only.

This wizard walks you through the process of installing or updating a license file, specifying the license server machine(s) (which updates the `ansyslmd.ini` file), and starting/restarting the license manager. The wizard will prompt you for the necessary information at each step. You should use this option when you receive a new license file as it will walk you through the necessary steps to activate the new license.

Advanced users can also use the wizard to configure their license servers to run the Licensing Interconnect without FLEXlm or independently of FLEXlm. See [Advanced Licensing Configuration Options](#) for more information.

You can alternatively perform each of these steps individually using the separate options on the **ANSLIC\_ADMIN** utility.

1. When the License Wizard is launched, you will need to specify the type of license server. You can choose from three options:
  - Run the ANSYS Licensing Interconnect with FLEXlm (default)
  - Run the ANSYS Licensing Interconnect without FLEXlm
  - Run the ANSYS Licensing Interconnect and FLEXlm independently

Accept or select the default and click **Continue**. Most customers will use the default option. The other options are more advanced and should be used only by experienced customers. Instructions for using the other options are found under [Advanced Licensing Configuration Options](#) in the *ANSYS, Inc. Licensing Guide* and are not included here.

2. Next, you must install a valid license file if one is not already installed. Click **Continue**, and then browse to and select the license file provided to you by ANSYS, Inc. You must complete this step to continue with the wizard. If you do not have a license file, you must exit the wizard. You can rerun the license wizard when you receive your license file.
3. The license wizard will next specify the local machine as the license server, meaning you will use the local machine as the license manager from which licenses are checked out. Click **Continue**. This step updates the `ansyslmd.ini` file to contain the local machine information as the license server. If you wish to specify a different or additional machine as the license server, you can use the [Specify the License Server Machine](#) option of the **ANSLIC\_ADMIN** utility.
4. The final step is to start the ANSYS, Inc. License Manager. Click **Continue** to start the license manager. If the license manager is already running, the license wizard will stop and restart it. Be aware that stopping the license manager could impact other users who are using this machine as their license server.

When all steps of the wizard have been successfully completed, you can click **Exit** to leave the license wizard.

---

### Note

You can choose to skip steps when applicable; however, if you choose to skip any steps and you have not otherwise configured your system, your license server may not function correctly and you may not be able to run ANSYS, Inc. products.

## 3.1.4. Install the License File

This option is used for license server machines only.

If you received a new license and your server and hostid are unchanged, use this option to replace your existing license file. This utility creates the license file, called `license.dat` by default, in the licensing directory. The license file contains your licensing information and allows you to run ANSYS, Inc. products.

After you run this option, select **Reread the License Manager Settings** to reread the FLEXlm license file, which allows you to update the license manager with the new license without the need to stop and restart the license server.

1. Save the license file that you received from your ANSYS sales representative to a temporary file.
2. Select this option from the **ANSLIC\_ADMIN** utility.
3. Browse to the file. If you do not already have a license file, the information from this temporary file will be used to create the license file. If you do have an existing license file, the existing file will be renamed to `license.dat.old`.

In a three-server (redundant triad) system, you must use this option on each license server machine.

---

### Caution

Do not save license files in Microsoft Word format.

If the license manager is running when you add or update a license, you must restart the license manager or [reread the license file](#) for the new file to take effect.

You can use this option to install ANSYS license files only. To determine if you have an ANSYS license file, see [Recognizing an ANSYS License File](#).

## 3.1.5. Start the ANSYS, Inc. License Manager

This option is used for license server machines only.

Use this option to start the license manager. This option starts `ansysli_monitor` and `ansysli_server`, which automatically starts the FLEXlm components of the license manager: `lmgrd` and `ansyslmd`. The license file must exist prior to starting the license manager. The license manager must be running to run an ANSYS, Inc. product.

---

### Note

If you chose to configure your license manager such that you run the Licensing Interconnect without FLEXlm or run the Licensing Interconnect and FLEXlm independently, then selecting this option will start only the Licensing Interconnect.

In a three-server license environment, the license manager *must* be running on at least two of the three servers before an ANSYS product can be run. In a one-license server system, the license manager must be started on that server.

---

### Note

In a three-server environment, you may see a delay in starting the license manager until two of the three servers are running. Do not press the **Start the License Manager** button more than once for the same server.

The `ansysli_server.log` and the `license.log` files in the licensing directory contain a history of the Licensing Interconnect and FLEXlm activity, respectively, across your network since the last time the license manager was started.

### 3.1.6. Stop the ANSYS, Inc. License Manager

This option is used for license server machines only.

Use this option to shut down the license manager. This option will shut down the ANSYS, Inc. License Manager, including `ansysli_monitor`, the Licensing Interconnect and the FLEXlm components of the license manager (`lmgrd` and `ansyslmd`) on all machines listed in the license file. You will be asked to confirm that the license manager should be shut down. Remember that the license manager must be running in order to run an ANSYS, Inc. product. Before shutting down the license manager, check that no ANSYS, Inc. product processes are currently running. Any processes that are running will be terminated when you shut down the license manager.

---

#### Note

If you chose to configure your license manager such that you run the Licensing Interconnect without FLEXlm or run the Licensing Interconnect and FLEXlm independently, then selecting this option will stop only the Licensing Interconnect.

---

#### Caution

Do not use `kill -9` (on UNIX/Linux systems) to shut down the license manager. Use the **ANSLIC\_ADMIN** utility option or `ansysli_server -k stop`.

---

#### Note

In a three-server (redundant triad) environment, you may see a delay in stopping the license manager. Do not press the **Stop the License Manager** button more than once for the same server.

This option will shut down all three server machines of a three-server (redundant triad) system, unless the license manager was started with the `-local` option. However, it will only stop the Licensing Interconnect running on the machine where this option was run. You will need to manually stop the Licensing Interconnects that are running on the other two machines of the three-server system, or those Licensing Interconnects will continue to restart FLEXlm on the other machines.

We do not recommend stopping the license manager manually; however, if you choose to do so, you must stop the components in the following order:

1. `ansysli_monitor` (`ansysli_monitor`)
2. Licensing Interconnect (`ansysli_server`)
3. FLEXlm components (`lmgrd` and `ansyslmd`)

### 3.1.7. Reread the License Manager Settings

This option is used for license server machines only.

Use this option to reread the FLEXlm license file and reread the license manager settings. You will need to use this option to reread the FLEXlm license file when the license manager is already running and you have updated the license file or the license options file so that changes will take effect. Otherwise, the license manager will not be aware of the changes until it is stopped and restarted.

This option:

- rereads the `ansyslmd.ini` file to get any new settings, such as log file size or change of name
- appends or opens a new log file
- rereads the site preferences
- Recaches the license file
- Issues a FLEXlm reread
- Recaches the servers
- Informs other servers about changes

Be aware that if you change the Licensing Interconnect port number, you will need to restart the license manager; a reread will not update the port number.

---

### Note

If you chose to configure your license manager such that you run the Licensing Interconnect without FLEXlm or run the Licensing Interconnect and FLEXlm independently, then selecting this option will affect only the Licensing Interconnect components and recache the license file.

## 3.1.8. Specify the License Server Machine

Use this option to set the ANSYS Licensing Interconnect and ANSYS FLEXlm servers. Using this option creates or updates (lists, adds, modifies, deletes, or reorders) entries in the `ansyslmd.ini` file in the licensing directory for the local machine. With this option, you can specify single servers, multiple single servers, or redundant triad servers. You also use this option to specify the Licensing Interconnect and ANSYS FLEXlm port numbers. Run this option on every machine where you installed an ANSYS product. We recommend using this method for specifying your license server preference.

This option is available for both the server and client versions of **ANSLIC\_ADMIN** on Windows machines. In most situations, you should use the client version. The client version is appropriate regardless of where you have installed the product. You should use the server version of this option only to specify the license server machines for Release 11.0 and lower versions. You can also use the server version to specify license server machines if you have installed the product into `<os drive>\Program Files`, but we recommend using the client version anytime you are working with the current release.

If the product is installed in `<os drive>\Program Files` (by a user with administrative privileges), you must have administrative privileges in order to add or modify your server specification. If you are running on a Windows 7/Vista machine, you must run as an administrator, preferably with UAC turned off. If you are running with UAC turned on, you must launch **ANSLIC\_ADMIN** using the Run as Administrator option. For more information on Windows 7/Vista behavior with UAC turned on, see [Platform Specifics](#) in the *ANSYS, Inc. Windows Installation Guide*.

To add a server:

1. Click **Add Server Specification**.



2. Enter the ANSYS Licensing Interconnect and FLEXlm port numbers (2325 and 1055 by default, respectively). Specify the number of servers and enter the host name(s) of your license server(s). You cannot enter a path or a filename in place of the hostname.

To specify multiple single servers, click **OK** on this screen after adding the first server. Then click the **Add Server Specification** button again and add information for the next server. Repeat this process for each single server.

To specify redundant triad servers, click the **3-server (redundant triad)** option and enter the hostname for each of the three servers in the spaces available.

3. Click **OK**.

To delete a server, highlight the server in the list and click **Delete Selected Server**.

To edit a server, click on one of the listed servers and click **Edit Selected Server**. You can then change the ANSYS Licensing Interconnect and the FLEXlm port numbers, the number of license servers, or the hostname(s).

You can choose to start the ANSYS Licensing Interconnect without FLEXlm, and to manage FLEXlm separately by caching the FLEXlm license. If you choose to run these advanced options, please read [Advanced Licensing Configuration Options](#) for detailed instructions, limitations, and caveats.

**The ansyslmd.ini File** The **Specify the License Server Machine** option creates or updates the `ansyslmd.ini` file that is located in the licensing directory. Entries in the `ansyslmd.ini` file tell ANSYS, Inc. products which license server(s) to query to find a valid license. Using this option allows all users at your site to use this setting without having to individually set the **ANSYSLMD\_LICENSE\_FILE** or **ANSYSLI\_SERVERS** environment variables to specify the license server machine(s). It also eliminates the need to have a copy of the license file on every system at your site.

The order that the SERVER lines are listed in the `ansyslmd.ini` file dictates the order in which the license request is granted. To reorder the entries in the list, highlight a server and use the **Move up** or **Move down** buttons.

You are not limited to designating one set of license server machines for your network. You can have multiple single-server or three-server (redundant triad) licensing systems on your network. In this situation, you would have certain licenses connected to a set of server machines (one or three) on the network, and other licenses connected to a different set of server machines (one or three) on the network.

The format of the `ansyslmd.ini` file created by **ANSLIC\_ADMIN** follows. Each server's specification entry in this file will typically begin with **ANSYSLI\_SERVERS=** and **SERVER=** to specify the Licensing Interconnect and FLEXlm port numbers, respectively.

On a single server:

```
ANSYSLI_SERVERS=<ansysliport>@<host>
SERVER=<flexlmpport>@<host>
```

For multiple single servers, each server should have its own **ANSYSLI\_SERVERS=** and **SERVER=** lines.

On redundant (triad) servers on UNIX/Linux platforms:

```
ANSYSLI_SERVERS=<ansysliport>@<host1>:<ansysliport>@<host2>:<ansysliport>@<host3>
SERVER=<flexlmpport>@<host1>:<flexlmpport>@<host2>:<flexlmpport>@<host3>
```

Windows platforms use semicolons (;) instead of colons:

```
ANSYSLI_SERVERS=<ansysliport>@<host1>;<ansysliport>@<host2>;<ansysliport>@<host3>
SERVER=<flexlmpport>@<host1>;<flexlmpport>@<host2>;<flexlmpport>@<host3>
```



The ANSYS Licensing Interconnect default port number is 2325 and the ANSYS FLEXlm default port number is 1055. The host is the license server hostname. For example, if the license server name is alpha1:

```
ANSYSLI_SERVERS=2325@alpha1
SERVER=1055@alpha1
```

The order of the ANSYSLI\_SERVERS and the SERVER lines in the `ansyslmd.ini` file specifies the order in which the requested license will be granted.

You must use the `port@host` format; you cannot enter a path or a filename in place of the hostname.

**Overriding the `ansyslmd.ini` File** If you want to override the server specification settings in the `ansyslmd.ini` file, you can do so by setting the **ANSYSLMD\_LICENSE\_FILE** and **ANSYSLI\_SERVERS** environment variables on individual machines. These environment variables are useful if you want to temporarily point to a different license server machine without disrupting the machine's configuration. Use the **ANSYSLI\_SERVERS** environment variable to specify the Licensing Interconnect port number. Use the **ANSYSLMD\_LICENSE\_FILE** to specify the FLEXlm port number.

If the **ANSYSLMD\_LICENSE\_FILE** environment variable is set but the **ANSYSLI\_SERVERS** environment variable is not set, the same server machines will be used to specify the Licensing Interconnect but the port number will be replaced by the Licensing Interconnect default port of 2325. When both variables are set, **ANSYSLMD\_LICENSE\_FILE** explicitly defines the FLEXlm servers while **ANSYSLI\_SERVERS** explicitly defines the Licensing Interconnect servers.

If you set the **ANSYSLMD\_LICENSE\_FILE** or **ANSYSLI\_SERVERS** environment variables on a three-server (redundant) system, specify all three systems, in the same order as the SERVER lines are listed in the license file. If you specify only the master and it is down, you could see a "License Server Down" or "No License Found" message and the search for a license could fail. Join redundant or multiple single server systems by separating the system names with colons on UNIX/Linux systems and semicolons on Windows systems.

To have your license server preference known each time you log in, set the environment variables **ANSYSLMD\_LICENSE\_FILE** and **ANSYSLI\_SERVERS** (both Windows and UNIX systems). On UNIX systems, place these environment variables in your login startup file (e.g., `.cshrc` file).

### 3.1.8.1. Sample Scenario

Suppose that you have a company with two departments, Design and Engineering. The Design Department has one license of ANSYS Mechanical and one license of DesignSpace with the server DES1, and all users in the Design Department run on DES1. The second department, Engineering, has two licenses of ANSYS Multiphysics with the server ENG1, and all users in the Engineering Department run on ENG1. The `ansyslmd.ini` file states `ANSYSLI_SERVERS=2325@DES1` and `SERVER=1055@DES1` for the Design Department, and `ANSYSLI_SERVERS=2325@ENG1` and `SERVER=1055@ENG1` for the Engineering Department. These settings limit users in each department to work only on their own server. If individual users in the Design Department need to run on the engineering server, ENG1, they could set the following environment variable:

```
setenv ANSYSLMD_LICENSE_FILE 1055@ENG1
```

This example would automatically use 2325@ENG1 for the Licensing Interconnect license path. For this example, you would need to set **ANSYSLI\_SERVERS** only if you were not using the default Licensing Interconnect port number.

## 3.1.9. Remove a Client License

This option is used for license server machines only.

You may need to free a license when a licensed user was running an ANSYS, Inc. product on a node that subsequently crashed. This situation will sometimes cause the license to become unusable. Free the license to allow the license to return to the pool of available licenses. Only those with FLEXlm administrative privileges (UNIX/Linux) or administrative privileges (Windows) can use this procedure because removing a user's license can be disruptive.

You will be prompted for the following information:

- feature name
- user name
- host name
- display

If you are not sure of any of the above items, run the *Display the License Status* (p. 47) option to obtain the correct information. The messages display in the following format:

```
user hostname display version servername/port start date
```

For example:

```
user1 hostabc /dev/pts/5 (v2009.0213) (pghlicserver/1055 301), start Mon 2/16 6:09
user2 hostdef NONE (v2009.0112) (pghlicserver/1055 2901), start Mon 2/16 6:10
user3 hostghi pghxpuser3 (v2007.0830) (pghlicserver/1055 5151), start Tue 2/17 10:30
user4 hostjkl /dev/tty (v2009.0213) (pghlicserver/1055 5062), start Mon 2/16 9:00
```

In the above example, the following are display names:

```
/dev/pts/5
NONE
pghxpuser3
/dev/tty
```

Note that display names at your site may or may not follow one of these conventions.

This option allows you to remove a client only from the license server machine on which the license is checked out.

A client must be running at least two minutes before you can remove it. If you attempt to remove a client in less than two minutes, you will get an error.

The license removal is not immediate.

### 3.1.10. Set License Preferences for User

Use this option to review the licenses available and prioritize the licenses. Settings specified here apply only to the username shown in the button title. When you select this option, you will first choose the release for which you want to set preferences. Select the release and click **OK**. All licenses available to you on your license server machine(s) are shown under the appropriate tab on the next dialog:

- Solver
- PrepPost
- Geometry
- HPC

You control the order in which the system attempts to check-out a license by moving the licenses up and down in the list using the **Move Up** or **Move Down** buttons. When you open an application, the license manager attempts to check-out the license listed first in the list. If unable to check-out that license, then the license manager attempts to check out the next license in the list that can be used by the application. The license manager will continue through the list until an available license is found. If none are found, a message displays.

By default, all licenses available from your license server are shown and marked as available for use. If you do not want a particular license to be available for your use, select the license in the list, and set the **Use/Don't Use** field to 0.

If you received a new license, you may need to reset your preferences. If you had previously set your license preferences using this option, add the new license to your list in the desired order.

If you have made changes to your license preferences, you can return the settings to the default state by selecting the **Reset to Default** button.

When you have completed your changes, click **OK** to make the changes and close the dialog box. Click **Apply** to make the changes without closing the dialog box. Selections you make here will take effect with any new sessions/license checkouts but will not affect the current session or change the license(s) currently in use.

ANSYS Workbench users can also specify whether to use the shared or separate licensing method.

For details on setting user license preferences, including shared vs. separate licensing for ANSYS Workbench and specifying HPC licensing settings, see [Establishing User Licensing Preferences](#).

### 3.1.11. Run the ANSYS Borrow Utility

License borrowing (available only on Windows machines) allows a user to take a license for use outside of the company facility, such as for an engineer to take a license home on his laptop. Before users can take advantage of license borrowing, you need to set up the borrowing environment.

License borrowing is available only if you have borrowable licenses and only on Windows machines; however, the license server machine that is managing the licenses can be any system that ANSYS, Inc. supports as a license server. You cannot run both borrowed and non-borrowed licenses simultaneously. For example, if you borrow licenses but remain connected to a network from which you can use licenses, you will be able to use only the borrowed licenses. Likewise, if the machine on which you borrowed licenses is also a license server from which you could normally use licenses, you would be able to use only the borrowed licenses.

This option is available for both the server and client versions of **ANSLIC\_ADMIN**.

#### 3.1.11.1. Setting up License Borrowing

To use license borrowing, you must have license keys that specify borrowable licenses. License features that can be borrowed will contain the keyword BORROW.

You can use the optional FLEXlm options file settings to further control borrowing at your site. To use these settings, you must first specify which users are allowed to borrow licenses by adding an INCLUDE\_BORROW group to the options file. Any user not in the INCLUDE\_BORROW group will not be allowed to borrow licenses. You may also wish to set up an EXCLUDE\_BORROW group to prevent specific users from borrowing licenses.

You can also limit the number of licenses that can be borrowed from your total pool of borrowable licenses. Set the BORROW\_LOWWATER option in the options file to the number of licenses that *cannot* be borrowed.

For example, if you have 15 licenses total and want to limit the number that can be borrowed to 5, set `BORROW_LOWWATER=10`.

The maximum number of hours a license can be borrowed at any one time is defined by your license file. You can set that limit lower for your users by setting the `MAX_BORROW_HOURS` option in the options file. `MAX_BORROW_HOURS` cannot exceed the limit specified in your license file. The total maximum number of borrowable hours is defined by your license agreement.

**Compatibility between versions** Flexera Software has created an incompatibility in license borrowing between FLEXlm releases. Since different versions of FLEXlm are used at most releases, the borrowed licenses are not compatible between different ANSYS releases. The ANSBORROW utility includes a selection for the ANSYS version you are planning to run using the borrowed license. Choose your version carefully, because you will only be able to run one of the versions at a time with borrowed licenses.

Another incompatibility exists within the same release, between platforms. If you borrow a license with one of the Windows versions (32-bit or x64), you can't use the borrowed license on one of the other Windows versions, even on the same computer. For example, you have an x64 machine running the Windows 32-bit version, and then you install the x64 version when it is released. If you borrow the license using the 32-bit ANSBORROW, you won't be able to run the x64 version of ANSYS.

The ANSBORROW utility allows you to borrow either initial release or service pack (SP) versions. The SP versions for a given platform use the same FLEXlm version as the initial release; therefore, licenses selected for SP should work for the initial release.

### **3.1.11.2. Running the Borrowing Utility**

To run the license borrowing utility, you must be running on a Windows machine connected to a network, and you must have borrowable licenses. For more information about borrowable licenses, please contact your ANSYS sales representative.

1. Connect to the network where the license server machine is running.
2. Launch the client license borrowing utility: **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN utility**. Select **Run the ANSYS Borrow Utility**.
3. On the Borrow tab, specify the version number and return date. The return date defaults to the third day from the current date.

If you change any of the borrowing criteria, the product list will automatically update in five seconds. To update the list immediately, click the **Update Delay (sec)** button.

4. Select the product(s) you want to borrow. The product list includes add-on features or geometry interfaces. If you do not see a product listed that you think should be available, verify that you have selected the correct version.
5. Click **Borrow**.
6. Disconnect from the network. Note that even when you've disconnected from the network, you must still run the products from the same account you used to borrow the licenses. For example, if you use a network account to borrow the licenses, you cannot then run from a local account.

Be sure to verify that you can run the correct product with the borrowed licenses from your machine AFTER you've disconnected from the network. If you encounter any difficulties, reconnect to the network, return the license, and then re-borrow the correct license.

If you keep a license for the full amount of time, there is no need to check the license back in. Your borrowed license will simply expire at the end of the borrow period, and the license will be released into the pool of available licenses.

To return a license early:

1. Reconnect to the same license server from which you borrowed the license(s).
2. Launch the client license borrowing utility: **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN utility**. Select **Run the ANSYS Borrow Utility**.
3. Select the **Return** tab. Click **Return all**. If you wish to keep some but not all of the licenses you borrowed, you will need to re-borrow them.
4. Disconnect from the network.

The license(s) you borrowed will be returned to the pool of available licenses.

---

### Note

You must return borrowed licenses from the same account you used to borrow the licenses originally.

Some multi-feature products include other products and/or add-on features. The utility will automatically check out any such included products or add-on features. If you borrow one of these multi-feature products and then click on the **Return** tab, you will see the product you selected, as well as any other products and add-ons that it includes. If you return a multi-feature product early, these products and add-on features will also be returned.

## 3.1.12. Set Site Preferences

This option is used for license server machines only.

The following site preference settings are available:

- [Edit the FLEXlm Options File \(p. 41\)](#)
- [Specify Product Order \(p. 45\)](#)
- [Modify Startup Options \(p. 45\)](#)
- [Specify License Servers to Cache \(p. 46\)](#)

### 3.1.12.1. Edit the FLEXlm Options File

To set specific user privileges with FLEXlm, you need to use the options file. The options file allows you, the license administrator, to control various operating parameters of FLEXlm:

- Allow or deny the use of features based on user, hostname, display name, group, etc.
- Reserve licenses based on user, hostname, display name, group, etc.
- Control the amount of information logged about license usage.

By using the options file, you can be as secure or as open with licenses as you like.

The default location of the options file, `ansyslmd.opt`, is in the licensing directory. If you have a three-server (redundant triad) system, the options file must match exactly on all three servers.

If you are using an options file, you must specify the pathname to the file on the **VENDOR** line in the license file, unless it resides in the same directory as the license file, which is the default when using the **ANSLIC\_ADMIN** utility. On UNIX/Linux systems, the **VENDOR** line would look like this for an options file named `my.opt`:

```
VENDOR ansyslmd options=/ansys_inc/shared_files/licensing/my.opt
```

On Windows systems, if the path has spaces in it, you must enclose it in quotes:

```
VENDOR ansyslmd options="c:\Program Files\Ansys Inc\Shared Files\Licensing\my.opt"
```

Use the **Edit the FLEXlm Options File** option to edit the license options file `ansyslmd.opt`. The license options files must match on all three systems if you are using redundant systems. Restart the license manager or *Reread the License Manager Settings* (p. 34) for changes to take effect.

### 3.1.12.1.1. The Options File Format

The maximum line length is 2048 characters. FLEXlm allows the "\" character as a continuation character in options file lines. You can include comments in your options file by starting each comment line with a pound sign "#." Everything in an options file is case sensitive. Be sure that user names and feature names, for example, are entered correctly.

Some commonly used lines in the options file are shown below. See the *FLEXnet Licensing End User Guide* (accessible from the **ANSLIC\_ADMIN** utility) for information on additional options.

EXCLUDE	Deny a user access to a feature.
EXCLUDEALL	Deny a user access to all features served by this vendor daemon.
GROUP	Define a group of users for use with any options.
HOST_GROUP	Define a group of hosts for use with any options.
INCLUDE	Allow a user to use a feature.
INCLUDEALL	Allow a user to use all features served by this vendor daemon.
RESERVE	Reserve licenses for a user.
BORROW_LOWWATER	Sets the number of licenses for a BORROW feature that <i>cannot</i> be borrowed.
EXCLUDE_BORROW	Excludes a user or predefined group of users from the list of who is allowed to borrow licenses for a BORROW feature.
INCLUDE_BORROW	Includes a user or predefined group of users in the list of who is allowed to borrow a BORROW feature.
MAX_BORROW_HOURS	Changes the maximum period for which a license can be borrowed. The new period must be less than that specified in the license file. If multiple MAX_BORROW_HOURS keywords appear in the options file, only the last one is applied to the license feature.

#### Note

You must stop and restart the license manager (rather than rereading the license file) when a change is made to the options file.

When creating an options file, you must understand the options file precedence. INCLUDE and EXCLUDE statements can be combined in the same options file and control access to the same features. When doing so, keep in mind the following:

- If there is only an EXCLUDE list, everyone who is not on the list will be allowed to use the feature.
- If there is only an INCLUDE list, only those users on the list will be allowed to use the feature.
- If neither list exists, everyone is allowed to use the feature.
- The EXCLUDE list is checked before the INCLUDE list; someone who is on both lists will not be allowed to use the feature.
- EXCLUDE\_BORROW supersedes INCLUDE\_BORROW.
- Anyone not in an INCLUDE\_BORROW statement is not allowed to borrow licenses.

Once you create an INCLUDE or EXCLUDE list, everyone else is implicitly "outside" the group. This feature allows you, as an administrator, the ability to control licenses without having to explicitly list each user that you wish to allow or deny access to. In other words, there are two approaches; you can either give most users access and list only the exceptions, or you can severely limit access and list only the those users that have access privileges.

**Keyword Format** The format for these keywords follow:

```
RESERVE number feature {USER | HOST | DISPLAY | PROJECT |
    GROUP | HOST_GROUP | INTERNET} name

INCLUDE feature {USER | HOST | DISPLAY | PROJECT | GROUP
    | HOST_GROUP | INTERNET} name

EXCLUDE feature {USER | HOST | DISPLAY | PROJECT | GROUP
    | HOST_GROUP | INTERNET} name

GROUP name list_of_users

INCLUDEALL {USER | HOST | DISPLAY | PROJECT | GROUP
    | HOST_GROUP | INTERNET} name

EXCLUDEALL {USER | HOST | DISPLAY | PROJECT | GROUP
    | HOST_GROUP | INTERNET} name

HOST_GROUP name host_list

BORROW_LOWWATER feature number

EXCLUDE_BORROW feature {USER | HOST | DISPLAY | PROJECT |
    GROUP | HOST_GROUP | INTERNET} name

INCLUDE_BORROW feature {USER | HOST | DISPLAY | PROJECT |
    GROUP | HOST_GROUP | INTERNET} name

MAX_BORROW_HOURS feature num_hours
```

Above, *number* is the number of tasks; *feature* is the license feature name (see the [Product Variable Table](#) for a list of license feature names for all ANSYS products); *name* is the user's login name or group; *list\_of\_users* is a blank-separated list of group member's login names; *host\_list* is a blank-separated list of host names; and *num\_hours* is the number of hours for which a license can be borrowed, up to the limit specified in the license file. On UNIX/Linux systems, DISPLAY requires the tty device name, and not the **DISPLAY** environment variable name. Also, PROJECT refers to the **LM\_PROJECT** environment variable.

---

## Note

You can make groups arbitrarily large by listing the GROUP more than once; FLEXlm concatenates such entries.



You can also specify *feature* as follows:

```
feature:name=value
```

You can specify a feature by any of the following fields, found in the license file INCREMENT lines:

```
VERSION HOSTID EXPDATE KEY VENDOR_STRING ISSUER NOTICE
dist_info user_info asset_info
```

If your license file has multiple INCREMENT lines of the same feature and any of these fields (such as VENDOR\_STRING and VERSION) differ between the INCREMENT lines, and you are reserving licenses for this feature, FLEXlm will reserve the same number of licenses from each INCREMENT line. This can result in more licenses being reserved than expected. Specify the differing field to reserve licenses from a single INCREMENT line.

For example, the following license contains two INCREMENT lines that differ only by the version date:

```
INCREMENT struct ansyslmd 2010.0331 permanent 1 AA1DE4A0E7BB \
  VENDOR_STRING=customer:12345678 \
  ISSUED=01-Jan-2010 START=01-Jan-2010

INCREMENT struct ansyslmd 2010.0430 permanent 1 3D967A3ECF4E \
  VENDOR_STRING=customer:12345678 \
  ISSUED=01-Jan-2010 START=01-Jan-2010
```

If you create the following entry in the options file:

```
RESERVE 1 struct USER smith
```

then you will reserve two licenses, one from each INCREMENT line, for user smith.

To reserve only one license, create an entry that contains the differing information:

```
RESERVE 1 struct:VERSION=2010.0430 USER smith
```

For a complete list of license features, see the [Product Variable Table](#).

For more information on the options file, see the *FLEXnet Licensing End User Guide* (accessible from the **ANSLIC\_ADMIN** utility).

---

## Note

ANSYS products use a TIMEOUT and TIMEOUTALL default of 55 minutes (3300 seconds). You cannot set the minimum value below 15 minutes (900 seconds). Application idle TIMEOUTs are not supported, i.e., the TIMEOUT options will not work for applications that are not actively in use.

### 3.1.12.1.2. Sample Options File

In the following example options file, one license to execute ANSYS Mechanical is reserved for user pat, three licenses for user lee, and one license for anyone on the computer with the hostname of client3. In addition, user joe is not allowed to use the ANSYS Mechanical license.

```
RESERVE 1 ansys USER pat
RESERVE 3 ansys USER lee
RESERVE 1 ansys HOST client3
EXCLUDE ansys USER joe
```



## Note

In a three license server environment, the same options file must be placed on all three systems. The options specified will not take effect until the license manager has been shut down and re-started.

### 3.1.12.2. Specify Product Order

This option launches a utility that allows you to designate the order that the licensed products are listed in the Mechanical APDL launcher and the order in which licenses are tried in all applications. You first will need to select the product release. You can choose one of the following:

- Release 11.0: Selecting this option affects the product ordering for Release 11.0 for this machine (including all users who run a Release 11.0 product on this machine or mount to this machine). If you select this option, the product ordering you specify will NOT affect any releases after 11.0.
- Release 12.0 and higher: Selecting this option affects the product ordering for Releases 12.0 or later and is applicable ONLY if this machine is a license server machine. The product ordering you select with this option will affect all users who use this machine as a license server machine.

After you choose a release level, click **OK**. You will then need to select the installation directory (Release 11.0 only), product category (Solver, PrepPost, or Geometry). To re-order the products, select a product and click the **Move up** or **Move down** button. When you have finished reordering all products that you wish to reorder, click **Save** and then **Close**.

All products available in a given category are shown, regardless of whether you have licenses for them.

If you have made changes to your product ordering, you can return the settings to the default state by selecting the **Reset to Default** button.

For Release 12.0 and higher, after making changes to the product order (including resetting to the default), you will need to reread the license or restart the license manager for the changes to take effect.

Only those logged in as root or superuser (UNIX/Linux) or with administrative privileges (Windows) can use this utility. You should set the product order before any of the users at your site run; once a user sets his preferences, the user preferences will take precedence.

### 3.1.12.3. Modify Startup Options

Use this option to modify your license manager startup options. License manager startup options that you can modify include:

#### ANSYS Licensing Interconnect Debug Log File

You can specify the path to the ANSYS Licensing Interconnect debug log file (`ansysli_server.log`, by default), the detail level of the file (standard or verbose), the size limit of the file, and the number of log files to save (default is 1; minimum is 1). The default file size is unlimited (blank), and the minimum size is 1 MB. By default, a new file is created each time; you can choose to have new information appended to the existing file instead.

#### ANSYS Licensing Interconnect Report File

The Licensing Interconnect has the ability to track and log your usage data into a report file (`<timestamp>.rprt`). By default, this feature is turned off. Use this option to enable this feature. Report files are saved and time-stamped for future reference. You can specify a directory other than the default where the report files will be saved, but you cannot change the file name. You can also specify the size

of the report file in megabytes. While we are able to collect this data into a report file, we currently do not offer a reporting tool that will display the data.

### **FLEXlm Options**

You can specify the path to the FLEXlm license file (`license.dat`, by default), as well as to the FLEXlm debug log file (`license.log`, by default). By default, a new `license.log` file is created each time you start the license manager; you can choose to have new information appended to the existing file instead.

You can also set the `-local` setting (UNIX/Linux only).

The `-local` option restricts the ability to shut down the FLEXlm components of the license manager to only an administrator running on the same machine where the FLEXlm components of the license manager was started. This option is off by default. See the *FLEXnet Licensing End Users Guide* (accessible from the **ANSLIC\_ADMIN** utility) for more information on using this option.

The `-local` option is not recommended if you will be using license borrowing.

You can also choose to start the ANSYS Licensing Interconnect without FLEXlm and to cache FLEXlm licenses. Use these two options together to run the ANSYS Licensing Interconnect and FLEXlm independently. These options are advanced configuration options and require additional steps. Please see [Advanced Licensing Configuration Options](#) for more information, including detailed procedures for using these options.

### **Miscellaneous Options**

You can specify the ANSYS Licensing Interconnect port number here. The default port number is 2325.

All machines that have this server machine in their Licensing Interconnect paths must use the same port number. If you change the Licensing Interconnect port number here, and other machines use this machine's Licensing Interconnect, then on each of those machines, you must change the Licensing Interconnect port number by one or both of the following options, as appropriate for your configuration:

- **Specify the License Server Machine** option of the **ANSLIC\_ADMIN** utility
- **Specify License Servers to Cache** option of the **ANSLIC\_ADMIN** utility

On UNIX/Linux machines only, you can also specify who can shut down the ANSYS, Inc. License Manager. By default, anyone can stop the license manager. You can restrict shutdown capabilities to only the user who started it, or to a group, such as `ladmin`. If you choose group, you will need to specify the name of the group. If you specify a group restriction, any users in that group who wish to stop the license manager must have the specified group as their primary group. Note that the user who started the license manager will still be able to shut it down, even if he is not part of a group with shutdown capabilities.

#### **Warning**

The `license.log` file and the `ansysli_server.log` file for each server should be located on a local disk. Writing to an NFS-mounted disk or remote file server creates a situation where the license server(s) may fail. If the remote system containing these files crashes, the license manager would be unable to log license transaction data. This would create a fatal error condition.

### **3.1.12.4. Specify License Servers to Cache**

When the Licensing Interconnect starts, it caches the licenses for any `ANSYSLI_SERVERS` lines that are in the `ansyslmd.ini` file in addition to its own licenses. As you run ANSYS applications and they connect with the Licensing Interconnect, it caches the licenses for any servers in your `ANSYSLI_SERVERS` path that have

not yet been cached. Sometimes, especially due to network traffic, this caching process can impact the time it takes to check out a license. Use this option to cache your servers when you start the ANSYS, Inc. License Manager, rather than at the point that you request individual licenses.

When you select this option:

1. On the **Specify License Servers to Cache** dialog, click **Add Server Specification**.
2. On the **Specify License Servers to Cache - Add Server Machine Specification** dialog box, enter the Licensing Interconnect port number. The default Licensing Interconnect port number is 2325.
3. Enter the hostname of the server.
4. Click **OK**.
5. Repeat steps 1 - 4 for any additional servers that you want to cache.
6. When you have finished specifying servers, click **Close** on the **Specify License Servers to Cache** dialog.

To delete a cached server, highlight the server in the list and click **Delete Selected Server**.

To edit a cached server, click on one of the listed servers and click **Edit Selected Server**. You can then change the ANSYS Licensing Interconnect port number or the hostname(s).

### 3.1.13. View Status/Diagnostic Options

The following reporting options are available:

- [Display the License Status](#) (p. 47)
- [Display Queued Licenses](#) (p. 48)
- [Display the Customer Number](#) (p. 49)
- [View the ANSYS Licensing Interconnect Debug Log File](#) (p. 49)
- [View the ANSYS FLEXlm License File](#) (p. 49)
- [View the ANSYS FLEXlm Debug Log File](#) (p. 49)
- [Gather Diagnostic Information](#) (p. 49)

When you select one of these options, the data will appear in the log area on the right side of the utility. **Display the Customer Number** appends the customer number to the session log itself; the other options create their own display in the log area. Each option is explained in the following sections.

Not all of these options are available for both the server and client versions of **ANSLIC\_ADMIN**.

#### 3.1.13.1. Display the License Status

Use this option to see the status of licensing activity. All features are displayed in the status window, along with the users of those features. You can append this information to the session log or write it directly to a file using the buttons at the bottom of the window. This option displays information based on the same [license path rules](#) as ANSYS, Inc. products, taking into account the settings in the **ANSYSLMD\_LICENSE\_FILE** environment variable and the `ansyslmd.ini` and license files. Setting the **ANS\_FLEXLM\_DISABLE\_DEFLICPATH** environment variable will affect the information displayed by this option.

3.1.13.2. Display Queued Licenses

Use this option to see a list of capabilities that are queued and awaiting availability, and the applicable licenses that are being used. The report will have the following format:

Capability	Timestamp	User	Count	Host	PID	Platform
Product1	Count					
	Timestamp	User1	Count	Host	PID	Platform
	Timestamp	User2	Count	Host	PID	Platform
	Timestamp	Usern	Count	Host	PID	Platform
Product2	Count					
	Timestamp	User1	Count	Host	PID	Platform
	Timestamp	User2	Count	Host	PID	Platform
	Timestamp	Usern	Count	Host	PID	Platform

Where the first line shows information for the requesting user, and subsequent lines under that capability show information for the users who are currently using the licenses for each of the licensed products that satisfy the requested capability.

An example report is shown below. Each line is numbered for reference in the discussion following the example. Note that the actual report does not contain line numbers.

Capability	Product	Timestamp	User	Count	Host	PID	Platform
1 ANS_SOLVER		2010/04/27 09:39:27	JQD	1	mach1.win.acme.com	1234	winx64
2	ane3fl			3			
3		2010/04/27 08:01:10	JXS	1	mach2.win.acme.com	5678	winx64
4		2010/04/27 07:29:13	MQD	2	mach3.win.acme.com	8765	winx64

Line 1: User *JQD* is running a process, *1234*, on the machine *mach1.win.acme.com*, which has requested one license with the *ANS\_SOLVER* capability.

Line 2: Based on the site/user preferences, the *ane3fl* product can satisfy the requested capability, and this site/user is permitted to use up to 3 licenses of this product.

Line 3: User *JXS* has been using 1 of the available licenses since *8:01:10*, from machine *mach2.win.acme.com*, running process number *5678*.

Line 4: User *MQD* has been using 2 of the available licenses since *7:29:13*, from machine *mach3.win.acme.com*, running process number *8675*.

Since users *JXS* and *MQD* are already using all of the available licenses, user *JQD* must either wait for one of those licenses to be freed, or contact one of those users and request that a license be freed.

Select the Auto-Refresh button to have the information automatically refresh every five (5) seconds.

You can also display the queuing report via command line, without launching the **ANSLIC\_ADMIN** GUI. To do so, run the following command on Windows systems:

```
"C:\Ansys Inc\Shared Files\Licensing\licadmin\anslic_admin" -queueinfo
```

If you need to view the queuing report frequently, you may find it convenient to set a desktop shortcut using the above command.

Run the following command on UNIX/Linux systems:

```
/ansys_inc/shared_files/licensing/lic_admin/anslic_admin -queueinfo
```

### 3.1.13.3. Display the Customer Number

Use this option to display your customer number (necessary when requesting customer support). The license manager must be running for you to obtain your customer number.

### 3.1.13.4. View the ANSYS Licensing Interconnect Debug Log File

This option is available only on the license server machine(s). Use this option to view the Licensing Interconnect debug log file (named `ansysli_server.log` by default).

### 3.1.13.5. View the ANSYS FLEXlm License File

This option is available only on the license server machine(s). This option displays the contents of your current license file in the reporting area. You can append this information to the session log or write it out to a separate file. Do not use this option to create a new license file. The **Write to File** option here will create a text file that includes extraneous information that is not valid in a license file, and should be used only to create a file for reference purposes.

### 3.1.13.6. View the ANSYS FLEXlm Debug Log File

This option is available only on the license server machine(s). Use this option to view the license log file (named `license.log` by default).

### 3.1.13.7. Gather Diagnostic Information

When you choose this option, you will be given the option to gather all relevant log and error files into a single directory. Files that will be gathered include the `license.log` file, the `ansysli_server.log` file, relevant `ansyslmd.ini` files, etc.

This feature is useful if you want to easily review the files from a single location, or if you need to compress the files and send them to a technical support representative. Every time you choose to gather these files, they will be copied into a new, date-stamped subdirectory in your `/var/tmp` directory on UNIX/Linux or in the directory specified by the **TEMP** environment variable on Windows. Be aware when choosing this option that some of these files can be large and could require significant disk space. You may wish to delete these directories when you are finished reviewing them because of the disk space usage.

After all relevant files are gathered (if selected), this option runs the `ansys_pid` utility, which queries the system for information that may be useful for troubleshooting certain problems. This operation may take a few minutes. You can append this information to the session log, or you can write it out directly to a file using the buttons at the bottom of the window.

When you run this utility, information about your licensing information is displayed in the **ANSLIC\_ADMIN** log area. Information displayed may include:

- operating system info
- environment information, including environment variable settings
- firewall information
- ANSYS, Inc. License Manager version and status information
- available license information

The contents of specific log and error files will not be displayed in the **ANSLIC\_ADMIN** log area. You will need to review these files individually for specific errors and other status/log information.

**Note**

If you have set the **ANS\_FLEXLM\_DISABLE\_DEFLICPATH** environment variable, the information will be displayed based on the **ANSYSLMD\_LICENSE\_FILE** environment variable setting; settings in the `ansyslmd.ini` file in the licensing directory will NOT be used.

### 3.1.14. Uninstall the License Manager

**UNIX/Linux** On UNIX/Linux machines only, you can use this **ANSLIC\_ADMIN** option to remove the entire ANSYS, Inc. license manager installation from your system. To use this option, you must be logged in as root or superuser. You should not uninstall the license manager if you still have products installed that use the license manager.

You will be asked to verify that the license manager should be uninstalled from this system.

The contents of the licensing directory will be deleted. Also, if the license manager is running, it will be shut down.

**Windows** On Windows machines, you must follow the procedure below. You should not uninstall the license manager if you still have products installed that use the license manager.

**License Servers** Follow these steps on a license server machine:

1. Stop the ANSYS, Inc. License Manager via the **ANSLIC\_ADMIN** utility (**Start> Programs> ANSYS Inc. License Manager> Server ANSLIC\_ADMIN Utility**).

2. Uninstall the ANSYS, Inc. License Manager service. You must use the following command to do so:

```
"C:\Program Files\Ansys Inc\Shared Files\Licensing\<platform>\ansysli_server"  
-k uninstall
```

3. Delete the licensing subdirectory (C:\Program Files\Ansys Inc\Shared Files\Licensing by default).
4. Remove the ANSYS, Inc. License Manager folder from the Start menu.
5. Remove the **ANSYSLIC\_DIR** and the **ANSYSLIC\_SYSDIR** environment variables, if set.

**Clients** Follow these steps on client machines:

1. Delete the licensing subdirectory (C:\Program Files\Ansys Inc\Shared Files\Licensing by default).
2. Remove the ANSYS 13.0> ANSYS Client Licensing folder from the Start menu.

---

## Chapter 4: End-User Settings

---

End users can set several controls for a customized licensing environment. These controls are explained in this chapter.

### 4.1. Client Environment Variable Settings

You can set following environment variables on individual machines to control their behavior, especially if you need it to be different from the general site configuration. These settings are especially useful for situations where you are testing a new product installation/license manager installation on a single machine before full site deployment, or other similar situations.

#### **ANSYSLMD\_LICENSE\_FILE**

Can be used to identify a license server machine or license file. If set, this specification is used before any other license path information. See [License Files Settings Precedence \(p. 51\)](#) for precedence information. The default port number assigned to ANSYS, Inc. is 1055. Therefore, if your server has the hostname alpha1 and the IP address of 10.3.1.69, you can identify the server to use as 1055@alpha1 or 1055@10.3.1.69.

#### **ANS\_FLEXLM\_DISABLE\_DEFLLCPTH**

Indicates that the default license path should not be searched when determining the licensing path in the ANSYS product:

- Only **ANSYSLMD\_LICENSE\_FILE** environment variable setting is used.
- Settings in the `ansyslmd.ini` file and the license file in the licensing directory will NOT be used.

#### **ANSYSLI\_SERVERS**

Used to identify the server machine for the Licensing Interconnect. Set to `port@host`. The default port is 2325. This setting takes precedence over settings specified in the `ansyslmd.ini` file.

#### **ANSYSLI\_TIMEOUT\_CONNECT**

Used to specify the amount of time that elapses before the client times out if it cannot connect to the server. Default is 20 seconds. Minimum timeout period you can specify is 5 seconds and the maximum is 60 seconds.

#### **ANSYSLI\_TIMEOUT\_TCP**

Used to specify the amount of time that elapses before the client times out if it cannot get a response from the server. Default is 60 seconds. Minimum timeout period you can specify is 30 seconds and the maximum is 300 seconds.

### 4.1.1. License Files Settings Precedence

ANSYS Licensing Interconnect settings have precedence in the following order:

1. **ANSYSLI\_SERVERS** environment variable
2. If **ANSYSLI\_SERVERS** is not set, use **ANSYSLMD\_LICENSE\_FILE** environment variable, replacing the FLEXlm port number with the default Licensing Interconnect port number of 2325.
3. Settings in the `ansyslmd.ini` file in the relative client licensing directory for **ANSYSLI\_SERVERS=** keyword



FLEXlm settings have precedence in the following order:

1. **ANSYSLMD\_LICENSE\_FILE** environment variable
2. Settings in the `ansyslmd.ini` file in the relative client licensing directory for **SERVER=** keyword.

---

**Note**

The FLEXlm environment variable **LM\_LICENSE\_FILE** is not supported with the ANSYS, Inc. License Manager.

## 4.2. Establishing User Licensing Preferences

Before running ANSYS, Inc. products, you should establish your licensing preferences, especially if you have multiple licenses at your site. Use the **Set License Preferences for User** option of the **ANSLIC\_ADMIN** utility or choose **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> User License Preferences** to set your own licensing preferences. Preferences specified with this option apply only to you and will not affect other users at your site.

You can use this option to review the licenses available, prioritize the licenses, and establish which licenses you will use. When you select this option, a dialog box opens. All licenses available to you on your license server machine(s) are shown under the appropriate tab:

- Solver
- PrepPost
- Geometry
- HPC

You control the order in which the system attempts to check-out a license by moving the licenses up and down in the list using the **Move Up** or **Move Down** buttons. When you open an application, the license manager attempts to check-out the license listed first in the list. If unable to check-out that license, then the license manager attempts to check out the next license in the list that can be used by the application. The license manager will continue through the list until an available license is found. If none are found, a message displays.

By default, all licenses available from your license server are shown and marked as available for use. If you do not want a particular license to be available for your use, select the license in the list, and set the **Use/Don't Use** field to 0.

If you received a new license, you may need to reset your preferences. If you had previously set your license preferences using this option, add the new license to your list in the desired order.

If you have made changes to your license preferences, you can return the settings to the default state by selecting the **Reset to Default** button.

When you have completed your changes, click **OK** to make the changes and close the dialog box. Click **Apply** to make the changes without closing the dialog box. Selections you make here will take effect with any new sessions/license checkouts but will not affect the current session or change the license(s) currently in use.

### 4.2.1. ANSYS Workbench Licensing Methods

ANSYS Workbench users can also specify the licensing method to use.



ANSYS Workbench offers two licensing methods at Release 13.0:

- Share a single license between applications (default) (shared mode)
- Use a separate license for each application (separate mode)

Use the Licensing Preferences dialog box (**Start > Programs > ANSYS 13.0 > ANSYS Client Licensing > User Licensing Preferences**) to specify which method to use and which licenses to use. You must specify the licensing method before starting an ANSYS Workbench session. If you access the Licensing Preferences dialog box from the ANSYS Workbench Tools menu, you will not be able to choose a licensing preference from there.

## Single License Sharing

ANSYS Workbench allows you to work across multiple applications and workspaces in ANSYS Workbench while consuming only one of a single type of license per user per session. Using shared licensing, the active application holds the license, preventing other applications that are sharing that license from using it during that time. The application or operation requiring use of the license is called a concurrency event. For example, meshing and solving would each be a concurrency event.

Single license sharing allows you to progress through your analysis, from specifying engineering data through building or attaching a geometry, meshing, setup, solving, and finally, reviewing your results, all under the same license. The application holding the license must close or issue a PAUSE command, or receive an automatic release request to release the license and allow another application to use it. Licenses cannot be released while an application is actively performing a licensed operation (for example, an application cannot release a license in the middle of a solve operation; the license cannot be released until the solve operation is completed).

Single license sharing applies only to licenses of the same type (e.g., Mechanical). Choosing this option does not affect your ability to use licenses of different types simultaneously (e.g., Mechanical for one task and Fluid Dynamics for another).

Because this method is the default, you do not have to take any action to run this way.

**Explanation of License Type and Examples** License type is primarily by license feature. It is possible to use both a Mechanical and an Emag license within a single ANSYS Workbench session. It is also possible to use both a Multiphysics and a Mechanical license within a single ANSYS Workbench session.

The first license checked out within a session will be based on your [preferences](#) and what capabilities are being requested. For all applications other than the first (subsequent) one opened (within ANSYS Workbench), ANSYS licensing will first look at what other licenses are opened within this session. These subsequent license requests will look at sharing first to satisfy their request: do any other licenses being used within this session fulfill the needed capabilities? If yes, share an existing license. If not, preferences are used and a new, different license is checked out.

**Example 1:** You have one license for Multiphysics and one license for Mechanical, with Multiphysics listed first in your preferences. The first application starts and only needs capabilities in Mechanical. Since Multiphysics contains Mechanical capabilities and is first in your preferences, Multiphysics will be checked out. The second application starts and needs Multiphysics; since Multiphysics is already checked out, the second application will share it with the first. Only the Multiphysics license is consumed in this session.

**Example 2:** You have one license for Multiphysics and one license for Mechanical, with Mechanical listed first in your preferences. The first application starts and only needs capabilities provided in Mechanical, so Mechanical is checked out. The second application starts and needs capabilities provided on Multiphysics;

since (the already in use) Mechanical cannot satisfy its requirements, it checks out Multiphysics. Both a Multiphysics and a Mechanical license are consumed in this session.

**Restrictions of Single License Sharing** You cannot run two concurrency events simultaneously (for example, you cannot mesh one model and solve another simultaneously) with one license.

If you are using a license for one application, other applications may still not be able to share that license if those applications require capabilities not supported by the license. For example, you cannot share a Mechanical license with a FLUENT application.

**Single License Sharing in ANSYS Workbench Applications** ANSYS Workbench applications handle single license sharing differently:

#### **The Mechanical Application:**

You can launch the Mechanical application and move between its components (such as Meshing, Setup, and Solve). The active component will control the license while completing its operations and will release the license as soon as the operation is completed. For example, when you mesh, the Meshing component will control the license during the meshing operation and then immediately release the license when the operation is completed. The other components will remain in a read-only mode while Meshing uses the license, allowing you to view the data in other components but not operate on it.

---

#### **Note**

Applications in read-only mode because of shared licensing do not refresh their license status automatically. Once the shared license is released by the editor that had consumed it, you must trigger Mechanical to query the license status. The most straightforward way to do this is click outside the Mechanical application window and then click back in the window to cause the license availability to be rechecked.

#### **The Mechanical APDL Application:**

This application consumes a license as soon as you launch it, and retains that license until it is finished. If you launch the Mechanical APDL application interactively, the license is retained until you either close the application or issue a PAUSE command at the Mechanical APDL command line. PAUSE allows you to temporarily release the license for another application to use. No other operation other than SAVE or /EXIT is permitted while PAUSED. When the second application has finished and releases the license, issue an UNPAUSE command from the Mechanical APDL command line to resume its use of the license.

#### **CFX, FLUENT, AUTODYN, POLYFLOW:**

These applications consume a license when launched and retain the license until they receive a request from another application to release it. For example, if you open CFX-Pre, CFX-Pre will obtain and control the license. It will retain the license until you close the application or until another application (such as the CFX solver) requests it.

AUTODYN and POLYFLOW also provide a manual PAUSE feature that allows you to interrupt AUTODYN or POLYFLOW and release the license, temporarily, for another application to use.

To unpause the Mechanical APDL application and resume its use of the license, issue UNPAUSE in Mechanical APDL.

### **Separate Licenses**

By using the separate licenses method, ANSYS Workbench requires a separate license for each application. By using this method, you can move freely between the many applications that you might require during an analysis in ANSYS Workbench, provided that you have sufficient licenses. You can leave each application

running and easily move between them at any point during the analysis, even if one of the applications is actively using the license (such as during a solve process). The disadvantage to this method is that you could potentially consume many licenses.

To activate the separate licenses method, choose **Use a separate license for each application** in the Licensing Preferences dialog box (**Start > Programs > ANSYS 13.0 > ANSYS Client Licensing > User License Preferences**). You must specify the licensing method before starting an ANSYS Workbench session.

**Examples of Using Separate Licenses** You have two Mechanical licenses. When you open and mesh or solve a model in the Mechanical application, you consume one Mechanical license. If you link that Mechanical analysis to a Mechanical APDL system, you would consume a second Mechanical license when you launch the Mechanical APDL application, if you have not closed out of the Mechanical application. Neither of these licenses would then be available for other users until you closed out of one or both of the applications.

## 4.2.2. HPC Licensing

ANSYS, Inc. offers multiple high performance computing license options, described below. These license options apply to ANSYS, Inc. commercial and academic associate licenses only. These license options do not apply to ANSYS LS-DYNA; see the [ANSYS LS-DYNA User's Guide](#) for details on parallel processing options with ANSYS LS-DYNA.

The HPC license options below cannot be combined with each other in a single solution; for example, you cannot use both ANSYS HPC and ANSYS HPC Packs in the same analysis solution.

See the applicable product documentation for instructions on configuring and running a distributed solution.

### ANSYS HPC

These physics-neutral licenses can be used to run multiple analysis jobs across multiple processors and work with most ANSYS, Inc. applications. Contact your ANSYS sales representative for a complete list of applications that can be used with ANSYS HPC.

The Mechanical APDL application allows you to use two processors without using any HPC licenses; ANSYS HPC licenses add to this base functionality. For example, a Mechanical APDL user using twelve processors (or cores) will consume only ten ANSYS HPC tasks.

ANSYS HPC licenses do work with ANSYS Workbench's Remote Solve capability; however, ANSYS HPC licenses are not subject to single license sharing as described in [ANSYS Workbench Licensing Methods](#) (p. 52).

You cannot combine ANSYS HPC licenses with any other type of HPC licenses, including HPC Packs, in the same solution.

ANSYS HPC licenses are not valid with academic products; instead, use the appropriate ANSYS Academic HPC license.

### ANSYS HPC Packs

These physics-neutral licenses can be used to run a single analysis across multiple processors and work with most ANSYS, Inc. applications. Contact your ANSYS sales representative for a complete list of applications that can be used with ANSYS HPC Packs. ANSYS HPC Packs enable parallel for single simulations according to the following schedule:

- 1 ANSYS HPC Pack per simulation: 8 processors
- 2 ANSYS HPC Packs per simulation: 32 processors

- 3 ANSYS HPC Packs per simulation: 128 processors
- 4 ANSYS HPC Packs per simulation: 512 processors
- 5 ANSYS HPC Packs per simulation: 2048 processors

You cannot use more than five HPC Packs on a single analysis. Individual HPC Packs cannot be split between multiple users or between multiple analyses.

Mechanical APDL allows you to use two processors without using any HPC licenses; ANSYS HPC Pack licenses function independently of this base capability. For example, a Mechanical APDL simulation using a single ANSYS HPC Pack can access up to eight processors (or cores), as noted above, and not ten processors (or cores).

HPC Packs do work with ANSYS Workbench's Remote Solve capability; however, HPC Packs are not subject to single license sharing as described in [ANSYS Workbench Licensing Methods \(p. 52\)](#).

You cannot combine ANSYS HPC Pack licenses with any other type of HPC licenses, including ANSYS HPC licenses, in the same solution.

ANSYS HPC Pack licenses are not valid with academic products.

GPU acceleration is allowed when using HPC Packs with Mechanical APDL or with the Mechanical Application. GPU acceleration is available on 64-bit Windows and Linux x64 systems only. One HPC Pack enables one GPU, in addition to enabling up to eight traditional processors. For more information on GPU acceleration, see [GPU Accelerator Capability](#) in the Mechanical APDL [Advanced Analysis Techniques Guide](#).

#### **Mechanical HPC, CFX Parallel Computing, CFD HPC, etc.**

These physics-specific licenses will continue to work. The Mechanical APDL application allows you to use two processors without using any HPC licenses; Mechanical HPC licenses accommodate this feature. For example, a Mechanical APDL user with two Mechanical HPC tasks can access four processors/cores.

You cannot combine physics-specific HPC licenses with any other type of HPC licenses, including ANSYS HPC or HPC Pack licenses, in the same solution.

#### **4.2.2.1. Specifying HPC License Order**

Use the **HPC** tab to define the order in which HPC licenses are used. By default, the physics-neutral HPC licenses are listed before the physics-specific HPC licenses. If you wish to change the order, select licenses and click the **Move up** or **Move down** button to reorder the list.

---

#### **Note**

Be aware that you can only use five (5) HPC Packs (a maximum of 2048 processors) for a single analysis. If you configure your distributed analysis such that it requires more than 2048 processors, the HPC Pack licenses will not be used, regardless of how many HPC Pack licenses are available. The analysis will use the next available HPC license category (for example, ANSYS HPC or Mechanical HPC), and will consume as many licenses as necessary to accommodate the specified number of processors.

### **4.3. Setting Up License Queuing**

The License Queuing option allows an ANSYS process to wait (queue) for an available license in the event all licenses are checked out. Activate this option by setting the environment variable **ANSWAIT** to 1.

If the **ANSWAIT** environment variable is not set, then the ANSYS process will terminate with the following message in the event all ANSYS licenses are checked out. This is the default condition.

```
Licensed number of users already reached (-4,132)
```

The License Queuing option may not be available on all products.

## 4.4. Using License Borrowing

License borrowing (available only on Windows machines) allows a user to take a license for use outside of the company facility, such as for an engineer to take a license home on his laptop. Before users can take advantage of license borrowing, the site administrator needs to set up the borrowing environment (see [Run the ANSYS Borrow Utility](#) (p. 39)).

License borrowing is available only if you have borrowable licenses and only on Windows machines; however, the license server machine that is managing the licenses can be any system that ANSYS, Inc. supports as a license server. You cannot run both borrowed and non-borrowed licenses simultaneously. For example, if you borrow licenses but remain connected to a network from which you can use licenses, you will be able to use only the borrowed licenses. Likewise, if the machine on which you borrowed licenses is also a license server from which you could normally use licenses, you would be able to use only the borrowed licenses.

1. Connect to the network where the license server machine is running.
2. Launch the client license borrowing utility: **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN utility**. Select **Run the ANSYS Borrow Utility**.
3. On the Borrow tab, specify the version number and return date. The return date defaults to the third day from the current date.

If you change any of the borrowing criteria, the product list will automatically update in five seconds. To update the list immediately, click the **Update Delay (sec)** button.

4. Select the product(s) you want to borrow. The product list includes add-on features or geometry interfaces. If you do not see a product listed that you think should be available, verify that you have selected the correct version.
5. Click **Borrow**.
6. Disconnect from the network. Note that even when you've disconnected from the network, you must still run the products from the same account you used to borrow the licenses. For example, if you use a network account to borrow the licenses, you cannot then run from a local account.

Be sure to verify that you can run the correct product with the borrowed licenses from your machine AFTER you've disconnected from the network. If you encounter any difficulties, reconnect to the network, return the license, and then re-borrow the correct license.

If you keep a license for the full amount of time, there is no need to check the license back in. Your borrowed license will simply expire at the end of the borrow period, and the license will be released into the pool of available licenses.

To return a license early:

1. Reconnect to the same license server from which you borrowed the license(s).
2. Launch the client license borrowing utility: **Start> Programs> ANSYS 13.0> ANSYS Client Licensing> Client ANSLIC\_ADMIN utility**. Select **Run the ANSYS Borrow Utility**.
3. Select the **Return** tab. Click **Return all**. If you wish to keep some but not all of the licenses you borrowed, you will need to re-borrow them.

4. Disconnect from the network.

The license(s) you borrowed will be returned to the pool of available licenses.

---

### Note

You must return borrowed licenses from the same account you used to borrow the licenses originally.

Some multi-feature products include other products and/or add-on features. The utility will automatically check out any such included products or add-on features. If you borrow one of these multi-feature products and then click on the **Return** tab, you will see the product you selected, as well as any other products and add-ons that it includes. If you return a multi-feature product early, these products and add-on features will also be returned.

---

## Chapter 5: Troubleshooting

---

This chapter lists problems and error messages that you may encounter while setting up licensing. After each situation description or error message is the user action required to correct the problem.

For more troubleshooting information, see the *FLEXnet Licensing End User Guide* (accessible from the **AN-SLIC\_ADMIN** utility) .

The following troubleshooting topics are available:

- [5.1. Getting Additional License Debug Information](#)
- [5.2. Gathering Diagnostic Information](#)
- [5.3. Problem Situations](#)
- [5.4. Licensing Installation Process Errors](#)
- [5.5. Licensing-Related Mechanical APDL Launcher Error Messages](#)
- [5.6. Licensing Error Messages](#)
- [5.7. ANSYS License Borrowing Errors](#)
- [5.8. FLEXlm License Log File Errors](#)

### 5.1. Getting Additional License Debug Information

Use the following suggestions to display or generate additional error messages and debugging information.

- View the `licdebug` file. The `licdebug` file is generated when you run an ANSYS, Inc. application and resides in the `.ansys` subdirectory under the directory specified by the **TEMP** environment variable (Windows) or in the `$HOME` directory (UNIX/Linux). The `licdebug` filename will vary depending on the product but will follow the format `licdebug.<product>.130.out`. For example:

If a `licdebug` file already exists and is dated today, the information is appended. If it is dated before today, the existing file will be renamed with a `.old` extension and a new file will be started.

- Mechanical APDL (ANSYS): `licdebug.ANS_SOLVER.130.out`
- ANSYS Workbench: `licdebug.ANS_WB.130.out`
- Mechanical: `licdebug.ANS_SIM.130.out`
- ANSYS FLUENT: `licdebug.FLUENT_SOLVER.130.out`
- ANSYS POLYFLOW: `licdebug.POLYFLOW.130.out`
- ANSYS CFX-Pre: `licdebug.CFX_PRE.130.out`
- ANSYS CFX Solver: `licdebug.CFX_SOLVER.130.out`
- ANSYS CFX-Post: `licdebug.CFX_POST.130.out`,
- ANSYS ICEM CFD (includes AI\*Environment): `licdebug.ICEM_CFD.130.out`
- ANSYS Icepak: `licdebug.ICE_PAK.130.out`
- ANSYS LS-DYNA: `licdebug.DYNA_SOLVER.130.out`
- Connection functionality: `licdebug.ANS_PM.130.out`



The directory specified by the **TEMP** environment variable may be hidden on your system. To view the directory and file, click on My Computer. Choose **Tools** from the menu, and then click on **Folder** options. Click on the **View** tab and select **Show hidden files and folders**. Click **OK**.

If after following these suggestions, the resulting debug information does not make sense, try these suggestions:

- Confirm that the license manager was restarted or the license file was reread after any changes were made to the license file. If the user did not make any changes to the license file for the server, check the date/time that it was last changed. Get the relevant path information from the debug output. Also, confirm that the same path is being used.
- Try restarting the license manager and then attempt to run again. See if the same situation occurs.
- If the user installed a new license file but is not seeing it even after restarting the license manager, confirm that the correct license file is being used to start the license manager. In this case, neither the client application nor the license manager is using the changed file. Also confirm that if site or user license preferences were set, the preferences were updated with the new license information.
- If the `license.dat` file is at the end of the path and it is a license file that uses the license manager daemon/service, then confirm that the license manager is started with the same path as the `license.dat` file's path. The license manager could be looking at one file in the client application but the license manager daemon/service was started with another file.

## 5.2. Gathering Diagnostic Information

You can use the **Gather Diagnostic Information** option of the **ANSLIC\_ADMIN** utility to query the system for various pieces of information that may be needed for troubleshooting certain problems. This option collects information about the system as well as about ANSYS licenses and sends all of the information that it collects to the log area. At times it may be necessary to provide the information for technical support. Use the **Write to File** button at the bottom of the **ANSLIC\_ADMIN** to write a file. Then forward the file to the appropriate person.

## 5.3. Problem Situations

This section describes problems you may encounter when setting up licensing or running an ANSYS product, as well as actions you can take to correct the problems.

### 5.3.1. License Manager Will Not Start

If the license manager will not start, perform the tests that are listed below. Perform the tests in the order that they are listed.

1. Check the `license.log` file in the licensing directory on the license server for errors. See the remainder of this chapter for a list of possible errors and their resolutions.
2. Check the `ansysli_server.log` file in the licensing directory on the license server for errors.
3. Verify that TCP/IP is installed and configured correctly. Verify that the IP address is static. See [Communications Requirements](#) (p. 12) for information about configuring TCP/IP.
4. Verify that the hostid has not changed. If the hostid has changed, you must obtain a new license.

**Windows Pentium III Systems** The version of FLEXlm used at ANSYS Release 13.0 no longer supports using a CPU ID as the hostid. As a result, if you are using a Windows Pentium III system as a license server, you will need to obtain a new license file to be able to run this release. To obtain your new license, run the **Register License Server Information** option on the **ANSLIC\_ADMIN** utility on the system you have designed.



nated as your license server. Send the resulting file `LICSERVER.INFO` to your ANSYS sales representative. We will then supply you with a new license file.

**Linux 32 and Linux x64 Systems** Linux 32 and Linux x64 systems running the ANSYS, Inc. License Manager require the Linux Standard Base (LSB) package. If this package is missing, you will see one of the following errors when the license manager attempts to run:

```
/lmgrd -h
./lmgrd: No such file or directory
```

In this case, try installing the LSB package from the Linux installation media.

**Three-Server Environment** In a three-server environment, you could also see a message similar to the following message when you start the first of the three servers:

**\*\*\*Attempting to start the license manager...**

**Start the License Manager status:**

**The license manager failed to start.**

This message results when a quorum of servers (2 of the 3) are not yet started. Start the other servers, or verify that they are already started. When at least two of the three servers are started, you will see the **ANSLIC\_ADMIN** status window of this server change to reflect that it is now running. Do NOT attempt to stop the license manager by pressing the **Stop the ANSYS, Inc. License Manager** button.

If you verify that at least two servers are started and you still see this message, and the status window does not update to running in a reasonable time, follow the three steps noted in [License Manager Will Not Start](#) (p. 60).

---

### Caution

When you see this message, do NOT press the **Start the ANSYS, Inc. License Manager** button again.

## 5.3.2. License Manager Will Not Stop

When you attempt to stop the license manager, it may not stop immediately. You should use the **Stop the ANSYS, Inc. License Manager** option of the **ANSLIC\_ADMIN** utility to stop the license manager. We do not recommend stopping the license manager manually; however, if you choose to do so, you must stop the components in the following order:

1. `ansysli_monitor`
2. Licensing Interconnect (`ansysli_server`)
3. FLEXlm components (`lmgrd` and `ansyslmd`)

## 5.3.3. License Manager Will Not Stop in a Three-Server Environment

When you attempt to stop the license manager in a three-server environment, the license manager does not stop immediately. You will see a message stating that the currently-installed license file is a three-server license file. With three-server license files, the license manager could take 60 seconds or longer to successfully stop. Wait at least 60 seconds. When the license manager has successfully stopped, the **ANSLIC\_ADMIN** status window will show that it is stopped.

### 5.3.4. The Application Does Not Show the Correct License(s)

If you do not see the correct licenses in your application, in **ANSLIC\_ADMIN's Set License Preferences for User** option, or in the Mechanical APDL Product Launcher, check the following:

1. If you received a new license but your application does not recognize it, you may need to reset your preferences. If you had previously set your license preferences, select **Set License Preferences for User** from the **ANSLIC\_ADMIN** utility and add the new license to your list in the desired order.
2. If you do not see the correct licenses in the **ANSLIC\_ADMIN's Set License Preferences for User** option or in the Mechanical APDL Product Launcher, and you updated your product order at a previous release, you may need to manually update your product order again. Use the **Specify Product Order** option under **ANSLIC\_ADMIN's Set Site Preferences**.

On the license server machine:

1. Select the **Reset to Default** button to see the current release's default product order.
2. Reorder the products as you wish, including any new products.

For more information on setting product order, see [Specify Product Order \(p. 45\)](#). For information on setting user license preferences, see [Establishing User Licensing Preferences \(p. 52\)](#).

### 5.3.5. I Do Not See an HPC Product Category in the Specify Product Order or the Set License Preferences for User Dialogs

If your site has HPC licenses but you do not see an HPC Product Category in the **Specify Product Order** or the **Set License Preferences for User** dialog boxes, you may already have site license preferences set (via the **Specify Product Order** option) from an earlier release. To correct this problem, reset the site preferences again after installing the current release by selecting the **Reset to Default** button and then resetting your product order preferences.

### 5.3.6. FLEXlm Log File Shows Unexpected Messages When the License Manager Is Stopped

On Windows systems, you may see unexpected error messages in the FLEXlm licensing log file when you shut down the license manager. Messages similar to the following could appear:

13:47:49 (lmgrd) Shutting down ansyslmd pid=1860 because of signal 15

13:47:49 (lmgrd) Can't connect to the license server system. Shutdown ansyslmd failed.

13:47:49 ((lmgrd)) 13:47:49 Loop info:(lmgrd) Cannot read data from license server system. (-16,10009:10054 "WinSock: Connection reset by peer")

MT:0 VD\_HB:5913:47:49 (lmgrd) reset:0Can't shutdown the license server system. Shutdown ansyslmd failed. clients:013:47:49 (lmgrd) fd's:0EXITING DUE TO SIGNAL 15

13:47:49 (lmgrd) ansyslmd exited with status 58 ()

13:47:49 (lmgrd) Since this is an unknown status, license server

13:47:49 (lmgrd) manager (lmgrd) will attempt to re-start the vendor daemon.

**13:47:49 (lmgrd) EXITING DUE TO SIGNAL 1**

**13:47:49 (lmgrd) Can't remove statfile C:\Documents and Settings\All Users\Application Data\Macrovision\FLEXlm\lmgrd.620: errno No such file or directory**

These or similar messages in the FLEXlm log file can be safely ignored in most cases. You can verify that the service is shown as stopped, and that the `ansysli_server`, `ansysli_monitor`, `lmgrd`, and `ansyslmd` processes are stopped via Task Manager.

### 5.3.7. Unable to Check Out Licenses

If the license manager appears to start but you cannot check out any licenses, perform the tests that are listed below. Perform the tests in the order that they are listed.

1. Review the Licensing Interconnect debug log file for more information. Use the **View Status/Diagnostic options > View the ANSYS Licensing Interconnect Debug Log File** option of the **ANSLIC\_ADMIN** utility (the Server **ANSLIC\_ADMIN** Utility on Windows) to view the file.
2. On the license server machine(s), review the FLEXlm `license.log` file for error messages. Use the **View Status/Diagnostic options > View the ANSYS FLEXlm Debug Log File** option of the **ANSLIC\_ADMIN** utility (the Server **ANSLIC\_ADMIN** Utility on Windows) to view the file.
3. If neither log file exists or has a zero length, verify that you have write permissions on the files and on the directory containing the files (the licensing directory by default).
4. If you have borrowed licenses, verify that you are not attempting to check out a non-borrowed license. Once you have borrowed one or more licenses, you will not be able to use non-borrowed licenses until you have returned all borrowed licenses.

You could also encounter problems with obtaining a license if the Licensing Interconnect is still attempting to restart the FLEXlm server. Check the FLEXlm `license.log` file. If you see the following lines multiple times, try rebooting your license server machine:

```
(lmgrd) ansyslmd exited with status 28 (Communications error)
(lmgrd) Since this is an unknown status, license server
(lmgrd) manager (lmgrd) will attempt to re-start the vendor daemon.
(lmgrd) REStarted ansyslmd (pid 103864)
```

### 5.3.8. Jobs Abort When License Manager Goes Down in Three-Server Environment

If you are running in a three-server environment and ANSYS program jobs abort when one of the license managers goes down, perform the tests that are listed below. Perform the tests in the order that they are listed.

---

#### Note

If this problem persists after you perform the steps below, you may want to consider switching to a single-server environment.

---

#### Note

When using a three-server environment, we strongly recommend that all three servers be on the same subnet. If this is not the case, you should consider using three machines which are in the same physical location and are on the same subnet.

1. Verify that two of the three license managers are still running. Use the **ANSLIC\_ADMIN** utility (**Display the License Status**). If the license manager is not already running on all servers, start it on all of them.
2. Check the `ansyslmd.ini` file in the licensing directory to make sure the server specification is correct.
3. Check to see whether the **ANSYSLI\_SERVERS** or **ANSYSLMD\_LICENSE\_FILE** environment variable is set. If it is, verify that the settings are correct.
4. Verify that the same license file exists on all three license servers in the licensing directory. The hostnames of all three servers must appear in these files and in the same order. If the license files are not the same on all three servers, make corrections as necessary and restart the license manager.
5. Verify that the date and time on all license servers are consistent and correct. Make corrections as necessary and restart the license manager.
6. Verify that the `hostid` has not changed on any of the license servers. If any `hostid` has changed, you need to obtain and install new licenses on all license servers.
7. Verify that the license manager and utilities are installed locally on each license server. If not, run the installation from the installation media and choose to install the license server only. See the installation manual for your product and platform for more information.
8. Verify that the `ansyslmd.opt` files match exactly on all three license servers.
9. Check the `FLEXlm license.log` and the licensing interconnect `ansysli_server.log` files for error messages. See the remainder of this chapter for a list of possible errors and their resolutions.

### 5.3.9. Licensing Log File Not Created

On Windows, if the license manager starts and licenses can be taken from the license server, but the FLEXlm licensing log file and the `ansysli_server.log` file are not written, check the Administrator account. Verify that the Administrator (as well as the individual user) has write privileges to the directory where the log files are to be written.

### 5.3.10. Queuing Does Not Work

If queuing does not work, refer to the following list of possible causes and corrections:

- Verify that the environment variable **ANSWAIT** is set to 1.
- The license manager is down. In this case, use **ANSLIC\_ADMIN** to start the license manager.
- The hostname is mistyped in the `ansyslmd.ini` file or the **ANSYSLMD\_LICENSE\_FILE** environment variable. If so, correct the name and retry.
- You are not licensed for the requested feature. In this case, request a product for which you are licensed.
- All licenses for the requested feature are reserved in the license options file. If so, have the license administrator correct the license options file.
- The user is excluded via the license options file from using the licenses for the feature(s) that are installed on this machine.
- You are running an application that does not support queuing.

### 5.3.11. The FLEXlm Utility `lmutil` Does Not Shut Down License Manager

If you attempt to use the FLEXlm utility `lmutil` from an earlier release (such as the FLEXlm 10.8.8 version of the daemons) to shut down the ANSYS, Inc. Release 13.0 License Manager, you may see the prompt to shut down the license manager repeatedly:

```

lmutil - Copyright (c) 1989-2008 Acresto Software Inc. All Rights Reserved.
[Detecting lmgrd processes...]

Port@Host          Vendors
1) 1055@pghxpuser7  ansyslmd

Are you sure (y/n)?  y

```

If the current version of the `lmutil` is not in the licensing directory, the older version of `lmutil` may or may not shut down the `ansyslmd` daemon, but it will not shut down the `lmgrd` daemon.

To correct this error, use the most current version of `lmutil` to shut down the license manager.

### 5.3.12. The FLEXlm Utility `lmcksum` May Give Misleading Information

The FLEXlm utility `lmcksum` may give misleading information. Do not rely on its results to verify the accuracy of your license file.

### 5.3.13. The Mechanical APDL Launcher is Excessively Slow to Start

If the Mechanical APDL launcher takes an excessively long time to startup (Windows only), make sure the hostnames in the `ansyslmd.ini` file and in the `ANSYSLMD_LICENSE_FILE` and the `ANSYSLI_SERVERS` environment variables are typed correctly and that the hosts specified by the hostnames exist. Replacing hostnames with IP addresses may improve the speed as well. Also verify that the port number is correct.

### 5.3.14. Mechanical APDL Launcher is Not Using `ANSYS130_PRODUCT` Environment Variable Correctly

If the Mechanical APDL Product Launcher does not use the product specified by the `ANSYS130_PRODUCT` environment variable the first time that the launcher is run, you may need to clear your profiles and settings. In the launcher, choose **Options >Delete All Settings/Profiles**. This option will remove any profiles you have saved for this release, and revert back to using the `ANSYS130_PRODUCT` setting. Please see the discussion on *Launcher Menu Options* in the *Operations Guide* for more details on using the **Delete All Settings/Profiles** option before choosing this option.

### 5.3.15. Cannot Run a Product Listed in the Mechanical APDL Product Launcher

When running the Mechanical APDL Product Launcher, if you see the products listed correctly under the License field but are not able to run a product (you may see various error messages), the FLEXlm license path (usually `<port>@<host>`) may not be specified correctly. If you see the products listed but are not able to run a product, select **Tools> Display License Status** and review the FLEXlm license path for errors.

### 5.3.16. Remove a Client Option Does Not Work

Due to FLEXlm limitations, the **Remove a Client** option may not work correctly for products running in the Workbench environment. If it does not, you will need to shut down and restart the license manager in order to remove a client.

### 5.3.17. No Licensing Interconnect or FLEXlm Path Available from Display the License Status Option

If you select the **Display the License Status** option from the **ANSLIC\_ADMIN** utility and see a message stating that the Licensing Interconnect or FLEXlm path is empty, try the following:

1. Use the **Specify the License Server Machine** option of the **ANSLIC\_ADMIN** utility to specify the missing Licensing Interconnect or FLEXlm port number. (On Windows, use the Client **ANSLIC\_ADMIN** utility.) If the machine is already specified but the error message states that no FLEXlm path has been specified, the FLEXlm port number may be missing from the license server specification shown in the Specify the License Server window. Add the machine's FLEXlm port number using the **Specify the License Server Machine** option to add this machine to your FLEXlm license path.

---

#### Note

This particular machine may have been configured to run the ANSYS Licensing Interconnect without starting FLEXlm. Verify that FLEXlm is running on this machine before changing this setting.

2. Check the **ANS\_FLEXLM\_DISABLE\_DEFLLCPATH** environment variable setting. If this environment variable is set, the ANSYS, Inc. License Manager will not use the license server machine specifications in the `ansyslmd.ini` file. In this case, only the **ANSYSLMD\_LICENSE\_FILE** and **ANSYSLI\_SERVERS** environment variable settings will be used.

See [Specify the License Server Machine](#) (p. 35) for more information on specifying license server machines.

### 5.3.18. Cannot Enter Data in Text Fields

On some SUSE Linux systems and possibly other systems, if you cannot enter data in text fields during the installation or when using the **ANSLIC\_ADMIN** utility, you may be encountering a Tcl incompatibility. To correct the problem, unset the following environment variables before running the installation or the **ANSLIC\_ADMIN** utility:

```
QT_IM_MODULE
XMODIFIERS
GTK_IM_MODULE
```

You should reset these environment variables when you are finished running the installation or using **ANSLIC\_ADMIN**. Do not permanently unset these environment variables as doing so could affect other applications.

### 5.3.19. Cannot See the Entire Run License Wizard Dialog Box

On some SUSE Linux systems and possibly other systems, you may not be able to see the entire dialog box when you select **Run License Wizard** from the **ANSLIC\_ADMIN** utility. To correct the problem, resize the window vertically until the entire dialog box is visible.

### 5.3.20. Licenses Remain Checked Out After Job Completes

On HP platforms, licenses might remain checked out after a job completes if the license server is disconnected or crashes. In this situation, restart the license server or run the FLEXlm `lmremove` utility to release the license(s).



## 5.4. Licensing Installation Process Errors

If you encounter a problem during the configuration steps of the licensing installation process that results in a failure or abort, or if you are concerned that the licensing installation did not complete correctly, try running the **Complete Unfinished Licensing Installation Configuration** option from the **ANSLIC\_ADMIN** utility's **Tools** menu. This option will only repeat the licensing installation configuration steps that correspond to the version of the **ANSLIC\_ADMIN** utility you are running (client or server) and cannot be used as an alternative means of installing.

## 5.5. Licensing-Related Mechanical APDL Launcher Error Messages

Some messages that you see from the Mechanical APDL launcher are related to product licensing. These messages follow. For additional Mechanical APDL launcher error messages, see the Troubleshooting section of the *ANSYS, Inc. Installation Guide* for your platform.

### Various Messages

If you see any licensing-related messages, the launcher has encountered an error while trying to find or retrieve a license. Possible causes are the license manager is not running, no licenses were found, or the existing licenses have expired. If any of these messages appear, perform the tests that are listed below in the order presented:

1. Verify that the license manager is running on all license servers by viewing the status window on the **ANSLIC\_ADMIN** utility. If the license manager is not running, start it on the license server machine(s).
2. Check the `license.log` and `ansysli_server.log` file on each license server for errors.
3. Run Mechanical APDL (ANSYS) from a command prompt to check for additional error messages.

Example (Windows):

```
"C:\Program Files\Ansys Inc\V130\ANSYS\bin\intel\ansys130"
```

Check the Mechanical APDL (ANSYS) Output window for error messages and additional information about the failure.

4. Use the **ping** command to test communication between the client machine and the license server.

If the **ping** command does not work, modify the hosts file or the DNS server to contain the correct host names and IP addresses. Verify that the license server and the client machines have unique IP addresses.

5. Verify that the `ansyslmd.ini` file exists in the client licensing directory and that it contains the correct hostname and port numbers of the license server. If this file does not exist, see [Specify the License Server Machine](#) (p. 35) to specify the license server. If the **ANSYSLMD\_LICENSE\_FILE** variable is used to specify the license server, verify that the hostname of the license server is correct.
6. Verify that the correct version of the license manager daemons (`lmgrd` and `ansyslmd`) is being used. The current version is FLEXlm 11.8.

**\*\*\*Unable to load profile <name> because simulation environment <simulation environment> is not available.**

If you see this message, the simulation environment specified in the named profile was uninstalled since the profile was created. You will need to specify a new simulation environment and re-specify any

launcher settings to continue. To avoid seeing this message in subsequent launcher sessions, update and resave the profile, or reinstall the desired simulation environment.

## 5.6. Licensing Error Messages

### \*\*\* Licensed number of users already reached.

You may have reached the number of ANSYS product tasks that you have licensed. Wait until a task has been freed up and try again. If this error occurs on a regular basis, you may want to talk to your ANSYS sales representative about obtaining additional licenses.

This error may also occur when licenses have been reserved for certain users, or when certain users have been excluded from licenses via the license options file.

### \*\*\* All licenses are reserved for others.

This error may occur if you have multiple INCREMENT lines for the same feature in the license file, and there are differing fields on each INCREMENT line (such as `VENDOR_STRING` and the version field), and licenses have been reserved for this feature. In this situation, the number of reserved licenses is reserved out of each INCREMENT, rather than the sum total of both.

### \*\*\* Cannot connect to license server.

or

### \*\*\* Unable to connect to FLEXlm license server

The license manager is currently not running. Use the **ANSLIC\_ADMIN** utility to start the license manager. Your license must exist on the license server prior to starting the license manager.

If you install the license manager and a license file, and it appears that the server is running, but you cannot connect to your license server, try rebooting.

If you think the license manager should already be running, check the FLEXlm license log file and the `ansysli_server.log` file for errors.

Some other possible causes for this error include:

- If Windows Firewall is enabled on the license server, `ansyslmd.exe`, `lmgrd.exe`, and `ansysli_server.exe` need to be included in the exceptions.
- The wrong hostname is in the license file (machine ID is correct in the file and when the license keys were generated). The license manager could not be started.
- The wrong license file is on the system (both hostname and machine ID are incorrect and the license keys were created with the wrong ID). The license manager could not be started.
- The wrong license file is on the system AND the user changed the server line to have the machine's `hostid` but not the `hostname`. The license manager could not be started.
- An incorrect port number was used on the `SERVER` line(s) in the `ansyslmd.ini` file or in the setting of the licensing path environment variable **ANSYSLMD\_LICENSE\_FILE**
- All of the installed license files have expired.

If none of the above causes are applicable, the Licensing Interconnect may not have started because it was not able to start the FLEXlm server or may have exited after failing to restart the FLEXlm server.



Check the FLEXlm license log file (`license.log`). If you see the following lines multiple times, try re-booting your license server machine:

```
(lmgrd) ansyslmd exited with status 28 (Communications error)
(lmgrd) Since this is an unknown status, license server
(lmgrd) manager (lmgrd) will attempt to re-start the vendor daemon.
(lmgrd) REStarted ansyslmd (pid 103864)
```

### \*\*\* Feature removed during Imrread, or wrong SERVER line hostid

Some possible causes of this message include:

- There is a typographical error in the license file's SERVER line. Typically, the machine ID was mistyped when manually entering the license file on a machine.
- The user tried to install another machine's license file on a machine and changed the SERVER line in the file to have this machine's hostname but not the machine ID.
- A laptop with a docking station (with an Ethernet card in it) is removed from the docking station and the license file was made using this card's Ethernet address.
- The feature was removed during Imrread but the client is reading an old copy of the license file that still contains the removed feature.

### \*\*\* Clock difference too large between client and server.

This message will appear if the date on the client machine has changed. The date should never be set ahead or behind the actual date.

### \*\*\* License server does not support this feature.

This message can be generated if the feature has expired (on the server), or has not yet started, or the version is greater than the highest supported version, or the license file has been edited while the license server is running and the license file is not reread. Additional causes for this message include:

- If the user is trying to install another machine's license file on a machine AND changes both the hostname and the machine ID information on the SERVER line to be this machine's information. The INCREMENT lines contained in the license file are still invalid for this machine.
- If new information for the requested feature was added to the license file but neither an Imrread was issued nor was the license manager restarted.
- If a laptop with a docking station (with an Ethernet card in it) is removed from the docking station, and the license was made using this card's Ethernet address, AND the user edited the SERVER line, replacing the Ethernet address with the disk signature of the machine (in an attempt to fix the problem). Such an attempt will not succeed because the INCREMENT lines were created using the docking station's Ethernet address, which in this situation can no longer be seen on this machine.
- If one or more INCREMENT lines that were made for machine A were installed in machine B's license file and those features were trying to be checked out from machine B.

### \*\*\* Feature has expired.

Your license has expired; please contact your ANSYS sales representative for a renewal.

### \*\*\* No such feature exists.

- A license does not exist for the requested product. Verify that the **ANSYSLMD\_LICENSE\_FILE** environment variable and the `ansyslmd.ini` file are pointing to the correct license server.

- Verify that you are running the correct product. Use the **Set License Preferences for User** option of the **ANSLIC\_ADMIN** utility to specify the products and the product order you want to use.
- If you are not licensed to run this product, contact your ANSYS sales representative to obtain a license.
- If you are licensed to run this product, install the license from the license supplied by ANSYS, Inc. or your ANSYS sales representative using the **ANSLIC\_ADMIN** utility. If you have already created the license, use **ANSLIC\_ADMIN** to reread the license file.
- You could also see this message if you are trying to use a license manager daemon from a previous release. The license manager daemons (`lmgrd` and `ansyslmd`) at this release are FLEXlm 11.8.

**Failover feature <product name> specified in license preferences is not available.**

You may see this message if you specify a particular license (such as with the `-p` command line option on the Mechanical APDL (ANSYS) product) and no license for that product is currently available in the license path. Check the following:

- The license may have expired.
- All of your licenses may be in use.
- You do not have the specified license.
- The license is not included in your license preferences as set with the **Specify License Preferences for User** in the **ANSLIC\_ADMIN** utility.
- Your TECS for this version may have expired.

**\*\*\* License file does not support this version.**

The build date in the program is newer than the version in the license file. You may not be authorized to run a new release or you may not have installed your new license. If this is the case, install the license using the **ANSLIC\_ADMIN** utility. If you have installed the new license, use **ANSLIC\_ADMIN** to reread the license file. To view the build date in Mechanical APDL (ANSYS), use the `-v` command line option.

**\*\*\* Invalid (inconsistent) license key**

The license manager daemons for this release are FLEXlm 11.8 (`lmgrd` and `ansyslmd`). Verify that you are not using the license manager daemons supplied with a previous release.

This error can also occur if the license key contains incorrect characters or format (sometimes caused by transferring the file).

**Invalid license file: None of the hostnames in the license file match the system hostname.**

This message occurs when you install the license file if the hostname of the system you are running on does not match any of the hostnames in your license file. Some possible reasons why this can happen include:

- You are attempting to run on a system that has not been included in your license.
- The hostname of the system you are working on has been changed since it was used to generate the license information.
- A typographical error was introduced when the hostname was entered when creating the license key.
- On some newer Linux machines, the system hostname is not listed consistently in the `/etc/hosts` file. When you run the **Register License Server Information** or the **Display the License Server**

**Hostid** option of the **ANSLIC\_ADMIN** utility, the utility uses the localhost information instead of the correct system hostname. An example of the problematic `/etc/hosts` file might look like this:

```
# Do not remove the following line, or various programs
# that require network functionality will fail.
127.0.0.1      localhost.localdomain  localhost      abclinux5
10.99.9.99     abclinux5             abclinux5
[root@abclinux5 ~]$ hostname
abclinux5
[root@abclinux5 ~]$ hostname -s
localhost
[root@abclinux5 ~]$
```

In this example, the **Register License Server Information** or the **Display the License Server Hostid** option would incorrectly use the localhost information.

To correct the problem, you need to edit the `/etc/hosts` file and change the order of the hostname lines. The example shown above would then look like this:

```
# Do not remove the following line, or various programs
# that require network functionality will fail.
10.99.9.99     abclinux5             abclinux5
127.0.0.1      localhost.localdomain  localhost      abclinux5
[root@abclinux5 ~]$ hostname
abclinux5
[root@abclinux5 ~]$ hostname -s
abclinux5
[root@abclinux5 ~]$
```

In the corrected example, the **Register License Server Information** or the **Display the License Server Hostid** option would correctly use the abclinux5 hostname.

### \*\*\* Local checkout filter rejected request.

View the license debug log files (`FLEXlm license.log` and the Licensing Interconnect `ansysli_server.log`) to find the specific cause of this message.

### \*\*\* Cannot find server hostname in network database.

This message will appear if one or more of the SERVER computer names do not appear in the client computer's `/etc/hosts` file (UNIX/Linux) or the `services` file (Windows).

### \*\*\*The ANSYS license manager server is down. Unless a connection is reestablished, ANSYS will exit in *nn* minutes.

A message similar to this one occurs in a one-server license environment if your license manager has quit running. In a three-license server environment, the ANSYS, Inc. license manager must be running on at least two of the three license server machines at all times. If two of the license server machines go down, or two of the machines are not running the license manager, this error message will appear in the program output or in a message box. The program will continue to run for *nn* minutes to allow the license manager to be restarted or to be started on a second machine if using redundant servers. When this error message appears, start the license manager on the other machines designated as license servers.

If you get this message and determine that the license manager **is** still running, and you are running in a one-server environment, then the IP address of the license server machine was changed while the ANSYS product was running (this is usually caused by connecting to or disconnecting from an Internet Service Provider (ISP) that dynamically allocates IP addresses). To correct this situation, you must return the IP address to the same address that the license server had when the ANSYS product was started.

If the IP address changes after you start the ANSYS product (either because you connected to or disconnected from your ISP), you can correct the error by restarting the ANSYS product. You should not need to restart the license manager.

You can avoid this problem by remaining connected to or disconnected from the ISP the entire time you are running the application.

**\*\*\*FLEXlm version of vendor daemon is too old.**

The license manager daemons at this release are FLEXlm 11.8 (lmgrd and ansyslm). Verify that you are not using the license manager daemons supplied with a previous release.

**\*\*\*Cannot read data from license server.**

The server and the client are having difficulties communicating. This error is usually caused by the network setup or by the network data being different from the data in the license file (for example, the server name, machine ID, port number, or vendor daemon).

- Confirm that the information on the SERVER line of the license file is the correct information for the server.
- Confirm that the syntax of the SERVER and VENDOR lines of the license file is correct.
- Verify that TCP/IP is enabled correctly.
- Check that the hosts file on the client machine contains the license server.
- Try using `<port>@<ipaddress>` when specifying the server.

**\*\*\*The license server is taking too long to respond. The application has stopped waiting for a reply. The license server may be experiencing a high demand or a temporary outage. Try again later.**

If you see the above message and the server is not experiencing any demand, a virus scanner may be blocking connections to your license server. Rebooting your license server may also resolve the issue.

**Could not bind socket on port 2325. Address already in use.**

If you see the above message in your licensing log file, your operating system was not able to free a port before the timeout period expired. This situation could happen if the port is being used by some other application, or after you shut down a server if you did not wait long enough for the ports to free before restarting it. This situation appears to occur more frequently on HP and Sun systems, because these operating systems have a default timeout period to free a port that may not be sufficient.

On HP systems, the default is unlimited for the FIN\_WAIT\_2 port state. If the machine has an unlimited default timeout period, then the port will never be freed. To determine the current timeout period, issue the command:

```
ndd -get /dev/tcp tcp_fin_wait_2_timeout
```

If the setting is unlimited, this command will return a value of 0.

You can adjust the default timeout period to 30 seconds by issuing the following command:

```
ndd -set /dev/tcp tcp_fin_wait_2_timeout 30000
```

To verify that the setting is now 30 seconds, re-issue the following command:

```
ndd -get /dev/tcp tcp_fin_wait_2_timeout
```

The command should return a value of 30000 (30 seconds).

If you wish to retain this setting after the machine is rebooted, you must edit the `/etc/rc.config.d/nddconf` file as follows:

```
TRANSPORT_NAME[1]=tcp
NDD_NAME[1]=tcp_fin_wait_2_timeout
NDD_VALUE[1]=30000
```

On Sun systems, the default is 4 minutes for the CLOSE\_WAIT and TIME\_WAIT port states. You can adjust the default timeout period to 30 seconds by issuing either of the following commands as appropriate for your operating system level:

```
ndd /dev/tcp -set tcp_time_wait_interval 30000
ndd /dev/tcp -set tcp_close_wait_interval 30000
```

Consult your system administrator before making changes to your default operating system settings or to determine which command is appropriate for your operating system level.

**\*\*\*Failed to retrieve license preferences. Please be sure that you are able to connect to the license server.**

If you attempt to set your license preferences and the client information is at a higher release than the server information, you may see this message. To verify, see the `getuserprefs.log` file (on UNIX/Linux machines, in `.ansys` in your home directory; on Windows, in `%TEMP%\ .ansys`). Note the last lines in the example below:

```
2009/08/24 17:18:00 INFO Not connected to a local port.
2009/08/24 17:18:00 NEW_CONNECTION
Connected to license server: 2325@linux10.ansys.com.
ANSYSLI_SERVERS: 2325@linux10.ansys.com
FLEXlm Servers: 1055@linux10.ansys.com
2009/08/24 17:18:01 SITE_PREFS 2009.0806
The version of the license server 2325@linux10.ansys.com [1.1.2]
must be greater or equal to the client ALI_UTIL version [1.1.3].
Please update your license server to the latest version.
```

**\*\*\*License File *filename* has changed. Please do a reread to update the server.**

The license file or the FLEXlm options file has been changed since the last restart of the license manager or reread of the license file. The changes will not be available until you restart the license manager or reread the license file. If you are running the ANSYS Licensing Interconnect and FLEXlm independently and have cached the license file, the cached license file may be out of date. Restart FLEXlm or reread the license file to update the cached license file. If you make changes to the license file, in addition to rereading the license file in FLEXlm, you must also recache the license file in the Licensing Interconnect by restarting the Licensing Interconnect or by using the **Reread the License Manager Settings** option of **ANSLIC\_ADMIN**.

## 5.7. ANSYS License Borrowing Errors

**\*\*\*Unable to return <Product>, encountered "feature not found" attempting to return license feature <Product>.**

or the following will appear in the log file:

**UNAUTHORIZED Imremove request from <user> at node <host>.**

If you see the above message when attempting to return a borrowed license before the expiration date, the `-2 -p` license manager startup option might have been set. The `-2 -p` option is not supported. If you are running an earlier release, you must manually unset the option.

## 5.8. FLEXlm License Log File Errors

### \*\*\*Not a valid server hostname, exiting. Valid server hosts are: xxx

The hostname in the SERVER line of the license file does not match the hostname of the system. In this case, change the hostname in the license file. Other possible causes include:

- The license file was put on a machine other than the one for which it was created; therefore, the hostname is not correct.
- The hostname of the license server changed. Change the hostname on the SERVER line of the license file to correct this problem. REMEMBER to send ANSYS the new hostname so that future license files are made with the correct hostname.
- The license file was put on a machine other than the one for which it was created AND the user changed the machine's ID on the SERVER line but did not change the hostname. If the machine ID is also wrong, you will also see a message about the wrong hostid on SERVER line after correcting the hostname.
- The wrong hostname was used to create the license file. As long as the correct machine ID was used, you should be able to change the hostname on the SERVER line of the license file to fix this problem. REMEMBER to send ANSYS the correct hostname so that future license files are made with the correct hostname.

### \*\*\*Invalid license key (inconsistent authentication code)

The INCREMENT lines in the license file are not valid for this system (HostID mismatch). This error will also occur if the license file was typed incorrectly. Additional causes for this message include:

- If a license file created for machine A is being installed on machine B AND the user changed the hostname and the machine ID to machine B's in the SERVER line.
- If INCREMENT lines created for machine A are appended to an existing `license.dat` file on machine B.
- If a laptop with a docking station (with an Ethernet card in it) is removed from the docking station, and the license was made using this card's Ethernet address, AND the user edited the SERVER line, replacing the Ethernet address with the disk signature of the machine (in an attempt to fix the problem). Such an attempt will not succeed because the INCREMENT lines were created using the docking station's Ethernet address, which in this situation can no longer be seen on this machine.
- In a three-server license environment, if the license file was created using an incorrect hostid for one of the license servers, this message will appear and will prohibit an ANSYS product from running on any machine. Verify that the hostid in the license file matches the hostid on all three license servers. If it does not, the license file must be created using the correct hostid for all license server machines.

### \*\*\*Wrong HostID on SERVER line for license file: <path to license file>

License file is not valid for this machine. Possible causes include:

- There is a typographical error. If the license file was entered manually, the machine ID may have been incorrectly typed in the SERVER line.

- The user installed a license file created for machine A on machine B and changed the hostname in the SERVER line but did not change the machine ID.
- The hostid changed on a machine but the hostname stayed the same.
- A laptop with a docking station (with an Ethernet card in it) is removed from the docking station, and the license file was made using this card's Ethernet address.
- A license file was made for a Pentium III CPU ID and the hyphen was not put after the first four characters of the ID (e.g., ABCDEF01 was entered as the ID rather than ABCD-EF01). Ethernet address would be expected.
- A license file was made using the Windows disk signature but the keyword 'DISK\_SERIAL\_NUM=' was not put before the ID. Ethernet address would be expected.

**\*\*\* "XXX" : Not a valid server hostname, exiting. Valid server hosts are: "YYY"**

The hostname listed in the `license.dat` (YYY) file does not match the hostname of the system (XXX).





---

## Chapter 6: Product Variable Table

---

The following table shows all ANSYS, Inc. products and their associated feature name as used in the INCREMENT lines.

This table is provided as a reference for both current and legacy products. Not all products listed are generally available but are included here for customers running an older release. Please contact your ANSYS sales representative or technical support representative for details regarding product availability.

**Table 6.1 Product/Feature Names for Licensed Products**

Product	Feature Names
ANSYS Multiphysics	ane3fl*
ANSYS Multiphysics/LS-DYNA	ane3flds*
ANSYS Multiphysics 1	ane3fl1*
ANSYS Multiphysics 2	ane3fl2*
ANSYS Multiphysics 3	ane3fl3*
ANSYS Multiphysics/LS-DYNA PrepPost	ane3fldp*
ANSYS Mechanical	ansys*
ANSYS Mechanical/Emag	ane3*
ANSYS Mechanical/FLOTRAN	anfl*
ANSYS Mechanical/LS-DYNA	ansysds*
ANSYS Mechanical/Emag/LS-DYNA	ane3ds*
ANSYS Mechanical/CFD-Flo/LS-DYNA	anflds*
ANSYS Mechanical/LS-DYNA PrepPost	ansysdp*
ANSYS Mechanical/Emag/LS-DYNA PrepPost	ane3dp*
ANSYS Mechanical/CFD-Flo/LS-DYNA PrepPost	anfldp*
ANSYS Mechanical/CFD-Flo	ancfx*
ANSYS Structural	struct*
ANSYS Structural/Emag	ste3*
ANSYS Structural/FLOTRAN	stfl*
ANSYS Structural/Emag/CFD-Flo	ste3fl*
ANSYS Structural/LS-DYNA	structds*
ANSYS Structural/Emag/LS-DYNA	ste3ds*
ANSYS Structural/CFD-Flo/LS-DYNA	stflds*
ANSYS Structural/Emag/CFD-Flo/LS-DYNA	ste3flds*
ANSYS Structural/LS-DYNA PrepPost	structdp*
ANSYS Structural/Emag/LS-DYNA PrepPost	ste3dp*

Product	Feature Names
ANSYS Structural/CFD-Flo/LS-DYNA PrepPost	stfldp*
ANSYS Structural/Emag/CFD-Flo/LS-DYNA PrepPost	ste3fldp*
ANSYS Structural/CFD-Flo	stcfx*
ANSYS Professional NLT	prf*
ANSYS Professional NLS	prfnls
ANSYS Professional/Emag	prfe3*
ANSYS Professional/FLOTRAN	prffl*
ANSYS Professional/Emag/FLOTRAN	prfe3fl*
ANSYS Emag	emag*
ANSYS Emag HF	emaghf*
ANSYS Emag/FLOTRAN	emagfl*
ANSYS FLOTRAN	flotran*
ANSYS Mechanical PrepPost	preppost*
ANSYS PrepPost/LS-DYNA PrepPost	prpostdy*
ANSYS LS-DYNA	dyna*
ANSYS LS-DYNA PrepPost	dynapp*
ANSYS LS-DYNA PC	dynapc*
ANSYS Multiphysics Solver	mpba*
ANSYS Mechanical Solver	meba*
ANSYS Professional Solver	prba*
ANSYS Structural Solver	stba*
ANSYS FLOTRAN Solver	flba*
ANSYS Emag Solver	e3ba*
ANSYS DesignSpace Batch Child	debatch
ANSYS Emag Batch Child	e3bach
ANSYS FLOTRAN Batch Child	flbach
ANSYS Mechanical Batch Child	mebach
ANSYS Multiphysics Batch Child	mpbach
ANSYS Professional Batch Child	prbach
ANSYS Structural Batch Child	stbach
ANSYS DesignSpace Entra	caewbpl1
ANSYS DesignSpace Structural	dsstruct
ANSYS DesignSpace	caewbpl3
ANSYS Workbench SDK	caewbplh, caetmpl, piautoin, pimedesk, pisolwor, pisoledg, piproe, dsdxm
ANSYS Workbench Runtime	caewbplh, caetmpl

Product	Feature Names
ANSYS CFX-CAD2Mesh with DesignModeler	cad2mesh, aimed, aioutput, aiprism, aitetra, agppi, aiiges, aiakis, aipips, aidxf, acfx_pre
ANSYS CFX-Mesh	cad2mesh
ANSYS CFX FLOTRAN Upgrade	acfx_flotran_upgrade
ANSYS CFX-Flo	acfx_anymodule
ANSYS CFX PrepPost	acfx_anymodule
ANSYS CFX Basic Capability Solver	acfx_pre, acfx_solver, acfx_nolimit, acfx_parallel
ANSYS CFX Multiple Frames of Reference	acfx_mfr
ANSYS CFX Multi-Phase Flows	acfx_multiphase
ANSYS CFX Reacting and Combusting Species	acfx_combustion
ANSYS CFX Radiation Models	acfx_radiation
ANSYS CFX Advanced Turbulence Models	acfx_advanced_turbulence, acfx_turbulence_transition
ANSYS CFX Parallel Computing	acfx_par_proc
ANSYS CFX Full Capability Solver	acfx_pre, acfx_solver, acfx_nolimit, acfx_mfr, acfx_multiphase, acfx_combustion, acfx_radiation, acfx_parallel, acfx_advanced_turbulence, acfx_turbulence_transition
ANSYS CFD-Post	acfx_post
ANSYS BladeModeler	acfx_bldmdlr
ANSYS TurboGrid	acfx_turbogrid
ANSYS CFD	acfd
ANSYS CFX	acfd_cfx
ANSYS CFX Solver	acfd_cfx_solver
ANSYS FLUENT	acfd_fluent
ANSYS FLUENT Solver	acfd_fluent_solver
ANSYS FLUENT V2F Module	acfd_v2f
ANSYS FLUENT PEM Fuel Cell Module	acfd_fcell
ANSYS FLUENT SOFC Fuel Cell Module	acfd_fcell
ANSYS FLUENT Immersed Boundary Module	acfd_ib
ANSYS POLYFLOW	acfd_polyflow
ANSYS POLYFLOW Extrusion	acfd_polyflow_extrusion
ANSYS POLYFLOW BlowMolding	acfd_polyflow_blowmolding
ANSYS POLYFLOW Solver	acfd_polyflow_solver
ANSYS CFX-RIF Flamelet Library Generator	acfd_rif

Product	Feature Names
ANSYS Vista TF	acfd_vista_tf
ANSYS CFD Solver	acfd_solver
ANSYS CFD PrepPost	acfd_preppost
ANSYS CFD-Flo	acfd_flo
ANSYS CFD HPC	acfd_par_proc
ANSYS CFD MHD	acfd_mhd
ANSYS HPC	anshpc
ANSYS HPC Pack	anshpc_pack
ANSYS ICEM CFD	aienv, aienvsub, aiprism, aiiges, aiacis, aiedgeom, aioutcfd, aidxf
ANSYS ICEM CFD AddOn	aiaddon
ANSYS ICEM CFD Hexa Add-on	aihexa
ANSYS ICEM CFD Basic with Advanced Geometry Support	aioutput, aiiges, aiacis, aiedgeom, aioutcfd, aidxf
ANSYS ICEM CFD Basic with Geometry Interface for NX	aioutput, aiiges, aiacis, aipi-ug, aioutcfd, aidxf
ANSYS ICEM CFD Basic with Geometry Interface for CATIA V4	aioutput, aiiges, aiacis, aipicat, aioutcfd, aidxf
ANSYS ICEM CFD Basic with Geometry Interface for Pro/ENGINEER	aioutput, aiiges, aiacis, aipipro, aioutcfd, aidxf
ANSYS ICEM CFD Basic with Geometry Interface for SDRC-IDEAS	aioutput, aiiges, aiacis, aipiidea, aioutcfd, aidxf
ANSYS ICEM CFD Basic with Geometry Interface for SolidWorks	aioutput, aiiges, aiacis, aip-isw, aioutcfd, aidxf
ANSYS ICEM CFD Basic with Geometry Interface for SolidEdge	aioutput, aiiges, aiacis, aip-ise, aioutcfd, aidxf
ANSYS ICEM CFD Hexa	aimed, aihexa, aimshcrt, aibfcart, aiiges, aiacis, aiout-put, aioutcfd, aidxf, aiquad
ANSYS ICEM CFD Tetra/Prism	aimed, aitetra, aiprism, aiiges, aiacis, aioutput, aioutcfd, aidxf, aiquad
ANSYS ICEM CFD Quad	aimed, aiquad
ANSYS ICEM CFD Global	aimed, aiglobal
ANSYS ICEM CFD Autohexa	aimed, aiautmdl, aiautmsh
ANSYS ICEM CFD Hexa CAA V5 Based (for CATIA V5)	aihext5, aihext5e
ANSYS ICEM CFD Mesh Prototyper	aiiges, aiacis, aimed, aitetra, aimshprt
ANSYS ICEM CFD Environment for CART3D	aivis3, aiiges, aiacis

Product	Feature Names
ANSYS ICEM CFD Cart3D Product	aimed, aimshcrt, aiflowcrt, aiflowcrtp
ANSYS ICEM CFD BF-Cart	aibfcart, aimed
ANSYS ICEM CFD Cabin Modeler Stand-alone	aimed, aitetra, aiprism, aicabin
ANSYS ICEM CFD IC3M Stand-alone	aimed, aitetra, aihexa, aiic3m, aiic3msb
ANSYS ICEM CFD Comak	aimed, aicomak
ANSYS ICEM CFD Visual3 / PV3	aivis3
ANSYS ICEM CFD OptiMesh	aioptmsh
ANSYS AUTODYN-2D	acdi_ad2dfull
ANSYS AUTODYN-3D	acdi_ad3dfull
ANSYS AUTODYN	acdi_ad3dfull, acdi_adhpc, acdi_adprepost
ANSYS AUTODYN HPC	acdi_adhpc
ANSYS AUTODYN PrepPost	acdi_adprepost
ANSYS Explicit STR	acdi_explprof
ANSYS AI*Environment	aienv, aienvsub
Direct CAD Interface - CATIA V4	aipicat
Direct CAD Interface - SDRC-IDEAS	aipiidea
ANSYS ICEM CFD Parasolid Reader	aipips
Direct CAD Interface - Pro/ENGINEER	aipipro
Direct CAD Interface -Solid Edge	aipise
Direct CAD Interface -SolidWorks	aipisw
Direct CAD Interface - NX	aipiug
Batch Meshing Module Add-on to AI*Environment	batmesh, bmeshsub
CFD Utilities Add-on for AI*Environment	aiprism, aioutput, aioutcfid
ANSYS Connection for CATIA V4	concatia
ANSYS Connection for CATIA V5	concatv5
ANSYS Connection for Parasolid	conpara
ANSYS Connection for Pro/ENGINEER	conproe
ANSYS Connection for SAT	consat
ANSYS Connection for NX	conug
Geometry Interface for Inventor MDT	piautoin, pimedesk
Geometry Interface for CATIA V4	picatia
Geometry Interface for CATIA V5	picatv5
Geometry Interface for Parasolid	rdpara
Geometry Interface for Pro/ENGINEER	piproe
Geometry Interface for SAT	rdacis

Product	Feature Names
Geometry Interface for SolidEdge	pisoledg
Geometry Interface for SolidWorks	pisolwor
Geometry Interface for NX	piug
Geometry Interface for CoCreate Modeling	1spacdes
CADNexus/CAPRI CAE Gateway for CATIA V5	capricatv5
ANSYS SpaceClaim Direct Modeler	a_spaceclaim_dirmod
ANSYS SpaceClaim CATIA V5 Interface	a_spaceclaim_catv5
ANSYS JT Reader	a_jt_reader
ANSYS Pro/E Reader	a_proe_reader
ANSYS Composite PrepPost	acpreppost
TAS	ahti_tas
TASTEC	ahti_tastec
TAS Pultrusion	ahti_tas_pultrusion
TAS Sinda/3D	ahti_tas_sinda3d
TASStress	ahti_tas_stress
FET Module for TAS	ahti_tas_fet
ICEBoard	anti_taspcb
ICEChip	ahti_ptd
ANSYS DesignModeler	agppi
ANSYS DesignXplorer	dsdxm
ANSYS CAE Templates	caetmpl
ANSYS Fatigue Module	dfatigue
Pressure Equipment Module	preqmo
ANSYS Meshing	amesh
ANSYS Extended Meshing	amesh_extended
ANSYS TGrid	amesh_tgrid
ANSYS LS-DYNA Parallel	dysmp
LS-DYNA MAT_161	lsmat161
LS-DYNA MAT_162	lsmat162
ANSYS Mechanical HPC	mechhpc
ANSYS Interface for Team Center Engineering	pdmiman
ANSYS MeshMorpher	paramesh
ANSYS MeshMorpher Target Geometry Module	pmeshgeo
ANSYS Academic Associate	aa_a*, aa_mcad
ANSYS Academic Associate CFD	aa_a_cfd, aa_mcad
ANSYS Academic Associate HPC	aa_a_hpc
ANSYS Academic Research	aa_r*, aa_mcad
ANSYS Academic Research CFD	aa_r_cfd, aa_mcad

Product	Feature Names
ANSYS Academic Research LS-DYNA	aa_r_dy*, aa_mcad
ANSYS Academic Research Offshore/Marine	aa_r_om, aa_mcad, aa_r_aql
ANSYS Academic Research AUTODYN	aa_r_ad, aa_mcad
ANSYS Academic Research Mechanical	aa_r_me, aa_mcad
ANSYS Academic Research HPC	aa_r_hpc
ANSYS Academic Research LS-DYNA HPC	aa_dy_p
ANSYS Academic Research POLYFLOW	aa_r_pf, aa_mcad
ANSYS Academic Research Electronics Thermal	aa_r_et, aa_mcad
ANSYS Academic Teaching Introductory	aa_t_i*, aa_mcad, aa_ds
ANSYS Academic Teaching Advanced	aa_t_a*, aa_mcad, aa_ds
ANSYS Academic Teaching Mechanical	aa_t_me*, aa_mcad, aa_ds
ANSYS Academic Teaching CFD	aa_t_cfd, aa_mcad
ANSYS Academic Meshing Tools	aa_mesh, aa_mcad
ANSYS Academic CFD Turbo Tools	aa_turbo
ANSYS Academic Fuel Cell Tools	aa_fcell
ANSYS Academic Mechanical HPC	aa_p_me
ANSYS Academic AUTODYN HPC	aa_p_ad
ANSYS Academic CFD HPC	aa_p_c
ANSYS Rigid Dynamics	kinemat, dynamics
EKM Workgroup 1	ekm_wg_server, ekm_named_user, ekm_user, ekm_test_server, ekm_test_named_user, ekm_test_user
EKM Workgroup 2	ekm_wg_server, ekm_named_user, ekm_user, ekm_test_server, ekm_test_named_user, ekm_test_user
EKM Enterprise Server	ekm_server, ekm_test_serv- er, ekm_test_named_user, ekm_test_user
EKM Desktop	ekm_desktop
EKM Datalink	ekm_datalink_server, ekm_datalink_user, ekm_datalink_named_user
ANSYS ASAS	acdi_asas, acdi_asaslink, acdi_asasnl, acdi_asas-vis, acdi_beamst, acdi_comped, acdi_loco, acdi_maxmin, acdi_post, acdi_postnl, acdi_prebeam, acdi_prenl,

Product	Feature Names
	acdi_response, acdi_xtract, preppost
ANSYS BEAMCHECK	acdi_asas-vis, acdi_beamst
ANSYS ASAS-OFFSHORE	acdi_asas, acdi_asaslink, acdi_asasnl, acdi_asas-vis, acdi_beamst, acdi_comped, acdi_fatjack, acdi_loco, acdi_mass, acdi_maxmin, acdi_post, acdi_postnl, acdi_prebeam, acdi_prenl, acdi_response, acdi_splinter, acdi_wave, acdi_windspec, acdi_xtract, preppost
ANSYS ASAS CONCRETE	acdi_concrete
ANSYS ASAS PANEL	acdi_panel
ANSYS ASAS FEMGV	acdi_femgv, acdi_fgv_asas
ANSYS AQWA DIFFRACTION	acdi_aqwags, acdi_aqwaline, acdi_aqwawave, acdi_hydrodiff
ANSYS AQWA SUITE	acdi_aqwadrft, acdi_aqwafer, acdi_aqwags, acdi_aqwalbrm, acdi_aqwaline, acdi_aqwanaut, acdi_aqwawave, acdi_hydrodiff
ANSYS AQWA FREQUENCY	acdi_aqwaline, acdi_aqwags, acdi_aqwawave, acdi_aqwafer, acdi_aqwalbrm, acdi_hydrodiff
ANSYS AQWA SUITE with Coupled Cable Dynamics	acdi_aqwaline, acdi_aqwawave, acdi_cdags, acdi_cd-drift, acdi_cdfer, acdi_cd-libr, acdi_cdnaut, acdi_hydrodiff
ANSYS FATJACK	acdi_fatjack
ANSYS Icepak	aice_pak
ANSYS Icepak Mesher	aice_mesher
ANSYS Icepak Solver	aice_solv
ANSYS Iceopt	aice_opt

An \* after a feature name indicates that an item is a product variable and can be used on a stand-alone basis to start a product run in the Mechanical APDL (ANSYS) application.

Products listed here use the ANSYS license manager. ANSYS, Inc. does supply licenses for ICEM CFD, CFX, Fluent, Century Dynamics, Inc., Harvard Thermal, and Ansoft products not listed above; however, those licenses will use their respective license manager. To determine the license manager to which the license applies, see [Recognizing an ANSYS, Inc. License File \(p. 6\)](#).



# Glossary

ANSLIC_ADMIN	<p>A utility that centralizes the various ANSYS product licensing administrative functions. Tasks include, but are not limited to:</p> <ul style="list-style-type: none"><li>• starting and stopping the license manager</li><li>• installing licenses</li><li>• displaying license status</li></ul>
ANSWAIT	<p>An environment variable that will allow you to queue your ANSYS job in the event that all ANSYS licenses are in use. Once a license becomes available and you are next on the queue, your ANSYS job will automatically start.</p>
ANSYS, Inc. License Manager	<p>The ANSYS, Inc. License Manager consists of three components. The first two are FLEXlm components; the last one is an ANSYS component. See the definitions of each for more information.</p> <ul style="list-style-type: none"><li>• <code>lmgrd</code></li><li>• <code>ansyslmd</code></li><li>• <code>ansysli</code></li></ul>
ansysli	<p>see Licensing Interconnect</p>
ansysli_client	<p>A component of the Licensing Interconnect that is run by ANSYS applications on client systems. No user configuration or administration is required or associated with this component.</p>
ansysli_monitor	<p>The <code>ansysli_monitor</code> runs on the same machine where the <code>ansysli_server</code> is running and ensures that the license server is functioning correctly. If the license server is not running, <code>ansysli_monitor</code> can restart it. If the server is running but is not responsive, <code>ansysli_monitor</code> can kill and restart it.</p>
ansysli_server	<p>A component of the Licensing Interconnect that is run on the license server systems. Use <b>ANSLIC_ADMIN</b> to manage (start, stop, etc.).</p>
ansysli_server.log	<p>The Licensing Interconnect log file, it provides a chronicle of Licensing Interconnect licensing activity, including problems. The Licensing Interconnect log file is located in the licensing directory by default.</p>
ansysli port number	<p>Communication channel by which the ANSYS, Inc. applications communicate with the Licensing Interconnect. The default <code>ansysli</code> port number is 2325.</p>
ANSYSLMD_LICENSE_FILE	<p>An environment variable that may be used to specify the license server machine from which you want to check out a license.</p>
ansyslmd	<p>This is one of the FLEXlm components of the ANSYS, Inc. license manager used to process ANSYS product licensing requests, including issuing and returning licenses. <code>ansyslmd</code>, often referred to as the vendor daemon, is started by <code>lmgrd</code> and must be running to perform the aforementioned tasks.</p>

ansyslmd.ini	File that resides in the licensing directory. It is created by <b>ANSLIC_ADMIN</b> utility options <b>Specify the license server machine</b> and <b>Modify license manager startup options</b> .
ASD	ANSYS Support Distributor.
backup server	In a three-server (redundant triad) network, the two servers not chosen to be the master are the backup servers. If the master server goes down, the backup server listed next in the license file automatically assumes the role of master. (A backup server is also known as a shadow server.)
borrowable license	A borrowable license is a license that you can use temporarily outside of the company facility (such as at home on a laptop). A special license key is required before any license can be borrowed.
build date	The build date is the year, month, and date the ANSYS application was built. The version field in the license file specifies the latest build date that can be run using that license. It may also appear as 9999.9999 if a maintenance agreement is not applicable.
Capability	ANSYS, Inc. has assigned identifiers to each of the specific areas of functionality in the software. We refer to these identifiers as capabilities, which you may see in ANSYS, Inc. licensing displays and logs. Each capability can be satisfied by at least one license feature; often, multiple license features can satisfy a particular capability.
client	A client is a machine that requests licenses but is not a license server machine (i.e., does not have licenses installed on it).
feature	The word feature, when used in the descriptions of the licensing utilities, refers to the ANSYS product. See <a href="#">Product Variable Table (p. 77)</a> for the list of ANSYS products and their corresponding license features.
FLEXlm	A component of the ANSYS, Inc. license manager used for all ANSYS products. Also called FLEXnet License Manager. The FLEXlm component authenticates and processes all license requests.
FLEXlm license log file	Typically referred to as the FLEXlm Debug Log File, it contains information relevant to licensing activity. This file provides a way of tracking licensing problems that may occur. The licensing log file, <code>license.log</code> , is located in the licensing directory by default.
FLEXlm port number	Communication channel by which the license manager communicates with the client (Licensing Interconnect). The default FLEXlm port number is 1055.
floating license	Anyone on the network can run a licensed ANSYS product, up to the limit specified in the license file. Floating licenses require the license manager daemon ( <code>lmgrd</code> ) and the vendor daemon ( <code>ansyslmd</code> ) to be running to count the concurrent usage of the licenses.
getcn	Getcn is a utility that will display your customer number, which is contained in your license. Getcn may be run from the <b>ANSLIC_ADMIN</b> utility by choosing <b>Display customer number</b> .
IA-32	IA-32 refers to the Intel architecture, 32-bit processor.

IA-64	IA-64 refers to the Intel architecture, 64-bit processor.
license	See license task.
license borrowing	License borrowing allows a user to take a license for use outside of the company facility, such as for an engineer to take a license home on his laptop.
license file	A license file grants access to run specified products. Each licensed product will have an entry in the license file, <code>license.dat</code> , which will reside in the licensing directory. Proper installation of a license file grants access to ANSYS products. Install the license file using the <b>Run License Wizard</b> or the <b>Install the License File</b> options of the <b>ANSLIC_ADMIN</b> utility or during the installation process.
license key	The actual full license for the product.
license.log	Typically referred to as the FLEXlm Debug Log File, it contains information relevant to licensing activity. This file provides a way of tracking licensing problems that may occur. The licensing log file is located in the licensing directory by default.
license manager	Software used for licensing ANSYS, Inc. products. The ANSYS, Inc. License Manager is comprised of two components: the FLEXlm software and the Licensing Interconnect.
license options file	<p>A FLEXlm file containing license manager-related resource information. This file is created on license server machines by the <b>ANSLIC_ADMIN</b> utility. By default, the file is named <code>ansyslmd.opt</code> and resides in the licensing directory. You may specify the following information in the options file:</p> <ul style="list-style-type: none"><li>• Licenses that are reserved for individuals, groups, or machines</li><li>• Those individuals, groups, or machines that are denied access to licenses</li></ul>
license server machine	A license server machine is a computer that you have designated to be the administrator of ANSYS product licenses; the licenses are installed on the server machine(s). One or three systems can be used to administer any particular set of ANSYS product licenses.
license task	Each concurrent use of an ANSYS product is a license task. Each use of an ANSYS product will take a number of license tasks from the total number available.
licensing directory	The default location for the licensing files. On UNIX/Linux systems, the licensing directory is <code>/ansys_inc/shared_files/licensing</code> . On Windows systems, the licensing directory is <code>\Program Files\Ansys Inc\Shared Files\Licensing</code> , located in the same drive as the operating system. The licensing directory cannot be changed from the default directory on Windows systems.
Licensing Interconnect	Communications between the ANSYS applications, <code>lmgrd</code> , and <code>ansyslmd</code> are handled by an intermediary process called the ANSYS Licensing Interconnect. The ANSYS Licensing Interconnect communicates with the FLEXlm license manager to authenticate and process all license requests.

Licensing Interconnect Log File	Provides a chronicle of Licensing Interconnect licensing activity, including problems. The Licensing Interconnect log file, <code>ansysli_server.log</code> , is located in the licensing directory by default.
ladmin	A group of users that you designate to have the ability to perform license administrative tasks that are considered disruptive.
lmgrd	One of the FLEXlm components of the ANSYS, Inc. License Manager, used to process ANSYS product licensing requests, including issuing and returning licenses.
LMTOOLS	LMTOOLS is a Windows-only utility that will perform various license administrator functions, such as listing the users of licensed products. LMTOOLS is no longer sufficient to manage the ANSYS, Inc. License Manager and the Licensing Interconnect. You should use the <b>ANSLIC_ADMIN</b> utility to start or stop the license manager, check the license status, etc. However, if you have well-established processes to manage FLEXlm, you can continue to use LMTOOLS (see <a href="#">Advanced Licensing Configuration Options</a> ).
lmutil	<code>lmutil</code> is a FLEXlm-supplied utility that will perform various license administrator functions including: rereading the license file, shutting down the license manager, starting the license manager, and listing the users of licensed products. <code>lmutil</code> is the underlying process in several of the <b>ANSLIC_ADMIN</b> options.
master server	In a three-server (redundant triad) network, one of the servers must be the master server. The server listed first in the license file automatically assumes the role of master. The other two license server machines are shadow servers (or backup servers). In a one-server network, that server is automatically the master.
options file	See license options file.
path	When used in the context of a license file path, the list of places that are searched in order to locate a valid license file. The path is built from values in the <b>ANSYSLMD_LICENSE_FILE</b> environment variable, settings in the <code>ansyslmd.ini</code> file, or an actual license file.
product variable	Each ANSYS product has been assigned a specific product variable value. Use this product variable value when starting a job to specify which ANSYS product is to be run. For Mechanical APDL, the product variable may be set by using the <b>-p</b> command line option, the <b>ANSYS130_PRODUCT</b> environment variable, or the Mechanical APDL launcher. See <a href="#">Product Variable Table (p. 77)</a> for a list of ANSYS products and their associated product variables/feature names.
quorum	You may designate either one or three machines to be license servers. In a three-server (redundant triad) network, license manager daemons must be running on the majority (2) of the server machines before requests for licenses will be processed.
redundant (triad) servers	Having multiple machines designated as license servers for an ANSYS product. Redundancy can be achieved either by having three license server machines working together in tandem, where two of the three must be running at all times but serving from a single set of licenses, or by having any number of

	license server machines running independently, each serving from different license files.
server	See license server.
session log	In the <b>ANSLIC_ADMIN</b> utility, all licensing activity is written to an area on the right-hand side of the utility, called the session log area. The session log maintains a running tabulation of all activity (starting or stopping the license manager, rereading the license file, errors, etc.). The session log can also be written to a file that can be used for troubleshooting.
shadow server	See backup server.
task	See license task.
timezones	The timezones keyword as part of the INCREMENT line's VENDOR_STRING specifies which time zone the client needs to be running in. The absence of the timezones keyword indicates that a client can run in any time zone. The format is <b>timezones:X[ ,Y,Z, . . . ]</b> , where <i>X</i> is a numeric value from 0-23, where 0 represents GMT, and 19 represents EST. Multiple time zones are separated by commas (,): for example, <b>timezones:17,18,19</b> . This keyword cannot be modified by the user.
vendor daemon	FLEXlm terminology used to refer to the vendor-specific component of the FLEXlm license manager. The ANSYS vendor daemon is <code>ansyslmd</code> .



# Index

## Symbols

### A

Academic licenses, 38, 52  
Agreement number, 49  
All licenses reserved, 68  
ANS\_FLEXLM\_DISABLE\_DEFLICPATH, 51  
ANSLIC\_ADMIN utility, 29  
    definition, 85  
    display customer number, 49  
    display license server hostid, 31  
    display queued licenses, 48  
    display the license status, 47  
    editing the FLEXlm license options file, 25  
    editing the license options file, 41  
    gather diagnostic information, 49  
    install license, 32  
    modify startup options, 45  
    register license server information, 31  
    remove a client license, 37  
    reread the license settings, 34  
    run the ANSYS borrow utility, 39  
    run the license wizard, 31  
    set license preferences, 38  
    set site preferences, 41  
    shutdown the license manager, 34  
    specify product order, 45  
    specifying the license server, 35  
    start the license manager, 33  
    uninstall the license manager, 50  
    update license, 32  
    using its options (UNIX/Linux), 29  
    view FLEXlm debug log file, 49  
    View status/diagnostic options, 47  
    view the ansysli debug log file, 49  
    view the FLEXlm license file, 49  
ANSWAIT environment variable, 56  
    definition, 85  
ANSYS  
    licenses, 7  
ansys\_pid utility, 60  
ansysli, 2–3  
    definition, 85  
    port number, 13  
ansysli port number  
    definition, 85  
ansysli\_client  
    definition, 85  
ansysli\_monitor

    definition, 85  
ansysli\_server  
    definition, 85  
ansysli\_server log file  
    definition, 85  
ANSYSLI\_SERVERS, 51  
ANSYSLI\_TIMEOUT\_CONNECT, 51  
ANSYSLI\_TIMEOUT\_TCP, 51  
ansyslmd, 2, 4–5  
    definition, 85  
ansyslmd.ini  
    definition, 86  
ansyslmd.ini file, 23, 35, 51  
ansyslmd.opt file, 25, 37, 41  
ANSYSLMD\_LICENSE\_FILE, 3, 23, 51  
ANSYSLMD\_LICENSE\_FILE environment variable  
    definition, 85  
ASD  
    definition, 86

### B

Backup server  
    definition, 86  
boot\_ansflex file, 21  
Borrowable license  
    definition, 86  
Borrowing  
    licenses, 39, 57  
Build date  
    definition, 86

### C

Caching FLEXlm licenses, 17, 45  
Caching license servers, 46  
Capability  
    definition, 86  
Client configuration, 51  
Clients  
    definition, 86  
Commands (UNIX/Linux)  
    PS, 21  
Commercial licenses, 38, 52  
Communications requirements, 12  
Complete unfinished licensing installation configuration, 29  
Configuring redundant (triad) servers, 26  
Configuring TCP/IP, 12  
Configuring the license server machine, 23  
.cshrc file, 35  
customer number, 49

**D**

- daemon
  - vendor, 4–5
- Daemons
  - starting at system boot time, 21
  - starting manually, 33
  - vendor (ansyslmd), 2
- Debugging, 59
- Diagnostic information
  - gathering, 49
- Display customer number, 49
- Display license server hostid, 31
- Display license status, 47
- Display queued licenses, 48

**E**

- Environment variables
  - ANS\_FLEXLM\_DISABLE\_DEFLICPATH, 51
  - ANSWAIT, 56
  - ANSYSLI\_SERVERS, 51
  - ANSYSLI\_TIMEOUT\_CONNECT, 51
  - ANSYSLI\_TIMEOUT\_TCP, 51
  - ANSYSLMD\_LICENSE\_FILE, 3, 23, 51
- Error messages, 59
  - launcher , 67
  - license borrowing, 73
  - license log file, 74
  - licensing, 21, 68
  - licensing installation, 67
- Establishing user licensing preferences, 52
- Expired license, 69

**F**

- Feature
  - definition, 86
- Feature names, 77
- Files
  - ansyslmd.ini, 23, 35, 51
  - ansyslmd.opt, 25, 37, 41
  - boot\_ansflex
    - init\_ansysli, 21
  - .cshrc, 35
  - init\_ansysli, 21
  - license, 2–3
    - installing, 23
  - license (license.dat), 23
  - license files format, 3
  - license.dat, 32, 51
  - license.log, 21, 33
    - viewing, 49
  - licserver.info, 23, 31

- sample license files, 6
- Firewall settings, 26
- FLEXlm
  - compatibility with other software, 1
  - definition, 86
  - port number, 13
- FLEXlm debug log file (license.log)
  - viewing, 49
- FLEXlm license options file (ansyslmd.opt)
  - editing, 25
- FLEXlm port number
  - definition, 86
- FLEXNet Manager, 17, 29
- floating license
  - definition, 86
- Floating licenses, 1, 7
- Format of license files, 3
- Freeing a license, 29, 37
- Functions requiring lmadm group privileges, 24

**G**

- Gather diagnostic information, 49
- getcn utility
  - definition, 86
- Group
  - creating, 24
- Group restrictions
  - defining, 24

**H**

- hostid, 4, 7, 31
- hostname, 4, 7, 31

**I**

- IA-32, 86
- IA-64, 87
- INCREMENT lines, 5, 77
- init\_ansysli, 21
- Installation
  - license, 32
  - uninstalling the license manager, 50
- Installing license files, 23
- Installing the license manager, 11
  - post-installation instructions, 21
  - UNIX/Linux, 15
  - Windows, 14
- installing the license manager, 13
  - Complete unfinished licensing installation configuration, 29



**L**

## Launcher

- error messages, 67

## License

- error message: has expired, 69
- installing, 32
- updating, 32

## License administration, 29

- creating ladmin group, 29

## License borrowing, 39, 57

- definition, 87
- running, 57
- running the borrow utility, 40
- setup, 39

## License borrowing error messages, 73

## License file installation, 23

## License files, 2–3

- definition, 87
- example, 6
- format, 3
- INCREMENT lines, 5
- recognizing, 6
- reread, 34
- SERVER lines, 4
- settings precedence, 51
- specifying the location, 23
- VENDOR lines, 4

## License key

- definition, 87

## License log file (license.log)

- definition, 86–87
- error messages, 74

## License log file error messages, 74

## License manager

- advanced configuration, 17
- compatibility with other software, 1
- Complete unfinished licensing installation configuration, 29
- components, 2
- daemon, 88
- defined, 1
- definition, 87
- installing, 11, 13
- installing on UNIX/Linux, 15
- installing on Windows, 14
- modifying startup command options, 24
- modifying startup options, 45
- post-installation instructions, 21
- shutting down, 34
- shutting down with ANSLIC\_ADMIN, 34
- starting at system boot time, 21
- starting manually, 33

- starting with ANSLIC\_ADMIN, 33

- uninstalling, 50

## License methods

- sharing vs. separate, 38

## License options file (ansyslmd.opt)

- editing, 41
- sample, 44

## License options file (ansyslmd.opt),

- definition, 87

## License preferences

- academic vs. commercial, 38, 52

## License queuing, 29, 56

## License server

- error message: is down, 68
- specifying, 25

## license server machines, 2

## License server machines

- definition, 7, 87
- display hostid, 31
- master, 7
- selecting, 7
- three-server network, 7

## License server manager, 2

## License servers

- caching, 46
- configuring, 23
- designating for license checkout, 29, 35
- error messages, 21
- network licensing, 2
- redundancy, 8
- registering, 23, 31
- shutting down the license manager daemon, 34
- specifying, 35

## License status

- display, 47–48

## License task

- definition, 87

## License wizard, 31

## license.dat file, 2–3, 23, 32, 51

## license.log file, 21, 33

- viewing, 49

## Licenses

- customer number, 49
- floating, 7
- freeing, 29, 37
- installing/updating, 32
- waiting for available, 56

## Licensing

- debugging, 59
- error messages, 59
- problem situations, 60
- process, 1

- sample scenario, 37
- selecting server machines, 7
- separate method, 38
- sharing method, 38
- tasks, 7
- three-server license network, 7
- Licensing directory, 1
- Licensing environment
  - setting up, 29
- Licensing error messages, 68
- Licensing installation
  - error messages, 67
- Licensing Interconnect, 2
  - defining group restrictions, 24
  - port number, 13
  - running independently of FLEXlm, 17
  - running without FLEXlm, 17
- Licensing Interconnect (ansysli), 3
- Licensing preferences
  - establishing for users, 52
- licserver.info file, 23, 31
- ladmin group, 24, 29
- lmdown, 24
- lmgd, 2, 33
  - utility definition, 88
- lmhostid, 31
- lmremove, 24
- lmreread, 24
- LMTOOLS, 17, 29
- lmutil
  - utility definition, 88
- Log file, 88
  - (see also license log file)

## M

- Master server, 7, 21
  - definition, 88

## N

- Network File System
  - licensing failures on, 24
- Network licensing
  - network licensing considerations, 7
  - queuing facility, 56
  - shutting down the license manager, 34
  - starting license manager at system boot time, 21
  - starting license manager manually, 33
- No license file found for feature error message, 70
- No licenses are currently available error message, 68

## P

- Path
  - definition, 88
- Pool of available tasks, 7
- Port number
  - FLEXlm, 4
- Port numbers
  - changing, 13
  - FLEXlm, 13
  - Licensing Interconnect, 13
- Preferences
  - setting, 38, 52
- Product list, 77
- Product order
  - specifying, 45
- Product variable
  - definition, 88
- PS command, 21

## Q

- Queued licenses
  - display, 48
- Queuing, 29, 56
- Quorum
  - definition, 88

## R

- Redundant (triad) servers
  - setting up, 26
- Redundant server options, 8
- Redundant servers
  - definition, 88
- Registering license server information, 23, 31
- Reread the license settings, 34
- root login, 29, 50
- Running the Licensing Interconnect and FLEXlm independently, 17, 19
- Running the Licensing Interconnect without FLEXlm, 17–18

## S

- Sample license files, 6
- Sample licensing scenario, 37
- SERVER lines, 4
- Session log
  - definition, 89
- Set site preferences, 41
- Shadow server
  - definition, 89
- Shutting down the license manager, 34
- Site preferences

- setting, 41
- Specify product order, 45
- Specifying firewall settings, 26
- Specifying the license server, 25, 35
- Starting the license manager, 33
- Starting the license manager at system boot time, 21
- Startup options for license manager
  - modifying, 45
- Status
  - viewing, 47
- superuser login, 29, 50
- Symbolic links, 1

## T

- Tasks, 7
- TCP/IP
  - dial-up connection, 12
  - testing whether TCP/IP is installed, 12
- TCP/IP requirements, 12
- Three-server network, 7
- timezones, 89
- Troubleshooting
  - ansys\_pid utility, 60
  - getting additional debugging information, 59
  - license borrowing errors, 73
  - license log file errors, 74
  - licensing error messages, 68
  - problem situations, 60
- Troubleshooting ANSYS error messages, 59

## U

- Unavailable licenses, 68
- Uninstalling the license manager, 50
- Updating license, 32
- User privileges, 29
  - Using the FLEXlm options file, 25
  - Using the options file, 41

## V

- Vendor daemon, 2, 5
- vendor daemon
  - definition, 89
- Vendor daemon (ansyslmd), 4
- VENDOR lines, 4
- View status/diagnostic options, 47
- View the ansysli debug log file, 49
- View the FLEXlm license file, 49
- Viewing the FLEXlm debug log file, 49

## W

- WANs, 7

- Wide area networks, 7





# **ANSYS, Inc. Installation Guide for UNIX/Linux**

---



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 13.0  
November 2010  
00XXXX

ANSYS, Inc. is certified to ISO 9001:2008.
--

---

## Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superceded by a newer version of this guide. This guide replaces individual product installation guides from previous releases.

## Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---

# Table of Contents

<b>1. Installation Prerequisites for UNIX/Linux</b>	1
1.1. System Prerequisites	2
1.1.1. CAD Support	4
1.2. Disk Space and Memory Requirements	5
1.3. GPU Requirements	6
1.4. Additional Hardware and Software Requirements	6
<b>2. Platform Details</b>	7
<b>3. Installing the Software</b>	13
3.1. Pre-Installation Instructions for Download Installations	13
3.1.1. Downloading the Installation Files	13
3.2. Mounting the DVD Instructions for DVD Installations	13
3.3. Installing ANSYS, Inc. Products	14
3.3.1. Product Installation	15
3.3.2. License Manager Installation	17
3.3.3. Network Installation and Product Configuration	18
3.3.3.1. Export the /ansys_inc Directory	18
3.3.3.2. Run the Product Configuration Utility on All Client Machines	20
3.3.3.3. Configure Licensing for File Server Installations	21
3.3.4. Silent Mode Operations	21
3.3.4.1. Silent Product and License Manager Installation	22
3.3.4.2. Silent Product Configuration/Unconfiguration	24
3.3.4.3. Silent Uninstall	24
3.3.5. Registering the License Server	25
3.4. Post-Installation Procedures	25
3.4.1. Post-Installation Procedures for Mechanical APDL (ANSYS) and ANSYS Workbench Products	26
3.4.2. Post-Installation Procedures for ANSYS CFX	27
3.4.2.1. Setting up ANSYS TurboGrid Release 13.0	29
3.4.2.2. Using the ANSYS CFX Launcher to Set Up Users	29
3.4.2.3. Verifying the Installation of ANSYS CFX Products	29
3.4.3. Post-Installation Procedures for ANSYS FLUENT	30
3.4.4. Post-Installation Procedures for ANSYS POLYFLOW	31
3.4.5. Post-Installation Procedures for ANSYS ICEM CFD	31
3.4.6. Post-Installation Procedures for ANSYS AUTODYN	31
3.5. Translated Message File Installation for Mechanical APDL (ANSYS)	31
3.6. Launching ANSYS, Inc. Products	32
3.7. Uninstalling ANSYS, Inc. Products	32
<b>4. Configuring CAD Products</b>	35
4.1. Using the CAD Configuration Manager on Linux	35
4.1.1. Unconfiguring	37
4.1.2. Running the <b>CAD Configuration Manager</b> in Batch Mode	37
4.1.3. Pro/ENGINEER Configuration	38
4.1.4. NX Configuration	38
4.2. Configuring the Connection for Pro/ENGINEER	39
4.2.1. Other Connection for Pro/ENGINEER Configuration Steps	39
4.2.2. The protk.dat File	40
4.2.3. The config.pro File	41
4.2.4. Configuring the config.anscon File for Connection for Pro/ENGINEER	41
4.3. Configuring Pro/ENGINEER Manually	41
4.3.1. Configuring the Geometry Interface for Pro/ENGINEER for ANSYS Workbench Products	42
4.4. Configuring NX	42

4.4.1. Configuring the Connection for NX .....	43
4.4.2. Configuring the Geometry Interface for NX for ANSYS Workbench Products .....	43
<b>5. Troubleshooting .....</b>	<b>45</b>
5.1. Installation Troubleshooting .....	45
5.1.1. Using ANSLIC_ADMIN to Gather Diagnostic Information .....	45
5.1.2. The GUI Installation Process Hangs .....	45
5.1.3. The Target Machine Does Not Have a DVD Drive .....	46
5.1.4. The Online Help System Does Not Run Properly .....	46
5.1.5. CAD Configuration Manager Help Does Not Load .....	47
5.1.6. Cannot Enter Data in Text Fields .....	47
5.1.7. Download and Installation Error Messages .....	47
5.1.8. System-related Error Messages .....	48
5.1.9. High Performance Computing Error Messages .....	48
5.2. Installation Troubleshooting - Mechanical APDL (ANSYS) .....	48
5.2.1. Your batch jobs terminate when you log out of a session .....	48
5.2.2. Japanese/Chinese characters display in status bar windows on Mechanical APDL (ANSYS) on Red Hat AS 4.0 .....	48
5.2.3. Mechanical APDL (ANSYS) Documentation File for User Interface Error Messages .....	49
5.2.4. Launcher Error Messages .....	49
5.2.5. FORTRAN Runtime Error Messages .....	49
5.2.5.1. HP Series Systems .....	49
5.2.5.2. Intel Linux 32 Systems .....	49
5.2.5.3. Intel Linux 64 Systems .....	49
5.2.5.4. Intel EM64T Linux x64 Systems .....	50
5.2.5.5. AMD Opteron Linux x64 Systems .....	50
5.3. Installation Troubleshooting - ANSYS Workbench .....	50
5.3.1. Startup or Graphics Problems .....	50
5.4. Contacting Technical Support .....	50

## List of Tables

1.1. Supported Platforms .....	3
1.2. Supported Platforms by Product .....	4
1.3. CAD Support by Platform .....	4
2.1. Compiler Requirements .....	11
3.1. Locally-Mounted DVD Procedures .....	14
3.2. Remotely-Mounted DVD Procedures .....	14
3.3. Startup Commands .....	32
4.1. Names for ANSYS and Pro/ENGINEER Platform Directories .....	41



---

## Chapter 1: Installation Prerequisites for UNIX/Linux

---

This document describes the steps necessary to correctly install and configure all ANSYS, Inc. products on UNIX and Linux platforms for Release 13.0. These products include:

- ANSYS Structural Mechanics
  - ANSYS Mechanical Products (includes Mechanical APDL and Mechanical, where supported)
  - ANSYS Customization Files
- ANSYS Explicit Dynamics
  - ANSYS AUTODYN
  - ANSYS LS-DYNA
- ANSYS Fluid Dynamics
  - ANSYS CFX (includes ANSYS CFD-Post)
  - ANSYS FLUENT (includes ANSYS CFD-Post)
  - ANSYS TurboGrid
  - ANSYS POLYFLOW (includes ANSYS CFD-Post)
  - ANSYS CFD-Post only
- ANSYS Additional Tools
  - ANSYS ICEM CFD
    - ANSYS ICEM CFD Pro/ENGINEER Interface
    - ANSYS ICEM CFD NX Interface
  - ANSYS Icepak (includes ANSYS CFD-Post)
- ANSYS Geometry Interfaces
  - ACIS
  - CATIA, Version 4
  - CATIA, Version 5
  - NX
  - Parasolid
  - Pro/ENGINEER

**Notes** Not all products may be available on all UNIX/Linux platforms. Please see the remainder of this document for detailed information on which products are available on which platforms.

At Release 13.0, ANSYS BladeGen, BladeEditor, Vista CCD, Vista CPD, Vista RTD, and Vista AFD are not supported on UNIX/Linux platforms.

ANSYS Workbench and ANSYS EKM Desktop are installed as product components to most ANSYS, Inc. product (Linux systems only). ANSYS Workbench is not installed as part of the products under ANSYS Additional Tools, nor with the CFD-Post only option. ANSYS Workbench includes the following applications:

- The Mechanical Application
- DesignModeler
- Design Exploration
- Meshing
- Remote Solve Manager
- TGrid
- FE Modeler
- EKM Desktop

Because some of these applications can be run as standalone applications without ANSYS Workbench, you will see some but not all of these listed separately as options when you uninstall. In such cases, you will need to select those options in order to uninstall those components.

### Important Notice

If you wish to run multiple releases of ANSYS, Inc. software, you **MUST** install them chronologically (i.e., Release 12.1 followed by Release 13.0). If you install an earlier release after installing Release 13.0, you will encounter licensing issues that may prevent you from running any products/releases. If you need to install an earlier release after you have already installed Release 13.0, you **MUST** uninstall Release 13.0, then re-install the releases in order.

### Summary of New and Changed Features

The following features are new or changed at Release 13.0. Please review these items carefully.

- ANSYS, Inc. has discontinued support for the HP-UX 64 PA-RISC and the Sun SPARC 64 platforms for all products. The ANSYS, Inc. License Manager will continue to support Sun SPARC 64.
- The installation and product configuration utilities have been improved.
- Silent mode operations have been extended to include installation, uninstall, product configuration, and product unconfiguration on all platforms.

### 1.1. System Prerequisites

ANSYS, Inc. Release 13.0 products are supported on the UNIX/Linux platforms and operating system levels listed in the following tables. Patches listed may be superseded by newer versions; see your vendor for the most current version. See the individual Platform Certification Specifics in this guide for more information

on specific platform requirements. For up-to-date information on hardware platforms or operating systems that have been certified, go to <http://www.ansys.com/hardware-support>. (This URL is case-sensitive.)

**Table 1.1 Supported Platforms**

Platform	Processor	Operating System	Platform architecture (directory name)	Availability
Linux 32	x86	Red Hat Enterprise Linux 5; SUSE Linux Enterprise 10 [1] and Enterprise 11 SP1	linia32	Download/DVD
Linux Itanium 64	Itanium2	Red Hat Enterprise Linux 5; SUSE Linux Enterprise 10 and Enterprise 11 SP1	linia64	Download only
Linux x64	EM64T/Opteron 64	Red Hat Enterprise Linux 5; SUSE Linux Enterprise 10 [2] and Enterprise 11 SP1	linx64	Download/DVD
HP-UX Itanium 64	Itanium2	HP-UX B.11.23	hpia64	Download only
Sun Solaris x64	x64 (EM64T/Opteron 64)	Solaris 10 [3]	solx64	Download only
IBM AIX 64	Power5 & Power6	AIX 5.3 [4] and 6.0	aix64	Download only

1. On Linux 32-bit systems, ANSYS Workbench is supported only on Red Hat Enterprise Linux 5 or higher. We recommend that you run on Red Hat Enterprise Linux 5.3 or higher. ANSYS Workbench is not supported on any SUSE version on Linux 32 systems.
2. We recommend that you run ANSYS Workbench on Red Hat Enterprise Linux 5.3 or higher or SUSE Linux Enterprise 10.2 or higher.
3. Solaris 10 Update 3 (or greater) is required for Mechanical APDL (ANSYS).
4. AIX 5.3 TL7 Service Pack 5 (or greater) is required for Mechanical APDL (ANSYS).

**Supported Platforms for High Performance Computing** Please see the discussions on [Configuring Distributed ANSYS](#) and [Configuring ANSYS CFX Parallel](#) later in this guide for detailed information on supported platforms for distributed computing.

**Table 1.2 Supported Platforms by Product**

	Linux 32	Linux Itanium 64	Linux x64	HP-UX Itanium 64	IBM AIX 64	Sun Solaris x64
Mechanical APDL	X	X	X	X	X	X
Workbench	X [2]		X [3]			
AUTODYN (Server only)	X	X	X			
LS-DYNA	X	X	X	X	X	
CFX	X	X	X	X [1]	X [1]	
FLUENT	X	X	X	X	X	
TurboGrid	X		X			
ICEM CFD	X	X	X	X	X	
POLYFLOW	X		X			
Icepak	X		X	X		

1. Support for ANSYS CFX-Solver only. No support for ANSYS CFX-Pre, ANSYS CFX-Solver Manager and CFD-Post.
2. We recommend that you run ANSYS Workbench on Red Hat Enterprise Linux 5.3 or higher on Linux 32 systems. ANSYS Workbench is not supported on any SUSE version on Linux 32 systems.
3. We recommend that you run ANSYS Workbench on Red Hat Enterprise Linux 5.3 or higher or SUSE Linux Enterprise 10.2 or higher on Linux x64 systems.

### 1.1.1. CAD Support

The following CAD and auxiliary programs are supported on the indicated products and platforms. Products are:

A = Mechanical APDL  
W = ANSYS Workbench  
I = ANSYS ICEM CFD

**Table 1.3 CAD Support by Platform**

	Linux 32	Linux Itanium 64	Linux x64	HP-UX Itanium 64	IBM AIX 64	Sun Solaris x64
CATIA 4.x	A, I	A, I	A, I	I	A, I	
CATIA V5 R20					A	

	Linux 32	Linux Itanium 64	Linux x64	HP-UX Itanium 64	IBM AIX 64	Sun Sol- aris x64
Parasolid 22.1	A, W, I	A	A, W, I	A, I	A, I	
Pro/ENGINEER Wildfire 4						A
SAT ACIS 20 [1]	A, W, I	A, I	A, W, I	A, I	I	
NX 6.0 [2]			A, W, I			
NX 7.5			A [3], W [3]			
STEP AP203, AP214	W, I	I	W, I	I	I	
IGES 4.0, 5.2, 5.3	W, I	I	W, I	I	I	
DWG	I	I	I	I		
GEMS	I	I	I	I	I	
IDI				I		

1. For ANSYS ICEM CFD standalone, ACIS 18.0.1 is the supported version for all platforms.
2. NX 6.0 has dropped support for all UNIX/Linux platforms except SUSE Linux.
3. NX 7.5 supports RedHat 53 and SUSE Linux Enterprise 10 SP2.

## 1.2. Disk Space and Memory Requirements

You will need the disk space shown here for each product for installation. The numbers listed here are the maximum amount of disk space you will need. Depending on the options selected for each product, you may require less.

Mechanical APDL (ANSYS): 7.4 GB  
 ANSYS AUTODYN: 5.6 GB  
 ANSYS LS-DYNA: 5.7 GB  
 ANSYS CFX: 6.6 GB  
 ANSYS TurboGrid: 5.8 GB  
 ANSYS FLUENT: 6.9 GB  
 POLYFLOW: 7.4 MB  
 ANSYS ICEM CFD: 1.6 GB  
 ANSYS Icepak: 1.8 GB  
 ANSYS TGrid: 5.2 GB  
 CFD Post only: 6.0 GB  
 ANSYS Geometry Interfaces: 100 MB

**Mechanical APDL (ANSYS)** A minimum of 2 GB of real memory is required.

**ANSYS Workbench** A minimum of 2 GB of real memory is required.

**ANSYS CFX** A minimum of 2 GB of real memory per CPU is recommended for 64-bit machines.

**ANSYS FLUENT** A minimum of 2 GB of real memory per CPU is required for 64-bit machines.

**ANSYS POLYFLOW** A minimum 2 GB of real memory recommended for 32-bit and 4 GB for 64-bit. Additional swap space of 2 GB is needed for 32-bit and 4 GB for 64-bit.

## 1.3. GPU Requirements

Your system must meet the following requirements to use the GPU capability in Mechanical APDL:

The machine being used for the simulation must contain at least one nVIDIA Tesla series GPU card (a Tesla 20-series card is recommended for optimal performance).

On Linux, the driver version for the nVIDIA Tesla series GPU card must be 256.35 or newer.

You must be running on a Linux x64 or a Windows 64-bit operating system. Linux Itanium (Linux IA-64) is not supported.

## 1.4. Additional Hardware and Software Requirements

- TCP/IP for the license manager (see the *ANSYS, Inc. Licensing Guide* for more information on TCP/IP)
- If you use a spaceball, ensure that you have the latest version of the drivers installed.
- Approximately twice as much swap space as memory. The amount of memory swap space on the system may limit the size of the model that can be created and/or solved.
- Graphics card compatible with the supported operating systems, capable of supporting 1024x768 High Color (16-bit) and a 17-inch monitor compatible with this type of graphics card; ANSYS CFX products and ANSYS TurboGrid require 24-bit color and that antialiasing on your graphics card be disabled. Refer to your operating system's documentation for specific instructions.
- The appropriate driver and OpenGL packages for the installed graphics card are recommended for Sun Solaris x64.
- X11, OpenGL graphics libraries
- For IBM systems, all OpenGL filesets must be installed.
- Mesa-libGL (OpenGL) is required to run data-integrated ANSYS Workbench applications such as Mechanical.
- If you are using Exceed to run ANSYS Workbench products, Exceed 3D is required. Exceed 3D 2008 Service Pack 9 or higher is recommended.
- The latest available patch bundle is recommended for HP systems.
- For FLUENT, CFX-Pre, and CFX-Post, a three-button mouse is required to access all available functionality.
- Adobe Acrobat Reader is required to read the installation guides and other user documentation.
- ANSYS CFX requires Exceed 2006 or newer to remote display from some platforms.
- FLUENT graphics are not officially supported when running under Exceed.
- For the Red Hat Enterprise Linux on Linux 32 systems, ANSYS Workbench supports only the GNOME desktop environment.

---

## Chapter 2: Platform Details

---

### Hewlett-Packard

**Setting the Maximum Data Segment Size** Any kernel parameters that are specific to the HP 64-bit operating system are appended with the suffix `_64bit` (for example, `maxdsiz_64bit`). If you should run into memory allocation problems, set the suffix `_64bit` parameters as large as the system will allow. The `maxdsiz_64bit` kernel parameter sets the maximum data segment size allowed by the operating system. The maximum allowable value is based on your hardware, and as a result, the operating system will not permit you to set a value higher than it allows. For information on increasing `maxdsiz`, see “Changing Kernel Parameters” in the *HP 9000 Computers System Administration Tasks Manual*.

**Setting the File Buffer Cache** You may want to limit the amount of RAM that the file buffer cache will use during installation. We recommend that you set the following parameters:

```
dbc_max_pct = 10
dbc_min_pct = 10
```

The value indicates the percentage of total RAM.

### HP Graphics Requirements

- When using the 3-D driver (OpenGL), we recommend using graphics boards greater than 8-plane.
- HP Itanium2 systems do not support spaceball devices.
- (Mechanical APDL (ANSYS) program only) When running the Mechanical APDL program using Motif, the following options must be set in the `.Xdefaults` file:

```
Mwm*FocusAutoRaise: False
Mwm*KeyboardFocusPolicy: Pointer
Mwm*KeyBindings: Motif
```

To use the HP OpenGL graphics driver, you must have a graphics device and a machine that are both capable of supporting OpenGL. This list changes often; we recommend that you verify with HP that your hardware meets the requirements. Once these requirements have been established, install the OpenGL software onto the HP system. Typically, OpenGL software is installed into the `/opt/graphics/OpenGL` directory.

### HP-UX Itanium 64

#### Patches

HPUX11i-TCOE, B.11.23.0712, HP-UX Technical Computing OE Component

HPUXBaseAux, B.11.23.0712, HP-UX Base OS Auxiliary

HPUXBaseOS, B.11.23, HP-UX Base OS

BUNDLE11i, B.11.23.0409.3, Required Patch Bundle for HP-UX 11i v2 (B.11.23), Sept. 2004

FEATURE11i, B.11.23.0809.073, Feature Enablement Patches for HP-UX 11i v2, Sept. 2008

QPKAPPS, B.11.23.0712.070a, Applications Quality Pack Bundle for HP-UX 11i v2, Dec. 2007

QPKBASE, B.11.23.0712.070a, Base Quality Pack Bundle for HP-UX 11i v2, Dec. 2007

## IBM

Increase the data segment size to unlimited to allow for the maximum possible memory availability. This can be done via **smit** or **smitty** for each ANSYS user or globally by changing the default in the `/etc/security/limits` file.

OpenGL must be installed on the system.

For best performance using ANSYS, IBM recommends that Simultaneous Multi-Threading be turned off. To disable SMT, run the following command as root:

```
smtctl -m off
```

See the `smtctl` man page for more information.

IBM system requirements: AIX 5.3 Technical Level 7 (5300-07) or higher. POE 4.3.2.6 is required for Distributed ANSYS. POE 4.3.2.2 or higher is required if using FLUENT with Infiniband on AIX.

### Patches

IBM C++ Runtime Environment Components for AIX V7.0 or greater (for CFX products)

## Sun Solaris x64

ANSYS was built and tested on the Solaris 10 i86pc 64-bit Intel and Opteron platforms (x64).

Solaris 10, Update 3 (11/06) or higher, and "Entire" or "Entire+OEM" software group is required.

HPC 7.1 is required for Distributed ANSYS.

Install the appropriate driver for the installed graphics card. OpenGL is required. For best graphics performance, the OpenGL that is associated with the graphics driver is recommended.

### Patches

118677-03	121016-05	125504-02
118855-36	121018-10	125548-02
118919-21	121020-05	125556-02
119255-59	121022-03	126420-01
119964-08	121616-04	126424-03
120012-14	122136-02	127128-11
120273-24	122143-03	127756-01
120754-05	122661-08	137138-09
120759-11	123840-04	138867-02
120762-03	125370-06	138884-01



## Linux

For ALL 32-bit and 64-bit Linux platforms, OpenMotif, OpenMP, and Mesa libraries should be installed. These libraries are typically installed during a normal Linux installation. You will also need the xpdf package to view the online help.

You can find the necessary OpenMotif libraries for your platform at <http://www.motifzone.net>.

SUSE Linux Enterprise 11 requires SP1. After installing the SP1 updates, you must also install OpenMotif and the prerequisites from the SLES11 SDK DVD, as well as the OpenMotif22 packages (openmotif22-libs-2.2.4-138.17 and openmotif22-libs-32bit-2.2.4-138.17) for OpenSUSE Desktop V11 from the OpenSUSE website.

Additional requirements for Intel, AMD Opteron, and EM64T Linux systems are detailed below.

**ANSYS, Inc. License Manager** Linux 32 and Linux x64 systems running the ANSYS, Inc. License Manager require the Linux Standard Base (LSB) 3.0 package.

**ANSYS Workbench** For ANSYS Workbench, install the following gamin .rpm for your platform to prevent ANSYS Workbench from hanging when the Linux file alteration monitor, gam\_server, is using 100% of the CPU.

- gamin-0.1.7-1.4.EL4.i386.rpm
- gamin-0.1.7-1.4.EL4.x86\_64.rpm

If you are running ANSYS Workbench using the KDE desktop environment, set the focus stealing prevention level to "None" to prevent the project save dialog boxes from appearing behind the application window:

1. Use the **kcontrol** command to launch the KDE Control Center.
2. In the Control Center window, select **Desktop> Window Behaviour> Advanced**.
3. Change **Focus Stealing Prevention Level** to None.
4. Click **Apply**.

**Using FLUENT with Infiniband** On some operating systems, the default amount of physical memory that can be pinned/locked by a user application is set to a low value and must be explicitly increased. A value recommended by Intel is 90% of the physical memory. Therefore, for a system with 8GB of memory, the following should be added to the `/etc/security/limits.conf` file:

```
* soft memlock 7500000
* hard memlock 7500000
```

The need for increasing the limits may be indicated by the following error message with HP-MPI:

```
fluent_mpi.6.3.26: Rank 0:0: MPI_Init: ibv_create_qp()
failed fluent_mpi.6.3.26: Rank 0:0:
MPI_Init: Can't initialize RDMA device
```

After setting soft and hard memlock to a proper value you still receive memory errors:

```
libibverbs: Warning: RLIMIT_MEMLOCK is 32768 bytes.
This will severely limit memory registrations
```

**Graphics Libraries** ANSYS Workbench is installed as a 32-bit application on 64-bit Linux platforms. Therefore, you must have the most recent 32-bit graphics libraries installed on your system before installing ANSYS Workbench.

**Semaphore Limit** On some Linux systems, ANSYS Workbench reaches a system limit on the number of semaphores in the Linux configuration. In this case, you will see a message similar to the following:

**sem\_lock->semop->op\_op: Invalid argument**

**sem\_unlock->semctl: Invalid argument**

To increase the number of semaphores, run the following command as owner or root:

```
% echo 256 40000 32 32000 > /proc/sys/kernel/sem
```

This modification takes effect immediately, but is reset at the next reboot. To avoid resetting the limit when rebooting, add the above command to one of your system's startup scripts by copying the command into a file called `mod_sem` and then setting up the following links to execute the file each time you restart your system:

```
cp mod_sem /etc/init.d
ln -s /etc/init.d/mod_sem /etc/rc3.d/S61mod_sem
ln -s /etc/init.d/mod_sem /etc/rc5.d/S61mod_sem
```

**Using the FLUENT Launcher** On Linux systems, you must have the following package (as appropriate for your platform) installed in order to use the FLUENT launcher:

Red Hat 4: compat-libstdc++-33-3.2.3-47.3  
 Red Hat 5: compat-libstdc++-33-3.2.3-61  
 SUSE 10: compat-libstdc++-5.0.7-22.2

**Using ANSCUSTOM** If you use ANSCUSTOM to link your own version of ANSYS Release 13.0 on a SUSE SLES10.x box, you may see two unsatisfied externals that are system, not ANSYS files, due to the linker looking for some system files in a Red Hat directory on a SUSE box.

To work around this problem, run the following as root:

```
mkdir -p /usr/lib/gcc/x86_64-redhat-linux/3.4.6
ln -sf /usr/lib/gcc/i586-suse-linux/2.95.3/crtbegin.o /usr/lib/gcc/x86_64-redhat-linux/3.4.6/crtbegin.o
ln -sf /usr/lib/gcc/i586-suse-linux/2.95.3/crtend.o /usr/lib/gcc/x86_64-redhat-linux/3.4.6/crtend.o
```

The revision numbers shown in the examples (2.95.3 and 3.4.6) may be different on your system. The linker will specify where it is trying to find the `crtbegin.o` and `crtend.o` files; that location will be the last part of the above commands. You can use the Linux **locate** command to find the existing `crtbegin.o` and `crtend.o` files; that location would be the first part of the above commands.

**System Libraries** On 64-bit Linux `linux64` systems (not Itanium systems), the ANSYS Release 13.0 executable is looking for system libraries that do not have revision numbers appended to the end of their file names. On some SUSE systems, the graphics libraries all have revision numbers appended to the end of the library filenames. In these cases, ANSYS quits because the loader cannot find all of the libraries that it is looking for. When running ANSYS Release 13.0, the loader will inform you that it is unable to locate a specific library (for example, `libXm.so`). Using the Linux **locate** command, find the library (`libXm.so` in this example) on your system and add the appropriate symbolic link as seen below.

To overcome this possible problem, run the following as root:

```
ln -sf /usr/lib64/libGLU.so.1.3.060402 /usr/lib64/libGLU.so
ln -sf /usr/X11R6/lib64/libXm.so.3.0.3 /usr/X11R6/lib64/libXm.so
ln -sf /usr/X11R6/lib64/libXp.so.6.2 /usr/X11R6/lib64/libXp.so
ln -sf /usr/X11R6/lib64/libXt.so.6.0 /usr/X11R6/lib64/libXt.so
ln -sf /usr/X11R6/lib64/libXext.so.6.4 /usr/X11R6/lib64/libXext.so
ln -sf /usr/X11R6/lib64/libXi.so.6.0 /usr/X11R6/lib64/libXi.so
```

```
ln -sf /usr/X11R6/lib64/libX11.so.6.2 /usr/X11R6/lib64/libX11.so
ln -sf /usr/X11R6/lib64/libSM.so.6.0 /usr/X11R6/lib64/libSM.so
ln -sf /usr/X11R6/lib64/libICE.so.6.4 /usr/X11R6/lib64/libICE.so
ln -sf /lib64/libgcc_s.so.1 /lib64/libgcc.so
```

For Red Hat 5.0, you will need the following:

```
sudo ln -sf /usr/lib64/libXm.so.4.0.0 /usr/lib64/libXm.so.3
```

The revision numbers appended to the filenames on the left may be different on your system.

### Intel Linux

ANSYS was built and tested on Red Hat using the compilers as noted in [Table 2.1: Compiler Requirements](#) (p. 11). The ANSYS solver is built on Red Hat Enterprise Linux AS release 4 (Update 5).

For ANSYS Workbench and ANSYS AUTODYN, you need to unlimit the stack size. Add the following to your `.cshrc` file:

```
limit stack unlimited
```

ANSYS TurboGrid 13.0 is not supported on Linux IA-64.

### AMD Opteron

ANSYS was tested on a generic Opteron™ system running Red Hat Enterprise Linux AS release 4 (Update 5).

### Intel Xeon EM64T

ANSYS was built and tested on a generic Intel EM64T system running Red Hat Enterprise Linux AS release 4 (Update 5).

If you are running on Intel's Xeon EM64T system, we recommend that you turn CPU hyperthreading off (default is on). A system administrator needs to reboot the system and enter the BIOS to turn the hyper-threading option off.

**Table 2.1 Compiler Requirements**

Mechanical APDL (ANSYS), ANSYS Workbench Compilers*	CFX Compilers*	FLUENT Compilers*	AUTODYN Compilers*
<b>Linux (all versions)</b>			
Intel 11.1.069 (FORTRAN, C, C++)	PGI Fortran 10.3	Intel 11.1.069 (FORTRAN, C, C++)	Intel 11.1.069 (FORTRAN, C, C++)
<b>HP-UX Itanium 64 (HP-UX B.11.23)</b>			
HP F90 (B.11.23.39) HP aC++/C A.06.20	HP F90 B.11.23.21 HP C C.11.23.04	Native installed OS C compiler	N/A
<b>IBM AIX 64 (AIX 5.3)</b>			
XL Fortran 11.1.0.2 XL C/C++ 9.0.0.5	XL Fortran 11.1 XL C/C++ 9.0	Native installed OS C compiler	N/A
<b>Sun Solaris x64 (Solaris 10)</b>			
Sun Studio 12	N/A	N/A	N/A

\* Compilers are required only if you will be using User Programmable Features or other customization options.



---

## Chapter 3: Installing the Software

---

### 3.1. Pre-Installation Instructions for Download Installations

Before downloading the installation files, you need to accurately determine your platform type. Versions that are optimized for different chip sets from the same vendor can have similar names, causing confusion. We strongly recommend that you run the `get_ansys_platform130` script on each machine first. This script will output the correct platform name for each machine on which it is run. You can download this script by clicking the **Which UNIX/Linux platform am I using?** button on the download site.

We strongly recommend that you review the **Read Download Instructions and Product Information** file included on the download site for the most current download instructions. We also recommend that you review the **What Should I Download?** file to understand which package(s) you need to download, depending on which product(s) you purchased and wish to run.

#### 3.1.1. Downloading the Installation Files

To download the installation files from our website, you will need to be a TECS customer.

Depending on the product/platform combination(s) you choose, you may need to download multiple tar files.

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.

### 3.2. Mounting the DVD Instructions for DVD Installations

If you install ANSYS, Inc. products from the installation media (DVD), you will need to run the installation procedure using either a locally- or remotely-mounted DVD, depending on your site's system.

For a locally-mounted DVD installation, issue the commands from [Table 3.1: Locally-Mounted DVD Procedures](#) (p. 14) for your specific platform. If the target machine does not have a DVD reader, first follow the steps for locally-mounted DVD, and then follow the appropriate procedure for your platform as shown in [Table 3.2: Remotely-Mounted DVD Procedures](#) (p. 14).

Table 3.1 Locally-Mounted DVD Procedures

Platform	Procedure
Linux	<pre>mkdir dvdrom_dir mount -t iso9660 /dev/cdrom dvdrom_dir</pre>

Table 3.2 Remotely-Mounted DVD Procedures

Platform	Procedure
Linux	<p>1. Add the <code>dvdrom_dir</code> directory to the <code>/etc/exports</code> file on the machine with the DVD device. A sample <code>/etc/exports</code> entry is:</p> <pre>/dvdrom_dir *(ro)</pre> <p>or</p> <pre>/dvdrom_dir (ro)</pre> <p>2. Run <b>exportfs</b> to export the <code>dvdrom_dir</code> directory:</p> <pre>exportfs -a</pre> <p>Check the manual page for 'exports' for the correct syntax, as different Linux versions can have different syntax.</p> <p>3. Log on to the machine where you wish to install ANSYS, Inc. products and issue the following commands:</p> <pre>mkdir dvdrom_dir2 mount -t nfs Host:cdrom_dir dvdrom_dir2</pre> <p>where <code>Host</code> is the hostname of the machine where the DVD device is located.</p> <p>Run <b>man exportfs</b> for more information.</p>

If you are installing from media, you will be prompted to change DVDs during the installation. Please make sure you have all installation DVDs before beginning the installation.

### 3.3. Installing ANSYS, Inc. Products

This section explains how to install ANSYS, Inc. products, including ANSYS client licensing, as well as the ANSYS, Inc. License Manager.

The default installation expects you to be logged in as root. You can override that setting and run the installation as a regular user by starting the installation with the `-noroot` command option. If you are not logged in as root, however, you will not be able to set the `/ansys_inc` symbolic link and may potentially experience permission problems. The inability to set the `/ansys_inc` symbolic link will in no way inhibit your ability to run ANSYS, Inc. products; it is provided as a convenience.

If you do not use the `/ansys_inc` symbolic link, we recommend that you install all releases into a common directory to ensure license manager compatibility and availability among releases and products.

### 3.3.1. Product Installation

1. To launch the product install, enter the full path to the installation program and run the `INSTALL` program:

```
/ <mount_dir> /INSTALL
```

To invoke the license manager only install, run `INSTALL.LM` and proceed directly to *License Manager Installation*, below. Use `/INSTALL.LM only` to install the license manager; use `INSTALL` to install the product(s).

We recommend that you run the installation command above from a command prompt. You could encounter unpredictable behavior if you launch the installation by double-clicking on the install program.

2. Select the language. Click **Next**. (Linux only)
3. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
4. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. products. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. You can choose as many platforms as you wish; however, you must run the platform configuration procedure (see *Run the Product Configuration Utility* in [Network Installation and Product Configuration](#) (p. 18)) for each platform other than your current machine type. See [Network Installation and Product Configuration](#) (p. 18) for specific instructions on how to configure a shared installation directory across multiple machines using a common network file system.

Click **Next** to continue.

5. Specify the installation directory. You can accept the default or specify an alternate directory name where the products are to be installed. When you choose an install directory via the Browse feature, the installation will automatically append `/ansys_inc/` to the chosen directory. The installation path can have a maximum of 100 characters.

Default platform-specific directory is `/ansys_inc`, if it exists, or `/usr/ansys_inc` if not.

We strongly recommend that you also set the symbolic link `/ansys_inc` to the directory where the ANSYS, Inc. product is installed. The `/ansys_inc` symbolic link is set by default. If you choose not to specify the symbolic link, substitute the directory path where you installed the product for all subsequent occurrences of `/ansys_inc` in this guide. The symbolic link option is available only if you are installing as root.

If you do not use the `/ansys_inc` symbolic link, we recommend that you install all releases into a common directory to ensure license manager compatibility and availability among releases and products.

If you have already installed the ANSYS Workbench Framework for Ansoft, you must install any additional ANSYS, Inc. products into the same directory.

On Linux, you can choose **Disable RSS** to disable automatic internet feeds to ANSYS, Inc. products.

Click **Next** to continue.

6. Select the components you want to install. You can select as many components as you wish to install. The amount of disk space required for the available components and the disk space available appears at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. However, if you choose to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully.

The components listed represent all available components for the platforms you selected earlier. Not all components are available on all platforms or with all products.

Click **Next** to continue.

7. If you selected Pro/ENGINEER, you may need to specify the Pro/ENGINEER language, the Pro/ENGINEER command, and the full Pro/ENGINEER installation path for an existing Pro/ENGINEER installation. Click **Next**.

If you have not yet installed Pro/ENGINEER, or do not know the requested information, you can choose to skip this configuration step. If you skip this step, you may need to run the **ANS\_ADMIN** utility (for the Mechanical APDL connection functionality) or the **CAD Configuration Manager** (for ANSYS Workbench) to configure Pro/ENGINEER. For Icem CFD Direct CAD Interface for Pro/ENGINEER, you must manually configure Pro/ENGINEER if you skip this step (see [Configuring Pro/ENGINEER Manually](#)). However, you can use the **CAD Configuration Manager** when installing on Linux via the unified installation.

If you selected NX and the UGII environment variables were not set, you may need to specify the NX installation path for an existing NX installation. Click **Next**.

8. A licensing file date verification summary appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.
9. A summary of the selected installation data appears. Information shown includes platform, installation directory, and products. Review the information carefully, and if correct, click **Next** to continue the installation.

The selected products and components are now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the licensing client.

10. The **Licensing Client Installation Configuration** box appears. As the licensing client is installed, progress messages appear in the box.
11. If you do not have an existing `ansyslmd.ini` file, the **Specify License Server Machine - Add Server Specification** box appears. Enter the hostname of your license server machine and click **Next**.

If you already have an existing `ansyslmd.ini` file, you will not see this box and you will proceed directly to the next step.

12. When the client installation is complete, click **Exit**.
13. On the product installation window, click **Next**. You will be asked to participate in an Install Survey. To take the survey, enter the path to a valid browser for your system, and click **Next**. You can also click **Finish** now to skip the survey or when you have completed the survey.

If you have installed ANSYS, Inc. products on a file server, follow the instructions under [Network Installation and Product Configuration](#) (p. 18).



### Caution

If you have a three-license server network, we do not recommend that you load ANSYS, Inc. products on a single file server; doing so eliminates the inherent safety of a redundant setup.

## 3.3.2. License Manager Installation

Follow the instructions below to install the ANSYS License Manager server on UNIX/Linux systems. Client licensing is installed automatically when the product is installed; you do not have to take any further steps to run as a client if you have installed a product. You may safely install the ANSYS License Manager over a client installation. Install the client first (as part of the product installation), and then install the ANSYS License Manager following the steps below. If you are installing both the client and the ANSYS License Manager to the same directory, you will need to do both installations in order to properly configure the ANSYS License Manager.

1. Run `INSTALL.LM` to launch the license manager installation. If you downloaded the license manager installation package, this file will reside in the directory where you untarred the files. If you are running from a DVD, this file will reside in the top level of the DVD.
2. You will be notified that the license manager, if running, will be shut down. Click **OK**.
3. Select the language. Click **Next**. (Linux only)
4. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
5. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. License Manager. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. Click **Next** to continue.
6. The installation directory is specified. You can accept the default or specify an alternate path and directory name where the products are to be installed.

Click **Next** to continue.

7. Select the component you want to install. The amount of disk space required and the disk space available appear at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. However, if you choose to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully.

Click **Next** to continue.

8. A licensing file date verification summary appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.
9. A summary of the selected installation data appears. Information shown includes platform, installation directory, and product. Review the information carefully, and if correct, click **Next** to continue the installation.

The ANSYS License Manager is now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the license manager.

10. The **Licensing Server Installation Configuration** box appears. As the license manager is installed, progress messages appear in the box.
11. The License Wizard will be launched. This wizard walks you through the process of installing or updating a license file, specifying the license server(s) (which updates the `ansyslmd.ini` file), and starting the license manager. The wizard will prompt you for the necessary information at each step. During this process, the license manager will be shut down if it is running. Be aware that this can impact any users currently running using the license manager.

Click **Continue** on the License Wizard to begin, and follow the instructions on the screen.

12. When the License Wizard is complete, click **Exit** on the wizard screen and then click **Finish** again on the Licensing Installation Configuration Log screen.
13. When the license manager installation is complete, click **Finish**.

### 3.3.3. Network Installation and Product Configuration

To complete a network installation (where the product is installed on one machine and one or more clients access that installation to run the product) to a file server machine or a cross-platform installation (where the product for one machine type is installed on a different machine type), follow the steps below. These steps apply to the following products: Mechanical APDL, ANSYS Workbench, ANSYS CFX, ANSYS ICEM CFD, ANSYS FLUENT, ANSYS POLYFLOW, ANSYS Icepak, Common/CAD, and Remote Solve Manager (RSM).

A network installation can be either homogenous or heterogeneous:

- Homogenous network: all clients that will access this installation are of the same UNIX/Linux platform type.
- Heterogeneous network: multiple UNIX/Linux platforms will be installed on a single UNIX/Linux file server, and clients of those platform types will access the product installation on that single UNIX/Linux file server.

You must complete the following steps to run products across a network:

[3.3.3.1. Export the /ansys\\_inc Directory](#)

[3.3.3.2. Run the Product Configuration Utility on All Client Machines](#)

[3.3.3.3. Configure Licensing for File Server Installations](#)

We strongly recommend that these steps be performed by the same non-root user. Installing and configuring as different users may create permissions problems. Likewise, installing and/or configuring as a root user may also result in permissions problems.

#### 3.3.3.1. Export the /ansys\_inc Directory

If you are installing an ANSYS, Inc. product on a file server, you need to export the `/ansys_inc` directory to all client machines so that all users can access the program. You will also need to share the ANSYS directory if the machine you are installing on does not have a DVD drive or an internet connection for downloading files and you need to share files with a machine that does have a DVD drive or internet connection.

The instructions below assume ANSYS, Inc. products were installed in the specified directory.

#### HP

1. Install the ANSYS, Inc. products. The following example uses `/usr/ansys_inc`.
2. Export the `ansys_inc` directory by adding the following line to the `/etc/exports` file:

```
/usr/ansys_inc
```

The default behavior on HP permits read/write access from all clients.

### 3. Run

```
exportfs -a
```

### 4. On all client computers, mount the ansys\_inc directory.

## Linux

### 1. Install the ANSYS, Inc. products. The following example uses /usr/ansys\_inc.

### 2. Export the ansys\_inc directory by adding the following line to the /etc/exports file:

```
/usr/ansys_inc
```

The default behavior on Linux provides read-only access from all clients. To enable read/write permission from all clients, use \*(rw):

```
/usr/ansys_inc *(rw)
```

### 3. Run

```
exportfs -a
```

### 4. On all client computers, mount the ansys\_inc directory.

## IBM

### 1. Install the ANSYS, Inc. products. The following example uses /usr/ansys\_inc.

### 2. Export the ansys\_inc directory by adding the following line to the /etc/exports file:

```
/usr/ansys_inc
```

The default behavior on IBM permits read/write access from all clients.

### 3. Run

```
exportfs -a
```

### 4. On all client computers, mount the ansys\_inc directory.

## Sun

### 1. Install the ANSYS, Inc. products. The following example uses /usr/ansys\_inc.

### 2. Designate the ansys\_inc directory to be exported by adding the following line to the /etc/dfs/dfstab file:

```
share -F nfs /usr/ansys_inc
```

The default behavior on Sun permits read/write access from all clients.

### 3. Issue the following command:

```
/etc/init.d/nfs.server start
```

### 4. Export the ansys\_inc directory to all client computers:

```
shareall
```

5. On all client computers, mount the `ansys_inc` directory.

If you perform a network install where the server and client are on the same platform and you want the clients to be able to modify the licensing configuration, you need to consider the NFS write options for the exported file system as shown in the above examples. You also need local permissions to the licensing directory (`/shared_files/licensing/<platform>/`) if you want to be able to create the `install_lic-config.log` that the license configuration produces.

If you need to transfer the files from a Windows machine with a DVD drive to a UNIX/Linux machine without one, copy the DVD contents using a Samba mount or some other transfer method that is safe to use between Windows and UNIX/Linux.

If sharing the ANSYS directory between UNIX/Linux machines, you must use the same mount point for both the client and server. For example, if you installed to a file server in a directory named `/apps/ansys_inc` and you did not choose the symbolic link to `/ansys_inc`, then you must mount this directory on the client machine using `/apps/ansys_inc` as the mount point. If you did choose the symbolic link to `/ansys_inc` during installation on the file server, you must either use `/ansys_inc` as the mount point on the client or you must create a symbolic link to `/ansys_inc` on the client machine. (The symbolic link is created by default during installation if you installed as root).

### **3.3.3.2. Run the Product Configuration Utility on All Client Machines**

For both network and cross-platform installations, you must run this step on every client machine.

1. On each client machine, issue the following command to run the **Product Configuration** utility:

```
/ansys_inc/v130/commonfiles/tools/<platform>/ProductConfig.sh
```

2. Select **Configure Products**.
3. Select the products you want to configure and click **Configure**.

---

#### **Note**

ANSYS Workbench is configured when Mechanical APDL, ANSYS CFX, ANSYS FLUENT, ANSYS POLYFLOW, ANSYS Icepak, or Remote Solve Manager (RSM) is selected. CFD-Post is configured when either ANSYS CFX or ANSYS FLUENT is configured.

Select **Common/CAD** to configure the Pro/ENGINEER and NX CAD packages.

4. If you selected **Common/CAD**, you will need to supply the following information for Pro/ENGINEER:
  - Pro/ENGINEER language
  - Pro/ENGINEER start command
  - Pro/ENGINEER installation directory path

If you selected **Common/CAD**, you will need to supply the following information for NX:

- NX installation directory path

For both Pro/ENGINEER and NX, you can choose to skip the configuration. If you skip the configuration here, you must manually configure them, or use the **ANS\_ADMIN** utility or the **CAD Configuration Manager** (Linux only) to configure these products. See [Configuring CAD Products \(p. 35\)](#) for detailed information on using alternative methods to configure these products.

- On the **Configuration Complete** dialog box, click **Finish**.

### 3.3.3.3. Configure Licensing for File Server Installations

When you run a cross-platform/file server installation where the product is installed on one platform and run on a different platform, you will need to complete the client licensing configuration on the client machines where you will be running the product.

- Run the **ANSLIC\_ADMIN** utility:

```
/ansys_inc/shared_files/licensing/lic_admin/anslic_admin
```

- Choose **Tools> Complete Unfinished Licensing Installation Configuration**.
- Follow the on-screen prompts to provide any necessary information.
- Click **Exit**.

### 3.3.4. Silent Mode Operations

ANSYS, Inc. supports silent mode operations, including installation, product configuration/unconfig, and uninstall.

You can specify the following product flags. These flags are all valid for a silent install. However, because of the way the products are packaged, not all of these flags may be valid for a silent configuration/unconfiguration, or uninstall.

### Product Flags

<b>Product</b>	<b><i>product_flag</i></b>
Mechanical APDL (ANSYS)	-mechapdl
ANSYS Customization Files	-ansyscust
ANSYS AUTODYN	-autodyn
ANSYS LS-DYNA	-lsdyna
ANSYS CFD-Post	-cfdpost
ANSYS CFX	-cfx
ANSYS TurboGrid	-turbogrid
ANSYS FLUENT	-fluent
ANSYS POLYFLOW	-polyflow
ANSYS ASAS	-asas
ANSYS AQWA	-aqwa
ANSYS Icepak	-icepak
ANSYS ICEM CFD	-icemcfd

*Note: Installing any of the above products will install ANSYS Workbench.*

ANSYS ICEM CFD Pro/ENGINEER Interface	-icemproe
ANSYS ICEM CFD Unigraphics NX Interface	-icemug
Pro/ENGINEER	-proe

<b>Product</b>	<b><i>product_flag</i></b>
NX	-ug
CATIA 4.x	-catia4
Parasolid	-parasolid
ACIS	-acis

### 3.3.4.1. Silent Product and License Manager Installation

You can deploy an ANSYS, Inc. product installation in silent mode. The general form to run a silent product installation, including the client licensing, is:

```
INSTALL -silent -install_dir path -product_flag
```

If no product flags from the list above are specified, all products will be installed. To install specific products, run the silent install with any combination of the product flags listed above (not all products are available on all platforms). For example, to install only TurboGrid and Icepak, issue the following command:

```
INSTALL -silent -install_dir "/ansys_inc/" -turbogrid -icepak
```

Additional command line arguments are available; please see the list below.

To install the ANSYS License Manager on UNIX/Linux systems that will act as license servers, you must run the `INSTALL.LM` command:

```
INSTALL.LM -silent -install_dir path
```

The silent license manager installation is valid only for the default Licensing Configuration option "Run the ANSYS Licensing Interconnect with FLEXlm." Please see the *ANSYS, Inc. Licensing Guide* for more information.

You can use the following arguments when running a silent installation. Note that some options are available only for a silent license manager installation.

-silent	Initiates a silent installation.
-install_dir path	Specifies the directory to which the product or license manager is to be installed. If you want to install to the default location, you can omit the <code>-install_dir</code> argument. The default location is <code>/ansys_inc</code> if the symbolic link is set; otherwise, it will default to <code>/usr/ansys_inc</code> .
-product_flag	Specifies one or more products to install specific products. If you omit the <code>-product_flag</code> argument, all products will be installed. See the list of valid <i>product_flags</i> below.
-productfile path	You can specify an options file that lists the products you want to install. To do so, you must provide a full path to a file containing desired products. See <a href="#">Specifying Products with an Options File</a> below for more details.
-disablerss	Disables automatic internet feeds to ANSYS, Inc. products (Linux only).
-licfilepath path	Specifies the location of the license file to install. If the path is not specified or if the path is the same as the existing license file, the license file will not be installed. Valid only when doing a silent license manager installation ( <code>INSTALL.LM</code> ).
-licserverinfo	Specifies information to be used by the client for the license server. Valid only in conjunction with a silent installation ( <code>INSTALL</code> ). The format is:  Single license server:

	<pre>LI port number:FLEXlm port number:hostname</pre> <p>Example:</p> <pre>2325:1055:abc</pre> <p>Three license servers:</p> <pre>LI port number:FLEXlm port number:hostname,hostname,hostname</pre> <p>Example:</p> <pre>"2325:1055:abc,def,xyz"</pre> <p>The default values for the Licensing Interconnect and FLEXlm port numbers (2325 and 1055, respectively) will be used if they are not specified. However, you do need to include the colons. In a three-server environment, you also need to enclose the values in quotes (Windows only).</p> <p>Example:</p> <pre>::abc</pre> <p>or</p> <pre>"::abc,def,xyz"</pre> <p>Information specified via <code>-licserverinfo</code> will be appended to existing information in the <code>ansyslmd.ini</code> file. To change information already in your <code>ansyslmd.ini</code> file, you must use the <b>ANSLIC_ADMIN</b> utility.</p>
--	--

Any messages will be written to the appropriate installation log files. Installation log files are located in the installation directory: `install.log` contains installation messages, and `install_licconfig.log` contains licensing installation messages. In rare circumstances with a silent licensing installation, the licensing installation messages may not be written to the `install_licconfig.log` (for example, if the silent licensing installation aborts); in these cases, you may find error messages in the `.ansys_install_temp_licconfig_<user>_<index>.log` file, located in `/var/tmp`.

### Caution

A silent license manager installation could shut down the ANSYS, Inc. License Manager, affecting other users who are using that license server machine.

For more information on the silent license manager installation, see the *ANSYS, Inc. Licensing Guide*.

## Specifying Products with an Options File

You can also specify an options file on the command line using the `-productfile path` option. The options file can have any name and extension, but the path must include the full path and filename, including any extension used. The options file can specify which products you want to install. The options file can contain all possible products, with the products you do not want to install commented out, or it can contain only the products you want to install. An example options file is shown below. In the example, Pro/ENGINEER and NX are commented out using the acceptable comment indicators. When using the options file, do not include the dash (-) before the product name.

```
mechapidl
ansyscst
autodyn
lsdyna
cfdpost
cfx
turbogrid
fluent
polyflow
asas
aqwa
icemproe
icemug
icepak
tceng
#proe
#ug
```

### 3.3.4.2. Silent Product Configuration/Unconfiguration

You can also run the ProductConfig utility via command line (i.e., silent mode) to configure products.

To run in silent mode, from each client machine, run the ProductConfig with the `-silent` option:

```
/ansys_inc/v130/commonfiles/tools/<platform>/ProductConfig.sh -silent
```

Use the `-product_flag` argument to specify which products should be configured. If you do not specify one or more products, all products that have been installed will be configured. The valid *product\_flags* are:

<b>Product</b>	<b><i>product_flag</i></b>
Mechanical APDL (ANSYS)	-mechapidl
ANSYS CFD-Post	-cfdpost
ANSYS CFX	-cfx
ANSYS TurboGrid	-turbogrid
ANSYS FLUENT	-fluent
POLYFLOW	-polyflow
ANSYS POLYFLOW/ANSYS Icepak	-icepak
ANSYS ICEM CFD	-icemcfd
ANSYS RSM	-rsm

Errors will be written to the `install.err` on the server machine if you have write access to that machine.

### 3.3.4.3. Silent Uninstall

You can also run the uninstall silently by issuing the following command:

```
/ansys_inc/v130/ans_uninstall130 -silent
```

The silent uninstall will automatically uninstall all products for this release and delete the v130 directory and all subdirectories. You will not be prompted for confirmation.

To uninstall individual products, use the following product options in conjunction with the `-silent` argument:

Mechanical APDL     -mechapidl



ANSYS CFX	-cfx
ANSYS FLUENT	-fluent
ANSYS POLYFLOW	-polyflow
ANSYS ICEM CFD	-icemcfd
ANSYS TurboGrid	-turbogrid
ANSYS Icepak	-icepak

For example, to uninstall only TurboGrid and Icepak, issue the following command:

```
/ansys_inc/v130/ans_uninstall1130 -silent -turbogrid -icepak
```

A record of the uninstall process will be written to `ansys_inc/install.log`. Any errors messages will be written to `ansys_inc/install.err`.

### 3.3.5. Registering the License Server

Follow this procedure to register your license server information if you are a new user or if you are adding or changing a license server machine. See the *ANSYS, Inc. Licensing Guide* for more information on selecting license servers and on using the **ANSLIC\_ADMIN** utility.

1. Follow the instructions in *Installing ANSYS, Inc. Products* (p. 14) to install the appropriate product(s) on the machine(s) designated as license servers.
2. Use the **ANSLIC\_ADMIN** utility to register license server information on each license server. To run the **ANSLIC\_ADMIN** utility on a UNIX/Linux platform, type:

```
/ansys_inc/shared_files/licensing/lic_admin/anslic_admin
```

3. Return the resulting file to your ANSYS, Inc. sales representative. ANSYS, Inc. or your sales representative will then supply you with your license keys. This file is located in `/ansys_inc/shared_files/licensing/licserver.info` by default.
4. Use the **Run the License Wizard** option of the **ANSLIC\_ADMIN** utility to enter your license key and start the license manager.

## 3.4. Post-Installation Procedures

The following post-installation procedures apply to all products. Individual products may have additional post-installation procedures; please refer to the following sections.

After the product is installed, you need to establish some system settings, including pathnames and environment variables. See your shell documentation or man pages file for the shell being used for specific instructions on setting paths and environment variables.

1. Add the following paths to all users' login startup files (i.e., `.cshrc`, `.profile`, or `.login` files).

<code>/ansys_inc/v130/ansys/bin</code>	(or appropriate path to the individual products' executables)
<code>/ansys_inc/shared_files/licensing/lic_admin</code>	(contains ANSLIC_ADMIN utility)
<code>/usr/bin/X11</code>	(HP, IBM)
<code>/usr/openwin/bin</code>	(Sun)

<code>/usr/X11R6/bin</code>	(Linux 32 and Itanium 64)
<code>/usr/X11R6/bin64</code>	(Linux x64)
<code>/opt/mpi/bin</code>	(HP, if using MPI)

2. Set the following environment variables based on the behavior you want. Set the environment variables following the conventions of your shell. For example, to set an environment variable in the C shell, type:

```
setenv environment_variable value
```

For example, to set the **DISPLAY** environment variable, you type the following, where *dev* is the workstation hostname or IP address on which to display graphics:

```
setenv DISPLAY dev:0.0
```

**DISPLAY** - set this to the IP address or hostname of the workstation to which the users will output their analysis results. Note that the X server must have permission to connect to the host machine.

If you will be using connection functionality, you may have additional environment variables to set. See the section [Configuring CAD Products](#) (p. 35) later in this guide for more information.

3. Set the license manager to start automatically at boot time. For platform-specific instructions, see [License Manager Automatic Startup Instructions](#) in the *ANSYS, Inc. Licensing Guide*.
4. Designate server(s) for license checkout and establish necessary user privileges (recommended but not required). For information on these tasks, see [Post-Installation Instructions for the License Manager](#) in the *ANSYS, Inc. Licensing Guide*.
5. Make a backup copy of the `/ansys_inc` directory using a tape backup command such as `tar`.
6. Verify the installation by logging out as root (if you installed as root) and logging back in as a regular user and then starting the product to verify that it starts and runs correctly.

### 3.4.1. Post-Installation Procedures for Mechanical APDL (ANSYS) and ANSYS Workbench Products

The following post-installation procedures apply only to the Mechanical APDL (ANSYS) and ANSYS Workbench products. These are in addition to the post-installation procedures noted above for all products.

1. Set the following environment variables based on the behavior you want. Set the environment variables following the conventions of your shell. Not all of these are required for all integrated ANSYS Workbench products (such as ANSYS AUTODYN), but setting them correctly for ANSYS Workbench will in no way hinder the performance of the other products.

The **ANSYS130\_DIR** environment variable sets the location of the ANSYS directory hierarchy. The default value is `/ansys_inc/v130/ansys`. You probably will not need to reset this variable, unless you change the location of the installed files.

The **ANSYSLIC\_DIR** environment variable sets the location of the ANSYS licensing directory hierarchy. The default value is `/ansys_inc/shared_files/licensing`. You probably will not need to reset this variable, unless you change the location of the licensing files.

**ANSYS130\_PRODUCT** - set this to the correct product variable to run Mechanical APDL (ANSYS) to start with the correct product without specifying the **-p** command modifier each time. See the [Product Variable Table](#) in the *ANSYS, Inc. Licensing Guide* for a list of valid product variables.

**ANSYS\_LOCK** - set to ON (default) to create file locks to prevent users from opening a new job with the same name and in the same directory as the current job.

**ANSYS130\_WORKING\_DIRECTORY** - set this variable to the directory you want designated as your working directory. The working directory setting in the launcher will reflect this setting.

**SPACEBALL2** - set this environment variable to 1 to use a spaceball with Mechanical APDL.

**ANSYS130\_MAT161** - set this environment variable to 1 to enable use of the LS-DYNA

\*MAT\_COMPOSITE\_MSC material (requires an LS-DYNA MAT\_161 license).

**ANSYS130\_MAT162** - set this environment variable to 1 to enable use of the LS-DYNA

\*MAT\_COMPOSITE\_DM3\_MSG material (requires an LS-DYNA MAT\_162 license).

**ANSBROWSER** - set this environment variable to the browser on your system (specify the full path) if the automatic browser detection fails. A browser is needed to view HTML reports and help for the **ANS\_ADMIN** utility and the Mechanical APDL (ANSYS) launcher. By default, **ANSBROWSER** points to one of several UNIX/Linux browsers, based on the browser specified in your path (if any).

If you will be using connection functionality, you may have additional environment variables to set. See the section *Configuring CAD Products* (p. 35) later in this guide for more information.

2. Create or update the `at.` files. The `at.allow` file should contain the username of all users allowed to run batch jobs; the `at.deny` file should contain the username of users who are not permitted to run batch jobs. The files consist of one username per line and can be modified only by the superuser. If neither file exists, only root will be able to run batch jobs.

The `at.` files are located in the `/usr/lib/cron` directory on HP and Sun machines, in the `/var/adm/cron` directory on IBM machines, and in the `/etc` directory on Linux machines.

3. Run the **ANS\_ADMIN** utility to properly configure ANSYS (depending on the products you are running) or relink ANSYS.
4. Specify the product order as it will appear in the Mechanical APDL (ANSYS) launcher (optional). If you want to specify product order, use the **ANSLIC\_ADMIN** utility. See the *ANSYS, Inc. Licensing Guide* for more information.

### Explicit Dynamics, Rigid Dynamics, My Computer Background, and Remote Solve Manager (RSM)

**Users:** If you are running ANSYS Workbench on a multi-user RSM machine, the 'My Computer, Background' Solve Process Settings will likely not function as expected due to write permissions for RSM working directories. In this situation, we strongly recommend that you set up RSM as a daemon.

This issue also affects Rigid Dynamics and Explicit Dynamics using both 'My Computer' and 'My Computer, Background' Solve Process Settings. Please see *Configuring a Multi-User RSM Machine* in the RSM documentation for more information.

**Quality Assurance Program** If you require verification of the ANSYS program, ANSYS, Inc. offers Quality Assurance programs for some ANSYS products. If you are interested in this service, go to <http://www.ansys.com/services/ss-quality-services.asp> or call the ANSYS, Inc. Quality Assurance Group at (724) 746-3304.

## 3.4.2. Post-Installation Procedures for ANSYS CFX

The following post-installation procedures apply only to the ANSYS CFX product.

The UNIX/Linux installation of ANSYS CFX or ANSYS TurboGrid automatically installs the Sun Java 2 Runtime Environment in the `/ansys_inc/v130/commonfiles/jre` directory. Regardless of whether you have modified your setup files, you can still run ANSYS CFX commands by specifying the full pathname of the commands you want to run. This procedure may be useful if you have several releases of ANSYS CFX installed and you want to run more than one release.

Unless you want to run ANSYS CFX commands by typing their full path names, for example `cfx-root/bin/cfx5` (where `cfxroot` is the directory in which ANSYS CFX is installed), your command search paths must be modified to search the directory `cfxroot/bin`. This can be done by one of the following methods:

**Modification of individual user setup files** You can select **Tools> Configure User Startup Files** from the ANSYS CFX Launcher to modify your own setup files: `.login` for the C shell, `.profile` for the Bourne and Korn shells. The utility can also be run from the command line by entering:

```
cfxroot/bin/cfx5setupuser
```

If this modification is done, the ANSYS CFX software will be available every time you log in, just by running the ANSYS CFX commands by name. This method has the advantage that it need not be done by the system administrator, but has the disadvantage that it must be done by each user.

User setup can also be run from the command line by entering:

```
cfxroot/bin/cfx5setupuser
```

Use the `-h` option to view the optional commands.

If you choose to modify your setup files, you will see a message indicating that your setup files have been changed. You will then need to log out and log in again or source your setup files before you can use the software.

**Manual execution of a setup script each time the software is used** You can also use the **Tools** menu of the launcher to launch an editor to create new setup scripts which need to be run each time you want to use the ANSYS CFX software. This method has the advantage of not requiring changes to existing setup files and allows you to use different versions of ANSYS CFX software by running different setup files. The disadvantages are that all users must create their own setup files and run them manually in every session in which they want to run ANSYS CFX software.

Having created the setup files, users of the C shell then need to do the following to run ANSYS CFX:

```
source ~/cfx5.login
cfx5
```

Having created the setup files, users of the Bourne or Korn shell then need to do the following before running ANSYS CFX:

```
.$HOME/cfx5.profile
cfx5
```

**Modification of system setup files** The system administrator modifies the setup files (normally `/etc/profile`), which are run by all users during login, to include the directory `cfxroot/bin` in the command search path. While this has the advantage of only one file needing to be modified in order to allow all users to use the software, it also:

- Affects users regardless of whether they use ANSYS CFX
- Can only be done by the root user
- Is system dependent

Refer to your system documentation for information about which files to change for your workstations.

### 3.4.2.1. Setting up ANSYS TurboGrid Release 13.0

**Manual modification of individual user setup files** To start ANSYS TurboGrid without using full pathnames every time you want to run the ANSYS TurboGrid software, your path must be altered to include the ANSYS TurboGrid directory. This can be done by adding the following line to the `.login` and `.cshrc` files in your home directory:

```
set path=(cfxroot/bin $path)
```

and these lines to the `.profile` file in your home directory, and also the `.bash_profile` if it exists:

```
PATH=cfxroot/bin:$PATH
export PATH
```

and these lines to the `.dtprofile` file in your home directory:

```
PATH=cfxroot/bin:$PATH \
export PATH
```

With the path altered in this way, you can start ANSYS TurboGrid in the current working directory by typing **cfx5tg**.

**Modification of system setup files** The system administrator modifies the setup files ( normally `etc/profile`), which are run by all users during login, to include the directory `cfxroot/bin` in the command search path. While this method has the advantage that only one file needs to be modified to allow all users to use the software, it:

- Affects users regardless of whether they use ANSYS TurboGrid
- Can only be done by the root user
- Is system dependent

Refer to your system documentation for information about which files to change for your workstations.

### 3.4.2.2. Using the ANSYS CFX Launcher to Set Up Users

To create setup files to be merged or run manually, start the ANSYS CFX Launcher using the command:

```
cfxroot/bin/cfx5
```

and select **Tools> Configure User Startup Files**. This option runs `cfxroot/bin/cfx5setupuser` that modifies your setup files or writes the necessary commands to files, which you can merge manually with your existing setup files.

If you choose to modify your setup files, you will see a message indicating that your setup files have been changed. You will then need to log out and log in again or source your setup files before you can use the software.

User setup can also be run from the command line by entering:

```
cfxroot/bin/cfx5setupuser
```

Enter the flag `-h` to view the optional commands.

### 3.4.2.3. Verifying the Installation of ANSYS CFX Products

If you are working on a UNIX/Linux machine and have not used ANSYS CFX before, you will first need to set yourself up to run the ANSYS CFX commands.

To keep all the files for the example together, you should first create a new directory in which to run the example. This can be done by typing:

```
mkdir cfx_example
```

at the command line. You should then change to this directory by typing:

```
cd cfx_example
```

Start the ANSYS CFX Launcher by typing **cfx5**.

To complete testing, perform all the steps in Flow in a Static Mixer.

Note: To speed up the process of testing, consider using the provided session files when working with ANSYS CFX-Pre.

### 3.4.3. Post-Installation Procedures for ANSYS FLUENT

To start ANSYS FLUENT without using full pathnames every time you want to run the FLUENT software, your path must be altered to include the ANSYS FLUENT bin directory. You can do this in the C shell and its derivatives by entering:

```
set path = (/ansys_inc/v130/fluvent/bin $path)
```

or in the Bourne/Korn shell or bash, by entering:

```
PATH=(/ansys_inc/v130/fluvent/bin $path)
export PATH
```

We recommend adding these statements to your \$HOME/.cshrc (C shell), \$HOME/.profile (Bourne/Korn shell), or \$HOME/.bashrc (bash shell) file for regular use.

After installing the ANSYS FLUENT software, you will need to reset the default values in the ANSYS FLUENT launcher as follows:

1. Verify that the **FLUENT\_INC** environment variable is not set.

Remove the **FLUENT\_INC** setting from your .cshrc, .profile, or .bashrc file if you have added it for previous versions. Verify that the environment variable is unset by typing:

```
printenv FLUENT_INC
```

This command should not return anything.

2. Add the following paths to all users' login startup files:

```
<install_dir>/ansys_inc/v130/fluvent/bin
```

3. Run the following command:

```
<install_dir>/ansys_inc/v130/fluvent/bin/fluvent
```

4. Click **Default**.
5. Click **Yes** when asked if you want to discard the LAUNCHER history.
6. Click **Cancel** if you do not wish to start FLUENT at this time. The new defaults will have been saved.

Please refer to the *ANSYS FLUENT Quick Start Guide* for more information.

### 3.4.4. Post-Installation Procedures for ANSYS POLYFLOW

POLYFLOW no longer requires the **FLUENT\_INC** environmental variable. It should be deleted as is recommended for FLUENT:

- Remove the **FLUENT\_INC** setting from your `.cshrc`, `.profile`, or `.bashrc` file if you have added it for previous versions. Verify that the environment variable is unset by typing:

```
printenv FLUENT_INC
```

This command should not return anything.

### 3.4.5. Post-Installation Procedures for ANSYS ICEM CFD

The following post-installation procedures apply only to the ANSYS ICEM CFD product.

- Add the following paths to all users' login startup files (i.e., `.cshrc` or `.login` files).

```
/ansys_inc/v130/icemcfd/<ICEMCFD_OS_DIR>/bin
```

ANSYS ICEM CFD operating system directories are:

Hardware	ICEMCFD_OS_DIR
HP-UX Itanium 64	hpit
IBM AIX 64	ibm64
Linux 32	linux
Linux Itanium 64	linux64
Linux x64 (AMD Opteron)	linux64_amd
Linux x64 (EM64T)	linux64_amd

- Add the following environment variable to all users' login startup files.

```
ICEM_ACN - set to /ansys_inc/v130/icemcfd/<ICEMCFD_OS_DIR>/bin
```

- Start ANSYS ICEM CFD by typing **icemcfd**.

### 3.4.6. Post-Installation Procedures for ANSYS AUTODYN

The following post-installation procedures apply only to the AUTODYN product.

Add the following paths to all users' login startup files (i.e., `.cshrc` or `.login` files).

```
/ansys_inc/v130/autodyn/bin
```

path to the AUTODYN executable

Please refer to the ANSYS AUTODYN *Quick Start Guide* and *What's New* documents in the ANSYS AUTODYN help directory for information on starting and running ANSYS AUTODYN.

## 3.5. Translated Message File Installation for Mechanical APDL (ANSYS)

If your ANSYS sales representative has supplied you with message files in your local language, use the following procedures to install and access these files. You must create new message files for each release because error messages may occur in a different order for each release.



1. Create a language-named subdirectory (for example, `fr` for French) under the `/docu` directory:

```
mkdir /ansys_inc/v130/ansys/docu/fr
```

2. Copy the message files `msgcat.130`, `msgidx.130`, and `msgfnm.130` into that subdirectory.
3. Access these files from the Language Selection option of the launcher or via the `-l` command line option:

```
ansys130 -l fr
```

## 3.6. Launching ANSYS, Inc. Products

To launch ANSYS, Inc. products on Unix/Linux platforms, issue the appropriate command from the list below. The paths specified assume that you installed the product using the symbolic link to `/ansys_inc`. If you did not, substitute your installation path for the path given below.

**Table 3.3 Startup Commands**

Product	Command	Notes
Mechanical APDL	<code>/ansys_inc/v130/ansys/bin/ansys130</code>	For a complete list of command line options, see <a href="#">Starting an ANSYS Session from the Command Level</a> in the <i>Operations Guide</i> .
ANSYS Workbench	<code>/ansys_inc/v130/Framework/bin/&lt;platform&gt;/runwb2</code>	
ANSYS CFX	<code>/ansys_inc/v130/CFX/bin/cfx5</code>	
ANSYS FLUENT	<code>/ansys_inc/v130/fluent/bin/fluent</code>	For a complete list of command line and launcher options, see <i>Starting ANSYS FLUENT</i> in the <i>FLUENT Users Guide</i> .
ANSYS ICEM CFD	<code>/ansys_inc/v130/icemcfd/&lt;platform&gt;/bin/icemcfd</code>	
ANSYS POLYFLOW	<code>/ansys_inc/v130/polyflow/bin/polyman</code>	Starts the POLYFLOW MANager. For any other tool, use <code>/ansys_inc/v130/polyflow/bin/&lt;tool&gt;</code>
ANSYS CFD-Post	<code>/ansys_inc/v130/CFD-Post/bin/cfdpost</code>	
ANSYS Icepak	<code>/ansys_inc/v130/Icepak/bin/icepak</code>	
ANSYS TurboGrid	<code>/ansys_inc/v130/TurboGrid/bin/cfxtg</code>	
ANSYS AUTODYN	<code>/ansys_inc/v130/autodyn/bin/autodyn130</code>	solver only

## 3.7. Uninstalling ANSYS, Inc. Products

To uninstall a product, issue the following command:

```
/ansys_inc/v130/ans_uninstall130
```



Alternatively, if you are using the Mechanical APDL product, you can use the **Uninstall** option of the **ANS\_ADMIN** utility. To launch **ANS\_ADMIN**, issue the following command:

```
/ansys_inc/v130/ansys/bin/ans_admin130
```

1. From the uninstall panel, choose one of the following uninstall options:

- **Select Products to Uninstall**
- **Uninstall all ANSYS Products**

If you are not a superuser, you will see a warning message, and then the uninstall continues.

2. If you chose **Select Products to Uninstall**, you will see a list of products that are installed. Select those products you want to uninstall and click **Continue**. Then click **OK** to confirm the list of products to be uninstalled.

If you chose **Uninstall all ANSYS Products**, you will be asked to confirm that you want to uninstall all products. Click **Yes**. This process will remove all files and directories under and including the /ansys\_inc/v130 directory.

3. When the uninstall has completed, click **Finish**.

In the case of a platform (file server) installation, the uninstall will remove the selected product(s) from all of the UNIX/Linux platforms.



---

## Chapter 4: Configuring CAD Products

---

The connection functionality of all supported CAD products is included with the ANSYS release media, and all CAD functionality except Pro/ENGINEER, NX, and CATIA v5 is installed by default. To use the connection functionality, you need to ensure that the product is properly licensed, and set any necessary environment variables or other configuration as appropriate. See the manuals for the individual CAD products for information about environment variables and other configuration requirements.

For complete information about the files you can import, see *Introduction to Import* in the *ANSYS Connection User's Guide*.

---

### Caution

Be sure to install Mechanical APDL (ANSYS) and the connection functionality from the same release. If you attempt to run the latest connection functionality on a machine that is running an earlier release of Mechanical APDL, or vice versa, the connection may fail.

If you are running Pro/ENGINEER or NX, some additional configuration may be required, especially if you chose to skip configuring these products during the installation process. The following sections describe any post-installation configuration procedures that are required and how to manually configure these products if you did not configure these products during installation or if you are updating your CAD versions. You can configure the Connection for Pro/ENGINEER by running the **ANS\_ADMIN** utility. You must configure the ICEM CFD Direct CAD Interface for Pro/ENGINEER manually (except when installing on Linux from the unified installation, in which case the **CAD Configuration Manager** is available), and you can configure NX using environment variables. These methods are all described in the following sections.

---

### Note

You cannot use the **ANS\_ADMIN** utility to configure the Connection for Pro/ENGINEER on Linux x64 systems.

## 4.1. Using the CAD Configuration Manager on Linux

The **CAD Configuration Manager** utility allows you to configure geometry interfaces for ANSYS Workbench and ICEM CFD Direct CAD interfaces on Linux systems. CAD configuration is typically handled during the product installation; however, if you chose to skip those steps, or if you make changes to your local CAD configuration between releases (for example, you move or update your CAD package, or remove it entirely), you can use this utility. Note that only Reader mode for NX and ProE are supported on Linux. You should continue to use the **ANS\_ADMIN** utility to configure your connection products on all Unix/Linux platforms.

The **CAD Configuration Manager** is organized into several tabs:

- CAD Selection
- Pro/ENGINEER
- NX

- Teamcenter Engineering (Windows only)
- CAD Configuration

An administrative user has the option to configure or unconfigure any selected CAD systems either for the present user's environment or for all users, as indicated by the **Configuration actions apply to** options. When the original installation was performed by a non-administrative user, an administrative user will only be allowed to configure or unconfigure for all users. In this situation, Pro/Engineer and NX configure and unconfigure actions are skipped. Non-administrative users will only be allowed to configure for themselves. Any user-specific configuration settings take precedence over global configuration settings.

This document describes how to use the **CAD Configuration Manager** as a wizard on Linux systems, beginning with the **CAD Selection** tab and using the **Next** button to progress through the configuration process. You can also manually select each tab to progress through the configuration process; both methods work the same. However, the applicable tabs will not be enabled until you choose the associated product on the **CAD Selection** tab.

You can choose **Help** to view instructions on using the **CAD Configuration Manager** at any time. If the help does not load into your default browser, set the **BROWSER** environment variable to the path of your HTML viewer (such as Mozilla or Firefox) and restart the **CAD Configuration Manager**.

1. Run the following command to start the **CAD Configuration Manager**, substituting the full installation path if different than `/ansys_inc`:  

```
/ansys_inc/v130/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigUtilityGUI.exe.
```
2. On the **CAD Selection** tab, choose the ANSYS products and the CAD products that you need to configure. You must select at least one ANSYS product and at least one CAD product to enable the remaining functionality.
3. If you selected Pro/ENGINEER as one of your CAD products, the **Pro/ENGINEER** tab opens.
  - a. Enter or browse to the Pro/ENGINEER installation location.
  - b. Enter or browse to the Pro/ENGINEER start command. Include the complete path if entering the command manually.
  - c. Click **Next**.
4. If you selected NX as one of your CAD products, the **NX** tab opens.
  - a. Enter or browse to the NX installation location.
  - b. The NX custom directory file is not applicable on Linux and can be ignored as it will be disabled.
  - c. Click **Next**.
5. The **CAD Configuration** tab opens.
  - a. Click the **Configure Selected CAD Interfaces** button.
  - b. When the configuration for all products is complete, log entries appear, listing those products that were successfully configured and those that were not. Address any errors and reconfigure.
  - c. For more details, click the **Display Configuration Log File** button to see a detailed log file.
6. When all of your CAD products have been successfully configured, click **Exit**.

### Note

Configuration of the ANSYS Connection for Pro/ENGINEER on Linux (as with other UNIX platforms) is still performed through the **ANS\_ADMIN** utility.

You can review the **CAD Configuration Manager** log file, `CADConfigurationMgr.log`, in `/ansys_inc`. If you do not have write permissions to the `/ansys_inc` directory, the log file will be written to `$TEMP`. If you have not defined `$TEMP`, the log file is written to `/tmp`.

### 4.1.1. Unconfiguring

If you need to unconfigure any of your CAD products, follow the steps above, but choose **Unconfigure Selected CAD Interfaces** on the **CAD Configuration** tab.

### 4.1.2. Running the CAD Configuration Manager in Batch Mode

You can configure ANSYS Geometry Interfaces by supplying the **CAD Configuration Manager** with arguments specific to the CAD sources you want to make available. The following table contains a list of supported arguments.

The command to run the **CAD Configuration Manager** in batch mode on Linux is:

```
/ansys_inc/v130/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigurationUtility.exe
-arguments
```

Argument	Value	Comment
unconfigure	None	Results in any specified CAD sources being unconfigured. When this flag is absent, the <b>CAD Configuration Manager</b> will attempt to configure all designated CAD sources.
PE_CONFIG_WB	None	Configure/unconfigure Pro/ENGINEER Geometry Interface to Workbench. The arguments PROELOADPOINT and PROE_START_CMD must also be specified.
PROELOADPOINT	Full path to Pro/ENGINEER installation (quotations are required if there are spaces in the path).	Not required with unconfigure operation.
PROE_START_CMD	Full path to command used to launch Pro/ENGINEER (quotations are required if there are spaces in the path).	Not required with unconfigure operation.
UG_CONFIG_WB	None	Configure/unconfigure NX Geometry Interface to Workbench. The argument UGII_BASE_DIR must also be specified.
UGII_BASE_DIR	Full path to NX installation (quotations are required if there are spaces in the path).	This should agree with environment variable <b>UGII_BASE_DIR</b> . Not required with unconfigure operation.
UG_USE_COLORS	None	Process NX entity colors as possible source of attributes and named selections. Not required with unconfigure operation.

Argument	Value	Comment
PE_CONFIG_ICEM	None	Configure/unconfigure the ICEM CFD Direct CAD Interface to Pro/ENGINEER. The arguments PROELOADPOINT and PROE_START_CMD must also be specified.

### Note

All arguments require a dash (-) before them in order to be properly recognized by the **CAD Configuration Manager**. Arguments' values should not have a dash preceding them.

For example, you can configure the Pro/ENGINEER Geometry Interface to ANSYS Workbench from the command line by using the following:

```
<installpath>/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigurationUtility.exe
-PE_CONFIG_WB -PROELOADPOINT "<pathtoproe>/proeWildfire 4.0"
-PROE_START_CMD "pathtoproe start cmd"
```

where *installpath* is the /v130 directory under the installation directory.

To unconfigure the same CAD Interface, the command would be:

```
<installpath>/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigurationUtility.exe
-unconfigure -PE_CONFIG_WB
```

Although the argument order does not matter, an argument value must immediately follow its argument.

## 4.1.3. Pro/ENGINEER Configuration

Running the **CAD Configuration Manager** for Pro/ENGINEER performs the following steps to activate the Pro/ENGINEER Pro/ENGINEER ANSYS Workbench reader:

- Updates references to **PROE\_START\_CMD130** and **PRO\_COMM\_MSG\_EXE** within the script /ansys\_inc/v130/aisol/.workbench to match user inputs for the Pro/ENGINEER starting command and install location.
- For ANSYS ICEM CFD, a config.pro file is created in \$HOME with the entry:

```
PROTKDAT $PROMIF_ACN/protk.dat
```

You must set the environment variable **PROMIF\_ACN** to the install point of the Direct CAD Interface (typically /ansys\_inc/v130/icemcfd/<platform>/dif/pro.

In order to be able export to Mechanical APDL, set the **LD\_LIBRARY\_PATH** variable as follows:

```
LD_LIBRARY_PATH /ansys_inc/v130/ansys/syslib/linia32
```

Previous releases of ANSYS, Inc. software installed the `protk.dat` file to the Pro/ENGINEER installation directory as listed in [The `protk.dat` File \(p. 40\)](#), below. If you have a previous release installed, you may want to verify that the `config.pro` file is pointing to the file located in the ANSYS installation directory for Linux platforms only. Other platforms that do not use the **CAD Configuration Manager** utility will continue to point to the Pro/ENGINEER installation directory for this file.

## 4.1.4. NX Configuration

Running the **CAD Configuration Manager** for NX performs the following steps to activate the NX reader:

- Registers the NX Reader for ANSYS Workbench by copying the file `UGNX#.Component.XML` from `/ansys_inc/v130/aisol/CADIntegration/UG` to either `/ansys_inc/v130/commonfiles/registry/linux64/append` when the configuration manager is in administrative mode or to `$HOME/.config/Ansys/130/UserRegFiles_NNNN/append`, when the **CAD Configuration Manager** is run by a non-administrator. The **CAD Configuration Manager** ensures that only one version of the file exists in the target location.

`UGNX#` is the NX version detected by the **CAD Configuration Manager**, and `NNNN` is a numeric identifier appended to the `UserRegFiles` directory.

- You must specify the environment variable **UGII\_BASE\_DIR**, which must match the install location supplied to the **CAD Configuration Manager**. The WorkBench Reader for NX will not work without this variable set properly, as it is required for proper startup of the CAD.

The **CAD Configuration Manager** does not configure the ICEM CFD Direct CAD Interface to NX on Linux. To manually configure this product, set the environment variable **UGII\_VENDOR\_DIR** to `/ansys_inc/v130/icemcfd/linux64_amd64/dif/ug/lib/ug/ugXX_vendor_dir`. Here, `XX` corresponds to the version of NX intended to load the NX Direct CAD Interface:

XX=22 for NX4

XX=23 for NX5

## 4.2. Configuring the Connection for Pro/ENGINEER

On UNIX/Linux systems, use the **ANS\_ADMIN** utility to configure the Connection for Pro/ENGINEER. To launch the **ANS\_ADMIN** utility, run the following command:

```
/ansys_inc/v130/ansys/bin/ans_admin130
```

Select the **Configure for Pro/ENGINEER** option.

When you select this option, you will be asked to specify the following:

- The Pro/ENGINEER language.
- The Pro/ENGINEER command. If you do not specify this setting, ANSYS will use the default setting in the **PROE\_START\_CMD130** environment variable. The default for this environment variable is `proe1`. If you enter a different command in this field, ANSYS will update the environment variable to use the setting specified here.
- The full Pro/ENGINEER installation path for an existing Pro/ENGINEER installation.

### 4.2.1. Other Connection for Pro/ENGINEER Configuration Steps

In addition to the configuration steps described above, you must also set up the `config.anscon` and `protk.dat` files. These files are required for the connection for Pro/ENGINEER. The `config.anscon` file is placed in the `/ansys_inc/v130/ansys/ac4/data` directory during the ANSYS installation. The `protk.dat` file is placed in the Pro/ENGINEER installation directory when you run the **ANS\_ADMIN's Configure for Pro/ENGINEER** option. This file defines:

- The name of the executable for the connection for Pro/ENGINEER
- The path to the executable
- The path to the message file
- The current revision of Pro/ENGINEER

**Note**

You must have permissions to create/update the `protk.dat` file in the Pro/ENGINEER installation directory when using the **ANS\_ADMIN's Configure for Pro/ENGINEER** option.

---

**Caution**

Check the `protk.dat` file for duplicate definitions. If you have installed other Pro/ENGINEER products before, you may have duplicate entries for the name of the executable and the revision. The **ANS\_ADMIN** utility creates the `protk.dat` file in the Pro/ENGINEER installation directory as listed in [The `protk.dat` File \(p. 40\)](#), below. If a `protk.dat` file already exists, the information is appended. Edit the `protk.dat` file and delete any duplicate definitions. The new file should have the following entries for the current release of this product, but may contain additional definitions if other Pro/ENGINEER products are in use:

```
name      ac4pro130
exec_path /ansys_inc/v130/ansys/ac4/bin/pro/<platform>/ac4pro
text_path /ansys_inc/v130/ansys/ac4/data/pro/text
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
revision  24.0
STARTUP  dll
end
```

See [The `protk.dat` File \(p. 40\)](#) for more information about the `protk.dat` file. The `config.anscon` file is documented in [Configuring the `config.anscon` File for Connection for Pro/ENGINEER \(p. 41\)](#).

---

**Note**

If the ANSGeom menu in Pro/ENGINEER does not appear correctly, copy the `config.anscon` file into your working directory and restart Pro/ENGINEER.

## 4.2.2. The `protk.dat` File

The `protk.dat` file defines the name of the connection executable for Pro/ENGINEER, the path to the executable, the path to the message file, and the current revision of the installed Pro/ENGINEER.

---

**Note**

Mechanical APDL (ANSYS) and ICEM CFD each use their own `protk.dat` file.

The `protk.dat` file may be stored in one of these directories.

1. Pro/ENGINEER working directory (directory from which Pro/ENGINEER was launched)
2. `/<ProE_install_dir>/<proe_platform>/text/usascii/`
3. `<ProE_install_dir>/<proe_platform>/text`
4. For ICEM CFD, in `$ICEM_ACN/dif/pro`, which is referenced from the `config.pro` file

`usascii` is the default language (character set). You may have specified a different language during installation.



The `<proe_platform>` variable is the name that Pro/ENGINEER gives to its platform directories. Pro/ENGINEER platform names are similar, but not identical, to the ANSYS platform names:

**Table 4.1 Names for ANSYS and Pro/ENGINEER Platform Directories**

Hardware Platform	ANSYS Platform Name <PLATFORM>	Pro/ENGINEER Platform Name <PROE_PLATFORM>
Sun Solaris x64	solx64	sun_solaris_x64
Linux	lin32 or linux64	i486_linux

### 4.2.3. The config.pro File

A typical ANSYS ICEM CFD entry in the `config.pro` file might look like the following example.

```
PROTKDAT $PROMIF_ACN/protk.dat
```

### 4.2.4. Configuring the config.anscon File for Connection for Pro/ENGINEER

Users who launch Mechanical APDL (ANSYS) from Pro/ENGINEER will need the information from the `config.anscon` file. This file is installed for all users in `/ansys_inc/v130/ansys/ac4/data/`. Here is a sample `config.anscon` file:

```
ANSYS_CMD           /ansys_inc/v130/ansys/bin/ansys130
ANSYS_GRAPHIC_DEVICE x11-stat
ANSYS_SOLVER         Sparse**
ANSYS_SELECTED_LAYERS 1-256**
ANSYS_GEOMETRY_TYPE   Solids Only**
ANSYS_NEUTRAL_FORMAT  Yes
ANSYS_PRODUCT_NAME    ANSYS
```

\*\*These variables are not supported by Pro/ENGINEER and are ignored by Pro/ENGINEER. They appear for compatibility with NX.

You can modify the `config.anscon` file to set information for all connection for Pro/ENGINEER users. Pro/ENGINEER users can copy this file to their working directory and configure it for their own projects. See *Setting ANSYS Configuration Parameters* in the *ANSYS Connection User's Guide* for more information about the `config.anscon` file.

## 4.3. Configuring Pro/ENGINEER Manually

For Mechanical APDL (ANSYS), the following environment variables are typically set during the installation process or by running **ANS\_ADMIN**. For ICEM CFD, you need to set them manually. They are described below should you need to modify or reset them.

**PROE\_START\_CMD130:** Specifies the name of the command which launches Pro/ENGINEER on the system. Do not use an alias to the Pro/ENGINEER executable. The path to the executable should be already set if the Pro/ENGINEER installation path has been defined for the **PATH** environment setting. Defaults to **proe1**.

Exporting from Pro/ENGINEER also requires that the **ANSYS130\_DIR** environment variable to be set. If the `/ansys_inc` symbolic link is not used, the **ANSYSLIC\_DIR** environment variable must also be set.

If you are using the Connection for Pro/ENGINEER on a Linux machine, you will need to set the following environment variable in order to export your model to Pro/ENGINEER (this environment variable is not needed to import a Pro/ENGINEER model):

**LD\_LIBRARY\_PATH** -- Set this environment variable to `/ansys_inc/v130/ansys/syslib/linia32`.

If you are using Pro/ENGINEER on a Linux machine with ICEM CFD, set **LD\_LIBRARY\_PATH** to `$ICEM_ACN/lib` on 32-bit machines or to `$PROMIF_ACN` on 64-bit machines.

## ICEM CFD Environment Variables

**ICEM\_ACN**: Specifies the ICEM CFD installation directory, typically `/ansys_inc/v130/icemcfd/<platform>`.

**PROMIF\_ACN**: Specifies the full pathname to the ICEM CFD Pro/ENGINEER installation, typically `$ICEM_ACN/dif/pro`.

### 4.3.1. Configuring the Geometry Interface for Pro/ENGINEER for ANSYS Workbench Products

If you chose to skip configuring the Geometry Interface for Pro/ENGINEER during the ANSYS Workbench installation, you may need to run the **CAD Configuration Utility's CAD Interface Configuration > Configure for Pro/ENGINEER** option. This option sets all necessary registry entries.

1. Launch **CAD Configuration Utility**.
2. Select **CAD Interface Configuration** to continue.
3. Select **Configure for Pro/ENGINEER** to continue.
4. Enter the name of the language (character set) used with Pro/ENGINEER (usascii is the default) and the Pro/ENGINEER installation path. Select **OK** to continue.
5. Select **EXIT** to close **CAD Configuration Utility**.

## 4.4. Configuring NX

To use NX in reader mode, you will not need to perform any additional configuration steps. Mechanical APDL (ANSYS) and ICEM CFD use the following three NX environment variables, which are set during the NX installation:

**UGII\_BASE\_DIR**  
**UGII\_ROOT\_DIR**  
**UGS\_LICENSE\_SERVER**

To use NX in plug-in mode, you need to set additional environment variables. ANSYS provides default settings for some of the necessary environment variables. In general, you will not need to reset these variables. They are described below should you need to modify or reset them.

**ANSCON\_CONFIG\_DIR**: Sets the location of the `config.anscon` configuration file. This environment variable is used only if the **ANSYS130\_DIR** has not been set. Defaults to `/ansys_inc/v130/ansys/ac4/data`.

**UGII\_VENDOR\_DIR**: This environment variable is not set during the installation process and can only be set in a user's start-up file or at command level before running NX. This environment variable defines the Mechanical APDL (ANSYS) or ICEM CFD menu for NX. You must set this environment variable if you will be running the connection for NX product from inside NX. This environment variable tells NX where to find the Mechanical APDL or ICEM CFD program.

**Note**

You cannot set the **UGII\_VENDOR\_DIR** for both Mechanical APDL and ICEM CFD. You must set it for only one product.

For Mechanical APDL, set this environment variable as follows:

```
setenv UGII_VENDOR_DIR /ansys_inc/v130/ansys/ac4/bin/ug50/<platform>
```

For ICEM CFD, set this environment variable as follows:

```
setenv UGII_VENDOR_DIR $ICEM_ACN/dif/ug/lib/ug/ugNN_vendor_dir
```

Where *NN* = 22 for NX4, 24 for NX6

**UGII\_OPTION:** This environment variable sets the case of characters in directory names on the current system. You must set this environment variable if the directory names in this environment variable or in **ANSCON\_CONFIG\_DIR** contain uppercase characters. Possible options are MIXED, LOWER or UPPER. We recommend the following setting:

```
setenv UGII_OPTION MIXED
```

### 4.4.1. Configuring the Connection for NX

No additional configuration is needed to configure the connection capability for NX.

However, to export to Mechanical APDL (ANSYS) from NX, you will need the `config.anscon` file. This file is installed for all users in `/ansys_inc/v130/ansys/ac4/data/`. Here is a sample `config.anscon` file:

```
ANSYS_CMD           /ansys_inc/v130/ansys/bin/ansys130
ANSYS_GRAPHIC_DEVICE x11-stat
ANSYS_SOLVER         Sparse**
ANSYS_SELECTED_LAYERS 1-256**
ANSYS_GEOMETRY_TYPE  Solids Only**
ANSYS_NEUTRAL_FORMAT Yes
ANSYS_PRODUCT_NAME   ANSYS
```

You can modify the `config.anscon` file to set information for all NX users. NX users can copy this file to their working directory and configure it for their own projects. NX users should see *Using the Configuration Editor* in the *ANSYS Connection User's Guide* for more information. See *Setting ANSYS Configuration Parameters* in the *ANSYS Connection User's Guide* for more information about the `config.anscon` file.

### 4.4.2. Configuring the Geometry Interface for NX for ANSYS Workbench Products

If you chose to skip configuring the Geometry Interface for NX during the ANSYS Workbench installation, you may need to run the **CAD Configuration Utility's CAD Interface Configuration > Configure UG NX Reader for Workbench** option.

1. Launch **CAD Configuration Utility**.
2. Select **CAD Interface Configuration** to continue.
3. Select **Configure UG NX Reader for Workbench** to continue.
4. Enter the directory path to the NX installation path. Select **OK** to continue.
5. Select **EXIT** to close **CAD Configuration Utility**.



---

## Chapter 5: Troubleshooting

---

### 5.1. Installation Troubleshooting

This section lists problems and error messages that you may encounter while installing and/or running ANSYS, Inc. products. After each situation description or error message is the user action required to correct the problem.

You can also find answers to commonly-asked questions on our customer portal. After you log in to the customer portal, select **Online Support**> **Installation/System FAQs**. Then select either the Windows or UNIX/Linux link.

For information on licensing-related errors, see the [Troubleshooting](#) section of the *ANSYS, Inc. Licensing Guide*.

In addition, this appendix describes the **ansys\_pid** utility, which is useful for troubleshooting some problems.

#### 5.1.1. Using ANSLIC\_ADMIN to Gather Diagnostic Information

You can use the **Gather Diagnostic Information** option of the **ANSLIC\_ADMIN** utility to query the system for various pieces of information that may be needed for troubleshooting certain problems. This option collects information about the system as well as about ANSYS licenses and sends all of the information that it collects to the log area. At times it may be necessary to provide the information for technical support. Use the **Write to File** button at the bottom of the **ANSLIC\_ADMIN** to write a file. Then forward the file to the appropriate person.

You can use the **Gather Diagnostic Information** option of the **ANSLIC\_ADMIN** utility to query the system for various pieces of information that may be needed for troubleshooting certain problems. This option collects information about the system as well as about ANSYS licenses and sends all of the information that it collects to the log area. At times it may be necessary to provide the information for technical support. Use the **Write to File** button at the bottom of the **ANSLIC\_ADMIN** to write a file. Then forward the file to the appropriate person.

#### 5.1.2. The GUI Installation Process Hangs

- If the GUI installation process appears to “hang” during file extraction, with no activity appearing in the message window, press **ENTER** on the command window used to start the installation. When the installation is complete, check the message window carefully for any installation errors or warnings; however, this situation rarely causes installation errors.
- If the installation does not progress beyond the extraction of any single file during the file extraction phase, you may have insufficient disk space in the temporary directory that the file extraction utility uses. Be aware that some components require a lengthy extraction time; we recommend allowing up to 30 minutes for these components to extract if you are running on a particularly old or slow system.

To resolve this problem, remove files from your \$TEMP directory to free up disk space, or increase the size of any disk quotas on your \$TEMP directory.

### 5.1.3. The Target Machine Does Not Have a DVD Drive

If the target machine does not have a DVD drive, we recommend that you download the installation files from the Customer Portal on [www.ansys.com](http://www.ansys.com) or follow the instructions in *Mounting the DVD Instructions for DVD Installations* (p. 13) to mount to a machine that does have a DVD drive. You can also mount the DVD on a Windows machine, then copy all of the files (using binary format) to the target UNIX/Linux machine in the exact same directory structure as is on the DVD, and follow the instructions in *Downloading the Installation Files*. However, we strongly recommend against copying the UNIX/Linux installation files from a Windows machine; such a file transfer can result in unpredictable behavior.

### 5.1.4. The Online Help System Does Not Run Properly

The online documentation for the UNIX/Linux versions of the ANSYS CFX and ANSYS ICEM CFD products uses the Oracle Help browser. Several unusual behaviors and their cause and resolution are listed.

If you request help and nothing happens:

- On a few machine configurations, the online help system can take up to 60 seconds longer than CFX or ICEM CFD to initialize, due to the large amount of documentation available. If you request help within 60 seconds of running the product, you may simply need to wait a few more seconds for help to start.
- If you have minimized the help windows, activating help will not restore them to their original size. Instead of minimizing help, you can close the help windows when you are done using help. This does not exit the help, so the help system will still be available quickly next time you request help.
- The help system may have aborted due to a system error. You cannot restart the help system from your current session. You can save your work and restart a new CFX or ICEM CFD session.

If the text in the online help is unclear (the fonts are too small, too large, or rough around the edges):

- Use the [+] or [-] keys to enlarge or reduce the fonts. Depending on your system's configuration and installed fonts, you may need to experiment with the [+] and [-] keys to adjust your font size. The [+] will increase the font size, and the [-] will decrease it. On some systems, you need to use the [+] and [-] keys on the standard keypad, rather than those on the number pad.

If, when starting CFX or ICEM CFD over Exceed, all of the UNIX/Linux window controls (borders, close buttons, etc.) disappear, or the help windows appear but are small and in the top left corner of the screen:

- You need to change your window manager settings in Exceed. See the section Note to Exceed Users earlier in this guide for information on changing your window manager settings.
- If changing the window manager settings does not correct the problem, we recommend upgrading to Exceed 7.0 or higher.
- If the problem still occurs, change the Window Manager to **X**. You must start an X Windows manager like mwm before running CFX or ICEM CFD.

Mouse clicks are slow to initiate an action.

- This is a result of the mouse click signals traveling over the network via an Exceed connection. Try adjusting the Exceed **Performance** settings. See the section Note to Exceed Users earlier in this guide for information on specific changes we suggest.

### 5.1.5. CAD Configuration Manager Help Does Not Load

If the help for the **CAD Configuration Manager** does not load into your default browser, set the **BROWSER** environment variable to the path of your HTML viewer (such as Mozilla or Firefox) and restart the **CAD Configuration Manager**.

### 5.1.6. Cannot Enter Data in Text Fields

**SUSE** On some SUSE Linux systems, if you cannot enter data in text fields during the installation or when using the **ANSLIC\_ADMIN** utility, you may be encountering a Tcl incompatibility. To correct the problem, unset the following environment variables before running the installation or the **ANSLIC\_ADMIN** utility:

```
QT_IM_MODULE
XMODIFIERS
GTK_IM_MODULE
```

You should reset these environment variables when you are finished running the installation or using **ANSLIC\_ADMIN**. Do not permanently unset these environment variables as doing so could affect other applications.

**Red Hat** If you are installing on Linux Red Hat 5 machines on the local console, and you are unable to enter text into any of the text fields in the installer, you may have to disable SCIM (Smart Common Input Method) as follows:

In KDE: Choose **[K Menu] >Settings >Input Method**

In GNOME: Choose **System >Preferences >More Preferences >Input Methods**.

In other window managers: Run program **im-chooser**.

Select **Never Use Input Methods**.

Then rerun the installation.

### 5.1.7. Download and Installation Error Messages

**The current platform type is not selected and is not included in current download files. Please make sure that you are using the correct media or downloaded file. Continuing with a platform installation may require additional post-install configuration.**

**Do you wish to continue?**

This message occurs if you have selected a platform for installation that does not match the files you are trying to install (either from the installation DVD or from downloaded installation files).

**Cannot find file <product>.tar in directory <dvd\_dir>**

This error may appear during the ANSYS installation if you have entered the wrong DVD pathname.

Check [Table 3.1: Locally-Mounted DVD Procedures \(p. 14\)](#) and enter the correct pathname for your platform.

**Licensing files currently installed for <platform> are more recent than those on the installation media. The <platform> files will not be installed and will be deselected.**

This UNIX/Linux-only message appears during an ANSYS, Inc. product installation if the installed license manager files are newer than the ones being installed. You should always use the newest files. However, due to system format changes or other unlikely scenarios, the date check could produce incorrect results. To override the date check and force the installation to always install the files from the media, regardless of the file dates, re-run the installation with the `-nodatecheck` option. We strongly recommend that you exercise caution when running the installation with the `-nodatecheck` option; installing older license files can result in licensing errors and the inability to run ANSYS, Inc. products.

### 5.1.8. System-related Error Messages

#### Error, could not open display.

Either the **DISPLAY** environment variable is not correct or the `xhosts` command was not properly set. See the *Basic Analysis Guide* for specific graphics information.

#### \*\*\*Error, ANSYS130\_DIR environment variable is not set. This is a fatal error – exiting.

This message indicates that the **ANSYS130\_DIR** environment variable was not set where necessary for licensing. This environment variable (which is set in the scripts that run ANSYS) should be set to the release-specific installation directory.

### 5.1.9. High Performance Computing Error Messages

The following error messages are associated with the High Performance Computing solvers.

#### mpid: Error: HP MPI version incompatibility detected

You may encounter this or a similar message if you attempt to use ANSYS 13.0 with a different version of MPI than is supported. See the *Distributed ANSYS Guide* for a complete list of supported MPI versions.

## 5.2. Installation Troubleshooting - Mechanical APDL (ANSYS)

The items listed below apply only to the Mechanical APDL (ANSYS) product.

### 5.2.1. Your batch jobs terminate when you log out of a session

On some systems, you may need to use the 'nohup' option to allow batch jobs to continue running after you log out of a session. If you are running via the Launcher, select **Options> Use 'nohup' To Start Batch Runs With Output Sent to 'File Only.'** We do not recommend using this setting on systems that automatically set 'nohup.'

### 5.2.2. Japanese/Chinese characters display in status bar windows on Mechanical APDL (ANSYS) on Red Hat AS 4.0

If you are running the traditional Mechanical APDL (ANSYS) interface and see Japanese/Chinese characters in the status bar windows, you will need to reinstall Red Hat, picking only English. This problem is caused by choosing to install "Everything" during the Red Hat Linux OS setup, thus installing all languages. If you then install Mechanical APDL and do a solve with the traditional interface, you will see Japanese or Chinese text in the solve status window. This bug will not be fixed.



## 5.2.3. Mechanical APDL (ANSYS) Documentation File for User Interface Error Messages

**Missing or erroneous documentation files for user interface. Command ignored.**

Verify that the documentation list file for the user interface exists in the `/ansys_inc/v130/ansys/gui/en-us/UIDL` subdirectory.

```
ls -l /ansys_inc/v130/ansys/gui/en-us/UIDL/menulist130.ans
```

The system should respond with:

```
-rw-r--r-- 1 root 23 Jan  8 11:50 /ansys_inc/v130/ansys/gui
/en-us/UIDL/menulist130.ans
```

Make sure that the pathnames in the `menulist130.ans` file are correct.

## 5.2.4. Launcher Error Messages

Some of the more common error messages follow. See the *ANSYS, Inc. Licensing Guide* for licensing-related launcher messages.

**\*\*\*Cannot create required <profile> file. Therefore, cannot write to profile information during this launcher session.**

If you see this error, you cannot add or modify profile information during this launcher session. Verify that you have write access to the directory and restart the launcher session. Typically, this directory is `C:\Documents and Settings\<user name>\Application Data\Ansys\v130\launcher` on Windows or `~/ansys/v130/launcher` on UNIX/Linux.

## 5.2.5. FORTRAN Runtime Error Messages

The following error messages occur if the user is running Mechanical APDL (ANSYS) in a directory in which the user does not have write permission, or if Mechanical APDL files (i.e., `Jobname.RST`, `Jobname.DB`) exist in the current directory but the user does not have write permissions to the files. The specific messages that appear on each system are shown below.

### 5.2.5.1. HP Series Systems

**\*\* FORTRAN I/O ERROR 908: COULD NOT OPEN FILE SPECIFIED FILE: ftn19, UNIT:19**

**LAST FORMAT:**

### 5.2.5.2. Intel Linux 32 Systems

**forrtl: Permission denied**

**forrtl: severe (9): permission to access file denied, unit 19, file /ansys/user/fort.19**

### 5.2.5.3. Intel Linux 64 Systems

**Input/Output Error 177: Creat Failure**

**In Procedure: fappnd**

**At Line: 72**

**Statement: Formatted WRITE**

**Unit: 19**

#### **5.2.5.4. Intel EM64T Linux x64 Systems**

**forrtl: Permission denied**

**forrtl: severe (9): permission to access file denied, unit 19, file /build/v130/ansys/objs**

#### **5.2.5.5. AMD Opteron Linux x64 Systems**

**\*\*\*ERROR**

**Unable to open file /build/v130/ansys/objs/file.err for WRITE. Check directory and file permissions.**

### **5.3. Installation Troubleshooting - ANSYS Workbench**

#### **5.3.1. Startup or Graphics Problems**

To minimize graphics problems, always verify that you are running the latest graphics drivers provided by your computer's hardware manufacturer.

If you are running ANSYS Workbench on Linux and experience problems at startup or with the GUI or graphics displaying correctly, and you are running in accelerated graphics mode, you may need to relaunch ANSYS Workbench using the `-oglmesa` flag to activate software rendering:

```
runwb2 -oglmesa
```

If ANSYS Workbench detects that graphics problems are causing crashes, it will automatically switch to software rendering. ANSYS Workbench also will use software rendering mode by default when running on a remote display, or on a local display if the hardware does not appear to be accelerated.

To revert to accelerated graphics mode, launch ANSYS Workbench using the `-ogllhw` flag:

```
runwb2 -ogllhw
```

If you are running under Exceed3D, try the following settings if you are having graphics problems:

- Turn off the graphics (hardware) acceleration option in Exceed3D options.
- If graphics acceleration is on, turn on the GLX 1.3 option.

Any version of Exceed that does not have the GLX 1.3 option is unlikely to function correctly with graphics acceleration.

### **5.4. Contacting Technical Support**

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to [www.ansys.com](http://www.ansys.com) and select **Support> Technical Support> Designated Service Providers**. The direct URL is: <http://www1.ansys.com/customer/public/supportlist.asp>. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website ([www.ansys.com](http://www.ansys.com)) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: <http://www.ansys.com/customerportal>.

One of the many useful features of the Customer Portal is the Knowledge Base Search, where you can find solutions to various types of problems, like FAQ. The Knowledge Base Search feature is located under **Online Support> Search Options> Solutions Search**.

Systems and installation Knowledge Resources and FAQs are easily accessible via the Customer Portal under Online Support > [Installation/System FAQs](#). These Knowledge Resources provide a plethora of solutions and direction on how to get installation and licensing issues resolved quickly and efficiently.

## **NORTH AMERICA**

### **All ANSYS, Inc. Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Toll-Free Telephone:** 1.800.711.7199

**Telephone:** 1.724.514.3600

**Fax:** 1.724.514.5096

For installation and licensing questions, visit our Knowledge Resources and FAQs on the [Customer Portal](#). Support for University customers is provided only through the ANSYS Customer Portal.

## **GERMANY**

### **ANSYS Mechanical Products**

**Telephone:** +49 (0) 8092 7005-55

**Fax:** +49 (0) 8092 7005-5

**Email:** [support@cadfem.de](mailto:support@cadfem.de)

### **CFX Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +49 (0) 8024 9054-44

**Fax:** +49 (0) 8024 9054-17

**Email:** [cfx-support-germany@ansys.com](mailto:cfx-support-germany@ansys.com)

### **FLUENT Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +49 (0) 6151 3644-0

**Fax:** +49 (0) 6151 3644-44

**Email:** [fluent-support-germany@ansys.com](mailto:fluent-support-germany@ansys.com)

### **ICEM CFD Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +49 (0) 511 288696-4

**Fax:** +49 (0) 511 288696-66

**Email:** icemcfd-support-germany@ansys.com

## UNITED KINGDOM

### All ANSYS, Inc. Products

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +44 (0) 870 142 0300

**Fax:** +44 (0) 870 142 0302

**Email:** support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

## JAPAN

### CFX, ICEM CFD and Mechanical Products

**Telephone:** +81-3-5324-8333

**Fax:** +81-3-5324-7308

**Email:** *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

### FLUENT Products

**Email:** *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

### Licensing and Installation

**Telephone:** +81-3-5324-7305

**Email:** japan-license-support@ansys.com

## INDIA

### ANSYS Products (including FLUENT, CFX, ICEM-CFD)

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

**Fax:** +91 80 2529 1271

**Email:** *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

## FRANCE

### ANSYS, CFX, FLUENT, and ICEM CFD Products

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +33 (0) 820 480 240

**Email:** *ANSYS:* ansys-support-france@ansys.com; *CFX:* cfx-support-france@ansys.com; *FLUENT:* fluent-support-france@ansys.com; *ICEM CFD:* icemcfd-support-france@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

**BELGIUM****All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +32 (0) 10 45 28 61

**Email:** support-belgium@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

**SWEDEN****All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +44 (0) 870 142 0300

**Email:** support-sweden@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

**SPAIN and PORTUGAL****All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +33 1 30 60 15 63

**Email:** support-spain@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

**ITALY****All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +39 02 89013378

**Email:** support-italy@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.





# **ANSYS, Inc. Installation Guide for Windows**

---



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 13.0  
November 2010  
00XXXX

ANSYS, Inc. is certified to ISO 9001:2008.
--

---

## Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superceded by a newer version of this guide. This guide replaces individual product installation guides from previous releases.

## Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---



# Table of Contents

<b>1. Installation Prerequisites for Windows</b>	1
1.1. System Prerequisites	3
1.1.1. CAD Support	4
1.2. Disk Space and Memory Requirements	5
1.3. Software Prerequisites	5
1.4. GPU Requirements	6
1.5. Additional Hardware and Software Requirements	6
<b>2. Platform Details</b>	9
<b>3. Installing the Software</b>	13
3.1. Pre-Installation Instructions for Download Installations	13
3.1.1. Downloading the Installation Files	13
3.2. Pre-Installation Instructions for DVD Installations	13
3.3. Installing ANSYS, Inc. Products	14
3.3.1. Product Installation	14
3.3.2. License Manager Installation	17
3.3.3. Network Installation and Product Configuration	18
3.3.4. Silent Mode Operations	20
3.3.4.1. Silent Product and License Manager Installation	21
3.3.4.2. Silent Product Configuration/Unconfiguration	24
3.3.4.3. Silent Uninstall	24
3.3.5. Setting the /3GB Switch on Windows 32-bit	25
3.3.6. Registering the License Server	26
3.3.7. SpaceBall Support	26
3.3.7.1. SpaceBall Support for CFX Components	26
3.3.7.2. SpaceBall Support for FLUENT Components	26
3.4. Post-Installation Procedures	27
3.4.1. Post-Installation Procedures for Mechanical APDL (ANSYS) and ANSYS Workbench Products	28
3.4.2. Post-Installation Procedures for ANSYS CFX and ANSYS CFD-Post	29
3.4.3. Post-Installation Procedures for ANSYS AUTODYN	29
3.4.4. Post-Installation Procedures for ANSYS FLUENT	29
3.4.5. Post-Installation Procedures for ANSYS POLYFLOW	30
3.4.6. Post-Installation Procedures for ANSYS ASAS	30
3.4.7. Post-Installation Procedures for Other Products	30
3.5. Launching ANSYS, Inc. Products	30
3.6. Running the ANS_ADMIN Utility for Mechanical APDL (ANSYS) /ANSYS Workbench Products	31
3.7. Product Localization	31
3.7.1. Translated Message File Installation for Mechanical APDL (ANSYS) /ANSYS Workbench Products	31
3.8. Uninstalling ANSYS, Inc. Products	32
3.8.1. Uninstalling Licensing Components	33
<b>4. Configuring CAD Products</b>	35
4.1. Using the CAD Configuration Manager	35
4.1.1. Unconfiguring	38
4.1.2. Running the <b>CAD Configuration Manager</b> in Batch Mode	38
4.1.3. Uninstalling	40
4.1.4. Pro/ENGINEER Configuration	40
4.1.4.1. Configuring the Connection for Pro/ENGINEER	41
4.1.4.1.1. The WBPlugInPE.dat File and config.pro File	41
4.1.4.1.2. The config.pro File	41
4.1.4.1.3. The config.anscon File	41

4.1.4.2. Pro/ENGINEER Environment Variables .....	42
4.1.5. NX Configuration .....	42
4.1.5.1. Configuring the Connection for NX .....	42
4.1.5.2. Configuring for Teamcenter Engineering .....	43
4.1.6. ANSYS ICEM CFD Configuration .....	43
4.2. Configuring CADNexus/CAPRI CAE Gateway for CATIA V5 .....	43
<b>5. Troubleshooting .....</b>	<b>45</b>
5.1. Installation Troubleshooting .....	45
5.1.1. Using ANSLIC_ADMIN to Gather Diagnostic Information .....	45
5.1.2. Uninstall Gives Access Denied Error Message .....	45
5.1.3. Uninstall on Vista Systems Gives Compatibility Error Message .....	46
5.1.4. A .chm File Does Not Display Properly Across a Network .....	46
5.1.5. License Manager Fails to Start on Windows Pentium III Systems .....	46
5.1.6. Products Crash with an Application Error .....	46
5.1.7. Product Installation Does Not Create Start Menu Item for ANSYS and CAD Plugins Do Not Work .....	46
5.1.8. System-related Error Messages .....	46
5.1.9. CasPol Error Messages .....	47
5.2. Installation Troubleshooting - Mechanical APDL (ANSYS) and ANSYS Workbench Products .....	47
5.2.1. The Mechanical APDL Launcher is Excessively Slow to Start .....	47
5.2.2. Display Problems on Windows XP .....	47
5.2.3. ANS_ADMIN Error Messages .....	47
5.2.4. Mechanical APDL Product Launcher Error Messages .....	48
5.2.5. Distributed ANSYS HP-MPI Error Messages .....	48
5.2.6. ANSYS Workbench Products Troubleshooting .....	48
5.3. Installation Troubleshooting - ANSYS CFX .....	49
5.3.1. TurboGrid Mouse Behavior Problems .....	49
5.4. Contacting Technical Support .....	49

## List of Tables

1.1. Supported Platforms .....	3
1.2. Supported Products by Platform .....	3
1.3. CAD Support by Platform .....	4

---

## Chapter 1: Installation Prerequisites for Windows

---

This document describes the steps necessary to correctly install and configure all ANSYS, Inc. products on Windows platforms for Release 13.0. These products include:

- ANSYS Structural Mechanics
  - ANSYS Mechanical Products (includes ANSYS Mechanical and Mechanical APDL)
  - ANSYS Customization Files
- ANSYS Explicit Dynamics
  - ANSYS AUTODYN
  - ANSYS LS-DYNA
- ANSYS Fluid Dynamics
  - ANSYS CFX (includes ANSYS CFD-Post)
  - ANSYS FLUENT (includes ANSYS CFD-Post)
  - ANSYS TurboGrid
  - ANSYS POLYFLOW (includes ANSYS CFD-Post)
  - ANSYS CFD-Post only
- ANSYS Offshore
  - ANSYS ASAS
  - ANSYS AQWA
- ANSYS Additional Tools
  - ANSYS ICEM CFD
    - ANSYS ICEM CFD Pro/ENGINEER Interface
    - ANSYS ICEM CFD NX Interface
  - ANSYS Icepak (includes ANSYS CFD-Post)
- ANSYS PDM Interfaces
  - TeamCenter Engineering
- ANSYS Geometry Interfaces
  - ACIS
  - CATIA, Version 4
  - CATIA, Version 5
  - CoCreate Modeling
  - Autodesk Inventor
  - JTOpen
  - NX

- Parasolid
- Pro/ENGINEER
- Solid Edge
- SolidWorks

---

### Note

ANSYS Workbench and ANSYS EKM Desktop are installed as product components to most ANSYS, Inc. products. ANSYS Workbench is not installed as part of the products under ANSYS Additional Tools, nor with the CFD-Post only option. ANSYS Workbench includes the following applications:

- The Mechanical Application
- DesignModeler
- Design Exploration
- Meshing
- Remote Solve Manager
- TGrid
- FE Modeler
- EKM Desktop

Because some of these applications can be run as standalone applications without ANSYS Workbench, you will see some but not all of these listed separately as options when you uninstall. In such cases, you will need to select those options in order to uninstall those components.

Where supported, IGES and STEP Geometry Interfaces will be installed.

Some procedures apply only to specific products and are so noted.

### Important Notice

If you wish to run multiple releases of ANSYS, Inc. software, you **MUST** install them chronologically (i.e., Release 12.1 followed by Release 13.0). If you install an earlier release after installing Release 13.0, you will encounter licensing issues that may prevent you from running any products/releases. If you need to install an earlier release after you have already installed Release 13.0, you **MUST** uninstall Release 13.0, then re-install the releases in order.

### Summary of New and Changed Features

The following features are new or changed at Release 13.0. Please review these items carefully.

- ANSYS, Inc. products now support Windows 7.
- On Windows systems, the unified installation process now automatically checks for the necessary prerequisites on your system and will install any prerequisites that are missing. You no longer have to choose to install the prerequisites as a separate step.
- The installation and product configuration utilities have been improved.
- Silent mode operations have been extended to include installation, uninstall, product configuration, and product unconfiguration on all platforms.

## 1.1. System Prerequisites

ANSYS 13.0 is supported on the following Windows platforms and operating system levels. For up-to-date information on hardware platforms or operating systems that have been certified, go to <http://www.ansys.com/hardwaresupport>. (This URL is case-sensitive.)

**Table 1.1 Supported Platforms**

Platform	Processor	Operating System	Platform architecture (directory name)	Availability
Windows (64-bit)	x64	Windows XP 64 SP2, Vista 64 SP1, Windows 7, Windows HPC Server 2008 [1]	winx64	Download, DVD
Windows (32-bit)	x86	Windows XP SP2, Vista SP1, Windows 7	intel	Download, DVD

1. Windows HPC Server 2008 is supported only for select solvers for parallel high performance execution.

### Note

Home editions of Windows operating systems are not supported.

If you are running Internet Explorer 8 on Windows XP 32-bit, you must use SP3.

**Supported Platforms for High Performance Computing** Please see the discussions on [Configuring Distributed ANSYS](#) and [Configuring ANSYS CFX Parallel](#) later in this guide for detailed information on supported platforms for distributed computing.

**Table 1.2 Supported Products by Platform**

Product	Windows 64-bit	Windows 32-bit	Windows Server 2008 HPC
ANSYS Workbench data-integrated and native applications	X	X	
<b>Standalone Solvers and Applications</b>			
Mechanical APDL (ANSYS)	X	X	<a href="#">2</a>
CFX	X	X	<a href="#">2</a>
TurboGrid	X	X	
ICEM CFD	X	X	
AUTODYN	X	X	
LS-DYNA	X	X	
FLUENT	X	X	<a href="#">2</a>
ASAS/AQWA	<a href="#">1</a>	X	
POLYFLOW	X	X	
Icepak	X	X	<a href="#">2</a>

Product	Windows 64-bit	Windows 32-bit	Windows Server 2008 HPC
EKM	X	X	

1. Supported as a 32-bit application
2. Solver only support

### 1.1.1.CAD Support

The following CAD and auxiliary programs are supported on the indicated products and platforms (where the CAD product is supported on the noted platforms). Products are:

A = Mechanical APDL (ANSYS)  
W = ANSYS Workbench  
I = ANSYS ICEM CFD standalone

**Table 1.3 CAD Support by Platform**

Product	Windows 64-bit	Windows 32-bit
CATIA 4.x [1]	W, I	A, W, I
CATIA V5 R20	A, W, I	A, W, I
CADNexus/CAPRI CAE Gateway for CATIA V5 R19 [1], [2]	W	W
Parasolid 22.1	A, W, I	A, W, I
Pro/ENGINEER Wildfire 3	A [3], W [3], I [3]	A [3], W [3], I
Pro/ENGINEER Wildfire 4 [6]	A, W, I	A, W, I
Pro/ENGINEER Wildfire 5 [7]	A, W, I	A, W, I
ACIS 20 [4]	A [5], W, I	A [5], W, I
NX 6.0	A [5], W [5], I	A [5], W [5], I
NX 7.0	A [5], W [5]	A [5], W [5]
NX 7.5	A, W	A, W
SolidWorks 2009 [5], 2010 [8]	W, I	W, I
Solid Edge v100 MP9 [5], v102MP4 [5]	W, I	W, I
Autodesk Inventor 2010, 2011	W, I	W, I
IGES 4.0, 5.2, 5.3	W, I	W, I
JTOpen 8.0, 8.1	W	W
STEP AP203, AP214	W, I	W, I
CoCreate Modeling 2008 [5], 17.0	W, I	W, I
DWG	I	I
GEMS	I	I
IDI MS8/9	I	I

1. ANSYS has tested these CAD packages on Windows 7, but the CAD vendor does not claim support for Windows 7.
2. If using CATIA V5 (R18 SP4+), Vista 32-bit and Windows 7 (32- and 64-bit) are not supported.

3. Windows XP only; Vista and Windows 7 are not supported.
4. For ICEM CFD standalone, ACIS 18.0.1 is the supported version for all platforms.
5. Windows 7 is not supported.
6. Requires Pro/ENGINEER Wildfire 4 Build Date M120 for Windows 7.
7. Requires Pro/ENGINEER Wildfire 5 Build Date M020 for Windows 7.
8. Requires SolidWorks 2010 SP1 for Windows 7.

## 1.2. Disk Space and Memory Requirements

You will need the disk space shown here for each product for installation. The numbers listed here are the maximum amount of disk space you will need for each ANSYS, Inc. product. Depending on the options selected for each product, you may require less.

Mechanical APDL (ANSYS): 6.1 GB  
 ANSYS AUTODYN: 3.1 GB  
 ANSYS LS-DYNA: 3.3 GB  
 ANSYS CFX: 3.8 GB  
 ANSYS TurboGrid: 3.1 GB  
 ANSYS FLUENT: 4.1 GB  
 POLYFLOW: 1.4 MB  
 ANSYS ASAS: 2.9 GB  
 ANSYS AQWA: 2.7 GB  
 ANSYS ICEM CFD: 1.4 GB  
 ANSYS Icepak: 1.7 GB  
 ANSYS TGrid: 2.7 GB  
 CFD Post only: 3.1 GB  
 ANSYS Geometry Interfaces: 1 GB  
 CATIA v5: 600 MB

Product installations also require an additional 500 MB of free disk space during an installation.

**Mechanical APDL (ANSYS)/ANSYS Workbench** A minimum of 2 GB of real memory is recommended.

**ANSYS CFX** A minimum of 2 GB of real memory is recommended for 64-bit machines.

**ANSYS FLUENT** A minimum of 2 GB of real memory is required.

**ANSYS POLYFLOW** A minimum of 2 GB of real memory recommended for 32-bit and 4 GB for 64-bit. Additional swap space of 2 GB is needed for 32-bit and 4 GB for 64-bit.

## 1.3. Software Prerequisites

You must have the following software installed on your system. Administrator privileges are required to install these files. These software prerequisites will be installed automatically when you launch the product installation. If you've completed an installation successfully, the executables are also located under the `\v130\prereq` directory should you need to manually install them at a later date. Prior to installing, you can find them in the root `/prereq` directory of the media/download package.

- Microsoft .NET Framework 3.5 SP1, with Update KB959209. Version 3.5 includes versions 2.0 and 3.0, and the following patches:

XP (32-bit)	XP (64-bit)	Vista (32-bit)	Vista (64-bit)
NDP20SP2-KB958481-x86.exe	NDP20SP2-KB958481-x64.exe	--	--
NDP30SP2-KB958483-x86.exe	NDP30SP2-KB958483-x64.exe	--	--
NDP35SP1-KB958484-x86.exe	NDP35SP1-KB958484-x64.exe	NDP35SP1-KB958484-x86.exe	NDP35SP1-KB958484-x64.exe

.NET Framework 3.5 SP1 is native on Windows 7 and does not require any additional patches.

- Microsoft Visual C++ 2005 SP1 Redistributable, 2.0.50727.762
- Microsoft Visual C++ 2008 SP1 ATL Redistributable Package

### **NX 5.0 or 6.0**

You will need one of these versions of NX if you wish to configure and use the Geometry Interface for NX.

### **Pro/ENGINEER Wildfire**

You will need Pro/ENGINEER Wildfire 3, 4, or 5 installed if you wish to configure and use Geometry interface for Pro/Engineer.

## **1.4. GPU Requirements**

Your system must meet the following requirements to use the GPU capability in Mechanical APDL:

The machine being used for the simulation must contain at least one nVIDIA Tesla series GPU card (a Tesla 20-series card is recommended for optimal performance).

On Windows, the driver version for the nVIDIA Tesla series GPU card must be 258.96 or newer.

You must be running on a Windows 64-bit or Linux x64 operating system.

---

### **Note**

On Windows, the use of Remote Desktop disables the use of a GPU, which means the GPU Accelerator Capability cannot function under Remote Desktop. Alternative means of remotely logging into a machine can be used (e.g., third-party software such as VNC or other).

## **1.5. Additional Hardware and Software Requirements**

- Pentium-class or EM64T / AMD64 system with the correct operating system version installed
- 1GB of RAM. On some 32-bit systems, you can use the /3GB switch to allocate additional RAM to the application and less to the Windows kernel. See [Setting the /3GB Switch on Windows 32-bit \(p. 25\)](#) for more information.
- For 32-bit systems, 4 GB free on the hard drive; up to 4.5 GB with CATIA V5. For 64-bit systems, 8 GB is recommended.
- DVD drive
- Certified graphics card with the latest drivers, compatible with the supported operating systems, capable of supporting 1024x768 High Color (32-bit), and a 17-inch monitor compatible with this type of graphics card. We recommend using certified graphics card from the Nvidia Quadro FX or ATI FireGL line.



ANSYS CFX products and ANSYS TurboGrid require at least 24-bit color and that antialiasing on your graphics card be disabled.

---

### Note

How you determine whether antialiasing is enabled varies according to the graphics card you have. To determine the status of an NVIDIA card on Windows, right-click on your desktop and select **NVIDIA Display > monitor type**. In the dialog box that appears, disable **Application-controlled** and slide the **Antialiasing settings** slider to **Off**. Click **OK** to save the changes.

---

### Note

Some combinations of graphics card type, operating system, and MPEG resolution fail to play MPEGs properly. You may be able to get normal playback results simply by changing the MPEG settings. Alternatively, you can upgrade your graphics card.

- For laptop systems, ANSYS Workbench requires the latest graphics drivers.
- Microsoft Mouse or a mouse compatible with the supported operating systems; ANSYS CFX-Pre and ANSYS CFX-Post require a three-button mouse.
- If you use a spaceball, ensure that you have the latest version of the drivers installed.
- Approximately twice as much swap space as memory. The amount of memory swap space on the system may limit the size of the ANSYS model that can be created and/or solved.
- TCP/IP for the license manager (see the *ANSYS, Inc. Licensing Guide* for more information on TCP/IP).

Although TCP/IP is included as part of the operating system, it may not be installed by default. When TCP/IP is installed, it must be bound to a network adapter. On machines that are connected to an internal network, TCP/IP must be bound to a network card such as an Ethernet adapter. The vast majority of systems using TCP/IP will fall into this category. On machines that connect to the internet or corporate intranet through a modem, TCP/IP can be bound to the Remote Access Service. Also, the Autodial option of the Internet Options must be disabled so that the machine does not attempt to connect to the Internet every time ANSYS is run. See the *ANSYS, Inc. Licensing Guide* for more information.

- Microsoft Internet Explorer (IE) 6 or 7 is recommended for proper operation of ANSYS Workbench. Once the correct version of IE is installed, it does not have to be your default Internet browser. After installation, simply run your preferred browser and reestablish it as the default.

If you are running Internet Explorer 8 on Windows XP 32-bit, you must use SP3.

## Notes to Exceed Users

If you run Hummingbird, Communications Ltd.'s Exceed<sup>TM</sup> product, we recommend that you adjust the Window Manager settings in order for the ANSYS Workbench GUI and the online help system to work properly. This setting affects the behavior of Windows, not just applications that are running through Exceed, and may vary depending on the version of Exceed you are running. Note that changing this setting may affect other applications that you run through Exceed; please check with your system administrator before making any changes.

FLUENT or Fluent legacy products do not support running from an Windows machine to a Linux or UNIX machine using Exceed or any remote software.

To ensure that the ANSYS Workbench GUI works properly, follow these steps:

1. Right-click on the **Exceed** icon in the Windows Task Bar.
2. Select **Tools> Configuration...** menu item.
3. Click on the **Display and Video** category.
4. Click on the **Common Settings** tab.
5. Set Native Window Manager Focus Policy to Click.

---

## Chapter 2: Platform Details

---

### Windows 7, Windows Vista

Generally speaking, you need full administrative privileges to install any software on Microsoft Windows 7 and Vista. Non-administrative accounts do not usually have the permissions required to access system areas that installation programs often need to modify. “Full administrative privileges” means that you are running as administrator with UAC turned off, or you are running as administrator with UAC turned on and **Run as Administrator** selected.

Windows 7 and Vista use a feature called User Account Control (UAC) to control privileges and automatically reduce the potential of security breaches in the operation system. However, UAC limits your accessibility to system areas and can cause unpredictable behavior in ANSYS, Inc. products, as noted below.

Full administrative options with Windows 7 and Vista include:

#### Recommended Options:

- UAC turned off and install as full administrator (preferred method)
- UAC turned on and install with full administrative privileges (using **Run as Administrator** from the context menu)

---

#### Note

If you are running with UAC on, even if you have administrative privileges, you will have limited account control unless you specifically choose **Run as Administrator**.

#### Other Options:

- UAC turned off and install as non-administrative user
- UAC turned on and not install with full administrative privileges (you did not select **Run as Administrator** from the context menu)
- UAC turned on and install as non-administrative user

When installing ANSYS, Inc. products, we strongly recommend that you use one of the two recommended options above. If UAC is enabled, we recommend that you turn UAC off before installing ANSYS, Inc. products, and then turn it back on when finished installing. You must be an administrative user in order to disable UAC.

Follow the instructions for your operating system for disabling UAC.

Once you have installed your applications, you can enable UAC again.

If you have upgraded to Vista from Windows XP, the TEMP environment variable may point to an inaccessible location. For example, it may point to %USERPROFILE%\Local Settings\Temp. As a result, the product installation could end without configuring the product correctly. You should ensure that the TEMP user variable points to a valid location, such as %USERPROFILE%\AppData\Local\Temp.

## Installing with UAC On

If you cannot disable UAC and need to install with UAC on, you must run all applications, including the installation setup, as an administrator. Otherwise, the files may be written to `AppData\Local\VirtualStore\<path>` (or similar) rather than directly to `<path>`, if `<path>` is a protected directory (such as `C:\Program Files`). As a result, some applications and utilities may behave unpredictably. If you cannot disable UAC, you should:

- Always run **ANSLIC\_ADMIN**, the **File Association** utility, and the **CAD Configuration Manager** by selecting the utility or application from the **Start** menu and right-mouse clicking and selecting **Run as Administrator**.
- Set your Start menu shortcuts to always run as administrator with the following procedure (or similar):
  1. Right-click the shortcut you wish to modify and select **Properties**.
  2. Click **Advanced** on the **Shortcut** tab.
  3. Select **Run as administrator** and click **OK**.
  4. Click **OK**.

Refer to your operating system documentation or visit [www.microsoft.com](http://www.microsoft.com) for more instructions on setting **Run as Administrator** at various levels within the operating system.

## Notes about UAC

When installing ANSYS, Inc. products, including the license manager, on Windows 7 or Vista machines, you need to be aware of several factors involving User Access Control (UAC):

You should install the ANSYS, Inc. License Manager using the same permissions as you used to install the product.

If you install both the product and the license manager with full administrative privileges, the product and licensing installation and configuration will proceed as normal.

If you *install* the product as a non-administrative user with UAC turned off, you will see some expected restrictions (i.e., you will not be able to install prerequisites, you may need to run some additional post-installation product configuration steps, etc.). Likewise, if you *run* as a non-administrative user with UAC turned off, you will also see expected restrictions (i.e., you will not be able to start and stop the license manager.)

If you install as an administrative user with UAC turned on, you may encounter unpredictable behavior with both the product and license manager installations and subsequent behavior. As an administrator, you should choose to install and run applications with full administrative privileges to avoid any unpredictable situations. We strongly recommend against running with UAC on and not selecting **Run as Administrator**. These situations are described below.

1. If you have UAC turned on and you do not install (and run) with full administrative privileges, be aware that files may be written to and read from the `%localappdata%\VirtualStore\` location (in the case of licensing, to `%localappdata%\VirtualStore\Program Files\Ansys Inc\Shared Files\Licensing`) instead of to the `<os drive>`.

In this situation, **ANSLIC\_ADMIN** settings, such as **Specify the License Server Machine**, will indicate that files are being written to and read from `<os drive>\Program Files\Ansys Inc\Shared Files\Licensing`. Log files may behave in the same way. However, when you view the an-

syslmd.ini file located in <os drive>\Program Files\Ansys Inc\Shared Files\Licensing, you will not see the **ANSLIC\_ADMIN** settings you specified.

2. To ensure that files are correctly written to and read from <os drive>\Program Files\Ansys Inc\Shared Files\Licensing, you should always run with full administrative privileges.
3. If you are not running with full administrative privileges, you will not be able to use those **ANSLIC\_ADMIN** options that require administrative privileges, such as starting and stopping the license manager.

## Compiler Requirements for Windows Systems

All ANSYS, Inc. products are built and tested using the Visual Studio 2008 SP1 (including the MS C++ compiler) and Intel FORTRAN 11.1 compilers. Compilers are required only if you will be using User Programmable Features or other customization options.



---

## Chapter 3: Installing the Software

---

### 3.1. Pre-Installation Instructions for Download Installations

We strongly recommend that you review the **Read Download Instructions and Product Information** file included on the download site for the most current download instructions. We also recommend that you review the **What Should I Download?** file to understand which package(s) you need to download, depending on which product(s) you purchased and wish to run.

#### 3.1.1. Downloading the Installation Files

To download the installation files from our website, you will need to be a TECS customer.

Depending on the product/platform combination(s) you choose, you may need to download multiple files.

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.

### 3.2. Pre-Installation Instructions for DVD Installations

To install ANSYS, Inc. products from the installation media, place the DVD in your DVD drive. If autorun is enabled, the installation will begin automatically. If autorun is disabled, choose **Start>Run** and type **D:\setup.exe** (replace D with the letter of your DVD drive) and continue with the steps in *Installing ANSYS, Inc. Products* (p. 14).

If you are installing from media, you will be prompted to change DVDs during the installation. Please make sure you have all installation DVDs before beginning the installation.

## 3.3. Installing ANSYS, Inc. Products

This section explains how to install ANSYS, Inc. products.

Although administrative privileges are not required for installation, we recommend that you use a login with administrative privileges. On Windows 7 and Vista systems, we prefer that you run as an administrator with UAC turned off. If you run as a non-administrative user on Windows 7 or Vista systems, you must have UAC turned off. See [Platform Details \(p. 9\)](#) for details on UAC. Some product components, including CAD programs, require administrative permissions to register. If you install with non-administrative privileges, you must follow the post-installation procedures as an administrator for your product to ensure that all components are successfully registered.

### 3.3.1. Product Installation

To install ANSYS, Inc. products, including client licensing, follow the steps below. You must also install the ANSYS, Inc. License Manager on at least one server machine in order to run ANSYS, Inc. products. See [License Manager Installation \(p. 17\)](#) for license manager installation instructions.

1. Save all data and close all Windows applications before continuing.
2. To install, double-click the `setup.exe` file.
3. The installation launcher appears. Choose a language. You will also see the following options:
  - Install ANSYS, Inc. Products
  - Install MPI for FLUENT and Distributed Mechanical APDL (ANSYS)
  - Install ANSYS, Inc. License Manager
  - Exit
4. If you will be running FLUENT Parallel or Distributed ANSYS and need to install HP-MPI or Intel MPI, choose **Install MPI for FLUENT Parallel and Distributed Mechanical APDL (ANSYS)**. On the following screen, choose to install HP-MPI (Distributed Mechanical APDL only) or Intel MPI (FLUENT Parallel and Distributed Mechanical APDL). The appropriate installation program will start. An installation README file will open simultaneously. Follow the instructions in the README file as you complete the installation.
5. After the installation of HP-MPI or Intel MPI has completed (if selected), choose **Install ANSYS, Inc. Products** from the installation launcher. During the installation process, the installation program will check your system to determine if you have the necessary prerequisites and will install any necessary prerequisites. You will need administrative privileges during the installation to install prerequisites. If you want to install as a non-administrative user but require prerequisites, you can install the prerequisites separately by running `InstallPreReqs.exe` from the top level directory as an administrator. Then you can log back in as a non-administrative user to continue with the installation.
6. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
7. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. products. You can choose as many platforms as you wish. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window.

When you run a platform installation (i.e., install another platform's version onto the current system), ANSYS installs all necessary files. You can then mount to that installation from the appropriate platform. However, you may still need to run platform-specific configuration procedures (such as using the **CAD**



**Configuration Manager**) from that platform before running an ANSYS, Inc. product. See [Network Installation and Product Configuration](#) (p. 18) for more information.

Click **Next** to continue.

8. Specify the installation directory. You can accept the default or specify an alternate path and the directory name where the products are to be installed. When you choose an install directory via the Browse feature, the installation will automatically append `\ANSYS Inc\` to the chosen directory. The installation path can have a maximum of 100 characters, except on AUTODYN and ASAS, which have a maximum of 40 characters. You should install all ANSYS, Inc. products into the same location. Installing products into different locations can cause product components to fail.

If you have already installed the ANSYS Workbench Framework for Ansoft, you must install any additional ANSYS, Inc. products into the same directory.

You can also choose to have file extensions associated with the appropriate ANSYS program. Associating file extensions allows you to open the files with the appropriate program by double-clicking the file in Windows Explorer. Otherwise, the files must be opened through the program with which they are associated. File association occurs only for those products that you install. If a product is not installed, the file association for that product's filename extension(s) will not be created. If you choose not to associate file extensions during the installation, you can do so later by running the File Association utility (**Start> Programs> ANSYS 13.0> Utilities> File Association**). Administrative privileges are required for file association; this option will not be available if you do not have administrative privileges.

Choose **Disable RSS** to disable automatic internet feeds to ANSYS, Inc. products.

Click **Next** to continue.

9. Select the components you want to install. You can select as many components as you wish to install. The amount of disk space required for the selected components and the disk space available appears at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. However, if you choose to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully. Note that an additional 500 MB of free disk space must be available during the installation process.

The components listed represent all available components for the platforms you selected earlier. Not all components are available on all platforms or with all products.

ICEM CFD: The CATIA v5 interface and the native ANSYS Workbench formats, such as `.mechdat` or `.meshdat` are only supported with the ANSYS Workbench interface. (Legacy formats such as `.agdb`, `.dsdb` and `.cmdb` are also supported.) To have access to those formats from within the ICEM CFD interface, you must install a product that includes ANSYS Workbench along with ICEM CFD.

Click **Next** to continue.

10. If you selected Pro/ENGINEER, you may need to specify the Pro/ENGINEER language, the Pro/ENGINEER command, and the full Pro/ENGINEER installation path (`C:\Program Files\proeWildfire 3` (or `4`) by default) for an existing Pro/ENGINEER installation. Click **Next**.

If you selected NX and the UGII environment variables were not set, you may need to specify the NX installation path for an existing NX installation. If you are an administrative user, a file required to load the NX plug-in is placed in the Administrative user's Application Data folder by default, which may not be accessible by other, non-administrative users. To allow non-administrative users to run, you will need to define the environment variable **UGII\_CUSTOM\_DIRECTORY\_FILE** prior to installation

and specify a location where other users have read access. Alternatively, you can run the **CAD Configuration Manager** after the installation and provide an updated location for the NX Custom Directory File Path under the **NX** tab.

If you selected CATIA v5 and any product that includes ANSYS Workbench, you will be required to select the appropriate CAD plugin for ANSYS Workbench. You can choose to configure Spatial, Capri, or skip and configure later (using the **CAD Configuration Manager**).

Click **Next**.

If you have not yet installed these programs, or do not know the requested information, you can choose to skip these configuration steps. If you skip these steps, you will need to manually configure Pro/ENGINEER, NX, and CATIA v5 using the **CAD Configuration Manager** before you can successfully import Pro/ENGINEER, NX, or CATIA v5 models into Mechanical APDL (ANSYS), ANSYS Workbench, or ICEM CFD. You can access the **CAD Configuration Manager** via **Start > Programs > ANSYS 13.0 > Utilities > CAD Configuration Manager**. See [Configuring CAD Products \(p. 35\)](#) for more information on using the **CAD Configuration Manager**.

11. A licensing file date verification summary appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.
12. A summary of the selected installation data appears. Information shown includes platform, installation directory, and products. Review the information carefully, and if correct, click **Next** to continue the installation.

The installation program will first check your system to determine if you have the necessary prerequisites. Installing prerequisites requires administrator privileges; if you do not have administrative privileges, you will not be able to continue. You will be asked to log out as a non-administrative user and log in as an administrative user to install the prerequisites. If you want to install as a non-administrative user but require prerequisites, you can install the prerequisites by running `InstallPreReqs.exe` from the top level directory as an administrator. Then you can log back in as a non-administrative user to continue with the installation.

If the prerequisites are not already on your system, and you have administrative privileges, the prerequisites will be installed automatically. In this case, you will see messages in the installation window indicating that prerequisites are being installed. You may be asked to run the necessary files (such as the .NET exe), or to reboot your machine, depending on your existing machine configuration. If you are asked, you will need to reboot in order to complete the prerequisite installation successfully.

The selected products and components are then installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the licensing client.

13. The **Licensing Client Installation Configuration** box appears. As the licensing client is installed, progress messages appear in the box.
14. If you do not have an existing `ansyslmd.ini` file, the **Specify License Server Machine - Add Server Specification** box appears. Enter the hostname of your license server machine and click **Next**.

If you already have an existing `ansyslmd.ini` file, you will not see this box and you will proceed directly to the next step.

15. When the client installation is complete, click **Exit**.

16. On the product installation window, click **Next**. You will be asked to participate in an Install Survey. To take the survey, enter the path to a valid browser for your system, and click **Next**. You can also click **Finish** now to skip the survey or when you have completed the survey.
17. Click **Exit** on the installation launcher, or choose **Install ANSYS, Inc. license manager** to go on to the License Manager Installation. You do not have to take any further steps to run as a client; however, each site must install the License Manager on at least one machine that will act as a license server.

If you installed as a non-administrative user, you will need to run additional post-installation steps as an administrative user to finish configuring CFD Viewer, BladeGen, and most CAD products. See [Post-Installation Procedures \(p. 27\)](#) for specific procedures for each product.

### 3.3.2. License Manager Installation

Follow the instructions below to install the ANSYS License Manager on Windows systems that will act as license servers. You must be an administrative user to install the ANSYS License Manager on Windows. Client licensing is installed automatically when the product is installed; you do not have to take any further steps to run as a client if you have installed a product.

---

#### Note

You do not have to install the prerequisites to install the ANSYS License Manager.

1. To install, double-click the `setup.exe` file.
2. The installation launcher appears. Choose a language. You will also see the following options:
  - Install ANSYS, Inc. Products
  - Install MPI for FLUENT and Distributed Mechanical APDL (ANSYS)
  - Install ANSYS, Inc. License Manager
  - Exit
3. Choose to **Install ANSYS, Inc. License Manager** from the installation launcher. You must be an administrative user to install the license manager; this option will not be available if you are not an administrative user.
4. You will be notified that the license manager, if running, will be shut down. Click **OK**.
5. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
6. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. License Manager. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. Click **Next** to continue.
7. The installation directory is shown; however, you cannot change the default or specify an alternate path or directory name where the license manager is to be installed.

Click **Next** to continue.

8. Select the component you want to install. The amount of disk space required and the disk space available appear at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. However, if you choose

to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully.

Click **Next** to continue.

9. A licensing file date verification summary appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.
10. A summary of the selected installation data appears. Information shown includes platform, installation directory, and product. Review the information carefully, and if correct, click **Next** to continue the installation.

The ANSYS License Manager is now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the license manager.

11. The **Licensing Server Installation Configuration** box appears. As the license manager is installed, progress messages appear in the box.
12. The License Wizard will be launched. This wizard walks you through the process of installing or updating a license file, specifying the license server(s) (which updates the `ansyslmd.ini` file), and starting the license manager. The wizard will prompt you for the necessary information at each step. During this process, the license manager will be shut down if it is running. Be aware that this can impact any users currently running using the license manager.

Click **Continue** on the License Wizard to begin, and follow the instructions on the screen. If you choose either of the non-default server configurations, please see [Advanced Licensing Configuration Options](#) in the *ANSYS, Inc. Licensing Guide* for detailed installation instructions.

13. When the License Wizard is complete, click **Exit** on the wizard screen and then click **Exit** again on the Licensing Installation Configuration Log screen.
14. When the license manager installation is complete, click **Finish**. A new Start Menu item named **ANSYS, Inc. License Manager** will be created automatically. It will include selections for the server **ANSLIC\_ADMIN** utility, the *ANSYS, Inc. Licensing Guide*, and the *FLEXnet Licensing End User Guide*.

---

### Note

Client machines will also include a Client **ANSLIC\_ADMIN** option under the ANSYS Release 13.0 Start Menu item. The client-only version of **ANSLIC\_ADMIN** offers access to a limited number of **ANSLIC\_ADMIN** features, such as status/reporting options.

## 3.3.3. Network Installation and Product Configuration

The procedure for a network, file server, or platform installation follows. Running AUTODYN from a file server is not recommended. When performing a network installation, you must install the products directly on the machine that will act as the server; you cannot install from a remote machine to the server.

To complete a network installation (where the product is installed on one machine and multiple clients access that installation to run the product) to a file server machine, follow the steps below. A network installation can be either homogenous or heterogeneous:

- Homogenous network: all clients that will access this installation are of the same Windows platform type.

- Heterogeneous network: multiple Windows platforms (i.e., 32- and 64-bit) will be installed on a single Windows file server, and clients of those Windows platform types will access the product installation on that single Windows file server.
1. Install the product(s) to be shared in a location that all clients can access. For heterogeneous network installations, install all platforms that this machine will host.

You must share the entire `\Ansys Inc` directory, not just the `\v130` directory. If mapping a drive to the shared location, it must be mapped to the `\Ansys Inc` folder.

2. Open `\\fileservermachine\Ansys Inc\v130\commonfiles\tools\<platform>\ProductConfig.exe`. Select **Install Required Prerequisites**. You must be logged in as an administrative user to install the prerequisites. This step must be completed on all client Windows systems.
3. If you are sharing the Mechanical APDL (ANSYS) product, edit the `tlbrlist130.ans` file, located in `\v130\ansys\gui\en-us\toolbars` on the Windows file server. The file should read as follows:

```
\\fileservermachine\Ansys Inc\v130\ansys\gui\en-us\toolbars\ANSYSSTANDARD.TLB
\\fileservermachine\Ansys Inc\v130\ansys\gui\en-us\toolbars\ANSYSABBR.TLB
\\fileservermachine\Ansys Inc\v130\ansys\gui\en-us\toolbars\ANSYSGRAPHICAL.TLB
```

4. On each Windows client, update the security settings. By default, the .NET Framework prevents applications from being run over a network. In order to install over a network, this restriction needs to be modified. You can allow network execution by running the Microsoft Caspol utility from the command line.

The Caspol utility is located under the .NET Framework installation, in `C:\Windows\Microsoft.NET\Framework\v2.0.50727` for a 32-bit machine. In this example, full trust is opened to files on a mapped drive, enabling software to run from that share:

```
C:\Windows\Microsoft.NET\Framework\v2.0.50727\Caspol.exe -m -ag <CodeGroup> -url
file://x:/* FullTrust
```

where *x* is the mapped drive or UNC path where the product is installed, and *CodeGroup* is the appropriate Caspol permissions level. We recommend a *CodeGroup* setting of 1.2. If that setting does not work or is not appropriate for your environment, please see the Caspol documentation available from Microsoft or contact your system administrator.

On a 64-bit machine, use the following command:

```
C:\Windows\Microsoft.NET\Framework64\v2.0.50727\Caspol.exe -m -ag <CodeGroup> -url
file://x:/* FullTrust
```

To enter this command, choose **Start> Run**, and enter the above command in the **Open:** field. Replace *CodeGroup* and *x:* in the above command with the correct values and press **Enter**.

You must install Caspol as an administrative user. If running this from the command line in Windows 7 or Vista, you must start the command prompt as Administrator, or the command will fail.

5. If you are installing multiple platforms, you must go to one client of each platform type and run the **Complete Unfinished Licensing Installation Configuration** from the **ANSLIC\_ADMIN** utility's **Tools** menu. Because Windows machines will not yet have the shortcuts, you will need to run the client **ANSLIC\_ADMIN** utility manually by issuing the following command:

```
<os drive>:\Program Files\Ansys Inc\Shared Files\Licensing\<platform>
\anslic_admin.exe -client
```

6. For all types of network installations, from each client Windows machine, navigate to the `v130\commonfiles\tools\<platform>` directory and run `ProductConfig.exe`.

7. Select **Configure Products**.
8. Select the products you want to configure. Click **Configure**.

---

### Note

CFD-Post is automatically configured when CFX, FLUENT, POLYFLOW, or Icepak is configured, or if CFD-Post is selected.

9. An informational message box appears as each product is configured.
10. Select **Configure CADs**.
11. The **CAD Configuration Manager** opens. Run this utility for all of the CAD products you need to configure. See [Using the CAD Configuration Manager](#) for detailed instructions on running this utility, or select **Help** from the menu bar in the **CAD Configuration Manager**.
12. When the configuration is completed, you will see shortcuts to each product under **Start> Programs> ANSYS 13.0**.

To unconfigure products that have previously been shared, use the following procedure:

1. From each client machine, select **Start> Programs> ANSYS 13.0> Utilities> Product Configuration**.
2. Select **Unconfigure Products**.
3. Select the products you want to unconfigure. Click **Unconfigure**. When unconfiguring products, be aware that WB/Common should be unconfigured last, after all other products are unconfigured. If you are unconfiguring some but not all products, do NOT unconfigure WB/Common, or your remaining products will not run correctly.

**CFD-Post** CFD-Post can be configured as a standalone product; however, as a standalone product, it is also automatically configured with FLUENT, POLYFLOW, and Icepak. To unconfigure CFD-Post, you must select all of these products to unconfigure, regardless if they are configured locally.

- If you choose to unconfigure standalone CFD-Post without unconfiguring the other products, you will not be able to proceed with the unconfigure. You will need to revise your selections before continuing.
  - If you choose to unconfigure all of the other products that include CFD-Post but you do not select CFD-Post, then CFD-Post will not be unconfigured.
4. An informational message box appears as each product is unconfigured. The shortcuts to each product under **Start> Programs> ANSYS 13.0** are now removed.

## 3.3.4. Silent Mode Operations

ANSYS, Inc. supports silent mode operations, including installation, product configuration/unconfig, and uninstall.

You can specify the following product flags. These flags are all valid for a silent install. However, because of the way the products are packaged, not all of these flags may be valid for a silent configuration/unconfig or uninstall. Not all products are available on all platforms.

For silent mode operations, if you do not specify any product arguments, all available products will be installed, configured, or uninstalled.

## Product Flags

Product	<i>product_flag</i>
Mechanical APDL (ANSYS)	-mechapdl
ANSYS Customization Files	-ansyscust
ANSYS AUTODYN	-autodyn
ANSYS LS-DYNA	-lsdyna
ANSYS CFD-Post	-cfdpost
ANSYS CFX	-cfx
ANSYS TurboGrid	-turbogrid
ANSYS FLUENT	-fluent
ANSYS POLYFLOW	-polyflow
ANSYS ASAS	-asas
ANSYS AQWA	-aqwa
ANSYS ICEM CFD	-icemcfd

*Note: Installing any of the above products will install ANSYS Workbench.*

ANSYS ICEM CFD Pro/ENGINEER Interface	-icemproe
ANSYS ICEM CFD Unigraphics NX Interface	-icemug
ANSYS Icepak	-icepak
ANSYS TeamCenter Engineering	-tceng
Pro/ENGINEER	-proe
NX	-ug
CATIA 4.x	-catia4
CATIA V5	-catia5
Parasolid	-parasolid
ACIS	-acis
SolidWorks	-solidworks
Solid Edge	-solidedge
Autodesk Inventor	-adinventor
CoCreate Modeling	-cocreate

### 3.3.4.1. Silent Product and License Manager Installation

You can deploy an ANSYS, Inc. product installation in silent mode. The general form to run a silent product installation, including the client licensing, from the **Start>Run** menu is:

```
setup.exe -silent -product_flag
```

The above form will install the products specified (see the list of *product\_flags* above). Additional command line arguments are available; please see the list below.

For example, to install TurboGrid and Icepak to the default installation directory, issue the following command:



```
setup.exe -silent -install_dir "C:\Program Files\ANSYS Inc" -turbogrid -icepak
```

To install the ANSYS License Manager on Windows systems that will act as license servers, you must run the setupLM.exe:

```
setupLM.exe -silent
```

The silent license manager installation is valid only for the default Licensing Configuration option "Run the ANSYS Licensing Interconnect with FLEXlm." Please see the *ANSYS, Inc. Licensing Guide* for more information.

You can use the following arguments when running a silent installation:

-silent	Initiates a silent installation.
-install_dir "path"	Specifies the directory to which the product is to be installed. For the installation directory, you must enclose the path in quotes if you have spaces in the pathname. If you want the product to install to the default location, you can omit the -install_dir argument. The default location is <os drive>:\Program Files\ANSYS Inc\. For the license manager installation, it will always be installed to <os drive>:\Program Files\ANSYS Inc\.
-product_flag	Specifies one or more products to install. If you omit the -product_flag argument, all products will be installed. See the list of valid product_flags above.
-productfile "path"	You can specify an options file that lists the products you want to install. To do so, you must provide a full path to a file containing desired products. See <a href="#">Specifying Products with an Options File</a> for more details.
-disablerss	Disables automatic internet feeds to ANSYS, Inc. products.
-licfilepath "path"	Specifies the location of the license file to install. If the path is not specified or if the path is the same as the existing license file, the license file will not be installed. Valid only when doing a silent license manager installation (setupLM.exe).
-licserverinfo	<p>Specifies information to be used by the client for the license server. Valid only in conjunction with a silent installation (setup.exe). The format is:</p> <p>Single license server:</p> <p><i>LI port number:FLEXlm port number:hostname</i></p> <p>Example:</p> <p>2325:1055:abc</p> <p>Three license servers:</p> <p><i>LI port number:FLEXlm port number:hostname,hostname,hostname</i></p> <p>Example:</p> <p>"2325:1055:abc,def,xyz"</p> <p>The default values for the Licensing Interconnect and FLEXlm port numbers (2325 and 1055, respectively) will be used if they are not specified. However, you do need to include the colons. In a three-server environment, you also need to enclose the values in quotes (Windows only).</p> <p>Example:</p>



	<pre> ::abc  or  "::abc,def,xyz"  Information specified via -licserverinfo will be appended to existing information in the ansyslmd.ini file. To change information already in your ansyslmd.ini file, you must use the <b>ANSLIC_ADMIN</b> utility. </pre>
--	---

The silent installation process will automatically close certain applications, possibly resulting in lost data. You should always close all programs before starting a silent install. A silent install will first install any necessary prerequisites that are not already on your system and then continue with the installation. You must have administrative privileges to install prerequisites; if not, the silent install will exit and write a message to the log file.

Installation log files are located in the installation directory: `install.log` contains installation messages, and `install_licconfig.log` contains licensing installation messages. In rare circumstances with a silent licensing installation, the licensing installation messages may not be written to the `install_licconfig.log` (for example, if the silent licensing installation aborts); in these cases, you may find error messages in the `.ansys_install_temp_licconfig_<user>_<index>.log` file, located in `%TEMP%`.

### Caution

A silent license manager installation could shut down the ANSYS, Inc. License Manager, affecting other users who are using that license server machine.

For more information on the silent license manager installation, see the *ANSYS, Inc. Licensing Guide*.

## Specifying Products with an Options File

You can also specify an options file on the command line using the `-productfile "path"` option. The options file can specify which products you want to install. The options file can contain all possible products, with the products you do not want to install commented out, or it can contain only the products you want to install. An example options file is shown below. In the example, Pro/ENGINEER and NX are commented out using the acceptable comment indicators. When using the options file, do not include the dash (-) before the product name.

```

mechapdl
ansyscust
autodyn
cfdpst
cfx
turbogrid
fluent
polyflow
icemproe
icemug
icepak
::proe
REM ug

```

To run with NX and Pro/ENGINEER:

- NX: Verify that your UGII environment variables are set, and the configuration will pick them up automatically.

- Pro/ENGINEER: Verify that the following environment variables are set:

**PROE\_START\_CMD130**=C:\Program Files\<ProE\_dir>\bin\proe1.bat (or your proe start command)

**PROLOADPOINT**= C:\Program Files\<ProE\_dir>

### 3.3.4.2. Silent Product Configuration/Unconfiguration

You can also run the `ProductConfig.exe` via command line (i.e., silent mode) to configure or unconfigure products.

From each client machine, run the `ProductConfig.exe` with the `-silent -config` (or `-unconfig`) options:

```
\\servermachine\ANSYS Inc\v130
\commonfiles\tools\<platform>\ProductConfig.exe
-silent -(un)config
```

You must use the `-config` or the `-unconfig` option with the `-silent` option. Use the `-product_flag` argument to specify which products should be configured; see the above list of `product_flags`. If you do not specify one or more products, all products that have been installed will be configured.

Use the `-help` option for a list of commands available for use with `ProductConfig.exe`.

You will still need to run all of the necessary prerequisite steps (including installing all prerequisites and updating the security settings). You can install the prerequisites silently by issuing the `-prereqs` argument. This argument will install any necessary prerequisites that are not already on your system.

An example entry in a batch file to silently install and configure might look like this:

```
CALL "\\machineabc\ANSYS Inc\v130\commonfiles\tools\winx64\
ProductConfig.exe" -silent -prereqs
CALL C:\Windows\Microsoft.NET\Framework64\v2.0.50727\Caspol.exe
-m -pp off -ag 1.2 -url file://"\\machineabc\ANSYS Inc\*" FullTrust
CALL "\\machineabc\ANSYS Inc\v130\commonfiles\tools\winx64\
ProductConfig.exe" -silent -config
```

Note that the above example includes an option to run the Caspol executable silently (`-pp off`).

If errors are encountered while using the silent product configuration/unconfiguration option, a log file (`productConfig.err`) will be written in the `%TEMP%` directory of the client machine. No log files are written to the machine that is hosting `ProductConfig.exe`.

### 3.3.4.3. Silent Uninstall

To run a silent uninstall, run the following from some location other than the `\v130` directory:

```
C:\> <installation path>\v130\Uninstall.exe -silent
```

The silent uninstall will unconfigure and remove all products and remove the entire `\v130` directory, including any user files or subdirectories that you have added to the `\v130` directory.

Because of the way ANSYS, Inc. products are packaged, not all of the `product_flags` shown above are valid. To uninstall individual products, use the following product options in conjunction with the `-silent` argument:

Mechanical APDL	-mechapdl
ANSYS AQWA	-aqwa

ANSYS ASAS	-asas
ANSYS CFX	-cfx
ANSYS CFD-Post	-cfdpost
ANSYS FLUENT	-fluent
ANSYS POLYFLOW	-polyflow
ANSYS ICEM CFD	-icemcfd
ANSYS TurboGrid	-turbogrid
ANSYS Icepak	-icepak
Remote Solve Manager	-rsm

For example, to uninstall only TurboGrid and Icepak, issue the following command:

```
C:\> <installation_path>\v130\Uninstall.exe -silent -turbogrid -icepak
```

**Uninstalling CFD-Post** CFD-Post can be installed as a standalone product. As a standalone product, it is also automatically installed with FLUENT, POLYFLOW, and Icepak. To uninstall the standalone version of CFD-Post, you must specify `-cfdpost` as well as all of the above products that you have installed. If you choose to uninstall CFD-Post without uninstalling the other products, the uninstall will stop and an error message will be written to the log file (`uninstall.err`). You will need to reissue your uninstall command with the correct product selections.

Note that CFD-Post is also installed with CFX; however, you do not need to uninstall CFX to uninstall the standalone version of CFD-Post, and uninstalling the standalone version of CFD-Post will not remove the CFD-Post capability from CFX.

**Uninstalling a Non-Administrative Installation** If an administrative user attempts to uninstall a non-administrative user's installation, the uninstall will write a log file and terminate. The log file, `uninstall.err`, will be located in the `\v130` directory. If you wish to proceed with the uninstall, issue the following command:

```
<installation_path>\v130\Uninstall.exe -silent -forceNonAdmin
```

### 3.3.5. Setting the /3GB Switch on Windows 32-bit

On Windows 32-bit systems, you can allocate additional memory to an application (such as Mechanical APDL (ANSYS)) via the /3GB switch. Normally, Windows allocates 2 GB of memory for the Windows kernel and 2 GB for the application. When /3GB is enabled, the OS reduces the potential memory allocated to the kernel and makes 3 GB of memory available to the ANSYS, Inc. product.

Be aware that when using the /3GB switch with Mechanical APDL, you will not be able to specify an Mechanical APDL `-m` setting any higher than was possible without this switch. However, with this switch enabled, Mechanical APDL can dynamically allocate more memory; dynamic memory allocation is the recommended memory allocation method.

**Windows XP** To enable this memory allocation on Windows XP systems, add the /3GB parameter to the startup line of the `boot.ini` file and reboot the system. The `boot.ini` file may be hidden. To view hidden files, go to Windows Explorer or My Computer and select **Tools>Folder Options>View**. Under **Hidden files and folders**, check **Show hidden files and folders**.

**Windows 7 and Vista** To enable this memory allocation on Windows 7 and Vista systems, open a command prompt window and type the following command:

```
bcdedit /set IncreaseUserVA 3072
```

Then reboot.

**Windows 64-bit** Windows 64-bit systems already have the capability to address greater amounts of memory, so the /3GB switch is not applicable to those systems.

### 3.3.6. Registering the License Server

Follow this procedure to register your license server information if you are a new user or if you add or change a license server machine. See the *ANSYS, Inc. Licensing Guide* for more information on selecting license servers and on using the **ANSLIC\_ADMIN** utility.

1. Follow the instructions in *Installing ANSYS, Inc. Products* (p. 14) to install the appropriate product(s) on the machine(s) designated as license servers.
2. Use the **ANSLIC\_ADMIN** utility to register license server information on each license server.
3. Return the resulting file to your ANSYS sales representative. ANSYS, Inc. or your sales representative will then supply you with your license keys. This file is located in Program Files\Ansys Inc\Shared Files\Licensing\licserver.info by default.
4. Use the **Run the License Wizard** option of the **ANSLIC\_ADMIN** utility to enter your license key and start the license manager.

### 3.3.7. SpaceBall Support

DesignModeler, Mechanical, Design Exploration, FE Modeler, and ANSYS ICEM CFD all support the 3Dconnexion SpaceBall® for manipulating the model (turning, zooming, panning, etc.). While the SpaceBall does provide significantly enhanced 3D control over viewing the model, it does not function as a mouse replacement and selection operations still require a mouse.

To use a SpaceBall with the listed products, you must download and install the appropriate Microsoft Windows® driver for your version of Windows. These are kept in the download area of the <http://www.3dconnexion.com> web site. At the time of publication of this document, the exact link was <http://www.3dconnexion.com/download.asp> and the file name for 32-bit versions of Windows was 3DxSoftware\_v1-1-0\_win32.exe; however, this is subject to change by 3Dconnexion.

#### 3.3.7.1. SpaceBall Support for CFX Components

SpaceBall is unsupported for CFX components on Windows 64 for Release 13.0.

On 32-bit Windows, you must also uninstall 3DOffice in order to use SpaceBall.

In all cases, it is important that you have the latest SpaceBall drivers installed when using a SpaceBall with ANSYS CFX components.

---

#### Note

No SpaceBall button functions are supported in ANSYS CFX or CFD-Post. However, you can record custom functions to SpaceBall buttons to perform actions in the Viewer.

#### 3.3.7.2. SpaceBall Support for FLUENT Components

FLUENT products do not support SpaceBall.

## 3.4. Post-Installation Procedures

The following post-installation procedures apply to all ANSYS, Inc. products. Individual products may have additional post-installation procedures; please refer to the following sections.

After the product is installed, you need to establish some system settings, including pathnames and environment variables. See your operating system documentation for specific instructions on setting paths and environment variables.

1. Set the following environment variables based on the behavior you want.
  - The **ANSYSLIC\_DIR** environment variable sets the location of the ANSYS licensing directory hierarchy. The default value is `c:\Program Files\ANSYS Inc\Shared Files\Licensing`. You probably will not need to reset this variable, unless you change the location of the licensing files.

To set environment variables:

**Windows XP:** Right click **My Computer** and choose **Properties> Advanced> Environment Variables**. Click **New**. Type the name in the Variable Name field and the desired setting in the Variable Value field. Click **OK** on all dialog boxes.

**Windows 7 / Vista:** Right click **Computer** and choose **Properties**. Click **Advanced System Settings**, and click **Environment Variables**. Click **New**. Type the name in the Variable Name field and the desired setting in the Variable Value field. Click **OK** on all dialog boxes.

If you have any command prompts or console windows open when you set or reset environment variables, those windows will need to be closed and restarted in order to pick up the new settings.

2. Set the home directory. To set a home directory in Windows, you need to set a **HOMEDRIVE** environment variable to the desired drive letter (including the colon) and a **HOMEPATH** environment variable to the desired path. For example:

```
HOMEDRIVE=C:
HOMEPATH=\Users\Mine
```

3. Set the license manager to start automatically at boot time. For platform-specific instructions, see [License Manager Automatic Startup Instructions](#) in the *ANSYS, Inc. Licensing Guide*.
4. Designate server(s) for license checkout and establish necessary user privileges (recommended but not required). For information on these tasks, see [Post-Installation Instructions for the License Manager](#) in the *ANSYS, Inc. Licensing Guide*.
5. Make a backup copy of the `\ANSYS Inc` directory.
6. Verify the product installation by selecting each product from the **Start** menu to verify that they each start and run correctly. You must be pointing to a valid license server machine before you can verify the installation.
7. If you chose not to set file associations during the installation, you may want to run the File Association utility now. Select **Start> Programs> ANSYS 13.0> Utilities> File Association**. Administrative privileges are required to run this utility. You can also run the File Association utility in silent mode:

```
<install_dir>\v130\commonfiles\tools\<platform>\fileassoc.exe -silent
```

You can see help on the `fileassoc.exe` by issuing `-help`. File associations are as follows:

ANSYS, Inc. Product	File Extension
Mechanical APDL (ANSYS)	.db, .dbb, .grph

ANSYS, Inc. Product	File Extension
ANSYS CFD-Post	.mres, .res
ANSYS CFX	.cfx, .cvf, .def, .mdef
ANSYS FLUENT	.cas
ANSYS ICEM CFD	.blk, .jrf, .prj, .rpl, .tin, .uns
ANSYS Workbench	.ad, .agdb, .aqdb, .bgd, .cmdb, .dsdb, .engd, .fedb, .mechdat, .meshdat, .wbdb, .wbpj, .wbpz

### 3.4.1. Post-Installation Procedures for Mechanical APDL (ANSYS) and ANSYS Workbench Products

The following post-installation procedures apply only to the Mechanical APDL (ANSYS) and ANSYS Workbench products. These are in addition to the post-installation procedures noted above for all products.

1. If you installed as a non-administrative user, run the **CAD Configuration Manager** as administrator to configure the CAD products appropriately. See [Configuring CAD Products \(p. 35\)](#) for details on running the **CAD Configuration Manager**. If you do not have the shortcut to access the **CAD Configuration Manager**, you can find it at:

```
<install_dir>\v130\commonfiles\CAD\bin<platform>
\Ans.CadInt.CADConfigUtilityGUI.exe
```

2. Set the following environment variables based on the behavior you want. Set the environment variables following the conventions of your operating system. Not all of these are required for all integrated ANSYS Workbench products (such as ANSYS AUTODYN), but setting them correctly for ANSYS Workbench will in no way hinder the performance of the other products.
  - The **ANSYS130\_DIR** environment variable sets the location of the ANSYS directory hierarchy. The default value is `c:\Program Files\Ansys Inc\V130\ANSYS`. You probably will not need to reset this variable, unless you change the location of the installed files.
  - **ANSYS130\_PRODUCT** - set this to the correct product variable to run Mechanical APDL to start with the correct Mechanical APDL product without specifying the **-p** command modifier each time.
  - **ANS\_CONSEC** - set this to YES to disable Mechanical APDL dialog boxes and allow multiple jobs to run consecutively without waiting for user input. Settings for **ANS\_CONSEC** are:

Value of <b>ANS_CONSEC</b>	Batch Dialog Box	GPF Dialog Box on Error
Not defined/default	No	Yes
YES	No	No
NO	Yes	Yes

- **ANSYS\_LOCK** - set to ON (default) to create file locks to prevent users from opening a new job with the same name and in the same directory as the current job.
- **ANSYS130\_MAT161** - set this environment variable to 1 to enable use of the LS-DYNA \*MAT\_COMPOSITE\_MSC material (requires an LS-DYNA MAT\_161 license).
- **ANSYS130\_MAT162** - set this environment variable to 1 to enable use of the LS-DYNA \*MAT\_COMPOSITE\_DMG\_MSG material (requires an LS-DYNA MAT\_162 license).

- **LSTC\_LICENSE** - This is an LS-DYNA environment variable that controls which license manager is used by the LS-DYNA executable. It is set to ANSYS during installation; changing this environment variable will prevent ANSYS LS-DYNA from using your ANSYS licenses when running the executable.

**Quality Assurance Program:** If you require verification of the ANSYS program, ANSYS, Inc. offers Quality Assurance programs for some ANSYS products. If you are interested in this service, go to <http://www.ansys.com/services/ss-quality-services.asp> or call the ANSYS, Inc. Quality Assurance Group at (724) 746-3304.

### 3.4.2. Post-Installation Procedures for ANSYS CFX and ANSYS CFD-Post

After installing the ANSYS CFX or ANSYS CFD-Post software, verify the installation as follows:

1. Select **Start> Programs> ANSYS 13.0> Fluid Dynamics> CFX**

The CFX 13.0 Launcher opens.

2. In the **Working Directory** field of the Launcher, type the path to the `examples` directory, which by default is: `C:\Program Files\ANSYS Inc\13.0\CFX\examples`
3. Click **CFD-Post 13.0**. CFD-Post opens.
4. Select **File > Load Results**. The **Load Results File** window opens.
5. Select **StaticMixer\_001.res** and click **Open**.

A wireframe of a mixer appears in the 3D Viewer.

6. Select **Session > Play Session**. The **Play Session File** window opens. Select **StaticMixerPost.cse** and click **Open**.

The mixer changes orientation, displays a plane colored by temperature, hides the plane, then loads a vector plot of temperature located on the plane.

7. Select **File > Close** and close the case without saving.
8. To exit CFD-Post, select **File > Quit**.

If any of the installation verification steps fails, contact your ANSYS Support representative.

### 3.4.3. Post-Installation Procedures for ANSYS AUTODYN

Please refer to the ANSYS AUTODYN *Quick Start Guide* and *What's New* sections in the ANSYS AUTODYN help for information on starting and running ANSYS AUTODYN in either GUI or batch modes.

### 3.4.4. Post-Installation Procedures for ANSYS FLUENT

After installing the ANSYS FLUENT software, you will need to reset the default values in the ANSYS FLUENT launcher as follows:

1. Verify that the **FLUENT\_INC** environment variable is not set. From the Control Panel, select **System** and click the **Advanced** tab. Click **Environment Variables**. Find and delete the **FLUENT\_INC** variable.
2. Select **Start> Programs> ANSYS 13.0> FLUENT> FLUENT 13.0.X**.
3. Click **Default**.
4. Click **Yes** when asked if you want to discard the LAUNCHER history.
5. Click **Cancel** if you do not wish to start FLUENT at this time. The new defaults will have been saved.



Please refer to the *ANSYS FLUENT Quick Start Guide* for more information.

### 3.4.5. Post-Installation Procedures for ANSYS POLYFLOW

POLYFLOW no longer requires the **FLUENT\_INC** environmental variable. It should be deleted as is recommended for FLUENT:

- From the Control Panel, select **System** and click the **Advanced** tab. Click **Environment Variables**. Find and delete the **FLUENT\_INC** variable.

### 3.4.6. Post-Installation Procedures for ANSYS ASAS

If you are running ASAS Visualizer, you must first manually install the 32-bit VC runtime prerequisite when running on 64-bit Windows systems. Navigate to `\asas\bin\winx64\asas_vis` under the installation directory and double-click the file `2008vcredist_x86.exe`.

### 3.4.7. Post-Installation Procedures for Other Products

**Standalone 3D Viewer** During installation, the Standalone 3D Viewer will not be configured. It can be configured separately after the overall installation is complete by running the following executable as administrator:

```
<install_dir>\v130\CFX\viewer\ANSYS_CFD-Viewer_130_Setup.exe
```

**BladeGen** If you are running BladeGen on Windows 7 and Vista, in order to access the BladeGen online help, you need to install `WinHlp32.exe`, which is available from Microsoft.

During installation, BladeGen will not be configured. It can be configured separately after the overall installation is complete by running the following executable as administrator:

```
<install_dir>\v130\AISOL\BladeModeler\BladeGen\BladeGenConfigure.exe
```

**BladeEditor** In order to use BladeEditor, you must set the Geometry license preference to ANSYS BladeModeler as follows:

1. In the ANSYS Workbench menu, select **Tools> License Preferences**.
2. In the **License Preferences** dialog box, click the **Geometry** tab.
3. If ANSYS BladeModeler is not the first license listed, then select it and click **Move up** as required to move it to the top of the list. If **ANSYS BladeModeler** is not in the list, then you need to obtain an ANSYS BladeModeler license.
4. Select ANSYS DesignModeler in the list and set its value to 0 (which means "Don't Use"). This step prevents DesignModeler from using an ANSYS DesignModeler license when an ANSYS BladeModeler license is not available.
5. Click **OK** to close the dialog box.

**FEMGV** For ANSYS ASAS and ANSYS AQWA customers, FEMGV is also available as a separate, standalone installation, available via media or the ANSYS Customer Download Center.

## 3.5. Launching ANSYS, Inc. Products

To launch ANSYS, Inc. products on Windows platforms, choose the product from the ANSYS 13.0 program group under the **Start** menu (**Start > Programs> ANSYS 13.0**).



For Mechanical APDL (ANSYS), you can also use the launcher: **Start > Programs> ANSYS 13.0> Mechanical APDL Product Launcher**.

## 3.6. Running the **ANS\_ADMIN** Utility for Mechanical APDL (ANSYS) /ANSYS Workbench Products

You may need to run the **ANS\_ADMIN** utility to relink Mechanical APDL (ANSYS) if you use the customization tools. For more information on individual options available with this utility, see the online help accessible from **Help** buttons on the utility dialog boxes. You can launch **ANS\_ADMIN** without administrative privileges, but some of the options require administrative privileges. If certain options are grayed out, you do not have the necessary system administrator privileges necessary for that option, or the corresponding product may not be installed.

To run the **ANS\_ADMIN** utility, choose **Start> Programs> ANSYS 13.0> Utilities> ANS\_ADMIN Utility**.

## 3.7. Product Localization

Many ANSYS, Inc. products are available in multiple languages, including English, German, French, and Japanese. For those products that are localized, you are able to view the GUI and messages in the specified language. See your specific product documentation for instructions on choosing a localized version of the product.

All products that are localized define the language via the `languagesettings.txt` file. In most cases, you will not have to manually edit this file. If you do need to edit it manually, you can use one of the following values:

- en-us (English, default)
- de (German)
- fr (French)
- ja (Japanese)

ANSYS, Inc. applications will look for the `languagesettings.txt` file in the following locations, in order:

1. `%appdata%\Ansys\v130`
2. `<install_dir>\Ansys Inc\v130\commonfiles\Language`

You will also need to set the **AWP\_LOCALE130** environment variable to the same language that is defined in the `languagesettings.txt` file to ensure that all applications will be able to use the localized version.

Some products are not fully localized but offer only the messages in a translated version. See the following section for instructions on translated message file installation.

### 3.7.1. Translated Message File Installation for Mechanical APDL (ANSYS) /ANSYS Workbench Products

If your ANSYS, Inc. sales representative has supplied you with message files translated into your local language, use the following procedure to install and access these files from within Mechanical APDL (ANSYS):

1. Create an appropriately named subdirectory to hold the message files. For example, if your local language is French, create a directory in the following location:

```
<os drive>\Program Files\Ansys Inc\V130\ANSYS\docu\fr
```

2. Copy the message files (`msgcat.130`, `msgidx.130`, and `msgfnm.130`) into the newly created subdirectory.
3. Access these message files by using the **-l** command line option. For example:

```
ansys130 -l fr
```

Or, when you are running Mechanical APDL from the Mechanical APDL launcher, choose the **Language Selection** option and then pick the desired language.

You must create a newly translated message file for each release of Mechanical APDL because error messages may occur in a different order for each release.

## 3.8. Uninstalling ANSYS, Inc. Products

Use this process to uninstall any ANSYS, Inc. product that was installed using the Unified Installation process.

**Note to CFX Users** The provided uninstall tool cannot uninstall MPI services. If you have CFX products installed that you are uninstalling, you must stop the MPI services before continuing with the uninstall:

1. As an administrator, go to **Start> Run** and type `Services.msc`.
2. In the list that appears, search for "MPICH2 Process Manager, Argonne National Lab" and if found, click **Stop**. Then search for "HP-MPI Remote Launch" and if found, click **Stop**.
3. Continue with the uninstallation procedure, below.

**Uninstall Procedure for All Users** All users should use the following procedure to uninstall ANSYS, Inc. products.

---

### Note

If you configured your CAD programs using the **CAD Configuration Manager** as an administrative user (required to register the necessary prerequisites), you must unconfigure your CAD programs as an administrative user before uninstalling ANSYS products as a non-administrative user. Failing to do so could result in undesirable CAD behavior.

1. Close down all applications that are currently running.
2. Select **Start> Programs> ANSYS 13.0> Uninstall**. You must uninstall with the same or higher privileges than were used to install the product, and we strongly recommend uninstalling as the same user who installed the product originally.

If you are a non-administrative user and you attempt to uninstall a product that was installed by an administrative user, you will not be able to proceed. Likewise, if you are a non-administrative user and you attempt to uninstall a product that was installed by a different non-administrative user, you will not be able to proceed.

If you are an administrative user and you are uninstalling a product that was installed by a non-administrative user, we recommend that you log in as the same non-administrative user who installed the product and uninstall the product as that user. If you wish to proceed with the uninstall as an administrative user, you should first run `%AWP_ROOT130%\commonfiles\tools\<platform>\Product-Config.exe` as the same non-administrative user who installed the product and unconfigure each product. After you have unconfigured the installed products as a non-administrative user, you can then proceed with the administrative uninstall. If you do not run the `ProductConfig.exe` as the

installing non-administrative user, the Start menu options and environment variables will not be removed for that user.

3. Select the product(s) to be uninstalled and unconfigured and click **Uninstall Selected Item(s)**. Not all products and product components may be listed individually.
4. You will be asked to save all data and close all Windows applications before continuing. Click **OK**.
5. After all products are uninstalled, click **Yes** to delete the \v130 directory. This will remove all files and subdirectories under the \v130 directory, including any user or customized files that you may have added. If you have added any files or subdirectories, you may want to copy them to a temporary location so they are not deleted.
6. A message appears indicating that the uninstallation is complete. Click **OK**.

**CFD-Post** CFD-Post can be installed as a standalone product; however, as a standalone product, it is also automatically installed with FLUENT, POLYFLOW, and Icepak. To uninstall CFD-Post, you must uninstall all of the above products that you have installed.

- If you choose to uninstall standalone CFD-Post without uninstalling the other products, you will not be able to proceed with the uninstall. You will need to revise your uninstall selections before continuing.
- If you choose to uninstall all of the other products that include standalone CFD-Post but you do not select CFD-Post, then CFD-Post will not be removed.

Individual product directories may not be removed if they contain components that are shared with other products that have not been uninstalled. However, the uninstalled product will no longer run.

In the case of a platform (file server) installation, the uninstall will remove the selected product(s) from all of the Windows platforms.

This procedure removes the specified ANSYS, Inc. program from your system but will not remove the ANSYS licensing components. Follow the steps in [Uninstalling Licensing Components \(p. 33\)](#) to uninstall the licensing components.

### 3.8.1. Uninstalling Licensing Components

Before proceeding with the following steps to remove the ANSYS licensing components from your system, make certain that there are no ANSYS, Inc. products on this machine or any other machines on the network that rely on these ANSYS licensing components.

**License Servers** Follow these steps on a license server machine:

1. Stop the ANSYS, Inc. License Manager via the **ANSLIC\_ADMIN** utility (**Start> Programs> ANSYS Inc. License Manager> Server ANSLIC\_ADMIN Utility**).
2. Uninstall the ANSYS, Inc. License Manager service. You must use the following command to do so:

```
"C:\Program Files\Ansys Inc\Shared Files\Licensing\<platform>\ansysli_server"
-k uninstall
```

3. Delete the licensing subdirectory.
4. Remove the **ANSYS, Inc. License Manager** folder from the **Start** menu.
5. Remove the **ANSYSLIC\_DIR** and the **ANSYSLIC\_SYSDIR** environment variables, if set.

**Clients** Follow these steps on client machines:

1. Delete the licensing subdirectory.

2. Remove the **ANSYS 13.0> ANSYS Client Licensing** folder from the **Start** menu.

---

## Chapter 4: Configuring CAD Products

---

The connection, geometry interface, and other CAD functionality for all supported CAD products is included with the ANSYS release media. Select the CAD systems you want to install during the installation process. If you choose not to install CAD systems during the installation, you can install them at a later date.

To run the connection or geometry interface functionality, you need to:

1. Ensure that the CAD product is correctly installed and licensed.
2. Ensure that you have the correct ANSYS, Inc. license(s).
3. Run the installation setup.

For complete information about the files you can import and the platforms supported by the connection capability, see *Introduction to Import* in the [ANSYS Connection User's Guide](#). For more information about geometry interface information in ANSYS Workbench, see the topics **Attach to Active CAD Geometry** and **Import External Geometry File** in the ANSYS DesignModeler help.

---

### Caution

Be sure to install Mechanical APDL (ANSYS) and the connection functionality from the same release. If you attempt to run the latest connection functionality on a machine that is running an earlier release of Mechanical APDL, or vice versa, the connection may fail.

The connection for Pro/ENGINEER requires you to run Mechanical APDL, Pro/ENGINEER and the connection for Pro/ENGINEER on the same machine. The connection for NX requires you to run Mechanical APDL, NX, and the connection for NX on the same machine. The connections for CATIA, SAT, and Parasolid do not require any additional CAD installation.

### 4.1. Using the CAD Configuration Manager

The **CAD Configuration Manager** allows you to configure geometry interfaces for Mechanical APDL (ANSYS), ANSYS Workbench, and ICEM CFD Direct CAD interfaces. CAD configuration is typically handled during the product installation; however, if you chose to skip those steps, or if you make changes to your local CAD configuration between releases (for example, you install, move, or update your CAD package, or remove it entirely), you can use this utility.

You can run the **CAD Configuration Manager** as an administrative or as a non-administrative user. However, in order for the necessary prerequisites to be registered to run the ANSYS Workbench readers, and some other components (see Step 3, below), you must run as an administrative user. Administrative rights are also required to fully configure several ANSYS CAD products, including the ANSYS Workbench reader for Catia V5, and the ANSYS Workbench Plug-ins for Inventor, CoCreate Modeling, Solid Edge, and SolidWorks. NX in reader mode also requires administrative rights to configure. Likewise, when unconfiguring, you should unconfigure CAD products as an administrative user before uninstalling the product as a non-administrative user. Otherwise, your CAD products could end up in an undesirable state, resulting in unpredictable behavior.

If you are running as an administrative user on a machine with a non-administrative installation, you may not see a **CAD Configuration Manager** shortcut. You can run the tool from `<os drive>\Ansys Inc\V130\commonfiles\CAD\bin\<platform>\Ans.CadInt.CADConfigUtilityGUI.exe`.

The **CAD Configuration Manager** is organized into several tabs:

- CAD Selection
- Pro/ENGINEER
- NX
- Teamcenter Engineering (active on Windows only)
- CAD Configuration

An administrative user has the option to configure or unconfigure any selected CAD systems either for the present user's environment or for all users, as indicated by the **Configuration actions apply to** options. When the original installation was performed by a non-administrative user, an administrative user will only be allowed to configure or unconfigure for all users. In this situation, Pro/Engineer and NX configure and unconfigure actions are skipped. Non-administrative users will only be allowed to configure for themselves. Any user-specific configuration settings take precedence over global configuration settings.

This document describes how to use the **CAD Configuration Manager** as a wizard, beginning with the **CAD Selection** tab and using the **Next** button to progress through the configuration process. You can also manually select each tab to progress through the configuration process; both methods work the same. However, the applicable tabs will not be enabled until you choose the associated product on the **CAD Selection** tab.

1. Run **Start> Programs> ANSYS 13.0> Utilities> CAD Configuration Manager**.
2. On the **CAD Selection** tab, choose the ANSYS products and the CAD products that you need to configure. You must select at least one ANSYS product to enable the remaining functionality. Click **Next**.
3. If you selected only **Workbench and ANSYS Geometry Interfaces** with no CAD Products selected, the necessary prerequisites will be registered, enabling Geometry (DesignModeler) and all the Readers. You must run the **CAD Configuration Manager** as an administrative user for some CAD registrations to take place. Select the **CAD Configuration** tab and click **Configure Selected CAD Interfaces** to register the prerequisites.
4. If you selected Pro/ENGINEER as one of your CAD products, you will have to choose which Pro/ENGINEER product to configure:
  - a. The **Reader** option specifies the non-associative source and doesn't require a Pro/ENGINEER installation.
  - b. The **Workbench Associative Plug-In** option, which requires Pro/ENGINEER to be installed. When you choose this option, the Pro/ENGINEER tab opens.
    - i. Enter or browse to the Pro/ENGINEER installation location (for example, `C:\Program Files\ProEWildfire 4.0`).
    - ii. Enter or browse to the Pro/ENGINEER start command (for example, `C:\Program Files\ProEWildfire 4.0\bin\proel.bat`). Include the complete path if entering the command manually.
    - iii. Click **Next**.

---

**Note**

If ANSYS Workbench is installed by an administrative user, any non-administrative users will need to run the **CAD Configuration Manager** for Pro/ENGINEER from their own accounts before the plug-in will load.

5. If you selected NX as one of your CAD products, the **NX** tab opens.
  - a. Enter or browse to the NX installation location (for example, C:\Program Files\UGS\NX 6.0).
  - b. Enter or browse to the NX custom directory file path (for example, C:\Documents and Settings\UserName\Application Data\Ansys\v130\Unigraphics).

---

**Note**

If you are installing as an administrator, this file path will be defined in your administrator's user space by default and non-administrative users will not be able to run. Use this setting to specify a new location for the custom directory file path that is accessible by all users.

- c. Choose to enable or disable color attribute processing.
  - d. Click **Next**.
6. If you selected Teamcenter Engineering as one of your CAD products, the Teamcenter Engineering tab opens:
  - a. Enter or browse to the Teamcenter Engineering installation location (for example, C:\SIEMENS\Teamcenter2007).
  - b. Choose **Next**.
7. If you selected Catia V5 as one of your CAD products, you will have the option to choose which CATIA V5 product to configure.
  - a. Select the **Standard Interface** option to configure the CATIA V5 standalone reader that does *not* require an installation of the CATIA V5 CAD program (this reader does *not* support CAD associativity or parameters).
  - b. Select the **CADNexus/CAPRI CAE Gateway** option to configure the CATIA V5 associative reader that *does* require an installation of the CATIA V5 CAD program and the CADNexus/CAPRI CAE Gateway (this geometry interface *does* support CAD associativity and parameter modifications).
  - c. Click **Next**.
8. The **CAD Configuration** tab opens.
  - a. Click the **Configure Selected CAD Interfaces** button.
  - b. When the configuration for all products is complete, log entries appear, listing those products that were successfully configured and those that were not. Address any errors and reconfigure.
  - c. For more details, click the **Display Configuration Log File** button to see a detailed log file.

9. When all of your CAD products have been successfully configured, click **Exit**.

You can review the **CAD Configuration Manager** log file, `CADConfigurationMgr.log`, in `\ANSYS Inc.`

### 4.1.1. Unconfiguring

If you need to unconfigure any of your CAD products, follow the steps above, but choose **Unconfigure Selected CAD Interfaces** on the **CAD Configuration** tab.

### 4.1.2. Running the CAD Configuration Manager in Batch Mode

You can configure ANSYS Geometry Interfaces by supplying the **CAD Configuration Manager** with arguments specific to the CAD sources you want to make available. The following table contains a list of supported arguments.

The command to run the **CAD Configuration Manager** in batch mode on Windows is:

```
"<installpath>\commonfiles\CAD\bin\<platform>\Ans.CadInt.CADConfigurationUtility.exe"  
-arguments
```

Argument	Value	Comment
unconfigure	None	Results in any specified CAD sources being unconfigured. When this flag is absent, the <b>CAD Configuration Manager</b> will attempt to configure all designated CAD sources.
Either: PE_CONFIG_WB or PE_CONFIG_WBSPATIAL	None	Specify Pro/ENGINEER source as either the associative plug-in or the reader (Spatial, no Pro/ENGINEER install required). When the plug-in is specified, additional arguments PROLOADPOINT and PROE_START_CMD are required.
PROLOADPOINT	Full path to Pro/ENGINEER installation (quotes required on Windows).	Not required with unconfigure operation.
PROE_START_CMD	Full path to command used to launch Pro/ENGINEER (quotes required on Windows)	Not required with unconfigure operation.
UG_CONFIG_WB	None	Configure/unconfigure NX Geometry Interface to Workbench. The argument UGII_BASE_DIR must also be specified.
UGII_BASE_DIR	Full path to NX installation	This should agree with environment variable <b>UGII_BASE_DIR</b> . Not required with unconfigure operation.
UG_USE_COLORS	None	Process NX entity colors as possible source of attributes and named selections. Not required with unconfigure operation.
Either: CATIA_SPATIAL or CATIA_CAPRI	None	Specify Catia V5 source as either the Reader (Spatial, no Catia install required) or CAPRI (CADNexus install required).



Argument	Value	Comment
OSDM_CONFIG	None	Configure/unconfigure CoCreate Modeling.
INVENTOR_CONFIG	None	Configure/unconfigure Inventor.
SOLIDWORKS_CONFIG	None	Configure/unconfigure SolidWorks.
SOLIDEDGE_CONFIG	None	Configure/unconfigure Solid Edge.
PE_CONFIG_ICEM	None	Configure/unconfigure the ICEM CFD Direct CAD Interface to Pro/ENGINEER. The arguments PROLOADPOINT and PROE_START_CMD must also be specified.
UG_CONFIG_ICEM	None	Configure/unconfigure the ICEM CFD Direct CAD Interface to NX. The argument UGII_BASE_DIR must also be specified.
TEAMCENTER_CONFIG	None	Configure/unconfigure Teamcenter Engineering; for configuration requires IMAN_ROOT flag.
IMAN_ROOT	Path to Teamcenter Engineering installation.	Not required with unconfiguration operation.
JTOPEN_CONFIG	None	Configure/unconfigure JTOpen.
CATIA4_CONFIG	None	Configure/unconfigure CATIA v4.

### Note

All arguments require a dash (-) before them in order to be properly recognized by the **CAD Configuration Manager**. Arguments' values should not have a dash preceding them.

For example, you can configure Pro/ENGINEER and SolidWorks Geometry Interfaces to ANSYS Workbench from the command line by using the following:

```
"<installpath>\commonfiles\CAD\bin\<platform>\Ans.CadInt.CADConfigurationUtility.exe"
-SW_CONFIG -PE_CONFIG_WB -PROLOADPOINT "C:\Program Files\proeWildfire 4.0"
-PROE_START_CMD "C:\Program Files\proeWildfire 4.0\bin\proeWildfire4.bat"
```

where *installpath* is the same as the value of environment variable **%AWP\_ROOT130%**, *platform* is the value of environment variable **%ANSYS\_SYSDIR%**, and Pro/ENGINEER is installed to C:\Program Files\proeWildfire 4.0.

To unconfigure the same CAD Interfaces, the command would be:

```
"<installpath>\commonfiles\CAD\bin\<platform>\Ans.CadInt.CADConfigurationUtility.exe"
-unconfigure -SW_CONFIG -PE_CONFIG_WB
```

Although the argument order does not matter, an argument value must immediately follow its argument.

### 4.1.3. Uninstalling

If you performed your installation as a non-administrative user and you are unconfiguring CADs as an administrative user in preparation for a product uninstall, then use the **Uninstall All Products** button on the **CAD Configuration** tab. This action will also unconfigure prerequisites for geometry and readers.

#### Warning

Do not proceed with an uninstall for CAD-specific unconfigure actions or you will leave some ANSYS Workbench products unusable.

### 4.1.4. Pro/ENGINEER Configuration

Running the **CAD Configuration Manager** for Pro/ENGINEER performs the following steps to activate the Pro/ENGINEER plug-in:

- Sets the environment variable **PROE\_START\_CMD130** to the file used to launch Pro/ENGINEER (for example C:\Program Files\proeWildfire 4.0\bin\proe1.bat).
- Sets the environment variable **ANSYS\_PROEWF\_VER** to ProE on Windows 32-bit, or to ProE\_Wildfire4 if running Wildfire 4.0 on 64-bit Windows.
- Adds the ANSYS 13.0 entry to the `config.pro` file located in either %HOME% or %HOMEDRIVE%%HOMEPATH%. An example of this entry is:

```
PROTKDAT E:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\%ANSYS_PROEWF_VER\
ProEPages\config\WBPlugInPE.dat
```

- Updates the `WBPlugInPE.dat` file referenced in the `config.pro` file, so that it contains information for loading the WorkBench Plug-In and the ANSYS Connection product.
- Registers the Plug-In file `WBPlugInPEU.dll` referenced in the `WBPlugInPE.dat` file.

**WBPlugInPE.dat File Contents** The `WBPlugInPE.dat` file should look like this example:

```
NAME WB130PluginProWF
EXEC_FILE E:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\%ANSYS_PROEWF_VER%\
%ANSYS_SYSDIR%\WBPlugInPEU.dll
TEXT_DIR E:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\%ANSYS_PROEWF_VER%\
ProEPages\Language\%AWP_LOCALE130%
STARTUP dll
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
REVISION ProEWildfire
END

NAME ac4pro130dll
exec_path E:\Program Files\Ansys Inc\V130\ANSYS\ac4\bin\pro\%ANSYS_SYSDIR%\ac4pro.exe
text_path E:\Program Files\Ansys Inc\V130\ANSYS\ac4\data\pro\text
STARTUP dll
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
revision 24.0
end
```

Do not delete any of these lines. If you modify this file, do NOT enter a carriage return after the END line. The file may be customized with other information. If these lines are deleted, or if the `WBPlugInPE.dat` file is not present in any of the directories in the search path, Pro/ENGINEER will not load ANSYS-related CAD interfaces. You should typically never have to edit these files for path information contained within

them. Paths are determined by environment variable settings, which are set automatically during installation. If you encounter problems when attempting to run Pro/ENGINEER, use the **CAD Configuration Manager** to reconfigure rather than attempting to edit files directly.

#### 4.1.4.1. Configuring the Connection for Pro/ENGINEER

All Pro/ENGINEER users must have copies of the `WBPlugInPE.dat` file and the `config.anscon` (connection for Pro/E) files. The `config.anscon` file is placed in the `Program Files\Ansys Inc\V130\ansys\ac4\data\<platform>` directory. `config.anscon` must be copied into the user's working directory at the time pro/engineer is started. The `WBPlugInPE.dat` file is placed in the `Program Files\Ansys Inc\V130\AISOL\CADIntegration\%ANSYS_PROEWF_VER%\ProEPages\config` directory. This file defines the name of the executable, the path to the executable, the path to the message file, and the current revision of Pro/ENGINEER.

##### Note

If the ANSGeom menu in Pro/ENGINEER does not appear correctly, copy the `config.anscon` file into your working directory and restart Pro/ENGINEER.

##### 4.1.4.1.1. The WBPlugInPE.dat File and config.pro File

Pro/ENGINEER uses the `WBPlugInPE.dat` file to locate related executables such as the connection for Pro/ENGINEER.

Mechanical APDL (ANSYS) /ANSYS Workbench also requires a `config.pro` file, which resides in the `%HOME%` directory, and when HOME is not defined, in the `%HOMEDRIVE%%HOMEPATH%` directory. This file contains a `PROTKDAT` line that points to the `WBPlugInPE.dat` file. This `config.pro` file is generated either during the product install or by running the **CAD Configuration Manager** to configure the Mechanical APDL (ANSYS) /ANSYS Workbench products for Pro/ENGINEER.

The `<proe_platform>` variable is the name that Pro/ENGINEER gives to its platform directories:

Hardware Platform	ANSYS Platform <code>&lt;platform&gt;</code>	Pro/ENGINEER Platform <code>&lt;proe_platform&gt;</code>
Intel 32-bit	win32	i486_nt
XP x64	winx64	x86e_win64

##### 4.1.4.1.2. The config.pro File

A typical `config.pro` file might look like the following example. This example has an entry for ANSYS Workbench and another for ANSYS ICEM CFD. You may have other Pro/ENGINEER specific customizations.

```
PROTKDAT E:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\%ANSYS_PROEWF_VER%\
ProEPages\config\WBPlugInPE.dat
PROTKDAT $PROMIF_ACN\protk.dat
```

##### 4.1.4.1.3. The config.anscon File

Users who launch Mechanical APDL (ANSYS) from Pro/ENGINEER will need the information from the `config.anscon` file. This file is installed for all users in the `\ac4\data\<platform>` subdirectory. You typically do not need to edit this file. Here is a sample `config.anscon` file:

```
ANSYS_CMD %AWP_ROOT130%\ANSYS\bin\%ANSYS_SYSDIR%\ansys.exe
ANSYS_GRAPHICS_DEVICE WIN32
ANSYS_SOLVER Sparse**
ANSYS_SELECTED_LAYERS 1-256**
ANSYS_GEOMETRY_TYPE Solids Only**
ANSYS_NEUTRAL_FORMAT Yes
ANSYS_PRODUCT_NAME ANFL
```

\*\*These variables are not supported by Pro/ENGINEER and are ignored by Pro/ENGINEER.

See *Setting ANSYS Configuration Parameters* in the [ANSYS Connection User's Guide](#) for more information about the `config.anscon` file.

#### 4.1.4.2. Pro/ENGINEER Environment Variables

Most environment variables are set during product installation. In general, you will not need to reset these variables. Check the manuals for the individual CAD products for information about environment variables and their settings.

##### PROE\_START\_CMD130

Specifies the name of the `.bat` file which launches Pro/ENGINEER on the system. The path to the executable should be already set if the Pro/ENGINEER installation path has been defined for the **PATH** environment setting.

**PROE\_START\_CMD130** Default = `C:\Program Files\PROEWildFire4.0\bin\proe1.bat`

#### 4.1.5. NX Configuration

Running the **CAD Configuration Manager** for NX performs the following steps to activate the NX plug-in:

- If not already existing, sets the environment variable **UGII\_CUSTOM\_DIRECTORY\_FILE** to `%APP-DATA%\Ansys\v130\Unigraphics\custom_dirs.dat`.
- If not already present, adds an entry to the `custom_dirs.dat` file specifying the location of the Plug-In, e.g., `C:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\UnigraphicsNX5\%ANSYS_SYSDIR%`.
- Registers the Plug-In file (either `DSPlugInUG5U.dll` or `DSPlugInUG6U.dll`). For the previous example, these files would be located in `C:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\UnigraphicsNX5\winx64\startup`.
- Registers `AGPSUtilU.dll`. For the previous example, this file would be located in `C:\Program Files\Ansys Inc\V130\AISOL\CADIntegration\UnigraphicsNX5\winx64`.

##### 4.1.5.1. Configuring the Connection for NX

**User-Defined Environment Variables** You will need to have the following environment variable set if you will be running the connection for NX product from inside NX.

##### UGII\_VENDOR\_DIR for Mechanical APDL

This environment variable is not set during the installation process and can only be set in a user's startup file or at command level before running NX. This environment variable defines the ANSYS menu for NX. You must set this environment variable if you will be running the connection for NX product from inside NX. This environment variable tells NX where to find the ANSYS program, for example: `Program Files\Ansys Inc\V130\ANSYS\ac4\bin\ug30\<platform>`.

**UGII\_BASE\_DIR**

The environment variable **UGII\_BASE\_DIR** must be defined for proper operation of NX and the ANSYS Workbench Plug-In to NX. This environment variable is typically created by the NX installer; however, in rare cases, you may have to create/update the variable. In these situations, the **NX Installation Location** field will be left blank within the **CAD Configuration Manager's** NX tab.

**4.1.5.2. Configuring for Teamcenter Engineering**

Running the **CAD Configuration Manager** for NX performs the following steps to activate the Teamcenter Engineering plug-in:

- Sets the environment variable **IMAN\_ROOT** to the Teamcenter Engineering installation path.

**4.1.6. ANSYS ICEM CFD Configuration**

Running the **CAD Configuration Manager** for ANSYS ICEM CFD performs the following steps to activate the ANSYS ICEM CFD plug-in:

- For Pro/ENGINEER, sets the environment variable **PROE\_START\_CMD130** to the file used to launch Pro/ENGINEER (for example C:\Program Files\proeWildfire 4.0\bin\proel.bat).
- Sets the **PROMIF\_ACN** environment variable to the directory where the ANSYS ICEM CFD interface resides (C:\Program Files\Ansys Inc\V130\icemcfd\<platform>\dif\pro by default).
- For NX, sets the **UGII\_VENDOR\_DIR** environment variable to the directory where the ANSYS ICEM CFD interface resides (C:\Program Files\Ansys Inc\V130\icemcfd\<platform>\dif\ug by default).

**4.2. Configuring CADNexus/CAPRI CAE Gateway for CATIA V5**

The CADNexus/CAPRI CAE Gateway for CATIA V5 product is available for ANSYS Workbench. The ANSYS Workbench-specific portions are included in the installation. However, to run this product, you must complete the configuration as follows:

1. Install ANSYS Workbench according to the instructions in this guide.
2. Install CATIA V5 and LUM licensing (requires a CATIAV5 license key MD2, HD2 or ME2).
3. From the ANSYS, Inc. Customer Portal, download the CADNexus/CAPRI CAE Gateway for CATIA V5 for your platform (Windows 32 or Windows x64) to a temporary folder. (Do not download to a folder containing blank spaces in the folder name, e.g., Program Files). Follow the download procedures described in [Pre-Installation Instructions for Download Installations](#) (p. 13).
4. Extract the contents of the zip file using your standard zip utility.
5. Install CADNexus/CAPRI CAE Gateway for CATIA V5 using the appropriate exe for your platform.
6. Follow the instructions on the CAPRI installation screens to complete the installation. When specifying the component to install, you will need to select the desired CatiaV5 release under CAPRI CAD Applications. When asked if CAPRI should set the environment variables, click **Yes**.
7. Run the **CAD Configuration Manager** and select the **Catia V5: CADNexus/CAPRI CAE Gateway** option to complete the configuration as described in [Using the CAD Configuration Manager](#) (p. 35).

### Note

Using the **CAD Configuration Manager** to set up the CADNexus/CAPRI CAE Gateway does not require administrative rights, although installing that product (step 5) requires these rights. However, if you wish to reconfigure to use the ANSYS Workbench Reader for Catia V5, administrative rights are required.

If you wish to revert to the standard CATIA V5 interface, run the **CAD Configuration Manager** and select the **Catia V5: Standard Interface** option to complete the configuration as described in [Using the CAD Configuration Manager](#) (p. 35).

You cannot run the standard CATIA V5 interface simultaneously with the CADNexus/CAPRI CAE Gateway for CATIA V5 interface.

---

## Chapter 5: Troubleshooting

---

### 5.1. Installation Troubleshooting

This section lists problems and error messages that you may encounter while installing and/or running ANSYS, Inc. products. After each situation description or error message is the user action required to correct the problem.

You can also find answers to commonly-asked questions on our customer portal. After you log in to the customer portal, select **Online Support**> **Installation/System FAQs**. Then select either the Windows or UNIX/Linux link.

For information on licensing-related errors, see the [Troubleshooting](#) section of the *ANSYS, Inc. Licensing Guide*.

In addition, this appendix describes the ANSYS diagnostic tools, which are useful for troubleshooting some problems.

#### 5.1.1. Using ANSLIC\_ADMIN to Gather Diagnostic Information

You can use the **Gather Diagnostic Information** option of the **ANSLIC\_ADMIN** utility to query the system for various pieces of information that may be needed for troubleshooting certain problems. This option collects information about the system as well as about ANSYS licenses and sends all of the information that it collects to the log area. At times it may be necessary to provide the information for technical support. Use the **Write to File** button at the bottom of the **ANSLIC\_ADMIN** to write a file. Then forward the file to the appropriate person.

You can use the **Gather Diagnostic Information** option of the **ANSLIC\_ADMIN** utility to query the system for various pieces of information that may be needed for troubleshooting certain problems. This option collects information about the system as well as about ANSYS licenses and sends all of the information that it collects to the log area. At times it may be necessary to provide the information for technical support. Use the **Write to File** button at the bottom of the **ANSLIC\_ADMIN** to write a file. Then forward the file to the appropriate person.

You can also use the `AnsysInstallationDiagnostics.exe`, in the `\v130\instutil` directory, for additional installation diagnostics. This tool is helpful to diagnose installation and runtime issues. One or more log files will be generated either at the same location as the tool or in your temporary directory, depending on directory permissions. The log files are intended to help ANSYS, Inc. technical support personnel expedite any technical issues. It would be useful to have the files ready to provide before contacting Technical Support.

#### 5.1.2. Uninstall Gives Access Denied Error Message

If you run the uninstall utility as a non-administrative user, you may get access denied error messages when removing product components. Run the uninstall as an administrative user, and close all applications before running the uninstall.

### 5.1.3. Uninstall on Vista Systems Gives Compatibility Error Message

If you uninstall an ANSYS, Inc. product on Vista systems using the ANSYS, Inc. Uninstall utility, you may see the following error message:

**This program might not have installed correctly.**

**If this program didn't install correctly, try reinstalling using settings that are compatible with this version of Windows.**

This message is caused by Vista's Program Compatibility Assistant and can safely be ignored. Click **Cancel** and continue with the uninstall.

### 5.1.4. A .chm File Does Not Display Properly Across a Network

Help system .chm files may not display topics correctly when you try to use a Universal Naming Convention (UNC) path to open the file on a network shared folder, typical with a Network Configuration install. This is a known issue with security update 896358, security update 840315, or Microsoft Windows Server 2003 Service Pack 1 (SP1). Please go to <http://support.microsoft.com/kb/896358> and <http://support.microsoft.com/kb/896054> for more information on this problem and Microsoft's recommended fix.

### 5.1.5. License Manager Fails to Start on Windows Pentium III Systems

The version of FLEXlm used at ANSYS Release 13.0 no longer supports using a CPU ID as the hostid. As a result, if you are using a Windows Pentium III system as a license server, you will need to obtain a new license file to be able to run this release. To obtain your new license, run the **Register License Server Information** option on the **ANSLIC\_ADMIN** utility on the system you have designated as your license server. Send the resulting file **LICSERVER.INFO** to your ANSYS sales representative. We will then supply you with a new license file.

### 5.1.6. Products Crash with an Application Error

On Windows Server 2003 or Windows XP systems, ANSYS applications may crash with an application error if the DEP (Data Execution Prevention) flag is set. To check your DEP setting, right mouse click on **My Computer**, and select **Properties**. Select the **Advanced** tab, and under **Performance**, click **Settings**. Then select the **Data Execution Prevention** tab.

If DEP is set, you will need to add ANSYS to the list of exceptions. For more information on DEP and how to change the settings, see <http://support.microsoft.com/kb/875351>.

### 5.1.7. Product Installation Does Not Create Start Menu Item for ANSYS and CAD Plugins Do Not Work

On Vista systems, if you install ANSYS, Inc. products but do not see a Start Menu item and your CAD plugins do not work, the TEMP environment variable may be pointing to an inaccessible location. For example, it may point to a location such as %USERPROFILE%\Local Settings\Temp. As a result, the product installation could end without configuring the product correctly. You should ensure that the TEMP variable points to a valid location, such as %USERPROFILE%\AppData\Local\Temp.

### 5.1.8. System-related Error Messages

**\*\*\*Error, ANSYS130\_DIR environment variable is not set. This is a fatal error – exiting.**



This message indicates that the **ANSYS130\_DIR** environment variable was not set where necessary for licensing. This environment variable should be set to the installation directory.

### 5.1.9. CasPol Error Messages

#### ERROR: Not enough arguments, no permission set specified

If you are configuring to a server with spaces in the path and get this error, then you must put quotes around your server specification as shown in the following example:

```
C:\Users\>C:\Windows\Microsoft.NET\Framework\v2.0.50727\CasPol.exe -m -a
g 1.2 -url file://"\\machineabc\ANSYS Inc\*" FullTrust
```

## 5.2. Installation Troubleshooting - Mechanical APDL (ANSYS) and ANSYS Workbench Products

The items listed below apply only to the Mechanical APDL (ANSYS) and ANSYS Workbench products.

### 5.2.1. The Mechanical APDL Launcher is Excessively Slow to Start

If the Mechanical APDL launcher takes an excessively long time to startup (Windows only), make sure the hostnames in the `ansyslmd.ini` file and in the **ANSYSLMD\_LICENSE\_FILE** and the **ANSYSLI\_SERVERS** environment variables are typed correctly and that the hosts specified by the hostnames exist. Replacing hostnames with IP addresses may improve the speed as well. Also verify that the port number is correct.

### 5.2.2. Display Problems on Windows XP

This situation applies only to 32-bit Windows systems running Mechanical APDL (ANSYS).

Windows XP includes a feature called Windows Ghosting that may cause Mechanical APDL graphing or plotting problems in 3-D mode on some XP systems. If this problem occurs, follow the procedure below to turn Windows Ghosting off for Mechanical APDL:

1. Open the file `ansys130.sdb`, located in `Program Files\Ansys Inc\V130\ANSYS\bin\intel`.
2. A **Compatibility Database Installer** dialog box stating "Installation of ANSYS130 complete" will be displayed. Click **OK**.

To reset Windows XP Windows Ghosting for Mechanical APDL, follow the procedure below:

1. Open a DOS command prompt and type the following command:  

```
sdbinst /U "%ANSYS130_DIR%\bin\intel\ansys130.sdb"
```
2. A **Compatibility Database Installer** dialog box stating "Uninstallation of ANSYS130 complete" will be displayed. Click **OK**.

### 5.2.3. ANS\_ADMIN Error Messages

#### Grayed out options

If certain options are grayed out, your account may not have the necessary system administrator privileges necessary to run those options.

### 5.2.4. Mechanical APDL Product Launcher Error Messages

Some of the more common error messages follow. See the *ANSYS, Inc. Licensing Guide* for licensing-related launcher messages.

**\*\*\*Cannot create required <profile> file. Therefore, cannot write to profile information during this launcher session.**

If you see this error, you cannot add or modify profile information during this launcher session. Verify that you have write access to the directory and restart the launcher session. Typically, this directory is `C:\Documents and Settings\<user name>\Application Data\Ansys\v130\launcher` on Windows or `~/ansys/v130/launcher/` on UNIX.

**\*\*\*No ANSYS product installations found.**

Verify your Mechanical APDL (ANSYS) product installation. If the desired product was not installed, install it.

### 5.2.5. Distributed ANSYS HP-MPI Error Messages

You may encounter the following message when setting up HP-MPI or running Distributed ANSYS using HP-MPI.

**Error executing ANSYS. Refer to System-related Error Messages in the ANSYS online help. If this was a Distributed ANSYS job, verify that your MPI software is installed correctly and is in your PATH, check your environment settings, or check for an invalid command line option.**

You may see this error if you did not correctly run the set password bat file or add `%MPI_ROOT%\bin` to your PATH. Verify that you completed both of those items according to the HP-MPI installation readme instructions.

You may also see this error if `Ansys Inc\v130\ansys\bin\<platform>` (where `<platform>` is intel or winx64) is not in your PATH.

If you need more detailed debugging information, use the following:

1. Open a Command Prompt window and set the following:

```
SET ANS_SEE_RUN=TRUE
SET ANS_CMD_NODIAG=TRUE
```

2. Run the following command line: `ansys130 -b -dis -i myinput.inp -o myoutput.out.`

### 5.2.6. ANSYS Workbench Products Troubleshooting

This section lists problems and error messages that you may encounter while installing and/or running ANSYS Workbench. After each situation description or error message is the user action required to correct the problem. Always try the suggested user action before contacting your technical support representative.

## Problem Situations

During setup, if you encounter any errors containing the text "0x8000FFFF", you will need to install the required installation prerequisites. Run the installation launcher (`setup.exe`) and choose **Install Required Prerequisites**.

**CAD System Plug-In Menus Do Not Appear for NX or Pro/ENGINEER** ANSYS Workbench on Windows platforms will append its information to an existing customization file for NX and/or Pro/ENGINEER. If no customization file exists, ANSYS Workbench will create a file. For NX, ANSYS Workbench looks for the `custom_dirs.dat` file in the directory specified via the **UGII\_CUSTOM\_DIRECTORY\_FILE** environment variable. For Pro/ENGINEER, ANSYS Workbench looks for the `config.pro` file in the `%HOMEDRIVE%%HOMEPATH%` directory. In addition, during setup of the Pro/ENGINEER Geometry Interface, ANSYS Workbench will also append its information to the `config.pro` file located in the Pro/ENGINEER installation path, under the `\text` directory (e.g., `Proewildfire2\text\config.pro`).

If ANSYS Workbench encounters a read-only file, it will not be able to write the necessary information to the file. In this case, you will need to revise the permissions on the file and manually add the appropriate ANSYS Workbench-specific information in order for the ANSYS menu to appear in NX or Pro/ENGINEER.

**Script Errors When Running ANSYS Workbench** If you encounter script errors such as "Error: Unable to create object microsoft.XMLDOM," you may need to install the latest version of Microsoft's MSXML. Please visit Microsoft's web site at <http://www.microsoft.com/downloads/details.aspx?FamilyID=993c0bcf-3bcf-4009-be21-27e85e1857b1&DisplayLang=en> for more information on downloading and installing MSXML.

## ANSYS Workbench Error Messages

### \*\*\*Unable to connect to Solver Manager.

Another application might be using the Solver Manager port (10002 by default). Try changing the port number by editing the `Ansys.SolverManager.exe.config` file located in the installation directory at `\AISOL\Bin\<platform>`.

If you are getting the "Unable to connect to Solver Manager" error message or are having difficulty launching other applications/editors, it is also possible that the Windows hosts file has been corrupted. Please make sure that localhost is specified in the Windows `<os drive>:\Windows\system32\drivers\etc\hosts` file.

## 5.3. Installation Troubleshooting - ANSYS CFX

The items listed below apply only to the ANSYS CFX products.

### 5.3.1. TurboGrid Mouse Behavior Problems

Depending on the graphics card and driver version, you may experience problems with the accuracy of mouse clicks in the 3D Viewer. For example, you may try to insert a control point at a given location by using the mouse, but the control point appears at a location far from where you clicked the mouse. If you experience such problems, try lowering the hardware acceleration setting of your graphics card.

## 5.4. Contacting Technical Support

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to [www.ansys.com](http://www.ansys.com) and select **Support> Technical**

**Support> Designated Service Providers.** The direct URL is: <http://www1.ansys.com/customer/public/supportlist.asp>. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website ([www.ansys.com](http://www.ansys.com)) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: <http://www.ansys.com/customerportal>.

One of the many useful features of the Customer Portal is the Knowledge Base Search, where you can find solutions to various types of problems, like FAQ. The Knowledge Base Search feature is located under **Online Support> Search Options> Solutions Search**.

Systems and installation Knowledge Resources and FAQs are easily accessible via the Customer Portal under Online Support > [Installation/System FAQs](#). These Knowledge Resources provide a plethora of solutions and direction on how to get installation and licensing issues resolved quickly and efficiently.

## **NORTH AMERICA**

### **All ANSYS, Inc. Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Toll-Free Telephone:** 1.800.711.7199

**Telephone:** 1.724.514.3600

**Fax:** 1.724.514.5096

For installation and licensing questions, visit our Knowledge Resources and FAQs on the [Customer Portal](#). Support for University customers is provided only through the ANSYS Customer Portal.

## **GERMANY**

### **ANSYS Mechanical Products**

**Telephone:** +49 (0) 8092 7005-55

**Fax:** +49 (0) 8092 7005-5

**Email:** [support@cadfem.de](mailto:support@cadfem.de)

### **CFX Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +49 (0) 8024 9054-44

**Fax:** +49 (0) 8024 9054-17

**Email:** [cfx-support-germany@ansys.com](mailto:cfx-support-germany@ansys.com)

### **FLUENT Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +49 (0) 6151 3644-0

**Fax:** +49 (0) 6151 3644-44

**Email:** [fluent-support-germany@ansys.com](mailto:fluent-support-germany@ansys.com)

### **ICEM CFD Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +49 (0) 511 288696-4

**Fax:** +49 (0) 511 288696-66

**Email:** [icemcfd-support-germany@ansys.com](mailto:icemcfd-support-germany@ansys.com)

## UNITED KINGDOM

### All ANSYS, Inc. Products

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +44 (0) 870 142 0300

**Fax:** +44 (0) 870 142 0302

**Email:** [support-uk@ansys.com](mailto:support-uk@ansys.com)

Support for University customers is provided only through the ANSYS Customer Portal.

## JAPAN

### CFX, ICEM CFD and Mechanical Products

**Telephone:** +81-3-5324-8333

**Fax:** +81-3-5324-7308

**Email:** CFX: [japan-cfx-support@ansys.com](mailto:japan-cfx-support@ansys.com); Mechanical: [japan-ansys-support@ansys.com](mailto:japan-ansys-support@ansys.com)

### FLUENT Products

**Email:** FLUENT: [japan-fluent-support@ansys.com](mailto:japan-fluent-support@ansys.com); POLYFLOW: [japan-polyflow-support@ansys.com](mailto:japan-polyflow-support@ansys.com); FfC: [japan-ffc-support@ansys.com](mailto:japan-ffc-support@ansys.com); FloWizard: [japan-flowizard-support@ansys.com](mailto:japan-flowizard-support@ansys.com)

### Licensing and Installation

**Telephone:** +81-3-5324-7305

**Email:** [japan-license-support@ansys.com](mailto:japan-license-support@ansys.com)

## INDIA

### ANSYS Products (including FLUENT, CFX, ICEM-CFD)

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

**Fax:** +91 80 2529 1271

**Email:** FEA products: [feasup-india@ansys.com](mailto:feasup-india@ansys.com); CFD products: [cfdsup-india@ansys.com](mailto:cfdsup-india@ansys.com); Installation: [installation-india@ansys.com](mailto:installation-india@ansys.com)

## FRANCE

### ANSYS, CFX, FLUENT, and ICEM CFD Products

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +33 (0) 820 480 240

**Email:** ANSYS: [ansys-support-france@ansys.com](mailto:ansys-support-france@ansys.com); CFX: [cfx-support-france@ansys.com](mailto:cfx-support-france@ansys.com); FLUENT: [fluent-support-france@ansys.com](mailto:fluent-support-france@ansys.com); ICFM CFD: [icemcfd-support-france@ansys.com](mailto:icemcfd-support-france@ansys.com)

Support for University customers is provided only through the ANSYS Customer Portal.

## **BELGIUM**

### **All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +32 (0) 10 45 28 61

**Email:** [support-belgium@ansys.com](mailto:support-belgium@ansys.com)

Support for University customers is provided only through the ANSYS Customer Portal.

## **SWEDEN**

### **All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +44 (0) 870 142 0300

**Email:** [support-sweden@ansys.com](mailto:support-sweden@ansys.com)

Support for University customers is provided only through the ANSYS Customer Portal.

## **SPAIN and PORTUGAL**

### **All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +33 1 30 60 15 63

**Email:** [support-spain@ansys.com](mailto:support-spain@ansys.com)

Support for University customers is provided only through the ANSYS Customer Portal.

## **ITALY**

### **All ANSYS Products**

**Web:** Go to the ANSYS Customer Portal (<http://www.ansys.com/customerportal>) and select the appropriate option.

**Telephone:** +39 02 89013378

**Email:** [support-italy@ansys.com](mailto:support-italy@ansys.com)

Support for University customers is provided only through the ANSYS Customer Portal.



## Configuring High Performance Computing

---



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 13.0  
November 2010  
002310

ANSYS, Inc. is  
certified to ISO  
9001:2008.



---

## Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superceded by a newer version of this guide. This guide replaces individual product installation guides from previous releases.

## Copyright and Trademark Information

© 2010 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

## Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## Third-Party Software

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---



# Table of Contents

Configuring Distributed ANSYS .....	1
1. Prerequisites for Running Distributed ANSYS .....	1
1.1. MPI Software .....	2
1.2. ANSYS Software .....	3
2. Setting Up the Environment for Distributed ANSYS .....	4
2.1. Optional Setup Tasks .....	6
2.2. Using the mpitest Program .....	7
2.3. Interconnect Configuration .....	8
2.4. Other Considerations .....	9
3. Running a Distributed Job .....	10
Configuring ANSYS CFX Parallel .....	11
1. ANSYS CFX UNIX Parallel Setup .....	11
1.1. Setting Up Remote Access on UNIX/Linux .....	11
1.1.1. Testing Remote Access .....	11
1.1.2. Global Set Up of rsh .....	12
1.1.3. Individual User Set Up for rsh .....	12
1.1.4. Set Up of ssh .....	12
1.2. hostinfo.ccl File .....	12
1.2.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility .....	14
1.3. Using HP MPI (Message Passing Interface Library) .....	15
1.3.1. Environment Variables .....	15
1.3.2. Interconnect Selection .....	16
1.4. Using SGI MPI on Altix (Linux IA64) .....	17
1.4.1. Configuring Your System for SGI MPI with ANSYS CFX .....	17
1.4.2. Selecting a CFX Start Method .....	19
2. ANSYS CFX Windows Parallel Setup .....	19
2.1. hostinfo.ccl file .....	19
2.1.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility .....	21
2.2. Setting Up HP MPI or MPICH2 for Windows .....	22
2.2.1. Setting Up Distributed Parallel or Remote Access .....	22
2.2.2. Installing the HP MPI/MPICH2 Service and Registering Users on Windows Vista .....	23
2.2.3. Installing the HP MPI/MPICH2 Service and Registering Users on Windows XP .....	23
2.2.4. Enabling Parallel Through a Windows Firewall .....	24
2.3. Setting up and Running CCS 2003/HPC 2008 .....	25

## List of Tables

1. Products with Limited HPC Functionality .....	1
2. Platforms and MPI Software .....	2
3. LS-DYNA MPP MPI Support on Windows and Linux .....	3



---

## Configuring Distributed ANSYS

This section explains how to configure your network/cluster to run Distributed ANSYS. It is important that you follow these steps in order to run successfully in a distributed environment.

For more information on using Distributed ANSYS, see the [Distributed ANSYS Guide](#).

You will need an ANSYS Mechanical HPC (HPC) license for each processor after the first two. For example, if you want to run four processors, you will need two HPC licenses. Some products cannot use all HPC functionality, as noted in the following table.

**Table 1 Products with Limited HPC Functionality**

Product	Distributed ANSYS Capability	SMP Capability	VT Accelerator Capability
ANSYS Academic Teaching Products	Yes	4 processors default limit	Yes
ANSYS LS-DYNA	No	No	No
ANSYS DesignSpace	No	2 processors max	No
ANSYS Professional NLS	No	Yes	Yes
Mechanical APDL Solver	Yes	Yes	Yes

### 1. Prerequisites for Running Distributed ANSYS

Your system must meet the following requirements to run Distributed ANSYS.

- Homogeneous network: All machines must be the same type, OS level, chip set, and interconnects.
- You must be able to remotely log in to all machines, and all machines in the cluster must have identical directory structures (including the ANSYS installation, MPI installation, and on some systems, working directories). Do not change or rename directories after you've launched ANSYS. For more information on files that are written and their location, see *Files that Distributed ANSYS Writes* in the [Distributed ANSYS Guide](#).
- All machines in the cluster must have ANSYS installed, or must have an NFS mount to the ANSYS installation. If not installed on a shared file system, ANSYS must be installed in the same directory path on all systems.
- Distributed ANSYS allows you to use two processors without using any HPC licenses. Additional licenses will be needed to run Distributed ANSYS with more than two processors. Several HPC license options are available. See [HPC Licensing for Distributed ANSYS](#) for more information.
- All machines must have the same version of MPI software installed and running. The table below shows the MPI software and version level supported for each platform.

If you plan to use only the AMG solver in shared-memory ANSYS, MPI software is not required. If you are using only the AMG solver, skip the rest of this document and continue with *Activating Parallel Processing in a Shared-Memory Architecture* in the *Advanced Analysis Techniques Guide*.

## 1.1. MPI Software

The MPI software you use depends on the platform. The following table lists the type of MPI software required for each platform. HP-MPI and Intel MPI are included on the Linux installation media and are installed automatically when you install ANSYS. Similarly, HP-MPI is included on the UNIX installation media and is installed automatically when you install ANSYS on an HP machine. For questions regarding the installation and configuration of the native MPI software versions, please contact your MPI or hardware vendor.

Distributed ANSYS runs on the following platforms:

- HP PA8000 / HP IA-64 (native MPI)
- Intel IA-32 Linux (HP-MPI, Intel MPI)
- Intel IA-64 Linux (HP-MPI, Intel MPI)
- SGI Altix 64-bit Linux (MPI MPT)
- AMD Opteron 64-bit Linux (HP-MPI, Intel MPI)
- Intel Xeon EM64T 64-bit Linux (HP-MPI, Intel MPI)
- Sun - single box, multiple processor only (native MPI only)
- IBM - single box, multiple processor only (native MPI only)
- Windows 32-bit (HP-MPI, Intel MPI)
- Windows 64-bit (HP-MPI, MS MPI, Intel MPI)

**Table 2 Platforms and MPI Software**

Platform	MPI Software	More Information
HP PA8000 64-bit / HP-UX 11.11 (64-bit)	HP-MPI 2.2.5	<a href="http://docs.hp.com/en/B6060-96023/index.html?jumpid=reg_R1002_USEN">http://docs.hp.com/en/B6060-96023/index.html?jumpid=reg_R1002_USEN</a>
HP IA-64 / HP-UX 11.23	HP-MPI 2.2.5	<a href="http://docs.hp.com/en/B6060-96023/index.html?jumpid=reg_R1002_USEN">http://docs.hp.com/en/B6060-96023/index.html?jumpid=reg_R1002_USEN</a>
IBM AIX64 64-bit / AIX 5.3	POE 4.3.2.6	Contact your IBM vendor. Earlier or later versions of POE may not work correctly with Distributed ANSYS.
Sun AMD Opteron 64-bit / Solaris 10	HPC CLUSTER-TOOLS 7.1	<a href="http://www.sun.com/software/products/clustertools/ct7/">http://www.sun.com/software/products/clustertools/ct7/</a>
Linux Intel (including EM64T) and AMD (32-bit and 64-bit)	HP-MPI 2.3.1 Intel MPI 4.0	HP-MPI: <a href="http://docs.hp.com/en/T1919-90018/T1919-90018.pdf">http://docs.hp.com/en/T1919-90018/T1919-90018.pdf</a> Intel MPI: <a href="http://software.intel.com/en-us/articles/intel-mpi-library-documentation/">http://software.intel.com/en-us/articles/intel-mpi-library-documentation/</a>
Windows 32-bit / Windows XP / Windows Vista  Windows 64-bit / Windows XP x64 / Windows Vista x64	HP-MPI 2.0 Intel MPI 4.0	HP-MPI: <a href="http://docs.hp.com/en/BA683-90006/BA683-90006.pdf?jumpid=reg_R1002_USEN">http://docs.hp.com/en/BA683-90006/BA683-90006.pdf?jumpid=reg_R1002_USEN</a> Intel MPI: <a href="http://software.intel.com/en-us/articles/intel-mpi-library-documentation/">http://software.intel.com/en-us/articles/intel-mpi-library-documentation/</a>
Windows HPC Server 2008 x64	Microsoft HPC Pack (MS MPI)	<a href="http://www.microsoft.com/hpc/">http://www.microsoft.com/hpc/</a>

**ANSYS LS-DYNA** If you are running ANSYS LS-DYNA, you can use LS-DYNA's parallel processing (MPP or SMP) capabilities. Use the launcher or the command line method as described in *Starting Distributed ANSYS* in the *Distributed ANSYS Guide* to run LS-DYNA MPP. To run LS-DYNA's MPP version with ANSYS, you need to use the MPI versions listed above for HP and IBM systems. Note that MPP is not available for Sun Solaris systems. For Windows and Linux systems, please see the following table for LS-DYNA MPP MPI support. For more information on using ANSYS LS-DYNA in general, and its parallel processing capabilities specifically, see the *ANSYS LS-DYNA User's Guide*.

**Table 3 LS-DYNA MPP MPI Support on Windows and Linux**

MPI version for DYNA MPP	32-bit Windows	64-bit Windows	32-bit Linux	64-bit Linux
HP-MPI	n/a	X	X	X
MS MPI	n/a	X		

## 1.2. ANSYS Software

To run Distributed ANSYS, you must install ANSYS on all machines in the cluster, or have an NFS mount to the ANSYS installation. Install ANSYS following the instructions in the *ANSYS, Inc. Installation Guide* for your platform. Be sure to complete the installation, including all required post-installation procedures. On Windows systems, you must use the Universal Naming Convention (UNC) for all file and path names for Distributed ANSYS to work correctly.

To run Distributed ANSYS on more than two processors, you must have a sufficient number of HPC licenses to accommodate the desired number of processors. See [HPC Licensing for Distributed ANSYS](#) for more information.

### Installing HP-MPI on Windows

You can install HP-MPI from the Installation launcher by choosing **Install MPI for Distributed Mechanical APDL (ANSYS)**. On the next screen, select **Install HP-MPI for Distributed Mechanical APDL (ANSYS)**. The HP-MPI installation program will start. An HP-MPI installation README file will open simultaneously. Follow the instructions in the README file as you complete the HP-MPI installation.

The instructions for installing HP-MPI are also found on the ANSYS distribution media in the following README files:

Program Files\Ansys Inc\V130\ANSYS\HP-MPI\INSTALL\_HP-MPI\_README.mht

or

Program Files\Ansys Inc\V130\ANSYS\HP-MPI\INSTALL\_HP-MPI\_README.doc

### Installing Intel MPI on Windows

You can install Intel MPI from the Installation launcher by choosing **Install MPI for Distributed Mechanical APDL (ANSYS)**. On the next screen, select **Install Intel MPI for Distributed Mechanical APDL (ANSYS)**. The Intel MPI installation program will start. An Intel MPI installation README file will open simultaneously. Follow the instructions in the README file as you complete the Intel MPI installation.

The instructions for installing Intel MPI are also found on the ANSYS distribution media in the following README files:

Program Files\Ansys Inc\V130\ANSYS\Intel-MPI\INSTALL\_Intel-MPI\_README.mht

or

Program Files\Ansys Inc\V130\ANSYS\Intel-MPI\INSTALL\_Intel-MPI\_README.doc

### **Microsoft HPC Pack (Windows HPC Server 2008)**

You must complete certain post-installation steps before running Distributed ANSYS on a Microsoft HPC Server 2008 system. The post-installation instructions provided below assume that Microsoft HPC Server 2008 and Microsoft HPC Pack (which includes MS MPI) are already installed on your system. The post-installation instructions can be found in the following README files:

Program Files\Ansys Inc\V130\ANSYS\WinHPC\README.mht

or

Program Files\Ansys Inc\V130\ANSYS\WinHPC\README.docx

Microsoft HPC Pack examples are also located in Program Files\Ansys Inc\V130\ANSYS\WinHPC. Jobs are submitted to the Microsoft HPC Job Manager either from the command line or the Job Manager GUI.

To submit a job via the GUI, go to **Start> All Programs> Microsoft HPC Pack> HPC Job Manager**. Then click on **Create New Job from Description File**.

## **2. Setting Up the Environment for Distributed ANSYS**

After you've ensured that your cluster meets the prerequisites and you have ANSYS and the correct version of MPI installed, you need to configure your distributed environment using the following procedure.

1. Obtain the machine name for each machine on the cluster.

#### **Windows:**

Right-click on **My Computer**, left-click on **Properties**, and select the **Network Identification** or **Computer Name** tab. The full computer name will be listed. Note the name of each machine (not including the domain).

#### **UNIX/Linux:**

Type **hostname** on each machine in the cluster. Note the name of each machine. You will need this name to set up the `.rhosts` file, as well as for the **ANS\_ADMIN** utility.

2. (UNIX/Linux only) Set up the `.rhosts` file on each machine. The `.rhosts` file lists all machines in the cluster. The machines should be listed using their complete system name, as taken from **hostname**. For example, an `.rhosts` file for a two-machine cluster might look like this:

```
golinux1.ansys.com jqd  
golinux2 jqd
```

Change/verify `.rhosts` file permissions on all machines by issuing:

```
chmod 600 .rhosts
```

Verify communication between machines via `rsh` or `ssh` (e.g., `rsh golinux2 ls`). You should not be prompted for a password. If you are, check the `.rhosts` permissions and machine names for correctness. For more information on using remote shells, see the man pages for `rsh` or `ssh`.

3. If you want the list of machines to be populated in the Mechanical APDL Product Launcher, you need to configure the `hosts130.ans` file. You can use the **ANS\_ADMIN** utility to configure this file. You

can manually modify the file later, but we strongly recommend that you use **ANS\_ADMIN** to create this file initially to ensure that you establish the correct format.

#### Windows:

**Start >Programs >ANSYS 13.0 >Utilities >ANS\_ADMIN**

Choose **Configuration options**, and then **Configure Cluster** to configure the `hosts130.ans` file.

1. Specify the directory in which the `hosts130.ans` will be configured: Select the **Configure a hosts130.ans file in a directory you specify** option and click **OK**. Enter a working directory. Click **OK**.
2. Enter the system name (from Step 1) in the **Machine hostname** field and click **Add**. On the next dialog box, enter the system type in the **Machine type** drop-down, and the number of processors in the **Max number of jobs/processors** field and click **OK** for each machine in the cluster. When you are finished adding machines, click **Close**, then **Exit**.

An example `hosts130.ans` file where machine1 has 2 processors and machine2 has 4 processors would look like this:

```
machine1 intel 0 2 0 0 MPI 1 1
machine2 intel 0 4 0 0 MPI 1 1
```

#### UNIX/Linux:

`/ansys_inc/v130/ansys/bin/ans_admin130`

Choose **ANSYS/Workbench Configuration**, and then click **Configure Cluster**. Under **Select file to configure**, choose the `hosts130.ans` file to be configured and choose **Configure for Distributed ANSYS**. Click **OK**. Then enter the system name (from Step 1) in the **Machine hostname** field and click **Add**. On the next dialog box, enter the system type in the **Machine type** drop-down, and the number of processors in the **Max number of jobs/processors** field for each machine in the cluster. Click **Add**. When you are finished adding machines, click **Close**.

The `hosts130.ans` should be located in your current working directory, your home directory, or the `apdl` directory.

4. (Windows only) Add `%MPI_ROOT%\bin` to the **PATH** environmental variable on all Windows machines. This line must be in your path for distributed processing to work correctly.
5. (Windows only) Setting up environment variables.

On the head node, where ANSYS and ANSYS Licensing is installed, set the following:

**ANSYS130\_DIR**=C:\Program Files\ANSYS Inc\v130\ansys

**ANSYSLIC\_DIR**=C:\Program Files\ANSYS Inc\Shared Files\Licensing

where `C:\Program Files\ANSYS Inc` is the location of the product install. If your installation location is different than this, specify it instead.

On Windows systems, you must use the Universal Naming Convention (UNC) for all ANSYS environment variables on the compute nodes for Distributed ANSYS to work correctly.

On the compute nodes, set the following:

**ANSYS130\_DIR**=\\head\_node\_machine\_name\ANSYS Inc\v130\ansys

**ANSYSLIC\_DIR** = \\head\_node\_machine\_name\ANSYS Inc\Shared Files\Licensing

For Distributed LS-DYNA:

On the head node and the compute nodes, set **LSTC\_LICENSE** to ANSYS. This tells the LS-DYNA executable to use ANSYS licensing.

Since the LS-DYNA run will use ANSYS licensing for LS-DYNA, you do not need to set **LSTC\_LICENSE\_SERVER**.

6. (Windows only) Share out the `ANSYS Inc` directory on the head node with full permissions so that the compute nodes can access it.

## 2.1. Optional Setup Tasks

The tasks explained in this section are optional. They are not required to get Distributed ANSYS to run correctly, but they may be useful for achieving the most usability and efficiency, depending on your system configuration.

- On UNIX/Linux systems, you can also set the following environment variables:
  - **ANSYS\_NETWORK\_START** - This is the time, in seconds, to wait before timing out on the start-up of the client (default is 15 seconds).
  - **ANSYS\_NETWORK\_COMM** - This is the time to wait, in seconds, before timing out while communicating with the client machine (default is 5 seconds).
  - **ANS\_SEE\_RUN\_COMMAND** - Set this ANSYS environment variable to 1 to display the actual mpirun command issued from ANSYS.

On HP or Linux systems running HP-MPI:

- **MPI\_REMSH** - This is the path to the remote shell (ssh or rsh). Set this environment variable to specify a full path to a remote shell. For example, setting **MPI\_REMSH** = `/usr/bin/ssh` will use ssh instead of the default remote shell (rsh). Note that selecting the **Use Secure Shell instead of Remote Shell** option on the launcher will override **MPI\_REMSH**, if **MPI\_REMSH** is not set or is set to a different location. You can also issue the `- usessh` command line option to use ssh instead of rsh. The command line option will override the environment variable setting as well.
- **MPI\_WORKDIR** - Set this environment variable to specify a working directory on either the master and all nodes, or on specific nodes individually. For more information, see [Files that Distributed ANSYS Writes](#).
- **MPI\_IC\_ORDER** - Set this environment variable to specify the order in which the interconnects on the system are to be used. The interconnects will be tried in the order listed from left to right. If an interconnect is listed in uppercase, no interconnects listed after that one will be tried. If **MPI\_IC\_ORDER** is not set, the fastest interconnect available on the system is used. See the HP-MPI documentation for more details.
- **MPI\_ICLIB\_<interconnect>** - Set this environment variable to the interconnect location if the interconnect is not installed in the default location, especially on AMD Opteron systems using the GM (Myrinet) interconnect:

```
setenv MPI_ICLIB_GM <path>/lib64/libgm.so
```

See the HP-MPI documentation for the specific interconnect names (e.g., **MPI\_ICLIB\_GM**).

- **MPRUN\_OPTIONS** - Set this environment variable to `-prot` to display a grid of interconnects among the systems being used for distributed processing.



- **ANS\_HPMPI\_SRUN\_OPTIONS** - Use this environment variable to define the `-n` (specifies the number of processes and assigns ranks in a block allocation) and `-N` (specifies the number of nodes allocated to the job and assigns ranks in a cyclic allocation) values for `-srun`. For example, to specify two machines with two processes on each node, you would set this environment variable as follows:

```
setenv ANS_HPMPI_SRUN_OPTIONS "-n4 -N2"
```

You cannot specify `-srun` or its options directly on the Distributed ANSYS command line. Valid only for HP XC clusters. See the *HP-MPI User's Guide* for details on `-srun`.

- **ANS\_HPMPI\_PRUN\_OPTIONS** - Use this environment variable to define the `-n` (specifies the number of processes and assigns ranks in a block allocation) and `-N` (specifies the number of nodes allocated to the job and assigns ranks in a cyclic allocation) values for `-prun`. For example, to specify two machines with two processes on each node, you would set this environment variable as follows:

```
setenv ANS_HPMPI_PRUN_OPTIONS "-n4 -N2"
```

You cannot specify `-prun` or its options directly on the Distributed ANSYS command line. Valid only for MPI on an Elan (Quadrics) interconnect. See the *HP-MPI User's Guide* for details on `-srun`.

If you set the **ANS\_HPMPI\_PRUN\_OPTIONS** environment variable and also specify `-np` on the Distributed ANSYS command line, the environment variable will override the `-np` setting. If you have `-prun` on your system and you use the `-np` command line option but do not set this environment variable, Distributed ANSYS will use the specified number of processors on a single machine (i.e., shared-memory parallel mode). If the number of processors specified with `-np` exceeds the number of processors available on a single machine, Distributed ANSYS will then use multiple machines connected via the Elan interconnect until the specified number of processors is reached.

When running jobs on HP XC clusters (`-srun`) or clusters that use the Quadrics interconnect (`-prun`), ANSYS highly recommends that you run from jobs in a writable directory that is visible from all hosts to ensure proper location and execution of files for your job.

- **ANS\_HPMPI\_ELAN\_OFF** - Set this environment variable to ignore the Elan (Quadrics) interconnect.

On Linux systems running Intel MPI:

- Issue the command line option `-usesh` to use `ssh` instead of `rsh`.
- See the Intel MPI reference manual (for Linux) for further information and additional environment variables and their settings: <http://software.intel.com/en-us/articles/intel-mpi-library-documentation/>.

On IBM systems:

- **LIBPATH** - on IBM, if POE is installed in a directory other than the default (`/usr/lpp/ppe.poe`), you must supply the installed directory path via the **LIBPATH** environment variable:

```
export LIBPATH=nondefault-directory-path/lib
```

To verify that these environment variables are set correctly on each machine, run:

```
rsh machine1 env
```

## 2.2. Using the mpitest Program

The `mpitest` program is a ping program to verify that MPI is set up correctly. The `mpitest` program should start without errors. If it does not, check your paths, `.rhosts` file, and permissions; correct any errors, and rerun. Additional communication tests are performed.

When running the `mpitest` programs, you must use an even number of nodes.

### On UNIX/Linux:

For HP-MPI (default), issue the following command:

```
mpitest130 -machines machine1:2
```

For Intel MPI, issue the following command:

```
mpitest130 -mpi intelmpi -machines machine1:2
```

You can use any of the same command line arguments (such as `-machines`) with the `mpitest` program as you can with full Distributed ANSYS.

### On Windows:

Issue the following command to run a local test on Windows using HP-MPI:

```
ansys130 -np 2 -mpitest
```

Use the following procedure to run a distributed test on Windows using HP-MPI:

1. Create a file named `machines` in your local/home directory. Open the `machines` file in an editor.
2. Add your master and slave machines in your cluster. For example, in this cluster of two machines, the master machine is **gowindows1**. List the machine name separately for each core on that machine. For example, if **gowindows1** has four processors and **gowindows2** has two, the `machines` file would look like this:

```
gowindows1
gowindows1
gowindows1
gowindows1
gowindows2
gowindows2
```

3. From a command prompt, navigate to your working directory. Run the following:

```
ansys130 -mpifile machines -mpitest
```

## 2.3. Interconnect Configuration

Low-end hardware, such as interconnects and cables, can reduce the speed improvements you see in a distributed analysis. We typically recommend that you use an interconnect with a communication speed of 200 megabits/second (20 megabytes/second) or higher.

Distributed ANSYS supports the following interconnects. Not all interconnects are available on all platforms; see <http://www.ansys.com/services/ss-platform-support.asp> for a current list of supported interconnects. Other interconnects may work but have not been tested.

- Elan (Quadrics) (recommended)
- InfiniBand (recommended)
- Myrinet (recommended)
- GigE (minimum recommended)
- Ethernet (not recommended)

Interconnects plug into a PCI (Peripheral Component Interconnect), PCI-X (extended), or PCIe (PCI Express) slot on the system. You will need a PCI-X or a PCIe slot for the faster interconnects to accommodate the higher speeds.

Hardware for specific types of interconnects is generally incompatible with other proprietary interconnect types (except Ethernet and GiGE).

Systems can have a network of several different types of interconnects. Each interconnect must be assigned a unique hostname and IP address.

On Windows x64 systems, use the Network Wizard in the Compute Cluster Administrator to configure your interconnects. See the Compute Cluster Pack documentation for specific details on setting up the interconnects. You may need to ensure that Windows Firewall is disabled for Distributed ANSYS to work correctly.

See [Optional Setup Tasks \(p. 6\)](#) for environment variables specific to the Elan (Quadrics) interconnect or refer to the HP-MPI documentation at <http://docs.hp.com>.

## 2.4. Other Considerations

Other factors can also affect your distributed analysis.

- A single 64-bit machine with multiple processors running under shared memory typically works as well as a cluster (multiple machines), and can often be faster. This type of comparison requires evaluating many parameters, such as the network speed, the model being solved, the distributed solver being used, and the hardware involved. The PCG solver performs very little I/O; therefore, running multiple processors on one machine works very well because the multiple processors can communicate rapidly with each other. However, the distributed sparse direct solver (**EQSLV**, SPARSE) runs in out-of-core mode by default, with high I/O. Running multiple processors on a single machine with the distributed sparse solver may not work well if the hard drive cannot quickly handle the multiple I/O requests generated by the distributed processes. For better performance in this case, run the distributed sparse solver using a single processor on multiple machines, assuming that you are using a high-speed network to quickly handle the communication between machines. See the PCG and distributed sparse solver descriptions in the *Basic Analysis Guide* for more information.
- PCG considerations: Review the following guidelines if you will be using the PCG solver in distributed mode. Note that these are not steadfast rules, but rather recommendations to help you get started.
  - The master machine needs more memory than the slave machines.
  - Deploy 64-bit platforms such as Linux on Itanium or AMD chips.
  - Use the /3GB switch on 32-bit platforms on Windows systems.
  - As a broad guideline, use the following formula to get a general idea of memory usage for the DPCG solver. In this formula,  $n$  is the total number of CPU processors used, and MDOF is million degrees of freedom.

Master machine (Machine 0):  $\text{MDOF}_{(\text{maximum})} = \text{Machine}(0) \text{ Memory}_{(\text{GB})} / (0.1 + 1.0/\text{No. of machines})$

Slave machines (Machine 1 -  $n$ ):  $\text{MDOF}_{(\text{maximum})} = \text{Machines } (1 \dots n) \text{ Memory}_{(\text{GB})} * n$

For example, if you have a master machine that is a 32-bit Windows machine with 2.2 GB available RAM, using the /3GB switch, and a total of four machines in the cluster, you could solve a problem up to 6.3 MDOF:

$$\text{MDOF} = 2.2 \text{ GB} / (0.1 + 1 / 4) = 6.3 \text{ MDOF}$$

In this scenario, the slave machines must have 6.3 / 4 or about 1.5 GB of available memory.

### 3. Running a Distributed Job

For information on running Distributed ANSYS after you have your environment configured, see the [Distributed ANSYS Guide](#).

---

## Configuring ANSYS CFX Parallel

If you have purchased the ANSYS CFX-Solver parallel option, you will need to follow the procedures in this chapter to ensure that users can execute ANSYS CFX jobs in parallel. Several parallel communication methods are available that use either PVM (Parallel Virtual Machine) or MPI (Message Passing Interface). Platform-specific versions of MPI are available in some instances.

The ANSYS CFX-Solver parallel option is supported on all platforms that support the CFX-Solver.

### 1. ANSYS CFX UNIX Parallel Setup

The following are required in order for you to execute ANSYS CFX parallel:

- The ANSYS CFX-Solver must be installed on both master and slave nodes.
- To run distributed parallel (where slave processes run on a different host to the master process), remote or secure shell access must be available from the master nodes (systems on which parallel runs will be started) to slave nodes (systems on which the ANSYS CFX-Solver will actually run). See [Setting Up Remote Access on UNIX/Linux](#) (p. 11), below.
- You must have the same user name on all systems.
- The `hostinfo.ccl` file should be set up, as described in [hostinfo.ccl File](#) (p. 12).

#### 1.1. Setting Up Remote Access on UNIX/Linux

Each system that will be used as a slave node must be configured to allow access via remote shell commands from the master node. This can be done globally for all users or on a per-user basis.

Often, networks where `rlogin` and `rsh` or `ssh` are used frequently will already be configured to allow remote access to all users. If this is the case, nothing more needs to be done.

The `CFX5RSH` environment variable is used to select either `ssh` or `rsh` access. The default is `rsh`. If you want to use `ssh`, then set "`CFX5RSH=ssh`" either in the environment or in the `cfx5rc` file (see [Resources Set in cfx5rc Files in the CFX Introduction](#)).

In either case, `rsh` or `ssh`, the remote machine must not prompt for a password when you run remote commands.

##### 1.1.1. Testing Remote Access

You can test remote access using `rsh` for a UNIX/Linux slave node using the command:

```
rsh unixhost echo working
```

---

#### Note

On systems running HP-UX, use "`remsh`" instead of "`rsh`".

You can test remote access using `ssh` using the command:

```
ssh unixhost echo working
```

### 1.1.2. Global Set Up of rsh

This method, which is used to allow remote access for all users, depends on whether NIS is used to maintain netgroups, as well as common password and group databases. If this is not the case, then you should log in to each slave node as root and create a file called `/etc/hosts.equiv` containing a line:

```
<master>
```

where `<master>` is the hostname of the master node. See your system documentation for more information about the use of rsh and the syntax of the `hosts.equiv` file.

If NIS is used to control remote access, then a netgroup must be created for users of ANSYS CFX by the root user on the NIS master server, and a line such as the one below added to `/etc/hosts.equiv` on each slave node by the root user:

```
+@<netgroup>
```

where `<netgroup>` is the name of a netgroup to which users of ANSYS CFX belong. A detailed description of how to configure NIS is beyond the scope of this manual. Please see your system documentation for more information about NIS.

### 1.1.3. Individual User Set Up for rsh

Individual users can create a file called `.rhosts` in their home directory on each slave containing a line:

```
<master> <user>
```

where `<master>` is the hostname of the master and `<user>` is their username on the master. This file should be made readable only by the user, for example, by running:

```
chmod 600 ~/.rhosts
```

### 1.1.4. Set Up of ssh

If you use ssh for remote access, please consult your system documentation on how to set up ssh between machines so that it does not require the user to enter a password.

## 1.2. hostinfo.ccl File

In order to use the Distributed Parallel mode of ANSYS CFX, the file `hostinfo.ccl` must exist in the `<CFXROOT>/config/` directory of the ANSYS CFX installation on the master node and be made readable by all users of the software. This file is a database containing information about the available hosts and where ANSYS CFX has been installed on each of them. The file is used by the ANSYS CFX-Solver when constructing a parallel run.

This file is written using the CFX Command Language. It defines a set of HOST DEFINITION objects, one for each available node. For example:

```
SIMULATION CONTROL:
EXECUTION CONTROL:
  PARALLEL HOST LIBRARY:
    HOST DEFINITION: kangaroo
      Installation Root = /ansys_inc/v%v/CFX
    END
    HOST DEFINITION: wallaby
      Installation Root = /usr/local/cfx
      Host Architecture String = linux-amd64
```

```

        Number of Processors = 16
        Relative Speed = 1.7
    END
    HOST DEFINITION: mypc
        Remote Host Name = my_pc
    END
END
END
END

```

---

## Note

The `SIMULATION CONTROL...END` wrapper is a requirement for Release 12.0 and later. If you manually create your `hostinfo.ccl` file, you must ensure that this wrapper is present.

None of the values for each host are mandatory, and the following empty host definition is perfectly valid:

```

HOST DEFINITION: parakeet
END

```

Host names are limited to the set of valid CCL object names. In particular, they must not contain full stops (.) or underscores (\_) and must start with a letter.

If a `hostinfo.ccl` file does not already exist when ANSYS CFX is installed, one will be created containing the installation host. You can add hosts to the `hostinfo.ccl` file using the `cfx5parhosts` utility, or by modifying the file using a text editor. Individual users may also create their own versions of this file in:

```
~/.hostinfo.ccl
```

which will be used in preference if it exists.

For most installations, it will be necessary to supply only the Installation Root parameter, which should point to the `<CFXROOT>` directory in which ANSYS CFX is installed. On mixed networks, you may find it useful to supply the Number of Processors and/or Relative Speed parameters. A little time may be saved at startup by giving the Host Architecture String parameter explicitly, for example, using the `cfx5parhosts` utility.

---

## Tip

If an individual user wants to use a host that is not present in either the `hostinfo.ccl` in the installation `config` directory or the user's own version of this file, then the user can add this host to the list for a particular run by using the CFX-Solver Manager. However, this would have to be done each time a run is started.

The available parameters for the `hostinfo.ccl` file are as follows:

### Installation Root

This is set to the `<CFXROOT>` installation directory on this host. If it is set to the special string `none`, this indicates that there is no ANSYS CFX installation on the remote host, which can sometimes be useful if only the solver binary is available.

### Host Architecture String

ANSYS CFX will use this value to select the most appropriate solver executable for this node. These strings can be obtained by giving the command `<CFXROOT>/bin/cfx5info -os` on the node in question. When these short `os` values (for example, `linux-amd64`) are given in this position, the generic solver executable will always be used for this host. The string can also be set to the actual architecture of the remote host (for example, `intel_xeon64.sse2_linux2.3.4`), which is determined by giving the command `<CFXROOT>/bin/cfx5info -arch`. If these longer strings are used then CFX could use this

information to select between solvers optimized for specific architectures. However, since there are currently no solvers optimized for specific architectures, this extra information is currently unused.

### Number of Processors

As implied, this is the number of processors on the machine. It is used for display purposes only and can be safely omitted.

### Relative Speed

The Relative Speed is a real number that is used by the ANSYS CFX-Solver when partition sizes are calculated. The ratio of relative speeds of each host is used to determine the size of each partition. As an example, consider a parallel run involving two machines, one with a relative speed of 2.0 and the other with a relative speed of 1.0. The faster machine would work on a partition size twice as large as the slower machine in this case.

The numbers themselves are arbitrary; as a guide you may wish to set 1.0 for a 1GHz processor, 0.75 for a 750 MHz processor, and so on. If a relative speed entry is not entered, a default of 1.0 is assumed for that host.

You can obtain relative speed values using the `cfx5parhosts` utility.

### Remote Host Name

To include hosts in the parallel run with names that contain, for example, underscore characters, you can add the "Remote Host Name" parameter to the HOST DEFINITION with the correct network name for the host and use a readable alias as the name of the object.

### Solver Executable

A solver executable may be explicitly supplied, if necessary. This is usually only required when using Installation Root = none, and is recommended for advanced users only. The following substitutions are made on the string:

%r	root directory of the installation
%p	parallel suffix for the executable
%v	version of ANSYS CFX being run
%o	operating system string
%a	architecture subdirectory specification; for example, linux/double

If it is not supplied, this parameter has the default value `%r/bin/%a/solver%p.exe`

## 1.2.1. Adding Hosts for Parallel Processing with the `cfx5parhosts` Utility

You can add new hosts to ANSYS CFX's database for parallel runs using the `cfx5parhosts` utility:

```
CFXROOT\bin\cfx5parhosts argument list
```

where *argument list* corresponds to one or more of the arguments listed below:

Argument	Description
-add <i>host</i> [, <i>host</i> ,...]	Add information about the named hosts to the file. This assumes that ANSYS CFX is installed in the same directory on each listed host as on the host on which you are running.  <i>host</i> may be specified as [ <i>user</i> @] <i>hostname</i> [:cfx-5 root] if the user name or the ANSYS CFX installation root directory differs from the local host.



Argument	Description
-benchmark	Runs a standard benchmark case, and fills in the Relative Speed for the local host. The benchmark case will usually take less than 5 minutes to run.
-file <i>file</i>	Use the specified file as the <code>hostinfo.ccl</code> file.
-merge <i>file</i>	Merge host information from the named file.
-no-update	After modifying the file, write back the information available without attempting to fill in any missing pieces.
-strict	Used with <code>-update</code> . Normally, hosts which exist on the network but cannot be connected to with <code>rsh</code> or <code>ssh</code> are included in the resulting file with a comment. This switch will exclude these hosts.
-system	Use the system host file. This is the default.
-update	Updates the specified host information file. If any hosts do not have an architecture specified in the existing <code>hostinfo.ccl</code> file (for example, because it was added via the <code>-add</code> switch), it will connect to the host and query it to fill in the Host Architecture String parameter. This is the default behavior. Note that if the Installation Root parameter is incorrect for the host, it will use the standard system commands to guess a generic architecture string. This can happen if you use <code>-add</code> to include a host with a different installation directory than the local one.
-user	Use the per-user host file.

## 1.3. Using HP MPI (Message Passing Interface Library)

Most UNIX/Linux systems support three parallel run modes: PVM, MPICH and HP MPI. HP MPI is the preferred parallel run mode on all the supported ANSYS CFX platforms, except SGI Altix systems; for SGI Altix systems, see [Using SGI MPI on Altix \(Linux IA64\)](#) (p. 17).

HP MPI supports several communication modes including shared memory and networked TCP/IP (Ethernet), luDAPL, VAPI, OpenFabrics (IB), GM-2, MX (Myrinet), Elan4 (Quadrics), QLogic InfiniPath, and EtherFabric (Level 5 Networks).

HP MPI is automatically installed with ANSYS CFX in the `<CFXROOT>/tools` directory. You do not need a license from Hewlett-Packard or Platform to run the ANSYS CFX-Solver using HP MPI.

If there is no CFX installation on any slave node, you must ensure that HP MPI is manually installed on any such node.

HP MPI run modes are selected in the ANSYS CFX-Solver Manager or on the command line (using the `-start-method` command line option) using the "HP MPI Local Parallel" or "HP MPI Distributed Parallel" start method options.

For HP MPI, the `-cpu_bind` option is used by default and binds processes to cores using a cyclic order. This leads to better performance but, on multi-user clusters having scheduling systems that do not enforce exclusive use of cluster nodes, may cause problems because multiple jobs may be bound to the same CPU on the same node. In this case, the `-cpu_bind` argument to `mpirun` should be removed from the HP MPI start methods in the `<CFXROOT>/etc/start_methods.ccl` file.

### 1.3.1. Environment Variables

ANSYS CFX uses the environment variable `CFX5_HPMPI_DIR` to select which HP MPI installation is used. The default setting points to the version installed by ANSYS CFX in the `<CFXROOT>/tools` directory, except

on HP-UX where this variable points to the installation in `/opt/mpi`. If you want to use a different HP MPI version than what is installed then you can install that version in an alternative location and set `CFX5_HP_MPI_DIR` to that location instead of the default.

There are several environment variables that can be used to control HP MPI (documented in the HP MPI documentation) that may provide additional flexibility not directly accessible through the ANSYS CFX startup scripts. If these environment variables are set the startup scripts automatically set them for your CFX runs. Some useful environment variables include:

#### **MPI\_REMSH**

Enables you to change which method is used by HP MPI to spawn jobs on remote machines. The HP MPI default is `ssh`. However, by default the CFX software sets this to be the same as the setting used for `CFX5RSH`, and `rsh` if `CFX5RSH` is not set. This variable is applicable only to Linux and HP-UX.

#### **MPI\_IC\_ORDER**

Enables you to change the default order in which HP MPI searches for interconnects. See the next section.

### **1.3.2. Interconnect Selection**

HP MPI start methods use the default communication mode selection built into HP MPI. The default order in which HP MPI checks for interconnects is OpenFabric, VAPI, UDAPL, ITAPI, PSM, Myrinet MX, Myrinet GM, Elan, TCP/IP.

If you want to force selection of a specific device, then you can either modify the start command in the `etc/start-methods.ccl` file and add the necessary command line flags (for example: `-GM` to force selection of the Myrinet interconnect) or use the appropriate HP MPI environment variables.

Some useful command line options include forcing other interconnect types (`-TCP`, `-HMP`, `-ELAN`, `-ITAPI`, `-VAPI`, `-UDAPL`, and so on), using `prun` on systems with quadrics support (`-prun`), XC support (`-srun`) and setting the subnet you want to use (`-subnet`, sometimes useful for selecting particular interconnects as well if your system is configured that way).

For some Linux-based cluster environments not supported by HP MPI, it is possible that the Myrinet, Infiniband or Quadrics hardware drivers are not located where HP MPI expects. If HP MPI cannot detect the drivers for fast interconnects, it will generally default back to the lowest speed connection that it can detect (usually TCP/IP socket communication).

The following table gives information on the default search path HP MPI uses to find drivers for the various communication interfaces:

<b>Interconnect</b>	<b>1st attempt</b>	<b>2nd attempt</b>	<b>3rd attempt</b>
Infiniband (IB)	Environment variable <code>MPI_ICLIB_ITAPI</code>	<code>libitapi.so</code>	<code>/usr/voltaire/lib/libitapi.so</code>
Myrinet (GM)	Environment variable <code>MPI_ICLIB_GM</code>	<code>libgm.so</code> OR <code>libgm32.so</code>	<code>/opt/gm/lib/libgm.so</code> OR <code>/opt/gm/lib/libgm32.so</code>
ELAN	Environment variable <code>MPI_ICLIB_ELAN</code>	<code>libelan.so</code>	none
UDAPL	Environment variable <code>MPI_ICLIB_UDAPL</code>	<code>libdat.so</code>	none
VAPI	Environment variable <code>MPI_ICLIB_VAPI</code>	Environment variable <code>MPI_ICLIB_VAPIDIR</code>	<code>libmtl_common.so</code> , <code>libmp-ga.so</code> , <code>libmosal.so</code> , <code>libvapi.so</code>

Interconnect	1st attempt	2nd attempt	3rd attempt
Infinipath	Environment variable MPI_ICLIB_PSM	libpsm_infinipath.so.1	/usr/lib64/ libpsm_in- finipath.so.1

For example, HP MPI expects that GM stack shared libraries for Myrinet interconnects are located either in the default library search path or in the /opt/gm/lib (32 bit x86) or /opt/gm/lib64 (64 bit x86) directories. If the shared libraries are not in either location, HP MPI will not take advantage of the high-speed interconnects. To fix this, you will have to copy the shared libraries into the correct location or set the environment variable MPI\_ICLIB\_GM to point to the correct GM stack shared library.

## 1.4. Using SGI MPI on Altix (Linux IA64)

The Altix SGI MPI (Message Passing Toolkit) libraries can be downloaded free of charge by SGI customers by downloading the Message Passing Toolkit (ANSYS CFX was compiled and tested with SGI Message Passing Toolkit version 1.11 for IA64). Additional information and installation instructions are available online at: <http://www.sgi.com/products/software/mpt/>.

### 1.4.1. Configuring Your System for SGI MPI with ANSYS CFX

---

#### Note

You must perform the following steps or SGI MPI will not correctly execute.

The first step is to prepare your SGI cluster by editing the /usr/lib/array/array.conf file on each machine.

1. Search for the entry:

```
array me
    machine localhost
```

and add the following lines:

```
array default
    host1
    host2
    host3
    host4
```

where *default* is the name of the array that will be automatically looked up by ANSYS CFX. You may add an array of a different name, but you must set an environment variable to have that array picked up instead of the default array:

```
setenv MPI_ARRAY array_name
```

where *array\_name* is the name of your array. *host1*, *host2*, etc., correspond to the hostnames you wish to use as nodes in the parallel run. Each host in the list must also contain the same entries in their *arrayd.conf* files. Each host may be a member of any other arrays as well. For further information, please refer to the *arrayd.conf* man pages.

2. Now search for the following section (usually near the bottom of the file):

```
local
user  guest
group guest
port  5434
options setmachid
```

```
# Tell kernel to generate global ASHs
destination array me
```

and change destination array me to point to the array default that you have just created:

```
local
user guest
group guest
options setmachid
# Tell kernel to generate global ASHs
destination array default
```

3. Edit the file `/usr/lib/array/arrayd.auth` and comment out (add a # at the beginning of the line) the line that reads `AUTHENTICATION NOREMOTE`, and remove the # at the beginning of the line that reads `AUTHENTICATION NONE`. More details are available in the `arrayd.auth` man pages.
4. `arrayd` must be running on all hosts. You can check this by entering `ps -ef | grep array`. The process `arrayd` will be found if `arrayd` is running. If the array daemon is already running, then you need to shut it down and restart it for your changes to take effect:

```
/etc/init.d/array stop
/etc/init.d/array start
```

5. If it was not running, then start it using:

```
/etc/init.d/array start
```

`arrayd` must be running on each node in the cluster, and the same version of `arrayd` is also required for each node.

6. In order for other hosts to have access to the subdirectories on the current machine, the `.rhosts` file must contain the following entries:

```
host1 user
host2 user
host3 user
host4 user
```

where `host1`, `host2`, etc. correspond to hostnames and `user` is your username (or the user who will be running SGI MPI).

7. Some environment variables associated with SGI MPI may need to be modified, as follows:

```
setenv MPI_DSM_MUSTRUN
setenv MPI_DSM_PPM <1,2,3 or 4> (see the mpi man pages)
setenv MPI_DSM_CPULIST range-of-cpu-ids
```

The following is an example of the use of *range-of-cpu-ids*:

```
setenv MPI_DSM_CPULIST 4-7,9,12-14
```

and tells `mpirun` to use CPU IDs 4,5,6,7,9,12,13, and 14 for this 8 partition run.

8. Ensure that your input case directory is the same when seen from every node in the cluster (that is, the paths to your working directory are the same over all cluster nodes).
9. The interconnect method is set through environment variables (and uses TCP/IP by default). The following are the available environment variables for the interconnect types:

MPI_USE_XPMEM	For a partitioned IRIX system
MPI_USE_GSN	Use the GSN (ST protocol), IRIX systems
MPI_USE_GM	For myrinet

MPI_USE_HIPPI	For hipp
MPI_USE_TCP	For TCP/IP like 100bT or 1000bT

### 1.4.2. Selecting a CFX Start Method

Once you have properly configured the array daemon, you can start ANSYS CFX jobs using the SGI MPI Local Parallel or SGI MPI Distributed Parallel run modes in the ANSYS CFX-Solver Manager or from the command line using the -start-method command line option.

## 2. ANSYS CFX Windows Parallel Setup

The following are required in order for you to use ANSYS CFX in parallel on Windows platforms:

- ANSYS CFX-Solver must be installed on all nodes to be used for a parallel run.
- To run distributed parallel (where slave processes run on a different host to the master process), you must either configure the parallel host setup properly or set up remote access to slave nodes.
- You must have the same user name on all systems.
- To run distributed parallel, you must install and configure the HP MPI or MPICH2 service and register your user name on all nodes involved in the parallel run. (See [Installing the HP MPI/MPICH2 Service and Registering Users on Windows Vista \(p. 23\).](#))

You must register your user name each time your password changes.

### 2.1. hostinfo.ccl file

In order to use the Distributed Parallel modes effectively, the file `hostinfo.ccl` must exist in the `<CFX-ROOT>/config/` directory of the ANSYS CFX installation on the master node and be made readable by all users of the software. This file is a database containing information about the available nodes and where ANSYS CFX has been installed on each of them. The file is used by the ANSYS CFX-Solver when constructing a parallel run.

The `hostinfo.ccl` file is written using the CFX Command Language. It defines a set of HOST DEFINITION objects, one for each available node. For example:

```
SIMULATION CONTROL:
EXECUTION CONTROL:
  PARALLEL HOST LIBRARY:
    HOST DEFINITION: hostname1
      Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
      Host Architecture String = winnt-amd64
    END # HOST DEFINITION hostname1
  END # PARALLEL HOST LIBRARY
END # EXECUTION CONTROL
END # SIMULATION CONTROL
```

---

#### Note

The `SIMULATION CONTROL...END` wrapper is a requirement for Release 12.0 and later. If you manually create your `hostinfo.ccl` file, you must ensure that this wrapper is present.

None of the values for each host are mandatory, and the following empty host definition is perfectly valid:

```
HOST DEFINITION: parakeet
END
```

Host names are limited to the set of valid CCL object names. In particular, they must not contain full stops (.) or underscores (\_) and must start with a letter.

If a `hostinfo.ccl` file does not already exist when ANSYS CFX is installed, one will be created containing the installation host. You can add hosts to the `hostinfo.ccl` file using the `cfx5parhosts` utility, or by modifying the file using a text editor.

For most installations, it will be necessary only to supply the Installation Root parameter, which should point to the <CFXROOT> directory in which ANSYS CFX is installed. On mixed networks, you may find it useful to supply the Number of Processors and/or Relative Speed parameters. A little time may be saved at startup by giving the Host Architecture String parameter explicitly, for example, using the `cfx5parhosts` utility.

The available parameters are as follows:

### Installation Root

This is set to the <CFXROOT> installation directory on this host. If it is set to the special string `none`, this indicates that there is no ANSYS CFX installation on the remote host, which can sometimes be useful if only the solver binary is available.

### Host Architecture String

This should be set to the actual architecture <arch> of the remote host. ANSYS CFX will use this value to select the most appropriate solver executable for this node. These strings can be obtained by giving the command `<CFXROOT>/bin/cfx5info -os` on the node in question. If the shorter <os> values (for example, `solaris`) are given in this position, the generic solver executable will always be used for this host.

### Number of Processors

As implied, this is the number of processors on the machine. It is used for display purposes only and can be safely omitted.

### Relative Speed

The Relative Speed is a real number that is used by the ANSYS CFX-Solver when partition sizes are calculated. The ratio of relative speeds of each host is used to determine the size of each partition. As an example, consider a parallel run involving two machines, one with a relative speed of 2.0 and the other with a relative speed of 1.0. The faster machine would work on a partition size twice as large as the slower machine in this case.

The numbers themselves are arbitrary; as a guide you may want to set 1.0 for a 1GHz processor, 0.75 for a 750 MHz processor, and so on. If a relative speed entry is not entered, a default of 1.0 is assumed for that host.

You can obtain relative speed values using the `cfx5parhosts` utility.

### Remote Host Name

To include hosts in the parallel run with names that contain, for example, underscore characters, you can add the "Remote Host Name" parameter to the HOST DEFINITION with the correct network name for the host and use a readable alias as the name of the object.

### Solver Executable

A solver executable may be explicitly supplied, if necessary. This is usually only required when using Installation Root = `none`, and is recommended for advanced users only. The following substitutions are made on the string:

%r	root directory of the installation
%p	parallel suffix for the executable
%v	version of ANSYS CFX being run

%o	operating system string
%a	architecture subdirectory specification, for example, linux/double

If it is not supplied, this parameter has the default value %r/bin/%a/solver%p.exe

### 2.1.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility

You can add new hosts to ANSYS CFX's database for parallel runs using the cfx5parhosts utility; this is done by running:

```
CFXROOT\bin\cfx5parhosts argument list
```

where *argument list* corresponds to one or more of the arguments listed below. On Windows Vista, cfx5parhosts must be run with administrator privileges. This is because the hostinfo.ccl file, like all files in Program Files, requires administrative privileges to edit.

Argument	Description
-add <i>host</i> [ <i>host</i> ,...]	Add information about the named host(s) to the file. This assumes that ANSYS CFX is installed in the same directory on each listed host as on the host on which you are running.  <i>host</i> may be specified as [ <i>user</i> @] <i>hostname</i> [:cfx-5 root] if the user name or the ANSYS CFX installation root directory differs from the local host.  To add a set of separately-installed Windows hosts to the <code>hostinfo.ccl</code> file, where the installation may be in a different place on each host, the recommended method is to gather the <code>hostinfo.ccl</code> files created on each host by the installation process, and merge them together using the <code>-merge</code> switch.
-benchmark	Runs a standard benchmark case, and fills in the Relative Speed for the local host. The benchmark case will usually take less than 5 minutes to run.
-file <i>file</i>	Use the specified file as the <code>hostinfo.ccl</code> file.
-merge <i>file</i>	Merge host information from the named file.
-no-update	After modifying the file, write back the information available without attempting to fill in any missing pieces.
-strict	Used with <code>-update</code> . Normally, hosts which exist on the network but cannot be connected to with <code>rsh</code> or <code>ssh</code> are included in the resulting file with a comment. This switch will exclude these hosts.
-system	Use the system host file. This is the default.
-update	Updates the specified host information file. If any hosts do not have an architecture specified in the existing <code>hostinfo.ccl</code> file (for example, because it was added via the <code>-add</code> switch), it will connect to the host and query it to fill in the Host Architecture String parameter. This is the default behavior. Note that if the Installation Root parameter is incorrect for the host, it will use the standard system commands to guess a generic architecture string. This can happen if you use <code>-add</code> to include a host with a different installation directory than the local one.
-user	Use the per-user host file.

## 2.2. Setting Up HP MPI or MPICH2 for Windows

Windows systems support two parallel run modes: MPICH2 and HP MPI. HP MPI is the preferred parallel run mode on Windows platforms.

Both HP MPI and MPICH2 require that you must either configure the parallel host setup properly or set up remote access to slave nodes. See [Setting Up Distributed Parallel or Remote Access \(p. 22\)](#) for details.

In addition, HP MPI and MPICH2 have other requirements as follows.

### For MPICH2 on Windows

---

#### Note

- The MPICH2 service must be installed on all hosts involved in the run.
- The MPICH2 service must be installed and started for MPICH2 local parallel as well as distributed parallel.
- MPICH2 requires user registration only for distributed parallel runs.

### For HP MPI on Windows

---

#### Note

- All nodes must be on the same domain and the user must have an account for that domain.
- The HP MPI service is required only for distributed parallel runs.
- HP MPI requires user registration only for distributed parallel runs.
- HP MPI uses the `-cpu_bind` option by default and binds processes to cores using a cyclic order. This leads to better performance but, on multi-user clusters having scheduling systems that do not enforce exclusive use of cluster nodes, may cause problems because multiple jobs may be bound to the same CPU on the same node. In this case, the `-cpu_bind` argument to `mpirun` should be removed from the HP MPI start methods in the `start_methods.ccl` file in the `<CFXROOT>/etc` directory.

### 2.2.1. Setting Up Distributed Parallel or Remote Access

When the ANSYS CFX distributed parallel starts, it needs to know where to find the CFX-Solver executable on every host and which solver executable it should run (for example, 64-bit or 32-bit). This can be set up in two ways:

- (Preferred method) Explicitly specify the required information:
  1. Ensure that every host used in the distributed parallel run is included in the `hostinfo.ccl` file, including the `Installation Root` and `Host Architecture String` parameters for each host. See [hostinfo.ccl file \(p. 19\)](#) for details on how to set this up.
  2. Set the environment variable `CFX_SOLVE_DISABLE_REMOTE_CHECKS` in your user or site `cfx5rc.txt` file. See [Resources Set in cfx5rc Files in the CFX Introduction](#) for details.
- Have ANSYS CFX determine the required information for itself.



If ANSYS CFX is to determine the required information for itself, then remote- or secure-shell access must be available from the master node (the system on which parallel runs will be started) to slave nodes (systems on which the ANSYS CFX-Solver will actually run). You must be able to execute an rsh or ssh command on the remote host *without typing in your password*. Unless you already have your Windows systems set up to allow this type of access, this method of setting up distributed parallel is not recommended.

### 2.2.2. Installing the HP MPI/MPICH2 Service and Registering Users on Windows Vista

On Windows Vista, you must start a command window and install the HP MPI/MPICH2 service as Administrator:

1. From My Computer, right-click on `C:\WINDOWS\system32\cmd.exe` and select **Run As**. In the **Run As** dialog, select **The following user** and login as Administrator.
2. Enter the username and password information.
  - The username should be entered in full (for example DOMAIN\user).
  - The password information is encrypted and added to the registry.

3. If necessary, change to the disk drive that has the ANSYS software installed. For example, to change to the C drive, enter:

```
c:
```

4. Run the HP MPI/MPICH2 Service installation with Administrator privileges:

```
<install_dir>\v130\CFX\bin\cfx5parallel -install-hpmpi-service
```

or

```
<install_dir>\v130\CFX\bin\cfx5parallel -install-mpich2-service
```

A message appears in the terminal when the installation is complete. On the master host, details of a user with access to all of the hosts in the run must now be set. The username and password information is used during the run to access the hosts.

5. Enter either:

```
<install_dir>\v130\CFX\bin\cfx5parallel -register-hpmpi-user
```

or

```
<install_dir>\v130\CFX\bin\cfx5parallel -register-mpich2-user
```

6. Start the service; enter either:

```
<install_dir>\v130\CFX\bin\cfx5parallel -start-hpmpi-service
```

or

```
<install_dir>\v130\CFX\bin\cfx5parallel -start-mpich2-service
```

### 2.2.3. Installing the HP MPI/MPICH2 Service and Registering Users on Windows XP

As a user with administrative privileges:

1. From the launcher, select **Tools > Command Line**.

## 2. Enter either:

```
cfx5parallel -install-hmpi-service
```

or

```
cfx5parallel -install-mpich2-service
```

A message appears in the terminal when the installation is complete. On the master host, details of a user with access to all of the hosts in the run must now be set. The username and password information is used during the run to access the hosts.

## 3. From the command line invoked from the launcher, enter either:

```
cfx5parallel -register-hmpi-user
```

or

```
cfx5parallel -register-mpich2-user
```

## 4. Enter your username and password information.

- If prompted, the username should be entered in full (for example DOMAIN\user).
- The password information is encrypted and added to the registry.

## 5. From the command line invoked from the launcher, enter either:

```
cfx5parallel -start-hmpi-service
```

or

```
cfx5parallel -start-mpich2-service
```

## 2.2.4. Enabling Parallel Through a Windows Firewall

To enable Parallel Processing through a Windows firewall:

1. Click on **Start** and select **Control Panel**.
2. On the **Control Panel** dialog, double-click **Security Center**.
3. Click **Windows Firewall**.
4. On the **Windows Firewall** dialog, click the **Exceptions** tab and then click **Add Program**.
5. Browse to the following programs, which are in your CFX\_ROOT:
  - For HP MPI:
    - bin\<CFX5OS>\solver-hmpi.exe
    - bin\<CFX5OS>\double\solver-hmpi.exe
    - tools\hmpi-win-1.1-1\sbin\hmpiwinservice32.exe
    - tools\hmpi-win-1.1-1\bin\mpirun.exe
    - tools\hmpi-win-1.1-1\bin\mpid.exe
  - For MPICH2:
    - bin\<CFX5OS>\solver-mpich2.exe
    - bin\<CFX5OS>\double\solver-mpich2.exe

- tools\mpich2-1.0.7\<CFX5OS>\bin\mpiexec.exe
- tools\mpich2-1.0.7\<CFX5OS>\bin\smpd.exe

After each selection, click **OK** to add that program to the list of programs that are permitted to pass through the Windows firewall.

6. When you are finished, click **OK** to close the **Windows Firewall** dialog, then close the **Windows Security Center** and the **Control Panel**.

## 2.3. Setting up and Running CCS 2003/HPC 2008

To set up running CCS 2003/HPC2008 steps must be taken so that:

- The installed software is on a shared location that can be accessed by all hosts
- The working directory is a shared location
- A mechanism is provided so that all local paths can be properly resolved.

For running ANSYS CFX with CCS 2003/HPC 2008, this can be accomplished by following the steps below (terms in angle brackets < > should be replaced with the required entry):

1. Install ANSYS Licensing, ANSYS Workbench and ANSYS CFX on the head node.
2. Share the installation directory. For example, on a typical installation share C:\Program Files\ANSYS Inc as \\<HeadNodeName>\ANSYS Inc where <HeadNodeName> is the name of the head node.
3. Install the ANSYS prerequisites on all of the cluster nodes. You can do this either by:
  - Executing the following commands directly on each node:
 

```
\\<HeadNodeName>\ANSYS Inc\vl30\prereq\vc redistrib_x64_SP1.exe
\\<HeadNodeName>\ANSYS Inc\vl30\prereq\vc redistrib_x86_SP1.exe
\\<HeadNodeName>\ANSYS Inc\vl30\prereq\2008vc redistrib_x64.exe /qn
```
  - or by using the clusrun command on the headnode to execute these commands on all the nodes (refer to your Windows CCS or HPC documentation for details).
4. Share the working directory on the head node. For example, share C:\Users\<UserName> as \\<HeadNodeName>\<UserName> where <UserName> is the user name.

Alternatively, share the working directory of the submitting machine. For example share C:\Users\<UserName> on the submitting machine as \\<SubmitHostName>\<UserName> where <SubmitHostName> is the name of the submitting machine.

5. On the submitting machine, create %USERPROFILE%\cfx\cfxccs\_options.txt with the following content to define the required CCS options:

```
PATHMAP=C:\Program Files\Ansys Inc;\\<HeadNodeName>\Ansys Inc
PATHMAP= C:\Users\<UserName>;\\<HeadNodeName>\<UserName> or
PATHMAP=C:\Users\<UserName>;\\<SubmitHostName>\<UserName> if the working
directory has been shared from the submitting machine.
CLUSTERHOST=<HeadNodeName> to be used when submitting jobs from machines other than
the headnode.
ACCOUNT=<OtherUserDomain>\<OtherUserName> to be used when submitting jobs using
different credentials, where <OtherUserDomain> and <OtherUserName> are the domain and
user names of another user, respectively.
```

PROCESSORSPERSOLVER=2 an optional setting (default setting is 1) that allocates the number of cores per partition. This is typically used on hosts that are limited by memory bandwidth such as Xeon-based machines.

6. Set up ANSYS Workbench for a network as described in [Network Installation and Product Configuration](#) (p. 18).

To submit a job:

1. Start the CFX-Solver Manager (either using CFX standalone or from ANSYS Workbench) on the submitting machine using the software installed on the headnode.
2. Ensure that the **Run Mode** on the **Run Definition** tab of the CFX-Solver Manager is set to Submit to Windows CCS or HPC Queue. Otherwise, set up the CFX-Solver Manager as normal.
3. Click **Start Run** on the CFX-Solver Manager to start the simulation. The first time a simulation is run, supply the required credentials that are prompted (this prompt may appear behind the CFX-Solver Manager window). You may elect to have these credentials saved to avoid repeated prompting of these credentials.

---

### Note

1. For simulations making use of User Fortran, you must ensure that the Fortran is linked against the `solver-msmpi.lib`. This is done by making the appropriate changes to `cfx5mkext.ccl`.
2. %USERPROFILE% is typically C:\Documents and Settings\<Username>\ on XP, or C:\Users\<UserName> on Vista or Server HPC 2008.
3. To use on ANSYS Workbench with Parameters and Design Points, start the CFX-Solver Manager ensure that the **Run Mode** on the **Run Definition** tab of the CFX-Solver Manager is set to Submit to Windows CCS or HPC Queue, set the number of processes, then click **Save Settings**. When the user clicks **Update all Design Points** each parameter or design point will be solved on the cluster.