

# Installation Guide for Windows

---



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

Release 16.2  
July 2015

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

---

## Revision Information

The information in this guide applies to all ANSYS, Inc. products released on or after this date, until superseded by a newer version of this guide.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

## Copyright and Trademark Information

© 2015 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

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

## Disclaimer Notice

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

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

## U.S. Government Rights

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

## Third-Party Software

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

Published in the U.S.A.

---

---

# Table of Contents

<b>1. Installation Prerequisites for Windows</b> .....	1
1.1. System Prerequisites .....	3
1.1.1. CAD Support .....	4
1.2. Disk Space and Memory Requirements .....	6
1.3. Software Prerequisites .....	6
1.4. Requirements for the GPU Accelerator in Mechanical APDL .....	7
1.5. Additional Hardware and Software Requirements .....	7
1.5.1. ANSYS AIM Requirements .....	9
1.5.2. 3Dconnexion Product Support .....	9
1.6. Third-Party Software and other Security Considerations .....	9
<b>2. Platform Details</b> .....	11
2.1. User Account Control (UAC) .....	11
2.2. Utilizing CPU Hyperthreading Technology with ANSYS CFD Solvers .....	13
2.3. Compiler Requirements for Windows Systems .....	13
2.4. Select Your Installation .....	14
<b>3. Installing the ANSYS Software for a Stand-alone Windows System</b> .....	15
3.1. Downloading the Installation Files .....	16
3.1.1. Installing from a USB Drive .....	16
3.1.2. Installing from a DVD .....	16
3.2. Installing ANSYS, Inc. Products .....	17
3.2.1. Product Installation with Client Licensing .....	17
3.2.1.1. Specifying CAD Configurations .....	20
3.2.2. ANSYS License Manager Installation .....	21
3.2.2.1. Registering the License Server .....	23
<b>4. Installing the ANSYS Products and the License Manager on Different Windows Machines</b> .....	25
4.1. Downloading the Installation Files .....	26
4.1.1. Installing from a USB Drive .....	26
4.1.2. Installing from a DVD .....	27
4.2. Installing ANSYS, Inc. Products .....	27
4.2.1. Product Installation with Client Licensing .....	27
4.2.1.1. Network Server and Product Configuration .....	31
4.2.1.2. Specifying CAD Configurations .....	33
4.2.2. ANSYS License Manager Installation .....	34
4.2.2.1. Registering the License Server .....	35
<b>5. Post-Installation Instructions</b> .....	37
5.1. Post-Installation Procedures for Mechanical APDL and ANSYS Workbench Products .....	38
5.1.1. Post-Installation Procedures for ANSYS CFX and ANSYS CFD-Post .....	39
5.1.2. Post-Installation Procedures for ANSYS Fluent .....	40
5.1.3. Post-Installation Procedures for ANSYS Polyflow .....	40
5.1.4. Post-Installation Procedures for Icem CFD .....	40
5.1.5. Post-Installation Procedures for Other Products .....	40
5.2. Launching ANSYS, Inc. Products .....	41
5.3. Running the ANS_ADMIN Utility for Mechanical APDL /ANSYS Workbench Products .....	41
5.4. Product Localization .....	41
5.4.1. Translated Message File Installation for Mechanical APDL /ANSYS Workbench Products .....	42
<b>6. Installing the ANSYS, Inc. Product Help Documentation Only</b> .....	43
<b>7. Silent Mode Operations</b> .....	45
7.1. Installing ANSYS, Inc. Products .....	45
7.1.1. Silent Mode Operations .....	45
7.1.1.1. Silent Product and License Manager Installation .....	46

7.1.1.2. Silent Product Configuration/Unconfiguration .....	50
7.1.1.3. Silent Media Installation .....	50
7.1.1.4. Silent Uninstall .....	50
<b>8. Configuring CAD Products .....</b>	<b>53</b>
8.1. Using the CAD Configuration Manager .....	53
8.1.1. Unconfiguring .....	56
8.1.2. Running the <b>CAD Configuration Manager</b> in Batch Mode .....	56
8.1.3. Network Considerations .....	58
8.1.4. Uninstalling .....	58
8.1.5. Creo Parametric Configuration .....	59
8.1.5.1. Configuring the Connection for Creo Parametric .....	60
8.1.5.1.1. The WBPlugInPE.dat File and config.pro File .....	60
8.1.5.1.2. The config.pro File .....	61
8.1.5.1.3. The config.anscon.162 File .....	61
8.1.5.2. Creo Parametric Environment Variables .....	62
8.1.6. NX Configuration .....	62
8.1.6.1. Configuring the Connection for NX .....	62
8.1.6.2. Configuring for Teamcenter .....	63
8.2. Configuring CADNexus/CAPRI CAE Gateway for CATIA V5 .....	63
8.3. Configuring AutoCAD .....	64
<b>9. Uninstalling the Software .....</b>	<b>67</b>
9.1. Uninstalling Licensing Components .....	68
<b>10. Troubleshooting .....</b>	<b>69</b>
10.1. Installation Troubleshooting .....	69
10.1.1. Using ANSLIC_ADMIN to Gather Diagnostic Information .....	69
10.1.2. Uninstall Gives Access Denied Error Message .....	69
10.1.3. Uninstall on a Windows 7 or Windows 8.1 System Gives Compatibility Error Message .....	69
10.1.4. A .chm File Does Not Display Properly Across a Network .....	70
10.1.5. Products Crash with an Application Error .....	70
10.1.6. Product Installation Does Not Create Start Menu Item for ANSYS and CAD Plugins Do Not Work .....	70
10.1.7. Product Installation Displays Security Warning Dialogs or Silent Product Installation Hangs on a Network Install .....	70
10.1.8. System-related Error Messages .....	71
10.1.9. CasPol Error Messages .....	71
10.1.10. Configuring NX 10 on Windows 8.1 Displays an Error .....	71
10.2. Installation Troubleshooting - Mechanical APDL and ANSYS Workbench Products .....	71
10.2.1. The Mechanical APDL Launcher is Excessively Slow to Start .....	71
10.2.2. Display Problems on Windows .....	71
10.2.3. ANS_ADMIN Error Messages .....	72
10.2.4. Mechanical APDL Product Launcher Error Messages .....	72
10.2.5. Distributed Mechanical APDL IBM Platform MPI Error Messages .....	72
10.2.6. ANSYS Workbench Products Troubleshooting .....	73
10.3. Installation Troubleshooting - ANSYS CFX .....	74
10.3.1. TurboGrid Mouse Behavior Problems .....	74
10.4. Contacting Technical Support .....	74
<b>11. Applications Included with Each Product .....</b>	<b>77</b>

---

# List of Tables

- 1.1. Supported Windows Platforms ..... 3
- 1.2. Supported Products by Platform ..... 3
- 1.3. CAD Support by Platform ..... 4
- 2.1. Compiler Requirements for All Windows Versions ..... 13

---

---

# Chapter 1: Installation Prerequisites for Windows

---

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

- ANSYS Structural Mechanics
  - ANSYS Mechanical Products (includes ANSYS Mechanical and Mechanical APDL)
  - ANSYS Customization Files for User Programmable Features
- ANSYS Explicit Dynamics
  - ANSYS Autodyn
  - ANSYS LS-DYNA
- ANSYS Fluid Dynamics
  - ANSYS CFX (includes ANSYS CFD-Post)
  - ANSYS Fluent (includes ANSYS CFD-Post)
  - ANSYS TurboGrid
  - ANSYS Polyflow (includes ANSYS CFD-Post)
  - ANSYS CFD-Post only
- ANSYS ICEM CFD
- ANSYS Offshore
  - ANSYS Aqwa
- ANSYS Additional Tools
  - ANSYS Icepak (includes ANSYS CFD-Post)
  - Remote Solve Manager Standalone Services
- ANSYS PDM Interfaces
  - Teamcenter
- ANSYS Geometry Interfaces
  - AutoCAD
  - ACIS

- CATIA, Version 4
- CATIA, Version 5
- CATIA, Version 6
- Creo Elements/Direct Modeling
- Autodesk Inventor
- JTOpen
- NX
- Parasolid
- Creo Parametric
- Solid Edge
- SolidWorks

---

### Note

ANSYS Workbench and ANSYS EKM Desktop are installed by default as product components to most ANSYS, Inc. products. ANSYS Workbench is not installed as part of the products under ANSYS Additional Tools, nor with the CFD-Post only option. ANSYS Workbench includes the following applications:

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

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

Where supported, IGES and STEP Geometry Interfaces will be installed.

---

Some procedures apply only to specific products and are so noted.



## Important Notice

If you want to run multiple releases of ANSYS, Inc. software, you **MUST** install them chronologically (i.e., Release 16.1 followed by Release 16.2). If you install an earlier release after installing Release 16.2, you will encounter licensing issues that may prevent you from running any products/releases. If you need to install an earlier release after you have already installed Release 16.2, you **MUST** uninstall Release 16.2, then re-install the releases in order.

## 1.1. System Prerequisites

ANSYS 16.2 is supported on the following Windows platforms and operating system levels. For up-to-date information on hardware platforms or operating systems that have been certified, go to <http://www.ansys.com/Support/Platform+Support>. (This URL is case-sensitive.)

**Table 1.1: Supported Windows Platforms**

Processor	Operating System	Platform architecture (directory name)	Availability
x64 (64-bit)	Windows 7, Windows 8.1, Microsoft Windows HPC Server 2008 R2 [1], Windows Server 2012 R2 [1]	winx64	Download, USB
<p>1. Microsoft Windows HPC Server 2008 R2 and Windows Server 2012 R2 are supported only for select solvers for parallel high performance execution. Any users running on Microsoft Windows HPC Server 2008 R2 or Windows Server 2012 R2 as non-admin should be a member of the HPC Users Group in the configuration panel of the <b>HPC Cluster Manager</b>.</p> <p>For detailed and current support information, see the Platform Support section of the ANSYS, Inc. website (<a href="http://www.ansys.com">www.ansys.com</a>).</p>			
<p><b>Note</b></p> <p>Home editions of Windows operating systems are not supported.</p>			

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

**Table 1.2: Supported Products by Platform**

Product	Windows 64-bit	Microsoft Windows HPC Server 2008	Windows Server 2012
ANSYS Workbench data-integrated and native applications	√		
<b>Standalone Solvers and Applications</b>			
Mechanical APDL	√	2	2
CFX	√	2	2

Product	Windows 64-bit	Microsoft Windows HPC Server 2008	Windows Server 2012
TurboGrid	✓		
ICEM CFD	✓		
Autodyn	✓		
LS-DYNA	✓		
Fluent	✓	2	2
Aqwa	✓		
Polyflow	✓		
Icepak	✓	2	2
EKM Server	✓		

1. Supported as a 32-bit application
2. Solver only support

### 1.1.1. CAD Support

The following CAD and auxiliary programs are supported on the indicated products and platforms (where the CAD product is supported on the noted platforms). Products are:

A = Mechanical APDL

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**

Product	Windows 7 64-bit	Windows 8.1 64-bit 6
CATIA 4.2.4	A, W, I	A, W, I
CATIA V5-6R2014	A, W, I	A, W, I
CATIA V5 (CADNexus CAPRI CAE Gateway V3.20.0) V5-6R2012, V5-6R2013, V5-6R2014	W, I	W, I 11
CATIA V6 R2013x	W, I	W, I
Parasolid 26.0 10	A, W, I	A, W, I
Creo Parametric Wildfire 5.0 2, 3	A, W, I	
Creo Parametric 2.0 12	A, W, I	W, I
Creo Parametric 3.0	A, W, I	A, W, I
ACIS 25 1	A, W, I	A, W, I
NX 8.5	A, W, I	A, W, I
NX 9.0	A, W, I	A, W, I
NX 10.0	A, W, I	A, W, I
SolidWorks 2014	W, I	W, I 13

Product	Windows 7 64-bit	Windows 8.1 64-bit 6
SolidWorks 2015	W, I	W, I
Solid Edge ST6	W, I	
Solid Edge ST7	W, I	W, I
Autodesk Inventor 2014	W, I	W, I 7
Autodesk Inventor 2015	W, I	W, I
AutoCAD 2013	W, I	
AutoCAD 2014	W, I	W, I 14
IGES	A, W, I	A, W, I
JT 9.5	W, I	W, I
STEP AP203, AP214	W, I	W, I
Creo Elements/Direct Modeling 18.1	W, I	W, I 8
Creo Elements/Direct Modeling 19.0	W, I	W, I
Teamcenter Unified Architecture 9.1 (2- & 4-tier) with NX 8.5.2 MP1 and NX 9.0		W 15
Teamcenter Unified Architecture 9.1.3 (2- & 4-tier) with NX 8.0 (8.0.2 or higher), NX 8.5.2 MP1 & NX 9.0	W	
Teamcenter Unified Architecture 10.1.0.1 (2- & 4-tier) with NX 8.5.2 MP1 & NX 9.0	W	
GEMS	I	I
IDI MS8/9	I	I
Rhinoceros	I	I

1. For ICEM CFD standalone, ACIS 18.0.1 is the supported version for all platforms.
2. Requires Pro/ENGINEER 5 Build Date M020 or higher for Windows 7.
3. Pro/ENGINEER Wildfire 5 is also named Creo Elements/Pro 5.0.
4. MAPDL supports 5.1 by default, but 5.2 is also supported if the IOPTN command is used.
5. IGES Versions 4.0, 5.2, and 5.3 are supported.
6. Professional and Enterprise
7. Requires Autodesk Inventor 2014 SP1.
8. Requires CoCreate 18.1 M060 or later.
9. Requires AutoCAD 2013 SP2.
10. For ICEM CFD direct interface, Parasolid 24.0 is the supported version on all platforms.
11. Requires CATIA V5-6R2014.
12. Requires Creo Parametric 2.0 M100.

- 13. Requires SolidWorks 2014 SP1.
- 14. Requires Installation of Autodesk’s Model Documentation Hotfix.
- 15. Requires patch 9.1.3 with NX 8.5.2 MP1 and NX 9.0.

---

**Note**

View the latest supported CAD packages at [ansys.com](http://ansys.com)> Support> Platform Support> Certified and Supported Computing Platforms.

---

## 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 for each ANSYS, Inc. product. Depending on the options selected for each product, you may require less.

Product	Disk Space
ANSYS Mechanical APDL	8.1 GB
ANSYS Autodyn	6.3 GB
ANSYS LS-DYNA	6.5 GB
ANSYS CFX	6.9 GB
ANSYS TurboGrid	6.2 GB
ANSYS Fluent	7.2 GB
ANSYS Polyflow	7.2 GB
ANSYS Aqwa	6.0 GB
ANSYS ICEM CFD	7.0 GB
ANSYS Icepak	7.3 GB
ANSYS CFD Post only	6.2 GB
ANSYS Geometry Interfaces	1.5 GB
CATIA v5	1.2 GB
CATIA v6	95 MB
ANSYS Remote Solve Manager Standalone Services	2.2 GB
ANSYS Composite PrepPost	2.9 GB

Product installations also require an additional 500 MB of free disk space during an installation.

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

## 1.3. Software Prerequisites

You must have the following software installed on your system. Administrator privileges are required to install these files. These software prerequisites will be installed automatically when you launch the product installation. If you've completed an installation successfully, the executables are also located under the `\v162\prereq` directory should you need to manually install them at a later date. Prior to installing, you can find them in the root `\prereq` directory of the media/download package.

- Microsoft .NET Framework 4.0

.NET Framework 4.0 is native on Windows 8.1 and does not require any additional patches.

.NET Framework 4.0 is required for Windows 7.

- Microsoft Visual C++ 2005 SP1 Redistributable, 2.0.50727.762
- Microsoft Visual C++ 2008 SP1 ATL Redistributable Package
- Microsoft Visual C++ 2008 Service Pack 1 Redistributable Package MFC Security Update
- Microsoft Visual C++ 2010 SP1 ATL Redistributable Package

## 1.4. Requirements for the GPU Accelerator in Mechanical APDL

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

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

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

- For nVIDIA GPU cards, the driver version must be 333.11 or newer. For optimal performance on Windows, the TCC (Tesla Compute Cluster) driver mode is recommended when using Tesla series GPU cards. Some limitations exist when using this driver mode (e.g., no support for Windows XP). Check your GPU card documentation for more details on how to set this driver mode and the existing limitations.
- For Intel Xeon Phi coprocessors, the driver version must be "MPSS 3.4" or newer.

---

### Note

On Windows, the use of Remote Desktop may disable the use of a GPU device. Launching Mechanical APDL through the ANSYS Remote Solve Manager (RSM) when RSM is installed as a service may also disable the use of a GPU. In these two scenarios, the GPU accelerator capability cannot be used. Using the TCC (Tesla Compute Cluster) driver mode, if applicable, can circumvent this restriction.

---

## 1.5. Additional Hardware and Software Requirements

- Intel 64 / AMD64 system with the correct operating system version installed
- 8 GB of RAM
- 128 GB free on the hard drive is recommended

- Certified graphics card with the latest drivers, compatible with the supported operating systems, capable of supporting 1024x768 High Color, and a 17-inch monitor compatible with this type of graphics card. We recommend using certified graphics card from the Nvidia Quadro FX or AMD FirePro line.

---

### Note

Use of Ultra High Definition (4K) graphics cards cause a number of cosmetic display issues (including enlarged or reduced text and incorrect positioning of option labels). These issues do not affect the functionality of the installation program.

---

ANSYS CFX products and ANSYS TurboGrid require at least 24-bit color and that antialiasing on your graphics card be disabled. How you determine whether antialiasing is enabled and how to disable the feature varies according to the graphics card you have. Please contact your system administrator or refer to your system documentation for detailed instructions on changing antialiasing settings.

---

### Note

Some combinations of graphics card type, operating system, and MPEG resolution fail to play MPEGs properly. You may be able to get normal playback results simply by changing the MPEG settings. Alternatively, you can upgrade your graphics card.

---

- For laptop systems, ANSYS Workbench requires the latest graphics drivers.
- Microsoft Mouse or a mouse compatible with the supported operating systems; ANSYS CFX-Pre and ANSYS CFD-Post require a three-button mouse.
- If you use a 3Dconnexion product, ensure that you have the latest version of the drivers installed.
- Approximately twice as much swap space as memory. The amount of memory swap space on the system may limit the size of the ANSYS model that can be created and/or solved.
- TCP/IP for the license manager (see the [ANSYS, Inc. Licensing Guide](#) for more information on TCP/IP).

Although TCP/IP is included as part of the operating system, it may not be installed by default. When TCP/IP is installed, it must be bound to a network adapter. On machines that are connected to an internal network, TCP/IP must be bound to a network card such as an Ethernet adapter. The vast majority of systems using TCP/IP will fall into this category. On machines that connect to the internet or corporate intranet through a modem, TCP/IP can be bound to the Remote Access Service. Also, the Autodial option of the Internet Options must be disabled so that the machine does not attempt to connect to the Internet every time ANSYS is run. See the [ANSYS, Inc. Licensing Guide](#) for more information.

- Microsoft Internet Explorer (IE) 9 or greater is recommended for proper operation of ANSYS Workbench. Once the correct version of IE is installed, it does not have to be your default Internet browser. After installation, simply run your preferred browser and reestablish it as the default.
- PDF reader software is required to read the installation guides and other user documentation.

To access our help system videos, ANSYS recommends that you set Windows Media Player as your default. Note--The default media player for Windows 8.1 is defined as **Video**; please change this setting to **Windows Media Player**.

### 1.5.1. ANSYS AIM Requirements

For basic usage and simple models, a workstation-class 3D-capable graphics card with at least 256 MB of memory that supports OpenGL version 3.3 or higher, as noted below, is needed to successfully run AIM.

To analyze larger models, use a recent NVidia Quadro or AMD FirePro card, including:

NVidia	NVidia Mobile	ATI	ATI Mobile
Quadro FX x700	Quadro x600M	FirePro Vx800	FirePro Mx9xx
Quadro FX x800	Quadro x700M	FirePro Vx900	FirePro Mx000
Quadro x000	Quadro x800M	FirePro Wx	
Quadro Kxxx	Quadro x000M	FirePro Sx	
Quadro Kxxxx	Quadro kx000M		
	Quadro kx100M		

**Note on Intel Graphics Cards:** Use of integrated Intel graphics cards is not recommended. These cards are typically found in lightweight laptops or notebooks, and are known to have OpenGL incompatibilities resulting in display issues in AIM. Examples are Intel HD 3000 and 4000 series cards.

**Note on Hybrid Graphics:** If your graphics system supports two graphics chips, one integrated and one dedicated (example is NVidia Optimus), please ensure that the dedicated graphics chip is used when launching AIM. Instructions on how to do this can typically be found on graphics card vendor's web site.

### 1.5.2. 3Dconnexion Product Support

Only the Mechanical APDL, ANSYS Meshing, DesignModeler, Mechanical, ANSYS ICEM CFD, and CFD-Post applications have been tested for manipulating the model (turning, zooming, panning, etc.) with the 3Dconnexion products. While these 3Dconnexion products do provide significantly enhanced 3D control over viewing the model, they do not function as a mouse replacement and selection operations still require a mouse.

To use a 3Dconnexion product with the listed products, you must download and install the appropriate Microsoft Windows® driver for your version of Windows. Please refer to <http://www.3dconnexion.com> for more information on using Space products and to download the appropriate drivers.

## 1.6. 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.

<b>Product Name</b>	<b>Executable Name</b>
Tcl	tclsh.exe
Tk	want.exe
Perl	perl.exe
7zip	7z.exe
Microsoft .NET Framework	.net



---

## Chapter 2: Platform Details

---

### 2.1. User Account Control (UAC)

#### Windows 7 and Windows 8.1

Windows 7 and Windows 8.1 use a feature called User Account Control (UAC) to control privileges and automatically reduce the potential of security breaches in the operation system. However, UAC limits your accessibility to system areas and can cause unpredictable behavior in ANSYS, Inc. products.

ANSYS, Inc. requires that you have full administrative privileges when installing any software on Microsoft Windows 7 or Windows 8.1. Non-administrative accounts do not usually have the permissions required to access system areas that installation programs often need to modify. "Full administrative privileges" means that you are running as administrator with UAC turned off, or you are running as administrator with UAC turned on and **Run as Administrator** selected. Because of changes to the UAC functionality, it is necessary that you right-click and select **Run as Administrator** from the context menu when installing to Windows 8.1.

#### Recommended Installation Options for Windows 7:

- UAC turned off and install as full administrator (preferred method)
- UAC turned on and install with full administrative privileges (using **Run as Administrator** from the context menu)

---

#### Note

If you are running with UAC on, even if you have administrative privileges, you will have limited account control unless you specifically choose **Run as Administrator**.

---

#### Recommended Installation Options for Windows 8.1:

- UAC turned off and install with full administrative privileges (using **Run as Administrator** from the context menu)

When installing ANSYS, Inc. products, we strongly recommend that you use the recommended options above. If UAC is enabled, we recommend that you turn UAC off before installing ANSYS, Inc. products, and then turn it back on when finished installing. You must be an administrative user in order to disable UAC.

Follow the instructions for your operating system for disabling UAC.

Once you have installed your applications, you can enable UAC again.

## Installing with UAC On

If you cannot disable UAC and need to install with UAC on, you must run all applications, including the installation setup, as an administrator. Otherwise, the files may be written to `AppData\Local\VirtualStore\<path>` (or similar) rather than directly to `<path>`, if `<path>` is a protected directory (such as `C:\Program Files`). As a result, some applications and utilities may behave unpredictably.

If you cannot disable UAC, you should perform the following functions:

### For Windows 7:

- Always run **ANSLIC\_ADMIN**, the **File Association** utility, and the **CAD Configuration Manager** by selecting the utility or application from the **Start** menu and right-mouse clicking and selecting **Run as Administrator**.
- Set your Start menu shortcuts to always run as administrator with the following procedure (or similar):
  1. Right-click the shortcut you wish to modify and select **Properties**.
  2. From the **Properties** dialog box, click **Advanced** on the **Shortcut** tab.
  3. Select **Run as administrator** and click **OK**.
  4. Click **OK**.

Refer to your operating system documentation or visit [www.microsoft.com](http://www.microsoft.com) for more instructions on setting **Run as Administrator** at various levels within the operating system.

### For Windows 8.1:

- Always run **ANSLIC\_ADMIN**, the **File Association** utility, and the **CAD Configuration Manager** by selecting the utility or application from the **Start** menu and right-mouse clicking and selecting **Run as Administrator**.
- Set your Start menu shortcuts to always run as administrator with the following procedure (or similar):
  1. Right-click the shortcut you wish to modify and select **Open File Location** from the options located at the bottom of your screen.
  2. Right-click the shortcut file that is displayed and click **Properties** from the menu.
  3. From the **Properties** dialog box, click **Advanced** on the **Shortcut** tab.
  4. Select **Run as administrator** and click **OK**.
  5. Click **OK**.

## Notes about UAC

When installing ANSYS, Inc. products, including the license manager, on Windows 7 and Windows 8.1 machines, you need to be aware of several factors involving User Access Control (UAC):

You should install the ANSYS, Inc. License Manager using the same permissions as you used to install the product.

If you install both the product and the license manager with full administrative privileges, the product and licensing installation and configuration will proceed as normal.

If you install as an administrative user with UAC turned on, you may encounter unpredictable behavior with both the product and license manager installations and subsequent behavior. As an administrator, you should choose to install and run applications with full administrative privileges to avoid any unpredictable situations. We strongly recommend against running with UAC on and not selecting **Run as Administrator**. These situations are described below.

1. If you have UAC turned on and you do not install (and run) with full administrative privileges, be aware that files may be written to and read from the %localappdata%\VirtualStore\ location (in the case of licensing, to %localappdata%\VirtualStore\Program Files\ANSYS Inc\Shared Files\Licensing) instead of to the <os drive>.

In this situation, **ANSLIC\_ADMIN** settings, such as **Specify the License Server Machine**, will indicate that files are being written to and read from <os drive>\Program Files\ANSYS Inc\Shared Files\Licensing. Log files may behave in the same way. However, when you view the an-syslmd.ini file located in <os drive>\Program Files\ANSYS Inc\Shared Files\Licensing, you will not see the **ANSLIC\_ADMIN** settings you specified.

2. To ensure that files are correctly written to and read from <os drive>\Program Files\ANSYS Inc\Shared Files\Licensing, you should always run with full administrative privileges by selecting **Run as Administrator**.
3. If you are not running with full administrative privileges, you will not be able to use those **ANSLIC\_ADMIN** options that require administrative privileges, such as starting and stopping the license manager.

## 2.2. Utilizing CPU Hyperthreading Technology with ANSYS CFD Solvers

Hyperthreading technology uses one processor core to run more than one task at a time. ANSYS does not recommend using hyperthreading technology in conjunction with ANSYS CFD Solvers (Fluent, CFX and AIM Fluids). We recommend that you turn CPU hyperthreading off (default is on). A system administrator needs to reboot the system and enter the BIOS to turn the hyperthreading option off.

## 2.3. Compiler Requirements for Windows Systems

**Table 2.1: Compiler Requirements for All Windows Versions**

<b>Mechanical APDL, ANSYS Workbench Compilers*</b>	<b>CFX Compilers*</b>	<b>Fluent Compilers*</b>	<b>AUTODYN Compilers*</b>
Visual Studio 2010 SP1 (including the MS C++ compiler) and Intel FORTRAN 14.0.3 compilers	Visual Studio 2010 SP1 (including the MS C++ compiler) and Intel FORTRAN 14.0.3 compilers	Visual Studio 2010 SP1 (including the MS C++ compiler)	Visual Studio 2010 SP1 (including the MS C++ compiler) and Intel FORTRAN 14.0.3 compilers

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

## 2.4. Select Your Installation

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

- [Installing the ANSYS Software for a Stand-alone Windows System \(p. 15\)](#)
- [Installing the ANSYS Products and the License Manager on Different Windows Machines \(p. 25\)](#)

---

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

---

This section includes the steps required for installing ANSYS, Inc. products and licensing configuration on one Windows machine.

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

### Before You Begin

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

- An account on the ANSYS Customer Portal. If you do not have an account, you may register at <https://support.ansys.com/portal/site/AnsysCustomerPortal> to receive your own account.
- Your license file from ANSYS, Inc., saved to a temporary directory. For more information, see [Registering the License Server \(p. 23\)](#).
- Open port numbers for FlexNet and ANSYS Licensing Interconnect. Defaults are 1055 and 2325, respectively. To verify that these port numbers are available, open a command line and enter the following command:

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

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

- Your local machine's hostname, to specify as the license server.

You should also verify that you are running on a supported platform. ANSYS, Inc. supports 64-bit systems running Windows 7 and Windows 8.1.

Verify that you have sufficient disk space to download, uncompress, and install the products you will be installing.

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

These instructions are divided into four sections:

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

- **Post-Installation Procedures for All Products:** This is a set of instructions that describes any configuration steps that may be required for the various products.

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

## 3.1. Downloading the Installation Files

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

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

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

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

2. Select your installation operating system (**Windows x64** or **Linux x64**).
3. Select the type of files you wish to download:
  - **Primary Packages:** Individual full packages for the primary ANSYS products.
  - **ISO Images:** ISO images for the DVD installation.
4. Click the appropriate download option.
5. Select your desired download directory and click **Save**.
6. Repeat this process for each download file.
7. To download **Add-On Packages, Tools** or **Academic Packages**, click the + to the right of the appropriate product group title to display the download options and download as required.
8. After all downloads have been completed, uncompress each package using standard uncompression utilities for your specific platform. We strongly recommend that you extract the files into new, temporary directories.
9. Begin the product installation as described in [Installing ANSYS, Inc. Products \(p. 17\)](#).

### 3.1.1. Installing from a USB Drive

To install ANSYS, Inc. products from a USB drive, insert the USB drive into a USB slot on your computer. Locate and right-click the **setup.exe** file stored at the root level of the USB and select **Run as administrator**. Continue with the steps described in [Installing ANSYS, Inc. Products \(p. 17\)](#).

### 3.1.2. Installing from a DVD

To install ANSYS, Inc. products from a DVD, place the DVD in your DVD drive. If autorun is enabled, the installation will begin automatically. If autorun is disabled, locate and right-click the `setup.exe` file

contained on the DVD and select **Run as administrator**. Continue with the steps described in [Installing ANSYS, Inc. Products \(p. 17\)](#).

You will be prompted to change DVDs during the installation. Please make sure you have all installation DVDs before beginning the installation.

## 3.2. Installing ANSYS, Inc. Products

This section explains how to install ANSYS, Inc. products.

---

### Note

ANSYS requires users to install all products by using the right-click option **Run as administrator**.

---

### 3.2.1. Product Installation with Client Licensing

To install ANSYS, Inc. products with client licensing, follow the steps below.

---

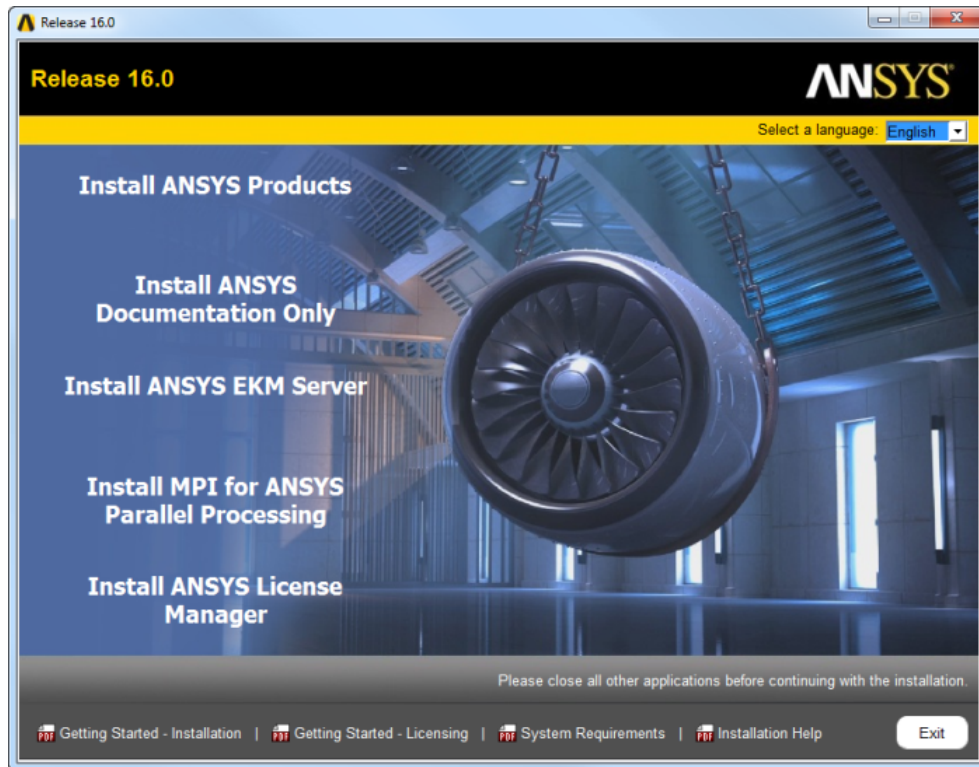
### Note

You must also install the ANSYS, Inc. License Manager on at least one server machine in order to run ANSYS, Inc. products. See [ANSYS License Manager Installation \(p. 21\)](#) for license manager installation instructions.

---

1. Save all data and close all Windows applications before continuing.
2. If you downloaded the installation files or if you are installing from a DVD or USB with autorun disabled, browse to the location of the `setup.exe` file. Right-click the `setup.exe` file and select **Run as administrator**.

The ANSYS, Inc. Installation Launcher appears.



From the options on the left side of the launcher you can install ANSYS products, ANSYS EKM Server, MPI for ANSYS Parallel Processing, and the ANSYS License Manager. You can access the installation guide for the ANSYS EKM Server from the **Downloads** page on the Customer Portal. The MPI for ANSYS Parallel Processing installation steps are displayed when you click the **Install MPI for ANSYS Parallel Processing** link.

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

3. Select the language you want to use from the drop-down menu in the upper right corner. English is the default.
4. Click the **Install ANSYS Products** option.  
The license agreement appears.
5. Read the agreement, and if you agree to the terms and conditions, select **I Agree**. Click **Next**.
6. The directory where you want to install the ANSYS, Inc. products is shown in the **Install Directory** field. You can install the products into any directory you want, but you must have write permissions to the directory you choose. The default is `C:\Program Files\ANSYS Inc.` We recommend using the default directory.

---

### Note

You must install all ANSYS, Inc. products into the same location. Installing products into different locations can cause product components to fail. If you choose an install directory via the Browse feature, the installation will automatically append `\ANSYS`



---

Inc\ to the chosen directory. The installation path can have a maximum of 100 characters, except on AUTODYN, which has a maximum of 40 characters.

---

- Leave the **Associate file extensions with ANSYS products** checkbox marked. This allows you to double-click files from Windows Explorer to launch the various products.
- Choose **Disable ANSYS RSS Feed** to disable automatic internet feeds to ANSYS, Inc. products.

Click **Next**.

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

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

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

Click **Next**.

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

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

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

If the installation program was unable to query your license server, this message is not displayed and the default installation options are selected.

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

If you select AutoCAD, Catia V5, Inventor, NX, Creo Parametric, Solid Edge, or SolidWorks, you will have additional installation steps. Please see [Specifying CAD Configurations \(p. 20\)](#).

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

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

---

## Note

On a first time installation, if you chose to install any ANSYS Geometry Interfaces, ensure that you have also selected at least one ANSYS, Inc. product as part of the in-

stallation. Installing an ANSYS Geometry Interface without an underlying ANSYS, Inc. product on a first time installation may cause installation errors.

---

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

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

9. A summary of the selected installation data appears. Review the information carefully, and if correct, click **Next** to continue the installation.
10. The installation progress screen displays a status bar towards the bottom of the installation window. This status bar tracks the percentage of packages that have been installed on your computer. Depending on the number of products you have selected, the installation time required could be lengthy. You will not be able to interrupt the installation process. Please be patient while the installation completes.

---

### Note

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

---

The installation program will first check your system to determine if you have the necessary prerequisites. If the prerequisites are not already on your system, the prerequisites will be installed automatically. You may be asked to run the necessary files (such as the .NET exe), or to reboot your machine, depending on your existing machine configuration. If you are asked, you will need to reboot in order to complete the prerequisite installation successfully.

Click **Next** to continue the installation.

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

---

### Note

If prerequisite components necessary for the installation of the ANSYS products were installed, the following message may be displayed:

*A system restart is required before using any ANSYS Products.*

---

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

## 3.2.1.1. Specifying CAD Configurations

### Creo Parametric

1. If you selected Creo Parametric, you will have to choose which Creo Parametric 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, you may need to specify the Creo Parametric language, the Creo Parametric command, and the full Creo Parametric installation path (C:\Program Files\PTC\Creo 1.0\Parametric by default) for an existing Creo Parametric installation. Click **Next**.

## NX

- 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. If you are an administrative user, a file required to load the NX plug-in is placed in the administrative user's Application Data folder by default, which may not be accessible by other, non-administrative users. To allow non-administrative users to run, you will need to define the environment variable **UGII\_CUSTOM\_DIRECTORY\_FILE** prior to installation and specify a location where other users have read access. Alternatively, you can run the **CAD Configuration Manager** after the installation and provide an updated location for the NX Custom Directory File Path under the **NX** tab.
- If you selected the ANSYS ICEM CFD NX Interface product but not the NX Geometry Interface product on the product selection screen, you may need to specify the NX installation path or choose to skip configuring NX and use the **CAD Configuration Manager** to configure later.

## CATIA v5

- If you selected CATIA v5 and any product that includes ANSYS Workbench, you will be required to select the appropriate CAD reader/plugin for ANSYS Workbench. You can choose to configure the Reader, CADNexus CAPRI CAE Gateway, or skip and configure later (using the **CAD Configuration Manager**).

## SolidWorks and Autodesk Inventor

- If you selected Inventor and/or SolidWorks as one of your CAD products, you will have to choose which Inventor and/or SolidWorks 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 have not yet installed these programs, or do not know the requested information, you can choose to skip these configuration steps. If you skip these steps, you will need to manually configure these CAD interfaces using the **CAD Configuration Manager** before you can successfully import models into Mechanical APDL, ANSYS Workbench, or ICEM CFD. You can access the **CAD Configuration Manager** in Windows 7 (**Start > All Programs > ANSYS 16.2 > Utilities > CAD Configuration Manager 16.2**) and in Windows 8.1 (**Right-click the Start Screen > All apps > ANSYS 16.2 > CAD Configuration Manager 16.2**). See [Configuring CAD Products \(p. 53\)](#) for more information on using the **CAD Configuration Manager**.

### 3.2.2. ANSYS License Manager Installation

Follow the instructions below to install and configure the ANSYS License Manager on your local Windows machine. You must be an administrative user to install the ANSYS License Manager on Windows. The License Manager controls access to the ANSYS, Inc. products you have purchased. You must complete the License Manager installation to run ANSYS, Inc. products.

- If the Installation Launcher is not open, right-click the setup.exe file and select **Run as administrator**.

2. Select **Install ANSYS License Manager**.

If you downloaded only the license manager installation, right-click `setupLM.exe` and select **Run as administrator**.

3. The installation launcher appears.

4. You will be notified that the license manager, if running, will be shut down. Click **OK**.

5. The License Agreement screen appears. Read the license agreement, and if you agree, click **I Agree** to accept the terms and click **Next**. You must select **I Agree** to continue with the installation.

6. Specify the installation directory and click **Next** to continue.. You can accept the default or specify an alternate path and the directory name where the license manager is to be installed. The installation directory is set to `<OS_Drive>\Program Files\ANSYS Inc` by default. You must have administrative privileges for the directory you specify. In addition, the directory:

- must be a local directory
- must be a local, fixed-media drive
- cannot be a UNC or relative path
- cannot be a short (8.3) file name format path
- cannot be a symbolic link or junction
- cannot use wide character names/paths

---

**Note**

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

---

The ANSYS, Inc. License Manager is the only component available and is selected to be installed. The amount of disk space required and the disk space available appear at the bottom of the window. Be sure that you have sufficient space before continuing.

7. Click **Next** to continue.

8. A summary of the selected installation data appears. 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. After the License Manager installation has been completed, the **Launch License Management Center upon exiting** option is included on the screen. The **ANSYS License Management Center** is a browser-based user interface that centralizes many of the ANSYS product licensing administrative functions. If you do not want to launch the License Management Center, uncheck the **Launch License Management Center upon exiting** option.

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

10. When the license manager installation is complete, click **Exit**. A new Start Menu item named **ANSYS, Inc. License Manager** will be created automatically. It will include selections for the **License Management Center**, the *ANSYS, Inc. Licensing Guide*, the *FlexNet License Administration Guide* and the **ANSLIC\_ADMIN** utility.

### 3.2.2.1. Registering the License Server

If you are a new user who has not received a license file for your server or if you add or change a license server machine, follow this procedure to register your license server information. See the [ANSYS, Inc. Licensing Guide](#) for more information on selecting license servers ([Selecting License Server Machines](#)) and on using the **ANSLIC\_ADMIN** server utility.

1. Use the **Register License Server Machine Information** option of the **ANSLIC\_ADMIN** server utility to register license server information on each license server.
2. Select **OK** and **Create File** to save this information to a file named `licserver.info` by default.
3. Return the resulting `licserver.info` file to your ANSYS sales representative. ANSYS, Inc. or your sales representative will then supply you with your license keys. This file is located in `Program Files\ANSYS Inc\Shared Files\Licensing\licserver.info` by default.
4. Use the **Run the License Wizard** option of the **ANSLIC\_ADMIN** server utility to install your license file and start the license manager.

After completing the installation process, please refer to [Post-Installation Instructions \(p. 37\)](#)



---

## Chapter 4: Installing the ANSYS Products and the License Manager on Different Windows Machines

---

This section describes installing ANSYS, Inc. products on one Windows machine and installing the ANSYS, Inc. License Manager on a different Windows machine that will act as a license server.

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

### Before You Begin

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

- An account on the ANSYS Customer Portal. If you do not have an account, you may register at <https://support.ansys.com/portal/site/AnsysCustomerPortal> to receive your own account.
- Your license file from ANSYS, Inc., saved to a temporary directory. For more information, see [Registering the License Server \(p. 35\)](#).
- Open port numbers for FlexNet and ANSYS Licensing Interconnect on your license server machine. Defaults are 1055 and 2325, respectively. To verify that these port numbers are available, open a command line on the server machine and enter the following command:

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

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

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

You should also verify that you are running on a supported platform. ANSYS, Inc. supports 64-bit systems running Windows 7, and Windows 8.1.

Verify that you have sufficient disk space to download, uncompress, and install the products you will be installing.

For network installations, the same UNC paths or mapped drive letters must be used on the server and all client computers. Failure to use matching UNC paths or mapped letter drives can cause server communication issues or loss of product functionality.

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

This installation is divided into four sets of instructions:

- Product Download/DVD Instructions: This set of instructions describes the download and extraction process.

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

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

## 4.1. Downloading the Installation Files

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

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

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

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

2. Select your installation operating system (**Windows x64** or **Linux x64**).
3. Select the type of files you wish to download:
  - **Primary Packages:** Individual full packages for the primary ANSYS products.
  - **ISO Images:** ISO images for the DVD installation.
4. Click the appropriate download option.
5. Select your desired download directory and click **Save**.
6. Repeat this process for each download file.
7. To download **Add-On Packages, Tools** or **Academic Packages**, click the + to the right of the appropriate product group title to display the download options and download as required.
8. After all downloads have been completed, uncompress each package using standard uncompression utilities for your specific platform. We strongly recommend that you extract the files into new, temporary directories.
9. Begin the product installation as described in [Installing ANSYS, Inc. Products \(p. 27\)](#)

### 4.1.1. Installing from a USB Drive

To install ANSYS, Inc. products from a USB drive, insert the USB drive into a USB slot on your computer. Locate and right-click the **setup.exe** file stored at the root level of the USB and select **Run as administrator**. Continue with the steps described in [Installing ANSYS, Inc. Products \(p. 27\)](#).



## 4.1.2. Installing from a DVD

To install ANSYS, Inc. products from a DVD, place the DVD in your DVD drive. If autorun is enabled, the installation will begin automatically. If autorun is disabled, locate and right-click the `setup.exe` file contained on the DVD and select **Run as administrator**. Continue with the steps in [Installing ANSYS, Inc. Products](#) (p. 27).

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.

## 4.2. Installing ANSYS, Inc. Products

This section explains how to install ANSYS, Inc. products.

---

### Note

ANSYS requires users to install all products by using the right-click option **Run as administrator**.

---

### 4.2.1. Product Installation with Client Licensing

To install ANSYS, Inc. products, including client licensing, follow the steps below.

---

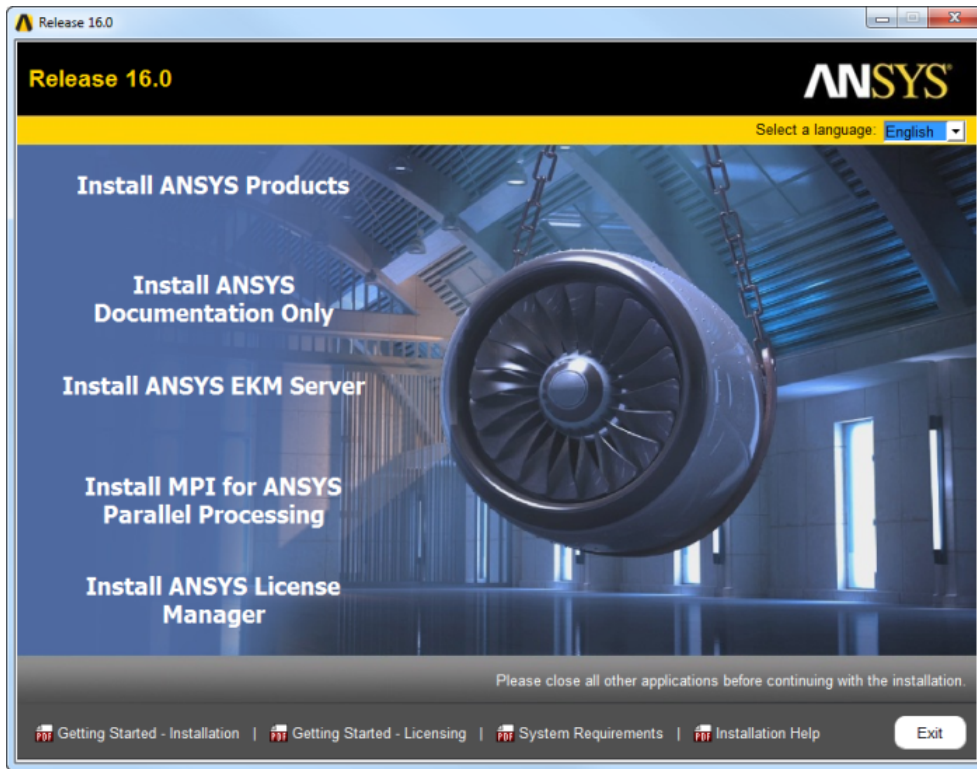
### Note

You must also install the ANSYS, Inc. License Manager on at least one server machine in order to run ANSYS, Inc. products. See [ANSYS License Manager Installation](#) (p. 34) for license manager installation instructions.

---

1. Save all data and close all Windows applications before continuing.
2. If you downloaded the installation files or if you are installing from a DVD or USB with autorun disabled, navigate to the location of the `setup.exe` file. Right-click the `setup.exe` file and select **Run as administrator**.

The ANSYS, Inc. Installation Launcher appears.



From the options on the left side of the launcher you can install ANSYS products, ANSYS EKM Server, MPI for ANSYS Parallel Processing, and the ANSYS License Manager. You can access the installation guide for the ANSYS EKM Server from the **Downloads** page on the Customer Portal. The MPI for ANSYS Parallel Processing installation steps are displayed when you click the **Install MPI for ANSYS Parallel Processing** link.

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

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

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

The license agreement appears.

5. Read the agreement, and if you agree to the terms and conditions, click **I Agree**. Click **Next**.
6. The directory where you want to install the ANSYS, Inc. products is shown in the **Install Directory** field. You can install the products into any directory you want, but you must have write permissions to the directory you choose. The default is `C:\Program Files\ANSYS Inc.` We recommend using the default directory.

---

### Note

You must install all ANSYS, Inc. products into the same location. Installing products into different locations can cause product components to fail. If you choose an install directory via the Browse feature, the installation will automatically append `\ANSYS Inc\` to the chosen directory. The installation path can have a maximum of 100 characters, except

on AUTODYN, which has a maximum of 40 characters. For a network installation, the same UNC paths or mapped drive letters must be used on the server and all client computers.

- Leave the **Associate file extensions with ANSYS products** checkbox marked. This allows you to double-click files from Windows Explorer to launch the various products.
- If you specify a network location as the installation directory, you should select the **Add .NET security exceptions on the above install directory** option. This option allows the .NET binaries that are located under the \ANSYS Inc directory on the shared machine to be run from the network. To understand the implications of this security exception for .NET, please contact your system administrator or see Microsoft's website [http://msdn.microsoft.com/en-us/library/7c9c2y1w\(v=vs.80\).aspx](http://msdn.microsoft.com/en-us/library/7c9c2y1w(v=vs.80).aspx).
- Choose **Disable ANSYS RSS Feed** to disable automatic internet feeds to ANSYS, Inc. products.
- Used with network installations, this option allows the .NET binaries that are located under the \ANSYS Inc directory on the shared machine to be run from the network.

Click **Next**.

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

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

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

Click **Next**.

8. All products available in the installation package(s) you downloaded are listed. The installation program attempts to query your license server to pre-select your installation options. If the query is successful, the following message is displayed:

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

If the installation program was unable to query your license server, this message is not displayed and the default installation options are selected.

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

If you select AutoCAD, Catia V5, Inventor, NX, Creo Parametric, Solid Edge, or SolidWorks, you will have additional installation steps. Please see [Specifying CAD Configurations \(p. 33\)](#).

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

You will also see an estimate of the disk space required to install all of the selected components, and the disk space you have available. The actual amount of disk space required may be less, but if you choose to run the installation with insufficient disk space available, we strongly recommend

that you review the log files at the end of the installation to verify that all products were installed correctly. Installation log files are written to the installation directory.

---

**Note**

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

---

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

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

9. A summary of the selected installation data appears. Review the information carefully, and if correct, click **Next** to continue the installation.
10. The installation progress screen displays a status bar towards the bottom of the installation window. This status bar tracks the percentage of packages that have been installed on your computer. Depending on the number of products you have selected, the installation time required could be lengthy. You will not be able to interrupt the installation process. Please be patient while the installation completes.

---

**Note**

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

---

The installation program will first check your system to determine if you have the necessary prerequisites. If the prerequisites are not already on your system, the prerequisites will be installed automatically. You may be asked to run the necessary files (such as the .NET exe), or to reboot your machine, depending on your existing machine configuration. If you are asked, you will need to reboot in order to complete the prerequisite installation successfully.

Click **Next** to continue the installation.

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

---

**Note**

If prerequisite components necessary for the installation of the ANSYS products were installed, the following message may be displayed:

*A system restart is required before using any ANSYS Products.*

---

### 4.2.1.1. Network Server and Product Configuration

To complete a network installation (where the product is installed on one machine and multiple clients access that installation to run the product) to a file server machine, follow the steps below. A network installation must be homogeneous, although you can install on different operating systems.

1. Install the product(s) to be shared in a location that all clients can access. Install all platforms that this machine will host as described in [Product Installation with Client Licensing \(p. 27\)](#).

You must share the entire `\ANSYS Inc` directory, not just the `\v162` directory. If mapping a drive to the shared location, it must be mapped to the `\ANSYS Inc` folder.

2. Open the Product Configuration Manager (`\\fileservermachine\ANSYS Inc\v162\Product-Config.exe`). Select **Install Required Prerequisites**. You must be logged in as an administrative user to install the prerequisites. This step must be completed on all client Windows systems.
3. A configuration status screen appears. This screen displays the required prerequisite installation process. Once the process is complete, click **Exit**. The Product Configuration Manager remains open.
4. On each Windows client, update the .NET security settings. By default, the .NET 3.5 Framework prevents applications from being run over a network. In order to install over a network, each Windows client that accesses the shared directory requires a modification to its security settings. This modification allows the .NET binaries that are located under the `\ANSYS Inc` directory on the shared machine to be run from the network. To make this modification from the Product Configuration Manager, select **Add .NET security exception for ANSYS, Inc. products**. You must have administrative privileges to modify the security policy.

To understand the implications of this security exception for .NET, please contact your system administrator or see Microsoft's website [http://msdn.microsoft.com/en-us/library/7c9c2y1w\(v=vs.80\).aspx](http://msdn.microsoft.com/en-us/library/7c9c2y1w(v=vs.80).aspx).

Alternatively, you can allow network execution manually rather than using the **Add .NET security exception for ANSYS, Inc. products** option of the Product Configuration Manager. The **Add .NET security exception for ANSYS, Inc. products** option runs the Caspol utility, which is located under the .NET Framework installation, in `C:\Windows\Microsoft.NET\Framework64\v2.0.50727` for a 64-bit machine. You can run this utility manually in a command prompt window to open full trust to files on the shared drive.

On a 64-bit machine, run the following command:

```
<os drive>\Windows\Microsoft.NET\Framework64\v2.0.50727\Caspol.exe -m -ag
<CodeGroup> -url file://x:/* FullTrust
```

where `x` is the mapped drive or UNC path where the product is installed, and `CodeGroup` is the appropriate Caspol permissions level. We recommend a `CodeGroup` setting of 1.2. If that setting does not work or is not appropriate for your environment, please see the Caspol documentation available from Microsoft or contact your system administrator.

You must install Caspol as an administrative user. If running this from the command line in Windows 7 or Windows 8.1, you must start the command prompt as Administrator, or the command will fail.

5. If you are sharing the Mechanical APDL product, edit the `tlbrlist162.ans` file, located in `\v162\ansys\gui\en-us\toolbars` on the Windows file server. The file should read as follows:

## Installing the ANSYS Products and the License Manager on Different Windows Machines

---

```
\\fileservermachine\ANSYS Inc\v162\ansys\gui\en-us\toolbars\ANSYSSTANDARD.TLB  
\\fileservermachine\ANSYS Inc\v162\ansys\gui\en-us\toolbars\ANSYSABBR.TLB  
\\fileservermachine\ANSYS Inc\v162\ansys\gui\en-us\toolbars\ANSYSGRAPHICAL.TLB
```

- For all types of network installations, from each client Windows machine, you will need to run `ProductConfig.exe`. If the Product Configuration Manager has been closed, please reopen it as described above.
- Select **Configure Products**.
- Select the products you want to configure. Click **Configure**.

---

### Note

CFD-Post is automatically configured when CFX, Fluent, Polyflow, or Icepak is configured, or if CFD-Post is selected.

---

- An informational message box appears as each product is configured.

All operations and error messages that are encountered while using the product configuration (or unconfiguration) option are written to the `productConfig.txt` file in the `%TEMP%` directory of the client machine. No log files are written to the machine that is hosting `ProductConfig.exe`.

---

### Note

`ProductConfig.exe` can also be run in batch mode using the '-silent' flag, which may be useful for automating configuration for multiple systems or across a Microsoft Windows HPC cluster. For more information see [Silent Mode Operations \(p. 45\)](#).

---

- Select **Configure CADs**.
- The **CAD Configuration Manager** opens. Run this utility for all of the CAD products you need to configure. See [Using the CAD Configuration Manager](#) for detailed instructions on running this utility, or select **Help** from the menu bar in the **CAD Configuration Manager**.
- When the configuration is completed, you will see shortcuts to each product in Windows 7 under **Start > All Programs > ANSYS 16.2** and in Windows 8.1 under **All apps > ANSYS 16.2**.

**Unconfigure** To unconfigure products that have previously been shared, use the following procedure:

- From each client machine, in Windows 7, select **Start > All Programs > ANSYS 16.2 > Utilities > Product Configuration 16.2** and in Windows 8.1, right-click the **Start Screen > All apps > ANSYS 16.2 > Product Configuration 16.2**.
- Select **Unconfigure Products**.
- Select the products you want to unconfigure. Click **Unconfigure**. When unconfiguring products, be aware that WB/Common should be unconfigured last, after all other products are unconfigured. If you are unconfiguring some but not all products, do NOT unconfigure WB/Common, or your remaining products will not run correctly.

**CFD-Post** CFD-Post can be configured as a standalone product; however, as a standalone product, it is also automatically configured with Fluent, Polyflow, and Icepak. To unconfigure CFD-Post, you must select all of these products to unconfigure, regardless if they are configured locally.

- If you choose to unconfigure standalone CFD-Post without unconfiguring the other products, you will not be able to proceed with the unconfigure. You will need to revise your selections before continuing.
  - If you choose to unconfigure all of the other products that include CFD-Post but you do not select CFD-Post, then CFD-Post will not be unconfigured.
4. The user interface billboard displays the status of each product as it is unconfigured. The shortcuts to each product are now removed.

### 4.2.1.2. Specifying CAD Configurations

#### Creo Parametric

1. If you selected Creo Parametric, you will have to choose which Creo Parametric 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**).
2. If you choose the Workbench Associative Interface, you may need to specify the Creo Parametric language, the Creo Parametric command, and the full Creo Parametric installation path (C:\Program Files\PTC\Creo 1.0\Parametric by default) for an existing Creo Parametric installation. Click **Next**.

#### NX

1. 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**).
2. 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. If you are an administrative user, a file required to load the NX plug-in is placed in the administrative user's Application Data folder by default, which may not be accessible by other, non-administrative users. To allow non-administrative users to run, you will need to define the environment variable **UGII\_CUSTOM\_DIRECTORY\_FILE** prior to installation and specify a location where other users have read access. Alternatively, you can run the **CAD Configuration Manager** after the installation and provide an updated location for the NX Custom Directory File Path under the **NX** tab.
3. If you selected the ANSYS ICEM CFD NX Interface product but not the NX Geometry Interface product on the product selection screen, you may need to specify the NX installation path or choose to skip configuring NX and use the **CAD Configuration Manager** to configure later.

#### CATIA v5

- If you selected CATIA v5 and any product that includes ANSYS Workbench, you will be required to select the appropriate CAD reader/plugin for ANSYS Workbench. You can choose to configure the Reader, CADNexus CAPRI CAE Gateway, or skip and configure later (using the **CAD Configuration Manager**).



## SolidWorks and Autodesk Inventor

- If you selected Inventor and/or SolidWorks as one of your CAD products, you will have to choose which Inventor and/or SolidWorks 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 have not yet installed these programs, or do not know the requested information, you can choose to skip these configuration steps. If you skip these steps, you will need to manually configure these CAD interfaces using the **CAD Configuration Manager** before you can successfully import models into Mechanical APDL, ANSYS Workbench, or ICEM CFD. You can access the **CAD Configuration Manager** in Windows 7 (**Start > All Programs > ANSYS 16.2 > Utilities > CAD Configuration Manager 16.2**) and in Windows 8.1 (**Right-click the Start Screen > All apps > ANSYS 16.2 > CAD Configuration Manager 16.2**). See [Configuring CAD Products \(p. 53\)](#) for more information on using the **CAD Configuration Manager**.

### 4.2.2. ANSYS License Manager Installation

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

1. On the server machine, navigate to the directory where the installation packages reside. Right-click the `setup.exe` file and select **Run as administrator**. The ANSYS, Inc. Installation Launcher appears. Click **Install ANSYS License Manager**.
2. You will see a warning stating that if the license manager is currently running, it will be shut down. You may safely ignore this message and click **OK**.
3. The license agreement appears. Read the agreement, and if you agree to the terms and conditions, click **I Agree**. Click **Next**.
4. The directory where the license manager will be installed is shown in the **Install Directory** field. Accept the default directory and click **Next**.

---

#### Note

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

---

5. The ANSYS, Inc. License Manager is selected as the only product available to install. As with the product installation, the required and available disk space numbers are shown. Click **Next**.
6. A summary screen appears that lists the products to be installed. Because this is a license manager installation, the ANSYS, Inc. License Manager is the only product listed.

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

After the License Manager installation has been completed, the **Launch License Management Center upon exiting** option is included on the screen. The **ANSYS License Management Center**



is a browser-based user interface that centralizes many of the ANSYS product licensing administrative functions. If you do not want to launch the License Management Center, uncheck the **Launch License Management Center upon exiting** option.

7. Click **Exit** to close the **License Manager Installation** screen.
8. Click **Exit** to close the ANSYS, Inc. Installation Launcher.
9. From the machine on which the products were installed, choose **Start> Programs> ANSYS 16.2> <product>** to launch the ANSYS, Inc. products that you have installed.

#### **4.2.2.1. Registering the License Server**

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

1. Open the **ANSYS License Management Center**.
2. Click the **Get System Hostid Information** option to display your system ID code(s).
3. Select the system ID you wish to use and click **SAVE TO FILE**.  
A text file containing your system ID information is created.
4. Forward this text file to your ANSYS sales representative so that a license file can be created for you.
5. Add your license files through the **ANSYS License Management Center**. For these steps, see [Adding a License with the ANSYS License Management Center](#).

After completing the installation process, please refer to [Post-Installation Instructions \(p. 37\)](#)



---

## Chapter 5: Post-Installation Instructions

---

The following post-installation procedures apply to all ANSYS, Inc. products. Individual products may have additional post-installation procedures; please refer to the following sections for each product:

- 5.1. [Post-Installation Procedures for Mechanical APDL and ANSYS Workbench Products](#)
- 5.2. [Launching ANSYS, Inc. Products](#)
- 5.3. [Running the ANS\\_ADMIN Utility for Mechanical APDL /ANSYS Workbench Products](#)
- 5.4. [Product Localization](#)

---

### Note

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

---

It may also be necessary to establish some system settings, including path names and environment variables. See your operating system documentation for specific instructions on setting paths and environment variables.

1. Set the following environment variables based on the behavior you want.
  - The **ANSYSLIC\_DIR** environment variable sets the location of the ANSYS licensing directory hierarchy. The default value is `c:\Program Files\ANSYS Inc\Shared Files\Licensing`. You probably will not need to reset this variable, unless you change the location of the licensing files.

To set environment variables:

**Windows 7:** Right-click **Computer** and choose **Properties**. Click **Advanced System Settings**, and click **Environment Variables**. Click **New**. Type the name in the Variable Name field and the desired setting in the Variable Value field. Click **OK** on all dialog boxes.

**Windows 8.1:** From the **Start Screen**, right-click, select **All apps** and then click the **File Explorer** icon. On the left side of the File Explorer, right-click **Computer** and choose **Properties**. Click **Advanced System Settings**, and click **Environment Variables**. Click **New**. Type the name in the Variable Name field and the desired setting in the Variable Value field. Click **OK** on all dialog boxes.

If you have any command prompts or console windows open when you set or reset environment variables, those windows need to be closed and restarted in order to pick up the new settings.

2. Set the home directory. To set a home directory in Windows, you need to set a **HOMEDRIVE** environment variable to the desired drive letter (including the colon) and a **HOMEPATH** environment variable to the desired path. For example:

```
HOMEDRIVE=C:  
HOMEPATH=\Users\Mine
```

3. Set the license manager to start automatically at boot time. For platform-specific instructions, see [License Manager Automatic Startup Instructions](#) in the [ANSYS, Inc. Licensing Guide](#).

4. Designate server(s) for license checkout and establish necessary user privileges (recommended but not required). For information on these tasks, see [Post-Installation Instructions for the License Manager](#) in the [ANSYS, Inc. Licensing Guide](#).
5. Make a backup copy of the \ANSYS Inc directory.
6. Verify the product installation by selecting each product from the **Start** menu to verify that they each start and run correctly. You must be pointing to a valid license server machine before you can verify the installation.
7. If you chose not to set file associations during the installation, you may want to run the File Association utility now to configure the ANSYS file types. Administrative privileges are required to run this utility

For Windows 7, select **Start> All Programs> ANSYS 16.2> Utilities> File Association 16.2** and for Windows 8.1, right-click the **Start Screen> All apps> ANSYS 16.2> File Association 16.2**.

For network installations, run the File Association utility on each client computer.

You can also run the File Association utility in silent mode:

```
<install_dir>\v162\commonfiles\tools\<platform>\fileassoc.exe -silent
```

You can see help on the fileassoc.exe by issuing -help. File associations are as follows:

ANSYS, Inc. Product	File Extension
Mechanical APDL	.db, .dbb, .grph
ANSYS CFD-Post	.mres, .res
ANSYS CFX	.cfx, .cvf, .def, .mdef
ANSYS Composite PrepPost	.acp
ANSYS Fluent	.cas
ANSYS ICEM CFD	.prj, .tin, .uns
ANSYS Space Claim Direct Modeler	.scdoc, .scjournal
ANSYS Workbench	.ad, .agdb, .aqdb, .bgd, .cmdb, .dsdb, .engd, .fedb, .mechdat, .meshdat, .wbdb, .wbpj, .wbpz

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

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

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

- Set the following environment variables based on the behavior you want. Set the environment variables following the conventions of your operating system. Not all of these are required for all integrated ANSYS

Workbench products (such as ANSYS Autodyn), but setting them correctly for ANSYS Workbench will in no way hinder the performance of the other products.

- The **ANSYS162\_DIR** environment variable sets the location of the ANSYS directory hierarchy. The default value is `c:\Program Files\ANSYS Inc\V162\ANSYS`. You probably will not need to reset this variable, unless you change the location of the installed files.
- **ANSYS162\_PRODUCT** - set this to the correct product variable to run Mechanical APDL to start with the correct Mechanical APDL product without specifying the **-p** command modifier each time.
- **ANS\_CONSEC** - set this to YES to disable Mechanical APDL dialog boxes and allow multiple jobs to run consecutively without waiting for user input. Settings for **ANS\_CONSEC** are:

Value of <b>ANS_CONSEC</b>	Batch Dialog Box	GPF Dialog Box on Error
Not defined/default	No	Yes
YES	No	No
NO	Yes	Yes

- **ANSYS\_LOCK** - set to ON (default) to create file locks to prevent users from opening a new job with the same name and in the same directory as the current job.
- **ANSYS162\_MAT161** - set this environment variable to 1 to enable use of the LS-DYNA \*MAT\_COMPOSITE\_MSC material (requires an LS-DYNA MAT\_161 license).
- **ANSYS162\_MAT162** - set this environment variable to 1 to enable use of the LS-DYNA \*MAT\_COMPOSITE\_DMG\_MSG material (requires an LS-DYNA MAT\_162 license).
- **LSTC\_LICENSE** - This is an LS-DYNA environment variable that controls which license manager is used by the LS-DYNA executable. It is set to ANSYS during installation; changing this environment variable will prevent ANSYS LS-DYNA from using your ANSYS licenses when running the executable.

### 5.1.1. Post-Installation Procedures for ANSYS CFX and ANSYS CFD-Post

After installing the ANSYS CFX or ANSYS CFD-Post software, verify the installation as follows:

1. In Windows 7, select **Start > All Programs > ANSYS 16.2 > Fluid Dynamics > CFX 16.2**. In Windows 8.1, right-click the **Start Screen > All apps > ANSYS 16.2 > CFX 16.2**.

The CFX 16.2 Launcher opens.

2. In the **Working Directory** field of the Launcher, type the path to the `examples` directory, which by default is: `C:\Program Files\ANSYS Inc\v162\CFX\examples`
3. Click **CFD-Post 16.2**. CFD-Post opens.
4. Select **File > Load Results**. The **Load Results File** window opens.
5. Select **StaticMixer\_001.res** and click **Open**.

A wireframe of a mixer appears in the 3D Viewer.

6. Select **Session > Play Session**. The **Play Session File** window opens. Select **StaticMixerPost.cse** and click **Open**.

The mixer changes orientation, displays a plane colored by temperature, hides the plane, then loads a vector plot of temperature located on the plane.

7. Select **File > Close** and close the case without saving.
8. To exit CFD-Post, select **File > Quit**.

If any of the installation verification steps fails, contact your ANSYS Support representative.

### 5.1.2. Post-Installation Procedures for ANSYS Fluent

After installing the ANSYS Fluent software, you will need to reset the default values in the ANSYS Fluent launcher as follows:

1. Verify that the **FLUENT\_INC** environment variable is not set. From the Control Panel, select **System** and click the **Advanced** tab. Click **Environment Variables**. Find and delete the **FLUENT\_INC** variable.
2. In Windows 7, select **Start > All Programs > ANSYS 16.2 > Fluid Dynamics > Fluent 16.2**. In Windows 8.1, right-click the **Start Screen > All apps > ANSYS 16.2 > FLUENT 16.2**.
3. Click **Default**.
4. Click **Yes** when asked if you want to discard the LAUNCHER history.
5. Click **Cancel** if you do not want to start Fluent at this time. The new defaults will have been saved.

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

### 5.1.3. Post-Installation Procedures for ANSYS Polyflow

Polyflow no longer requires the **FLUENT\_INC** environmental variable. It should be deleted as is recommended for Fluent:

- From the Control Panel, select **System** and click the **Advanced** tab. Click **Environment Variables**. Find and delete the **FLUENT\_INC** variable.

### 5.1.4. Post-Installation Procedures for ICEM CFD

If you are running ICEM CFD over the network, you may not be able to access ICEM CFD Help using the Help menu. To access ICEM CFD Help, navigate to `commonfiles\help\en-us` in the installation directory on the server machine and copy the file `icemcfd_help.chm` to your local machine. Double-click the local copy to view the help.

### 5.1.5. Post-Installation Procedures for Other Products

**BladeGen** If you are running BladeGen on Windows 7, in order to access the BladeGen online help, you need to install `WinHlp32.exe`, which is available from Microsoft.

**BladeEditor** In order to use BladeEditor, you must set the Geometry license preference to ANSYS BladeModeler as follows:

1. In the ANSYS Workbench menu, select **Tools> License Preferences**.
2. In the **License Preferences** dialog box, click the **Geometry** tab.
3. If ANSYS BladeModeler is not the first license listed, then select it and click **Move up** as required to move it to the top of the list. If **ANSYS BladeModeler** is not in the list, then you need to obtain an ANSYS BladeModeler license.
4. Select ANSYS DesignModeler in the list and set its value to 0 (which means “Don't Use”). This step prevents DesignModeler from using an ANSYS DesignModeler license when an ANSYS BladeModeler license is not available.
5. Click **OK** to close the dialog box.

**FEMGV** For ANSYS Aqwa customers, FEMGV is also available as a separate, standalone installation, available via media or the ANSYS Customer Download Center.

## 5.2. Launching ANSYS, Inc. Products

To launch ANSYS, Inc. products on Windows 7 platforms, choose the product from the ANSYS 16.2 program group under the **Start** menu (**Start> All Programs> ANSYS 16.2**) and on Windows 8.1, under **All apps**.

For Mechanical APDL, you can also use the launcher:

**Windows 7: Start> All Programs> ANSYS 16.2> Mechanical APDL Product Launcher 16.2.**

**Windows 8.1: Start Screen> All apps> ANSYS 16.2> Mechanical APDL Product Launcher 16.2.**

## 5.3. Running the ANS\_ADMIN Utility for Mechanical APDL /ANSYS Workbench Products

You may need to run the **ANS\_ADMIN** utility to relink Mechanical APDL if you use the customization tools. For more information on individual options available with this utility, see the online help accessible from **Help** buttons on the utility dialog boxes. You can launch **ANS\_ADMIN** without administrative privileges, but some of the options require administrative privileges. If certain options are grayed out, you do not have the necessary system administrator privileges necessary for that option, or the corresponding product may not be installed.

To run the **ANS\_ADMIN** utility from Windows 7, choose **Start> All Programs> ANSYS 16.2> Utilities> ANS\_ADMIN 16.2**. To run the **ANS\_ADMIN** utility from Windows 8.1, right-click the **Start Screen> All apps> ANSYS 16.2> ANS\_ADMIN 16.2**.

## 5.4. Product Localization

Many ANSYS, Inc. products are available in multiple languages, including English, German, French, and Japanese. For those products that are localized, you are able to view the GUI and messages in the specified language. See your specific product documentation for instructions on choosing a localized version of the product.

All products that are localized define the language via the `languagesettings.txt` file. In most cases, you will not have to manually edit this file. If you do need to edit it manually, you can use one of the following values:

en-us (English, default)  
de (German)  
fr (French)  
ja (Japanese)

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

1. `%appdata%\ANSYS\v162`
2. `<install_dir>\ANSYS Inc\v162\commonfiles\Language`

ANSYS, Inc. licensing also looks for the `languagesettings.txt` in the licensing languages subdirectories in order to display the server **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.

### 5.4.1. Translated Message File Installation for Mechanical APDL /ANSYS Workbench Products

If your ANSYS, Inc. sales representative has supplied you with message files translated into your local language, use the following procedure to install and access these files from within Mechanical APDL:

1. Create an appropriately named subdirectory to hold the message files. For example, if your local language is French, create a directory in the following location:

```
<os drive>\Program Files\ANSYS Inc\V162\ANSYS\docu\fr
```

2. Copy the message files (`msgcat.162`, `msgidx.162`, and `msgfnm.162`) into the newly created subdirectory.
3. Access these message files by using the `-l` command line option. For example:

```
ansys162 -l fr
```

Or, when you are running Mechanical APDL from the Mechanical APDL launcher, choose the **Language Selection** option and then pick the desired language.

You must create a newly translated message file for each release of Mechanical APDL because error messages may occur in a different order for each release.



---

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

---

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

### To perform the installation:

1. Save all data and close all Windows applications before continuing.
2. If you downloaded the installation files or if you are installing from a DVD/USB with autorun disabled, navigate to the location of the **setup.exe** file. Right-click the **setup.exe** file and select **Run as administrator**.

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

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

The license agreement appears.

4. Read the agreement, and if you agree to the terms and conditions, click **I Agree**. Click **Next**.
5. The directory where you want to install the ANSYS, Inc. help documentation is shown in the **Install Directory** field. You can install the products into any directory you want, but you must have write permissions to the directory you choose. The default is **C:\Program Files\ANSYS Inc**. We recommend using the default directory.

Click **Next**.

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

Click **Next** to begin the installation.

7. When the help documentation installation has completed, click **Exit**. The product installation window reappears. You can close this window at this time.

---

### Note

Alternatively, you can run the document installation by running the **SetupDocOnly.exe** installation file. This file is located in the *windows* directory on USB drives.

---

The ANSYS help documentation can be installed silently by running the **SetupDocOnly.exe** installation file with the `-silent` flag. For more information, see [Silent Mode Operations](#).



---

## Chapter 7: Silent Mode Operations

---

### 7.1. Installing ANSYS, Inc. Products

This section explains how to install ANSYS, Inc. products.

ANSYS, Inc. requires that you have full administrative privileges when installing any software on Microsoft Windows 7 or Windows 8.1. Non-administrative accounts do not usually have the permissions required to access system areas that installation programs often need to modify. “Full administrative privileges” means that you are running as administrator with UAC turned off, or you are running as administrator with UAC turned on and **Run as Administrator** selected. Because of changes to the UAC functionality, it is necessary that you right-click and select **Run as Administrator** from the context menu when installing to Windows 8.1.

Windows 7 and Windows 8.1 use a feature called User Account Control (UAC) to control privileges and automatically reduce the potential of security breaches in the operation system. However, UAC limits your accessibility to system areas and can cause unpredictable behavior in ANSYS, Inc. products, as noted below.

#### 7.1.1. Silent Mode Operations

ANSYS, Inc. supports silent mode operations, including installation, configuring/unconfiguring, and uninstalling.

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. Not all products are available on all platforms.

For silent mode operations, if you do not specify any product arguments, all available products will be installed, configured, or uninstalled.

#### **Product Flags**

<b>Product</b>	<i>product_flag</i>
Mechanical APDL	-mechapdl
ANSYS Customization Files	-ansyscust
ANSYS Autodyn	-autodyn
ANSYS LS-DYNA	-lsdyna
ANSYS CFD-Post	-cfdpost
ANSYS CFX	-cfx
ANSYS TurboGrid	-turbogrid
ANSYS Fluent	-fluent
ANSYS Polyflow	-polyflow
ANSYS Aqwa	-aqwa

<b>Product</b>	<b>product_flag</b>
ANSYS ICEM CFD	-icemcfd
ANSYS AIM	-aim

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

ANSYS Composite PrepPost	-acp
ANSYS ICEM CFD Creo Parametric Interface	-icemproe
ANSYS Icepak	-icepak
Remote Solve Manager Standalone Services	-rsm
ANSYS TeamCenter Engineering	-tceng
Creo Parametric	-proe
NX	-ug
CATIA 4.x	-catia4
CATIA V5	-catia5
CATIA V6	-catia6
Parasolid	-parasolid
ACIS	-acis
SolidWorks	-solidworks
Solid Edge	-solidedge
Autodesk Inventor	-adinventor
Creo Elements/Direct Modeling	-cocreate
JTOpen	-jtopen
AutoCAD	-acad
SolidWorks Reader	-solidworks_reader
SolidWorks Plugin	-solidworks_plugin
CATIA 5 Reader	-catia5_reader
CATIA 5 Plugin	-catia5_plugin
Autodesk Inventor Reader	-adinventor_reader
Autodesk Inventor Plugin	-adinventor_plugin
Creo Parametric Reader	-proe_reader
Creo Parametric Plugin	-proe_plugin
NX Reader	-ug_reader
NX Plugin	-ug_plugin

### **7.1.1.1. Silent Product and License Manager Installation**

#### **Caution**

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

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

```
setup.exe -silent -product_flag
```

---

## Note

In Windows 8.1 it is necessary to run silent commands from an administrator prompt. To access a Run as Administrator command prompt, click **Start Screen> All apps>** right-click the **Command Prompt Icon>** select **Run as Administrator**.

---

The above form will install the products specified (see the list of *product\_flags* above). Additional command line arguments are available; please see the list below.

For example, to install TurboGrid and Icepak to the default installation directory, issue the following command:

```
setup.exe -silent -install_dir "C:\Program Files\ANSYS Inc" -turbogrid -icepak
```

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

```
setupLM.exe -silent
```

The silent license manager installation is valid only for the default Licensing Configuration option "Run the ANSYS Licensing Interconnect with FLEXlm." Please see [Installing the ANSYS License Manager of the ANSYS, Inc. Licensing Guide](#) for more information.

If you are installing the license manager to a non-default installation directory, you can use the silent installation method, but only for the initial installation. To change the license manager installation directory for future installations, you will need to uninstall the license manager before reinstalling to a different directory.

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

-silent	Initiates a silent installation.
-install_dir "path"	Specifies the directory to which the product is to be installed. For the installation directory, you must enclose the path in quotes if you have spaces in the pathname. If you want the product to install to the default location, you can omit the <code>-install_dir</code> argument. The default location is <code>&lt;os drive&gt;:\Program Files\ANSYS Inc\</code> .
-product_flag	Specifies one or more products to install. If you omit the <code>-product_flag</code> argument, all products will be installed. See the list of valid <i>product_flags</i> above.
-productfile "path"	You can specify an options file that lists the products you want to install. To do so, you must provide a full path to a file containing desired products. See <a href="#">Specifying Products with an Options File (p. 49)</a> for more details.
-disablerss	Disables automatic internet feeds to ANSYS, Inc. products.
-caspol	Allows the .NET binaries that are located under the <code>\ANSYS Inc</code> directory on the shared machine to be run from the network. You must have administrative privileges to modify the security policy. Use this option when you specify a network location as the installation directory. To understand the implications of this security

	exception for .NET, please contact your system administrator or see Microsoft's website <a href="http://msdn.microsoft.com/en-us/library/7c9c2y1w(v=vs.80).aspx">http://msdn.microsoft.com/en-us/library/7c9c2y1w(v=vs.80).aspx</a> .
-help	Displays a list of valid arguments for a silent installation.
-licfilepath "path"	Specifies the location of the license file to install. If the path is not specified or if the path is the same as the existing license file, the license file will not be installed. Valid only when doing a silent license manager installation (setupLM.exe).
-setliclang language	Specifies a language to use for the server <b>ANSLIC_ADMIN</b> 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 <i>language</i> value. This flag can be used during a GUI installation as well. Valid only when doing a license manager installation (setupLM.exe).
-licserverinfo	<p>Specifies information to be used by the client for the license server. Valid only in conjunction with a silent installation (setup.exe). The format is:</p> <p>Single license server:</p> <p><i>LI port number:FLEXlm port number:hostname</i></p> <p>Example:</p> <p>2325:1055:abc</p> <p>Three license servers:</p> <p><i>LI port number:FLEXlm port number:hostname1,hostname2,hostname3</i></p> <p>Example:</p> <p>"2325:1055:abc,def,xyz"</p> <p>The default values for the Licensing Interconnect and FlexNet port numbers (2325 and 1055, respectively) will be used if they are not specified. However, you do need to include the colons. In a three-server environment, you also need to enclose the values in quotes (Windows only).</p> <p>Example:</p> <p>::abc</p> <p>or</p> <p>"::abc,def,xyz"</p> <p>Information specified via -licserverinfo will be appended to existing information in the ansyslmd.ini file. To change information already in your ansyslmd.ini file, you must use the <b>ANSLIC_ADMIN</b> utility.</p>
-nohelp	By default, for the products that are being installed, all help documentation is included. The -nohelp argument removes all help from the installation.
-lang	Specifies a language to use for the products.

The silent installation process will automatically close certain applications, possibly resulting in lost data. You should always close all programs before starting a silent install. A silent install will first install any necessary prerequisites that are not already on your system and then continue with the installation. You must have administrative privileges to install prerequisites; if not, the silent install will exit and write a message to the log file.

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

For more information on the silent license manager installation, see [Installing the ANSYS License Manager of the ANSYS, Inc. Licensing Guide](#).

## Specifying Products with an Options File

You can also specify an options file on the command line using the `-productfile "path"` option. The options file can specify which products you want to install. The options file can contain all possible products, with the products you do not want to install commented out, or it can contain only the products you want to install. An example options file is shown below. In the example, Creo Parametric and NX are commented out using the acceptable comment indicators. When using the options file, do not include the dash (-) before the product name.

```
mechapid
ansyscust
autodyn
cfdpost
cfx
turbogrid
fluent
polyflow
icemproe
icepak
::proe
REM ug
```

To run with NX and Creo Parametric:

- NX: Verify that your UGII environment variables are set, and the configuration will pick them up automatically.
- Creo Parametric: Verify that the following environment variables are set:

```
PROE_START_CMD162=C:\Program Files\PTC\Creo 1.0\Parametric\bin\para-
metric1.bat (or your Creo Parametric start command)
PROELOADPOINT= C:\Program Files\<Creo_dir>
```

## Silent Prerequisites Installation

The ANSYS, Inc. product installation requires several prerequisites. These prerequisites often require administrative permissions to install. If you want to install as a non-administrative user but require prerequisites, you can install the prerequisites separately by running `InstallPreReqs.exe` from the top level directory as an administrator. You can also install the prerequisites silently using the following command:

```
InstallPreReqs.exe -silent
```

### 7.1.1.2. Silent Product Configuration/Unconfiguration

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

From each client machine, run the `ProductConfig.exe -silent` with the following options: `-config`, `-unconfig`, `-prereqs`, `-caspol`, and `-product_flag`. These options are described below.

- `-config/-unconfig` -- Use with the `-silent` option to indicate that products should be configured/unconfigured.
- `-prereqs` -- Required to install the necessary prerequisites on your system. Prerequisites must be installed once on each machine.
- `-caspol` -- Required to modify the necessary .NET security settings to allow the product to run across the network. This option needs to be run once on each machine.
- `-product_flag` -- Use to specify which products should be configured; see the above list of `product_flags`. If you do not specify one or more products, all products that have been installed will be configured.
- Use the `-help` option for a list of commands available for use with `ProductConfig.exe`.

An example command for silent configuration is:

```
"\\machineabc\Program Files\ANSYS Inc\v162\ProductConfig.exe" -silent -prereqs  
"\\machineabc\Program Files\ANSYS Inc\v162\ProductConfig.exe" -silent -caspol  
"\\machineabc\Program Files\ANSYS Inc\v162\ProductConfig.exe" -silent -config
```

All operations and error messages that are encountered while using the silent product configuration/unconfiguration option are written to the `productConfig.txt` file in the `%TEMP%` directory of the client machine. No log files are written to the machine that is hosting `ProductConfig.exe`.

### 7.1.1.3. Silent Media Installation

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

- Copy the contents of each DVD to a folder on the machine's hard disk such that the `162-number.dvd` files of each DVD are located in the same directory. You can then proceed with the silent installation as described earlier.
- Place all of the media in separate drives (any combination of virtual ISO mounts or hardware drives) so that they can be accessed simultaneously during the installation. Then run the silent installation as described earlier, but include the additional `-media_dir2 <path>` option for each drive:

```
setup.exe -silent -product_flag -media_dir2 <path>
```

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

### 7.1.1.4. Silent Uninstall

To run a silent uninstall, run the following from some location other than the `\v162` directory:

```
C:\> <installation_path>\v162\Uninstall.exe -silent
```



The silent uninstall will unconfigure and remove all products and remove the entire `\v162` directory, including any user files or subdirectories that you have added to the `\v162` directory.

Because of the way ANSYS, Inc. products are packaged, not all of the *product\_flags* shown above are valid. To uninstall individual products, use the following product options in conjunction with the `-silent` argument:

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

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

```
C:\> <installation path>\v162\Uninstall.exe -silent -turbogrid -icepak
```

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

**Uninstalling CFD-Post** CFD-Post can be installed as a standalone product. As a standalone product, it is also automatically installed with Fluent, Polyflow, and Icepak. To uninstall the standalone version of CFD-Post, you must specify `-cfdpost` as well as all of the above products that you have installed.

Note that CFD-Post is also installed with CFX; however, you do not need to uninstall CFX to uninstall the standalone version of CFD-Post, and uninstalling the standalone version of CFD-Post will not remove the CFD-Post capability from CFX.



---

## Chapter 8: Configuring CAD Products

---

The connection, geometry interface, and other CAD functionality for all supported CAD products is included with the ANSYS release media. Select the CAD systems you want to install during the installation process. If you choose not to install CAD systems during the installation, you can install them at a later date.

To run the connection or geometry interface functionality, you need to:

1. Ensure that the CAD product is correctly installed and licensed.
2. Ensure that you have the correct ANSYS, Inc. license(s).
3. Run the installation setup.

For complete information about the files you can import and the platforms supported by the connection capability, see the *ANSYS Connection User's Guide*. For more information about geometry interface information in ANSYS Workbench, see the topics **Attach to Active CAD Geometry** and **Import External Geometry File** in the ANSYS DesignModeler help.

---

### Caution

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

---

The connection for Creo Parametric requires you to run Mechanical APDL, Creo Parametric and the connection for Creo Parametric on the same machine. The connection for NX requires you to run Mechanical APDL, NX, and the connection for NX on the same machine. The connections for CATIA, SAT, and Parasolid do not require any additional CAD installation.

### 8.1. Using the CAD Configuration Manager

The **CAD Configuration Manager** allows you to configure geometry interfaces for Mechanical APDL and ANSYS Workbench. CAD configuration is typically handled during the product installation; however, if you chose to skip those steps, or if you make changes to your local CAD configuration between releases (for example, you install, move, or update your CAD package, or remove it entirely), you can use this utility.

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

- CAD Selection
- Creo Parametric
- NX
- Teamcenter (active on Windows only)

- CAD Configuration

This document describes how to use the **CAD Configuration Manager** as a wizard, beginning with the **CAD Selection** tab and using the **Next** button to progress through the configuration process. You can also manually select each tab to progress through the configuration process; both methods work the same. However, the applicable tabs will not be enabled until you choose the associated product on the **CAD Selection** tab.

1. From Windows 7, click **Start > All Programs > ANSYS 16.2 > Utilities > CAD Configuration Manager 16.2**. From Windows 8.1, right-click the **Start Screen > All apps > ANSYS 16.2 > CAD Configuration Manager 16.2**.
2. On the **CAD Selection** tab, choose the CAD products that you need to configure. Click **Next**.
3. If you did not select any CAD interfaces the necessary prerequisites will be registered, enabling **Geometry (DesignModeler)**. You must run the **CAD Configuration Manager** as an administrative user for some CAD registrations to take place. Select the **CAD Configuration** tab and click **Configure Selected CAD Interfaces** to register the prerequisites.
4. If you selected Creo Parametric as one of your CAD products, you will have to choose which Creo Parametric product to configure:
  - a. The **Reader** option specifies the non-associative source and doesn't require a Creo Parametric installation. (This reader does *not* support CAD associativity or parameters.)
  - b. The **Workbench Associative Plug-In** option requires Creo Parametric to be installed. When you choose this option, the Creo Parametric tab opens.
    - i. Enter or browse to the Creo Parametric installation location (for example, `C:\Program Files\PTC\Creo 2.0\Parametric`).
    - ii. Enter or browse to the Creo Parametric start command (for example, `C:\Program Files\PTC\Creo 2.0\Parametric\bin\parametric.bat`). Include the complete path if entering the command manually.
    - iii. Click **Next**.

---

### Note

When ANSYS Workbench is installed by an administrative user, the Creo Parametric plug-in will load for all users of the designated CAD install. However, when multiple Creo Parametric installations are present on the system, the plug-in should only be configured for a single installation of Creo Parametric; otherwise unconfiguring the plug-in will cause load problems for all other installations of the CAD.

---

5. If you selected NX as one of your CAD products, you will have to choose which NX product to configure:
  - a. The **Reader** option specifies the non-associative source and doesn't require an NX installation. (This reader does *not* support CAD associativity or parameters.)
  - b. The **Workbench Associative Plug-In** option requires NX to be installed. When you choose this option, the **NX** tab opens.

- i. Enter or browse to the NX installation location (for example, `C:\Program Files\Siemens\NX 8.5`).
- ii. Enter or browse to the NX custom directory file path (for example, `C:\Documents and Settings\UserName\Application Data\ANSYS\v162\Unigraphics`).

---

**Note**

If you are installing as an administrator, this file path will be defined in your administrator's user space by default and non-administrative users will not be able to run. Use this setting to specify a new location for the custom directory file path that is accessible by all users.

---

- iii. Click **Next**.
6. If you selected Inventor and/or SolidWorks as one of your CAD products, you will have to choose which Inventor and/or SolidWorks product to configure:
    - a. The **Reader** option specifies the non-associative source and doesn't require an Inventor or SolidWorks installation. (This reader does *not* support CAD associativity or parameters.)
    - b. The **Workbench Associative Plug-In** option requires Inventor or SolidWorks to be installed.
  7. If **SpaceClaim Engineer** is installed, you will have to choose to configure the **ANSYS Workbench Plug-In** for either **SpaceClaim Direct Modeler** or **SpaceClaim Engineer**. Otherwise you are limited to configuring the **SpaceClaim Direct Modeler Plug-In**.
  8. If you selected Teamcenter as one of your CAD products, the Teamcenter tab opens:
    - a. Enter or browse to the Teamcenter installation location (for example, `C:\SIEMENS\Teamcenter2008`).
    - b. Discriminator is a client-defined identifier for a session. For example, `ANSYSWBPlugInTC`.
    - c. Choose **Next**.
  9. If you selected Catia V5 as one of your CAD products, you will have the option to choose which CATIA V5 product to configure.
    - a. Select the **Reader** option to configure the CATIA V5 standalone reader that does *not* require an installation of the CATIA V5 CAD program. (This reader does *not* support CAD associativity or parameters.)
    - b. Select the **CADNexus/CAPRI CAE Gateway** option to configure the CATIA V5 associative reader that *does* require an installation of the CATIA V5 CAD program and the CADNexus/CAPRI CAE Gateway (this geometry interface *does* support CAD associativity and parameter modifications).
    - c. Click **Next**.
  10. 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.

11. 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.`

---

### Important

If you change your source from reader to associative plug-in or vice-versa (Creo Parametric, NX, SolidWorks, Autodesk Inventor, or Catia V5), you must first unconfigure the existing source and then configure using the new source.

---

## 8.1.1. Unconfiguring

If you need to unconfigure any of your CAD products, follow the steps above, but choose **Unconfigure Selected CAD Interfaces** on the **CAD Configuration** tab.

## 8.1.2. Running the CAD Configuration Manager in Batch Mode

You can configure ANSYS Geometry Interfaces by supplying the **CAD Configuration Manager** with arguments specific to the CAD sources you want to make available. The following table contains a list of supported arguments.

The command to run the **CAD Configuration Manager** in batch mode on Windows is:

```
"<installpath>\commonfiles\CAD\bin\winx64\Ans.CadInt.CADConfigurationUtility.exe"
-arguments
```

Argument	Value	Comment
UNCONFIGURE_SPECIFIED	None	Results in any specified CAD sources being unconfigured. When this flag is absent, the <b>CAD Configuration Manager</b> will attempt to configure all designated CAD sources.
UNCONFIGURE	None	Results in all CAD sources being disabled along with prerequisite libraries.
Either: PE_CONFIG_WB or PE_CONFIG_WBSPATIAL	None	Specify Creo Parametric source as either the associative plug-in or the reader (Spatial, no Creo Parametric install required). When the plug-in is specified, additional arguments PROLOADPOINT and PROE_START_CMD are required.
PROLOADPOINT	Full path to Creo Parametric installation (quotes required on Windows).	Not required with unconfigure operation.

Argument	Value	Comment
PROE_START_CMD	Full path to command used to launch Creo Parametric (quotes required on Windows)	Not required with unconfigure operation.
Either: UG_CONFIG_WB or UG_CONFIG_WBSPATIAL	None	Specify NX source as either the associative plug-in or the reader (Spatial, no NX install required). When the plug-in is specified, the argument UGII_BASE_DIR must also be specified.
UGII_BASE_DIR	Full path to NX installation	This should agree with environment variable <b>UGII_BASE_DIR</b> . Not required with unconfigure operation.
Either: CATIA_SPATIAL or CATIA_CAPRI	None	Specify Catia V5 source as either the Reader (Spatial, no Catia install required) or CAPRI (CADNexus install required).
OSDM_CONFIG	None	Configure/unconfigure Creo Elements/Direct Modeling.
Either: INVENTOR_CONFIG or INVENTOR_CONFIG_WBSPATIAL	None	Specify Inventor source as either the associative plug-in or the reader (Spatial, no Inventor install required).
Either: SOLIDWORKS_CONFIG or SW_CONFIG_WBSPATIAL	None	Specify SolidWorks source as either the associative plug-in or the reader (Spatial, no SolidWorks install required).
SOLIDEDGE_CONFIG	None	Configure/unconfigure Solid Edge.
ACAD_CONFIG	None	Configure/unconfigure AutoCAD.
TEAMCENTER_CONFIG	None	Configure/unconfigure Teamcenter; for configuration requires TC_ROOT flag.
IMAN_ROOT	Path to Teamcenter installation.	Not required with unconfiguration operation.
JTOPEN_CONFIG	None	Configure/unconfigure JT.
CATIA4_CONFIG	None	Configure/unconfigure CATIA v4.

### Note

All arguments require a dash (-) before them in order to be properly recognized by the **CAD Configuration Manager**. Arguments' values should not have a dash preceding them.

For example, you can configure Creo Parametric and SolidWorks Geometry Interfaces to ANSYS Workbench from the command line by using the following:

```
"<installpath>\commonfiles\CAD\bin\winx64\Ans.CadInt.CADConfigurationUtility.exe"
-SW_CONFIG -PE_CONFIG_WB -PROELOADPOINT "C:\Program Files\PTC\Creo 2.0\Parametric"
-PROE_START_CMD "C:\Program Files\PTC\Creo 2.0\Parametric\bin\parametric.bat"
```

where *installpath* is the same as the value of environment variable **%AWP\_ROOT162%**, *platform* is the value of environment variable **%ANSYS\_SYSDIR%**, and Creo Parametric is installed to `C:\Program Files\PTC\Creo 2.0\Parametric`.

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

```
"<installpath>\commonfiles\CAD\bin\winx64\Ans.CadInt.CADConfigurationUtility.exe"  
-unconfigure -SW_CONFIG -PE_CONFIG_WB
```

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

### 8.1.3. Network Considerations

When Workbench is installed to a network location (made available from a mapped network drive), any configuration actions will make the selected geometry interfaces available/unavailable only to the user who is currently logged in. To make changes that apply to all users, the configure/unconfigure action must be run as an administrator logged into the system hosting the Workbench installation. Configuration actions performed as an administrator on this server machine will impact all users on all machines referencing that install, except for users who have overridden the server configuration by configuring for themselves.

Prior to running the **CAD Configuration Manager** from a network location, the product configuration tool must be executed. This process will result in the client system being configured to match the geometry interface selections on the server. Products requiring additional information such as interfaces to Creo Parametric, NX and Teamcenter may fail to configure. This can be remedied either by manually running CAD Configuration Manager following Product Configuration, or by setting appropriate environment variables prior to execution of Product Configuration such as UGII\_CUSTOM\_DIRECTORY\_FILE, PROE\_START\_CMD150 and TC\_ROOT. TC\_ROOT can be set to the path of your Teamcenter installation. Further details regarding environment variables pertaining to Creo Parametric and NX are offered in the interface specific help sections below. For more information on using the Product Configuration manager, please see [Network Server and Product Configuration \(p. 31\)](#).

For loading associative geometry interfaces into their respective CAD environments, when the Workbench installation is referenced from a network location:

- A local administrator's configuration changes will set up associative CAD interfaces to load into the CAD environment of all local users, when that scope is selected (applicable to AutoCAD, Inventor, SolidWorks, Solid Edge, Creo Elements/Direct and NX).
- The associative geometry interface for Creo Parametric will load into the CAD environment of all users when the CAD installation is writable for that administrative user. Otherwise, it will load only into the CAD environment of the administrator who is currently logged in, and each additional user must be configured individually.

CAD Readers for JT, Creo Parametric, Inventor, NX and Solidworks may not fully unconfigure on a client of a network installation, even when this action is attempted by a client's administrator. An administrative user on the server must execute an unconfigure action, using CAD Configuration Manager, in order for these interfaces to be fully deactivated on the client.

### 8.1.4. Uninstalling

#### **Warning**

Do not proceed with an uninstall for CAD-specific unconfigure actions or you will leave some ANSYS Workbench products unusable.



## 8.1.5. Creo Parametric Configuration

Running the **CAD Configuration Manager** for Creo Parametric performs the following steps to activate the Creo Parametric plug-in:

- Sets the environment variable **PROE\_START\_CMD162** to the file used to launch Creo Parametric (for example `C:\Program Files\PTC\Creo 2.0\Parametric\bin\parametric.bat`).
- For administrative installations, adds the ANSYS 16.2 entry to the `config.pro` file located in `<creo_path>\text`. An example for Creo Parametric would be:

```
PROTKDAT C:\Program Files\PTC\Creo 2.0\Common Files\M100\text
```

An example for Wildfire 5.0 would be:

```
PROTKDAT C:\Program Files\proeWildfire 5.0\text
```

- Updates the `WBPlugInPE.dat` file referenced in the `config.pro` file, so that it contains information for loading the WorkBench Plug-In and the ANSYS Connection product.
- Registers the Plug-In file `WBPlugInPECOM.dll` (Creo Parametric) or `DSPlugInPEWF5COM.dll` (Wildfire 5.0) referenced in the `WBPlugInPE.dat` file.
- When Workbench is installed on a network location and the local administrator has write permissions to `<creo_path>\text`, a version-specific `WBPlugInPE162.dat` file will be created and placed in that folder. However, when `<creo_path>\text` is unwritable, both the `config.pro` and the `WBPlugInPE162.dat` will be placed in either `%HOME%` or `%HOMEDRIVE%%HOMEPATH%`.

If you do not have write access to your ANSYS Workbench installation, you may encounter the following error when attempting to import Creo Parametric models without an active CAD session:

**No write access, please choose another startup directory for trail file creation.**

To prevent this issue, add the following line to your `config.pro` file:

```
trail_dir $TEMP
```

**WBPlugInPE.dat File Contents** The `WBPlugInPE.dat` file should look like this example:

```
NAME WB162PluginProWF
EXEC_FILE E:\Program Files\ANSYS Inc\V162\AISOL\CADIntegration\ProE\
  winx64\WBPlugInPECOM.dll
TEXT_DIR E:\Program Files\ANSYS Inc\V162\AISOL\CADIntegration\ProE\
  ProEPages\Language\<locale>
STARTUP dll
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
REVISION ProeWildfire
END

NAME ac4pro162dll
exec_path E:\Program Files\ANSYS Inc\V162\ANSYS\ac4\bin\pro\winx64\ac4pro.exe
text_path E:\Program Files\ANSYS Inc\V162\ANSYS\ac4\data\pro\text
STARTUP dll
delay_start FALSE
allow_stop TRUE
unicode_encoding FALSE
unicode_encoding FALSE
```

```
revision 24.0  
end
```

---

### Note

When configuring the plug-in to run with Wildfire 5.0, the reference to `WBPlugInPECOM.dll` will instead be `DSPlugInPEWF5COM.dll` and `ac4pro.exe` should be `ac4pro_wf5.exe`.

<locale> should be "en-us" for English, "de" for German, "fr" for French, or "ja" for Japanese.

---

Do not delete any of these lines. If you modify this file, do NOT enter a carriage return after the END line. The file may be customized with other information. If these lines are deleted, or if the `WBPlugInPE.dat` file is not present in any of the directories in the search path, Creo Parametric will not load ANSYS-related CAD interfaces. You should typically never have to edit these files for path information contained within them. Paths are determined by environment variable settings, which are set automatically during installation. If you encounter problems when attempting to run Creo Parametric, use the **CAD Configuration Manager** to reconfigure rather than attempting to edit files directly.

Following an update of an existing Creo Parametric 2.0 or 3.0 installation to a different maintenance release (for example upgrading from datecode F000 to M010), the ANSYS Workbench Associative Plug-In will no longer be configured. To reactivate your plug-in, follow the steps described in [Using the CAD Configuration Manager \(p. 53\)](#). This situation does not apply to Pro/ENGINEER Wildfire 5.0, or Creo Elements/Pro 5.0.

In order for **CAD Configuration Manager** to successfully perform configuration of the Creo Parametric Associative Geometry Interface it is required that the Windows Operating System's decimal separator be the same symbol when running configuration as it was when the original Creo Parametric installation was performed.

### 8.1.5.1. Configuring the Connection for Creo Parametric

All Creo Parametric users must have copies of the `WBPlugInPE.dat` file and the `config.anscon.162` (connection for Creo Parametric) files. The `config.anscon.162` file is placed in the `Program Files\ANSYS Inc\V162\ansys\ac4\data\winx64` directory. `config.anscon.162` must be copied into the user's working directory at the time Creo Parametric is started. The `WBPlugInPE.dat` file is placed in the `Program Files\ANSYS Inc\V162\AISOL\CADIntegration\%ANSYS_PROEWF_VER%\ProEPages\config` directory. This file defines the name of the executable, the path to the executable, the path to the message file, and the current revision of Creo Parametric.

---

### Note

If the ANSGeom menu in Creo Parametric does not appear correctly, copy the `config.anscon.162` file into your working directory and restart Creo Parametric.

---

#### 8.1.5.1.1. The WBPlugInPE.dat File and config.pro File

Creo Parametric uses the `WBPlugInPE.dat` file to locate related executables such as the connection for Creo Parametric.

Mechanical APDL/ANSYS Workbench also requires a `config.pro` file.

- The `config.pro` file resides in `<path-to-proe>\text` for any Pro/ENGINEER Wildfire release and `<path-to-creo-parametric>\..\Common Files\<Creo Ship Code>\text` for Creo Parametric. The *Creo Ship Code* is a three digit number preceded by a single letter. The ship code is specified in the Creo Parametric installation log, which can be found in `<path-to-creo-parametric>\uninstall\instlog.txt` for version 1.0, or in `<path-to-creo-parametric>\bin\pim\xml\creo-base.xml` for version 2.0. For version 2.0, in the cases that the `crebase.xml` file is missing, the ship code can be determined by looking in the Windows Registry using `regedit`. That value is located in either `HKLM\Software\PTC\Creo Parametric\2.0\Shipcode` or `SOFTWARE\Wow6432Node\PTC\Creo Parametric\2.0\Shipcode`.

The `config.pro` file contains a `PROTKDAT` line that points to the `WBPlugInPE.dat` file. This file is generated either during the product install or by running the **CAD Configuration Manager** to configure the Mechanical APDL /ANSYS Workbench products for Creo Parametric.

The `<proe_platform>` variable is the name that Creo Parametric gives to its platform directories:

WBPlugInPECOM

Hardware Platform	ANSYS Platform <platform>	Creo Parametric Platform <proe_platform>
Win 7 and Win 8.1 x64	winx64	x86e_win64

### 8.1.5.1.2. The config.pro File

A typical `config.pro` file might look like the following example. This example has an entry for ANSYS Workbench. You may have other Creo Parametric specific customizations.

```
PROTKDAT E:\Program Files\ANSYS Inc\V162\AISOL\CADIntegration\${ANSYS_PROEWF_VER}\
ProEPages\config\WBPlugInPE.dat
```

### 8.1.5.1.3. The config.anscon.162 File

Users who launch Mechanical APDL from Creo Parametric will need the information from the `config.anscon.162` file. This file is installed for all users in the `\ac4\data\winx64` subdirectory. You typically do not need to edit this file. Here is a sample `config.anscon.162` file:

```
ANSYS_CMD %AWP_ROOT162%\ANSYS\bin\%ANSYS_SYSDIR%\ansys162.exe
ANSYS_GRAPHIC_DEVICE WIN32
ANSYS_SOLVER Sparse**
ANSYS_SELECTED_LAYERS 1-256**
ANSYS_GEOMETRY_TYPE Solids Only**
ANSYS_NEUTRAL_FORMAT Yes**
ANSYS_PRODUCT_NAME ANE3FL
```

\*\*These variables are not supported by Creo Parametric and are ignored by Creo Parametric.

See *Setting ANSYS Configuration Parameters* in the [Connection User's Guide](#) for more information about the `config.anscon.162` file.

### 8.1.5.2. Creo Parametric Environment Variables

Most environment variables are set during product installation. In general, you will not need to reset these variables. Check the manuals for the individual CAD products for information about environment variables and their settings.

#### PROE\_START\_CMD162

Specifies the name of the .bat file which launches Creo Parametric on the system. The path to the executable should be already set if the Creo Parametric installation path has been defined for the **PATH** environment setting.

**PROE\_START\_CMD162** Default = C:\Program Files\PTC\Creo 2.0\Parametric\bin\parametric.bat

### 8.1.6. NX Configuration

Running the **CAD Configuration Manager** for NX performs the following steps to activate the NX plug-in:

- If not already existing, sets the environment variable **UGII\_CUSTOM\_DIRECTORY\_FILE** to %APPDATA%\ANSYS\v162\Unigraphics\custom\_dirs.dat.
- If not already present, adds an entry to the custom\_dirs.dat file specifying the location of the Plug-In, e.g., C:\Program Files\ANSYS Inc\V162\AISOL\CADIntegration\UnigraphicsNX\winx64.
- Registers the Plug-In file (DSPlugInUGCOM.dll). For the previous example, these files would be located in C:\Program Files\ANSYS Inc\V162\AISOL\CADIntegration\UnigraphicsNX\winx64\startup.

Specific to Windows 8.1, when NX 10 is configured for the associative Workbench reader during the end of the install, the CAD Configuration Manager incorrectly shows an error that says "Configure Failed - Registration Error". However the configuration is still successful, and this error message can be ignored.

#### 8.1.6.1. Configuring the Connection for NX

**User-Defined Environment Variables** You will need to have the following environment variable set if you will be running the connection for NX product from inside NX.

#### UGII\_VENDOR\_DIR for Mechanical APDL

This environment variable is not set during the installation process and can only be set in a user's start-up file or at command level before running NX. This environment variable defines the ANSYS menu for NX. You must set this environment variable if you will be running the connection for NX product from inside NX. This environment variable tells NX where to find the ANSYS program, for example: C:\Program Files\ANSYS Inc\V162\ANSYS\ac4\bin\ug50\winx64.

#### UGII\_BASE\_DIR and UGII\_ROOT\_DIR

The environment variables **UGII\_BASE\_DIR** and **UGII\_ROOT\_DIR** must be defined for proper operation of NX and the ANSYS Workbench Plug-In to NX. These environment variables are typically created by the NX installer; however, in rare cases, you may have to create/update these two variables. In these situations, the **NX Installation Location** field will be left blank within the **CAD Configuration Manager's** NX tab.

When multiple NX versions are installed on a system, the **UGII\_BASE\_DIR** environment variable must be set to the install path of the NX version to be launched, prior to starting that version of the CAD. Otherwise errors will be encountered during CAD startup and the plug-in will not load.

For example, if switching from NX 8.5 to run NX 9.0, you must update **UGII\_BASE\_DIR** to the NX 9.0 path before starting NX 9.0. Similarly, the **UGII\_ROOT\_DIR** environment variable must be set to the `ugii` subdirectory in the install path of the NX version to be launched.

### 8.1.6.2. Configuring for Teamcenter

Running the **CAD Configuration Manager** for Teamcenter performs the following step to activate the Teamcenter plug-in:

- When Teamcenter Unified is detected, the environment variable **TC\_ROOT** is set to the Teamcenter Unified installation path.

## 8.2. Configuring CADNexus/CAPRI CAE Gateway for CATIA V5

The CADNexus/CAPRI CAE Gateway for CATIA V5 product is available for ANSYS Workbench. The ANSYS Workbench-specific portions are included in the installation. However, to run this product, you must complete the configuration as follows:

1. Install ANSYS Workbench according to the instructions in this guide.
2. Install CATIA V5 and LUM licensing (requires a CATIAV5 license key MD2, HD2 or ME2).
3. From the ANSYS, Inc. Customer Portal, download the CADNexus/CAPRI CAE Gateway for CATIA V5 for your platform to a temporary folder. (Do not download to a folder containing blank spaces in the folder name, e.g., Program Files). Follow the download procedures described in [Downloading the Installation Files \(p. 16\)](#).
4. Extract the contents of the zip file using your standard zip utility.
5. Install CADNexus/CAPRI CAE Gateway for CATIA V5 using the appropriate exe for your platform.
6. Follow the instructions on the CAPRI installation screens to complete the installation. When specifying the component to install, you will need to select the desired CatiaV5 release under CAPRI CAD Applications. When asked if CAPRI should set the environment variables, click **Yes**.
7. Run the **CAD Configuration Manager** and select the **Catia V5: CADNexus/CAPRI CAE Gateway** option to complete the configuration as described in [Using the CAD Configuration Manager \(p. 53\)](#).

---

### Note

Using the **CAD Configuration Manager** to set up the CADNexus/CAPRI CAE Gateway does not require administrative rights, although installing that product (step 5) requires these rights. However, if you want to reconfigure to use the ANSYS Workbench Reader for Catia V5, administrative rights are required.

---

If you want to revert to the standard CATIA V5 interface, run the **CAD Configuration Manager** and select the **Catia V5: Standard Interface** option to complete the configuration as described in [Using the CAD Configuration Manager \(p. 53\)](#).

You cannot run the standard CATIA V5 interface simultaneously with the CADNexus/CAPRI CAE Gateway for CATIA V5 interface.

## 8.3. Configuring AutoCAD

Running the CAD Configuration Manager for AutoCAD performs the following steps to active the plug-in:

1. Creates the folders %PROGRAMDATA%\AnsysWB.bundle and %PROGRAMDATA%\Autodesk\ApplicationPlugins\AnsysWB.bundle\Contents\v162\Winx64\AutoCAD
2. Copies the following folders from the Workbench installation into the bundle's AutoCAD sub-folders:
  - All files within <ANSYS 16.0 Install Path>\aiso\CADIntegration\DWG\winx64 are copied to %PROGRAMDATA%\ApplicationPlugins\AnsysWB.bundle\Contents\v162\winx64\AutoCAD\2013
  - All files within <ANSYS 16.0 Install Path>\aiso\CADIntegration\DWG\images are copied to %PROGRAMDATA%\Autodesk\ApplicationPlugins\AnsysWB.bundle\Contents\v162\winx64\AutoCAD\Resources
  - All folders within <ANSYS 16.0 Install Path>\aiso\CADIntegration\DWG\Language are copied to %PROGRAMDATA%\Autodesk\ApplicationPlugins\AnsysWB.bundle\Contents\v162\winx64\AutoCAD\Resources\Language
3. Finally a %PROGRAMDATA%\Autodesk\ApplicationPlugins\AnsysWB.bundle\PackageContents.XML file is created with the following contents:

```
<?xml version="1.0" encoding="utf-8"?>
<ApplicationPackage ProductCode="{6C6B872A-2ECF-4272-B7B7-1CA69B1EAD08}" HelpFile="./Contents/v162/Winx64/AutoCAD/Resources/cadAutoCAD.html" Icon="./Contents/v162/Winx64/AutoCAD/Resources/ansys70.bmp"
Description="ANSYS Workbench Geometry Interface 16.2" AppVersion="16.2" Name="AnsysWB162
" ProductType="Application" SchemaVersion="1.0" AutodeskProduct="AutoCAD" FriendlyVersion="15.0
" SupportedLocales="Enu" OnlineDocumentation="www.ansys.com
" Author="ANSYS, Inc." UpgradeCode="{12D6EF98-CEE6-431E-B699-DE9DE4F32B7E}">
  <CompanyDetails Name="ANSYS, Inc." Email=" " Url="www.ansys.com" Phone=" " />
  <RuntimeRequirements SeriesMax="R19.1" SeriesMin="R19.0" Platform="AutoCAD" OS="Win64" />
  <Components Description="64bit Component">
    <RuntimeRequirements SeriesMax="R19.1" SeriesMin="R19.0" Platform="AutoCAD" OS="Win64" />
    <ComponentEntry AppName="WB150PlugInDWG" ModuleNameWin64="./Contents/v162/Winx64/AutoCAD/2013/
WB150PlugInDWG.arx" AppDescription="Windows 64bit support" LoadOnCommandInvocation="True"
  LoadOnRequest="True">
      <Commands GroupName="ANSYS_Workbench_Geometry_Interface">
        <Command Local="WB150STARTWB" Global="WB150STARTWB" />
        <Command Local="WB150STARTASIM" Global="WB150STARTASIM" />
        <Command Local="WB150HELP" Global="WB150HELP" />
        <Command Local="WB150ABOUT" Global="WB150ABOUT" />
      </Commands>
    </ComponentEntry>
    <ComponentEntry AppName="AnsysWB150" ModuleName="./Contents/v162/Winx64/
AutoCAD/Resources/Language/
```

```
en-us/Workbench150.cuix" />
```

```
</Components>
```

```
</ApplicationPackage>
```





---

## Chapter 9: Uninstalling the Software

---

Use this process to uninstall any ANSYS, Inc. product that was installed using the Unified Installation process.

**Note to CFX Users** The provided uninstall tool cannot uninstall MPI services. If you have CFX products installed that you are uninstalling, you must stop the MPI services before continuing with the uninstall:

1. As an administrator, go to the **Run** window and type **services.msc**.
2. Search for "IBM Platform MPI Remote Launch" and if found, click **Stop**.
3. Search for "Intel(R) MPI Library Process Manager" and if found, click **Stop**.
4. Continue with the uninstallation procedure, below.

**Uninstall Procedure for All Users** All users should use the following procedure to uninstall ANSYS, Inc. products.

1. Close down all applications that are currently running.
2. From Windows 7, select **Start > All Programs > ANSYS 16.2 > Uninstall 16.2**. From Windows 8.1, right-click the **Start Screen > All apps > ANSYS 16.2 > Uninstall 16.2**. You must uninstall with the same or higher privileges than were used to install the product, and we strongly recommend uninstalling as the same user who installed the product originally.
3. Select the product(s) to be uninstalled and unconfigured and click **Uninstall Selected Item(s)**. Not all products and product components may be listed individually.
4. You will be asked to save all data and close all Windows applications before continuing. Click **OK**.
5. You will be asked if you are sure you want to uninstall the selected products. Click **Yes**.
6. A message appears indicating that the uninstallation is complete. Click **OK**.

---

### Note

After all the desired products are uninstalled, you are presented with a message stating that any files or directories that were not recognized by our uninstall program remain. These remaining files typically include such things as .udf files, user customized documentation, and various routines. If these files are no longer required, please remove them manually.

---

**CFD-Post** CFD-Post can be installed as a standalone product; however, as a standalone product, it is also automatically installed with Fluent, Polyflow, and Icepak. To uninstall CFD-Post, you must uninstall all of the above products that you have installed.

- If you choose to uninstall standalone CFD-Post without uninstalling the other products, the uninstall program will not uninstall CFD-Post.
- If you choose to uninstall all of the other products that include standalone CFD-Post but you do not select CFD-Post, then CFD-Post will not be removed.

Individual product directories may not be removed if they contain components that are shared with other products that have not been uninstalled. However, the uninstalled product will no longer run.

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

This procedure removes the specified ANSYS, Inc. program from your system but will not remove the ANSYS licensing components. Follow the steps in [Uninstalling Licensing Components \(p. 68\)](#) to uninstall the licensing components.

### 9.1. Uninstalling Licensing Components

Before proceeding with the following steps to remove the ANSYS licensing components from your system, make certain that there are no ANSYS, Inc. products on this machine or any other machines on the network that rely on these ANSYS licensing components.

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

1. Stop the ANSYS, Inc. License Manager via the **ANSYS License Management Center**.
2. Uninstall the ANSYS, Inc. License Manager service. You must use the following command to do so:

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

3. Delete the Licensing subdirectory.
4. Remove the **ANSYS, Inc. License Manager** folder from the **Start** menu.
5. Remove the **ANSYSLIC\_DIR** and the **ANSYSLIC\_SYSDIR** environment variables, if set.

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

1. Delete the Licensing subdirectory.
2. Remove the **ANSYS 16.2> ANSYS Client Licensing** folder from the **Start** menu or from the **Apps** screen.

---

## Chapter 10: Troubleshooting

---

### 10.1. Installation Troubleshooting

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

You can also find answers to commonly-asked questions on our customer portal. After you log in to the customer portal, select **Knowledge Resources**> **Solutions**.

For information on licensing-related errors, see the [Troubleshooting](#) section of the [ANSYS, Inc. Licensing Guide](#).

#### 10.1.1. Using ANSLIC\_ADMIN to Gather Diagnostic Information

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

The client-related diagnostic information can be gathered by using the Client ANSLIC\_ADMIN utility. For more information, see, [Gather Client Diagnostic Information](#).

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

You can also use the `AnsysInstallationDiagnostics.exe`, in the `\v162\instutil` directory, for additional installation diagnostics. This tool is helpful to diagnose installation and runtime issues. One or more log files will be generated either at the same location as the tool or in your temporary directory, depending on directory permissions. The log files are intended to help ANSYS, Inc. technical support personnel expedite any technical issues. It would be useful to have the files ready to provide before contacting Technical Support.

#### 10.1.2. Uninstall Gives Access Denied Error Message

If you run the uninstall utility as a non-administrative user, you may get access denied error messages when removing product components. Run the uninstall as an administrative user, and close all applications before running the uninstall.

#### 10.1.3. Uninstall on a Windows 7 or Windows 8.1 System Gives Compatibility Error Message

If you uninstall an ANSYS, Inc. product on a Windows 7 or Windows 8.1 system using the ANSYS, Inc. Uninstall utility, you may see the following error message:

**This program might not have installed correctly.**

**If this program didn't install correctly, try reinstalling using settings that are compatible with this version of Windows.**

This message is caused by the Windows Program Compatibility Assistant and can safely be ignored. Click **Cancel** and continue with the uninstall.

### **10.1.4. A .chm File Does Not Display Properly Across a Network**

Help system .chm files may not display topics correctly when you try to use a Universal Naming Convention (UNC) path to open the file on a network shared folder, typical with a Network Configuration install. This is a known issue with security update 896358, security update 840315, or Microsoft Windows Server 2003 Service Pack 1 (SP1). Please go to <http://support.microsoft.com/kb/896358> and <http://support.microsoft.com/kb/896054> for more information on this problem and Microsoft's recommended fix.

If you are running ICEM CFD over the network and you have applied this security update, you may not be able to access ICEM CFD Help using the Help menu. To access ICEM CFD Help, navigate to `common-files\help\en-us` in the installation directory on the server machine and copy the file `icem-cfd_help.chm` to your local machine. Double-click the local copy to view the help.

### **10.1.5. Products Crash with an Application Error**

On Windows Server 2008 systems, ANSYS applications may crash with an application error if the DEP (Data Execution Prevention) flag is set. To check your DEP setting, right mouse click on **My Computer**, and select **Properties**. Select the **Advanced** tab, and under **Performance**, click **Settings**. Then select the **Data Execution Prevention** tab.

If DEP is set, you will need to add ANSYS to the list of exceptions. For more information on DEP and how to change the settings, see <http://support.microsoft.com/kb/875351>.

### **10.1.6. Product Installation Does Not Create Start Menu Item for ANSYS and CAD Plugins Do Not Work**

On a Windows 7 or Windows 8.1 system, if you install ANSYS, Inc. products but do not see a Start Menu/Apps item and your CAD plugins do not work, the TEMP environment variable may be pointing to an inaccessible location. For example, it may point to a location such as `%USERPROFILE%\Local Settings\Temp`. As a result, the product installation could end without configuring the product correctly. You should ensure that the TEMP variable points to a valid location, such as `%USERPROFILE%\AppData\Local\Temp`.

### **10.1.7. Product Installation Displays Security Warning Dialogs or Silent Product Installation Hangs on a Network Install**

If you see a security warning dialog box during a GUI installation/product configuration, or if your installation hangs during a silent installation, you may need to modify your security settings. Set the machine you are installing from as a trusted site and allow launching of unsafe applications so that the prerequisite installation can complete. Then re-run the installation. Contact your system administrator or refer to your operating system documentation for details on modifying security settings.

Alternatively, you can run the prerequisite check to verify that you have all the necessary prerequisites. If necessary, install any prerequisites and rerun the installation:

1. Run `PreReqCheck.exe` from the top level directory.

2. Install any missing prerequisites by running `InstallPreReqs.exe` from the top level directory as an administrator.
3. When the prerequisites have finished installing, run the installation as you normally would (GUI or silently).

### 10.1.8. System-related Error Messages

**\*\*\*Error, ANSYS162\_DIR environment variable is not set. This is a fatal error – exiting.**

This message indicates that the **ANSYS162\_DIR** environment variable was not set where necessary for licensing. This environment variable should be set to the installation directory.

**Unexpected error: Request for the permission of type ... failed.**

If you encounter this type of message on a server configuration, you may need to set the machine as a trusted site. On a Windows 7 or Windows 8.1 machine, in the Control Panel, select **Network and Internet**, select **Internet Options**, and select **Security**. Click on **Trusted Sites** and click **Sites**. Add the machine as `file://servername`.

### 10.1.9. CasPol Error Messages

**ERROR: Not enough arguments, no permission set specified**

If you are configuring to a server with spaces in the path and get this error, then you must put quotes around your server specification as shown in the following example:

```
C:\Users\>C:\Windows\Microsoft.NET\Framework\v2.0.50727\CasPol.exe -m -a
g 1.2 -url file://"\\machineabc\ANSYS Inc\*" FullTrust
```

### 10.1.10. Configuring NX 10 on Windows 8.1 Displays an Error

Specific to Windows 8.1, when NX 10 is configured for the associative Workbench reader during the end of the install, the CAD Configuration Manager incorrectly shows an error that says "Configure Failed - Registration Error". However the configuration is still successful, and this error message can be ignored.

## 10.2. Installation Troubleshooting - Mechanical APDL and ANSYS Workbench Products

The items listed below apply only to the Mechanical APDL and/or ANSYS Workbench products.

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

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

### 10.2.2. Display Problems on Windows

This situation applies only to Windows systems running Mechanical APDL.

Windows includes a function called Aero that may cause Mechanical APDL graphing or plotting problems in 3-D mode on some systems. If this problem occurs, disable the Windows Aero theme in Windows 7

when running Mechanical APDL. If you are using Windows 8.1, please follow the steps below to correct this issue.

### Disabling Aero in Windows 8.1

1. Open the Windows Control Panel.
2. Click **Ease of Access**.
3. Click **Optimize Visual Display**.

The **Make the computer easier to see** window opens. Please note that you can access this window in a number of ways.

4. Enable the **Turn off all unnecessary animations (when possible)** option.
5. Click **Ok**.
6. Close the Windows Control Panel.

### 10.2.3. ANSYS\_ADMIN Error Messages

#### Grayed out options

If certain options are grayed out, your account may not have the necessary system administrator privileges necessary to run those options.

### 10.2.4. Mechanical APDL Product Launcher Error Messages

Some of the more common error messages follow. See the [ANSYS, Inc. Licensing Guide](#) for licensing-related launcher messages.

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

If you see this error, you cannot add or modify profile information during this launcher session. Verify that you have write access to the directory and restart the launcher session. Typically, this directory is C:\Documents and Settings\*user name*\Application Data\ANSYS\v162\launcher on Windows or ~/.ansys/v162/launcher/ on UNIX.

**\*\*\*No ANSYS product installations found.**

Verify your Mechanical APDL product installation. If the desired product was not installed, install it.

### 10.2.5. Distributed Mechanical APDL IBM Platform MPI Error Messages

You may encounter the following message when setting up IBM Platform MPI or running Distributed ANSYS using IBM Platform MPI.

**Error executing Mechanical APDL. Refer to System-related Error Messages in the Mechanical APDL online help. If this was a Distributed Mechanical APDL job, verify that your MPI software is installed correctly, check your environment settings, or check for an invalid command line option.**

You may see this error if you did not correctly run the set password bat file. Verify that you completed this item according to the IBM Platform MPI installation readme instructions.

You may also see this error if `ANSYS Inc\vs162\ansys\bin\<winx64>` is not in your PATH.

If you need more detailed debugging information, use the following:

1. Open a Command Prompt window and set the following:

```
SET ANS_SEE_RUN=TRUE
SET ANS_CMD_NODIAG=TRUE
```

2. Run the following command line: `ansys162 -b -dis -i myinput.inp -o myoutput.out`.

## 10.2.6. ANSYS Workbench Products Troubleshooting

This section lists problems and error messages that you may encounter while installing and/or running ANSYS Workbench. After each situation description or error message is the user action required to correct the problem. Always try the suggested user action before contacting your technical support representative.

### **Problem Situations**

**CAD System Plug-In Menus Do Not Appear for NX or Creo Parametric** ANSYS Workbench on Windows platforms will append its information to an existing customization file for NX and/or Creo Parametric. If no customization file exists, ANSYS Workbench will create a file. For NX, ANSYS Workbench looks for the `custom_dirs.dat` file in the directory specified via the **UGII\_CUSTOM\_DIRECTORY\_FILE** environment variable. For Creo Parametric, ANSYS Workbench looks for the `config.pro` file in the `%HOMEDRIVE%%HOMEPATH%` directory. In addition, during setup of the Creo Parametric Geometry Interface, ANSYS Workbench will also append its information to the `config.pro` file located in the Creo Parametric installation path, under the `\text` directory (e.g., `Proewildfire2\text\config.pro`).

If ANSYS Workbench encounters a read-only file, it will not be able to write the necessary information to the file. In this case, you will need to revise the permissions on the file and manually add the appropriate ANSYS Workbench-specific information in order for the ANSYS menu to appear in NX or Creo Parametric.

**Script Errors When Running ANSYS Workbench** If you encounter script errors such as "Error: Unable to create object microsoft.XMLDOM," you may need to install the latest version of Microsoft's MSXML. Please visit Microsoft's web site at <http://www.microsoft.com/downloads/details.aspx?FamilyID=993c0bcf-3bcf-4009-be21-27e85e1857b1&DisplayLang=en> for more information on downloading and installing MSXML.

### **ANSYS Workbench Error Messages**

#### **\*\*\*Unable to connect to Solver Manager.**

Another application might be using the Solver Manager port (10002 by default). Try changing the port number by editing the `Ansys.SolverManager.exe.config` file located in the installation directory at `\AISOL\Bin\<platform>`.

If you are getting the "Unable to connect to Solver Manager" error message or are having difficulty launching other applications/editors, it is also possible that the Windows hosts file has been corrupted.

Please make sure that localhost is specified in the Windows `<os drive>:\Windows\system32\drivers\etc\hosts` file.

## 10.3. Installation Troubleshooting - ANSYS CFX

The items listed below apply only to the ANSYS CFX products.

### 10.3.1. TurboGrid Mouse Behavior Problems

Depending on the graphics card and driver version, you may experience problems with the accuracy of mouse clicks in the 3D Viewer. For example, you may try to insert a control point at a given location by using the mouse, but the control point appears at a location far from where you clicked the mouse. If you experience such problems, try lowering the hardware acceleration setting of your graphics card.

## 10.4. Contacting Technical Support

Technical Support for ANSYS, Inc. products is provided either by ANSYS, Inc. directly or by one of our certified ANSYS Support Providers. Please check with the ANSYS Support Coordinator (ASC) at your company to determine who provides support for your company, or go to [www.ansys.com](http://www.ansys.com) and select **Contacts > Contacts and Locations**.

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

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

### NORTH AMERICA

#### All ANSYS Products except Esterel, Apache and Reaction Design products

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

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

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

### GERMANY

#### ANSYS Mechanical Products

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

**Email:** [support@cadfem.de](mailto:support@cadfem.de)

#### All ANSYS Products

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

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

German language: 0800 181 8499

English language: 0800 181 1565

Austria: 0800 297 835



Switzerland: 0800 546 318

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

German language: +49 6151 152 9981

English language: +49 6151 152 9982

**Email:** support-germany@ansys.com

## UNITED KINGDOM

### All ANSYS Products

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

**Telephone:** Please have your Customer or Contact ID ready.

UK: 0800 048 0462

Republic of Ireland: 1800 065 6642

Outside UK: +44 1235 420130

**Email:** support-uk@ansys.com

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

## JAPAN

### Mechanical Products

**Telephone:** +81-3-5324-8333

**Email:**

Mechanical: japan-ansys-support@ansys.com

### Fluids Products

**Telephone:** +81-3-5324-7305

**Email:**

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

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

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

### Icepak

**Telephone:** +81-3-5324-7444

**Email:** japan-icepak-support@ansys.com

### Licensing and Installation

**Email:** japan-license-support@ansys.com

## INDIA

### All ANSYS Products

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

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

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

## FRANCE

### All ANSYS Products

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

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

**Email:** support-france@ansys.com

## BELGIUM

### All ANSYS Products

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

**Telephone:** +32 (0) 10 45 28 61

**Email:** support-belgium@ansys.com

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

## SWEDEN

### All ANSYS Products

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

**Telephone:** +46 (0) 10 516 49 00

**Email:** support-sweden@ansys.com

## SPAIN and PORTUGAL

### All ANSYS Products

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

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

**Email:** support-spain@ansys.com, support-portugal@ansys.com

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

## ITALY

### All ANSYS Products

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

**Telephone:** +39 02 89013378

**Email:** support-italy@ansys.com

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

## Chapter 11: Applications Included with Each Product

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

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

<b>Product Install Option</b>	<b>What is Included?</b>
Catia, Version 4	Geometry Interface for Catia, Version 4
Catia, Version 5	Geometry Interface for Catia, Version 5
Catia, Version 6	Geometry Interface for Catia, Version 6
Creo Elements/Direct Modeling	Geometry Interface for Creo Elements/Direct Modeling
Inventor	Geometry Interface for Inventor
JTOpen	Geometry Interface for JTOpen
NX	Geometry Interface for NX
Parasolid	Geometry Interface for Parasolid
Creo Parametric (formerly Pro/ENGINEER)	Geometry Interface for Creo Parametric (formerly Pro/ENGINEER)
Solid Edge	Geometry Interface for Solid Edge
SolidWorks	Geometry Interface for SolidWorks

1. Workbench includes the Workbench Framework, DesignModeler, DesignXplorer, SpaceClaim Direct Modeler, SpaceClaim Catia V5 Interface and Remote Solve Manager.