ANSYS Electromagnetics Suite 17.0
Windows Installation Guide

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 17.0
December 2015

ANSYS Inc. is
certified to ISO
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 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.
Conventions Used in this Guide
Please take a moment to review how instructions and other useful information are presented in this guide.

- Procedures are presented as numbered lists. A single bullet indicates that the procedure has only one step.
- Bold type is used for the following:
  - Keyboard entries that should be typed in their entirety exactly as shown. For example, “copy file1” means the word copy must be typed, then a space must be typed, and then file1 must be typed.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by carats. For example, “click HFSS>Excitations>Assign>Wave Port.”
  - Labeled keys on the computer keyboard. For example, “Press Enter” means to press the key labeled Enter.
- Italic type is used for the following:
  - Emphasis.
  - The titles of publications.
  - Keyboard entries when a name or a variable must be typed in place of the words in italics. For example, “copy file name” the word copy must be typed, then a space must be typed, and then name of the file must be typed.
- The plus sign (+) is used between keyboard keys to indicate that you should press the keys at the same time. For example, “Press Shift+F1” means to press the Shift key and the F1 key at the same time.
- Toolbar buttons serve as shortcuts for executing commands. Toolbar buttons are displayed after the command they execute. For example,

“On the Draw menu, click Line” means that you can click the Draw Line toolbar button to execute the Line command.

Getting Help: ANSYS Technical Support
For information about ANSYS Technical Support, go to the ANSYS corporate Support website, http://www.ansys.com/Support. You can also contact your ANSYS account manager in order to obtain this information.

All ANSYS software files are ASCII text and can be sent conveniently by e-mail. When reporting difficulties, it is extremely helpful to include very specific information about what steps were taken or what stages the simulation reached, including software files as applicable. This allows more rapid and effective debugging.

Help Menu
To access online help from the menu bar, click Help and select from the menu:

<Product> Contents - click here to open the contents of the online help.
<Product> Search - click here to open the search function of the online help.
Context-Sensitive Help

To access online help from the user interface, do one of the following:

- To open a help topic about a specific menu command, press Shift+F1, and then click the command or toolbar icon.
- To open a help topic about a specific dialog box, open the dialog box, and then press F1.
# Table of Contents

## 1 - Getting Started

- Installation Prerequisites ...................................................... 1-1
  - ANSYS Electromagnetics Suite Software ................................. 1-1
  - License File ........................................................................ 1-1
  - License Server ..................................................................... 1-1
  - TCP/IP .............................................................................. 1-2
  - OpenGL ............................................................................. 1-2
  - Firewall Exceptions ............................................................. 1-3
  - Windows Requirements ......................................................... 1-3
  - Additional Product-Specific Prerequisites .............................. 1-3
  - Additional Resources ............................................................ 1-3

## 2 - Installing ANSYS Electromagnetics Suite Software

- Obtaining a License File .......................................................... 2-1
- Installing the Software .............................................................. 2-1
- Integrating ANSYS Electromagnetics Products with ANSYS Workbench ................................................................. 2-7
  - Integration with ANSYS Workbench during ANSYS Electromagnetics Product Installation ......................................................... 2-7
  - Integration with ANSYS after ANSYS Electromagnetics Product Installation ................................................................. 2-8

## 3 - High Performance Computing

- Options for Distributed Analysis ............................................... 3-1
- Remote Simulation Manager (RSM) ........................................... 3-2
  - When do you need RSM? ....................................................... 3-3
  - Installing RSM .................................................................... 3-4
- Registering ANSYS Electromagnetics Product Engines for use with RSM .............................................................. 3-6
- Supported Job Schedulers .......................................................... 3-6
Custom Scheduler Integration .......................................................... 3-7
Running HPC Test Jobs on a Cluster ............................................... 3-7
  Non-distributed job ................................................................. 3-8
  Non-MPI job distributed to multiple hosts .................................. 3-9
  MPI job distributed to multiple hosts ........................................ 3-9
Submitting and Running the Tests .................................................. 3-10
  Submitting and running a non-distributed job .............................. 3-11
  Submitting and running a non-MPI job distributed to multiple hosts 3-12
  Submitting and running an MPI job distributed to multiple hosts .... 3-14
Scheduler Specific Configuration and Testing .................................. 3-16
Windows HPC ............................................................................. 3-16
  Cluster Configuration Guidelines ............................................... 3-16
  Running Test Jobs ..................................................................... 3-17
  Non-distributed job ................................................................. 3-17
  Non-MPI job distributed to multiple hosts .................................. 3-18
  MPI job distributed to multiple hosts ........................................ 3-18
IBM Platform LSF ................................................................. 3-19
  Cluster Configuration Guidelines ............................................... 3-19
  Running Test Jobs ..................................................................... 3-19
  Non distributed job .................................................................... 3-19
  Non-MPI job distributed to multiple hosts .................................. 3-19
  MPI job distributed to multiple hosts ........................................ 3-20
4 - Install Common Libraries ......................................................... 4-1
  Procedure for Installing Libraries .............................................. 4-1
5 - Updating Software and Licenses ................................................. 5-1
  Using Maintenance Mode ......................................................... 5-1
  Updating a Software License .................................................... 5-2
6 - Message Passing Interface (MPI) Installation .............................. 6-1
Windows ................................................................. 6-1
Install Intel MPI .......................................................... 6-1
Install IBM Platform MPI .............................................. 6-3
General Notes ............................................................ 6-9
Scenario 1 ................................................................ 6-10
Scenario 2 ................................................................ 6-10
Scenario 3 ................................................................ 6-10
Windows HPC Scheduler .............................................. 6-11
Network Installation ...................................................... 6-12
Troubleshooting Section ................................................ 6-13

A - Silent Installation .................................................... A-1
Running the Silent Installation ...................................... A-1

B - License hostid ......................................................... B-1
Network Interface Card ................................................ B-1
Windows ................................................................ B-1
Linux ...................................................................... B-2
Hardware Key ............................................................ B-2
Installing a Hardware Key on a USB Port .................... B-2
1 - Getting Started

Installation Prerequisites

Before installing ANSYS Electromagnetics Suite software, you need the following:

- The ANSYS Electromagnetics product CD or product download (see "Additional Resources" on page 1-3 for download information).
- A license file based on the ansoftd vendor daemon.
- The ANSYS License Manager software download.
- A computer with TCP/IP installed, for counted evaluation and purchased licenses.
- A computer that meets the minimum or recommended system requirements, which depends on the specific ANSYS Electromagnetics products you plan to run.

ANSYS Electromagnetics Suite Software

The ANSYS Electromagnetics product or equivalent download contains:

- The ANSYS Electromagnetics Suite software and project examples.
- Third-party software, when necessary, for viewing online documentation (for example, HTML Help or Adobe Acrobat Reader).
- Documentation files.

License File

Most ANSYS Electromagnetics products require a license, which is sent as an e-mail attachment. After you receive a license file, save it to a temporary location.

The license file requires a license hostid, which is a hardware device used to validate the license. A list of supported hostids, and a general explanation of hostids, is available in "License hostid" on page B-1

If you do not have a license file for the ANSYS Electromagnetics software you are about to install, please contact your ANSYS sales representative.

License Server

Once you have purchased the software, you must designate a machine on your network to act as a license server. The ANSYS License Manager must be installed in one of the following two locations:
On the computer where the software will be installed.

On another machine accessible over the network.

**Note** You can install the ANSYS License Manager before or after you install the ANSYS Electromagnetics software.

See the ANSYS License Manager instructions available as part of the ANSYS License Manager download.

**TCP/IP**

**Transmission Control Protocol/Internet Protocol (TCP/IP)** is required for any ANSYS Electromagnetics software—even for a single computer in stand-alone mode.

- TCP/IP must be installed on the machine that is running ANSYS License Manager (the server) and on each machine that obtains licenses from that server.
- By default, TCP/IP is usually already installed on a computer running a Windows platform.
- TCP/IP is usually bound to a network card. If a network card is already installed and set up for TCP/IP, no additional action is needed during installation.
- If a network card is not installed, then you can satisfy the TCP/IP requirement by installing the Microsoft Loopback Adapter (MLA). The MLA is a software module that emulates a network card. For more details, see the appropriate help topics in Microsoft Windows (on the Windows taskbar, click **Start>Help** or **Start>Help and Support**, depending on the operating system).

**OpenGL**

All ANSYS Electromagnetics software (excluding ANSYS License Manager) requires that the OpenGL libraries be accessible from the machine running ANSYS Electromagnetics software.

Most graphics card drivers automatically come with OpenGL software for Windows.

**Graphics Card and Performance**

**Note** For some graphics cards, there may be slowdown for Field animations, Rotating and zooming for complex models, and selection display. This includes

- NVidia Quadro 2000M
- NVidia Quadro FX 3800M
- NVidia Quadro 4000
- NVidia Quadro FX 1800
- The problem does not show on the following graphic adapters:
  - ATI FirePro V5800
  - Quadro NVS 295
Quadro NVS 290

To correct the problem, set the **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Global Settings** > **Global Presets: Workstation App - Dynamic Streaming**

**Firewall Exceptions**

Firewall exceptions are needed for the COM engines and ANSYS Electromagnetics products if solving remotely (something that does not necessarily require RSM). Exceptions are also required for RSM as well if it is installed. This comes up most frequently with the Windows firewall.

**Windows Requirements**

For supported platforms and system requirements, go to:


**Additional Product-Specific Prerequisites**

The readme file (readme.txt) may show additional product-specific prerequisites, including system requirements. You can find the readme file in the AnsysEM directory of the installation program or in the Help directory after installing the software.

**Additional Resources**

**General:** The ANSYS web site: [http://www.ansys.com](http://www.ansys.com).

**Downloads:** Software downloads (including ANSYS License Manager) and installation-related documentation are available at the ANSYS customer portal, [http://www.ansys.com/customerportal](http://www.ansys.com/customerportal)

**Online Technical Support:** Available through the ANSYS customer portal:

[http://www.ansys.com/customerportal](http://www.ansys.com/customerportal)

Under **Online Support**, choose **ANSYS Electromagnetics Customer Support**.

The ANSYS Electromagnetics technical support site provides access to frequently asked questions, application notes, presentations, and example scripts.

**Readme File:** Before installation, you can read the readme file (readme.txt) in the AnsysEM
directory of the installation program. After installation, the readme file is available in the Help directory.

**Product Documentation:**

Online help files are accessible from the Help menu of each application installed, and they are located in the product subdirectories under the Help directory. The online help system includes two formats: Microsoft HTML Help and Portable Document Format (PDF).

*Getting Started Guides* and other documentation may be available as PDFs in the product subdirectories under the Help directory.
2 - Installing ANSYS Electromagnetics Suite Software

To install ANSYS Electromagnetics Suite software on a Windows operating system:

- Obtain a license file for the software you are installing. See "License File" on page 1-1 for more information.
- Install the ANSYS Electromagnetics Suite software.
- Set up a license server if prompted to do so.

Obtaining a License File

ANSYS Electromagnetics requires a license for all software. If you do not have a license file for the ANSYS Electromagnetics software you are about to install, contact your ANSYS sales representative. See "License File" on page 1-1 for more information.

Installing the Software

To install ANSYS Electromagnetics software:

1. Log in as Administrator, or log on to an account with administrator privileges.
2. If you downloaded the ANSYS Electromagnetics product ("Additional Resources" on page 1-3 ), unzip the download, and run the autorun.exe file in the newly created directory. The Installation Shell appears.
3. Click Install Electromagnetics Suite.
   The initial screen appears.
4. Click Next.
   The License Agreement screen appears.
5. Read the license agreement carefully. Click Yes if you accept the terms of the agreement.
6. Click Next.
   The Choose Destination Location screen appears.
7. Under Destination Folder, do one of the following to accept the default path or specify a new one:
• Accept the default installation directory listed under Destination Folder.

• To select a different directory, click Browse. The Choose Folder window appears. Select a directory, and click OK. If you have other ANSYS Electromagnetics software, ANSYS recommends that you install new software in the same directory.

• To create a new directory, click Browse. The Choose Folder window appears. Enter the new name in the Path field, and click OK.

8. Click Next.

Depending on the software and version being installed, you may see one or both of the following screens before the Select Program Folder screen: the Choose Default Temporary Directory Location screen and/or the Choose Library Option screen.

9. If the Default Temporary Directory Location screen appears, select a default location for all simulations (including local) to use as a temporary work space.

• If desired, change the temporary folder assignment. You can edit the field directly or use the Browse button to display file browser to let you navigate the folders on your machine and network.

• If desired, check the option to add read/write permissions for Everyone. If the installation will be used for remote simulation by multiple users, all users must have permissions on the temporary folder.

10. If the Choose Library Option screen appears, do one of the following to specify the location of ANSYS Electromagnetics libraries:

   Note  ANSYS Electromagnetics libraries are large databases of electronic components, developed by ANSYS and various manufacturers.

• Click This local installation to copy and install the default ANSYS Electromagnetics component libraries, and then click Next.

• Click The following common location to use libraries that are already installed (locally or on the network). Click Browse, select the appropriate path, and click
OK. Click Next.

For the Install Libraries option, see the "Install Common Libraries" on page 4-1.

11. Click Next.

The Integration with ANSYS screen may appear.

If ANSYS is installed on the machine, you have the choice of integrating "Integrating ANSYS Electromagnetics Products with ANSYS Workbench" on page 2-7 or "Integration with ANSYS after ANSYS Electromagnetics Product Installation" on page 2-8. If ANSYS is not installed, the choice of integrating now is disabled.

12. Click Next.

The License Information screen appears.
For additional information about the three license options, click **Help me decide which of the following options to choose**. This displays a browser page with additional explanations.

**Note** You can review the current license configuration by clicking **View current license configuration**.
13. Select one of the following three license options:

<table>
<thead>
<tr>
<th>I have a new license file</th>
<th>I want to specify a license server</th>
<th>I want to keep the current license configuration</th>
</tr>
</thead>
<tbody>
<tr>
<td>Select this if you recently received a license file from ANSYS (either evaluation or purchased). a. Click Next. Another License Information window appears, asking you to specify a directory and license file. b. Click Browse. The Select License File window appears. c. Find and select the license file you received from ANSYS Electromagnetics. Click Open, and then click Next. d. If prompted, enter the name of the license server. You may also select or clear the Specify server TCP/IP port number (advanced) option. Specifying a TCP/IP port means that the software looks only at that port for licenses. The default port for the ANSYS License Manager is 1055. e. Click Next.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Select this if you are using a counted license and already know on which machine the ANSYS License Manager will be installed. a. Click Next. Another License Information window appears, asking if you want to use a single server or a redundant server. b. Select or clear the Specify redundant servers check box, and enter the server name(s) in the field(s). The redundant-server setup requires three license servers, each serving an identical license file. The order in which the servers are entered must match the license files on each server. c. Select or clear the Specify server TCP/IP port number (advanced) option. For redundant servers you must enter a port. Specifying a TCP/IP port means that the software only looks at that port for licenses. The default port for the ANSYS License Manager is 1055. d. Click Next.</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Select this if you already have a license file for previous ANSYS software and want to keep the same configuration.</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

14. On the Select Program Folder screen, do one of the following to specify a name for the program folder:
   - Accept the default name displayed in the Program Folder box.
   - Select an item from the Existing Folders list.
• Type a new name for the folder in the Program Folder box.
  
  **Note** The program folder, which appears on the Windows Start menu, is a grouping of related software that includes executable files and other components. The default programs folder is ANSYS Electromagnetics (Start>All Programs>ANSYS Electromagnetics).

15. Click **Next**.
   
The **Review settings before copying files** screen appears.

16. In the **Review settings before copying files** screen, review the information displayed under Current Settings, and then do one of the following:
   
   • If the settings are correct, click **Next** to start copying files.
   
   • If the settings are incorrect, click **Back** as many times as necessary, and change your selections. Verify each screen before clicking **Next** to go to the next screen. When all information is correct, return to the **Review settings before copying files** screen, and click **Next**.

The Setup Status screen appears, displaying a progress bar, and installation begins. You can click **Cancel** at any time to exit the installation.

  **Note** If you receive a message about read-only files, you typically want to overwrite them with the newer versions.

17. Once the files have all been copied, one of the following messages may appear:
   
   • If you identified as the license server the same machine where you installed the ANSYS Electromagnetics software, then a message appears that informs you that you must install the ANSYS License Manager on this machine. Click **OK**.
   
   • If you identified as the license server a machine other than where you installed the ANSYS Electromagnetics software, then a message appears that identifies that machine as the license server and informs you that you must install the ANSYS License Manager on that machine. Click **OK**.

18. When the installation is complete, the InstallShield Wizard Complete window appears. You will see one of two check box options:
   
   • If RSM is not installed, you will see a check box to launch the RSM installation and, once the installation is complete, register this ANSYS Electromagnetics product with RSM. If you select this option and click **Finish**, the product installation ends and the RSM installation is automatically started.
   
   • If RSM is installed, you will see a check box to register this ANSYS Electromagnetics product with RSM.
   
   See chapter 3 for more details on RSM.

19. Click **Finish**.
   
The initial setup Installation Shell reappears.
Integrating ANSYS Electromagnetics Products with ANSYS Workbench

You can integrate ANSYS Electromagnetics products with ANSYS Workbench in one of two ways:

- "Integration with ANSYS Workbench during ANSYS Electromagnetics Product Installation" below, or
- "Integration with ANSYS after ANSYS Electromagnetics Product Installation" on the next page

Integration with ANSYS Workbench during ANSYS Electromagnetics Product Installation

You can choose to integrate ANSYS Electromagnetics products with ANSYS Workbench during installation as follows:

1. If you have not already installed ANSYS, do so before proceeding. Refer to the Installation Guide for ANSYS Electromagnetics Suite.
2. Proceed through the installation dialogs.
3. When the Integration with ANSYS dialog displays, select the Yes radio button to have the installer automatically integrate the ANSYS Electromagnetics product with ANSYS.

Note If you choose not to integrate the product with ANSYS during installation, you can perform this step "Integration with ANSYS after ANSYS Electromagnetics Product Installation" on the next page

4. Complete the product installation and exit the installer.
Integration with ANSYS after ANSYS Electromagnetics Product Installation

If you want to integrate ANSYS Electromagnetics Suite with ANSYS after installation, do the following:

1. From the Start menu, select All Programs>ANSYS Electromagnetics>ANSYS Electromagnetics Suite 17.0 >Modify Integration with ANSYS.

2. In the Integration with ANSYS dialog box, ensure that the Yes radio button is selected,

Then click OK to complete the integration process. Then OK the information dialogue.
You can confirm that ANSYS is “aware” of the ANSYS Electromagnetics integrated applications via the ANSYS Workbench Options dialog, which shows the paths to the integrated ANSYS Electromagnetics applications.
3 - High Performance Computing

ANSYS Electromagnetics products have a variety of ways to use multiple threads, multiple cores or multiple hosts to improve productivity. These additional computational resources may be used to solve larger problems, to solve a given problem faster, or to improve the quality of a solution (using more stringent convergence criteria, for example) in the same amount of time. This section describes the configuration and requirements for using additional resources by ANSYS Electromagnetics products.

To make effective use of the resources allocated to the analysis, the job settings must match the job requirements. In addition, the cluster configuration and the resources must be allocated to the job in a way that the products are able to make effective use of the resources.

Options for Distributed Analysis

There are several options for configuring ANSYS Electromagnetics products to take advantage of additional computational resources. The simplest approach is to use the Remote Simulation Manager (RSM) for ANSYS Electromagnetics products. RSM may be installed, configured, and started on one or more hosts. It runs as a Windows Service. An analysis may be distributed to any of the hosts where the Remote Simulation Manager is running. RSM is bundled with ANSYS Electromagnetics products.

There are a number of job schedulers available from third parties, such as IBM Platform LSF. A job scheduler may also be called a distributed resource management system, a cluster manager, a batch system or a distributed resource manager. These systems provide queuing of jobs, load balancing, assigning resources to jobs, and many other management and monitoring features. ANSYS Electromagnetics products integrate with several of the most widely used job schedulers. The user is able to submit jobs to the scheduler. When the job runs, the product uses the resources allocated to the job to accomplish the analysis.

Users may be able to use job schedulers which are not included in the list of supported job schedulers via “custom scheduler integration.” The end user, working with ANSYS Electromagnetics products support engineers, will provide an IronPython script or code library to act as an interface between the scheduler and the ANSYS Electromagnetics products.

The RSM for ANSYS Electromagnetics products may be used for running an analysis on a remote host or for running an analysis distributed to multiple hosts. However, the user is required to manage the jobs and resources used by these jobs; no ability to queue jobs until needed resources are available is provided. In general, a job scheduler is more appropriate if there are a number of hosts shared between a number of users, because a scheduler is better able to manage the jobs and the resources. It can queue a job and delay the start the job until appropriate resources become available. Custom scheduler integration is needed to allow jobs to be run on a cluster that is not managed by one of the supported job schedulers.
Remote Simulation Manager (RSM)

The Remote Simulation Manager (RSM) for ANSYS Electromagnetics products may be used for running an analysis on a remote host or for running an analysis distributed to multiple hosts. When using RSM, the users are required to manage the jobs and resources used by these jobs. No load balancing or queueing of jobs is provided. The Remote Simulation Manager is not used if the analysis is run as a job on one of the supported job schedulers, such as LSF or Windows HPC. See "Supported Job Schedulers" on page 3-6 for a list of these job schedulers. RSM is also not used if the analysis is run as a job using custom scheduler integration with a scheduler that is not supported by default. See "Custom Scheduler Integration" on page 3-7 for more information.

If the Remote Simulation Manager is running on a host, by default, jobs started on remote hosts may distribute jobs to the local host. A user may disable this access or re-enable this access using the Options dialog. The **Tools>Options>General Options** menu item pops up the Options dialog:
The **Disable access by remote machines** check box, in the **RSM Service Options** group box, on the **General>Remote Analysis** page controls access. If this check box is not checked, the default setting, then remote hosts can distribute jobs to the local host. If this check box is checked, then remote hosts cannot distribute jobs to the local host.

---

**When do you need RSM?**

RSM is needed only to run an analysis on a remote host or an analysis distributed to multiple hosts. RSM is also required if some of the solvers are running on a hosts that run under a different OS than the OS host that is running ANSYS Electronics Desktop. If RSM is required, then the RSM daemon or service must be configured and running on all remote hosts for the analysis.

Please note that RSM is **not** needed:

- To run an analysis only on the local host.
- To run an analysis using integration with a supported job scheduler or using custom scheduler integration.

You should not install RSM on hosts that are managed by a scheduler.
Installing RSM

To install ANSYS Electromagnetics RSM software:

1. If you checked to install the Remote Simulation Manager during the software installation, the ANSYS Electromagnetics Remote Simulation Manager installer appears. You can also install the Remote Simulation Manager as an installation option on the installation CD. The following screen appears.

   ![Remote Simulation Manager Installer Screen]

   **ANSYS Electromagnetics Remote Simulation Manager: Windows 64-bit Edition**

   This wizard will install ANSYS Electromagnetics Remote Simulation Manager on your computer.

   It is strongly recommended that you exit all Windows programs before running this setup program.

2. To install the ANSYS Electromagnetics Remote Simulation Manager, click **Next**.
   The **License Agreement** screen appears.

3. Read the license agreement carefully. Click **Yes** if you accept the terms of the agreement.
   The **Choose Destination Location** screen appears.

4. Under **Destination Folder**, do one of the following to accept the default path or specify a new one:
   - Accept the default installation directory listed under **Destination Folder**.
To select a different directory, click **Browse**. The Choose Folder window appears. Select a directory, and click **OK**. If you have other ANSYS Electromagnetics software, ANSYS recommends that you install new software in the same directory.

To create a new directory, click **Browse**. The Choose Folder window appears. Enter the new name in the **Path** field, and click **OK**.

5. Click **Next**.
   
The Remote Simulation Manager Service Configuration window appears.

6. Choose to run the ANSYS Electromagnetics Communication service as the **System User** or as a **specific user** (who must have administrative privileges).
   
If you select a **specific user**, the fields for User name, Password, and Machine or domain name are enabled.

**Note** The user selected will run remote and distributed analyses when no user is specified in the product interface.

7. On the **Select Program Folder** screen, do one of the following to specify a name for the program folder:
   
   - Accept the default name displayed in the **Program Folders** box.
   - Select an item from the **Existing Folders** list.
   - Type a new name for the folder in the **Program Folders** box.

   **Note** The program folder, which appears on the Windows **Start** menu, is a grouping of related software that includes executable files and other components. The default programs folder is (**Start>**All Programs>**ANSYS Electromagnetics**).

8. Click **Next**.

   The **Review settings before copying files** screen appears.

9. In the **Review settings before copying files** screen, review the information displayed under **Current Settings**, and then do one of the following:
   
   - If the settings are correct, click **Next** to start copying files.
   - If the settings are incorrect, click **Back** as many times as necessary, and change your selections. Verify each screen before clicking **Next** to go to the next screen. When all information is correct, return to the **Review settings before copying files** screen, and click **Next**.

   The **Setup Status** screen appears, displaying a progress bar, and installation begins. You can click **Cancel** at any time to exit the installation.

   **Note** If you receive a message about read-only files, you typically want to overwrite them with the newer versions.

At the end of the installation, RSM is automatically started as a service on this machine. The InstallShield Wizard complete window appears. If you want to view the README file (recommended before using the software), check the box.
10. Click **Finish** to close the install window.

   If you use RSM, it must be installed on all remote machines. In addition, the product engines must be registered on RSM.
   
   - If this installation was automatically launched from a product installation, then the registration will occur automatically.
   
   - To register the engines at any time on Windows, choose **Start> All Programs> ANSYS Electromagnetics> product> Register with RSM**. You can also run **RegisterEnginesWithRSM.exe**, located in the product subdirectory (for example, C:\Program Files\AnsysEM\AnsysEM17.0\Win64\RegisterEnginesWithRSM.exe). In each case, you see a dialog confirming the registration. **OK** the dialog.

**Registering ANSYS Electromagnetics Product Engines for use with RSM**

You must make each installation of RSM aware of each ANSYS Electromagnetics product, so that remote simulation requests are properly routed to the product. For network installations, you need only to register each ANSYS Electromagnetics product once, regardless of the number of machines. Otherwise, you need to register each ANSYS Electromagnetics product on each machine running RSM.

- To register an ANSYS Electromagnetics product with RSM, either:
  
  Click **Start> All Programs> ANSYS Electromagnetics Suite 17.0> Register with RSM**.
  
  Or run **RegisterEnginesWithRSM.exe**, located in the product installation directory (for example, c:\Program Files\AnsysEM\AnsysEM17.0\Win64\RegisterEnginesWithRSM.exe).

- To unregister an ANSYS Electromagnetics product from RSM, either:
  
  Click **Start> All Programs> ANSYS Electromagnetics Suite 17.0> Unregister with RSM**.
  
  Or run **UnregisterEnginesWithRSM.exe**, located in the product installation directory (for example, c:\Program Files\AnsysEM\AnsysEM17.0\Win64\UnregisterEnginesWithRSM.exe).

**Supported JobSchedulers**

The supported Windows job schedulers of ANSYS Electromagnetics Suite 17.0 are listed in the following table.

<table>
<thead>
<tr>
<th>Job Scheduler (DRM)</th>
<th>Latest Tested Version</th>
</tr>
</thead>
<tbody>
<tr>
<td>Windows HPC</td>
<td>Windows HPC Server 2012 R2</td>
</tr>
</tbody>
</table>
The versions earlier than the latest tested version indicated in the table may also work fine and might have been tested in previous versions of the product. Newer versions may also work if backward compatibility is maintained by the scheduler.

**Custom Scheduler Integration**

For the above job schedulers in a standard configuration, ANSYS Electromagnetics products provide an out-of-the-box integrated solution. For integration with other schedulers or non-standard scheduler configurations, user (or scheduler vendor) needs to provide a component called "custom scheduler proxy." Using this small component, AnsysEM mainly identifies the compute nodes allocated by a scheduler to the AnsysEM job, and launches solver processes on those nodes.

A custom scheduler proxy is usually an IronPython script file that has a class that implements a specified interface. Functionally, the methods of this class mediate the interactions between AnsysEM processes and the scheduler. Physically, such a proxy is a small text file, and does not require to be compiled/linked. This is often the preferred approach, as it is easier to install and test. The details of this interface and its methods are described elsewhere. Installing a custom scheduler proxy usually means just adding a .py file under schedulers subdirectory of the AnsysEM installation directory. By providing a custom scheduler proxy written in IronPython, you can even override the default scheduler proxy for a supported scheduler.

**Note:** IronPython is an implementation of the Python programming language targeting the .NET runtime. You do not need to install IronPython, as AnsysEM already comes with IronPython. AnsysEM supports it on both Windows and Linux. For more information, see the "Introduction to IronPython" section.

Alternately, a custom scheduler proxy can be a dynamically linked library (on Microsoft Windows) that implements a set of functions. This library is loaded by the AnsysEM at runtime, and if it is running as part of a scheduler job, the AnsysEM interacts with the library to get information from the scheduler, and to start additional processes on specified hosts. Each such library implements a set of extern "C" functions needed to mediate the interactions between AnsysEM and the scheduler. The details of these functions are described elsewhere.

**Limitations**

Implementing your own custom scheduler proxy does not allow use of the job management GUI. It also does not provide tight-integration of scheduler for MPI-based jobs.

**Running HPC Test Jobs on a Cluster**

This section describes how to submit and run a small number of test jobs to verify that the product installation, the cluster configuration, and the MPI installation and configuration are satisfactory for running ANSYS Electromagnetics analysis jobs. These tests are not comprehensive, but successful completion of all of the jobs indicates that the basic requirements for running ANSYS Elec-
Electromagnetics jobs have been met. Before running these tests, the following steps should be completed:

- The ANSYS Electromagnetics product or products should be installed
- The cluster should be configured for running ANSYS Electromagnetics analysis jobs, as described in "Scheduler Specific Configuration and Testing" on page 3-16
- MPI should be installed and configured as described in "Message Passing Interface (MPI) Installation" on page 6-1

Each of the test jobs should run to completion in a few minutes or less. They should be run in the order specified, proceeding to the next test only if any problems found in the preceding test have been addressed. Three different test jobs are described.

- **Non-distributed job**
- **Non-MPI job distributed to multiple hosts**
- **MPI job distributed to multiple hosts**

### Non-distributed job

This is a simple job that uses one core on one host. This test uses the Tee.aedt example project, which is located in the "Examples/HFSS/RF Microwave" subdirectory of the product installation directory. The files Tee.PDF and Tee.aedt are needed for this test. This test uses Setup1, but it does not use the frequency sweep, Sweep1, so the frequency sweep must be disabled.

Use the following steps to copy the project files and disable the frequency sweep, if needed.

1. Copy the files Tee.aedt and Tee.PDF from the Examples/HFSS/RF Microwave subdirectory of the product installation directory to another directory.
2. Run the product (ansysedt.exe).
3. Open the copy of the project.
4. In the Project Manager window, expand the tree item TeeModel (Driven Model)>Analysis>Setup1>Sweep1. If frequency sweep Sweep1 is enabled, right-click on TeeModel (Driven Model>Analysis>Setup1> Sweep1, and select the item **Disable Sweep** in the popup menu to disable the frequency sweep.
5. Save and close the project.

Successful completion of this job indicates that the following requirements have been met:

- The job submission host is correctly configured for submitting jobs to the cluster.
- The job user account is authorized to run jobs on the cluster.
- The product installation is accessible from the cluster host allocated to the job.
- The project directory is accessible from the cluster host allocated to the job.
- Communication between different job processes on the cluster host allocated to the job is not blocked.
Non-MPI job distributed to multiple hosts

This job uses multiple cores distributed to multiple hosts. It does not use MPI. This job will start on one host, but portions of the analysis will be distributed to the other hosts allocated to the job. This test also uses the Tee.aedt example project, which is located in the Examples/HFSS/RF Microwave subdirectory of the product installation directory. The files Tee.PDF and Tee.aedt are needed for this test, the same as the previous test. This test uses Setup1, and it uses the frequency sweep, Sweep1, so the frequency sweep must be enabled.

Use the following steps to copy the project files and enable the frequency sweep, if needed.

1. Run the product (ansysedt.exe).
2. Open the copy of the project.
3. In the Project Manager window, expand the tree item TeeModel (Driven Model)>Analysis>Setup1>Sweep1. If frequency sweep Sweep1 is disabled, right click on TeeModel (Driven Model)>Analysis>Setup1>Sweep1, and select the item Enable Sweep in the popup menu to enable the frequency sweep.
4. Save and close the project.

Successful completion of this job indicates that the following additional requirements have been met:

- A job process on one cluster host allocated to the job is able to use the cluster to start additional job processes on other cluster hosts allocated to the job.
- Communications between job processes running on different cluster hosts allocated to the job are not blocked.

MPI job distributed to multiple hosts

This job uses MPI processes distributed to multiple hosts. This job will start on one host, but portions of the analysis will be distributed to MPI ranks running on the first execution host and ranks running on other hosts allocated to the job. This test uses the differential_stripline.aedt example project, which is located in the Examples/HFSS/Transmission Lines subdirectory of the product installation directory. The files differential_stripline.pdf and differential_stripline.aedt are needed for this test. This test uses Setup1, but it does not use the frequency sweep, DC_10GHz, so the frequency sweep must be disabled.

Use the following steps to copy the project files and disable the frequency sweep, if needed.

1. Copy the files differential_stripline.aedt and differential_stripline.pdf from the “Examples/HFSS/Transmission Lines” subdirectory of the product installation directory to another directory.
2. Run the product (ansysedt.exe).
3. Open the copy of the project. In the Project Manager window, expand the tree item differential_stripline -> Differential Stripline (Driven Terminal)>Analysis>GHz>DC_10GHz. If frequency sweep DC_10GHz is enabled, right-click on differential_stripline> Differential
If Stripline (Driven Terminal)>Analysis>GHz>DC_10GHz, and select the item **Disable Sweep** in the popup menu to disable the frequency sweep.

4. Save and close the project.

Successful completion of this job indicates that the following additional requirements have been met:

- If needed, MPI is installed and configured on the cluster hosts.
- If needed, user passwords are registered with MPI on the cluster hosts.
- If needed, the MPI prerequisites are installed on the cluster hosts.

### Submitting and Running the Tests

This section includes general instructions for submitting and running the jobs. For additional notes on running any of the test jobs on a specific cluster, see the information for the specific scheduler in "Scheduler Specific Configuration and Testing" on page 3-16

The test jobs may be submitted to a cluster using the Ansys Electromagnetics job submission GUI. To use this GUI, start the product, ansysedt.exe, on a host which is configured as a submission host for the cluster. The product should be started in an environment configured for submitting jobs to the cluster.

Before submitting the first job, the scheduler must be selected. Use menu item **Tools>Job Management>Select Scheduler** to pop up the **Select Scheduler** dialog.

For a Windows HPC cluster, select the **Use this computer** radio button, and then select **Windows HPC** in the **Choose scheduler** drop-down list. Enter the name of the head node in the **Head node** edit control. The “...” button may be used to access a dialog to browse computers accessible via a network.

For other clusters, select the **Use this computer** radio button, and then select the scheduler type in the **Choose scheduler** drop-down list. Select lsf for IBM Platform LSF.

Pressing the **Refresh** button causes the connection to the cluster to be verified, and basic cluster information to be displayed in the **Scheduler info** text control. When finished, click **OK**. The selected scheduler (and head node, if applicable) will be used for subsequent jobs submission using the Submit Job GUI. To change the scheduler or head node, pop up the **Select Scheduler** dialog again, and select a different scheduler or head node.

To submit a job, use menu item **Tools>Job Management>Select Scheduler** to pop up the **Submit Job To** dialog. The controls applicable to options that are used less frequently are only shown if the **Show advanced options** check box is checked. Three tabs are shown on this dialog:

- **Analysis Specification**: Basic analysis parameters, including the product pathname, the project pathname, and the setup or setups to analyze. Several additional options may be
specified using this tab.

- **Compute Resources**: The user selects the resources to be allocated to the job and options for job distribution. Scheduler specific resources may be specified using the “Resource selection parameters.”

- **Scheduler Options**: Options for the job to be submitted to the scheduler, such as the job name and job priority.

Use the **Preview Submission** button to display information about the job to be submitted to the cluster. For Windows HPC, this pops up a dialog showing the parameters of the job to be submitted, and the parameters of the first execution task, including the full command line of this task; for distributed jobs, additional tasks will be started automatically, as needed, to complete the job. For other clusters, the dialog shows the job submission command and the contents of the job script for the job. If a problem is detected with the selected job parameters, a message box containing an error message may be shown, instead of the job submission parameters.

Use the **Submit Job** button to submit the job to the selected cluster. If there is an error submitting the job, then an error message will be shown. If the job is submitted successfully, then a dialog box showing the job ID is displayed to the user. If the **Begin monitoring this job now** check box is checked when this dialog is dismissed, then the **Monitor Job** dialog pops up to monitor the job. The **Monitor Job** dialog may be dismissed at any time, and the job will continue to run. To resume monitoring the same job, select menu item **Tools> Job Management>Monitor Jobs** to access the **Monitor Job** dialog. Selecting a project file using the **Recent Jobs** drop-down list or the file browser popped up by the “...” button will resume monitoring of the most recent job submitted for the selected project. Monitoring will not be possible from the **Monitor Job** dialog if there is no network connection between the submission host and the cluster execution host where the Desktop process runs, or if this connection is blocked.

"Submitting and running a non-distributed job " below

"Submitting and running a non-MPI job distributed to multiple hosts" on the next page

"Submitting and running an MPI job distributed to multiple hosts" on page 3-14

**Submitting and running a non-distributed job**

1. Copy the files Tee.aedt and Tee.PDF from the Examples/HFSS/RF Microwave subdirectory of the product installation directory to a directory which is accessible from the cluster hosts using the same path. This directory, the project directory, must be readable and writeable by the user running the job. Disable the frequency sweep as described above.
   
   The scheduler (and head node, if applicable) should be set as described above before submitting the job to the cluster.

2. Select menu item **Tools>Job Management>Select Scheduler** to activate the **Submit Job To** dialog.

3. In the **Analysis Specification** tab, ensure that the **Product path** edit control contains a pathname of the product that is valid for the cluster execution nodes.
a. Specify the path to the project file in the Project edit control.
b. Specify the setup “TeeModel:Nominal:Setup1” in the Analysis Setups group box.

4. Switch to the Compute Resources tab, and make sure that the Use automatic settings check box is not checked.
   a. For the Method, select Number of Tasks and Cores in the drop-down list.
   b. Set the Total number of tasks to 1.
   c. Set the Cores per distributed task to 1.
   d. Ensure that the Limit number of tasks per node to: check box is not checked. The other options on all tabs may be set to any values.

5. Press the Preview Submission button. If no errors are found, then the Preview Submit Job Results dialog should display information on how the job will be submitted to the cluster. If an error is displayed instead of the Preview Submit Job Results dialog, then address the error before continuing. This dialog should indicate that only one host is needed for the job. If this is not the case, then review the job resources specified on the Compute Resources tab, and retry. If the job and task parameters are correct, then close the Preview Submit Job Results dialog and continue.

6. Submit the job using the Submit Job button. If an error occurs on job submission, address the error before submitting the job again. If the cluster is Windows HPC, you may be prompted for a password if it is not cached. If this occurs, enter the password and continue. If possible, monitor the job until it finishes.

If the job fails to complete successfully, check the following items:

- The user submitting the job must be authorized to run jobs on the cluster.
- The pathname of the product must be accessible on the cluster execution node(s) where the job runs, and the job user must have permission to run the product executables.
- The pathname of the project must be accessible on the cluster execution node(s) where the job runs. The job user must have permission to read from and write to the directory containing the project file, and there must be adequate disk space for the job results.
- The temp directory used by the job must exist and it must have adequate disk space for the job temporary files.
- Check for errors or warnings in the Monitor Job dialog, and address these issues.
- Check the cluster for errors or warnings related to the job and address these issues. The job ID is displayed in a dialog when the job is successfully submitted. The job ID of the last job run on the specified project is also shown in the Monitor Job dialog.

Submitting and running a non-MPI job distributed to multiple hosts

The cluster must have at least two hosts with the same number of cores per host. Let N be the smallest number of cores per host, such that the cluster contains at least two hosts with N cores per host. The job will be submitted to run on two hosts with N cores per host, requesting a total of 2 * N cores. For many clusters, selecting N = 4 may be a good choice.
1. Copy the files Tee.aedt and Tee.PDF from the “Examples/HFSS/RF Microwave” sub-directory of the product installation directory to a directory which is accessible from the cluster hosts using the same path. This directory, the project directory, must be readable and writeable by the user running the job. Enable the frequency sweep as described above. The scheduler (and head node, if applicable) should be set as described above before submitting the job to the cluster.

2. Select menu item Tools>Job Management>Select Scheduler to activate the Submit Job To: dialog.

3. In the Analysis Specification tab, ensure that the Product path edit control contains a pathname of the product that is valid for the cluster execution nodes.
   a. Specify the path to the project file in the Project edit control.
   b. Specify the setup “TeeModel:Nominal:Setup1” in the Analysis Setups group box.

4. Switch to the Compute Resources tab, and make sure that the Use automatic settings check box is not checked.
   a. For the Method, select Number of Tasks and Cores in the drop-down list.
   b. Set the Total number of tasks to 2 * N.
   c. Set the Cores per distributed task to 1.
   d. Ensure that the Limit number of tasks per node to: check box is not checked. The other options on all tabs may be set to any values.
   e. In the Job distribution group box, press the Modify button to access the Job Distribution dialog.
   f. In the Enable Distribution Types section, uncheck the Use defaults check box, and check only the Frequencies check box.
   g. In the Distribution levels section, select the Single level only radio button.
   h. Click OK to close the Job Distribution dialog.

5. For the schedulers which support setting the minimum and maximum cores per node for a job (Windows HPC and LSF), set both of these values to the value N, determined earlier, as follows:
   a. Click on the “...” button to activate the Compute Resource Selection Parameters dialog.
   b. For the MinCoresPerNode parameter, check the Specify Parameter check box, and set the value to N.
   c. For the MaxCoresPerNode parameter, check the Specify Parameter check box, and set the value to N.
   d. Click OK to close the Compute Resource Selection Parameters dialog. The other options on all tabs may be set to any values.

6. Press the Preview Submission button. If no errors are found, then the Preview Submit Job Results dialog should display information on how the job will be submitted to the cluster. If an error is displayed instead of the Preview Submit Job Results dialog, then
address the error before continuing. This dialog should indicate that a total of 2 * N cores are required for the job, and that N cores will be needed on each host allocated to the job. If this is not the case, then review the job resources specified on the Compute Resources tab, and retry. If the job and task parameters are correct, then close the Preview Submit Job Results dialog and continue.

7. Submit the job using the Submit Job button. If an error occurs on job submission, address the error before submitting the job again. If the cluster is Windows HPC, you may be prompted for a password if it is not cached. If this occurs, enter the password and continue. If possible, monitor the job until it finishes.

If the job fails to complete successfully, check the following items, in addition to the items mentioned for the first test:

- The Desktop process will use the cluster to launch processes on the other hosts allocated to the job. Verify that the cluster configuration allows a job to launch processes on the other hosts allocated to the job.
- Job processes on different hosts allocated to the job need to communicate. Verify that communication between these job processes is not blocked.

Submitting and running an MPI job distributed to multiple hosts

Determine which MPI Vendor will be used for MPI jobs, Intel MPI or IBM Platform MPI. This test should use the same MPI Vendor that will be used by cluster users. If no MPI Vendor is specified by the user, then the default MPI Vendor (Platform Computing) will be used. Intel MPI is denoted by the string “Intel.” IBM Platform MPI is denoted by the string “Platform Computing.” If both MPI Vendors may be needed by cluster users, then this test should be run twice, once with each MPI Vendor specified. Consult "Message Passing Interface (MPI) Installation" on page 6-1 to determine if the MPI provided by the selected MPI Vendor needs to be installed and to determine how it should be configured.

The cluster must have at least two hosts with the same number of cores per host. Let N be the smallest number of cores per host, such that the cluster contains at least two hosts with N cores per host. The job will be submitted to run on two hosts with N cores per host, requesting a total of 2 * N cores.

1. Copy the files differential_stripline.aedt and differential_stripline.pdf from the Examples/HFSS/Transmission Lines subdirectory of the product installation directory to a directory which is accessible from the cluster hosts using the same path. This directory, the project directory, must be readable and writeable by the user running the job. Disable the frequency sweep as described above.

   The scheduler (and head node, if applicable) should be set as described above before submitting the job to the cluster.

2. Select menu item Tools>Job Management>Select Scheduler to activate the Submit Job To: dialog.
3. In the **Analysis Specification** tab, ensure that the **Product path**: edit control contains a pathname of the product that is valid for the cluster execution nodes.
   a. Specify the path to the project file in the **Project**: edit control.
   b. Specify the setup Differential Stripline:Nominal:3GHz in the **Analysis Setups** group box.
   c. In the **Analysis options** group box, set the HFSS/MPIVendor batchoption to the desired MPI Vendor, “Intel” or “Platform Computing.” If this batchoption is present, but set to a different value, then you can use the **Edit** button to change the value; edit the value in the **Edit Batchoption** dialog and press the **Update** button to register the new value before closing the dialog with the **Done** button. If it is not set, then use the **Add** button to add it; in the **Add Batchoption** dialog, select the registry key “HFSS/MPIVendor” in the grid, and then edit the text in the **Value** control, if needed, and press **Add** to register the new value, before closing the dialog with the **Done** button.

4. Switch to the **Compute Resources** tab, and make sure that the **Use automatic settings** check box is not checked.
   a. For the Method, select **Number of Tasks and Cores** in the drop-down list.
   b. Set the **Total number of tasks** to 2 * N.
   c. Set the **Cores per distributed task** to 1.
   d. Ensure that the **Limit number of tasks per node to**: check box is not checked.
   e. In the **Job distribution** group box, press the **Modify** button to access the **Job Distribution** dialog.
   f. In the **Enable Distribution Types** section, uncheck the **Use defaults** check box, and check only the **Direct Solver Memory** check box.
   g. In the **Distribution levels** section, select the **Single level only** radio button.
   h. Click **OK** to close the **Job Distribution** dialog.

5. For the schedulers which support setting the minimum and maximum cores per node for a job (Windows HPC and LSF), set both of these values to the value N, determined earlier, as follows:
   a. Click on the “…” button to activate the **Compute Resource Selection Parameters** dialog.
   b. For the MinCoresPerNode parameter, check the **Specify Parameter** check box, and set the value to N.
   c. For the MaxCoresPerNode parameter, check the **Specify Parameter** check box, and set the value to N.
   d. Click **OK** to close the Compute Resource Selection Parameters dialog.

The other options on all tabs may be set to any values.

6. Press the **Preview Submission** button. If no errors are found, then the **Preview Submit Job Results** dialog should display information on how the job will be submitted to the cluster. If an error is displayed instead of the **Preview Submit Job Results** dialog, then
address the error before continuing. This dialog should indicate that a total of 2 * N cores are required for the job, and that N cores will be needed on each host allocated to the job. If this is not the case, then review the job resources specified on the Compute Resources tab, and retry. If the job and task parameters are correct, then close the Preview Submit Job Results dialog and continue.

7. Submit the job using the Submit Job button. If an error occurs on job submission, address the error before submitting the job again. If the cluster is Windows HPC, you may be prompted for a password if it is not cached. If this occurs, enter the password and continue. If possible, monitor the job until it finishes.

If the job fails to complete successfully, in addition to the items mentioned for the first and second tests, MPI configuration issues may cause job failures. Check the following MPI configuration issues:

- Verify that the correct MPI Vendor ("Intel" or "Platform Computing") is specified using a batchoption setting in the Submit Job To: dialog. The controls for specifying batchoptions are located in the Analysis Options group box on the Analysis Specification tab of the Submit Job To: dialog.

- Verify that the specified MPI Vendor is installed (if needed) and configured correctly for the user submitting the job, as described in "Message Passing Interface (MPI) Installation" on page 6-1. Pay close attention to the registration of user passwords because the lack of a cached password when one is required, or a cached password that is incorrect or out of date (even if no cached password is required) may cause authentication errors. Note that passwords are cached separately on each cluster host, so a job may have authentication errors on some hosts but not on other hosts.

### Scheduler Specific Configuration and Testing

The sections below describe configuration requirements for each supported scheduler. Methods for running tests to verify the configuration are included.

"Windows HPC" below

"IBM Platform LSF" on page 3-19

### Windows HPC

#### Cluster Configuration Guidelines

The Windows HPC cluster affinity setting should be set to “Non-Exclusive jobs” or “No jobs.” Setting the Windows HPC cluster affinity to “All jobs” may prevent the job from using all of the resources allocated to the job, for the reasons outlined below.
Each ANSYS Electromagnetics analysis job is started as a single process on a single host. If the job needs to distribute a portion of the analysis to a different host, a single core task is started on the remote host to start the analysis processes on that host. Only one core is allocated for the task used to start these processes, even if multiple processes are started by the task or multiple cores are needed by any of the processes started by the task. If the Windows HPC cluster affinity setting is set to “All jobs,” then the remote processes would be limited to using a single core on the remote host, even if the job was allocated multiple cores on the remote host.

Running Test Jobs

When attempting to submit a job to the cluster, if there is a problem connecting to the head node, check the following items:

- The head node name is correct
- There is a network connection between the job submission node and the head node
- No firewall blocks communication between the job submission node and the head node
- The 64 bit Windows HPC 2008 R2 Client Utilities are installed on the job submission node
- The user making the connection is an authorized Windows HPC cluster user

If the job cannot be monitored from the ANSYS Electromagnetics Monitor Job dialog because there is no network connection between the submission node and the cluster execution node (or the connection is blocked), then the HPC Job Manager installed with the Windows HPC Client Utilities can be used to monitor the job.

If errors occur with any of the test jobs, then information on the failure can be found by viewing the job in the HPC Job Manager. Check for messages in the Job Progress window. For any failed tasks, check the Results, Details, and Allocated Nodes tabs for more information on possible problems.

Non-distributed job

If the job fails to complete successfully, check the following items:

- The user submitting the job must be authorized to run jobs on the cluster.
- The pathname of the product must be accessible by the user running the job from the cluster execution node(s) where the job runs.
- The pathname of the project must be accessible by the user running the job from the cluster execution node(s) where the job runs. The directory containing the project file must be writable by the user running the job from the cluster execution node(s) where the job runs, and there must be adequate disk space for the job results.
- The temp directory specified by batchoptions or using the installation default setting on the cluster execution node(s) where the job runs must have adequate disk space for the job temporary files.
- Check for errors or warnings in the Monitor Job dialog, and address these issues.
Also check the Job using the **Windows HPC Job Manager**. Check for error and warning messages associated with the job. Also check for failed tasks, and check the task information for any failed tasks.

**Non-MPI job distributed to multiple hosts**

This job will be submitted to run on two hosts with N cores per host, requesting a total of 2 * N cores, where N is the smallest number of cores per host, such that the cluster contains at least two hosts with N cores per host.

If the job fails to complete successfully, check the following items, in addition to the items mentioned for the first test:

- In addition to the task used to start the job (the Desktop task), an additional task is started each of the other job hosts to start processes on these hosts. For this task, there should be one such task (a Remote Engine Task). Verify that the Remote Engine Task is able to start and communicate with the Desktop task.

**MPI job distributed to multiple hosts**

This job will be submitted to run on two hosts with N cores per host, requesting a total of 2 * N cores, where N is the smallest number of cores per host, such that the cluster contains at least two hosts with N cores per host.

The default MPI Vendor is IBM Platform MPI. If the MPI Vendor is “Platform Computing” (IBM Platform MPI), then installation of IBM Platform MPI on the cluster hosts is not required, and registration of user passwords on the cluster hosts is not required. We recommend verifying that user passwords are not registered with IBM Platform MPI on the cluster hosts, because an incorrect or out of date password registered with IBM Platform MPI can cause authentication errors in this case. If the MPI Vendor is “Intel,” then installation of Intel MPI on all cluster nodes is required as described in the "Install Intel MPI" on page 6-1.

If the job fails to complete successfully, in addition to the items mentioned for the first and second tests, MPI configuration issues may cause job failures. Check the following MPI configuration issues:

- Verify that the correct MPI Vendor (“Intel” or “Platform Computing”) is specified via a batch-option setting when submitting the job.
- Verify that the specified MPI Vendor is installed and configured correctly for the user submitting the job, as described in "Message Passing Interface (MPI) Installation" on page 6-1. Note that for both IBM Platform MPI and for Intel MPI, MPI password registration is done separately for each cluster host. If authentication errors occur for some cluster hosts but not for other cluster hosts, then the errors may be due to an incorrect password or no password registered on some hosts.
IBM Platform LSF

Cluster Configuration Guidelines

To allow users to submit exclusive jobs, the users must have access to queues which are configured with the EXCLUSIVE property set to Y. Appropriate queues should be configured with the EXCLUSIVE property set if users need to submit exclusive jobs.

Using the ANSYS Electromagnetics job submission GUI, a user may submit jobs where the user specifies tasks automatically and specifies the “Number of Cores and (Optional) RAM.” However, the RAM per core may only be specified if the LSF cluster is configured with parameter RESOURCE_RESERVE_PER_SLOT set. This LSF cluster configuration parameter should be set to allow user to specify the RAM per core for the job.

Running Test Jobs

If there is a problem connecting to the scheduler when the job submission node is the local node (where the GUI is running) and the lsf scheduler is specified, check the following items:

- The cluster is up and running normally.
- The local node is configured as a job submission node for the cluster.
- The user running the GUI has permission to submit jobs to the cluster.
- The environment was configured for submission of jobs to the cluster before starting the product, ansysedt.

Non-distributed job

- If the job fails to complete successfully, check the following items:
  - The user submitting the job must be authorized to run jobs on the cluster.
  - The pathname of the product must be accessible by the user running the job from the cluster execution node(s) where the job runs.
  - The pathname of the project must be accessible by the user running the job from the cluster execution node(s) where the job runs. The directory containing the project file must be writable by the user running the job from the cluster execution node(s) where the job runs, and there must be adequate disk space for the job results.
  - The temp directory specified by batchoptions or using the installation default setting on the cluster execution node(s) where the job runs must have adequate disk space for the job temporary files.
  - Check for errors or warnings in the Monitor Job dialog, and address these issues.

Non-MPI job distributed to multiple hosts

This job is submitted to run on two hosts with N cores per host, requesting a total of $2 \times N$ cores, where N is the smallest number of cores per host, such that the cluster contains at least two hosts with N cores per host.
For this test, the product, ansysedt, runs on one of the hosts, and the LSF blaunch command is used to start an ansoftsm process on the other host. The ansoftsm process is used to start any additional processes needed by the job on the other host. In some cases, multiple ansoftsm processes are started on the other host, but only one such process should be running on a given host at any time.

If the job fails to complete successfully, check the following items, in addition to the items mentioned for the first test:

- Verify that the LSF blauch command is able to start the ansoftsm process on the other host.

**MPI job distributed to multiple hosts**

This job is submitted to run on two hosts with N cores per host, requesting a total of 2 \* N cores, where N is the smallest number of cores per host, such that the cluster contains at least two hosts with N cores per host.

The default MPI Vendor is IBM Platform MPI. If the MPI Vendor is “Platform Computing” (IBM Platform MPI), then installation of Intel MPI on all cluster nodes is required as described in "Install IBM Platform MPI" on page 6-3. If the MPI Vendor is “Intel,” then installation of Intel MPI on all cluster nodes is required as described in "Install Intel MPI" on page 6-1. If the job fails to complete successfully, in addition to the items mentioned for the first and second tests, MPI configuration issues may cause job failures. Check the following MPI configuration issues:

- Verify that the correct MPI Vendor (“Intel” or “Platform Computing”) is specified via a batch-option setting when submitting the job; otherwise the default, “Platform Computing,” is used for the job.

- Verify that the specified MPI Vendor is installed and configured correctly for the user submitting the job, as described in "Message Passing Interface (MPI) Installation" on page 6-1. Note that for both IBM Platform MPI and for Intel MPI, MPI password registration is done separately for each cluster host. If authentication errors occur for some cluster hosts but not for other cluster hosts, then the errors may be due to an incorrect password or no password registered on some hosts.
4 - Install Common Libraries

The Installation CD includes an Install Common Libraries option which installs a superset of materials, components, and other libraries. This option allows customers to place ANSYS Electromagnetics libraries in a common location, so multiple users and multiple ANSYS Electromagnetics products may share the same libraries. You can also add custom materials and components that will be automatically accessible to all users and products that use this common library location.

Procedure for Installing Libraries

To install ANSYS Electromagnetics software:

1. Log in as Administrator, or log on to an account with administrator privileges.

2. If you have received a CD for the ANSYS Electromagnetics product you wish to install, insert it into the CD-ROM drive. The Installation Shell should appear automatically. If it does not appear, run the autorun.exe file located in the top-level directory of the CD-ROM drive.
   If you downloaded the ANSYS Electromagnetics product ("Additional Resources" on page 1-3), unzip the download, and run the autorun.exe file in the newly created directory. The Installation Shell appears.

3. Click Install Common Libraries.
   The Install Wizard for the ANSYS Electromagnetics Common Libraries appears. (The initial options differ depending on whether you have previously installed libraries or are doing a first or clean install: if you have a version installed, you have the option to modify, repair, or remove features.)

4. Click Next.
The License Agreement screen appears.
5. Read the license agreement carefully. Click Yes if you accept the terms of the agreement. The Customer Information screen appears.
6. Enter your information in the User Name and Company Name boxes.
7. Click Next. The Choose Destination Location screen appears.
8. Under Destination Folder, do one of the following to accept the default path or specify a new one:
   - Accept the default installation directory listed under Destination Folder.
   - To select a different directory, click Browse. The Choose Folder window appears. Select a directory, and click OK. If you have other ANSYS Electromagnetics software we recommend that you install new software in the same directory.
   - To create a new directory, click Browse. The Choose Folder window appears. Enter the new name in the Path field, and click OK.
9. Click Next. The Review Settings Before Copying Files screen appears. If you want to change anything, click Back.
10. To go ahead, click Next.
11. The Setup Status dialog shows the progress of the install. When the install has completed, the Finish dialog displays.
12. Click Finish to close the dialog.
5 - Updating Software and Licenses

After installation, you may need to reinstall the software, update a license, or add a software component, such as a library. If you have already installed the software, the next time you run the Setup program, the installation starts in Maintenance mode.

Using Maintenance Mode

The following options are available in Maintenance mode:

- **Modify**: Allows you to modify the existing installation. Use this option to select new software components or to select a new license file.
- **Repair**: Upgrades the existing software with new features and defect patches and refreshes files.
- **Remove**: Removes all installed software components.
- **Update/Remove** (This displays only if the version you are installing is newer than what is installed.)

To start the software in Maintenance mode:

1. Use the Windows Control Panel to access the Programs and Features window.
2. From the Programs and Features program list, click the desired ANSYS Electromagnetics product program folder.
3. Right-click `<ANSYS Electromagnetics Product>` and select Change.
   A Setup window appears for the desired ANSYS Electromagnetics software.
4. Select one of the following:
   - **Modify**: Change the program features installed by the previous setup. You can also choose this option to specify new license files or a new server.
   - **Repair**: Reinstall all program features installed by the previous setup.
   - **Remove**: Remove the ANSYS Electromagnetics product previously installed.

5. Click **Next**, and follow the instructions that appear.

**Updating a Software License**

There may be times when you need to update a software license. This may occur for a number of reasons, such as:

- When you change servers. You need to
  1. Install ANSYS License Manager on the new server machine, and add the new license file.
  2. Run each ANSYS Electromagnetics software installation in **Maintenance mode**, and specify the names of the new servers.
- When you receive an updated purchased license file. You only need to update the license file using the ANSYS License Manager.

If you are changing the name of the license server used by the software, you need to modify the client installation:
1. Click **Start> All Programs> ANSYS Electromagnetics.**

2. Click the desired ANSYS product program folder, and then click `<ANSYS ElectromagneticsProduct> Maintenance.`
   
   The **Maintenance Mode** screen appears for the desired ANSYS Electromagnetics software.

3. Click **Modify.**

4. Click **Next** in each window.

5. On the **License Information** screen, select the new license file.

   **Note** Select **I have a new license file** to change the license file (if you received a new one from ANSYS Electromagnetics, either evaluation or purchased).

   Select **I want to specify a license server** if you need to change the name of the machine where the ANSYS License Manager is installed.

   See "Installing the Software" on page 2-1 for more details on available license options.
6 - Message Passing Interface (MPI) Installation

The two supported Message Passing Interface (MPI) options in ANSYS Electromagnetics Suite are IBM Platform Computing and Intel. MPI is required when you solve using multiple nodes. It is not required to solve using a single node.

This chapter provides information about the Windows installations of MPI:

Windows

To obtain an ethernet address as a valid license hostid when running Microsoft Windows:

1. If you are using a laptop, disconnect the laptop from the docking station if docked.
2. Go to the command prompt.
3. Type `ipconfig /all`, and press Enter.
   A listing of network interface cards is shown.
4. For a valid network interface card, look for the **Physical Address** line. The ethernet address for that device is the value of the physical address, without dashes.

   **Note** Other types of adapters, including Point-to-Point Tunneling Protocol (PPTP) adapters and the Microsoft Loopback Adapter, may be listed here. These are not valid for license hostids.

Install Intel MPI

1. After you click **Install Intel MPI**, you see the following dialog.

2. Specify the installation directory and click **Extract**.
3. You then see the installation wizard for Intel(R) Library Runtime Environment.

4. Click **Next** and follow the prompts to accept the license terms and Install the software as directed.

   Intel MPI typically requires passwords to be registered on the simulation host machine, which is typically the first machine in the distributed machine list. Password registration may also be required on other machines to allow access to network resources, such as shared drives. Go to All Programs > Intel® Software Development Products > Intel MPI Library Runtime Environment > Build environment for Intel® 64 > wmpiregister and enter your account and password information.

   **Note** You can work around the password registration for cases where accessing network resources is not needed by performing the Active Directory setup (see [http://software.intel.com/sites/products/documentation/hpc/ics/impi/41/win/Reference_Manual/Active_Directory_Setup.htm](http://software.intel.com/sites/products/documentation/hpc/ics/impi/41/win/Reference_Manual/Active_Directory_Setup.htm)) and setting the environment variable I_MPI_MTHOD to "delegate".
5. Define the temp directory on all machines such that the designated user is allowed full control.
6. Disable the Windows firewall, or configure it as specified in the Firewall Configuration topic in the ANSYS Electromagnetics product online help.
7. Select which MPI to use in the MPI Vendor box on the Options tab under Tools > Options > HPC and Analysis Options.

**Install IBM Platform MPI**

For IBM Platform MPI jobs submitted to a Windows HPC cluster, an installation of IBM Platform MPI is not required because IBM Platform MPI is provided in the ANSYS Electromagnetics Suite installation. When using IBM Platform MPI for analyses not running on a Windows HPC cluster, IBM Platform MPI must be installed as described below.

If a distributed analysis uses IBM Platform MPI but it is not running as a Windows HPC Job, then IBM Platform MPI must be installed locally on each of the analysis hosts in Service Mode.

1. After choosing Install IBM Platform MPI, you see an IBM Platform-MPI dialog.

![IBM Platform-MPI Installation](image)

**Introduction**

InstallAnywhere will guide you through the installation of IBM Platform-MPI.

It is strongly recommended that you quit all programs before continuing with this installation.

Click the 'Next' button to proceed to the next screen. If you want to change something on a previous screen, click the 'Previous' button.

You may cancel this installation at any time by clicking the 'Cancel' button.

2. Click **Next**.

After a few moments, the **Software License Agreement** screen opens.
3. Click I accept the terms in the license agreement. The License screen closes and the Choose Install Set screen opens.

4. Select Service Mode.

5. Click Next to continue. The Service Listening Port screen displays.
6. Click **Next**.
   
The Choose Install Folder screen displays.
Select the Destination location for the install. You can specify a location.
7. Click **Next** to go to the **Choose Shortcut Folder** screen.

![Choose Shortcut Folder Screen](image)

8. Make your selections and click **Next**.

The **Pre-Installation Summary for Service Mode** screen displays.
9. Review the settings and click **Install**.
   
The Install Screen displays and is replaced by the **Install Complete** screen.
10. Click **Done**.

Platform MPI does not require a password to be registered for it to work with ANSYS Electromagnetics Suite software. However, password registration may be needed to allow access to network resources, such as shared drives. For example, if the ANSYS Electromagnetics Suite is installed on a shared drive, then password registration is required. If you want to register a password, open a command window, go to `<install_dir>/win64-common\fluent_mpi\multiport\mpi\win64\pcmpi\bin`, and run the following command:

```
mpidiag -s <your machine name> -at -cache
```

Enter your password at the prompt.

If password registration with IBM Platform MPI is required, and the password for a given user is changed because it has expired or for any other reason, then the new password for the user must be registered with IBM Platform MPI.

**General Notes**

**Note**  The standard setup described here assumes that the user is part of a domain and is running on computers within the given domain.

Using network resources, such as shared drives, may require password registration on all machines for both Platform and Intel MPI.
It is easiest if share permissions are set to Full Control for the Everyone group for the temp directories.

The following provides information about specific scenarios:

**Scenario 1**

The user is administrator or standard user on the local machine and needs to run on one or more remote computer as an admin or a standard user. In addition, the user always puts the local machine first in the distributed machine list so that the local machine is the simulation host.

This is the standard situation and does not require further specifications. MPI should be installed on all machines, and the solver options and MPI configuration on the local machine will be used.

**Scenario 2**

The user is administrator or standard user on the local machine and is not defined as a user on the remote machines. In addition, the user always puts the local machine first in the distributed machine list so that the local machine is the host.

This situation is similar to Scenario 1, but the MPI jobs will run as a user that does not exist on the remote machines. ANSYS strongly recommends setting the temp directories to Full Control for the Everyone group to avoid issues with permissions.

**Scenario 3**

The user is administrator or standard user on the local machine, but needs to run only on remote machines. In this scenario the remote machine that is first in the distributed machine list will be the host machine for the simulation.

RSM must be installed and registered on the host machine. There are two RSM settings that can affect the simulation:

- The first setting is under RSM properties, which can be set to System User or A specific user. This setting is chosen at installation and can be changed through the Services panel or by going to Remote Simulation Manager > Remote Simulation Manager Maintenance > Modify.

- The second setting is in the Desktop in Tools > Options > General Options, on the Remote Analysis Options tab. Here you can chose either Service User or Specified User. Here is additional information about these settings:

<table>
<thead>
<tr>
<th>RSM Properties</th>
<th>Remote Simulation Options</th>
<th>Simulation runs on remote machine as</th>
<th>Notes</th>
</tr>
</thead>
<tbody>
<tr>
<td>System</td>
<td>Service User</td>
<td>Local System user</td>
<td>Local System user will not have ANSYS Electromagnetics Suite user preferences and may not have necessary privileges or access to resources.</td>
</tr>
</tbody>
</table>
### Windows HPC Scheduler

If you are using the Windows HPC Scheduler, you have the option to install the ANSYS Electromagnetics Suite at the same location on each node individually. The simulation can then be run from one of the compute nodes or from the head node using the Windows HPC Scheduler GUI that is selected in **Tools>Job Management>Select Scheduler** before using **Tools>Job Management>Submit Job**. The MPI setup is the same as described previously. If using IBM Platform MPI, no installation of IBM Platform of MPI is needed, and no password registration is required. For this case, we recommend clearing any previously registered passwords. If a registered password expires or is changed for any other reason, then MPI failures may occur due to the incorrect password. To clear a previously registered password, open a command window, go to `<install_dir>\win64\common\fluent_mpi\multiport\mpi\win64\pcmpi\bin`, and run the following command:

```
mpidiag -s <your machine name> -at -clearcache
```

The following command may be used to check whether a password is cached for the current user:

```
mpidiag -s <your machine name> -at -iscached
```

For general information on using the Windows Scheduler, see the Integration with Microsoft Windows HPC Scheduler topic in the ANSYS Electromagnetics Suite online help.

**Note** The LSF scheduler is not supported on Windows networks for ANSYS Electromagnetics Suite simulations that require MPI (for example, DDM).
Network Installation

If you want to use a network installation with the Windows HPC Scheduler, follow these steps:

1. Install ANSYS Electromagnetics Suite on the head node.

2. Share the AnsysEM directory in the head node installation (setting Full Control for Everyone).

3. Install Intel MPI on all nodes. Register your password with MPI on all nodes. Perform Active Directory Setup on all nodes. To avoid password registration on all nodes, see the note in "Install Intel MPI" on page 6-1. See http://software.intel.com/sites/products/documentation/hpc/ics/impi/41/win/Reference_Manual/Active_Directory_Setup.htm

Or

Installation of IBM Platform MPI is not required because IBM Platform MPI is provided in the ANSYS Electromagnetics Suite installation. Password registration is not required for Windows HPC jobs. It is recommended that you read the information on using Platform MPI with Windows HPC for MPI Based Solves in the online help before submitting a job.

4. Install the following Microsoft Visual C++ Redistributables on all nodes:

   - Microsoft Visual C++ 2008 (x86 and x64)
   - Microsoft Visual C++ 2010 (x86 and x64)
   - Microsoft Visual C++ 2012 (x64 only)
   - Microsoft Visual C++ 2013 (x86 and x64)

   The necessary installers are provided in the installation folder in the Prerequisites directory.

5. Bring up ANSYS Electronics Desktop on each node using the network installation to verify the installation was done properly.

6. Place the pj in a shared folder (where Everyone has Full Control).

7. Launch ANSYS Electronics Desktop on the head node. Set the head node from the ANSYS Electronics Desktop using Tools > Job Management > Select Scheduler.

8. Open the Windows Scheduler GUI from the ANSYS Electronics Desktop to submit the job (see Tools > Job Management > Submit Job). Information is available in the Integration with Microsoft Windows HPC Scheduler topic in the ANSYS Electronics Desktop online help.

9. Be sure to use the UNC pathname for the product and project (see the Integration with Microsoft Windows HPC Scheduler topic in the online help for more details).

10. Name the job and set other options as required for your setup. We recommend setting the version of MPI (Platform Computing or Intel) in the Batchoptions by adding the following, for example:

'HFSS/MPIVendor'=Intel
'HFSS-IE/MPIVendor'=Intel
'Q3D Extractor/MPIVendor'=Intel
or
HFSS/MPIVendor'=Platform Computing
'HFSS-IE/MPIVendor'=Platform Computing
'Q3D Extractor/MPIVendor'=Platform Computing

After the job is submitted, you can monitor the job using the **HPC Job Manager**.

Testing has shown that when using Windows Server 2012 with a Network installation, the head node should not be one of the computer nodes.

**Troubleshooting Section**

- **I am getting an error message that mentions AnsoftRSMService. Is there a problem with my installation?**
  
  `[error] Unable to locate or start COM engine on 'your_machine_name_here': Unable to reach AnsoftRSMService. Check if the service is running and if the firewall allows communication…`

  Do not immediately assume there is a problem with RSM, especially if `your_machine_name_here` is your local machine. The message is stating that it cannot start the Com engine. This can be the result of a misspelled machine name or an issue with the firewall. Check these first.

  If the host machine is a remote machine (`your_machine_name_here` is a remote computer), there might be an issue with the RSM service on the remote computer. Check the RSM installation and registration on that machine.

- **Is there an easy way to test the computers in my Distributed Machine Configurations?**
  
  1. Select **HPC and Analysis Options**.
  2. Click **Edit**, and select one or more machine names in the list on the **Machines for Distributed Analysis** dialog.
  3. Click the **Test Machines** button. A set of tests will be run from the current machine to the selected machines that includes pinging the computer and checking the RSM availability.

- **When I attempt to simulate, I get the following error message:**

  `[error] The attempted launch of solvers via MPI failed while connecting to communication pipes. The probable cause is failure to install the vendor MPI on one or more machines or password authentication failure for MPI during the launch attempt. Followed by
  
  Unable to create child process: mpirun.**

  Possible causes for the messages are:

  - MPI is not installed on one or more machines.
  - The wrong version of MPI is installed on one or more machines.
The password registration is incorrect on one or more machines.

ANSYS Electromagnetics Suite is not installed in the same location on all machines.

The user is not a registered user on the remote machines for a workgroup network.

There is an issue with the firewall.

**When I attempt to simulate, I get the following error message:**

```plaintext
[error] Distributing matrix solutions for domains, process mpirun error: Simulation directory creation failed for one or more MPI solvers. Check access privileges and disk space for the following:…
```

This message means you do not have the correct privilege for the temp directory on one or more of the remote machines. In many cases the software uses the installation temp directory that is defined during the installation on the given machine. It is recommended that this directory be set for Everyone to have full Read/Write privileges on all machines.

**My setup was working last week, but today I am getting a password authentication failure error.**

This error may occur if anything changed in your network configuration. The most common cause is your login password was changed as part of a periodic update for network security. If the passwords were registered for MPI, they need to be updated on all machines every time the login passwords are changed.
When you use the ANSYS Electromagnetics silent installation feature, the software is installed on your computer with no windows appearing during the installation process. InstallShield suppresses all dialog boxes, including error messages and informational message boxes.

To install ANSYS Electromagnetics software using a silent installation:

- Obtain a license file for the software you are installing.
- Obtain and install a hardware key if necessary.
- On a representative machine, install the ANSYS Electromagnetics software once from the command prompt, as described in this chapter.

Running the Silent Installation

To install ANSYS Electromagnetics software using a silent installation, you need to record a response file, and then use that file to install the software on other computers.

To record a response file, you must install the software one time on one machine:

1. Open a Windows command prompt.
2. Browse to the directory where setup.exe (full download) or the self-extracting executable (patch) is located.
3. Type the following, and press Enter:
   - Full download:
     ```
     setup.exe -r -f1<.iss response file>
     ```
   - Self-extracting patch:
     ```
     <path to patch .exe> -a -r
     -f1<path to iss response file>
     ```
4. Follow the normal prompts for installation. Once complete, the iss response file will contain information that can be used to install the same product on other Windows machines.

To install the software on other computers running Windows, perform the following steps on each machine:

1. Open a Windows command prompt.
2. Browse to the directory where setup.exe (full download) or the self-extracting executable (patch) is located.
3. Type the following, and press Enter:
   - Full download:
     ```
     setup.exe -s -f1<.iss response file>
     -f2<path to log file>
     ```
   - Self-extracting patch:
<path to patch .exe> -a -s
-f1<path to iss response file>
-f2<path to log file>

In silent mode, no confirmation windows of any kind are displayed. In this mode, InstallShield suppresses all dialog boxes, including error messages and informational message boxes.

4. To verify that the silent setup succeeded, open the setup.log file, and look at the ResultCode value in the [ResponseResult] section. If the value is zero, the installation was successful.

If a failure occurs in silent mode, Setup aborts just as if you clicked the Abort button in a wizard dialog box during a normal installation. In this case, an error is written to the log file.

**Note** The installation typically fails for one of the following two reasons:
- An invalid path was specified on the command line.
- There were differences in the dialog box sequences between the recorded response file and the current silent installation - for example, an additional message appearing during the playback due to differences in the two computer systems.

**Silent Codes**

<table>
<thead>
<tr>
<th>Result Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Success.</td>
</tr>
<tr>
<td>-1</td>
<td>General error.</td>
</tr>
<tr>
<td>-2</td>
<td>Invalid mode.</td>
</tr>
<tr>
<td>-3</td>
<td>Required data not found in the Setup.iss file.</td>
</tr>
<tr>
<td>-4</td>
<td>Not enough memory available.</td>
</tr>
<tr>
<td>-5</td>
<td>File does not exist.</td>
</tr>
<tr>
<td>-6</td>
<td>Cannot write to the response file.</td>
</tr>
<tr>
<td>-7</td>
<td>Unable to write to the log file.</td>
</tr>
<tr>
<td>-8</td>
<td>Invalid path to the InstallShield Silent response file.</td>
</tr>
<tr>
<td>-9</td>
<td>Not a valid list type (string or number).</td>
</tr>
<tr>
<td>-10</td>
<td>Data type is invalid.</td>
</tr>
<tr>
<td>-11</td>
<td>Unknown error during setup.</td>
</tr>
<tr>
<td>-12</td>
<td>Dialog boxes are out of order.</td>
</tr>
<tr>
<td>-51</td>
<td>Cannot create the specified folder.</td>
</tr>
<tr>
<td>Code</td>
<td>Description</td>
</tr>
<tr>
<td>------</td>
<td>---------------------------</td>
</tr>
<tr>
<td>-52</td>
<td>Cannot access the specified file or folder.</td>
</tr>
<tr>
<td>-53</td>
<td>Invalid option selected.</td>
</tr>
</tbody>
</table>
Software licenses for each module of ANSYS Electromagnetics products you purchased are contained in one or more license files provided by ANSYS. Each license file requires one or more license hostids, which are physical devices that contain a unique identification. License hostids are always required for the license server, and they may also be required for each machine running ANSYS software. Each supported operating system supports certain hostids:

<table>
<thead>
<tr>
<th>Operating System</th>
<th>Permitted hostids</th>
</tr>
</thead>
</table>
| Windows          | • The ethernet address of a network interface card (MAC address).  
                  | • Flexera FLEXid USB port hardware key.                   
                  | • Disk serial number.                                    |
|                  | To obtain the disk serial number, open a command prompt. Type `vol <drive letter>;`, for example `vol C:` |
| Linux            | • The ethernet address of a network interface card (MAC address).  
                  | • Flexera FLEXid USB port hardware key.                   |

This utility will provide the disk id by default:

http://www.ansys.com/Support/Licensing/Capture+License+Server+Information

Network Interface Card

A network interface card is a physical device used to connect to a network (also known as an ethernet card). Each network interface card includes a unique ID that can be used as a license hostid. You can obtain an ethernet address to use as a valid license hostid. The method for doing so differs depending on whether you are running Windows or Red Hat Enterprise Linux.

Windows

To obtain an ethernet address as a valid license hostid when running Microsoft Windows:

1. If you are using a laptop, disconnect the laptop from the docking station if docked.
2. Go to the command prompt.
3. Type `ipconfig /all`, and press Enter.

A listing of network interface cards is shown.
4. For a valid network interface card, look for the **Physical Address** line. The ethernet address for that device is the value of the physical address, without dashes.

   **Note** Other types of adapters, including Point-to-Point Tunneling Protocol (PPTP) adapters and the Microsoft Loopback Adapter, may be listed here. These are not valid for license hostids.

**Linux**

To obtain an ethernet address as a valid license hostid when running Red Hat Enterprise Linux:

1. If you are using a laptop, disconnect the laptop from the docking station if docked.
2. Start a terminal session.
3. Type `ip addr` (normally located in the `/sbin` directory) to see a listing of all active network devices.
4. Devices with an ethernet address show a **Link encap** value of **Ethernet**. The ethernet address for each device is the value in the **HWaddr** field, without colons.

**Hardware Key**

A Flexera FLEXid hardware key may be provided to you by ANSYS Electromagnetics as your physical device. The license hostid of a hardware key is affixed to the hardware key itself. The hostid is a 17- to 21-character value that begins with `FLEXID=`.

**Installing a Hardware Key on a USB Port**

Only a single device can be installed on each USB port.