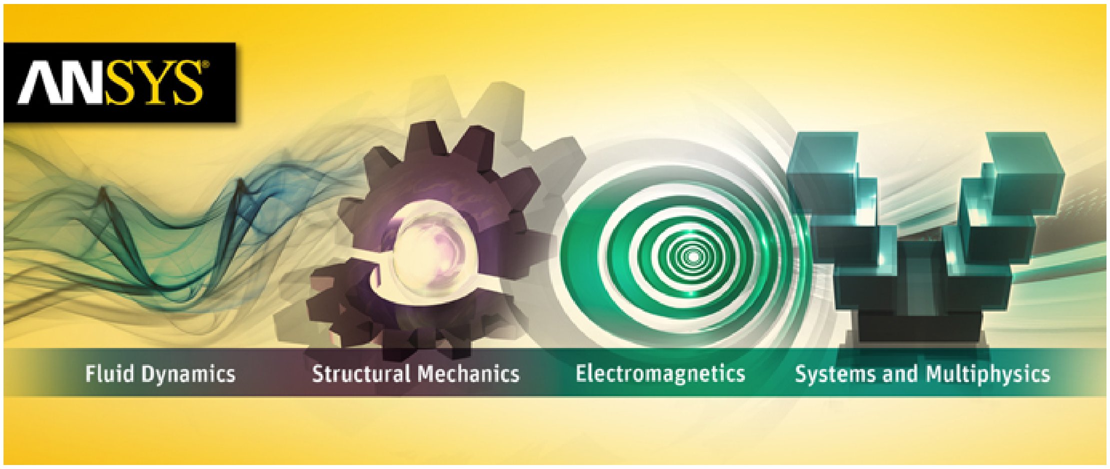

ANSYS, Inc. Installation Guide for Linux

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

Release 14.0
November 2011
000288

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

Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superceded by a newer version of this guide.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

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Published in the U.S.A.

Table of Contents

1. Installation Prerequisites for Linux 1
1.1. System Prerequisites 4
1.1.1. CAD Support 5
1.2. Disk Space and Memory Requirements 6
1.3. GPU Requirements 7
1.4. Additional Hardware and Software Requirements 7
1.5. Third-Party Software and Other Security Considerations 8
2. Platform Details 11
3. Pre-Installation Instructions 17
3.1. Pre-Installation Instructions for Download Installations 17
3.1.1. Downloading the Installation Files 17
3.2. Mounting the DVD Instructions for DVD Installations (Linux x64 Only) 18
4. Installing the Software 21
4.1. Installing ANSYS, Inc. Products 21
4.1.1. Product Installation 21
4.1.2. License Manager Installation 24
4.1.2.1. Registering the License Server 26
4.1.3. Network Installation and Product Configuration 26
4.1.3.1. Export the `/ansys_inc` Directory 27
4.1.3.2. Run the Product Configuration Utility on All Client Machines 28
4.1.3.3. Configure Licensing for File Server Installations 29
4.1.4. Silent Mode Operations 29
4.1.4.1. Silent Product and License Manager Installation 30
4.1.4.2. Silent Product Configuration/Unconfiguration 33
4.1.4.3. Silent Media Installation 34
4.1.4.4. Silent Uninstall 35
5. Post-Installation Instructions 37
5.1. Post-Installation Procedures for All Products 37
5.1.1. Post-Installation Procedures for Mechanical APDL (ANSYS) and ANSYS Workbench Products 38
5.1.2. Post-Installation Procedures for ANSYS CFX 40
5.1.2.1. Setting up ANSYS TurboGrid Release 14.0 42
5.1.2.2. Using the ANSYS CFX Launcher to Set Up Users 42
5.1.2.3. Verifying the Installation of ANSYS CFX Products 43
5.1.3. Post-Installation Procedures for ANSYS FLUENT 43
5.1.4. Post-Installation Procedures for ANSYS POLYFLOW 44
5.1.5. Post-Installation Procedures for ANSYS ICEM CFD 45
5.1.6. Post-Installation Procedures for ANSYS AUTODYN 45
5.2. Product Localization 46

5.2.1. Translated Message File Installation for Mechanical APDL (ANSYS) 46
5.3. Launching ANSYS, Inc. Products 47
6. Uninstalling the Software 49
6.1. Uninstalling ANSYS, Inc. Products 49
7. Configuring CAD Products 51
7.1. Using the CAD Configuration Manager 51
7.1.1. Unconfiguring 53
7.1.2. Running the **CAD Configuration Manager** in Batch Mode 53
7.1.3. NX Configuration 55
7.2. Configuring NX 55
7.2.1. Configuring the Connection for NX 56
7.2.2. Configuring the Geometry Interface for NX for ANSYS Workbench Products 57
8. Troubleshooting 59
8.1. Installation Troubleshooting 59
8.1.1. Using ANSLIC_ADMIN to Gather Diagnostic Information 59
8.1.2. The GUI Installation Process Hangs 59
8.1.3. The Target Machine Does Not Have a DVD Drive 60
8.1.4. The Online Help System Does Not Run Properly 60
8.1.5. CAD Configuration Manager Help Does Not Load 61
8.1.6. Cannot Enter Data in Text Fields 61
8.1.7. Download and Installation Error Messages 62
8.1.8. System-related Error Messages 63
8.1.9. High Performance Computing Error Messages 63
8.2. Installation Troubleshooting - Mechanical APDL (ANSYS) 64
8.2.1. Your batch jobs terminate when you log out of a session 64
8.2.2. Japanese/Chinese characters display in status bar windows on Mechanical APDL (ANSYS) on Red Hat AS 4.0 64
8.2.3. Mechanical APDL (ANSYS) Documentation File for User Interface Error Messages 64
8.2.4. Launcher Error Messages 65
8.2.5. FORTRAN Runtime Error Messages 65
8.2.5.1. Intel Linux 64 Systems 65
8.2.5.2. Intel EM64T Linux x64 Systems 65
8.2.5.3. AMD Opteron Linux x64 Systems 66
8.3. Installation Troubleshooting - ANSYS Workbench 66
8.3.1. Startup or Graphics Problems 66
8.4. Installation Troubleshooting - ANSYS CFX 67
8.4.1. CFX Distributed Parallel Runs Fail 67
8.5. Contacting Technical Support 67
Configuring Distributed ANSYS 71
1. Setting up Distributed ANSYS 71

1.1. Prerequisites for Running Distributed ANSYS 72
1.1.1. MPI Software 72
1.1.2. Installing the Software 74
1.2. Setting Up the Cluster Environment for Distributed ANSYS 76
1.2.1. Optional Setup Tasks 78
1.2.2. Using the mpitest Program 80
1.2.3. Interconnect Configuration 81
2. Running a Distributed Job 81
Configuring ANSYS CFX Parallel 83
1. ANSYS CFX UNIX Parallel Setup 83
1.1. Setting Up Remote Access on UNIX/Linux 83
1.1.1. Testing Remote Access 84
1.1.2. Global Set Up of rsh 84
1.1.3. Individual User Set Up for rsh 85
1.1.4. Set Up of ssh 85
1.2. hostinfo.ccl File 85
1.2.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility 88
1.3. Using Platform MPI (Message Passing Interface Library) 89
1.3.1. Environment Variables 90
1.3.2. Interconnect Selection 91
2. ANSYS CFX Windows Parallel Setup 92
2.1. hostinfo.ccl file 92
2.1.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility 95
2.2. Setting Up Platform MPI for Windows 96
2.2.1. Setting Up Distributed Parallel or Remote Access 97
2.2.2. Installing the Platform MPI Service and Registering Users on Windows Vista 98
2.2.3. Installing the Platform MPI Service and Registering Users on Windows XP 98
2.2.4. Enabling Parallel Through a Windows Firewall 99
2.3. Setting up and Running CCS 2003/HPC 2008 100

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 14.0. These products include:

- ANSYS Structural Mechanics
  - ANSYS Mechanical Products (includes Mechanical APDL and Mechanical, where supported)
  - ANSYS Customization Files
- ANSYS Explicit Dynamics
  - ANSYS AUTODYN
  - ANSYS LS-DYNA
- ANSYS Fluid Dynamics
  - ANSYS CFX (includes ANSYS CFD-Post)
  - ANSYS FLUENT (includes ANSYS CFD-Post)
  - ANSYS TurboGrid
  - ANSYS POLYFLOW (includes ANSYS CFD-Post)
  - ANSYS CFD-Post only
- ANSYS Additional Tools
  - ANSYS ICEM CFD
    - → ANSYS ICEM CFD Creo Parametric Interface
    - → ANSYS ICEM CFD NX Interface
    - → ANSYS Icepak (includes ANSYS CFD-Post)
    - → Remote Solve Manager Standalone Services
- ANSYS Geometry Interfaces
  - ACIS

- CATIA, Version 4
- 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 14.0, ANSYS BladeGen, BladeEditor, 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. product (Linux 64-bit systems only).ANSYS Workbench is not installed as part of the products under ANSYS Additional Tools, nor with the CFD-Post only option. ANSYS Workbench includes the following applications:

- The Mechanical Application
- DesignModeler
- Design Exploration
- Meshing
- Remote Solve Manager
- TGrid
- FE Modeler
- EKM Desktop

Because some of these applications can be run as standalone applications without ANSYS Workbench, you will see some but not all of these listed separately as options when you uninstall. In such cases, you will need to select those options in order to uninstall those components.

Important Notice

If you wish to run multiple releases of ANSYS, Inc. software, you **MUST** install them chronologically (i.e., Release 13.0 followed by Release 14.0). If you install an earlier release after installing Release 14.0, you will encounter licensing issues that may prevent you from running any products/releases. If you need to install an earlier release after you have already installed Release 14.0, you **MUST** uninstall Release 14.0, then re-install the releases in order.

Summary of New and Changed Features

The following features are new or changed at Release 14.0. Please review these items carefully.

- ANSYS, Inc. has discontinued support for the HP-UX Itanium 64, the Sun Solaris x64, IBM AIX 64, and the Linux 32-bit platforms for all products.
- ANSYS, Inc. has discontinued support for the Linux Itanium 64 platform for the ICEM CFD product.
- Third-party products that are used as part of the installation process are now documented in the *ANSYS, Inc. Installation Guides*.
- The ASAS product has been retired. The FATJACK, BEAMCHECK, and Splinter products are now installed automatically with the Mechanical application.
- The Pro/ENGINEER CAD product has been rebranded to Creo Parametric.
- The ANSYS, Inc. product installation now supports Creo Parametric, NX, SolidWorks, and Autodesk Inventor reader options. You can now choose to install the Reader (no CAD installation required) or the Associative Plug-in (CAD installation required) options for these CAD products.
- The release version now appears with each product selection in the Start menu on Windows.
- You can now specify two DVD drives during a silent installation to accommodate the installation process spanning two DVDs. See the discussion on Silent Mode Operations in the Installation Guide for your platform for more information.
- You can now choose to install and uninstall only Remote Solve Manager (RSM). RSM will continue to be installed as part of ANSYS Workbench.
- You can now choose to install and uninstall only the EKM Server on Windows platforms.
- The use of files requiring 777 permissions on Linux has been minimized. For more information on remaining full-permission files and softlinks, see the section Third-Party Software and Other Security Considerations in the ANSYS, Inc. Linux Installation Guide.
- The PDF version of the documentation that is available on the Customer Portal is now unprotected, allowing you to copy and paste content from the PDFs into other locations. This capability is especially useful if you want to use command snippets that are available throughout the documentation.

1.1. System Prerequisites

ANSYS, Inc. Release 14.0 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.)

Note

The ANSYS, Inc. License Manager supports additional UNIX/Linux platforms. Please see the *ANSYS, Inc. Licensing Guide* for the complete list of platforms supported by the ANSYS, Inc. License Manager.

Table 1.1 Supported Platforms

Platform	Processor	Operating System	Platform architecture (directory name)	Availability
Linux Itanium 64	Itanium2	Red Hat Enterprise Linux 5; SUSE Linux Enterprise 10 and Enterprise 11 SP1	linia64	Download only
Linux x64	EM64T/Opteron 64	Red Hat Enterprise Linux 5; Red Hat Enterprise Linux 6, SUSE Linux Enterprise 10 [1] and Enterprise 11 SP1	linx64	Download / DVD

1. We recommend that you run ANSYS Workbench on Red Hat Enterprise Linux 5.3 or higher or SUSE Linux Enterprise 10.2 or higher.

Supported Platforms for High Performance Computing Please see the discussions on *Configuring Distributed ANSYS* and *Configuring ANSYS CFX Parallel* later in this guide for detailed information on supported platforms for distributed computing.

Table 1.2 Supported Platforms by Product

	Linux Itanium 64	Linux x64
Mechanical APDL	X	X
Workbench		X [*1*]
AUTODYN (solver only)	X	X
LS-DYNA	X	X
CFX	X	X
FLUENT	X	X
TurboGrid		X
ICEM CFD		X
POLYFLOW		X
Icepak		X

1. We recommend that you run ANSYS Workbench on Red Hat Enterprise Linux 5.3 or higher or SUSE Linux Enterprise 10.2 or higher on Linux x64 systems.

1.1.1. CAD Support

The following CAD and auxiliary programs are supported on the indicated products and platforms. Products are:

A = Mechanical APDL
W = ANSYS Workbench
I = ANSYS ICEM CFD standalone (some CAD systems may require the integrated ANSYS Workbench Reader)

Table 1.3 CAD Support by Platform

	Linux Itanium 64	Linux x64
CATIA 4.2.4	A [*4*]	A [*4*], I
Parasolid 22.1	A	A, W, I
Parasolid 24		A, W, I
SAT ACIS 21 [*1*]	A [*4*]	A, W, I

	Linux Itanium 64	Linux x64
NX 6.0 [*2*]		A, W, I
NX 7.5		W [*3*], I [*3*]
NX 8.0		W [*5*]
STEP AP203, AP214		W, I
IGES 4.0, 5.2, 5.3		W, I
DWG		I
GEMS		I

1. For ANSYS ICEM CFD standalone, ACIS 18.0.1 is the supported version for all platforms.
2. NX 6.0 has dropped support for all Linux platforms except Linux SUSE 10 SP2.
3. NX 7.5 supports Red Hat 5.3 and SUSE Linux Enterprise 10 SP2.
4. Red Hat requires the installation of i686 (32-bit) glibc & libstdc++ libraries.
5. NX 8.0 supports Red Hat 6.0 and SUSE Linux Enterprise 11 SP1.

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
Mechanical APDL (ANSYS)	8.1 GB
ANSYS AUTODYN	7.1 GB
ANSYS LS-DYNA	7.3 GB
ANSYS CFX	9.1 GB
ANSYS TurboGrid	8.0 GB
ANSYS FLUENT	8.8 GB
POLYFLOW	10.2 GB
ANSYS ICEM CFD	2.2 GB
ANSYS Icepak	2.7 GB
ANSYS TGrid	5.5 GB

Product	Disk Space
CFD Post only	8.0 GB
ANSYS Geometry Interfaces	170 MB

Memory Requirements We recommend that you have at least 2 GB of memory available for running product installations.

1.3. GPU Requirements

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

The machine(s) being used for the simulation must contain at least one nVIDIA Tesla series GPU card (a Tesla 20-series card is recommended for optimal performance) or one Quadro 6000 card. If both a Tesla and Quadro card are detected, Mechanical APDL will choose the Tesla card.

On Linux, the driver version for the nVIDIA Tesla series GPU card must be 275.09.07 or newer.

You must be running on a Linux x64 operating system. Linux Itanium (Linux IA-64) is not supported.

1.4. Additional Hardware and Software Requirements

- TCP/IP for the license manager (see the *ANSYS, Inc. Licensing Guide* for more information on TCP/IP)
- If you use a spaceball, ensure that you have the latest version of the drivers installed. Spaceball devices are not supported on Linux with ANSYS Workbench applications, including the Mechanical Application, DesignModeler, Meshing, etc.
- 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.
- X11, OpenGL graphics libraries

- Mesa-libGL (OpenGL) is required to run data-integrated ANSYS Workbench applications such as Mechanical.
- If you are using Exceed to run ANSYS Workbench products, Exceed 3D is required. Exceed 3D 2008 Service Pack 9 or higher is recommended.
- For FLUENT, CFX-Pre, and CFD-Post, a three-button mouse is required to access all available functionality.
- Adobe Acrobat Reader is required to read the installation guides and other user documentation.
- ANSYS CFX requires Exceed 2006 or newer to remote display from some platforms.
- FLUENT graphics are not officially supported when running under Exceed.

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:

- MainWin
- CPython
- Mono
- Python
- Perl
- GCC

- Qt
- Qwt
- HPMPI
- PCMPI
- IMPI
- MPICH
- OPENMPI
- INTELMPI
- PCMPI

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 SP1. After installing the SP1 updates, you must also install OpenMotif and the prerequisites from the SLES11 SDK DVD, as well as the OpenMotif22 packages (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
- libXp.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 32 and 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 For ANSYS Workbench, install the following gamin `.rpm` for your platform to prevent ANSYS Workbench from hanging when the Linux file alteration monitor, gam_server, is using 100% of the CPU.

- gamin-0.1.7-1.4.EL4.x86_64.rpm

If you are running ANSYS Workbench using the KDE desktop environment, set the focus stealing prevention level to "None" to prevent the project save dialog boxes from appearing behind the application window:

1. Use the **`kcontrol`** command to launch the KDE Control Center.
2. In the Control Center window, select **Desktop> Window Behaviour> Advanced**.
3. Change **Focus Stealing Prevention Level** to None.

4. Click **Apply**.

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 Behaviour> Focus**.
3. Change **Focus Stealing Prevention Level** to None.
4. Click **Apply**.

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 `/v140/aisol/WBMWRegistry/` directory has write permissions. From the `/v140/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*>`/v140/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, for a system with 8GB of memory, the following should be added to the `/etc/security/limits.conf` file:

```
* soft memlock 7500000
* hard memlock 7500000
```

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

```
fluent_mpi.6.3.26: Rank 0:0: MPI_Init: ibv_create_qp()
     failed fluent_mpi.6.3.26: Rank 0:0:
MPI_Init: Can't initialize RDMA device
```

After setting soft and hard memlock to a proper value you still receive memory errors:

```
libibverbs: Warning: RLIMIT_MEMLOCK is 32768 bytes.
This will severely limit memory registrations
```

Semaphore Limit On some Linux systems, ANSYS Workbench reaches a system limit on the number of semaphores in the Linux configuration. In this case, you will see a message similar to the following:

sem_lock->semop->op_op: Invalid argument

sem_unlock->semctl: Invalid argument

To increase the number of semaphores, run the following command as owner or root:

```
% echo 256 40000 32 32000 > /proc/sys/kernel/sem
```

This modification takes effect immediately, but is reset at the next reboot. To avoid resetting the limit when rebooting, add the above command to one of your system's startup scripts by copying the command into a file called `mod_sem` and then setting up the following links to execute the file each time you restart your system:

```
cp mod_sem /etc/init.d
ln -s /etc/init.d/mod_sem /etc/rc3.d/S61mod_sem
ln -s /etc/init.d/mod_sem /etc/rc5.d/S61mod_sem
```

Using the FLUENT Launcher On Linux systems, you must have the following package (as appropriate for your platform) installed in order to use the FLUENT launcher:

Red Hat 4: compat-libstdc++-33-3.2.3-47.3
Red Hat 5: compat-libstdc++-33-3.2.3-61
SUSE 10: compat-libstdc++-5.0.7-22.2

Using ANSCUSTOM If you use ANSCUSTOM to link your own version of ANSYS Release 14.0 on a SUSE SLES10.x box, you may see two unsatisfied externals that are system, not ANSYS files, due to the linker looking for some system files in a Red Hat directory on a SUSE box.

To work around this problem, run the following as root:

```
mkdir –p /usr/lib/gcc/x86_64-redhat-linux/3.4.6
ln -sf /usr/lib/gcc/i586-suse-linux/2.95.3/crtbegin.o /usr/lib/gcc/x86_64-redhat-linux/
     3.4.6/crtbegin.o
ln -sf /usr/lib/gcc/i586-suse-linux/2.95.3/crtend.o   /usr/lib/gcc/x86_64-redhat-linux/
     3.4.6/crtend.o
```

The revision numbers shown in the examples (2.95.3 and 3.4.6) may be different on your system. The linker will specify where it is trying to find the `crtbegin.o` and `crtend.o` files; that location will be the last part of the above commands. You can use the Linux **`locate`** command to find the existing `crtbegin.o` and `crtend.o` files; that location would be the first part of the above commands.

System Libraries On 64-bit Linux linx64 systems (not Itanium systems), the ANSYS Release 14.0 executable is looking for system libraries that do not have revision numbers appended to the end of their file names. On some SUSE systems, the graphics libraries all have revision numbers appended to the end of the library filenames. In these cases, ANSYS quits because the loader cannot find all of the libraries that it is looking for. When running ANSYS Release 14.0, the loader will inform you that it is unable to locate a specific library (for example, `libXm.so`). Using the Linux **`locate`** command, find the library (`libXm.so` in this example) on your system and add the appropriate symbolic link as seen below.

To overcome this possible problem, run the following as root:

```
ln -sf /usr/lib64/libGLU.so.1.3.060402  /usr/lib64/libGLU.so
ln -sf /usr/X11R6/lib64/libXm.so.3.0.3  /usr/X11R6/lib64/libXm.so
ln -sf /usr/X11R6/lib64/libXp.so.6.2    /usr/X11R6/lib64/libXp.so
ln -sf /usr/X11R6/lib64/libXt.so.6.0    /usr/X11R6/lib64/libXt.so
ln -sf /usr/X11R6/lib64/libXext.so.6.4  /usr/X11R6/lib64/libXext.so
ln -sf /usr/X11R6/lib64/libXi.so.6.0    /usr/X11R6/lib64/libXi.so
ln -sf /usr/X11R6/lib64/libX11.so.6.2   /usr/X11R6/lib64/libX11.so
ln -sf /usr/X11R6/lib64/libSM.so.6.0    /usr/X11R6/lib64/libSM.so
ln -sf /usr/X11R6/lib64/libICE.so.6.4   /usr/X11R6/lib64/libICE.so
ln -sf /lib64/libgcc_s.so.1             /lib64/libgcc.so
```

For Linux, you may need the following:

```
sudo ln -sf /usr/lib64/libXm.so.4.0.0 /usr/lib64/libXm.so.3
```

The revision numbers appended to the filenames on the left may be different on your system.

Intel Linux

ANSYS was built and tested on Red Hat using the compilers as noted in *Table 2.1: Compiler Requirements* (p. 16). The ANSYS solver is built on Red Hat Enterprise Linux AS release 4 (Update 5).

For ANSYS Workbench and ANSYS AUTODYN, you need to unlimit the stack size. Add the following to your `.cshrc` file:

```
limit stack unlimit
```

ANSYS TurboGrid 14.0 is not supported on Linux IA-64.

AMD Opteron

ANSYS was tested on a generic Opteron^tm system running Red Hat Enterprise Linux AS release 4 (Update 5).

Intel Xeon EM64T

ANSYS was built and tested on a generic Intel EM64T system running Red Hat Enterprise Linux AS release 4 (Update 5).

If you are running on Intel's Xeon EM64T system, we recommend that you turn CPU hyperthreading off (default is on). A system administrator needs to reboot the system and enter the BIOS to turn the hyperthreading option off.

Table 2.1 Compiler Requirements

Mechanical APDL (ANSYS), ANSYS Workbench Compilers*	**CFX Compilers***	**FLUENT Compilers***	**AUTODYN Compilers***
Linux (all versions)			
Intel 11.1.069 (FORTRAN, C, C++)	PGI Fortran 10.3	Intel 11.1.069 (FORTRAN, C, C++)	Intel 11.1.069 (FORTRAN, C, C++)

* Compilers are required only if you will be using User Programmable Features or other customization options.

Chapter 3: Pre-Installation Instructions

3.1. Pre-Installation Instructions for Download Installations

Before downloading the installation files, you need to accurately determine your platform type. Versions that are optimized for different chip sets from the same vendor can have similar names, causing confusion. We strongly recommend that you run the `get_ansys_platform140` script on each machine first. This script will output the correct platform name for each machine on which it is run. You can download this script by clicking the **Which UNIX/Linux platform am I using?** button on the download site.

We strongly recommend that you review the **Read Download Instructions and Product Information** file included on the download site for the most current download instructions. We also recommend that you review the **What Should I Download?** file to understand which package(s) you need to download, depending on which product(s) you purchased and wish to run.

3.1.1. Downloading the Installation Files

To download the installation files from our website, you will need to be a TECS customer.

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

1. From the Customer Portal, click on **Download Software**.
2. The ANSYS Download Center Wizard page displays an overview of the download process. Review the overview and click **Next**.
3. For the Download Type, choose **Current Release and Updates**. Click **Next Step**.
4. Choose the hardware platform for which you want to download installation packages. You can select only one platform at a time. You will need to repeat

the download procedure for each platform that you want to download. Click **Next Step**.

5. Choose the products you wish to download. Products are listed by product names that correspond to the licensing product names. You can choose to list products grouped by product group or alphabetically. All products that you currently have licensed are highlighted and pre-selected for your convenience. After you have selected all products that you wish to download, click **Next Step**.
6. The license manager, product, and documentation packages that you've selected to download are listed. Click on each link provided to begin the downloads.
7. After the downloads have completed, uncompress each package using standard uncompression utilities for your specific platform. We strongly recommend that you extract the files into a new, temporary directory.
8. Begin the product installation as described in *Installing the Software* (p. 21).

3.2. 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.

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.

If you are installing from media, you will be prompted to change DVDs during the installation. Please make sure you have all installation DVDs before beginning the installation.

Chapter 4: Installing the Software

4.1. Installing ANSYS, Inc. Products

This section explains how to install ANSYS, Inc. products, including ANSYS client licensing, as well as the ANSYS, Inc. License Manager.

The default installation expects you to be logged in as root. You can override that setting and run the installation as a regular user by starting the installation with the `-noroot` command option. If you are not logged in as root, however, you will not be able to set the `/ansys_inc` symbolic link and may potentially experience permission problems. The inability to set the `/ansys_inc` symbolic link will in no way inhibit your ability to run ANSYS, Inc. products; it is provided as a convenience.

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

4.1.1. Product Installation

1. To launch the product install, enter the full path to the installation program and run the `INSTALL` program:

   ```
   /<mount_dir>/INSTALL
   ```

   To invoke the license manager only install, run `INSTALL.LM` and proceed directly to *License Manager Installation*, below. Use `/INSTALL.LM` *only* to install the license manager; use `INSTALL` to install the product(s).

   To invoke the EKM Server installation, run `INSTALL.EKMSVR` to launch the EKM server setup. You must install the EKM client, ANSYS Workbench, and any other desired components before installing the EKM Server.
2. Select the language. Click **Next**.

3. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
4. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. products. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. You can choose as many platforms as you wish; however, you must run the platform configuration procedure (see *Run the Product Configuration Utility* in *Network Installation and Product Configuration* (p. 26)) for each platform other than your current machine type. See *Network Installation and Product Configuration* (p. 26) for specific instructions on how to configure a shared installation directory across multiple machines using a common network file system.

   Click **Next** to continue.
5. Specify the installation directory. You can accept the default or specify an alternate directory name where the products are to be installed. When you choose an install directory via the Browse feature, the installation will automatically append `/ansys_inc/` to the chosen directory. The installation path can have a maximum of 100 characters. You must install all ANSYS, Inc. products into the same location. Installing products into different locations can cause product components to fail.

   Default directory is `/ansys_inc`, if it exists, or `/usr/ansys_inc` if not.

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

   If you do not use the `/ansys_inc` symbolic link, 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** to continue.

6. Select the components you want to install. You can select as many components as you wish to install. The amount of disk space required for the available components and the disk space available appears at the bottom of the window. If the disk space required exceeds the disk space available, be sure that you have sufficient space before continuing. The disk space required as calculated by the installation program may be greater than the actual amount needed. If you choose to continue the installation, you should carefully review any log and error files at the end of the installation to ensure that the installation completed successfully.

   The components listed represent all available components for the platforms you selected earlier. Not all components are available on all platforms or with all products.

   Click **Next** to continue.

7. If you selected NX, you will have to choose which NX product to configure. You can choose the Reader (non-associative), Workbench Associative Interface (requires that the CAD product be installed), or skip and configure later (using the **CAD Configuration Manager**).

   If you choose the Workbench Associative Interface and the UGII environment variables were not set, you may need to specify the NX installation path for an existing NX installation. Click **Next**.

8. A licensing file date verification summary appears. If the date verification finds a conflict, a message box appears with details of the conflict and steps for resolution. If no conflicts are found, click **Next**.

9. A summary of the selected installation data appears. Information shown includes platform, installation directory, and products. Review the information carefully, and if correct, click **Next** to continue the installation.

   The selected products and components are now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the licensing client.

10. The **Licensing Client Installation Configuration** box appears. As the licensing client is installed, progress messages appear in the box.

11. If you do not have an existing `ansyslmd.ini` file, the **Specify License Server Machine - Add Server Specification** box appears. Enter the hostname of your license server machine and click **Next**.

    If you already have an existing `ansyslmd.ini` file, you will not see this box and you will proceed directly to the next step.
12. When the client installation is complete, click **Exit**.
13. On the product installation window, click **Next**. You will be asked to participate in an Install Survey. To take the survey, enter the path to a valid browser for your system, and click **Next**. You can also click **Finish** now to skip the survey or when you have completed the survey.

If you have installed ANSYS, Inc. products on a file server, follow the instructions under *Network Installation and Product Configuration* (p. 26).

Caution

If you have a three-license server network, we do not recommend that you load ANSYS, Inc. products on a single file server; doing so eliminates the inherent safety of a redundant setup.

4.1.2. License Manager Installation

Follow the instructions below to install the ANSYS License Manager server on Linux systems. Client licensing is installed automatically when the product is installed; you do not have to take any further steps to run as a client if you have installed a product. You may safely install the ANSYS License Manager over a client installation. Install the client first (as part of the product installation), and then install the ANSYS License Manager following the steps below. If you are installing both the client and the ANSYS License Manager to the same directory, you will need to do both installations in order to properly configure the ANSYS License Manager.

1. Run `INSTALL.LM` to launch the license manager installation. If you downloaded the license manager installation package, this file will reside in the directory where you untarred the downloaded files. If you are running from a DVD, this file will reside in the top level of the DVD.
2. You will be notified that the license manager, if running, will be shut down. Click **OK**.
3. Select the language. Click **Next**.

4. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.
5. If you are installing more than one platform or if you are installing a platform other than your current machine type, you will need to select the platform(s) on which you want to install the ANSYS, Inc. License Manager. The platform on which you launched the installation will be selected by default and is shown at the bottom of the window. Click **Next** to continue.
6. The installation directory is specified. You can accept the default or specify an alternate path and directory name where the products are to be installed.

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

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

   The ANSYS License Manager is now being installed and configured on your system. The installation window displays the individual actions as they occur. When the installation is complete, the window displays any installation errors or warnings. Review this information carefully. Click **Next** to continue the installation and install the license manager.
10. The **Licensing Server Installation Configuration** box appears. As the license manager is installed, progress messages appear in the box.
11. The License Wizard will be launched. This wizard walks you through the process of installing or updating a license file, specifying the license server(s) (which updates the `ansyslmd.ini` file), and starting the license manager. The wizard

will prompt you for the necessary information at each step. During this process, the license manager will be shut down if it is running. Be aware that this can impact any users currently running using the license manager.

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

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

4.1.2.1. Registering the License Server

Follow this procedure to register your license server information if you are a new user or if you are adding or changing a license server machine. See the *ANSYS, Inc. Licensing Guide* for more information on selecting license servers and on using the **ANSLIC_ADMIN** utility.

1. Use the **ANSLIC_ADMIN** utility to register license server information on each license server. To run the **ANSLIC_ADMIN** utility on a Linux platform, type:

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

2. Return the resulting file to your ANSYS, Inc. sales representative. ANSYS, Inc. or your sales representative will then supply you with your license keys. This file is located in `/ansys_inc/shared_files/licensing/licserver.info` by default.
3. Use the **Run the License Wizard** option of the **ANSLIC_ADMIN** utility to enter your license key and start the license manager.

4.1.3. Network Installation and Product Configuration

To complete a network installation (where the product is installed on one machine and one or more clients access that installation to run the product) to a file server machine or a cross-platform installation (where the product for one machine type is installed on a different machine type), follow the steps below. These steps apply to the following products: Mechanical APDL, ANSYS Workbench, ANSYS CFX, ANSYS ICEM CFD, ANSYS FLUENT, ANSYS POLYFLOW, ANSYS Icepak, Common/CAD, and Remote Solve Manager (RSM).

A network installation must be homogeneous, although you can install on different operating systems and architectures within the same platform. For example, you can install multiple Linux platforms on a single Linux file server, and clients of those platform types will access the product installation on that single Linux file server. However, you must install different platforms into different directories.

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

4.1.3.1. Export the /ansys_inc Directory
4.1.3.2. Run the Product Configuration Utility on All Client Machines
4.1.3.3. Configure Licensing for File Server Installations

We strongly recommend that these steps be performed by the same non-root user. Installing and configuring as different users may create permissions problems. Likewise, installing and/or configuring as a root user may also result in permissions problems.

4.1.3.1. Export the `/ansys_inc` Directory

If you are installing an ANSYS, Inc. product on a file server, you need to export the `/ansys_inc` directory to all client machines so that all users can access the program. You will also need to share the ANSYS directory if the machine you are installing on does not have a DVD drive or an internet connection for downloading files and you need to share files with a machine that does have a DVD drive or internet connection.

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

1. Install the ANSYS, Inc. products. The following example uses `/usr/ansys_inc`.
2. Export the `ansys_inc` directory by adding the following line to the `/etc/exports` file:

   ```
   /usr/ansys_inc
   ```

   The default behavior on Linux provides read-only access from all clients. To enable read/write permission from all clients, use *(rw):

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

3. Run

   ```
   exportfs -a
   ```

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

If you perform a network install where the server and client are on the same platform and you want the clients to be able to modify the licensing configuration, you need to consider the NFS write options for the exported file system as shown in the above examples. You also need local permissions to the licensing directory (`/shared_files/licensing/<platform>/`) if you want to be able to create the `install_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).

4.1.3.2. Run the Product Configuration Utility on All Client Machines

For both network and cross-platform installations, you must run this step on every client machine.

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

   ```
   /ansys_inc/v140/commonfiles/tools/<platform>/ProductConfig.sh
   ```

2. Select **Configure Products**.
3. Select the products you want to configure and click **Configure**.

   Note

   ANSYS Workbench is configured when Mechanical APDL, ANSYS CFX, ANSYS FLUENT, ANSYS POLYFLOW, ANSYS Icepak, or Remote Solve Manager Standalone Services is selected. CFD-Post is configured when either ANSYS CFX or ANSYS FLUENT is configured.

   Select **Common/CAD** to configure the NX CAD packages.

4. If you selected **Common/CAD** and are configuring NX, you will need to supply the NX installation directory path.

   You can choose to skip the configuration. If you skip the configuration here, you must manually configure NX or use the **ANS_ADMIN** utility or the **CAD Configuration Manager** to configure NX later. See *Configuring CAD Products* (p. 51) for detailed information on using alternative methods to configure NX.

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

4.1.3.3. Configure Licensing for File Server Installations

When you run a cross-platform/file server installation where the product is installed on one platform and run on a different platform, you will need to complete the client licensing configuration on the client machines where you will be running the product.

1. Run the **ANSLIC_ADMIN** utility:

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

2. Choose **Tools> Complete Unfinished Licensing Installation Configuration**.
3. Follow the on-screen prompts to provide any necessary information.
4. Click **Exit**.

4.1.4. 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 (ANSYS)	-mechapdl
ANSYS Customization Files	-ansyscust
ANSYS AUTODYN	-autodyn

Product	*product_flag*
ANSYS LS-DYNA	-lsdyna
ANSYS CFD-Post	-cfdpost
ANSYS CFX	-cfx
ANSYS TurboGrid	-turbogrid
ANSYS FLUENT	-fluent
ANSYS POLYFLOW	-polyflow
ANSYS AQWA	-aqwa
ANSYS Icepak	-icepak
ANSYS ICEM CFD	-icemcfd

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

ANSYS ICEM CFD Unigraphics NX Interface	-icemug
ANSYS Remote Solve Manager Standalone Services	-rsm
NX Geometry Interface	-ug
CATIA 4.x Geometry Interface	-catia4
Parasolid Geometry Interface	-parasolid
ACIS Geometry Interface	-acis

4.1.4.1. Silent Product and License Manager Installation

You can deploy an ANSYS, Inc. product installation in silent mode. The general form to run a silent product installation, including the client licensing, is:

```
INSTALL -silent -install_dir path -product_flag
```

If no product flags from the list above are specified, all products will be installed. To install specific products, run the silent install with any combination of the product flags listed above (not all products are available on all platforms). For example, to install only TurboGrid and Icepak, issue the following command:

```
INSTALL -silent –install_dir "/ansys_inc/" -turbogrid -icepak
```

Additional command line arguments are available; please see the list below.

To install the ANSYS License Manager on 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* 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`*:*`host-name`* Example: 2325:1055:abc Three license servers: *`LI_port_number`*:*`FLEXlm_port_number`*:*`host-name1`*,*`hostname2`*,*`hostname3`* Example: 2325:1055:abc,def,xyz The default values for the Licensing Interconnect and FLEXlm port numbers (2325 and 1055, respectively) will be used if they are not specified. However, you do need to include the colons. Example: ::abc or ::abc,def,xyz Information specified via `-licserverinfo` will be appended to existing information in the `an-syslmd.ini` file. To change information already in your `ansyslmd.ini` file, you must use the **AN-SLIC_ADMIN** utility.

Any messages will be written to the appropriate installation log files. Installation log files are located in the installation directory: `install.log` contains installation messages, and `install_licconfig.log` contains licensing installation messages. In rare circumstances with a silent licensing installation, the licensing installation messages may not be written to the `install_licconfig.log` (for example, if the silent licensing installation aborts); in these cases, you may find error messages in the `.ansys_install_temp_licconfig_<user>_<index>.log` file, located in `/var/tmp`.

Caution

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

For more information on the silent license manager installation, see the *ANSYS, Inc. Licensing Guide*.

Specifying Products with an Options File

You can also specify an options file on the command line using the `-productfile path` option. The options file can have any name and extension, but the path must include the full path and filename, including any extension used. The options file can specify which products you want to install. The options file can contain all possible products, with the products you do not want to install commented out, or it can contain only the products you want to install. An example options file is shown below. In the example, 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
icemug
icepak
#ug
```

4.1.4.2. Silent Product Configuration/Unconfiguration

You can also run the `ProductConfig` utility via command line (i.e., silent mode) to configure products.

To run in silent mode, from each client machine, run the `ProductConfig` with the `-silent` option:

```
/ansys_inc/v140/commonfiles/tools/<platform>/ProductConfig.sh -silent
```

Use the *`-product_flag`* argument to specify which products should be configured. If you do not specify one or more products, all products that have been installed will be configured. The valid *`product_flags`* are:

Product	*product_flag*
Mechanical APDL (ANSYS)	-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 `install.err` on the server machine if you have write access to that machine.

4.1.4.3. 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 `140-<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_path2 <path>` option for each drive:

  ```
  INSTALL -silent -install_dir path -product_flag -media_path2 <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_path2` option shown in the example above.

4.1.4.4. Silent Uninstall

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

```
/ansys_inc/v140/ans_uninstall140 –silent
```

The silent uninstall will automatically uninstall all products for this release and delete the v140 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/v140/ans_uninstall140 –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 errors messages will be written to `ansys_inc/install.err`.

Chapter 5: Post-Installation Instructions

5.1. Post-Installation Procedures for All Products

The following post-installation procedures apply to all products. Individual products may have additional post-installation procedures; please refer to the following sections.

After the product is installed, you need to establish some system settings, including pathnames and environment variables. See your shell documentation or man pages file for the shell being used for specific instructions on setting paths and environment variables.

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

`/ansys_inc/v140/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. 51) 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 Program: If you require verification of Mechanical APDL, the Mechanical Application, FLUENT, or CFX, ANSYS, Inc. offers Quality Assurance programs. If you are interested in this service, go to http://www.ansys.com/About+ANSYS/Quality+Assurance/Quality+Services or call the ANSYS, Inc. Corporate Quality Group at (724) 746-3304.

5.1.1. Post-Installation Procedures for Mechanical APDL (ANSYS) and ANSYS Workbench Products

The following post-installation procedures apply only to the Mechanical APDL (ANSYS) and ANSYS Workbench products. These are in addition to the post-installation procedures noted above for all products.

1. Set the following environment variables based on the behavior you want. Set the environment variables following the conventions of your shell. Not all of these are required for all integrated ANSYS Workbench products (such as ANSYS AUTODYN), but setting them correctly for ANSYS Workbench will in no way hinder the performance of the other products.

   The **ANSYS140_DIR** environment variable sets the location of the ANSYS directory hierarchy. The default value is `/ansys_inc/v140/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 `/an-`

`sys_inc/shared_files/licensing`. You probably will not need to reset this variable, unless you change the location of the licensing files.
ANSYS140_PRODUCT - set this to the correct product variable to run Mechanical APDL (ANSYS) to start with the correct product without specifying the **-p** command modifier each time. See the Product Variable Table in the *ANSYS, Inc. Licensing Guide* for a list of valid product variables.
ANSYS_LOCK - set to ON (default) to create file locks to prevent users from opening a new job with the same name and in the same directory as the current job.
ANSYS140_WORKING_DIRECTORY - set this variable to the directory you want designated as your working directory. The working directory setting in the launcher will reflect this setting.
SPACEBALL2 - set this environment variable to 1 to use a spaceball with Mechanical APDL.
ANSYS140_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).
ANSYS140_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 (ANSYS) 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. 51) 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 (ANSYS) launcher (optional). If you want to specify product order, use the **ANSLIC_ADMIN** utility. See the *ANSYS, Inc. Licensing Guide* for more information.

Explicit Dynamics, Rigid Dynamics, My Computer Background, and Remote Solve Manager (RSM) Users: If you are running ANSYS Workbench on a multi-user RSM machine, the 'My Computer, Background' Solve Process Settings will likely not function as expected due to write permissions for RSM working directories. In this situation, we strongly recommend that you set up RSM as a daemon.

This issue also affects Rigid Dynamics and Explicit Dynamics using both 'My Computer' and 'My Computer, Background' Solve Process Settings. Please see Configuring a Multi-User RSM Machine in the RSM documentation for more information.

5.1.2. 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/v140/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.

User setup can also be run from the command line by entering:

```
cfxroot/bin/cfx5setupuser
```

Use the `-h` option to view the optional commands.

If you choose to modify your setup files, you will see a message indicating that your setup files have been changed. You will then need to log out and log in again or source your setup files before you can use the software.

Manual execution of a setup script each time the software is used You can also use the **Tools** menu of the launcher to launch an editor to create new setup scripts which need to be run each time you want to use the ANSYS CFX software. This method has the advantage of not requiring changes to existing setup files and allows you to use different versions of ANSYS CFX software by running different setup files. The disadvantages are that all users must create their own setup files and run them manually in every session in which they want to run ANSYS CFX software.

Having created the setup files, users of the C shell then need to do the following to run ANSYS CFX:

```
source ~/cfx5.login
cfx5
```

Having created the setup files, users of the Bourne or Korn shell then need to do the following before running ANSYS CFX:

```
.$HOME/cfx5.profile
cfx5
```

Modification of system setup files The system administrator modifies the setup files (normally `/etc/profile`), which are run by all users during login, to include the directory *cfxroot*`/bin` in the command search path. While this has the advantage of only one file needing to be modified in order to allow all users to use the software, it also:

- Affects users regardless of whether they use ANSYS CFX
- Can only be done by the root user
- Is system dependent

Refer to your system documentation for information about which files to change for your workstations.

5.1.2.1. Setting up ANSYS TurboGrid Release 14.0

Manual modification of individual user setup files To start ANSYS TurboGrid without using full pathnames every time you want to run the ANSYS TurboGrid software, your path must be altered to include the ANSYS TurboGrid directory. This can be done by adding the following line to the `.login` and `.cshrc` files in your home directory:

```
set path=(cfxroot/bin $path)
```

and these lines to the `.profile` file in your home directory, and also the `.bash_profile` if it exists:

```
PATH=cfxroot/bin:$PATH
export PATH
```

and these lines to the `.dtprofile` file in your home directory:

```
PATH=cfxroot/bin:$PATH \
export PATH
```

With the path altered in this way, you can start ANSYS TurboGrid in the current working directory by typing **`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.

5.1.2.2. Using the ANSYS CFX Launcher to Set Up Users

To create setup files to be merged or run manually, start the ANSYS CFX Launcher using the command:

```
cfxroot/bin/cfx5
```

and select **Tools> Configure User Startup Files**. This option runs `cfx-root/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.

5.1.2.3. 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.

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

5.1.3. Post-Installation Procedures for ANSYS FLUENT

To start ANSYS FLUENT without using full pathnames every time you want to run the FLUENT software, your path must be altered to include the ANSYS FLUENT `bin` directory. You can do this in the C shell and its derivatives by entering:

```
set path = (/ansys_inc/v140/fluent/bin $path)
```

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

```
PATH=(/ansys_inc/v140/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/v140/fluent/bin
   ```

3. Run the following command:

   ```
   <install_dir>/ansys_inc/v140/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 wish to start FLUENT at this time. The new defaults will have been saved.

Please refer to the *ANSYS FLUENT Quick Start Guide* for more information.

5.1.4. Post-Installation Procedures for ANSYS POLYFLOW

POLYFLOW no longer requires the **FLUENT_INC** environmental variable. It should be deleted as is recommended for FLUENT:

- Remove the **FLUENT_INC** setting from your `.cshrc`, `.profile`, or `.bashrc` file if you have added it for previous versions. Verify that the environment variable is unset by typing:

  ```
  printenv FLUENT_INC
  ```

This command should not return anything.

5.1.5. 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/v140/icemcfd/<*ICEMCFD_OS_DIR*>/bin

   ANSYS ICEM CFD operating system directories are:

Hardware	***ICEMCFD_OS_DIR***
Linux Itanium 64	linux64
Linux x64 (AMD Opteron)	linux64_amd
Linux x64 (EM64T)	linux64_amd

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

   ICEM_ACN - set to `/ansys_inc/v140/icemcfd/<ICEMCFD_OS_DIR>/bin`

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

5.1.6. Post-Installation Procedures for ANSYS AUTODYN

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

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

`/ansys_inc/v140/autodyn/bin`	path to the AUTODYN executable

Please refer to the ANSYS AUTODYN *Quick Start Guide* and *What's New* documents in the ANSYS AUTODYN help directory for information on starting and running ANSYS AUTODYN.

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/v140`
2. `<install_dir>/ansys_inc/v140/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 (ANSYS)

If your ANSYS sales representative has supplied you with message files in your local language, use the following procedures to install and access these files. You must create new message files for each release because error messages may occur in a different order for each release.

1. Create a language-named subdirectory (for example, `fr` for French) under the `/docu` directory:

   ```
   mkdir /ansys_inc/v140/ansys/docu/fr
   ```

2. Copy the message files msgcat.140, msgidx.140, and msgfnm.140 into that subdirectory.
3. Access these files from the Language Selection option of the launcher or via the **-l** command line option:

```
ansys140 -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/v140/ansys/bin/ansys140	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/v140/Framework/bin/<*platform*>/runwb2	
ANSYS CFX	/ansys_inc/v140/CFX/bin/cfx5	
ANSYS FLUENT	/ansys_inc/v140/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/v140/icemcfd/<*platform*>/bin/icemcfd	
ANSYS POLYFLOW	/ansys_inc/v140/polyflow/bin/polyman	Starts the POLYFLOW MANager. For any other tool, use /ansys_inc/v140/polyflow/bin/<*tool*>

Product	Command	Notes
ANSYS CFD-Post	`/ansys_inc/v140/CFD-Post/bin/cfdpost`	
ANSYS Icepak	`/ansys_inc/v140/Icepak/bin/icepak`	
ANSYS TurboGrid	`/ansys_inc/v140/TurboGrid/bin/cfxtg`	
ANSYS AUTODYN	`/ansys_inc/v140/autodyn/bin/autodyn140`	solver only

Chapter 6: Uninstalling the Software

6.1. Uninstalling ANSYS, Inc. Products

To uninstall a product, issue the following command:

```
/ansys_inc/v140/ans_uninstall140
```

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/v140/ansys/bin/ans_admin140
```

1. From the uninstall panel, choose one of the following uninstall options:

   - **Select Products to Uninstall**
   - **Uninstall all ANSYS Products**

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

2. If you chose **Select Products to Uninstall**, you will see a list of products that are installed. Select those products you want to uninstall and click **Continue**. Then click **OK** to confirm the list of products to be uninstalled.

   If you chose **Uninstall all ANSYS Products**, you will be asked to confirm that you want to uninstall all products. Click **Yes**. This process will remove all files and directories under and including the /ansys_inc/v140 directory.

3. When the uninstall has completed, click **Finish**.

In the case of a platform (file server) installation, the uninstall will remove the selected product(s) from all of the Linux platforms.

Chapter 7: 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 *Introduction to Import* in the *ANSYS Connection User's Guide*.

Caution

Be sure to install Mechanical APDL (ANSYS) and the connection functionality from the same release. If you attempt to run the latest connection functionality on a machine that is running an earlier release of Mechanical APDL, or vice versa, the connection may fail.

If you are running 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.

7.1. Using the CAD Configuration Manager

The **CAD Configuration Manager** utility allows you to configure geometry interfaces for ANSYS Workbench and ICEM CFD Direct CAD interfaces on Linux systems. CAD configuration is typically handled during the product installation; however, if you chose to skip those steps, or if you make changes to your local CAD configuration between releases (for example, you move or update your CAD package, or remove it entirely), you can use this utility. Note that only Reader mode for NX is supported 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 Engineering (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/v140/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.

7.1.1. 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.

7.1.2. 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/v140/commonfiles/CAD/bin/ansmono Ans.CadInt.CADConfigurationUtility.exe
     -arguments
```

Argument	Value	Comment
unconfigure	None	Results in any specified CAD sources being unconfigured. When this flag is absent, the **CAD Configuration Manager** will attempt to configure all designated CAD sources.
UG_CON-FIG_WB	None	Configure/unconfigure NX Geometry Interface to Workbench. The argument UGII_BASE_DIR must also be specified.
UGII_BASE_DIR	Full path to NX installation (quotations are required if there are spaces in the path).	This should agree with environment variable **UGII_BASE_DIR**. Not required with unconfigure operation.
UG_USE_COL-ORS	None	Process NX entity colors as possible source of attributes and named selections. Not required with unconfigure operation.

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:

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

where `installpath` is the `/v140` directory under the installation directory.

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

```
<installpath>/commonfiles/CAD/bin/ansmono  Ans.CadInt.CADConfigurationUtility.exe
-unconfigure -UG_CONFIG_WB
```

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

7.1.3. NX Configuration

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

- Registers the NX Reader for ANSYS Workbench by copying the file `UGNX#.Component.XML` from `/ansys_inc/v140/aisol/CADIntegration/UG` to either `/ansys_inc/v140/commonfiles/registry/linx64/append` when the configuration manager is in administrative mode or to `$HOME/.config/Ansys/140/UserRegFiles_NNNN/append`, when the **CAD Configuration Manager** is run by a non-administrator. The **CAD Configuration Manager** ensures that only one version of the file exists in the target location.

  `UGNX#` is the NX version detected by the **CAD Configuration Manager**, and *NNNN* is a numeric identifier appended to the `UserRegFiles` directory.

- You must specify the environment variable **UGII_BASE_DIR**, which must match the install location supplied to the **CAD Configuration Manager**. The WorkBench Reader for NX will not work without this variable set properly, as it is required for proper startup of the CAD.

The **CAD Configuration Manager** does not configure the ICEM CFD Direct CAD Interface to NX on Linux. To manually configure this product, set the environment variable **UGII_VENDOR_DIR** to `/ansys_inc/v140/icemcfd/linux64_amd/dif/ug/lib/ug/ugXX_vendor_dir`. Here, *XX* corresponds to the version of NX intended to load the NX Direct CAD Interface:

XX=22 for NX4
XX=23 for NX5

7.2. Configuring NX

To use NX in reader mode, you will not need to perform any additional configuration steps. Mechanical APDL (ANSYS) and ICEM CFD use the following three NX environment variables, which are set during the NX installation:

UGII_BASE_DIR
UGII_ROOT_DIR
UGS_LICENSE_SERVER

To use NX in plug-in mode, you need to set additional environment variables. ANSYS provides default settings for some of the necessary environment variables.

In general, you will not need to reset these variables. They are described below should you need to modify or reset them.

ANSCON_CONFIG_DIR: Sets the location of the `config.anscon.140` configuration file. This environment variable is used only if the **ANSYS140_DIR** has not been set. Defaults to `/ansys_inc/v140/ansys/ac4/data`.

UGII_VENDOR_DIR: This environment variable is not set during the installation process and can only be set in a user's start-up file or at command level before running NX. This environment variable defines the Mechanical APDL (ANSYS) or ICEM CFD menu for NX. You must set this environment variable if you will be running the connection for NX product from inside NX. This environment variable tells NX where to find the Mechanical APDL or ICEM CFD program.

Note

You cannot set the **UGII_VENDOR_DIR** for both Mechanical APDL and ICEM CFD. You must set it for only one product.

For Mechanical APDL, set this environment variable as follows:

```
setenv UGII_VENDOR_DIR /ansys_inc/v140/ansys/ac4/bin/ug50/<platform>
```

For ICEM CFD, set this environment variable as follows:

```
setenv UGII_VENDOR_DIR $ICEM_ACN/dif/ug/lib/ug/ugNN_vendor_dir
```

Where *NN* = 22 for NX4, 24 for NX6

UGII_OPTION: This environment variable sets the case of characters in directory names on the current system. You must set this environment variable if the directory names in this environment variable or in **ANSCON_CONFIG_DIR** contain uppercase characters. Possible options are MIXED, LOWER or UPPER. We recommend the following setting:

```
setenv UGII_OPTION MIXED
```

7.2.1. Configuring the Connection for NX

No additional configuration is needed to configure the connection capability for NX.

However, to export to Mechanical APDL (ANSYS) from NX, you will need the `config.anscon.140` file. This file is installed for all users in `/an-`

`sys_inc/v140/ansys/ac4/data/`. Here is a sample `config.anscon.140` file:

```
ANSYS_CMD                 /ansys_inc/v140/ansys/bin/ansys140
ANSYS_GRAPHIC_DEVICE      x11-stat
ANSYS_SOLVER              Sparse**
ANSYS_SELECTED_LAYERS     1-256**
ANSYS_GEOMETRY_TYPE       Solids Only**
ANSYS_NEUTRAL_FORMAT      Yes
ANSYS_PRODUCT_NAME        ANSYS
```

You can modify the `config.anscon.140` file to set information for all NX users. NX users can copy this file to their working directory and configure it for their own projects. NX users should see *Using the Configuration Editor* in the *ANSYS Connection User's Guide* for more information. See *Setting ANSYS Configuration Parameters* in the *ANSYS Connection User's Guide* for more information about the `config.anscon.140` file.

7.2.2. 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 8: Troubleshooting

8.1. Installation Troubleshooting

This section lists problems and error messages that you may encounter while installing and/or running ANSYS, Inc. products. After each situation description or error message is the user action required to correct the problem.

You can also find answers to commonly-asked questions on our customer portal. After you log in to the customer portal, select **Online Support> Installation/System FAQs**. Then select either the Windows or Linux link.

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

In addition, this appendix describes the **ansys_pid** utility, which is useful for troubleshooting some problems.

8.1.1. Using ANSLIC_ADMIN to Gather Diagnostic Information

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

8.1.2. 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.

8.1.3. 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 or follow the instructions in *Mounting the DVD Instructions for DVD Installations (Linux x64 Only)* (p. 18) to mount to a machine that does have a DVD drive. You can also mount the DVD on a Windows machine, then copy all of the files (using binary format) to the target Linux machine in the exact same directory structure as is on the DVD, and follow the instructions in *Downloading the Installation Files*. However, we strongly recommend against copying the Linux installation files from a Windows machine; such a file transfer can result in unpredictable behavior.

8.1.4. The Online Help System Does Not Run Properly

The online documentation for the Linux versions of the ANSYS ICEM CFD and ANSYS Icepak products uses the Oracle Help browser. Several unusual behaviors and their cause and resolution are listed.

If you request help and nothing happens:

- On a few machine configurations, the online help system can take up to 60 seconds longer than the product to initialize, due to the large amount of documentation available. If you request help within 60 seconds of running the product, you may simply need to wait a few more seconds for help to start.
- If you have minimized the help windows, activating help will not restore them to their original size. Instead of minimizing help, you can close the help windows when you are done using help. This does not exit the help, so the help system will still be available quickly next time you request help.
- The help system may have aborted due to a system error. You cannot restart the help system from your current session. You can save your work and restart a new session.

If the text in the online help is unclear (the fonts are too small, too large, or rough around the edges):

- Use the [+] or [-] keys to enlarge or reduce the fonts. Depending on your system's configuration and installed fonts, you may need to experiment with the [+] and [-] keys to adjust your font size. The [+] will increase the font size, and the [-] will decrease it. On some systems, you need to use the [+] and [-] keys on the standard keypad, rather than those on the number pad.

If, when starting ICEM CFD or Icepak over Exceed, all of the Linux window controls (borders, close buttons, etc.) disappear, or the help windows appear but are small and in the top left corner of the screen:

- You need to change your window manager settings in Exceed. See the section Note to Exceed Users earlier in this guide for information on changing your window manager settings.
- If changing the window manager settings does not correct the problem, we recommend upgrading to Exceed 7.0 or higher.
- If the problem still occurs, change the Window Manager to **X**. You must start an X Windows manager like mwm before running ICEM CFD or Icepak.

Mouse clicks are slow to initiate an action.

- This is a result of the mouse click signals traveling over the network via an Exceed connection. Try adjusting the Exceed **Performance** settings. See the section Note to Exceed Users earlier in this guide for information on specific changes we suggest.

8.1.5. CAD Configuration Manager Help Does Not Load

If the help for the **CAD Configuration Manager** does not load into your default browser, set the **BROWSER** environment variable to the path of your HTML viewer (such as Mozilla or Firefox) and restart the **CAD Configuration Manager**.

8.1.6. Cannot Enter Data in Text Fields

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

QT_IM_MODULE
XMODIFIERS
GTK_IM_MODULE

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

Red Hat If you are installing on Linux Red Hat 5 machines on the local console, and you are unable to enter text into any of the text fields in the installer, you may have to disable SCIM (Smart Common Input Method) as follows:

In KDE: Choose **[K Menu] >Settings >Input Method**

In GNOME: Choose **System >Preferences >More Preferences >Input Methods**.

In other window managers: Run program `im-chooser`.

Select **Never Use Input Methods**.

Then rerun the installation.

8.1.7. Download and Installation Error Messages

The current platform type is not selected and is not included in current download files. Please make sure that you are using the correct media or downloaded file. Continuing with a platform installation may require additional post-install configuration.

Do you wish to continue?

This message occurs if you have selected a platform for installation that does not match the files you are trying to install (either from the installation DVD or from downloaded installation files).

Cannot find file <*product*>.tar in directory <*dvd_dir*>

This error may appear during the ANSYS installation if you have entered the wrong DVD pathname. Check *Mounting the DVD Instructions for DVD Installa-*

tions (Linux x64 Only) (p. 18) 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 Linux-only message appears during an ANSYS, Inc. product installation if the installed license manager files are newer than the ones being installed. You should always use the newest files. However, due to system format changes or other unlikely scenarios, the date check could produce incorrect results. To override the date check and force the installation to always install the files from the media, regardless of the file dates, re-run the installation with the `-nodatecheck` option. We strongly recommend that you exercise caution when running the installation with the `-nodatecheck` option; installing older license files can result in licensing errors and the inability to run ANSYS, Inc. products.

8.1.8. System-related Error Messages

Error, could not open display.

Either the **DISPLAY** environment variable is not correct or the `xhosts` command was not properly set. See the *Basic Analysis Guide* for specific graphics information.

*****Error, ANSYS140_DIR environment variable is not set. This is a fatal error – exiting.**

This message indicates that the **ANSYS140_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.

8.1.9. High Performance Computing Error Messages

The following error messages are associated with the High Performance Computing solvers.

mpid: Error: HP MPI version incompatibility detected

You may encounter this or a similar message if you attempt to use ANSYS 14.0 with a different version of MPI than is supported. See the *Parallel Processing Guide* for a complete list of supported MPI versions.

8.2. Installation Troubleshooting - Mechanical APDL (ANSYS)

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

8.2.1. 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.'

8.2.2. Japanese/Chinese characters display in status bar windows on Mechanical APDL (ANSYS) on Red Hat AS 4.0

If you are running the traditional Mechanical APDL (ANSYS) interface and see Japanese/Chinese characters in the status bar windows, you will need to reinstall Red Hat, picking only English. This problem is caused by choosing to install "Everything" during the Red Hat Linux OS setup, thus installing all languages. If you then install Mechanical APDL and do a solve with the traditional interface, you will see Japanese or Chinese text in the solve status window. This bug will not be fixed.

8.2.3. Mechanical APDL (ANSYS) Documentation File for User Interface Error Messages

Missing or erroneous documentation files for user interface. Command ignored.

Verify that the documentation list file for the user interface exists in the `/ansys_inc/v140/ansys/gui/en-us/UIDL` subdirectory.

```
ls -l  /ansys_inc/v140/ansys/gui/en-us/UIDL/menulist140.ans
```

The system should respond with:

```
-rw-r--r-- 1 root 23 Jan  8 11:50 /ansys_inc/v140/ansys/gui
     /en-us/UIDL/menulist140.ans
```

Make sure that the pathnames in the `menulist140.ans` file are correct.

8.2.4. Launcher Error Messages

Some of the more common error messages follow. See the *ANSYS, Inc. Licensing Guide* for licensing-related launcher messages.

*****Cannot create required <*profile*> file. Therefore, cannot write to profile information during this launcher session.**

If you see this error, you cannot add or modify profile information during this launcher session. Verify that you have write access to the directory and restart the launcher session. Typically, this directory is `C:\Documents and Settings\<user name>\Application Data\Ansys\v140\launcher` on Windows or `~/.ansys/v140/launcher` on Linux.

8.2.5. FORTRAN Runtime Error Messages

The following error messages occur if the user is running Mechanical APDL (ANSYS) in a directory in which the user does not have write permission, or if Mechanical APDL files (i.e., `Jobname.RST`, `Jobname.DB`) exist in the current directory but the user does not have write permissions to the files. The specific messages that appear on each system are shown below.

8.2.5.1. Intel Linux 64 Systems

Input/Output Error 177: Creat Failure

In Procedure: fappnd

At Line: 72

Statement: Formatted WRITE

Unit: 19

8.2.5.2. Intel EM64T Linux x64 Systems

forrtl: Permission denied

forrtl: severe (9): permission to access file denied, unit 19, file /build/v140/ansys/objs

8.2.5.3. AMD Opteron Linux x64 Systems

*****ERROR**

Unable to open file /build/v140/ansys/objs/file.err for WRITE. Check directory and file permissions.

8.3. Installation Troubleshooting - ANSYS Workbench

8.3.1. Startup or Graphics Problems

To minimize graphics problems, always verify that you are running the latest graphics drivers provided by your computer's hardware manufacturer.

If you are running ANSYS Workbench on Linux and experience problems at startup or with the GUI or graphics displaying correctly, and you are running in accelerated graphics mode, you may need to relaunch ANSYS Workbench using the -oglmesa flag to activate software rendering:

```
runwb2 -oglmesa
```

If ANSYS Workbench detects that graphics problems are causing crashes, it will automatically switch to software rendering. ANSYS Workbench also will use software 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
```

If you are running under Exceed3D, try the following settings if you are having graphics problems:

- Turn off the graphics (hardware) acceleration option in Exceed3D options.
- If graphics acceleration is on, turn on the GLX 1.3 option.

Any version of Exceed that does not have the GLX 1.3 option is unlikely to function correctly with graphics acceleration.

8.4. Installation Troubleshooting - ANSYS CFX

8.4.1. 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.

8.5. 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 **About ANSYS> Contacts and Locations**. The direct URL is: http://www1.ansys.com/customer/public/supportlist.asp. Follow the on-screen instructions to obtain your support provider contact information. You will need your customer number. If you don't know your customer number, contact the ASC at your company.

If your support is provided by ANSYS, Inc. directly, Technical Support can be accessed quickly and efficiently from the ANSYS Customer Portal, which is available from the ANSYS Website (www.ansys.com) under **Support> Technical Support** where the Customer Portal is located. The direct URL is: http://www.ansys.com/customerportal.

One of the many useful features of the Customer Portal is the Knowledge Resources Search, which can be found on the Home page of the Customer Portal.

Systems and installation Knowledge Resources are easily accessible via the Customer Portal by using the following keywords in the search box: `Systems/Installation`. These Knowledge Resources provide solutions and guidance on how to resolve installation and licensing issues quickly.

NORTH AMERICA

All ANSYS, Inc. Products

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

Toll-Free Telephone: 1.800.711.7199

Fax: 1.724.514.5096

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

GERMANY

ANSYS Mechanical Products

Telephone: +49 (0) 8092 7005-55

Email: support@cadfem.de

All ANSYS Products

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

National Toll-Free Telephone:

German language: 0800 181 8499

English language: 0800 181 1565

International Telephone:

German language: +49 6151 3644 300

English language: +49 6151 3644 400

Email: support-germany@ansys.com

UNITED KINGDOM

All ANSYS, Inc. Products

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

Telephone: +44 (0) 870 142 0300

Fax: +44 (0) 870 142 0302

Email: support-uk@ansys.com

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

JAPAN

CFX , ICEM CFD and Mechanical Products

Telephone: +81-3-5324-8333

Fax: +81-3-5324-7308

Email: *CFX:* japan-cfx-support@ansys.com; *Mechanical:* japan-ansys-support@ansys.com

FLUENT Products

Telephone: +81-3-5324-7305

Email: *FLUENT:* japan-fluent-support@ansys.com; *POLYFLOW:* japan-polyflow-support@ansys.com; *FfC:* japan-ffc-support@ansys.com; *FloWizard:* japan-flowizard-support@ansys.com

Icepak

Telephone: +81-3-5324-7444

Email: japan-icepak-support@ansys.com

Licensing and Installation

Email: japan-license-support@ansys.com

INDIA

ANSYS Products (including FLUENT, CFX, ICEM-CFD)

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

Telephone: +91 1 800 233 3475 (toll free) or +91 1 800 209 3475 (toll free)

Fax: +91 80 2529 1271

Email: *FEA products:* feasup-india@ansys.com; *CFD products:* cfdsup-india@ansys.com; *Installation:* installation-india@ansys.com

FRANCE

All ANSYS, Inc. Products

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

Toll-Free Telephone: +33 (0) 800 919 225

Email: support-france@ansys.com

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

BELGIUM

All ANSYS Products

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

Telephone: +32 (0) 10 45 28 61

Email: support-belgium@ansys.com

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

SWEDEN

All ANSYS Products

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

Telephone: +44 (0) 870 142 0300

Email: support-sweden@ansys.com

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

SPAIN and PORTUGAL

All ANSYS Products

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

Telephone: +33 1 30 60 15 63

Email: support-spain@ansys.com

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

ITALY

All ANSYS Products

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

Telephone: +39 02 89013378

Email: support-italy@ansys.com

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

Configuring Distributed ANSYS

This section explains how to configure your network/cluster to run Distributed ANSYS. It is important that you follow these steps in order to run successfully in a distributed environment.

For more information on using Distributed ANSYS, see the *Parallel Processing Guide*.

You will need an ANSYS Mechanical HPC (HPC) license for each processor after the first two. For example, if you want to run four processors, you will need two HPC licenses. Some products cannot use all HPC functionality, as noted in the following table.

Table 1 Products with Limited HPC Functionality

Product	Distributed ANSYS Capability	SMP Capability	VT Accelerator Capability
ANSYS Academic Teaching Products	Yes	4 processors default limit	Yes
ANSYS LS-DYNA	No	No	No
ANSYS DesignSpace	No	2 processors max	No
ANSYS Professional NLS	No	Yes	Yes
Mechanical APDL Solver	Yes	Yes	Yes

1. Setting up Distributed ANSYS

This section describes the prerequisites, including software requirements, for running Distributed ANSYS and the steps necessary to set up the environment for Distributed ANSYS.

1.1. Prerequisites for Running Distributed ANSYS

Whether you are running on a single machine or multiple machines, the following condition is true:

- Distributed ANSYS allows you to use two cores without using any HPC licenses. Additional licenses will be needed to run Distributed ANSYS with more than two cores. Several HPC license options are available. For more information, see HPC Licensing in the *Parallel Processing Guide*.

If you are running on a single machine, there are no additional requirements for running Distributed ANSYS.

If you are running across multiple machines (e.g., a cluster), your system must meet these additional requirements to run Distributed ANSYS.

- Homogeneous network: All machines in the cluster must be the same type, OS level, chip set, and interconnects.
- You must be able to remotely log in to all machines, and all machines in the cluster must have identical directory structures (including the ANSYS installation, MPI installation, and on some systems, working directories). Do not change or rename directories after you've launched ANSYS. For more information on files that are written and their location, see Controlling Files that Distributed ANSYS Writes in the *Parallel Processing Guide*.
- All machines in the cluster must have ANSYS installed, or must have an NFS mount to the ANSYS installation. If not installed on a shared file system, ANSYS must be installed in the same directory path on all systems.
- All machines must have the same version of MPI software installed and running. The table below shows the MPI software and version level supported for each platform. For Linux platforms, the MPI software is included with the ANSYS installation. For Windows platforms, you must install the MPI software as described later in this document.

1.1.1. MPI Software

The MPI software supported by Distributed ANSYS depends on the platform. The following table lists the type of MPI software supported for each platform. Platform MPI and Intel MPI are included on the Linux installation media and are installed automatically when you install ANSYS. Instructions for installing the MPI software on Windows systems can be found later in this document (see Installing the Software).

Distributed ANSYS runs on the following platforms:

Intel IA-64 Linux (Platform MPI, SGI MPT)
Intel Xeon EM64T 64-bit Linux (Platform MPI, Intel MPI)
AMD Opteron 64-bit Linux (Platform MPI, Intel MPI)
Windows 32-bit (Platform MPI, Intel MPI)
Windows 64-bit (Platform MPI, MS MPI, Intel MPI)
Windows HPC Server 2008 x64 (Microsoft HPC Pack (MS MPI))

Table 2 Platforms and MPI Software

Platform	MPI Software	More Information
Linux, Intel IA-64	Platform MPI 8.1.2 SGI MPT 1.14 (SuSE 10) SGI MPT 1.26 (SuSE 11)	Platform MPI: http://www.platform.com/cluster-computing/platform-mpi SGI MPT
Linux, Intel Xeon EM64T and AMD Opteron 64-bit	Platform MPI 8.1.2 Intel MPI 4.0.2	Platform MPI: http://www.platform.com/cluster-computing/platform-mpi Intel MPI: http://software.intel.com/en-us/articles/intel-mpi-library-documentation/
Windows 32-bit / Windows XP / Windows Vista / Windows 7 Windows 64-bit / Windows XP x64 / Windows Vista x64 / Windows 7 x64	Platform MPI 8.1.2 Intel MPI 4.0.1	Platform MPI: http://www.platform.com/cluster-computing/platform-mpi Intel MPI: http://software.intel.com/en-us/articles/intel-mpi-library-documentation/
Windows HPC Server 2008 x64	Microsoft HPC Pack (MS MPI)	http://www.microsoft.com/hpc/

ANSYS LS-DYNA If you are running ANSYS LS-DYNA, you can use LS-DYNA's parallel processing (MPP or SMP) capabilities. Use the launcher or the command line method as described in *Activating Distributed ANSYS* in the *Parallel Processing Guide* to run LS-DYNA MPP. For Windows and Linux systems, please see the following table for LS-DYNA MPP MPI support. For more information on using ANSYS LS-DYNA in general, and its parallel processing capabilities specifically, see the *ANSYS LS-DYNA User's Guide*.

Table 3 LS-DYNA MPP MPI Support on Windows and Linux

MPI version for DYNA MPP	32-bit Windows	64-bit Windows	64-bit Linux
Platform MPI	n/a	X	X
MS MPI	n/a	X	n/a

1.1.2. Installing the Software

To run Distributed ANSYS on a cluster, you must install ANSYS on all machines in the cluster, or have an NFS mount to the ANSYS installation. Install ANSYS following the instructions in the *ANSYS, Inc. Installation Guide* for your platform. Be sure to complete the installation, including all required post-installation procedures. On Windows systems, you must use the Universal Naming Convention (UNC) for all file and path names for Distributed ANSYS to work correctly.

Installing Platform MPI on Windows

You can install Platform MPI from the installation launcher by choosing **Install MPI for ANSYS, Inc. Parallel Processing**. On the following screen, choose to install Platform MPI. The Platform MPI installation program will start. A Platform MPI installation README file will open simultaneously. Follow the instructions in the README file as you complete the Platform MPI installation.

The instructions for installing Platform MPI are also found in the ANSYS installation directory in the following README files:

```
Program Files\Ansys Inc\V140\commonfiles\MPI\Platform\8.1.2\Windows\INSTALL_PLATFORM-MPI_README.mht
```

or

```
Program Files\Ansys Inc\V140\commonfiles\MPI\Platform\8.1.2\Windows\INSTALL_PLATFORM-MPI_README.doc
```

Installing Intel MPI on Windows

You can install Intel MPI from the Installation launcher by choosing **Install MPI for ANSYS, Inc. Parallel Processing**. On the following screen, choose to install Intel MPI. The Intel MPI installation program will start. An Intel MPI installation README file will open simultaneously. Follow the instructions in the README file as you complete the Intel MPI installation.

The instructions for installing Intel MPI are also found in the ANSYS installation directory in the following README files:

`Program Files\Ansys Inc\V140\commonfiles\MPI\Intel\4.0.2\Windows\INSTALL_INTEL-MPI_README.mht`

or

`Program Files\Ansys Inc\V140\commonfiles\MPI\Intel\4.0.2\Windows\INSTALL_INTEL-MPI_README.doc`

Microsoft HPC Pack (Windows HPC Server 2008)

You must complete certain post-installation steps before running Distributed ANSYS on a Microsoft HPC Server 2008 system. The post-installation instructions provided below assume that Microsoft HPC Server 2008 and Microsoft HPC Pack (which includes MS MPI) are already installed on your system. The post-installation instructions can be found in the following README files:

`Program Files\Ansys Inc\V140\commonfiles\MPI\MicrosoftHPC2008\README.mht`

or

`Program Files\Ansys Inc\V140\commonfiles\MPI\MicrosoftHPC2008\README.docx`

Microsoft HPC Pack examples are also located in `Program Files\Ansys Inc\V140\commonfiles\MPI\MicrosoftHPC2008`. Jobs are submitted to the Microsoft HPC Job Manager either from the command line or the Job Manager GUI.

To submit a job via the GUI, go to **Start> All Programs> Microsoft HPC Pack> HPC Job Manager**. Then click on **Create New Job from Description File**.

1.2. Setting Up the Cluster Environment for Distributed ANSYS

After you've ensured that your cluster meets the prerequisites and you have ANSYS and the correct version of MPI installed, you need to configure your distributed environment using the following procedure.

1. Obtain the machine name for each machine on the cluster.

   Windows:
   Right-click on **My Computer**, left-click on **Properties**, and select the **Network Identification** or **Computer Name** tab. The full computer name will be listed. Note the name of each machine (not including the domain).

   Linux:
   Type **`hostname`** on each machine in the cluster. Note the name of each machine. You will need this name to set up the `.rhosts` file, as well as for the **ANS_ADMIN** utility.

2. **Linux only:** Set up the `.rhosts` file on each machine. The `.rhosts` file lists all machines in the cluster. The machines should be listed using their complete system name, as taken from **`hostname`**. For example, an `.rhosts` file for a two-machine cluster might look like this:

   ```
   golinux1.ansys.com jqd
   golinux2 jqd
   ```

   Change/verify `.rhosts` file permissions on all machines by issuing:

   ```
   chmod 600 .rhosts
   ```

   Verify communication between machines via rsh or ssh (e.g., **`rsh golinux2 ls`**). You should not be prompted for a password. If you are, check the `.rhosts` permissions and machine names for correctness. For more information on using remote shells, see the man pages for rsh or ssh.

3. If you want the list of machines to be populated in the Mechanical APDL Product Launcher, you need to configure the `hosts140.ans` file. You can use the **ANS_ADMIN** utility to configure this file. You can manually modify the file later, but we strongly recommend that you use **ANS_ADMIN** to create this file initially to ensure that you establish the correct format.

   Windows:
   Start >Programs >ANSYS 14.0 >Utilities >ANS_ADMIN 14.0

Choose **Configuration options**, and then **Configure Cluster** to configure the `hosts140.ans` file.

1. Specify the directory in which the `hosts140.ans` will be configured: Select the **Configure a `hosts140.ans` file in a directory you specify** option and click **OK**. Enter a working directory. Click **OK**.
2. Enter the system name (from Step 1) in the **Machine hostname** field and click **Add**. On the next dialog box, enter the system type in the **Machine type** drop-down, and the number of processors in the **Max number of jobs/processors** field and click **OK** for each machine in the cluster. When you are finished adding machines, click **Close**, then **Exit**.

   An example `hosts140.ans` file where machine1 has 2 processors and machine2 has 4 processors would look like this:

   ```
   machine1 intel 0 2 0 0 MPI 1 1
   machine2 intel 0 4 0 0 MPI 1 1
   ```

Linux:

`/ansys_inc/v140/ansys/bin/ans_admin140`

Choose **ANSYS/Workbench Configuration**, and then click **Configure Cluster**. Under **Select file to configure**, choose the `hosts140.ans` file to be configured and choose **Configure for Distributed ANSYS**. Click **OK**. Then enter the system name (from Step 1) in the **Machine hostname** field and click **Add**. On the next dialog box, enter the system type in the **Machine type** drop-down, and the number of processors in the **Max number of jobs/processors** field for each machine in the cluster. Click **Add**. When you are finished adding machines, click **Close**.

The `hosts140.ans` should be located in your current working directory, your home directory, or the `apdl` directory.

4. **Windows only:** Setting up environment variables.

   On the head node, where ANSYS and ANSYS Licensing is installed, set the following:

   ANSYS140_DIR=C:\Program Files\ANSYS Inc\v140\ansys

   ANSYSLIC_DIR=C:\Program Files\ANSYS Inc\Shared Files\Licensing

   where `C:\Program Files\ANSYS Inc` is the location of the product install. If your installation location is different than this, specify it instead.

On Windows systems, you must use the Universal Naming Convention (UNC) for all ANSYS environment variables on the compute nodes for Distributed ANSYS to work correctly.

On the compute nodes, set the following:

ANSYS140_DIR=*head_node_machine_name*\ANSYS Inc\v140\ansys

ANSYSLIC_DIR=*head_node_machine_name*\ANSYS Inc\Shared Files\Licensing

For Distributed LS-DYNA:

On the head node and the compute nodes, set **LSTC_LICENSE** to ANSYS. This tells the LS-DYNA executable to use ANSYS licensing.

Since the LS-DYNA run will use ANSYS licensing for LS-DYNA, you do not need to set **LSTC_LICENSE_SERVER**.

5. **Windows only:** Share out the `ANSYS Inc` directory on the head node with full permissions so that the compute nodes can access it.

1.2.1. Optional Setup Tasks

The tasks explained in this section are optional. They are not required to get Distributed ANSYS to run correctly, but they may be useful for achieving the most usability and efficiency, depending on your system configuration.

On Linux systems, you can also set the following environment variables:

- **ANSYS_NETWORK_START** - This is the time, in seconds, to wait before timing out on the start-up of the client (default is 15 seconds).
- **ANSYS_NETWORK_COMM** - This is the time to wait, in seconds, before timing out while communicating with the client machine (default is 5 seconds).
- **ANS_SEE_RUN_COMMAND** - Set this ANSYS environment variable to 1 to display the actual mpirun command issued from ANSYS.

On Linux systems running Platform MPI:

- **MPI_REMSH** - This is the path to the remote shell (ssh or rsh). Set this environment variable to specify a full path to a remote shell. For example, setting **`MPI_REMSH = /usr/bin/ssh`** will use ssh instead of the default remote shell (rsh). Note that selecting the **Use Secure Shell instead of Remote Shell** option on the launcher will override **MPI_REMSH**, if **MPI_REMSH** is not set or is set to a different location. You can also issue the `- usessh` command line option to use ssh instead of rsh. The command line option will override the environment variable setting as well.

- **MPI_WORKDIR** - Set this environment variable to specify a working directory on either the master and all nodes, or on specific nodes individually. For more information, see Controlling Files that Distributed ANSYS Writes.
- **MPI_IC_ORDER** - Set this environment variable to specify the order in which the interconnects on the system are to be used. The interconnects will be tried in the order listed from left to right. If an interconnect is listed in uppercase, no interconnects listed after that one will be tried. If **MPI_IC_ORDER** is not set, the fastest interconnect available on the system is used. See the Platform MPI documentation for more details.
- **MPI_ICLIB_<*interconnect*>** - Set this environment variable to the interconnect location if the interconnect is not installed in the default location:

  ```
  setenv MPI_ICLIB_GM <path>/lib64/libgm.so
  ```

  See the Platform MPI documentation for the specific interconnect names (e.g., **MPI_ICLIB_GM**).
- **MPIRUN_OPTIONS** - Set this environment variable to -prot to display a grid of interconnects among the systems being used for distributed processing.

On Linux systems running Intel MPI:

- Issue the command line option -usessh to use ssh instead of rsh.
- See the Intel MPI reference manual (for Linux) for further information and additional environment variables and their settings: http://software.intel.com/en-us/articles/intel-mpi-library-documentation/.

To verify that these environment variables are set correctly on each machine, run:

```
rsh machine1 env
```

On Windows systems, you can set the following environment variables to display the actual mpirun command issued from ANSYS:

- **ANS_SEE_RUN** = TRUE
- **ANS_CMD_NODIAG** = TRUE

1.2.2. Using the mpitest Program

The `mpitest` program performs a simple communication test to verify that the MPI software is set up correctly. The `mpitest` program should start without errors. If it does not, check your paths, `.rhosts` file, and permissions; correct any errors, and rerun.

When running the mpitest program, you must use an even number of nodes.

On Linux:

For Platform MPI (default), issue the following command:

```
mpitest140 -machines machine1:2
```

For Intel MPI, issue the following command:

```
mpitest140 -mpi intelmpi -machines machine1:2
```

You can use any of the same command line arguments (such as `-machines`) with the `mpitest` program as you can with Distributed ANSYS.

On Windows:

Issue the following command to run a local test on Windows using Platform MPI:

```
ansys140 -np 2 -mpitest
```

Use the following procedure to run a distributed test on Windows using Platform MPI:

1. Create a file named `machines` in your local/home directory. Open the `machines` file in an editor.
2. Add your master and slave machines in your cluster. For example, in this cluster of two machines, the master machine is **`gowindows1`**. List the machine name separately for each core on that machine. For example, if **`gowindows1`** has four processors and **`gowindows2`** has two, the `machines` file would look like this:

   gowindows1
   gowindows1
   gowindows1
   gowindows1
   gowindows2
   gowindows2

3. From a command prompt, navigate to your working directory. Run the following:

```
ansys140 -mpifile machines -mpitest
```

1.2.3. Interconnect Configuration

Low-end hardware, such as slow interconnects, will reduce the speed improvements you see in a distributed analysis. For optimal performance, we typically recommend that you use an interconnect with a communication speed of 1000 megabytes/second or higher.

Distributed ANSYS supports the following interconnects. Not all interconnects are available on all platforms; see http://www.ansys.com/services/ss-platform-support.asp for a current list of supported interconnects. Other interconnects may work but have not been tested.

- InfiniBand (recommended)
- Myrinet
- GigE
- Ethernet (not recommended)

Interconnects plug into a PCI (Peripheral Component Interconnect), PCI-X (extended), or PCIe (PCI Express) slot on the system. You will need a PCI-X or a PCIe slot for the faster interconnects to accommodate the higher speeds.

Hardware for specific types of interconnects is generally incompatible with other proprietary interconnect types (except Ethernet and GiGE).

Systems can have a network of several different types of interconnects. Each interconnect must be assigned a unique hostname and IP address.

On Windows x64 systems, use the Network Wizard in the Compute Cluster Administrator to configure your interconnects. See the Compute Cluster Pack documentation for specific details on setting up the interconnects. You may need to ensure that Windows Firewall is disabled for Distributed ANSYS to work correctly.

2. Running a Distributed Job

For information on running Distributed ANSYS after you have your environment configured, see the *Parallel Processing Guide*.

Configuring ANSYS CFX Parallel

If you have purchased the ANSYS CFX-Solver parallel option, you will need to follow the procedures in this chapter to ensure that users can execute ANSYS CFX jobs in parallel. Several parallel communication methods are available that use either PVM (Parallel Virtual Machine) or MPI (Message Passing Interface). Platform-specific versions of MPI are available in some instances.

The ANSYS CFX-Solver parallel option is supported on all platforms that support the CFX-Solver.

1. ANSYS CFX UNIX Parallel Setup

The following are required in order for you to execute ANSYS CFX parallel:

- The ANSYS CFX-Solver must be installed on both master and slave nodes.
- To run distributed parallel (where slave processes run on a different host to the master process), remote or secure shell access must be available from the master nodes (systems on which parallel runs will be started) to slave nodes (systems on which the ANSYS CFX-Solver will actually run). See *Setting Up Remote Access on UNIX/Linux* (p. 83), below.
- You must have the same user name on all systems.
- The hostinfo.ccl file should be set up, as described in *hostinfo.ccl File* (p. 85).

1.1. Setting Up Remote Access on UNIX/Linux

Each system that will be used as a slave node must be configured to allow access via remote shell commands from the master node. This can be done globally for all users or on a per-user basis.

Often, networks where rlogin and rsh or ssh are used frequently will already be configured to allow remote access to all users. If this is the case, nothing more needs to be done.

The CFX5RSH environment variable is used to select either ssh or rsh access. The default is rsh. If you want to use ssh, then set "CFX5RSH=ssh" either in the environment or in the cfx5rc file (see Resources Set in cfx5rc Files in the *CFX Introduction*).

In either case, rsh or ssh, the remote machine must not prompt for a password when you run remote commands.

1.1.1. Testing Remote Access

You can test remote access using rsh for a UNIX/Linux slave node using the command:

```
rsh unixhost echo working
```

Note

On systems running HP-UX, use "remsh" instead of "rsh".

You can test remote access using ssh using the command:

```
ssh unixhost echo working
```

1.1.2. Global Set Up of rsh

This method, which is used to allow remote access for all users, depends on whether NIS is used to maintain netgroups, as well as common password and group databases. If this is not the case, then you should log in to each slave node as root and create a file called `/etc/hosts.equiv` containing a line:

```
<master>
```

where *<master>* is the hostname of the master node. See your system documentation for more information about the use of rsh and the syntax of the `hosts.equiv` file.

If NIS is used to control remote access, then a netgroup must be created for users of ANSYS CFX by the root user on the NIS master server, and a line such as the one below added to `/etc/hosts.equiv` on each slave node by the root user:

```
+@<netgroup>
```

where <*netgroup*> is the name of a netgroup to which users of ANSYS CFX belong. A detailed description of how to configure NIS is beyond the scope of this manual. Please see your system documentation for more information about NIS.

1.1.3. Individual User Set Up for rsh

Individual users can create a file called `.rhosts` in their home directory on each slave containing a line:

```
<master> <user>
```

where <*master*> is the hostname of the master and <*user*> is their username on the master. This file should be made readable only by the user, for example, by running:

```
chmod 600 ~/.rhosts
```

1.1.4. Set Up of ssh

If you use ssh for remote access, please consult your system documentation on how to set up ssh between machines so that it does not require the user to enter a password.

1.2. hostinfo.ccl File

In order to use the Distributed Parallel mode of ANSYS CFX, the file `hostinfo.ccl` must exist in the <CFXROOT>`/config/` directory of the ANSYS CFX installation on the master node and be made readable by all users of the software. This file is a database containing information about the available hosts and where ANSYS CFX has been installed on each of them. The file is used by the ANSYS CFX-Solver when constructing a parallel run.

This file is written using the CFX Command Language. It defines a set of HOST DEFINITION objects, one for each available node. For example:

```
SIMULATION CONTROL:
  EXECUTION CONTROL:
    PARALLEL HOST LIBRARY:
      HOST DEFINITION: kangaroo
        Installation Root = /ansys_inc/v%v/CFX
      END
      HOST DEFINITION:wallaby
        Installation Root = /usr/local/cfx
        Host Architecture String = linux-amd64
        Number of Processors = 16
        Relative Speed = 1.7
```

```
      END
      HOST DEFINITION: mypc
        Remote Host Name = my_pc
      END
    END
  END
END
```

Note

The `SIMULATION CONTROL...END` wrapper is a requirement for Release 12.0 and later. If you manually create your `hostinfo.ccl` file, you must ensure that this wrapper is present.

None of the values for each host are mandatory, and the following empty host definition is perfectly valid:

```
HOST DEFINITION: parakeet
END
```

Host names are limited to the set of valid CCL object names. In particular, they must not contain full stops (.) or underscores (_) and must start with a letter.

If a `hostinfo.ccl` file does not already exist when ANSYS CFX is installed, one will be created containing the installation host. You can add hosts to the `hostinfo.ccl` file using the cfx5parhosts utility, or by modifying the file using a text editor. Individual users may also create their own versions of this file in:

```
~/.hostinfo.ccl
```

which will be used in preference if it exists.

For most installations, it will be necessary to supply only the Installation Root parameter, which should point to the <CFXROOT> directory in which ANSYS CFX is installed. On mixed networks, you may find it useful to supply the Number of Processors and/or Relative Speed parameters. A little time may be saved at startup by giving the Host Architecture String parameter explicitly, for example, using the `cfx5parhosts` utility.

> **Tip**
>
> If an individual user wants to use a host that is not present in either the `hostinfo.ccl` in the installation `config` directory or the user's own version of this file, then the user can add this host to the list for a particular run by using the CFX-Solver Manager. However, this would have to be done each time a run is started.

The available parameters for the hostinfo.ccl file are as follows:

Installation Root
: This is set to the <CFXROOT> installation directory on this host. If it is set to the special string `none`, this indicates that there is no ANSYS CFX installation on the remote host, which can sometimes be useful if only the solver binary is available.

Host Architecture String
: ANSYS CFX will use this value to select the most appropriate solver executable for this node. These strings can be obtained by giving the command **`<CFXROOT>/bin/cfx5info -os`** on the node in question. When these short **`os`** values (for example, linux-amd64) are given in this position, the generic solver executable will always be used for this host. The string can also be set to the actual architecture of the remote host (for example, intel_xeon64.sse2_linux2.3.4), which is determined by giving the command **`<CFXROOT>/bin/cfx5info -arch`**. If these longer strings are used then CFX could use this information to select between solvers optimized for specific architectures. However, since there are currently no solvers optimized for specific architectures, this extra information is currently unused.

Number of Processors
: As implied, this is the number of processors on the machine. It is used for display purposes only and can be safely omitted.

Relative Speed
: The Relative Speed is a real number that is used by the ANSYS CFX-Solver when partition sizes are calculated. The ratio of relative speeds of each host is used to determine the size of each partition. As an example, consider a parallel run involving two machines, one with a relative speed of 2.0 and the other with a relative speed of 1.0. The faster machine would work on a partition size twice as large as the slower machine in this case.

The numbers themselves are arbitrary; as a guide you may wish to set 1.0 for a 1GHz processor, 0.75 for a 750 MHz processor, and so on. If a relative speed entry is not entered, a default of 1.0 is assumed for that host.

You can obtain relative speed values using the cfx5parhosts utility.

Remote Host Name

To include hosts in the parallel run with names that contain, for example, underscore characters, you can add the "Remote Host Name" parameter to the HOST DEFINITION with the correct network name for the host and use a readable alias as the name of the object.

Solver Executable

A solver executable may be explicitly supplied, if necessary. This is usually only required when using Installation Root = none, and is recommended for advanced users only. The following substitutions are made on the string:

%r	root directory of the installation
%p	parallel suffix for the executable
%v	version of ANSYS CFX being run
%o	operating system string
%a	architecture subdirectory specification; for example, linux/double

If it is not supplied, this parameter has the default value %r/bin/%a/solver%p.exe

1.2.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility

You can add new hosts to ANSYS CFX's database for parallel runs using the cfx5parhosts utility:

```
CFXROOT\bin\cfx5parhosts argument list
```

where `argument list` corresponds to one or more of the arguments listed below:

Argument	Description
-add `host`[,`host`,...]	Add information about the named hosts to the file. This assumes that ANSYS CFX is installed in the same directory on each listed host as on the host on which you are running.

Argument	Description
	`host` may be specified as [*user@*]*hostname*[:cfx-5 root] if the user name or the ANSYS CFX installation root directory differs from the local host.
-benchmark	Runs a standard benchmark case, and fills in the Relative Speed for the local host. The benchmark case will usually take less than 5 minutes to run.
-file *file*	Use the specified file as the `hostinfo.ccl` file.
-merge *file*	Merge host information from the named file.
-no-update	After modifying the file, write back the information available without attempting to fill in any missing pieces.
-strict	Used with `-update`. Normally, hosts which exist on the network but cannot be connected to with rsh or ssh are included in the resulting file with a comment. This switch will exclude these hosts.
-system	Use the system host file. This is the default.
-update	Updates the specified host information file. If any hosts do not have an architecture specified in the existing `hostinfo.ccl` file (for example, because it was added via the `-add` switch), it will connect to the host and query it to fill in the Host Architecture String parameter. This is the default behavior. Note that if the Installation Root parameter is incorrect for the host, it will use the standard system commands to guess a generic architecture string. This can happen if you use `-add` to include a host with a different installation directory than the local one.
-user	Use the per-user host file.

1.3. Using Platform MPI (Message Passing Interface Library)

Most UNIX/Linux systems support three parallel run modes: PVM, MPICH, and Platform MPI. Platform MPI is the preferred parallel run mode on all the supported ANSYS CFX platforms.

Platform MPI supports several communication modes including shared memory and networked TCP/IP (Ethernet), IuDAPL, VAPI, OpenFabrics (IB), GM-2, MX (Myrinet), Elan4 (Quadrics), QLogic InfiniPath, and EtherFabric (Level 5 Networks).

Platform MPI is automatically installed with ANSYS CFX in the `<CFX-ROOT>/tools` directory. You do not need a license from Platform to run the ANSYS CFX-Solver using Platform MPI.

If there is no CFX installation on any slave node, you must ensure that Platform MPI is manually installed on any such node.

Platform MPI run modes are selected in the ANSYS CFX-Solver Manager or on the command line (using the `-start-method` command line option) using the “Platform MPI Local Parallel” or “Platform MPI Distributed Parallel” start method options.

For Platform MPI, the `–cpu_bind` option is used by default and binds processes to cores using a cyclic order. This leads to better performance but, on multi-user clusters having scheduling systems that do not enforce exclusive use of cluster nodes, may cause problems because multiple jobs may be bound to the same CPU on the same node. In this case, the `–cpu_bind` argument to `mpirun` should be removed from the Platform MPI start methods in the `<CFX-ROOT>/etc/start_methods.ccl` file.

1.3.1. Environment Variables

ANSYS CFX uses the environment variable CFX5_PCMPI_DIR to select which Platform MPI installation is used. The default setting points to the version installed by ANSYS CFX in the `<CFXROOT>/../commonfiles/MPI/Platform` directory. If you want to use a different Platform MPI version than what is installed then you can install that version in an alternative location and set CFX5_PCMPI_DIR to that location instead of the default.

There are several environment variables that can be used to control Platform MPI (documented in the Platform MPI documentation) that may provide additional flexibility not directly accessible through the ANSYS CFX startup scripts. If these environment variables are set the startup scripts automatically set them for your CFX runs. Some useful environment variables include:

MPI_REMSH

Enables you to change which method is used by Platform MPI to spawn jobs on remote machines. The Platform MPI default is ssh. However, by default the CFX software sets this to be the same as the setting used for CFX5RSH, and rsh if CFX5RSH is not set. This variable is applicable only to Linux and HP-UX.

MPI_IC_ORDER

Enables you to change the default order in which Platform MPI searches for interconnects. See the next section.

1.3.2. Interconnect Selection

Platform MPI start methods use the default communication mode selection built into Platform MPI. The default order in which Platform MPI checks for interconnects is OpenFabric, VAPI, UDAPL, ITAPI, PSM, Myrinet MX, Myrinet GM, Elan, TCP/IP.

If you want to force selection of a specific device, then you can either modify the start command in the `etc/start-methods.ccl` file and add the necessary command line flags (for example: -GM to force selection of the Myrinet interconnect) or use the appropriate Platform MPI environment variables.

Some useful command line options include forcing other interconnect types (-TCP, -HMP, -ELAN, -ITAPI, -VAPI, -UDAPL, and so on), using prun on systems with quadrics support (-prun), XC support (-srun) and setting the subnet you want to use (-subnet, sometimes useful for selecting particular interconnects as well if your system is configured that way).

For some Linux-based cluster environments not supported by Platform MPI, it is possible that the Myrinet, Infiniband or Quadrics hardware drivers are not located where Platform MPI expects. If Platform MPI cannot detect the drivers for fast interconnects, it will generally default back to the lowest speed connection that it can detect (usually TCP/IP socket communication).

The following table gives information on the default search path Platform MPI uses to find drivers for the various communication interfaces:

Interconnect	1st attempt	2nd attempt	3rd attempt
Infiniband (IB)	Environment variable MPI_ICLIB_ITAPI	libitapi.so	/usr/voltaire/lib/libitapi.so
Myrinet (GM)	Environment variable MPI_ICLIB_GM	libgm.so OR libgm32.so	/opt/gm/lib/libgm.so *OR* /opt/gm/lib/libgm32.so
ELAN	Environment variable MPI_ICLIB_ELAN	libelan.so	none
UDAPL	Environment variable MPI_ICLIB_UDAPL	libdat.so	none

Intercon-nect	1st attempt	2nd attempt	3rd attempt
VAPI	Environment variable MPI_ICLIB_VAPI	Environment variable MPI_ICLIB_VAPIDIR	libmtl_common.so,libmpga.so, libmosal.so, libvapi.so
Infinipath	Environment variable MPI_ICLIB_PSM	libpsm_infinipath.so.1	/usr/lib64/ libpsm_infinipath.so.1

For example, Platform MPI expects that GM stack shared libraries for Myrinet interconnects are located either in the default library search path or in the /opt/gm/lib (32 bit x86) or /opt/gm/lib64 (64 bit x86) directories. If the shared libraries are not in either location, Platform MPI will not take advantage of the high-speed interconnects. To fix this, you will have to copy the shared libraries into the correct location or set the environment variable MPI_ICLIB_GM to point to the correct GM stack shared library.

2. ANSYS CFX Windows Parallel Setup

The following are required in order for you to use ANSYS CFX in parallel on Windows platforms:

- ANSYS CFX-Solver must be installed on all nodes to be used for a parallel run.
- To run distributed parallel (where slave processes run on hosts other than the host with the master process), you must either configure the parallel host setup properly or set up remote access to slave nodes.
- You must have the same user name on all systems.
- To run distributed parallel, you must install and configure the Platform MPI service and register your user name on all nodes involved in the parallel run. (See *Installing the Platform MPI Service and Registering Users on Windows Vista* (p. 98).)

  You must register your user name each time your password changes.

2.1. hostinfo.ccl file

In order to use the Distributed Parallel modes effectively, the file `hostinfo.ccl` must exist in the `<CFXROOT>/config/` directory of the ANSYS CFX installation on the master node and be made readable by all users of the software. This file is a database containing information about the available nodes and where ANSYS

CFX has been installed on each of them. The file is used by the ANSYS CFX-Solver when constructing a parallel run.

The `hostinfo.ccl` file is written using the CFX Command Language. It defines a set of HOST DEFINITION objects, one for each available node. For example:

```
SIMULATION CONTROL:
  EXECUTION CONTROL:
    PARALLEL HOST LIBRARY:
      HOST DEFINITION: hostname1
        Installation Root = C:\Program Files\ANSYS Inc\v%v\CFX
        Host Architecture String = winnt-amd64
      END # HOST DEFINITION hostname1
    END # PARALLEL HOST LIBRARY
  END # EXECUTION CONTROL
END # SIMULATION CONTROL
```

Note

The `SIMULATION CONTROL...END` wrapper is a requirement for Release 12.0 and later. If you manually create your `hostinfo.ccl` file, you must ensure that this wrapper is present.

None of the values for each host are mandatory. For example, the following empty host definition is perfectly valid:

```
HOST DEFINITION: parakeet
END
```

Host names are limited to the set of valid CCL object names. In particular, they must not contain full stops (.) or underscores (_) and must start with a letter.

If a `hostinfo.ccl` file does not already exist when ANSYS CFX is installed, one will be created containing the installation host. You can add hosts to the `hostinfo.ccl` file using the cfx5parhosts utility, or by modifying the file using a text editor.

For most installations, it will be necessary only to supply the Installation Root parameter, which should point to the <CFXROOT> directory in which ANSYS CFX is installed. On mixed networks, you may find it useful to supply the Number of Processors and/or Relative Speed parameters. A little time may be saved at startup by giving the Host Architecture String parameter explicitly, for example, using the cfx5parhosts utility.

The available parameters are as follows:

Installation Root

This is set to the <CFXROOT> installation directory on this host. If it is set to the special string none, this indicates that there is no ANSYS CFX installation on the remote host, which can sometimes be useful if only the solver binary is available.

Host Architecture String

This should be set to the actual architecture <arch> of the remote host. ANSYS CFX will use this value to select the most appropriate solver executable for this node. These strings can be obtained by giving the command **`<CFX-ROOT>/bin/cfx5info -os`** on the node in question. If the shorter <os> values (for example, solaris) are given in this position, the generic solver executable will always be used for this host.

Number of Processors

As implied, this is the number of processors on the machine. It is used for display purposes only and can be safely omitted.

Relative Speed

The Relative Speed is a real number that is used by the ANSYS CFX-Solver when partition sizes are calculated. The ratio of relative speeds of each host is used to determine the size of each partition. As an example, consider a parallel run involving two machines, one with a relative speed of 2.0 and the other with a relative speed of 1.0. The faster machine would work on a partition size twice as large as the slower machine in this case.

The numbers themselves are arbitrary; as a guide you may want to set 1.0 for a 1GHz processor, 0.75 for a 750 MHz processor, and so on. If a relative speed entry is not entered, a default of 1.0 is assumed for that host.

You can obtain relative speed values using the cfx5parhosts utility.

Remote Host Name

To include hosts in the parallel run with names that contain, for example, underscore characters, you can add the "Remote Host Name" parameter to the HOST DEFINITION with the correct network name for the host and use a readable alias as the name of the object.

Solver Executable

A solver executable may be explicitly supplied, if necessary. This is usually only required when using Installation Root = none, and is recommended for advanced users only. The following substitutions are made on the string:

%r	root directory of the installation
%p	parallel suffix for the executable

%v	version of ANSYS CFX being run
%o	operating system string
%a	architecture subdirectory specification, for example, linux/double

If it is not supplied, this parameter has the default value %r/bin/%a/solver%p.exe

2.1.1. Adding Hosts for Parallel Processing with the cfx5parhosts Utility

You can add new hosts to ANSYS CFX's database for parallel runs using the cfx5parhosts utility; this is done by running:

```
CFXROOT\bin\cfx5parhosts argument list
```

where `argument list` corresponds to one or more of the arguments listed below. On Windows Vista, cfx5parhosts must be run with administrator privileges. This is because the `hostinfo.ccl` file, like all files in Program Files, requires administrative privileges to edit.

Argument	Description
-add *host*[,*host*,...]	Add information about the named host(s) to the file. This assumes that ANSYS CFX is installed in the same directory on each listed host as on the host on which you are running. *host* may be specified as [*user@*]*hostname*[:cfx-5 root] if the user name or the ANSYS CFX installation root directory differs from the local host. To add a set of separately-installed Windows hosts to the `hostinfo.ccl` file, where the installation may be in a different place on each host, the recommended method is to gather the `hostinfo.ccl` files created on each host by the installation process, and merge them together using the `-merge` switch.
-benchmark	Runs a standard benchmark case, and fills in the Relative Speed for the local host. The benchmark case will usually take less than 5 minutes to run.

Argument	Description
-file *file*	Use the specified file as the `hostinfo.ccl` file.
-merge *file*	Merge host information from the named file.
-no-update	After modifying the file, write back the information available without attempting to fill in any missing pieces.
-strict	Used with `-update`. Normally, hosts which exist on the network but cannot be connected to with rsh or ssh are included in the resulting file with a comment. This switch will exclude these hosts.
-system	Use the system host file. This is the default.
-update	Updates the specified host information file. If any hosts do not have an architecture specified in the existing `hostinfo.ccl` file (for example, because it was added via the `-add` switch), it will connect to the host and query it to fill in the Host Architecture String parameter. This is the default behavior. Note that if the Installation Root parameter is incorrect for the host, it will use the standard system commands to guess a generic architecture string. This can happen if you use `-add` to include a host with a different installation directory than the local one.
-user	Use the per-user host file.

2.2. Setting Up Platform MPI for Windows

Platform MPI requires that you must either configure the parallel host setup properly or set up remote access to slave nodes. See *Setting Up Distributed Parallel or Remote Access* (p. 97) for details.

In addition, Platform MPI has other requirements as follows.

Note

- All nodes must be on the same domain and the user must have an account for that domain.
- The Platform MPI service is required only for distributed parallel runs.
- Platform MPI uses the `-cpu_bind` option by default and binds processes to cores using a cyclic order. This leads to better performance but, on multi-user clusters having scheduling systems that do not enforce exclusive use of cluster nodes, may cause problems because multiple jobs may be bound to the same CPU on the same node. In this case, the `-cpu_bind` argument to `mpirun` should be removed from the Platform MPI start methods in the `start_methods.ccl` file in the `<CFXROOT>/etc` directory.

2.2.1. Setting Up Distributed Parallel or Remote Access

When the ANSYS CFX distributed parallel starts, it needs to know where to find the CFX-Solver executable on every host and which solver executable it should run (for example, 64-bit or 32-bit). This can be set up in two ways:

- (Preferred method) Explicitly specify the required information:

  1. Ensure that every host used in the distributed parallel run is included in the `hostinfo.ccl` file, including the `Installation Root` and `Host Architecture String` parameters for each host. See *hostinfo.ccl file* (p. 92) for details on how to set this up.
  2. Set the environment variable CFX_SOLVE_DISABLE_REMOTE_CHECKS in your user or site `cfx5rc.txt` file. See Resources Set in cfx5rc Files in the *CFX Introduction* for details.

- Have ANSYS CFX determine the required information for itself.

  If ANSYS CFX is to determine the required information for itself, then remote- or secure-shell access must be available from the master node (the system on which parallel runs will be started) to slave nodes (systems on which the ANSYS CFX-Solver will actually run). You must be able to execute an rsh or ssh command on the remote host *without typing in your password*. Unless you already have your Windows systems set up to allow this type of access, this method of setting up distributed parallel is not recommended.

2.2.2. Installing the Platform MPI Service and Registering Users on Windows Vista

On Windows Vista, you must start a command window and install the Platform MPI service as Administrator:

1. From Explorer, right-click C:\WINDOWS\system32\cmd.exe and select **Run As**. In the **Run As** dialog, select **The following user** and login as Administrator.
2. Enter the username and password information.
   - The username should be entered in full (for example DOMAIN\user).
   - The password information is encrypted and added to the registry.
3. If necessary, change to the disk drive that has the ANSYS software installed. For example, to change to the C drive, enter:

   ```
   c:
   ```

4. Run the Platform MPI Service installation with Administrator privileges:

   ```
   "<install_dir>\v140\CFX\bin\cfx5parallel" -install-pcmpi-service
   ```

   A message appears in the terminal when the installation is complete. On the master host, details of a user with access to all of the hosts in the run must now be set. The username and password information is used during the run to access the hosts.
5. Start the service; enter:

   ```
   "<install_dir>\v140\CFX\bin\cfx5parallel" -start-pcmpi-service
   ```

If you need to change the port number used by the service, you can run a command of the following form:

```
"<install_dir>\v140\CFX\bin\cfx5parallel" -setportkey-pcmpi-service <port number>
```

For details on the cfx5parallel utility, enter "<install_dir>\v&ansys_version;\CFX\bin\cfx5parallel" -help at the command prompt.

2.2.3. Installing the Platform MPI Service and Registering Users on Windows XP

As a user with administrative privileges:

1. From the launcher, select **Tools > Command Line**.
2. Enter:

   ```
   cfx5parallel -install-pcmpi-service
   ```

   A message appears in the terminal when the installation is complete. On the master host, details of a user with access to all of the hosts in the run must now be set. The username and password information is used during the run to access the hosts.
3. Enter your username and password information.
   - If prompted, the username should be entered in full (for example DOMAIN\user).
   - The password information is encrypted and added to the registry.
4. From the command line invoked from the launcher, enter:

   ```
   cfx5parallel -start-pcmpi-service
   ```

If you need to change the port number used by the service, you can run a command of the following form:

```
cfx5parallel -setportkey-pcmpi-service <port number>
```

For details on the cfx5parallel utility, enter `cfx5parallel -help` at the command prompt.

2.2.4. Enabling Parallel Through a Windows Firewall

To enable Parallel Processing through a Windows firewall:

1. Click on **Start** and select **Control Panel**.
2. On the **Control Panel** dialog, double-click **Security Center**.
3. Click **Windows Firewall**.
4. On the **Windows Firewall** dialog, click the **Exceptions** tab and then click **Add Program**.
5. Browse to the following programs:
   - For Platform MPI:
     - <CFXROOT>\bin\<CFX5OS>\solver-pcmpi.exe
     - <CFXROOT>\bin\<CFX5OS>\double\solver-pcmpi.exe

- <CFXROOT>\..\commonfiles\MPI\Platform\8.1\<winx64|win32>\sbin\pcmpi-win32service.exe
- <CFXROOT>\..\commonfiles\MPI\Platform\8.1\<winx64|win32>\bin\mpirun.exe
- <CFXROOT>\..\commonfiles\MPI\Platform\8.1\<winx64|win32>\bin\mpid.exe

After each selection, click **OK** to add that program to the list of programs that are permitted to pass through the Windows firewall.

6. When you are finished, click **OK** to close the **Windows Firewall** dialog, then close the **Windows Security Center** and the **Control Panel**.

2.3. Setting up and Running CCS 2003/HPC 2008

To set up running CCS 2003/HPC2008 steps must be taken so that:

- The installed software is on a shared location that can be accessed by all hosts
- The working directory is a shared location
- A mechanism is provided so that all local paths can be properly resolved.

For running ANSYS CFX with CCS 2003/HPC 2008, this can be accomplished by following the steps below (terms in angle brackets < > should be replaced with the required entry):

1. Install ANSYS Licensing, ANSYS Workbench and ANSYS CFX on the head node.
2. Share the installation directory. For example, on a typical installation share `C:\Program Files\ANSYS Inc` as `\\<HeadNodeName>\ANSYS Inc` where `<HeadNodeName>` is the name of the head node.
3. Install the ANSYS prerequisites on all of the cluster nodes. You can do this either by:

   - Executing the following commands directly on each node:

   ```
   \\<HeadNodeName>\ANSYS Inc\v140\prereq\vcredist_x64_SP1.exe
   \\<HeadNodeName>\ANSYS Inc\v140\prereq\vcredist_x86_SP1.exe
   \\<HeadNodeName>\ANSYS Inc\v140\prereq\2008vcredist_x64.exe /qn
   ```

   - or by using the`clusrun` command on the headnode to execute these commands on all the nodes (refer to your Windows CCS or HPC documentation for details).

4. Share the working directory on the head node. For example, share C:\Users\<*UserName*> as \\<*HeadNodeName*>\<*UserName*> where <*UserName*> is the user name.

   Alternatively, share the working directory of the submitting machine. For example share C:\Users\<*UserName*> on the submitting machine as \\<*SubmitHostName*>\<*UserName*> where <*SubmitHostName*> is the name of the submitting machine.

5. If the submitting machine is different from the server head node, it is necessary to install 'HPC Pack 2008 R2 Client Utilities' on the submitting machine. This is available for download from Microsoft. This sets the CCP_HOME environment variable, enabling the 'Submit to Windows CCS or HPC Queue' start method in the CFX-Solver Manager.

6. On the submitting machine, create %USERPROFILE%\.cfx\cfxccs_options.txt with the following content to define the required CCS options:

   PATHMAP=C:\Program Files\Ansys Inc;\\<*HeadNodeName*>\Ansys Inc
   PATHMAP= C:\Users\<*UserName*>;\\<*HeadNodeName*>\<*UserName*> or PATHMAP=C:\Users\<*UserName*>;\\<*SubmitHostName*>\<*UserName*> if the working directory has been shared from the submitting machine.
   CLUSTERHOST=<*HeadNodeName*> to be used when submitting jobs from machines other than the headnode.
   ACCOUNT=<*OtherUserDomain*>\<*OtherUserName*> to be used when submitting jobs using different credentials, where <*OtherUserDomain*> and <*OtherUserName*> are the domain and user names of another user, respectively.
   PROCESSORSPERSOLVER=2 an optional setting (default setting is 1) that allocates the number of cores per partition. This is typically used on hosts that are limited by memory bandwidth such as Xeon-based machines.

7. Set up ANSYS Workbench for a network as described in ???.

To submit a job:

1. Start the CFX-Solver Manager (either using CFX standalone or from ANSYS Workbench) on the submitting machine using the software installed on the headnode.

2. Ensure that the **Run Mode** on the **Run Definition** tab of the CFX-Solver Manager is set to `Submit to Windows CCS or HPC Queue`. Otherwise, set up the CFX-Solver Manager as normal.
3. Click **Start Run** on the CFX-Solver Manager to start the simulation. The first time a simulation is run, supply the required credentials that are prompted (this prompt may appear behind the CFX-Solver Manager window). You may elect to have these credentials saved to avoid repeated prompting of these credentials.

Note

1. For simulations making use of User Fortran, you must ensure that the Fortran is linked against the `solver-msmpi.lib`. This is done by making the appropriate changes to `cfx5mkext.ccl`.
2. `%USERPROFILE%` is typically `C:\Documents and Settings\<Username>\` on XP, or `C:\Users\<UserName>` on Vista or Server HPC 2008.
3. To use on ANSYS Workbench with Parameters and Design Points, start the CFX-Solver Manager ensure that the **Run Mode** on the **Run Definition** tab of the CFX-Solver Manager is set to `Submit to Windows CCS or HPC Queue`, set the number of processes, then click **Save Settings**. When the user clicks **Update all Design Points** each parameter or design point will be solved on the cluster.