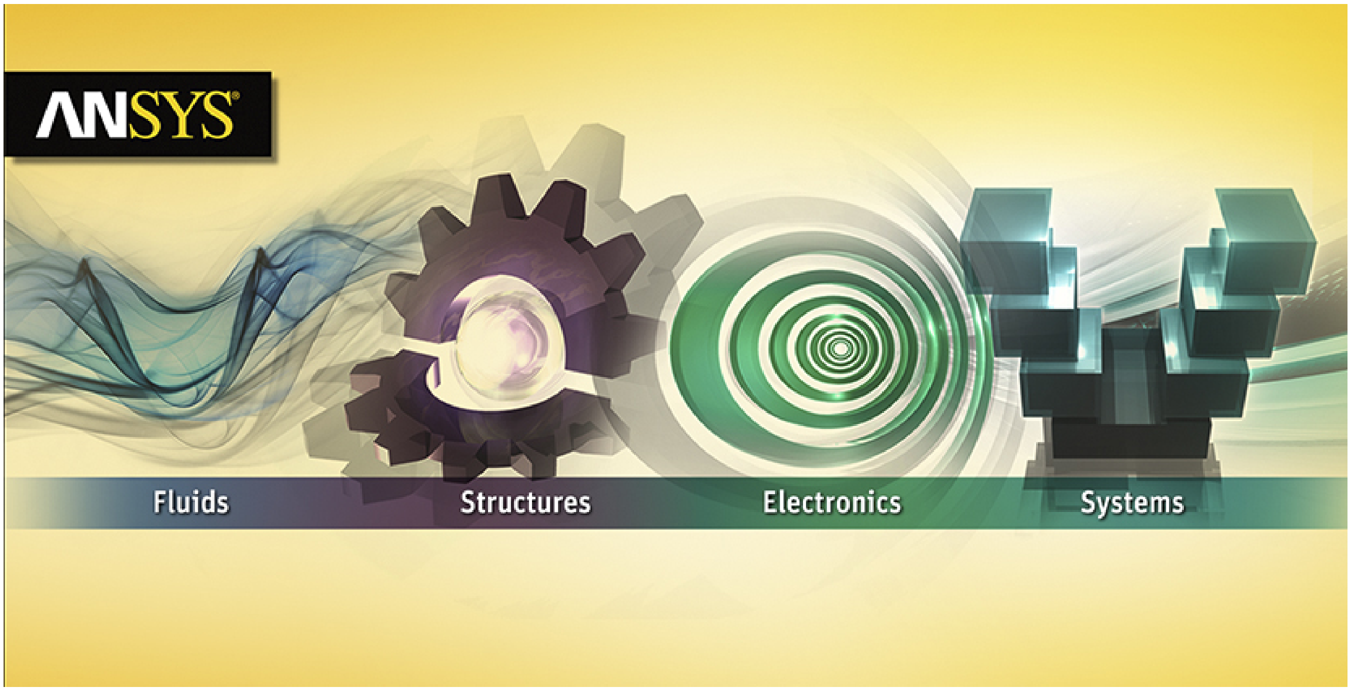


Installation Guide for Linux

ANSYS, Inc.
Southpointe
2600 ANSYS Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 16.2
July 2015
000408
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The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superseded by a newer version of this guide. This guide replaces individual product installation guides from previous releases.

Copyright and Trademark Information

Disclaimer Notice

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

U.S. Government Rights

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

List of Tables

Chapter 1: Installation Prerequisites for Linux

This document describes the steps necessary to correctly install and configure all ANSYS, Inc. products on Linux platforms for Release 16.2. These products include:

- ANSYS Structural Mechanics
  - ANSYS Mechanical Products (includes Mechanical APDL and Mechanical, where supported)
  - ANSYS Customization Files for User Programmable Features
- 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 ICEM CFD
- ANSYS Additional Tools
  - ANSYS Composite PrepPost
  - ANSYS Icepak (includes ANSYS CFD-Post)
  - Remote Solve Manager Standalone Services
- ANSYS Geometry Interfaces
  - ACIS
  - CATIA, Version 4
  - CATIA, Version 5
  - NX
  - Parasolid

Notes Not all products may be available on all Linux platforms. Please see the remainder of this document for detailed information on which products are available on which platforms.

At Release 16.2, ANSYS BladeGen, Vista CCD, Vista CPD, Vista RTD, and Vista AFD are not supported on Linux platforms.

ANSYS Workbench and ANSYS EKM Desktop are installed by default as product components to most ANSYS, Inc. products. ANSYS Workbench is not installed as part of the products under ANSYS Additional Tools. ANSYS Workbench includes the following applications:

- DesignModeler
- Design Exploration
- Meshing
- Remote Solve Manager
- Fluent Meshing
- FE Modeler
- EKM Client

Important Notice

If you wish to run multiple releases of ANSYS, Inc. software, you **MUST** install them chronologically (i.e., Release 16.1 followed by Release 16.2). If you install an earlier release after installing Release 16.2, 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 16.2, you **MUST** uninstall Release 16.2, then re-install the releases in order.

1.1. System Prerequisites

ANSYS, Inc. Release 16.2 products are supported on the 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/Support/Platform+Support. (This URL is case-sensitive.)

Table 1.1: Supported Linux Platforms

Platform	Processor	Operating System	Platform architecture (directory name)	Availability
Linux x64	EM64T/Opteron 64	Red Hat Enterprise Linux 6.5 through 6.6, SUSE Linux Enterprise 11 SP2 and SP3	linx64	Download / USB

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.

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 standalone (some CAD systems may require the integrated ANSYS Workbench Reader)

Table 1.2: CAD Support by Platform

	Linux x64
CATIA 4.2.4	A , I
CATIA V5–6R2014	W, I
GAMBIT 2.4	W, I
Parasolid 26.0	A, W, I
ACIS 25 1	A, W, I
NX 8.5	A, W, I
NX 9.0	A, W, I
NX 10.0	A, W, I
STEP AP203, AP214	W,
IGES	A 2, W, I 3
GEMS	I
Rhinoceros	I

1. For ANSYS ICEM CFD standalone, ACIS 18.0.1 is the supported version for all platforms.

2. MAPDL supports 5.1 by default, but 5.2 is also supported if the IOPTN command is used.

3. IGES Versions 4.0, 5.2, and 5.3 are supported.

1.2. Disk Space and Memory Requirements

You will need the disk space shown here for each product for installation and proper functioning. 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.

Product	Disk Space
ANSYS Mechanical APDL	10.0 GB
ANSYS Autodyn	7.1 GB
ANSYS LS-DYNA	7.6 GB
ANSYS CFX	9.6 GB
ANSYS TurboGrid	8.1 GB
ANSYS Fluent	9.5 GB
ANSYS Polyflow	9.2 GB

Product	Disk Space
ANSYS ICEM CFD	2.8 GB
ANSYS Icepak	5.5 GB
ANSYS CFD Post only	8.0 GB
ANSYS Geometry Interfaces	302 MB
ANSYS Composite PrepPost	2.9 GB

Memory Requirements You must have a minimum of 8 GB of memory to run product installations; 16 or 32 GB of memory is recommended.

1.3. Requirements for the GPU Accelerator in Mechanical APDL

Your system must meet the following requirements to use the GPU accelerator capability in Mechanical APDL:

- The machine(s) being used for the simulation must contain at least one supported nVIDIA GPU card or one Intel Xeon Phi coprocessor. The following cards are supported:

  nVIDIA Tesla series (any model)
  nVIDIA Quadro 6000
  nVIDIA Quadro K5000
  nVIDIA Quadro K6000
  Intel Xeon Phi 7120
  Intel Xeon Phi 5110
  Intel Xeon Phi 3120

- For nVIDIA GPU cards, the driver version must be 331.75 or newer.
- For Intel Xeon Phi coprocessors, the driver version must be "MPSS 3.4" or newer.

Note

Mechanical APDL assumes that the Intel Xeon Phi coprocessor driver is installed in /opt/intel/mic/coi. If the installation is located in a different folder, set the **ANS_MIC_LIB_PATH** environment variable to the path where the Xeon Phi driver is installed (for example, /*mypath*/mic/coi).

1.4. Additional Hardware and Software Requirements

- Intel 64 / AMD64 system with a supported operating system version installed
- 8 GB of RAM
- 128 GB free on the hard drive is recommended
- TCP/IP for the license manager (see the ANSYS, Inc. Licensing Guide for more information on TCP/IP)
- 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.

> **Note**
>
> – A number of cosmetic display issues have been reported when running the installation program on Linux platforms using AMD graphics cards. These issues do not affect the installation functionality.
>
> – Use of Ultra High Definition (4K) graphics cards cause a number of cosmetic display issues (including enlarged or reduced text and incorrect positioning of option labels). These issues do not affect the installation functionality.

- X11, OpenGL graphics libraries
- Mesa-libGL (OpenGL) is required to run data-integrated ANSYS Workbench applications such as Mechanical.
- For Fluent, CFX-Pre, and CFD-Post, a three-button mouse is required to access all available functionality.
- PDF reader software is required to read the installation guides and other user documentation.
- CFX-RIF requires 32-bit X11 libraries when running on a 64-bit machine.

1.5. Third-Party Software and Other Security Considerations

The following third-party products are used as part of the installation process. In order for the installation to work properly, you must allow access to these products.

Product Name	Executable Name
Tcl	tclsh
Tk	wish
Perl	perl
GNU gzip	gzip
GNU tar	tar

ANSYS, Inc. products may have softlinks that require 777 permissions. In addition, the following third-party products are known to contain softlinks that require 777 permissions:

- CPython
- GCC
- HPMPI
- IMPI
- INTELMPI
- MainWin

- Mono
- MPICH
- OPENMPI
- PCMPI
- Perl
- Python
- Qt
- Qwt

Chapter 2: Platform Details

Linux

For ALL 64-bit Linux platforms, OpenMotif, 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 SP2 or SP3. After installing the SP2/SP3 updates, you must also install OpenMotif and the prerequisites from the SLES11 SDK DVD, as well as the OpenMotif22 packages (recommended packages are openmotif22-libs-2.2.4-138.18.1.x86_64.rpm and openmotif22-libs-32bit-2.2.4-138.18.1.x86_64.rpm from SLED11 SP1 x86_64 distribution. You may need to use "rpm -iv -force" to install these).

Red Hat Enterprise Linux 6 Red Hat Enterprise Linux 6 base install requires patch kernel-2.6.32-71.14.1.el6.x86_64.rpm and kernel-devel-2.6.32-71.14.1.el6.x86_64.rpm.

In addition, you need to install the following libraries:

- libXp.x86_64
- xorg-x11-fonts-cyrillic.noarch
- xterm.x86_64
- openmotif.x86_64
- compat-libstdc++-33.x86_64
- libstdc++.x86_64
- libstdc++.i686
- gcc-c++.x86_64
- compat-libstdc++-33.i686
- libstdc++-devel.x86_64
- libstdc++-devel.i686
- compat-gcc-34.x86_64
- gtk2.i686
- libXxf86vm.i686
- libSM.i686

- libXt.i686
- xorg-x11-fonts-ISO8859-1-75dpi.noarch

Red Hat no longer includes the 32-bit libraries in the base configuration so you must install those separately.

Additional requirements for Intel, AMD Opteron, and EM64T Linux systems are detailed below.

ANSYS, Inc. License Manager Linux x64 systems running the ANSYS, Inc. License Manager require the Linux Standard Base (LSB) 3.0 package.

CATIA V4 with Mechanical APDL To run CATIA V4 with Mechanical APDL on Linux 64-bit platforms, you must have the following Linux 32-bit i686 libraries:

- glibc
- libstdc++

ANSYS Workbench 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 Behavior> Advanced**.
3. Change **Focus Stealing Prevention Level** to None.
4. Click **Apply**.

If you are running on KDE 4 or if the `kcontrol` command does not exist, use System Settings to set the focus stealing prevention setting level to "None":

1. Use the `systemsettings` command to launch the System Settings.
2. In the System Settings window, select **General> Window Behavior> Focus**.
3. Change **Focus Stealing Prevention Level** to None.
4. Click **Apply**.

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
```

Mechanical, Meshing, DesignModeler, and FE Modeler Applications If you are using a localized operating system (such as French or German), you must set the `mwcontrol` VisualMainWin control on any machines running these applications in order for these applications to recognize the correct numerical format. ANSYS Workbench must already be installed before setting this control.

First, you need to ensure that the `/v162/aisol/WBMWRegistry/` directory has write permissions. From the `/v162/aisol/` directory, issue the following command:

```
chmod -R 777 WBMWRegistry/
```

Then, use the following procedure to set `mwcontrol` for your locale:

1. cd to <*wb_install directory*>`/v162/aisol`

2. Issue the following command:

   ```
   ./.workbench -cmd mwcontrol
   ```

3. On the **MainWin Control Panel**, select **Regional Settings**.

4. Select the **Regional Settings** tab.

5. Change the language in the dropdown to match the language you want to use.

6. Check the **Set as system default locale** option.

7. Click **Apply** to accept the changes, and then click **OK** to dismiss the **Change Regional Settings** notification.

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, the following should be added to the `/etc/security/limits.conf` file:

```
* hard memlock unlimited
* soft memlock unlimited
```

The need for increasing the limits may be indicated by the following error message with Platform MPI:

```
fluent_mpi.16.0.0: Rank 0:1: MPI_Init: ibv_create_cq() failed
fluent_mpi.16.0.0: Rank 0:1: MPI_Init: Can't initialize RDMA device
fluent_mpi.16.0.0: Rank 0:1: MPI_Init: MPI BUG: Cannot initialize RDMA protocol
libibverbs: Warning: RLIMIT_MEMLOCK is 32768 bytes.
This will severely limit memory registrations
```

When Fluent is launched by a scheduler (like PBSPro, SGE, etc.), these limits may be reset by the scheduler. You may check running the “limit” command within and without the scheduler to compare. Any differences indicate the issue above.

If you use PBSPro, add **`ulimit -l unlimited`** directly in the `pbs_mom` startup script, and let PBS reload the `pbs_mom`.

If you use SGE, you must change the startup script `/etc/init.d/sgeexecd` to include the **`ulimit -l unlimited`** command as shown in the following example:

```
-- BEGIN --
if [ "$startup" = true ]; then
```

```
# execution daemon is started on this host!
echo " starting SGE_execd"
exec 1>/dev/null 2>&1
ulimit -l unlimited
$bin_dir/SGE_execd
else
-- END --
```

Once this modification is in place, restart SGE to set the correct memory limits on the SGE daemon and any invoked processes.

Using Mechanical APDL 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, the following should be added to the `/etc/security/limits.conf` file:

```
* hard memlock unlimited
* soft memlock unlimited
```

This behavior has been observed on Intel MPI 4.1.3, but might also occur on Platform MPI 9.1.2.1.

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:

SUSE 11: libstdc++33

Using ANSCUSTOM If you use ANSCUSTOM to link your own version of ANSYS Release 16.2 on a SUSE SLES 11.x computer, 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 computer.

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 linx64 systems, the ANSYS Release 16.2 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 16.2, 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 Linux, you may need the following:

```
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 for All Linux Versions (p. 11). The ANSYS solver is built on Red Hat Enterprise Linux AS release 4 (Update 5).

For ANSYS Workbench and ANSYS Autodyn, you may need to increase the stack size; we recommend setting it to 1 GB. Add the following to your configuration file:

For the Bourne (bash) shell:

```
ulimit -s 1024000
```

For the C (csh) shell:

```
limit stacksize 1024000
```

Other shells may have different settings; please refer to your shell documentation for specific details.

Intel Xeon EM64T

ANSYS was built and tested on a generic Intel EM64T system running Red Hat Enterprise Linux AS release 5.3.

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 hyperthreading option off.

2.1. Utilizing CPU Hyperthreading Technology with ANSYS CFD Solvers

Hyperthreading technology uses one processor core to run more than one task at a time. ANSYS does not recommend using hyperthreading technology in conjunction with ANSYS CFD Solvers (Fluent, CFX and AIM Fluids). 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 hyperthreading option off.

2.2. Compiler Requirements for Linux Systems

Table 2.1: Compiler Requirements for All Linux Versions

Mechanical APDL, ANSYS Workbench Compilers*	CFX Compilers*	Fluent Compilers*	AUTODYN Compilers*
Intel 14.0.3 (FORTRAN, C, C++)	Intel 14.0.3 (FORTRAN, C, C++)	GCC 4.6.1 (typically installed as part of the operating system)	Intel 14.0.3 (FORTRAN, C, C++)

* Compilers are required only if you will be using User Programmable Features (UPF), User Defined Functions (UDF), or other customization options.

2.3. Select Your Installation

The next step is to select your installation type. Please select the option below that matches your installation.

- Installing the ANSYS Software for a Stand-alone Linux System (p. 13)
- Installing the ANSYS Products and the License Server on Different Linux Machines (p. 21)

Chapter 3: Installing the ANSYS Software for a Stand-alone Linux System

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 install as non-root; however, if you are not logged in as root, 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, you must install all releases into a common directory to ensure license manager compatibility and availability among releases and products.

If you did not use the `/ansys_inc` symbolic link, you must replace all references to `/ansys_inc/v162` or `<install_dir>` with the actual installation path you used.

Before You Begin

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

- An account on the ANSYS Customer Portal. If you do not have an account, you may register at https://support.ansys.com/portal/site/AnsysCustomerPortal to receive your own account.
- Your license file from ANSYS, Inc., saved to a temporary directory. For more information, see Registering the License Server (p. 20).
- Open port numbers for both the FLEXlm and ANSYS Licensing Interconnect. Defaults are 1055 and 2325, respectively. To verify that these port numbers are available, open a command line and enter the following command:

  ```
  netstat -a -t
  ```

  You will see a list of active ports. If 1055 and 2325 are listed, they are already in use and cannot be used for ANSYS, Inc. licensing. In this case, you will need to specify different port numbers where indicated later in this installation.
- Your local machine's name, to specify as the license server.

You should also verify that you are running on a supported platform. ANSYS, Inc. products support 64-bit Linux systems running Red Hat 6 (6.4 and 6.5) and SUSE Linux Enterprise 11 (SP2 or SP3).

Verify that you have sufficient disk space to download, uncompress, and install the products you will be installing. Approximate disk space requirements for each product are shown in Disk Space and Memory Requirements (p. 3).

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

This section is divided into four sets of instructions:

- Product Download Instructions: This set of instructions describes the download and extraction process.
- Product Installation with Client Licensing: This set of instructions describes the product installation, including the client licensing portion.
- License Manager Installation: This set of instructions describes the license manager installation.
- Post-Installation Procedures for All Products: This is a set of instructions that describes any configuration steps that may be required for the various products.

Both the product and the license manager will be installed on the same machine. *You must complete both the client licensing portion and the license manager installation in order to run ANSYS, Inc. products.*

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 `getFLEXid` script on each machine first. This script will output the correct platform name for each machine on which it is run. This script can be obtained from the ANSYS website, by clicking **Support>Licensing>Capture License Server Info**.

3.2. Product Download Instructions

To download the installation files from our website, you will need to have a current technical support agreement.

Depending on the product/platform combination(s) you choose, you may need to download multiple files.

1. From the Customer Portal, https://support.ansys.com/portal/site/AnsysCustomerPortal, click **Downloads > Current Release**.

   Step 1 of the ANSYS Download Center Select Full Products page is displayed.

2. Select your installation operating system (**Windows x64** or **Linux x64**).

3. Select the type of files you wish to download:

   - **Primary Packages**: Individual full packages for the primary ANSYS products.
   - **ISO Images**: ISO images for the DVD installation.

4. Click the appropriate download option.

5. Select your desired download directory and click **Save**.

6. Repeat this process for each download file.

7. To download **Add-On Packages**, **Tools** or **Academic Packages**, click the + to the right of the appropriate product group title to display the download options and download as required.

8. After all downloads have been completed, uncompress each package using standard uncompression utilities for your specific platform. We strongly recommend that you extract the files into new, temporary directories.

- Continue with the steps described in Product Installation (p. 16).

3.3. Installing from a USB Drive

Insert the USB drive into an appropriate USB slot on your computer. Navigate to the root directory of the USB drive and Run ./INSTALL. Continue with the steps described in Product Installation (p. 16).

3.4. Mounting the DVD Instructions for DVD Installations (Linux x64 Only)

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.

Note

Typically, a Linux system will mount a local DVD for you under /media/ANSYS162.

For a locally-mounted DVD installation, issue the following commands:

```
mkdir dvdrom_dir
mount -t iso9660 /dev/cdrom dvdrom_dir
```

If the target machine does not have a DVD reader, first follow the steps for locally-mounted DVD, and then follow the procedure below for remotely-mounted DVDs:

Remotely-Mounted DVD Procedure

1. Add the *`dvdrom_dir`* directory to the `/etc/exports` file on the machine with the DVD device. A sample `/etc/exports` entry is:

   ```
   /dvdrom_dir *(ro)
   ```

   or

   ```
   /dvdrom_dir (ro)
   ```

2. Run **exportfs** to export the *`dvdrom_dir`* directory:

   ```
   exportfs –a
   ```

   Check the manual page for 'exports' for the correct syntax, as different Linux versions can have different syntax.

3. Log on to the machine where you wish to install ANSYS, Inc. products and issue the following commands:

   ```
   mkdir dvdrom_dir2
   mount -t nfs Host:cdrom_dir dvdrom_dir2
   ```

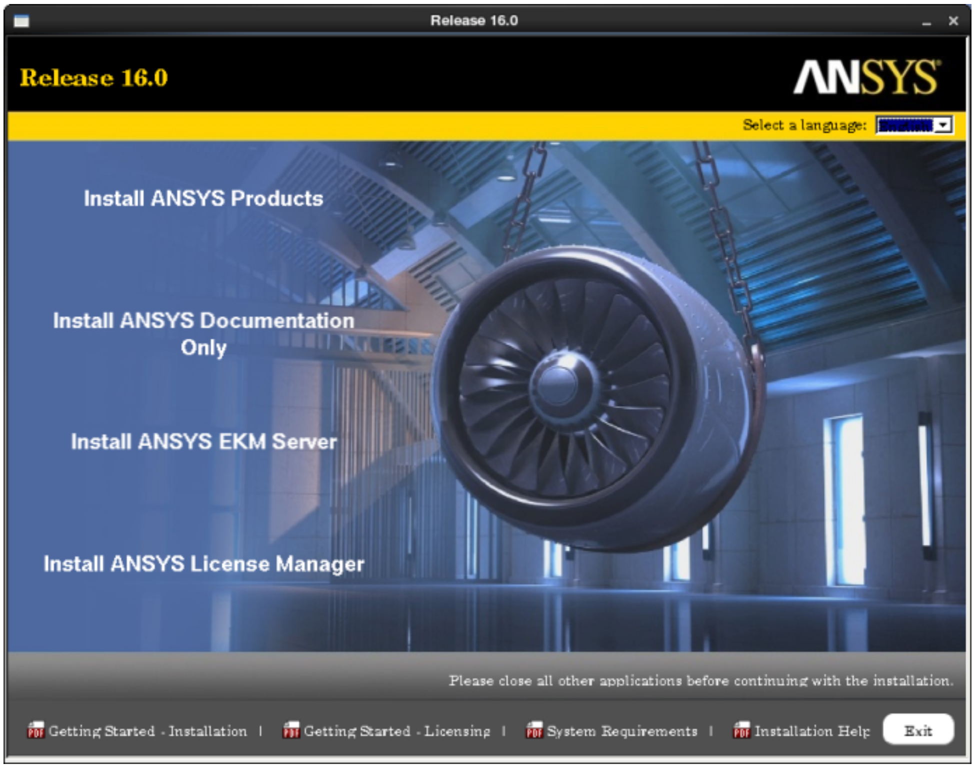
   where *`Host`* is the hostname of the machine where the DVD device is located.

   Run **`man exports`** for more information.

3.5. Product Installation

1. Navigate to the directory where you extracted the files. Run `./INSTALL`. If you downloaded the installation package, this file will reside in the directory where you untarred the downloaded files. If you are running from a DVD, this file will reside in the top level of the DVD.

2. The ANSYS, Inc. Installation Launcher appears.



   From the options along the left side of the launcher you can install ANSYS products, Documentation Only, ANSYS EKM Server, and the ANSYS License Manager. You can access the installation guide for the ANSYS EKM Server from the **Downloads** page on the Customer Portal.

   The ANSYS, Inc. Quick Start Installation Guide, ANSYS, Inc. Quick Start Licensing Guide, System Requirements Guide and complete Installation Help Guide can be accessed through the options located along the bottom of the launcher.

   Select the language you want to use from the drop-down menu in the upper right corner. English is the default.

3. Click the **Install ANSYS Products** option.

4. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, select **I Agree**. Click **Next**.

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

   The **Symbolic Link** option is available only if you are installing as root and is enabled (checked) by default. If you choose to disable (uncheck) the symbolic link or are installing as a non-root user, substitute the directory path where you installed the product for all subsequent occurrences of `/ansys_inc` in this guide.

We strongly recommend that you use the **Symbolic Link** option. If you do not use this option, you must 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**.

6. If this is a first time installation, you are prompted to enter your license server specification. If you already have an existing license server specification file, you will not see this window and proceed directly to the next step.

   Enter your ANSYS Licensing Interconnect port number and your ANSYS FlexNet port number. Defaults are provided and will work in most cases. You may need to check with your IT department to confirm that the default port numbers are valid or to get different port numbers if necessary.

   Specify the hostname for your license server machine(s).

   Click **Next**.

7. All products available in the installation package(s) you downloaded are listed.

   The installation program attempts to query your license server to pre-select your installation options. If the query is successful, the following message is displayed:

   Your installation options were pre-selected based upon information from your license server.

   You can select or deselect any combination of products. ANSYS Workbench is automatically installed with most ANSYS, Inc. products; there is no individual product selection for ANSYS Workbench.

   If you select NX or Catia V5, you will have additional installation steps. This installation assumes you are not installing these CAD packages. Deselect these options before continuing.

   By default, the **Install Documentation** option is enabled (checked). When enabled, help documentation is included as part of the installation process for all products selected. No help documentation is included with the installation if this box is disabled (unchecked).

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

   ### Note

   On a first time installation, if you chose to install any ANSYS Geometry Interfaces, ensure that you have also selected at least one ANSYS, Inc. product as part of the installation. Installing an ANSYS Geometry Interface without an underlying ANSYS, Inc. product on a first time installation may cause installation errors.

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

The dates on the licensing files being installed are compared to any that may already exist on your machine. (This may take a few moments.)

8. A summary screen appears listing your installation selections. Please review this list carefully to verify that it is correct. When you are sure all selections are correct, click **Next** to begin the installation.

9. The installation progress screen displays a status bar towards the bottom of the installation window. This status bar tracks the percentage of packages that have been installed on your computer. Depending on the number of products you have selected, the installation time required could be lengthy. You will not be able to interrupt the installation process. Please be patient while the installation completes.

Note

Clicking the **View Details Progress Log** button opens a second window that displays the name of each product package as it is uncompressed and installed.

Click **Next** to continue the installation.

10. The product installation window reappears with a message noting that the installation is complete. A **Launch Survey Upon Exiting** option is included here. Clicking **Exit** while the **Launch Survey Upon Exiting** is enabled causes your default browser to open, displaying the product survey. Disabling (unchecking) the **Launch Survey Upon Exiting** option and then clicking **Exit** skips the survey.

The ANSYS, Inc. Installation Launcher appears. For this stand-alone installation, you must complete the License Manager installation (next) to run ANSYS, Inc. products.

3.5.1. ANSYS License Manager Installation

Follow the instructions below to install the ANSYS, Inc. License Manager on your Linux machine. Because you will not be using a network server, you must install and configure the ANSYS, Inc. License Manager on your machine. The License Manager controls access to the ANSYS, Inc. products you have purchased.

1. Navigate to the directory where you extracted the files. Run `./INSTALL.LM`. If you downloaded the license manager installation package, this file will reside in the directory where you untarred the downloaded files. If you are running from a DVD, this file will reside in the top level of the DVD.

2. You may see a warning stating that if the license manager is currently running, it will be shut down. This installation assumes that you have not previously installed the products or the licensing on this machine, and you are not pointing to a network license server machine. You may safely ignore this message and click **OK**.

3. Select the language you want to use. English is the default.

4. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, select **I Agree**. Click **Next**.

5. The directory where you wish to install the ANSYS, Inc. License Manager is shown in the **Install Directory** field. You can install the License Manager into any directory you wish, but you must have write permissions

to the directory you choose. The default is `/ansys_inc`. We recommend using the default directory. You must use the same directory where the products were installed.

Note

You are unable to change the installation directory for a computer that currently contains an instance of the ANSYS, Inc. License Manager or ANSYS Electromagnetics License Manager. To change the installation directory location, you must first uninstall any previous versions of both products.

6. The ANSYS, Inc. License Manager is selected as the only product available to install. As with the product installation, the required and available disk space numbers are shown. Click **Next**.

7. A summary screen appears that lists the products to be installed. Because this is a license manager installation, the ANSYS, Inc. License Manager is the only product listed.

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

   The ANSYS, Inc. License Manager is now being installed and configured on your system. After the License Manager installation has been completed, the **Launch License Management Center upon exiting** option is included on the screen. The **ANSYS License Management Center** is a browser-based user interface that centralizes many of the ANSYS product licensing administrative functions. If you do not wish to launch the License Management Center, uncheck the **Launch License Management Center upon exiting** option.

8. Click **Exit** to close the **License Manager Installation** screen.

9. Click **Exit** to close the ANSYS, Inc. Installation Manager.

10. Open a terminal window and navigate to your ANSYS installation. The following table lists the default paths to start the ANSYS products you have installed.

Application Locations

Application	How to Launch
ACP	`/ansys_inc/v162/ACP.sh`
CFD Post	`/ansys_inc/v162/cfdpost`
CFX	`/ansys_inc/v162/CFX/bin/`*<productname>* Where *<productname>* can be *cfx5*, *cfx5launch*, *cfx5pre*, *cfx-solve*, or *cfx5post*.
FLUENT	`/ansys_inc/v162/fluent/bin/fluent`
ICEM CFD	`/ansys_inc/v162/icemcfd/icemcfd`
ICEPAK	`/ansys_inc/v162/icepak/icepak`
PolyFlow	`/ansys_inc/v162/polyflow/bin/polyflow`
Turbogrid	`/ansys_inc/v162TurboGrid/bin/cfxtg`
Workbench	`/ansys_inc/v162/Framework/bin/Linux64/runwb2`

3.5.1.1. Registering the License Server

If you are a new user who has not received a license file for your server or if you add or change a license server machine, follow this procedure to register your license server information. See the ANSYS, Inc. Licensing Guide for more information on selecting license servers (Selecting License Server Machines) and on using the ANSYS License Management Center (License Server Administration Using ANSYS License Management Center).

1. Open the **ANSYS License Management Center**. To run the **ANSYS License Management Center** on Linux run the following script:.

   ```
   /shared_files/licensing/start_lmcenter
   ```

2. Click the **Get System Hostid Information** option to display your system ID code(s).

3. Select the system ID you wish to use and click **SAVE TO FILE**.

   A text file containing your system ID information is created.

4. Forward this text file to your ANSYS sales representative so that a license file can be created for you.

5. Add your license files though the **ANSYS License Management Center**. For these steps, see Adding a License with the ANSYS License Management Center.

After completing the installation process, please refer to Post-Installation Instructions (p. 31).

Chapter 4: Installing the ANSYS Products and the License Server on Different Linux Machines

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 install as non-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 when navigating to the `/ansys_inc` directory.

If you do not use the `/ansys_inc` symbolic link, you must install all releases into a common directory to ensure license manager compatibility and availability among releases and products.

If you did not use the `/ansys_inc` symbolic link, you must replace all references to `/ansys_inc/v162` or `<install_dir>` with the actual installation path you used.

Before You Begin

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

- An account on the ANSYS Customer Portal. If you do not have an account, you may register at https://support.ansys.com/portal/site/AnsysCustomerPortal to receive your own account.
- Your license file from ANSYS, Inc., saved to a temporary directory. For more information, see Registering the License Server (p. 28).
- Open port numbers for both the FLEXlm and ANSYS Licensing Interconnect. Defaults are 1055 and 2325, respectively. To verify that these port numbers are available, open a command line and enter the following command:

  ```
  netstat -a -t
  ```

  You will see a list of active ports. If 1055 and 2325 are listed, they are already in use and cannot be used for ANSYS, Inc. licensing. In this case, you will need to specify different port numbers where indicated later in this installation.
- Your local machine's name, to specify as the license server.

You should also verify that you are running on a supported platform. ANSYS, Inc. products support 64-bit Linux systems running Red Hat 6 (6.4 and 6.5) and SUSE Linux Enterprise 11 (SP2 or SP3). For detailed information on which products are supported on which platforms, please see the *ANSYS, Inc. Linux Installation Guide*, which is available for download when you download ANSYS, Inc. products or as a printed manual if you receive installation media.

Verify that you have sufficient disk space to download, uncompress, and install the products you will be installing. Approximate disk space requirements for each product are shown in Disk Space and Memory Requirements (p. 3).

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

This section is divided into four sets of instructions:

- Product Download Instructions: This set of instructions describes the download and extraction process.
- Product Installation with Client Licensing: This set of instructions describes the product installation, including the client licensing portion.
- License Manager Installation: This set of instructions describes the license manager installation.
- Post-Installation Procedures for All Products: This is a set of instructions that describes any configuration steps that may be required for the various products.

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

4.1. 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 `getFLEXid` script on each machine first. This script will output the correct platform name for each machine on which it is run. This script can be obtained from the ANSYS website, in the Licensing Support area under "Capture License Server Info".

4.2. Product Download Instructions

To download the installation files from our website, you will need to have a current technical support agreement.

Depending on the product/platform combination(s) you choose, you may need to download multiple files.

1. From the Customer Portal, https://support.ansys.com/portal/site/AnsysCustomerPortal, click **Downloads > Current Release**.

   Step 1 of the ANSYS Download Center Select Full Products page is displayed.

2. Select your installation operating system (**Windows x64** or **Linux x64**).

3. Select the type of files you wish to download:

   - **Primary Packages**: Individual full packages for the primary ANSYS products.
   - **ISO Images**: ISO images for the DVD installation.

4. Click the appropriate download option.

5. Select your desired download directory and click **Save**.

6. Repeat this process for each download file.

7. To download **Add-On Packages**, **Tools** or **Academic Packages**, click the + to the right of the appropriate product group title to display the download options and download as required.

8. After all downloads have been completed, uncompress each package using standard uncompression utilities for your specific platform. We strongly recommend that you extract the files into new, temporary directories.

- Continue with the steps described in Product Installation (p. 24).

4.3. Installing from a USB Drive

Insert the USB drive into an appropriate USB slot on your computer. Navigate to the root directory of the USB drive and Run ./INSTALL. Continue with the steps described in Product Installation (p. 24).

4.4. Mounting the DVD Instructions for DVD Installations (Linux x64 Only)

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.

> **Note**
>
> Typically, a Linux system will mount a local DVD for you under /media/ANSYS162.

For a locally-mounted DVD installation, issue the following commands:

```
mkdir dvdrom_dir
mount -t iso9660 /dev/cdrom dvdrom_dir
```

If the target machine does not have a DVD reader, first follow the steps for locally-mounted DVD, and then follow the procedure below for remotely-mounted DVDs:

Remotely-Mounted DVD Procedure

1. Add the *`dvdrom_dir`* directory to the `/etc/exports` file on the machine with the DVD device. A sample `/etc/exports` entry is:

   ```
   /dvdrom_dir *(ro)
   ```

   or

   ```
   /dvdrom_dir (ro)
   ```

2. Run **exportfs** to export the *`dvdrom_dir`* directory:

   ```
   exportfs –a
   ```

   Check the manual page for 'exports' for the correct syntax, as different Linux versions can have different syntax.

3. Log on to the machine where you want to install ANSYS, Inc. products and issue the following commands:

   ```
   mkdir dvdrom_dir2
   mount -t nfs Host:cdrom_dir dvdrom_dir2
   ```

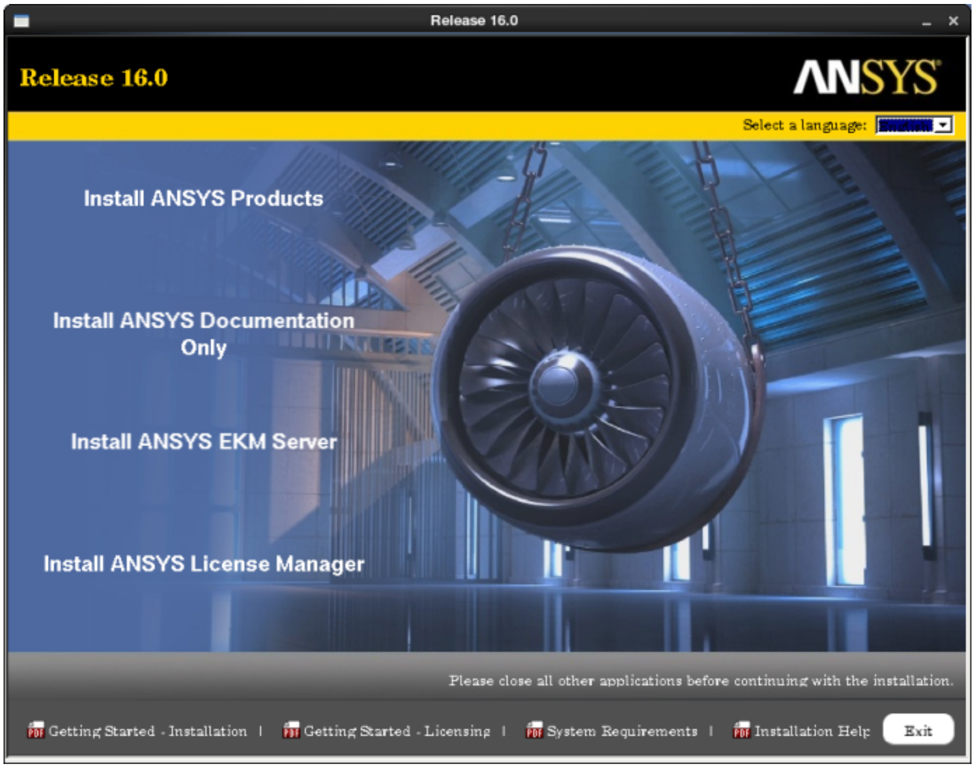
   where *`Host`* is the hostname of the machine where the DVD device is located.

   Run **`man exports`** for more information.

4.5. Product Installation

1. Navigate to the directory where you extracted the files. Run `./INSTALL`. If you downloaded the installation package, this file will reside in the directory where you untarred the downloaded files. If you are running from a DVD, this file will reside in the top level of the DVD.

2. The ANSYS, Inc. Installation Launcher appears.



   From the options along the left side of the launcher you can install ANSYS products, Documentation Only, ANSYS EKM Server, and the ANSYS License Manager. You can access the installation guide for the ANSYS EKM Server from the **Downloads** page on the Customer Portal.

   The ANSYS, Inc. Quick Start Installation Guide, ANSYS, Inc. Quick Start Licensing Guide, System Requirements Guide and complete Installation Help Guide can be accessed through the options located along the bottom of the launcher.

   Select the language you want to use from the drop-down menu in the upper right corner. English is the default.

3. Click the **Install ANSYS Products** option.

4. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, select **I Agree**. Click **Next**.

   When 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 want; however, you must run the platform configuration procedure (see *Run the Product Configuration Utility* in Network Installation and Product Configuration (p. 28)) for each platform other than your current machine type. See Network Installation and Product Configuration (p. 28) for specific instructions on how to configure a shared installation directory across multiple machines using a common network file system.

5. The directory where you want to install the ANSYS, Inc. products is shown in the **Install Directory** field. You can install the products into any directory you want, but you must have write permissions to the directory you choose. The default is `/ansys_inc`. We recommend using the default directory.

   The **Symbolic Link** option is available only if you are installing as root and is enabled (checked) by default. If you choose to disable (uncheck) the symbolic link or are installing as a non-root user, substitute the directory path where you installed the product for all subsequent occurrences of `/ansys_inc` in this guide.

   We strongly recommend that you use the **Symbolic Link** option. If you do not use this option, you must 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**.

6. If this is a first time installation, you are prompted to enter your license server specification. If you already have an existing license server specification file, you will not see this window and proceed directly to the next step.

   Enter your ANSYS Licensing Interconnect port number and your ANSYS FlexNet port number. Defaults are provided and will work in most cases. You may need to check with your IT department to confirm that the default port numbers are valid or to get different port numbers if necessary.

   Specify the hostname for your license server machine(s).

   Click **Next**.

7. All products available in the installation package(s) you downloaded are listed.

   The installation program attempts to query your license server to pre-select your installation options. If the query is successful, the following message is displayed:

   Your installation options were pre-selected based upon information from your license server.

   You can select or deselect any combination of products. ANSYS Workbench is automatically installed with most ANSYS, Inc. products; there is no individual product selection for ANSYS Workbench.

   If you select NX or Catia V5, you will have additional installation steps. This installation assumes you are not installing these CAD packages. Deselect these options before continuing.

   By default, the **Install Documentation** option is enabled (checked). When enabled, help documentation is included as part of the installation process for all products selected. No help documentation is included with the installation if this box is disabled (unchecked).

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

Note

On a first time installation, if you chose to install any ANSYS Geometry Interfaces, ensure that you have also selected at least one ANSYS, Inc. product as part of the installation. Installing an ANSYS Geometry Interface without an underlying ANSYS, Inc. product on a first time installation may cause installation errors.

Select the products you want to install and click **Next**.The dates on the licensing files being installed are compared to any that may already exist on your machine. (This may take a few moments.)

8. A summary screen appears listing your installation selections. Please review this list carefully to verify that it is correct. When you are sure all selections are correct, click **Next** to begin the installation.

9. The installation progress screen displays a status bar towards the bottom of the installation window. This status bar tracks the percentage of packages that have been installed on your computer. Depending on the number of products you have selected, the installation time required could be lengthy. You will not be able to interrupt the installation process. Please be patient while the installation completes.

Note

Clicking the **View Details Progress Log** button opens a second window that displays the name of each product package as it is uncompressed and installed.

Click **Next** to continue the installation.

10. The product installation window reappears with a message noting that the installation is complete. A **Launch Survey Upon Exiting** option is included here. Clicking **Exit** while the **Launch Survey Upon Exiting** is enabled causes your default browser to open, displaying the product survey. Disabling (unchecking) the **Launch Survey Upon Exiting** option and then clicking **Exit** skips the survey.

11. If you have installed ANSYS, Inc. products on a file server, follow the instructions under Network Installation and Product Configuration (p. 28).

4.5.1. ANSYS License Manager Installation

After you have installed the ANSYS products on your client machines, follow the instructions below to install the ANSYS, Inc. License Manager on your license server machine. The License Manager controls access to the ANSYS, Inc. products you have purchased.

1. Navigate to the directory where you extracted the files. Run `./INSTALL.LM`. If you downloaded the license manager installation package, this file will reside in the directory where you untarred the downloaded files. If you are running from a DVD, this file will reside in the top level of the DVD.

2. You may see a warning stating that if the license manager is currently running, it will be shut down. If you have not previously installed the products or the licensing on this machine, and you are not pointing to a network license server machine, you may safely ignore this message and click **OK**.

3. Select the language you want to use. English is the default.

4. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, select **I Agree**. Click **Next**.

5. The directory where you want to install the ANSYS, Inc. License Manager is shown in the **Install Directory** field. You can install the License Manager into any directory you want, but you must have write permissions to the directory you choose. The default is `/ansys_inc`. We recommend using the default directory. You must use the same directory where the products were installed.

   Note

   You are unable to change the installation directory for a computer that currently contains an instance of the ANSYS, Inc. License Manager or ANSYS Electromagnetics License Manager. To change the installation directory location, you must first uninstall any previous versions of both products.

6. The ANSYS, Inc. License Manager is selected as the only product available to install. As with the product installation, the required and available disk space numbers are shown. Click **Next**.

7. A summary screen appears that lists the products to be installed. Because this is a license manager installation, the ANSYS, Inc. License Manager is the only product listed.

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

   The ANSYS, Inc. License Manager is now being installed and configured on your system. After the License Manager installation has been completed, the **Launch License Management Center upon exiting** option is included on the screen. The **ANSYS License Management Center** is a browser-based user interface that centralizes many of the ANSYS product licensing administrative functions. If you do not want to launch the License Management Center, uncheck the **Launch License Management Center upon exiting** option.

8. Click **Exit** to close the **License Manager Installation** screen.

9. Click **Exit** to close the ANSYS, Inc. Installation Manager.

10. Open a terminal window and navigate to your ANSYS installation. The following table lists the default paths to start the ANSYS products you have installed.

Application Locations

Application	**How to Launch**
ACP	`/ansys_inc/v162/ACP.sh`
CFD Post	`/ansys_inc/v162/cfdpost`
CFX	`/ansys_inc/v162/CFX/bin/<productname>` Where *<productname>* can be `cfx5`, `cfx5launch`, `cfx5pre`, `cfx-solve`, or `cfx5post`.
FLUENT	`/ansys_inc/v162/fluent/bin/fluent`
ICEM CFD	`/ansys_inc/v162/icemcfd/icemcfd`
ICEPAK	`/ansys_inc/v162/icepak/icepak`
PolyFlow	`/ansys_inc/v162/polyflow/bin/polyflow`

Application	How to Launch
Turbogrid	`/ansys_inc/v162TurboGrid/bin/cfxtg`
Workbench	`/ansys_inc/v162/Framework/bin/Linux64/runwb2`

4.5.1.1. Registering the License Server

If you are a new user who has not received a license file for your server or if you add or change a license server machine, follow this procedure to register your license server information. See the ANSYS, Inc. Licensing Guide for more information on selecting license servers (Selecting License Server Machines) and on using the ANSYS License Management Center (License Server Administration Using ANSYS License Management Center).

1. Open the **ANSYS License Management Center**. To run the **ANSYS License Management Center** on Linux run the following script:.

   ```
   /shared_files/licensing/start_lmcenter
   ```

2. Click the **Get System Hostid Information** option to display your system ID code(s).

3. Select the system ID you wish to use and click **SAVE TO FILE**.

   A text file containing your system ID information is created.

4. Forward this text file to your ANSYS sales representative so that a license file can be created for you.

5. Add your license files though the **ANSYS License Management Center**. For these steps, see Adding a License with the ANSYS License Management Center.

4.5.2. 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, 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, NX, and Remote Solve Manager (RSM).

A network installation must be homogeneous, although you can install on different operating systems.

You must complete the following steps to run products across a network:

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.

4.5.2.1. 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/USB drive or an internet connection for downloading files and you need to share files with a machine that does have a DVD/USB drive or internet connection.

The instructions below assume ANSYS, Inc. products were installed in the specified directory.

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)
   ```

   Alternatively, if the installing user is root, use:

   ```
   /usr/ansys_inc *(rw,no_root_squash)
   ```

3. Run

   ```
   exportfs -a
   ```

4. On all client computers, mount the `ansys_inc` directory.

If you perform a network install where 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/`) if you want to be able to create the `install_licconfig.log` that the license configuration produces.

If you need to transfer the files from a Windows machine with a DVD drive to a 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 Linux.

If sharing the ANSYS directory between 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).

4.5.2.2. Run the Product Configuration Utility on All Client Machines

For network installations, you must run this step on every client machine.

> **Note**
>
> The username and user ID utilized when running the Product Configuration Utility must be the same as the installing server user.

1. On each client machine, issue the following command to run the **Product Configuration** utility:

   ```
   /ansys_inc/v162/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, or Remote Solve Manager Standalone Services is selected. CFD-Post is configured when either ANSYS CFX, ANSYS Fluent, ANSYS Polyflow or ANSYS Icepak is configured.

4. On the **Configuration Complete** dialog box, click **Finish**.

After completing the installation process, please refer to Post-Installation Instructions (p. 31).

Chapter 5: Post-Installation Instructions

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 for each product.

Note

If the installation program reported any errors, please review the installation error file (install.err) located in the ANSYS Inc directory. Contact your ANSYS Support representative if you have any questions.

After the product is installed, you need to establish some system settings, including path names 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).

`/ansys_inc/v162/ansys/bin`	(or appropriate path to the individual products' executables)
`/ansys_inc/shared_files/licensing/lic_admin`	(contains ANSLIC_ADMIN utility)

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. 49) 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.

Quality Assurance Services: If you require verification of Mechanical APDL, the Mechanical Application, Fluent, or CFX, ANSYS, Inc. offers Quality Assurance services. If you are interested in this service, go to http://www.ansys.com/Support/Quality+Assurance/Quality+Services or call the ANSYS, Inc. Corporate Quality Group at (724) 746-3304.

5.1. Post-Installation Procedures for Mechanical APDL and ANSYS Workbench Products

The following post-installation procedures apply only to the Mechanical APDL and ANSYS Workbench products. These are in addition to the post-installation procedures in the previous section.

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 **ANSYS162_DIR** environment variable sets the location of the ANSYS directory hierarchy. The default value is `/ansys_inc/v162/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.
   ANSYS162_PRODUCT - set this to the correct product variable to run Mechanical APDL 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.
   ANSYS162_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.
   ANSYS162_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).
   ANSYS162_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).
   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 launcher. By default, **ANSBROWSER** points to one of several 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. 49) 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 `/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 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.

5.1.1. Post-Installation Procedures for ANSYS CFX

The following post-installation procedures apply only to the ANSYS CFX product.

The Linux installation of ANSYS CFX or ANSYS TurboGrid automatically installs the Sun Java 2 Runtime Environment in the `/ansys_inc/v162/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 *`cfxroot`*`/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.

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.

5.1.1.1. Setting up ANSYS TurboGrid Release 16.2

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 **`cfxtg`**.

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.

5.1.1.2. 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.

5.1.1.3. Verifying the Installation of ANSYS CFX Products

If you are working on a 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. This can be found in the Tutorials & Training Materials section of the ANSYS Customer Portal.

Note

To speed up the process of testing, consider using the provided session files when working with ANSYS CFX-Pre.

5.1.2. 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/v162/fluent/bin $path)
```

or in the Bourne/Korn shell or bash, by entering:

```
PATH=(/ansys_inc/v162/fluent/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/v162/fluent/bin
   ```

3. Run the following command:

   ```
   <install_dir>/ansys_inc/v162/fluent/bin/fluent
   ```

4. Click **Default**.

5. Click **Yes** when asked if you want to discard the LAUNCHER history.

6. Click **Cancel** if you do not want 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.

5.1.3. 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.

5.1.4. Post-Installation Procedures for ANSYS ICEM CFD

The following post-installation procedures apply only to the ANSYS ICEM CFD product.

1. Add the following paths to all users' login startup files (i.e., `.cshrc` or `.login` files).

   /ansys_inc/v162/icemcfd/linux64_amd/bin

2. Add the following environment variable to all users' login startup files.

   ICEM_ACN - set to `/ansys_inc/v162/icemcfd/linux64_amd`

3. Start ANSYS ICEM CFD by typing **`icemcfd`**.

5.1.5. 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/v162/autodyn/bin
```

5.1.6. Post-Installation Procedures for Other Products

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.

You can inadvertently destroy your BladeEditor models by updating a project or otherwise processing a Geometry cell, depending on your license preference settings. This problem can affect you if you have any ANSYS DesignModeler licenses. To avoid this problem, configure the license preferences as described above. For more details, please see Configuring the BladeModeler License in the ANSYS BladeEditor documentation.

5.2. Product Localization

Some ANSYS, Inc. products are available in multiple languages, including English, German, and French. 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)

ANSYS, Inc. applications will look for the `languagesettings.txt` file in the following locations, in order:

1. `$Home/.ansys/v162`

2. <install_dir>/ansys_inc/v162/commonfiles/language

ANSYS, Inc. licensing also looks for the `languagesettings.txt` in the licensing languages subdirectories in order to display the **ANSLIC_ADMIN** utility and the ANSYS, Inc. Licensing Interconnect message and log files in a different language.

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.

5.2.1. Translated Message File Installation for Mechanical APDL

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/v162/ansys/docu/fr
   ```

2. Copy the message files `msgcat.162`, `msgidx.162`, and `msgfnm.162` into that subdirectory.

3. Access these files from the Language Selection option of the launcher or via the **-l** command line option:

   ```
   ansys162 -l fr
   ```

5.3. Launching ANSYS, Inc. Products

To launch ANSYS, Inc. products on 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 5.1: Startup Commands

Product	Command	Notes
Mechanical APDL	`/ansys_inc/v162/ansys/bin/ansys162`	For a complete list of command line options, see Starting an ANSYS Session from the Command Level in the *Operations Guide*.
ANSYS Workbench	`/ansys_inc/v162/Framework/bin/<platform>/runwb2`	
ANSYS CFX	`/ansys_inc/v162/CFX/bin/cfx5`	
ANSYS Fluent	`/ansys_inc/v162/fluent/bin/fluent`	For a complete list of command line and launcher options, see *Starting ANSYS Fluent* in the *Fluent Users Guide*.
ANSYS ICEM CFD	`/ansys_inc/v162/icemcfd/<platform>/bin/icemcfd`	
ANSYS Polyflow	`/ansys_inc/v162/polyflow/bin/polyman`	Starts the POLYFLOW MANager. For any other tool, use `/ansys_inc/v162/polyflow/bin/<tool>`

Product	Command	Notes
ANSYS CFD-Post	/ansys_inc/v162/CFD-Post/bin/cfd-post	
ANSYS Icepak	/ansys_inc/v162/Icepak/bin/icepak	
ANSYS TurboGrid	/ansys_inc/v162/TurboGrid/bin/cfx-tg	
ANSYS Autodyn	/ansys_inc/v162/autodyn/bin/autodyn162	solver only
ANSYS ACP	/ansys_inc/v162/ACP/ACP.sh	

Chapter 6: Installing the ANSYS, Inc. Product Help Documentation Only

Performing the ANSYS product help documentation installation places the entire help library onto your computer.

To perform the installation:

1. Navigate to the directory where you extracted the installation files. Run ./INSTALL.

   - If you downloaded the installation package, this file will reside in the directory where you untarred the downloaded files.

   - If you are running from a DVD or USB, this file will reside in the top level of the DVD or USB.

   The **ANSYS, Inc. Installation Launcher** appears.

2. From the options along the left side of the launcher click **Install ANSYS Documentation Only**.

   The license agreement appears.

3. Read the agreement, and if you agree to the terms and conditions, click **I Agree**. Click **Next**.

4. The directory where you want to install the ANSYS, Inc. help documentation is shown in the **Install Directory** field. You can install the help documentation into any directory you want, but you must have write permissions to the directory you choose. The default is **/ansys_inc**. We recommend using the default directory.

   We strongly recommend that you also set the symbolic link **/ansys_inc** to the directory where the ANSYS, Inc. product is installed. The symbolic link option is available only if you are installing as root and is set by default. If you choose not to specify the symbolic link or are installing as a non-root user, substitute the directory path where you installed the product for all subsequent occurrences of**/ansys_inc** in this guide.

   If you do not use the **/ansys_inc** symbolic link, you must install all releases into a common directory to ensure license manager compatibility and availability among releases and products.

   Click **Next**.

5. A summary screen appears listing your help documentation installation. You will also see an estimate of the disk space required to install all of the selected components, and the disk space you have available.

   Click **Next** to begin the installation.

6. When the help documentation installation has completed, click **Exit**. The product installation window re-appears. You can close this window at this time.

> **Note**
>
> Alternatively, you can run the document installation by running the **INSTALL.DOCONLY** installation file. This file is located in the *linux* directory on USB drives.

The ANSYS help documentation can be installed silently by running the **INSTALL.DOCONLY** installation file with the –silent flag. For more information, see *Silent Mode Operations*.

Chapter 7: Silent Mode Operations

ANSYS, Inc. supports silent mode operations, including installation, product configuration/unconfiguration, 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

Product	***product_flag***
Mechanical APDL	-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 Aqwa	-aqwa
ANSYS ICEM CFD	-icemcfd

Note: Installing any of the above products will install ANSYS Workbench.

ANSYS Composite PrepPost	-acp
ANSYS Remote Solve Manager Standalone Services	-rsm
NX Geometry Interface	-ug
CATIA 4.x Geometry Interface	-catia4
CATIA 5.x Geometry Interface	-catia5
Parasolid Geometry Interface	-parasolid
ACIS Geometry Interface	-acis
NX Geometry Interface Reader	-ug_reader
NX Geometry Interface Plugin	-ug_plugin
ANSYS Icepak	-icepak

7.1. Silent Product and License Manager Installation

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

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 available 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, Inc. License Manager on 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.
`-help`	Displays a list of valid arguments for 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 `-install_dir` argument. The default location is `/ansys_inc` if the symbolic link is set; otherwise, it will default to `/usr/ansys_inc`.
`-product_flag`	Specifies one or more products to install specific products. If you omit the *`-product_flag`* argument, all products will be installed. See the list of valid *`product_flags`* 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 Specifying Products with an Options File (p. 46) 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 (`INSTALL.LM`).
`-setliclang` *`language`*	Specifies a language to use for the **ANSLIC_ADMIN** utility and the ANSYS, Inc. Licensing Interconnect log file. Use the language directory name in the language subdirectory of the licensing directory (`en-us`, `fr`, `de`, etc.) as the *`language`*

	value. This flag can be used during a GUI installation as well. Valid only when doing a license manager installation (`INSTALL.LM`).
`-licserverinfo`	Specifies information to be used by the client for the license server. Valid only in conjunction with a silent installation (`INSTALL`). The format is: Single license server: *LI_port_number*:*FLEXlm_port_number*:*hostname* Example: 2325:1055:abc Three license servers: *LI_port_number*:*FLEXlm_port_number*:*hostname1*,*hostname2*,*hostname3* Example: 2325:1055:abc,def,xyz The default values for the Licensing Interconnect and FlexNet port numbers (2325 and 1055, respectively) will be used if they are not specified. However, you do need to include the colons. Example: ::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 **ANSLIC_ADMIN** utility.
`-nohelp`	By default, for the products that are being installed, all help documentation is included. The `-nohelp` argument removes all help from the installation.
`-lang`	Specifies a language to use for the products.

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 `$HOME/.ansys`.

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, NX is commented out using an acceptable comment indicator. When using the options file, do not include the dash (-) before the product name.

```
mechapdl
ansyscust
autodyn
lsdyna
cfdpost
cfx
turbogrid
fluent
polyflow
icepak
#ug
```

7.2. 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/v162/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:

Product	***product_flag***
Mechanical APDL	-mechapdl
ANSYS CFD-Post	-cfdpost
ANSYS CFX	-cfx
ANSYS TurboGrid	-turbogrid
ANSYS Fluent	-fluent
ANSYS Polyflow	-polyflow
ANSYS Icepak	-icepak
ANSYS ICEM CFD	-icemcfd
ANSYS Remote Solve Manager Standalone Services	-rsm

Errors will be written to the ${HOME}/.ansys/v160/ directory on the client machine.

7.3. Silent Media Installation

To run a silent installation from the media, you can either:

- Copy the contents of each DVD to a folder on the machine's hard disk such that the `162-`<*`number`*>`.dvd` files of each DVD are located in the same directory. You can then proceed with the silent installation as described earlier.

- Place all of the media in separate drives (any combination of virtual ISO mounts or hardware drives) so that they can be accessed simultaneously during the installation. Then run the silent installation as described earlier, but include the additional `-media_dir2 <path>` and `-media_dir3 <path>` options as needed for each drive:

```
INSTALL -silent -install_dir path -product_flag -media_dir2 <path -media_dir3 <path>
```

  The installer uses the mount directory from which it was launched as the first media path; you need to specify only the location of the subsequent DVD(s) using the `-media_dir2` and `-media_dir3` options shown in the example above.

7.3.1. Silent Uninstall

You can also run the uninstall silently by issuing the following command:

```
/ansys_inc/v162/ans_uninstall162 –silent
```

The silent uninstall will automatically uninstall all products for this release and delete the v162 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	-mechapdl
ANSYS CFX	-cfx
ANSYS CFD-Post	-cfdpost
ANSYS Fluent	-fluent
ANSYS Polyflow	-polyflow
ANSYS ICEM CFD	-icemcfd
ANSYS TurboGrid	-turbogrid
ANSYS Icepak	-icepak
ANSYS Remote Solve Manager Standalone Services	-rsm

For example, to uninstall only TurboGrid and Icepak, issue the following command:

```
/ansys_inc/v162/ans_uninstall162 –silent -turbogrid -icepak
```

You can also issue the `-help` option to see a list of valid arguments for a silent uninstall.

A record of the uninstall process will be written to `ansys_inc/install.log`. Any error messages will be written to `ansys_inc/install.err`.

Chapter 8: Configuring CAD Products

The connection functionality of all supported CAD products is included with the ANSYS release media, and all CAD functionality except NX 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 the *Mechanical APDL ANSYS Connection User's Guide*.

Caution

Be sure to install Mechanical APDL 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 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 NX if you did not configure NX during installation or if you are updating your CAD versions. These methods are all described in the following sections.

8.1. Using the CAD Configuration Manager

The **CAD Configuration Manager** utility allows you to configure geometry interfaces for ANSYS Workbench 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 Associative Interface mode for NX is supported on Linux. Consequently, the ANSYS NX menu and the active connection for the Associative Interface is NOT available on Linux. ANSYS uses a passive mode previously known as the Associative Reader on Linux.

The **CAD Configuration Manager** on Linux is organized into several tabs:

- CAD Selection
- Creo Parametric (options are active on Windows only)
- NX
- Teamcenter (options are 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, 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/v162/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 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**.

4. 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.

5. 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`. 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`. When `/tmp` is not accessible, this file will be written to the current working directory.

8.1.1. 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.

8.1.2. 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/v162/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigurationUtility.exe
     -arguments
```

Argument	Value	Comment
UNCONFIGURE_SPECIFIED	None	Results in any specified CAD sources being unconfigured. When this flag is absent, the **CAD Configuration Manager** will attempt to unconfigure all designated CAD sources.
UNCONFIGURE	None	Results in all CAD sources being disabled along with prerequisite libraries.
UG_CONFIG_WB	None	Configure/unconfigure NX Geometry Interface to Workbench. The argument UGII_BASE_DIR must also be specified for the configure operation.
UGII_BASE_DIR	Full path to NX installation (quotations are required if there are spaces in the path).	This should agree with environment variable **UGII_BASE_DIR**. Not required with unconfigure operation.
CATIA_READER	None	Configure/unconfigure CatiaV5 Reader.

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 NX Geometry Interface to ANSYS Workbench from the command line by using the following:

```
/ansys_inc/v162/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigurationUtility.exe
-UG_CONFIG_WB -UGII_BASE_DIR "<pathtonx>"
```

where /ansys_inc/v162 is the installation directory you specified.

To unconfigure the same CAD Interface, the command would be:

```
/ansys_inc/v162/commonfiles/CAD/bin/ansmono  Ans.CadInt.CADConfigurationUtility.exe
-unconfigure -UG_CONFIG_WB
```

Although the argument order does not matter, an argument value must immediately follow its argument.

8.1.3. NX Configuration

Running the **CAD Configuration Manager** for NX performs the following steps to activate the NX reader:

- Registers the NX Associative Geometry Interface for ANSYS Workbench by copying the file `UGNX#.Component.XML` from `/ansys_inc/v162/aisol/CADIntegration/UG` to either `/ansys_inc/v162/commonfiles/registry/linx64/append` when the configuration manager is in administrative mode or to `$HOME/.config/ANSYS/162/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 variables **UGII_BASE_DIR** and **UGII_ROOT_DIR**. The **UGII_BASE_DIR** must point to the install location supplied to the **CAD Configuration Manager**. The **UGII_ROOT_DIR** must point to the `/ugii` or `/bin` subdirectory in the NX install location. The Workbench Associative Interface for NX will not work without these two variables set properly, as these are required for proper startup of the CAD.

- Prior to configuring a client of a network install, the client system must have its **UGII_BASE_DIR** point to the same version of NX as designated on the server system. This must be done before running **CAD Configuration Manager** or the **Product Configuration Tool**. For more information, see Network Installation and Product Configuration (p. 28).

8.2. 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 9: Uninstalling the Software

To uninstall a product, issue the following command:

```
/ansys_inc/v162/ans_uninstall162
```

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/v162/ansys/bin/ans_admin162
```

1. From the uninstall panel, click **Select Products to Uninstall**.

   If you are not a superuser, you will see a warning message, and then the uninstall continues.

2. A list of products that are installed appears. Select those products you want to uninstall and click **Continue**. Then click **OK** to confirm the list of products to be uninstalled.

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 Linux platforms.

Chapter 10: Troubleshooting

10.1. 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 **Knowledge Resources> Solutions**.

For information on licensing-related errors, see the Troubleshooting section of the ANSYS, Inc. Licensing Guide.

10.1.1. Gathering Diagnostic Information

There are situations which require licensing-related information to be gathered for diagnostic and troubleshooting purposes. At times it may be necessary to provide this information to technical support.

The client-related diagnostic information can be gathered by using the Client ANSLIC_ADMIN utility. For more information, see, Gather Client Diagnostic Information.

The server-related diagnostic information can be gathered by using **ANSYS License Management Center** or by using the standalone `gatherdiagnostics` script. For more information, see, Gathering Diagnostic Information.

10.1.2. 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.

10.1.3. 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, use a USB Flash drive or follow the instructions in Mounting the DVD Instructions for DVD Installations (Linux x64 Only) (p. 15) to mount to a machine that does have a DVD drive.

10.1.4. 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**.

10.1.5. 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.

10.1.6. 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 want 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, USB 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 Mounting the DVD Instructions for DVD Installations (Linux x64 Only) (p. 15) 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 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.

10.1.7. 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 *Mechanical APDL Basic Analysis Guide* for specific graphics information.

*****Error, ANSYS162_DIR environment variable is not set. This is a fatal error – exiting.**

This message indicates that the **ANSYS162_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.

10.1.8. 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 16.2 with a different version of MPI than is supported. See the *Parallel Processing Guide* for a complete list of supported MPI versions.

10.2. Installation Troubleshooting - Mechanical APDL

The items listed below apply only to the Mechanical APDL product.

10.2.1. 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.'

10.2.2. Mechanical APDL 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/v162/ansys/gui/en-us/UIDL` subdirectory.

```
ls -l  /ansys_inc/v162/ansys/gui/en-us/UIDL/menulist162.ans
```

The system should respond with:

```
-rw-r--r-- 1 root 23 Jan  8 11:50 /ansys_inc/v162/ansys/gui
     /en-us/UIDL/menulist162.ans
```

Make sure that the pathnames in the `menulist162.ans` file are correct.

10.2.3. 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\v162\launcher` on Windows or `~/.ansys/v162/launcher` on Linux.

10.2.4. FORTRAN Runtime Error Messages

The following error messages occur if you are running Mechanical APDL 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 you do not have write permissions to the files. The specific messages that appear on each system are shown below.

10.2.4.1. Intel Linux 64 Systems

Input/Output Error 177: Create Failure

In Procedure: fappnd

At Line: 72

Statement: Formatted WRITE

Unit: 19

10.2.4.2. Intel EM64T Linux x64 Systems

forrtl: Permission denied

forrtl: severe (9): permission to access file denied, unit 19, file /build/v162/ansys/objs

10.2.4.3. AMD Opteron Linux x64 Systems

*****ERROR**

Unable to open file /build/v162/ansys/objs/file.err for WRITE. Check directory and file permissions.

10.3. Installation Troubleshooting - ANSYS Workbench

10.3.1. 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 re-launch 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 rending 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 -oglhw flag:

```
runwb2 -oglhw
```

10.4. Installation Troubleshooting - ANSYS CFX

10.4.1. CFX Distributed Parallel Runs Fail

On some SLES machines (typically ones with more than one network card), the default configuration of `/etc/hosts` will cause CFX distributed parallel runs to fail. In such cases, the problem might be solved by editing the `/etc/hosts` file to remove all lines that contain redundant loopback addresses. Do not remove the line with the first loopback address, which is typically 127.0.0.1.

10.5. 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 and select **Contacts> Contacts and Locations**.

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) under **Support > Customer Portal**. The direct URL is: support.ansys.com.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal. To use this feature, enter relevant text (error message, etc.) in the Knowledge Resources Search box and click the magnifying glass icon. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS Products except Esterel, Apache and Reaction Design products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Toll-Free Telephone: 1.800.711.7199 (Please have your Customer or Contact ID ready.)

Support for University customers is provided only through the ANSYS Customer Portal.

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55 (CADFEM)

Email: support@cadfem.de

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

National Toll-Free Telephone: (Please have your Customer or Contact ID ready.)

German language: 0800 181 8499

English language: 0800 181 1565

Austria: 0800 297 835

Switzerland: 0800 546 318

International Telephone: (Please have your Customer or Contact ID ready.)

German language: +49 6151 152 9981

English language: +49 6151 152 9982

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Telephone: Please have your Customer or Contact ID ready.

UK: 0800 048 0462

Republic of Ireland: 1800 065 6642

Outside UK: +44 1235 420130

Email: support-uk@ansys.com

Support for University customers is provided only through the ANSYS Customer Portal.

JAPAN

Mechanical Products

Telephone: +81-3-5324-8333

Email:

Mechanical: japan-ansys-support@ansys.com

Fluids Products

Telephone: +81-3-5324-7305

Email:

Fluent: japan-fluent-support@ansys.com;

CFX: japan-cfx-support@ansys.com;

Polyflow: japan-polyflow-support@ansys.com;

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Telephone: +91 1 800 209 3475 (toll free) or +91 20 6654 3000 (toll) (Please have your Customer or Contact ID ready.)

Support for University customers is provided only through the ANSYS Customer Portal.

FRANCE

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Toll-Free Telephone: +33 (0) 800 919 225 **Toll Number:** +33 (0) 170 489 087 (Please have your Customer or Contact ID ready.)

Email: support-france@ansys.com

BELGIUM

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) 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://support.ansys.com) and select the appropriate option.

Telephone: +46 (0) 10 516 49 00

Email: support-sweden@ansys.com

SPAIN and PORTUGAL

All ANSYS Products

Web: Go to the ANSYS Customer Portal (http://support.ansys.com) and select the appropriate option.

Telephone: +34 900 933 407 (Spain), +351 800 880 513 (Portugal)

Email: support-spain@ansys.com, support-portugal@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://support.ansys.com) 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.

Chapter 11: Applications Included with Each Product

The following table displays which ANSYS, Inc. applications are included with each of the product installation options.

Product Install Option	What is Included?
ANSYS, Inc. Products	
ANSYS Structural Mechanics	
ANSYS Mechanical Products	Workbench [1], Mechanical APDL
ANSYS Customization Files	Workbench [1], Mechanical APDL User-Programmable Feature
ANSYS Explicit Dynamics	
ANSYS Autodyn	Workbench [1], Autodyn
ANSYS LS-DYNA	Workbench [1], ANSYS LS-DYNA
ANSYS Fluid Dynamics	
ANSYS CFX (includes ANSYS CFD-Post)	Workbench [1], CFX, CFD-Post
ANSYS Fluent (includes ANSYS CFD-Post)	Workbench [1], Fluent, CFD-Post
ANSYS TurboGrid	Workbench [1], TurboGrid
ANSYS Polyflow (includes ANSYS CFD-Post)	Workbench [1], CFD-Post, Polyflow
ANSYS CFD-Post only	Workbench [1], CFD-Post
ANSYS ICEM CFD	Workbench [1], ICEM CFD
ANSYS Additional Tools	
ANSYS Composite PrepPost	ACP Only
ANSYS Icepak (includes ANSYS CFD-Post)	CFD-Post, Icepak
ANSYS Remote Solve Manager Stand Alone Services	Remote Solve Manager
ANSYS Geometry Interfaces	
ACIS	Geometry Interface for ACIS
Catia, Version 4	Geometry Interface for Catia, Version 4
NX	Geometry Interface for NX
Parasolid	Geometry Interface for Parasolid

1. Workbench includes the Workbench Framework, DesignModeler, DesignXplorer, and Remote Solve Manager.