Ansys Electromagnetics Suite 2024 R2 Windows Installation Guide

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

Release 2024 R2
July 2024

Copyright and Trademark Information

Disclaimer Notice

U.S. Government Rights

Third-Party Software

Conventions Used in this Guide

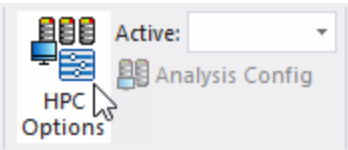
Please take a moment to review how instructions and other useful information are presented in this documentation.

- 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 you must type the word **copy**, then type a space, and then type **file1**.
  - On-screen prompts and messages, names of options and text boxes, and menu commands. Menu commands are often separated by greater than signs (>). 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** *filename*" means you must type the word **copy**, then type a space, and then type the name of the file.
- 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, while holding it down, press the **F1** key also. You should always depress the modifier key or keys first (for example, Shift, Ctrl, Alt, or Ctrl+Shift), continue to hold it/them down, and then press the last key in the instruction.

Accessing Commands: *Ribbons, menu bars,* and *shortcut menus* are three methods that can be used to see what commands are available in the application.

- The *Ribbon* occupies the rectangular area at the top of the application window and contains multiple tabs. Each tab has relevant commands that are organized, grouped, and labeled. An example of a typical user interaction is as follows:

  "Click **Simulation** > **HPC Options**"



This instruction means that you should click the **HPC Options** command on the **Simulation** ribbon tab. An image of the command icon, or a partial view of the ribbon, is often included with the instruction.

- The *menu bar* (located above the ribbon) is a group of the main commands of an application arranged by category such File, Edit, View, Project, etc. An example of a typical user interaction is as follows:

  "On the **File** menu, click the **Open Examples** command" means you can click the **File** menu and then click **Open Examples** to launch the dialog box.

- Another alternative is to use the *shortcut menu* that appears when you click the right-mouse button. An example of a typical user interaction is as follows:

  “Right-click and select **Assign Excitation** > **Wave Port**” means when you click the right-mouse button with an object face selected, you can execute the excitation commands from the shortcut menu (and the corresponding sub-menus).

Getting Help: Ansys Technical Support

For information about Ansys Technical Support, go to the Ansys corporate Support website, https://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 help from the Help menu, click **Help** and select from the menu:

- **[product name] Help** – opens the contents of the help. This help includes the help for the product and its *Getting Started Guides*.
- **[product name] Scripting Help** – opens the contents of the *Scripting Guide*.
- **[product name] Getting Started Guides** – opens a topic that contains links to Getting Started Guides in the help system.

Context-Sensitive Help

To access help from the user interface, press **F1**. The help specific to the active product (design type) opens.

You can press **F1** while the cursor is pointing at a menu command or while a particular dialog box or dialog box tab is open. In this case, the help page associated with the command or open dialog box is displayed automatically.

Table of Contents

1 - Getting Started

Installation Prerequisites

Before installing Ansys Electromagnetics Suite software, you will need:

- Ansys Electromagnetics physical product (USB or DVDs), or digital download.
  - Includes software, example projects, and documentation.
  - For download information, see: Additional Resources (Page 1-3).
- A license file (see: License File below).
- Ansys License Manager (see: Ansys License Manager below).
- TCP/IP installed, for counted evaluation and purchased licenses (see: TCP/IP below).
- A computer that meets the minimum or recommended system requirements for the Electromagnetics products you plan to run.
  - For detailed requirements, visit https://www.ansys.com/it-solutions/platform-support.

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. For a list of supported hostids and a general explanation of hostids, see: Appendix B.

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

Ansys License Manager

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 locations:

- On the computer where the software will be installed.
- On another machine accessible over the network.

> **Note:**
> You can install Ansys License Manager before or after installing Ansys Electromagnetics software.

Additional instructions are included as part of the Ansys License Manager download.

TCP/IP

Transmission Control Protocol/Internet Protocol (TCP/IP) is required to use any Ansys Electromagnetics Suite 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 already installed on computers running Windows.
- 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.
- If a network card is not installed, 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.

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.

Processor Selection (for Hybrid Graphics Hardware)

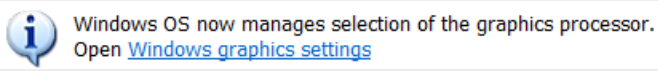
In the case of laptops that have both Intel integrated graphics and an nVidia graphics accelerator, you should ensure that the following two programs both use the "**High-performance NVIDIA processor**" and **not** the "*Integrated processor*":

- Ansys Electronics Desktop (ansysedt.exe)
- Ansys SIwave (siwave.exe)

Go to the **NVIDIA Control Panel** > **3D Settings** > **Manage 3D Settings** > **Program Settings** to verify the settings and change them as needed.

Note:

For Microsoft Windows 11 and newer builds of Windows 10, the NVIDIA Control Panel might indicate the following:



Click the provided link and ensure that the two Ansys executables specified above are added to the list under *Graphics performance preference* and that the **High performance** option is specified. You can browse the \AnsysEM\v*xxx*\Win64 folder, typically under C:\Program Files, to add the two Ansys executables to the list.

Graphics Card Recommendations

For details about graphics card recommendations and tested cards, refer to the following PDF documents listed on the Ansys Platform Support website:

- Ansys 2024 R2 - Graphics Cards Tested
- Ansys 2024 R2 - GPU Accelerator Capabilities

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.

Additional Resources

Ansys Website:	https://www.ansys.com
Downloads:	Software downloads (including Ansys License Manager) and installation-related documentation are available at the Ansys customer portal, https://support.ansys.com
Online Technical Support:	Available through the Ansys customer portal: https://support.ansys.com Under **Training & Support**, select from tutorials and training materials or click **Support Contacts** for contact information.
Product Documentation:	The online help is accessible from any Electronics product's **Help** menu. Information about the release can be found in the *What's New* documents. PDF versions of the documentation can be found in the online

	help, or downloaded via the **Downloads** link on the customer portal: https://support.ansys.com

2 - Installing Ansys Electromagnetics Suite Software

Installing Ansys Electromagnetics Suite on a Windows operating system generally involves the following steps:

1. Ensuring the machine meets installation prerequisites.

   See: Installation Prerequisites (Page 1-1).

2. Installing the software.
3. Setting up a license server, if prompted to do so.

Installing the Software

To install Ansys Electromagnetics software:

1. Log in as an administrator, or using an account with administrator privileges.
2. Launch the **Ansys Electromagnetics Installation Shell**:
   - If you have a physical disc, it will launch automatically. If it does not, locate the drive and launch autorun.exe.
   - If you downloaded a digital copy, unzip the download and launch autorun.exe.
3. Click **Install Electromagnetics Suite**.

   The Installation Wizard 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**.

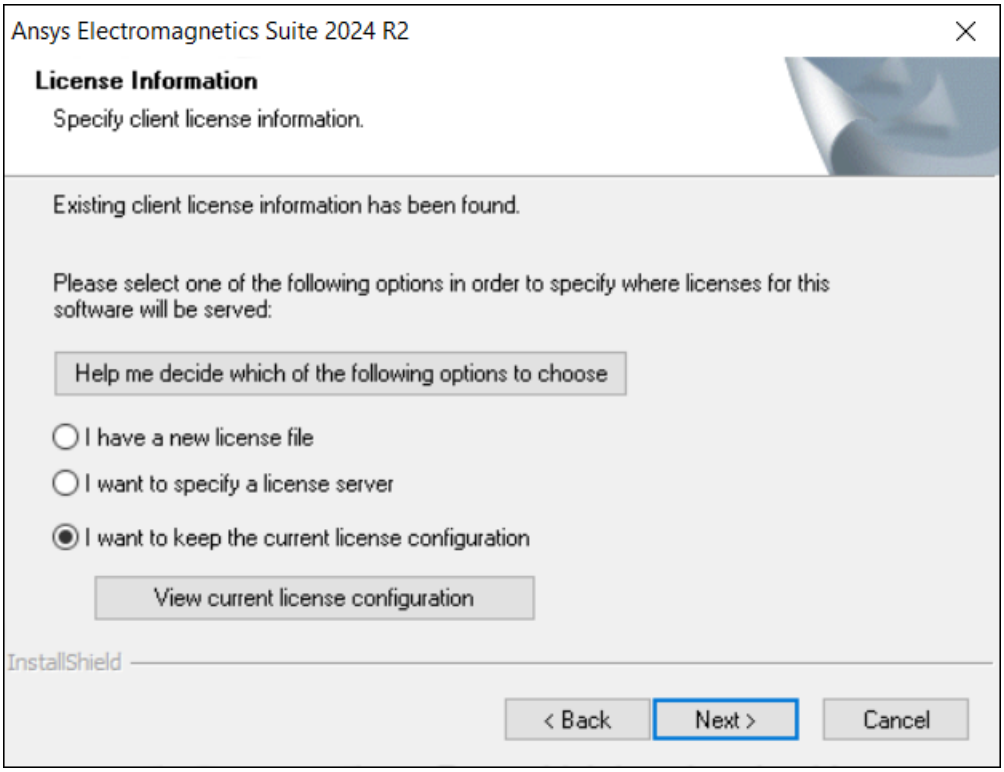
   Depending on the software and version being installed, you may see one or both of the following screens before the **Select Program Folder** screen: the **Choose Default Temp Directory Location** screen and/or the **Choose Library Option** screen.

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**.
11. 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**.

12. Select one of the following three license options:
    - **I have a new license file** – Select this option if you recently received a license file from Ansys.
      a. Click **Next**.
      b. Browse and select a license file.
      c. Click **Open**.
      d. Click **Next**.
      e. If prompted, enter the name of the license server. Select or clear the **Specify TCP/IP Port** option. Specifying a TCP/IP Port means that the software *only* looks to that port for licenses. The default port for Ansys License Manager is 1055.
      f. Click **Next**.

- **I want to specify a license server** – Select this option if you are using a counted license and already know the machine on which the Ansys License Manager is/will be installed.
  a. Click **Next**.
  b. Select whether you want to use a **Single** server or **Redundant** servers, and enter server names.

     A **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 TCP/IP Port** option. Specifying a TCP/IP Port means that the software *only* looks to that port for licenses. For **Redundant** servers, you must enter a port. The default port for Ansys License Manager is 1055.
  d. Click **Next**.
- **I want to keep the current license configuration** – Select this option if you already have a license file for a previous version of Ansys software and want to keep the same configuration.
  a. Click **Next**.

> **Note:**
>
> If you want to change the license server after installation, modify the ansyslmd.ini file in AnsysEM/Shared Files/Licensing.
>
> See: Updating Software and Licenses (Page 4-1).

13. 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** field.
    - Select an item from the **Existing Folders** list.
    - Type a new name for the folder in the **Program Folder** field.

    > **Note:**
    >
    > The program folder, which appears on the Windows **Start** menu, is a group of related software that includes executable files and other components. The default programs folder is **Ansys EM Suite 20*yy* R*r***, where *yy* is the last two digits of the version year, and *r* is the minor release number.

14. Click **Next**.

    The *Review settings before copying files* screen appears.

15. 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.

16. 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**.
17. 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.

    For details, see: Installing RSM (Page 3-4).

18. Click **Finish**.

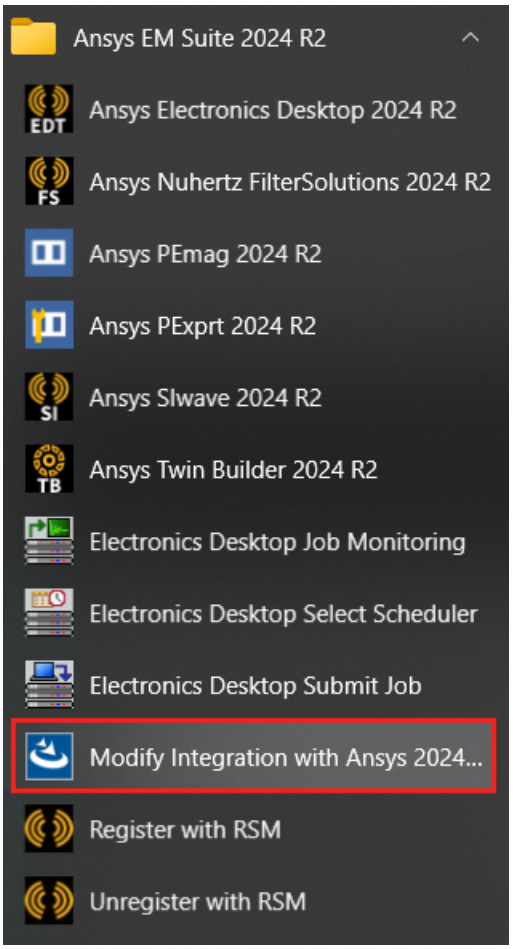
    The initial setup Installation Shell reappears.

19. Optionally, install any of the following additional software:
    - Distributed Simulation Technologies, which includes MPI and RSM.
    - Layout Integrations for ECAD.
20. Click **Exit** to terminate the Ansys Electromagnetics Installation Shell

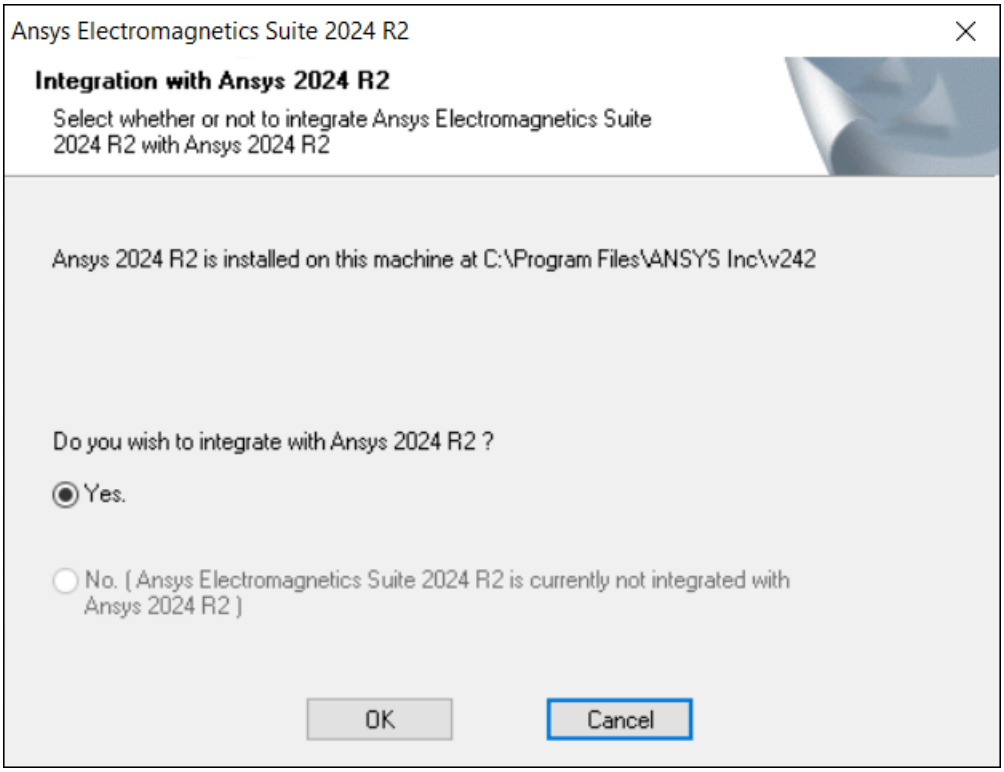
Integrating Ansys EM Suite with Ansys (*Workbench*)

You can integrate the **Ansys Electromagnetics Suite** with the **Ansys** software (*Workbench*) if both products have been installed on the same system and are the same version. The instructions are as follows:

1. From the Start menu, select **Ansys EM Suite 2024 R2** > **Modify Integration with Ansys 2024 R2**.

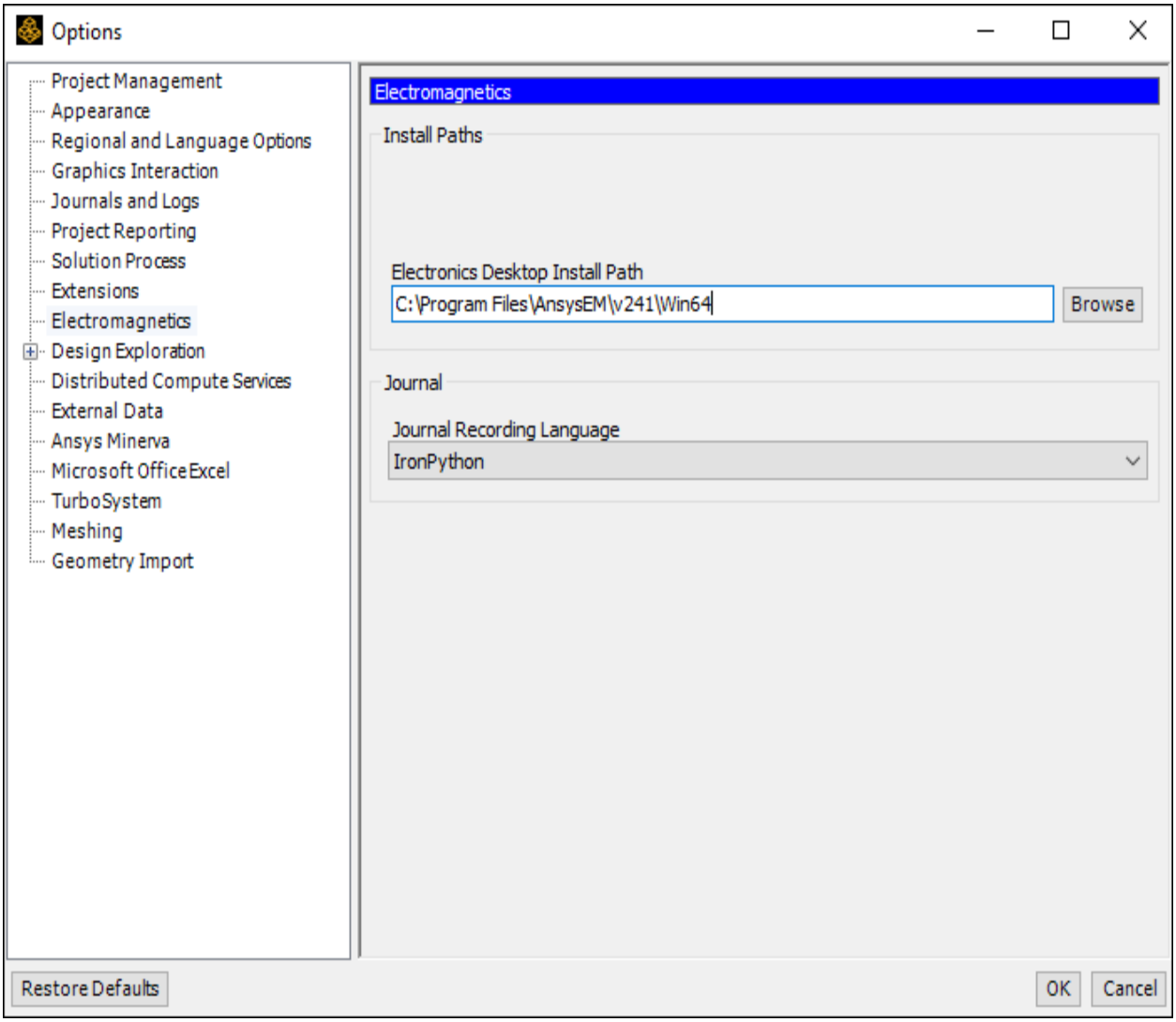


2. In the **Integration with Ansys 2024 R2** dialog box, ensure that the **Yes** radio button is selected.



3. Click **OK** to complete the integration process.
4. Click **Finish** to close the information dialog box.

> **Note:**
>
> You can confirm that Ansys is aware of the Ansys Electromagnetics integrated applications via Ansys Workbench. From Workbench, the *Options* dialog box (**Tools** > **Options** > **Electromagnetics**) shows the path to the *Electronics Desktop Install Path*.



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 in the same amount of time. This section describes the configuration and requirements for using additional resources.

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.

> **Note:**
>
> For additional information about high performance computing not in this guide, see the *Ansys Electromagnetics HPC Administrator's Guide*.

Distributing Analysis to Multiple Hosts

If an analysis is distributed to multiple hosts, then the Ansys Electromagnetics Suite must generally be installed at the same path on all hosts used for the analysis. This may be a shared network installation that is accessible using the same path on all analysis hosts (such as \\servername\sharename\AnsysEM\v242\Win64).

> **Important:**
>
> If this approach is used, then there *must not* be any local installations of Ansys Electromagnetics Suite on any of the hosts with the same major and minor version numbers. Installing a service pack does not affect the major and minor version numbers.

Alternatively, a separate, local installation on each host may also be used. In this case, the *path name* of the installation directory (such as C:\Program Files\AnsysEM\v242) must be the same on all hosts, and all installations must include the same service pack or packs.

A distributed analysis that does not use MPI distribution does not need to meet the above requirement. Because MPI is used for all distribution types except for variations, MPI will not be used only for the following cases:

- If the design type does not support the Auto option and does not allow distribution types other than distribution of variations, then MPI will not be used for distribution. This currently includes the Circuit Design, Circuit Netlist, EMIT, RMxprt, Maxwell Circuit, Twin

Builder, and Mechanical design types.

- If the Auto option is not selected, and no distribution type other than Variations is enabled, then MPI will not be used.

For these cases, Ansys Electromagnetics Suite may be installed at different locations on different hosts. These locations must be registered with the RSM Service running on each host.

Distributing Analysis Across Platforms

In general, all analysis hosts should be on the same platform. That is, either all analysis hosts should be Windows hosts or all analysis hosts should be Linux hosts. One additional configuration is supported, with limitations: Ansys Electronics Desktop may run on a Windows host, with solvers running on Windows or Linux hosts. This configuration is supported except for cases in which 1) auto mode is enabled and 2) variations are distributed. If auto mode is disabled or if all variations are analyzed sequentially, then this mode is enabled.

For this type of analysis, the machine list specified in the command line (for batch jobs) or the machines specified in the UI (for interactive jobs) should only include the Linux solver machines. This configuration is not supported in a scheduler environment. RSM must be installed and running on all of the Linux hosts, and the software must be registered with the RSM Service on each of the Linux hosts. If the analysis uses MPI distribution, then the installation path must be the same on all of the Linux hosts; this is the same requirement as for other configurations.

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 Spectrum 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, provides 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, 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 (Page 3-15). 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 (Page 3-15).

When Is RSM Needed?

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.

If you use RSM with MPI tight integraton you must set the environmental variable ANSYS_EM_EXEC_DIR,

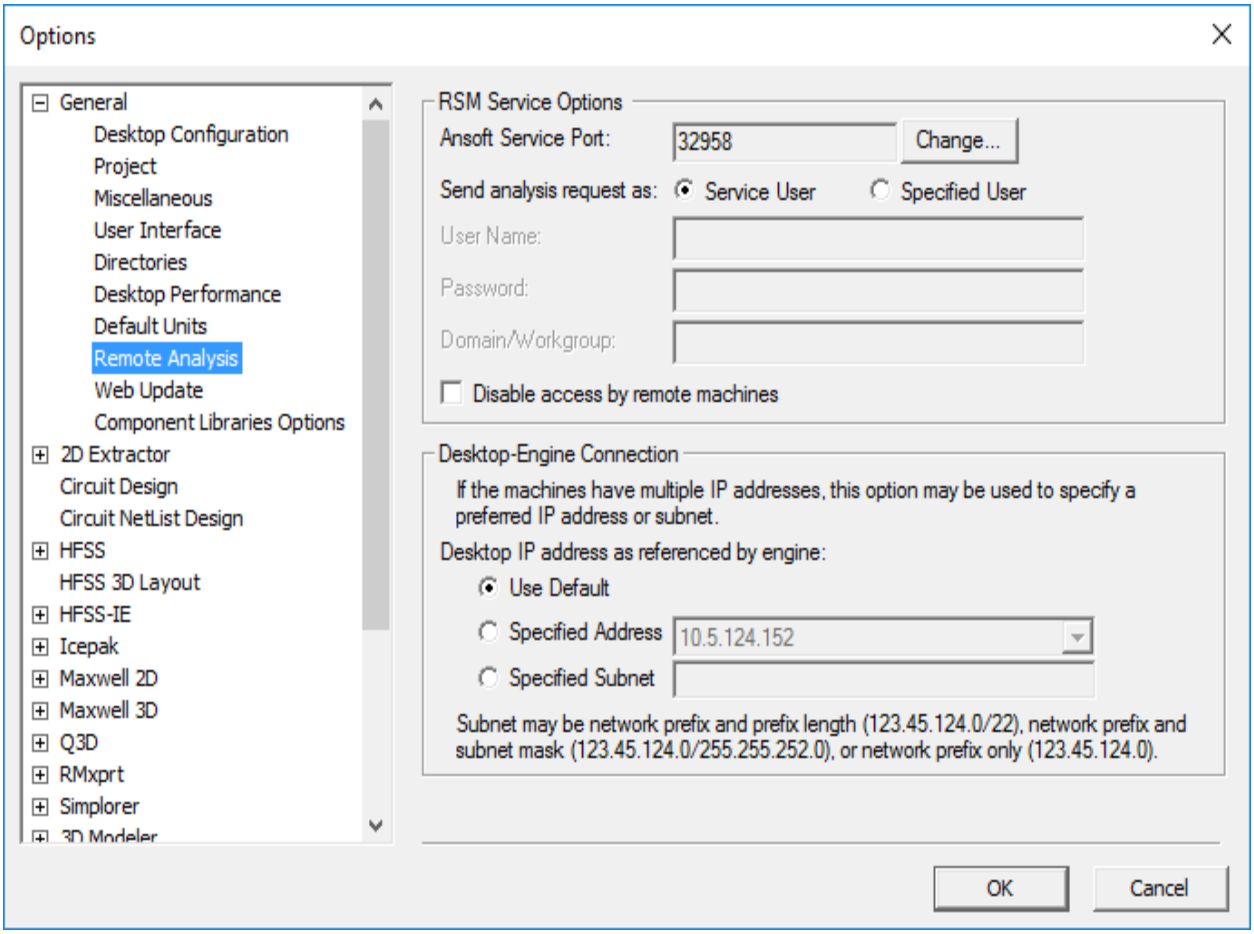
Set this variable to the same value with ANSYS_EM_INSTALL_DIR (not the RSM service path). To ensure the RSM service can accurately locate the essential script files required for MPI integration, it is necessary to set this variable within the RSM service environment.

Enabling and Disabling Remote Access

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 or enable this access using the **Options**

dialog box (**Tools** > **Options** > **General Options** > **Remote Analysis**).

The **Disable access by remote machines** check box, in the **RSM Service Options** group box controls access. If this check box is deselected, as is the default setting, remote hosts can distribute jobs to the local host. If this check box is selected, remote hosts cannot distribute jobs to the local host.

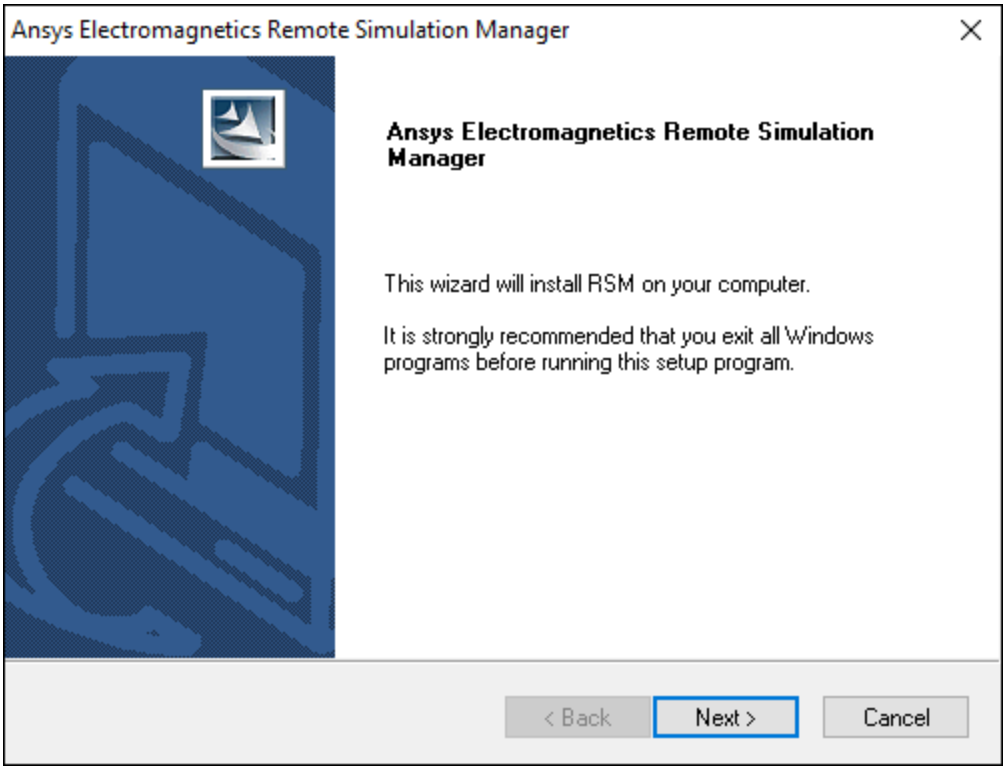


Installing RSM

To install Ansys Electromagnetics RSM software:

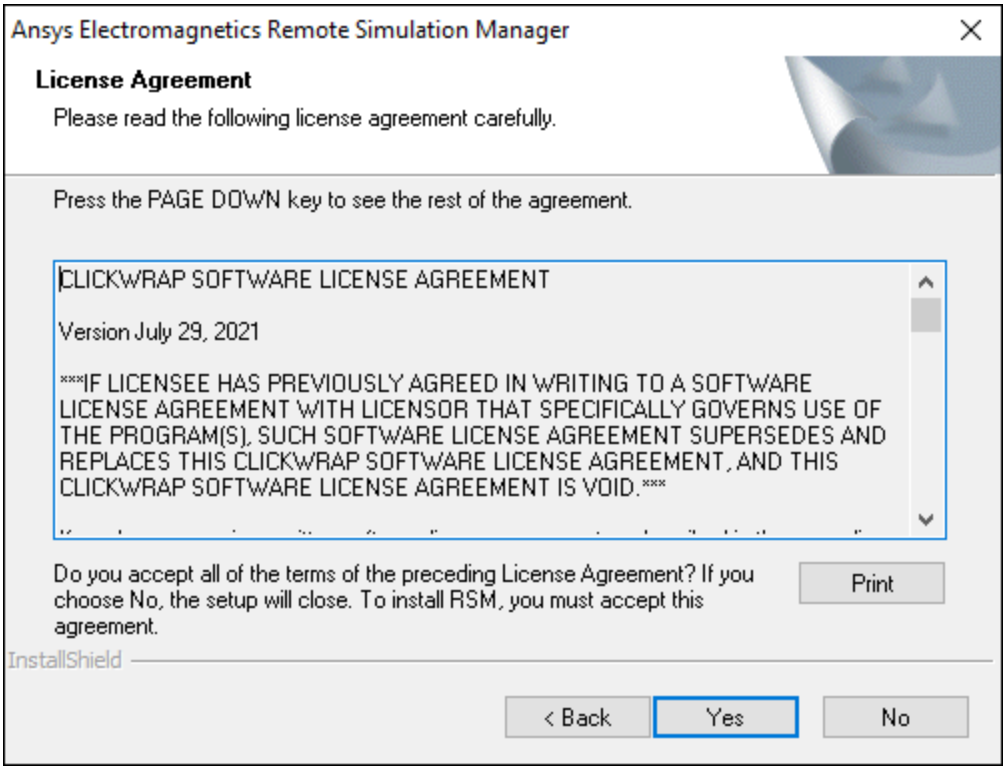
1. Launch the Remote Simulation Manager installer:
   - From the Ansys Electromagnetics installer, select **Install Distributed Simulation Technologies**.
   - Click **Install Electromagnetics RSM**.

   The installer appears.



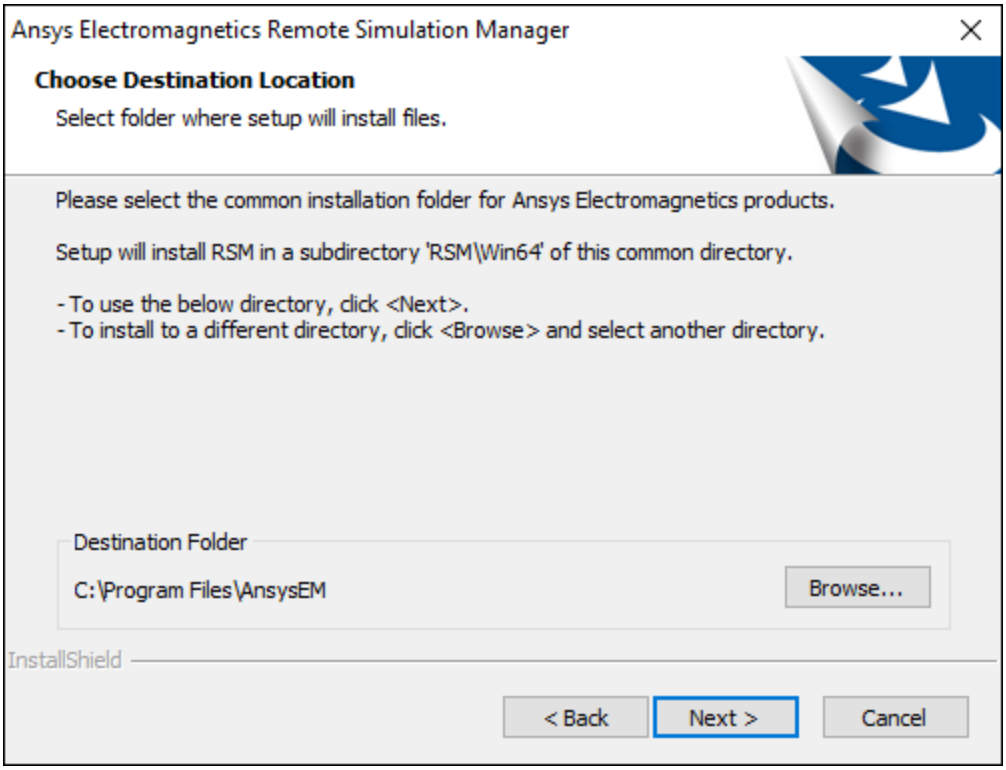
2. Click **Next**.

   A licensing agreement appears.



3. Read and agree to the license agreement. Click **Yes** to continue.

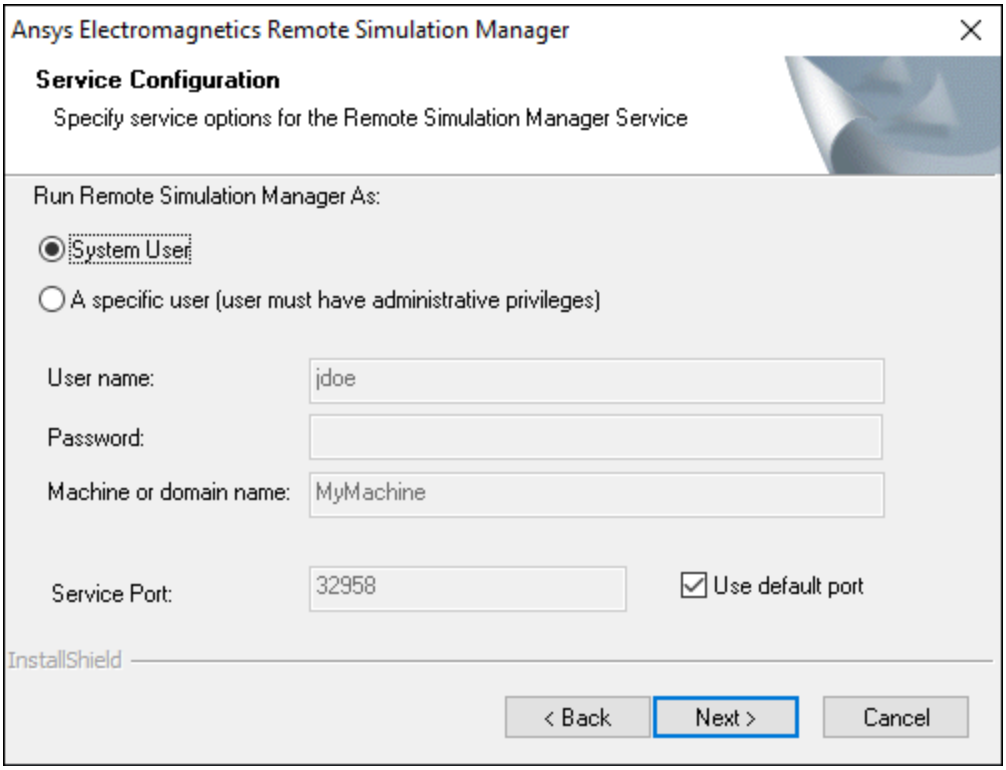
   The **Choose Destination Location** screen appears.



If you have other Ansys software installed, Ansys recommends using the same installation directory.

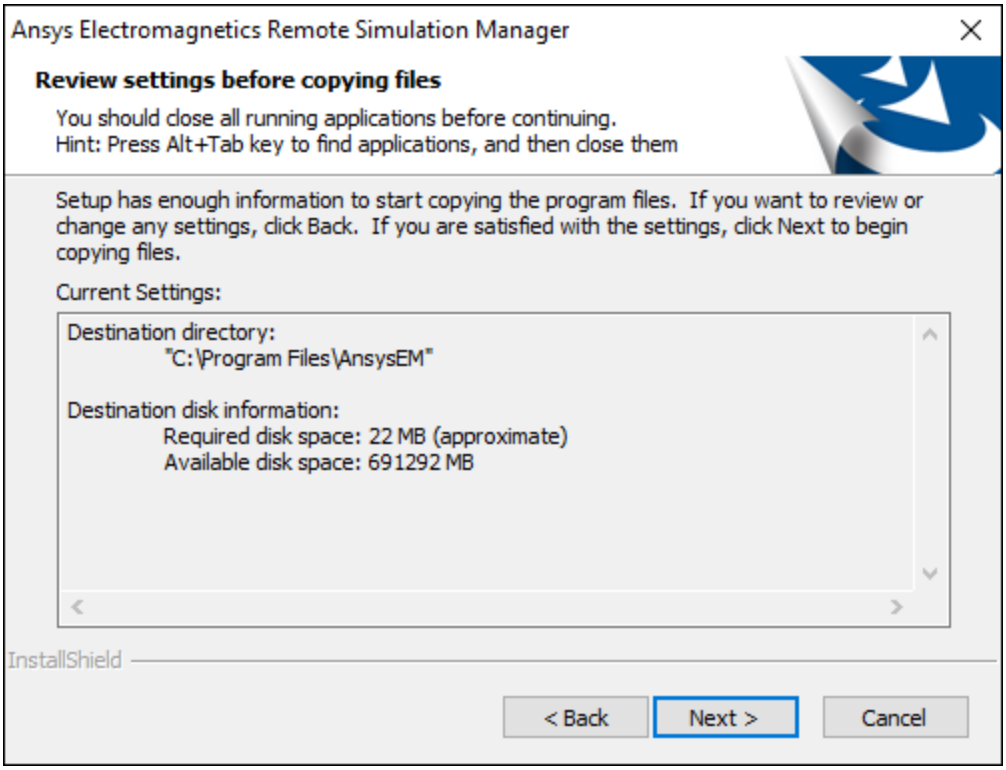
4. Select a destination folder. Click **Next** to continue.

   The **Service Configuration** screen appears.



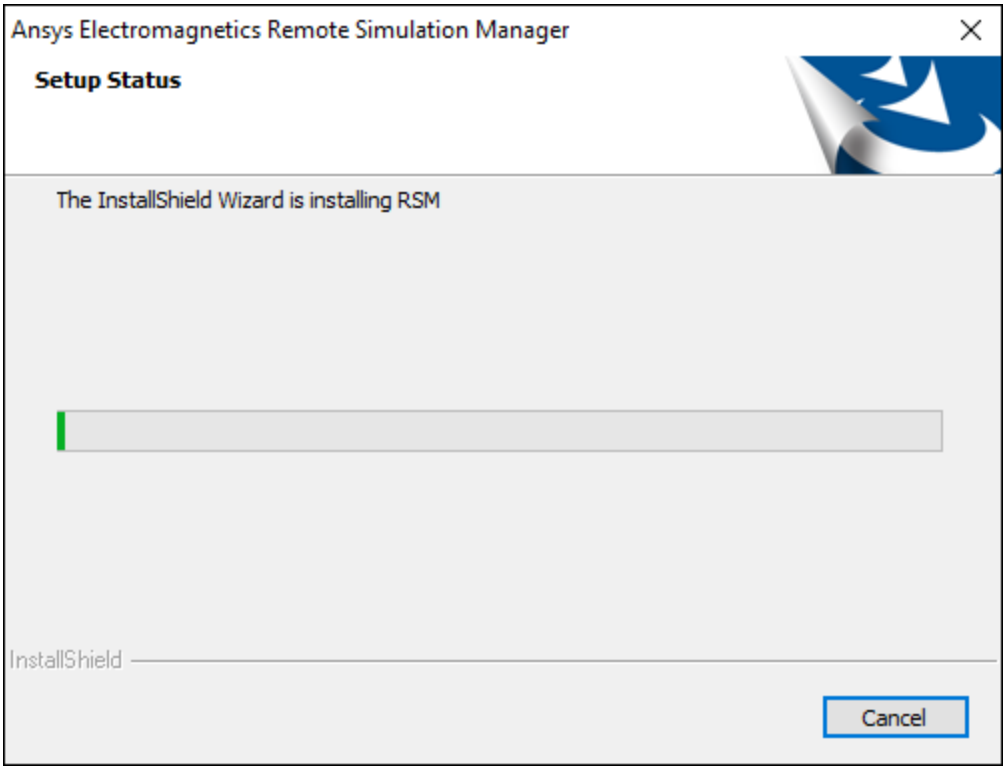
5. Select the **System User**, or specify a **Specific User** with administrative privileges. The selected user will run remote and distributed analyses when no user is specified in the product interface. Click **Next**.

   A review screen appears.

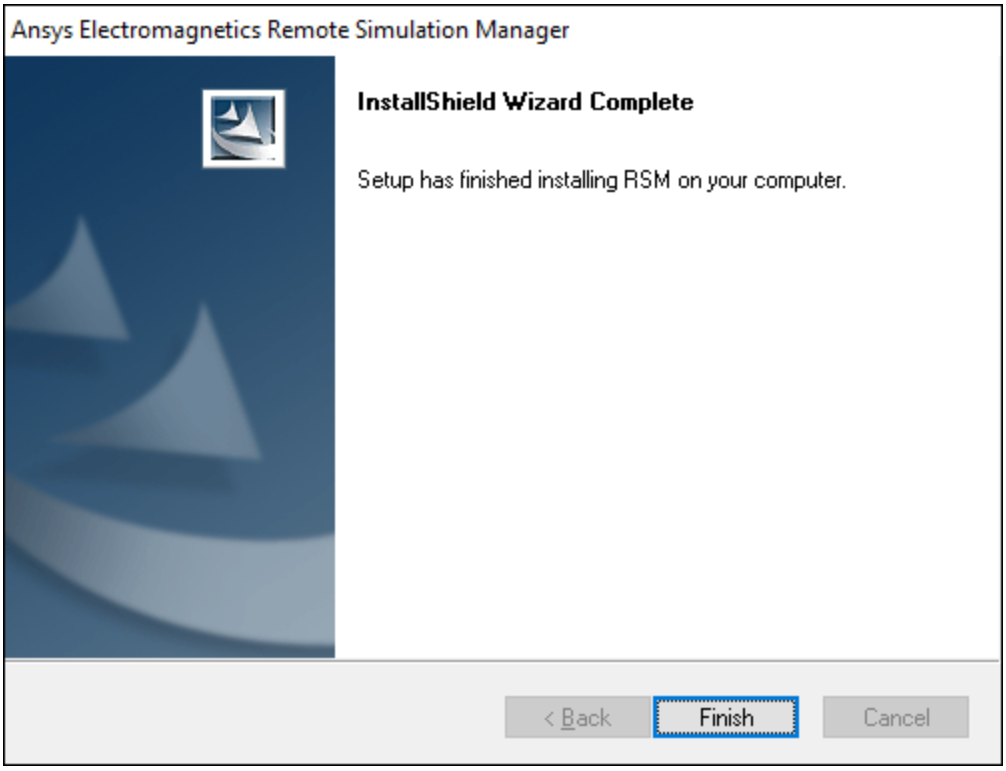


6. Review the settings. If they are correct, click **Next**.

   A progress bar displays installation progress.



When installation has finished, a message appears.



7. Click **Finish**. At the end of the installation, RSM is automatically started as a service on this machine.

Troubleshooting Installation

If you receive a message about read-only files, elect to overwrite them.

If you use RSM, it must be installed on all remote machines. In addition, the product engines must be registered on RSM. See below.

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, do one of the following:

- From Windows, select **Start** > **Ansys EM Suite 2024 R2** > **Register with RSM**.
- Run **RegisterEngineswithRSM.exe**, located in the installation directory. The command should be run from the installation path to be registered with RSM.

To un register an Ansys Electromagnetics product with RSM, do one of the following:

- From Windows, select **Start** > **Ansys EM Suite 2024 R2** > **Unregister with RSM**.
- Run **UnregisterEngineswithRSM.exe**, located in the installation directory.

Changing the RSM Listening Port

For Remote Analysis or Distributed Analsysis, processes may need to be started on multiple hosts. If Ansys Electronics Desktop needs to start a process on a remote host, the RSM service is used to start these remote processes. By default, the RSM service listens for socket connections from the Ansys Electronics Desktop on port 32958.

This section describes how to change the port number used by the RSM service.

To change the port number, both the RSM service and Ansys Electronics Desktop must be configured to use the new port number. The same port number must be used for Ansys Electronics Desktop and for the RSM service process running on each host used for the analysis.

RSM Service Configuration

The RSM service port number is configured in the ansoftrsmservice.cfg configuration file. This configuration file is located in the platform specific subdirectory of the RSM installation directory.

In Windows, the default location of this directory is C:\Program Files\AnsysEM\RSM\Win64.

To modify the RSM service configuration:

1. Stop the service.
2. Modify the ansoftrsmservice.cfg configuration file.
3. Restart the service.

The beginning of the configuration file should appear as follows:

```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If there are additional lines between the following two lines, then they should not be modified:

```
$begin 'AnsoftCOMDaemon'
```

```
$begin 'Default:CommDetails'
```

To change the port number, modify the ListenPort setting within the single quotes from 32958 to the desired port number. The single quotes should not be removed or changed.

For previous versions of the software, the ansoftrsmservice.cfg file may not contain the lines:

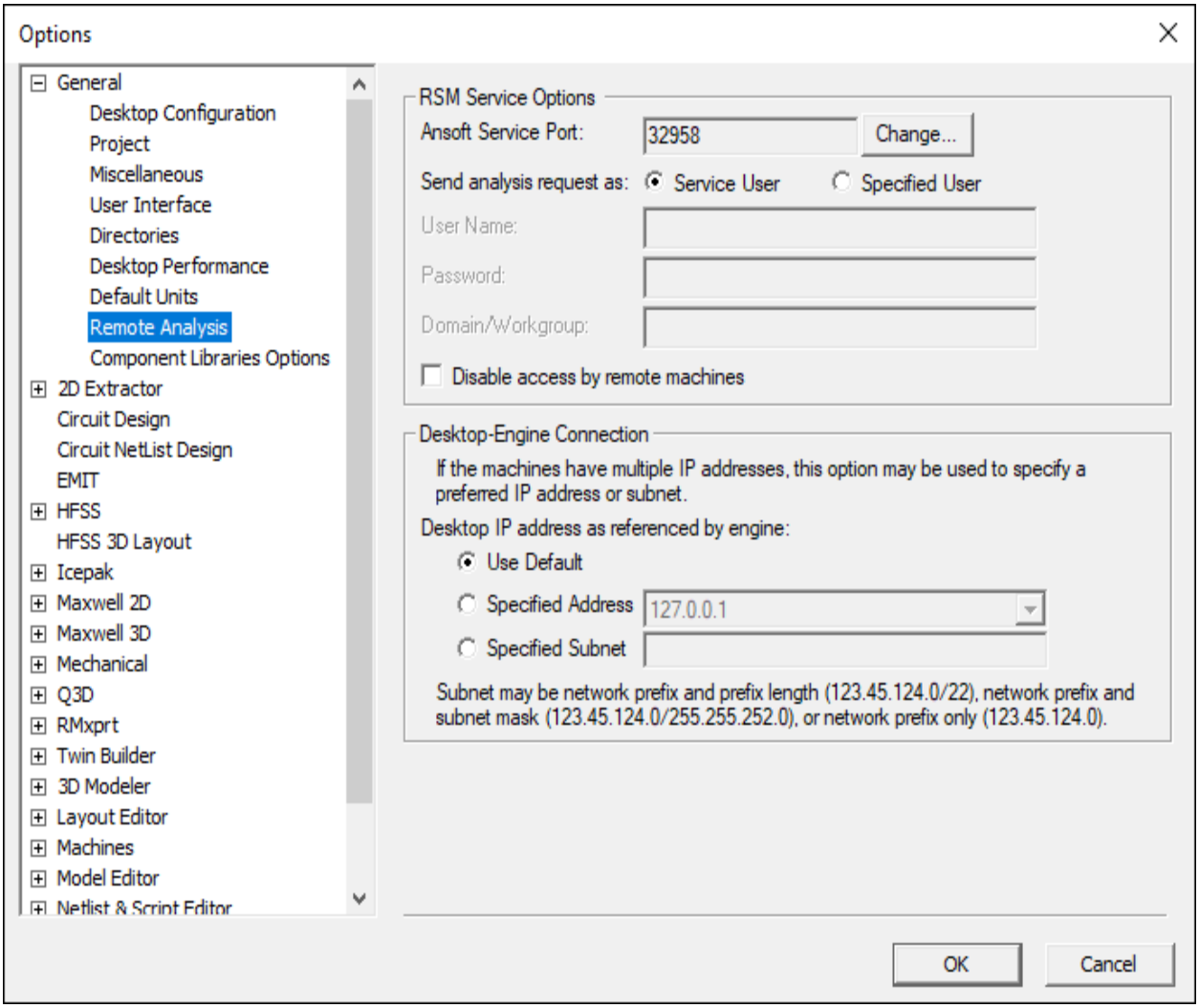
```
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

If these lines are not present, then add them to the ansoftrsmservice.cfg file after the first line of the file, and change the ListenPort to the desired port number. For example, to change the port number to 32000, the beginning of the ansoftrsmservice.cfg file should look like the following after the changes:

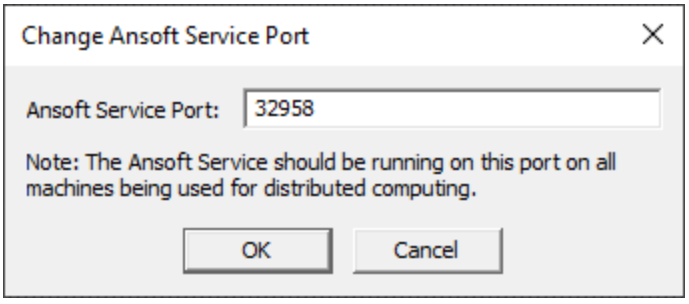
```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

Ansys Electromagnetics Desktop Configuration

The port number for connecting to the RSM service is configured using the **General Options** window (**Tools** > **Options** > **General Options**). From the left pane, select **Remote Analysis** to view **RSM Service Options**.



Click **Change** to open the **Change Ansoft Service Port** dialog box.



Enter a new port number and click **OK**.

Supported Job Schedulers

Supported job schedulers are listed at the Ansys Platform Support page:

https://www.ansys.com/Solutions/Solutions-by-Role/IT-Professionals/Platform-Support

Versions earlier than the latest tested version indicated may work and may have been tested in previous versions of the product. Newer versions may also work if backward compatibility is maintained by the scheduler.

This document covers the Windows HPC and IBM Spectrum LSF schedulers.

Custom Scheduler Integration

For supported 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, users (or scheduler vendors) can 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 Scripting section of the online help.

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 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.

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 Electromagnetics jobs have been met.

Before running these tests, the following steps should be completed:

- Ansys Electromagnetics product(s) should be installed.
- The cluster should be configured for running jobs, as described in Scheduler-Specific Configuration and Testing (Page 3-22).
- MPI should be installed and configured, as described in MPI Installation (Page 5-1).

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.

Background: Selecting a Scheduler

The test jobs in this section may be submitted to a cluster using the Electronics Desktop job submission GUI.

1. Launch Ansys Electronics Desktop on a host that is configured as a submission host for the cluster.
2. Click **Tools** > **Job Management** > **Select Scheduler**.

   The **Select Scheduler** window appears.

3. For a Windows HPC cluster, use the **Select scheduler** drop-down menu to select **Windows HPC**. In the **Head Node** field, type the name of a computer or click the ellipses (**...**) button to view a list of accessible computers in the network.

   For information about selecting other types of clusters (for example, LSF), consult the *HPC Administrator User's Guide*.

4. Click **Refresh** to verify the connection to the cluster.

   The **Scheduler Info** field updates with this information.

Test #1: Submitting and Running a 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 file Tee.aedt is needed for this test. This test uses Setup1, but it does not use the frequency sweep, Sweep1, so the frequency sweep must be disabled.

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.

Perform the following steps:

1. Copy the file Tee.aedt from <InstallationDirectory>/Examples/HFSS/RF Microwave subdirectory of the product installation directory to another directory.
2. Launch Ansys Electronics Desktop.
3. Open the copy of the project.
4. In the Project Manager, expand **TeeModel (Driven Model)** > **Analysis** > **Setup1**> **Sweep1**. If frequency sweep Sweep1 is enabled, right-click it and select **Disable Sweep**.
5. Click **Tools** > **Job Management** > **Submit Job**.

   The **Submit Job To** window appears, on the **Analysis Specification** tab.
6. Ensure the **Product path** contains a path that is valid for the cluster execution nodes.
7. In the **Analysis Setups** area, select **Single Setup**. Use the drop-down menu to select **TeeModel:Nominal:Setup1**.
8. Click the **Compute Resources** tab to view additional options.
9. Deselect **Use automatic settings**, and set the following:
   - For **Method**, use the drop-down menu to select **Number of Tasks and Cores**.
   - Set the **Total number of tasks** to 1.
   - Set the **Cores per distributed task** to 1.
   - Ensure that the **Limit number of tasks per node to:** check box is deselected.
10. Ignore any other settings, and click **Preview Submission**.

    A preview window appears. If no errors are found, the preview should display information on how the job will be submitted to the cluster. If an error appears, address the error before continuing. The preview should indicate that only one host is needed for the job. If

this is not the case, review the job resources specified on the **Compute Resources** tab and retry. If the job and task parameters are correct, close preview window and continue.

11. Click **Submit Job**.

    If an error occurs on job submission, address the error before submitting the job again. If you are prompted for a password, enter it and continue. If possible, monitor the job until it finishes.

12. Save and close the project.

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 path name of the product must be accessible on the cluster execution node(s) where the job runs, and the user must have permission to run the product executables.
- The path name 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 temporary 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 box, 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 box.

Test #2: Submitting and Running a 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 uses the Tee.aedt example project, which is located in the Examples/HFSS/RF Microwave subdirectory of the product installation directory. This test uses Setup1, and it uses the frequency sweep Sweep1, so frequency sweep must be enabled.

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.

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.
- Communication between job processes running on different cluster hosts allocated to the job is not blocked.

Perform the following steps:

1. Copy the file Tee.aedt from <InstallationDirectory>/Examples/HFSS/RF Microwave to another directory.
2. Launch Ansys Electronics Desktop.
3. Open Tee.aedt.
4. In the Project Manager, expand **TeeModel (Driven Model)** > **Analysis** > **Setup1** > **Sweep1**. If frequency sweep Sweep1 is disabled, right click it and select **Enable Sweep**.
5. Click **Tools** > **Job Management** > **Submit Job**.

   The **Submit Job To** window appears, on the **Analysis Specification** tab.
6. Ensure the **Product path** contains a path that is valid for the cluster execution nodes.
7. In the **Analysis Setups** area, select **Single Setup**. Use the drop-down menu to select **TeeModel:Nominal:Setup1**.
8. Click the **Compute Resources** tab to view additional options.
9. Deselect **Use automatic settings**, and set the following:
   - For **Method**, use the drop-down menu to select **Number of Tasks and Cores**.
   - Set the **Total number of tasks** to 2 * N.
   - Set the **Cores per distributed task** to 1.
   - Ensure that the **Limit number of tasks per node to:** check box is deselected.
   - In the **Job distribution** group box, press the **Modify** button to access the **Job Distribution** dialog box.
   - In the **Enable Distribution Types** section, deselect the **Use defaults** check box, and select only the **Frequencies** check box.
   - In the **Distribution levels** section, select the **Single level only** radio button.
   - Click **OK** to close the **Job Distribution** dialog box.
10. For the schedulers:
    - Click the ellipses (**...**) button to open the **Compute Resource Selection Parameters** dialog box.
    - For **MinCoresPerNode** , select **Specify Parameter** and set the value to N.
    - For **MaxCoresPerNode**, select **Specify Parameter** and set the value to N.
    - Click **OK** to close the **Compute Resource Selection Parameters** dialog box.

11. Ignore any other settings, and click **Preview Submission**.

    A preview window appears. If no errors are found, the preview should display information on how the job will be submitted to the cluster. If an error appears, address the error before continuing. This preview 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, review the job resources specified on the **Compute Resources** tab and retry. If the job and task parameters are correct, close the preview window and continue.

12. Click **Submit Job**.

    If an error occurs on job submission, address the error before submitting the job again. If you are prompted for a password, enter it and continue. If possible, monitor the job until it finishes.

13. Save and close the project.

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.

Test #3: Submitting and Running an 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 file differential_stripline.aedt is needed for this test. This test uses Setup1, but does not use the frequency sweep, DC_10GHz. The frequency sweep must be disabled.

Determine which MPI Vendor will be used for MPI jobs, Intel MPI or Microsoft 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 (Microsoft) is used. Intel MPI is denoted by the string “Intel.” Microsoft MPI is denoted by the string “Microsoft.” If both MPI Vendors may be needed by cluster users, then this test should be run twice, once with each MPI Vendor specified. Consult MPI Installation (Page 5-1) to determine whether 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.

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

- MPI is installed and configured on the cluster hosts.
- User passwords are registered with MPI on the cluster hosts.
- MPI prerequisites are installed on the cluster hosts.

Perform the following steps:

1. Copy the file differential_stripline.aedt from <InstallationDirectory>/Examples/HFSS/Transmission Lines to another directory.
2. Launch Ansys Electronics Desktop.
3. Open differential_stripline.aedt.
4. In the Project Manager window, expand **differential_stripline** > **Differential Stripline (Driven Terminal)** > **Analysis** > **GHz** > **DC_10GHz**. If frequency sweep DC_10GHz is enabled, right-click it and select **Disable Sweep**.
5. Click **Tools** > **Job Management** > **Submit Job**.

   The **Submit Job To** window appears, on the **Analysis Specification** tab.
6. Ensure the **Product path** contains a path that is valid for the cluster execution nodes.
7. In the **Analysis Setups** area, select **Single Setup**. Use the drop-down menu to select **Stripline:Nominal:3GHz**.
8. Under **Analysis Options**, set the **HFSS/MPIVendor** batchoption to the desired MPI Vendor, either **Intel** or **Microsoft**. If this batchoption is not present, click **Add** to open the **Add Batchoption** dialog box. Locate **HFSS/MPIVendor** in the list, select it, and click **Add**.
9. Click the **Compute Resources** tab to view additional options.
10. Deselect **Use automatic settings**, and set the following:
    - For **Method**, use the drop-down menu to select **Number of Tasks and Cores**.
    - Set the **Total number of tasks** to 2 * N.
    - Set the **Cores per distributed task** to 1.
    - Ensure that the **Limit number of tasks per node to:** check box is deselected.
    - In the **Job distribution** group box, press the **Modify** button to access the **Job Distribution** dialog box.
    - In the **Enable Distribution Types** section, deselect the **Use defaults** check box, and select only the **Direct Solver Memory** check box.
    - In the **Distribution levels** section, select the **Single level only** radio button.
    - Click **OK** to close the **Job Distribution** dialog box.

11. For the schedulers:
    - Click the ellipses (**...**) button to open the **Compute Resource Selection Parameters** dialog box.
    - For **MinCoresPerNode** , select **Specify Parameter** and set the value to N.
    - For **MaxCoresPerNode**, select **Specify Parameter** and set the value to N.
    - Click **OK** to close the **Compute Resource Selection Parameters** dialog box.
12. Ignore any other settings, and click **Preview Submission**.

    A preview window appears. If no errors are found, the preview should display information on how the job will be submitted to the cluster. If an error appears, address the error before continuing. This preview 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, review the job resources specified on the **Compute Resources** tab and retry. If the job and task parameters are correct, close the preview window and continue.

13. Click **Submit Job**.

    If an error occurs on job submission, address the error before submitting the job again. If you are prompted for a password, enter it and continue. If possible, monitor the job until it finishes.

14. Save and close the project.

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 “Microsoft”) is specified using a batchoption setting in the **Submit Job To:** dialog box. The controls for specifying batchoptions are located on the **Analysis Specification** tab, under **Analysis Options**.
- Verify that the specified MPI Vendor is installed (if needed) and configured correctly for the user submitting the job, as described in MPI Installation (Page 5-1). 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

As a companion to the scheduler tests above, the sections below describe configuration requirements for the Windows HPC and IBM Spectrum LSF schedulers.

For integration with Linux schedulers, consult the *Ansys Electromagnetics Suite 2024 R2 Unix/Linux Installation Guide*.

For integration with third party or custom schedulers, consult the Ansys Electronics online help.

Windows HPC Scheduler

Job Submission Host Configuration

In order to submit jobs to a Windows HPC cluster from the Ansys Electronics Desktop application, Windows HPC Client Utilities must be installed on the job submission host. The recommended version is Microsoft (R) HPC Pack 2019. Only the Client Components are required on the job submission host.

The Bin subdirectory of the Windows HPC Client Utilities installation directory (typically "C:\Program Files\Microsoft HPC Pack 2019\Bin") must be included in the PATH environment variable or in the ANSYSEM_WINHPC_CLIENT_UTILS_PATH environment variable. If this folder is not added to the PATH environment variable during the installation of the Client Utilities, or if it has been removed from the PATH environment variable, then it may need to be manually added to one of these environment variables. The ANSYSEM_WINHPC_CLIENT_UTILS_PATH environment variable may be used instead of the PATH environment variable to prevent the PATH environment variable from becoming too long. The ANSYSEM_WINHPC_CLIENT_UTILS_PATH environment variable may contain multiple directories separated by semicolon (;) characters, just like the PATH environment variable.

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 process 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.

- Windows HPC Server 2008 R2 (or later) 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** window 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 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 Electronics Desktop's **Monitor Job** window, 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.

If the MPI Vendor is “Intel,” then installation of Intel MPI on all cluster nodes is required as described in MPI Installation (Page 5-1).

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 (“Microsoft”) is specified via a batchoption 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 MPI Installation (Page 5-1). Note that 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 Spectrum LSF Scheduler

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 box, 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 * 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 ansoftrsm process on the other host. The ansoftrsm process is used to start any additional processes needed by the job on the other host. In some cases, multiple ansoftrsm 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 blaunch command is able to start the ansoftrsm 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.

If the MPI Vendor is “Intel,” then installation of Intel MPI on all cluster nodes is required as described in 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”) is specified via a batchoption 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 MPI Installation (Page 5-1). 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 - 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** – displays only if the version you are installing is newer than the current installation.)

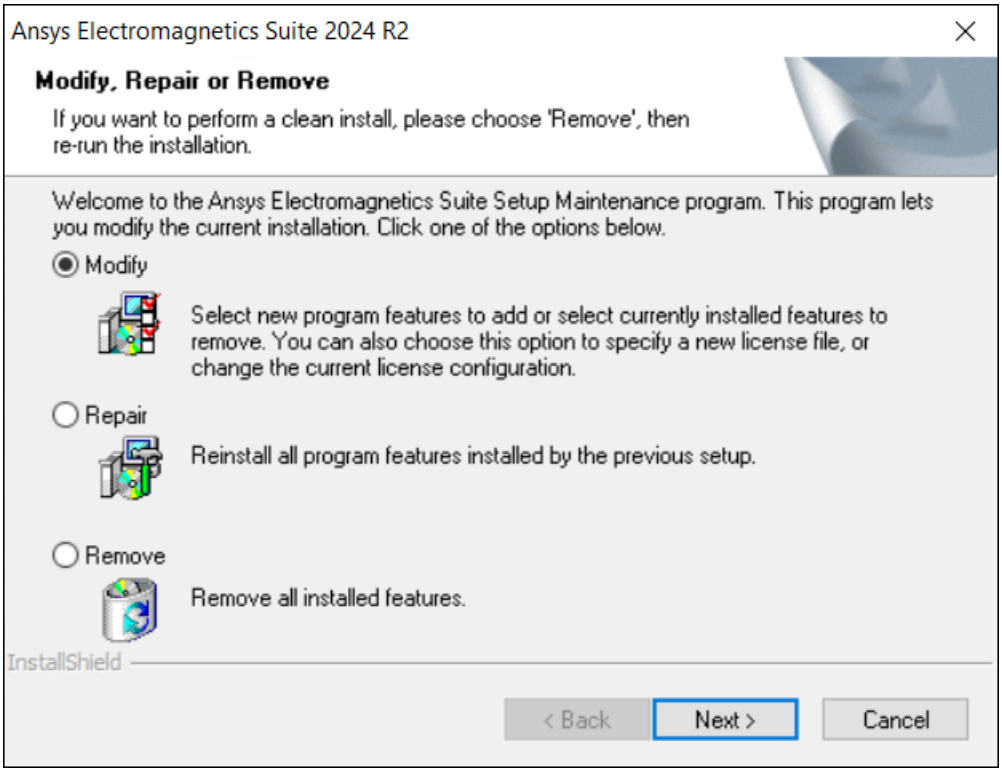
To start the Ansys Electromagnetics Suite Installation Wizard in **Maintenance Mode**:

1. From Windows, access the list of **Programs and Features**.

   Steps for doing so vary, depending on your version of Windows. Consult Microsoft Windows help.

2. From the list of applications, click the applicable Ansys Electromagnetics product.
3. Click **Change**.

   The **Modify, Repair, or Remove** window appears.



4. Select the desired option from: **Modify**, **Repair**, **Remove**, and **Update/Remove** (if available).

Updating a License

There may be times when you need to update a license. Most often, this happens when you change license servers.

When you receive a new license file:

If you receive an updated purchased license file, you need only update the license file using Ansys License Manager.

When you change license servers:

1. Install Ansys License Manager on the new server machine, and add the new license file.
2. Run each Ansys Electromagnetics Suite installation in Maintenance Mode, and specify the names of the new servers.

When changing the name of the license server:

When changing the name of the license server used by the software, you need to modify the client installation:

1. Run Maintenance Mode.
2. Click **Modify**.
3. Click **Next** on each screen to move through the steps.
4. On the **License Information** screen, select the new license file.
   - If you received a new file from Ansys evaluation or purchase, select **I have a new license file**.
   - If you need to change the name of the machine where Ansys License Manager is installed, select **I want to specify a license server**.

   See: Installing the Software (Page 2-1).

Specifying License Server Machines via `ansyslmd.ini` or Environment Variables

Specifying the license server during installation creates or updates the `ansyslmd.ini` file located in the licensing directory. Entries in this file tell Ansys products which license server(s) to query to find a valid license. This allows all users at your site to use this setting without having to individually set the **ANSYSLMD_LICENSE_FILE** or **ANSYSLI_SERVERS** environment variables to specify the license server machine(s). It also eliminates the need to have a copy of the license file on every system at your site.

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

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

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

On a single server:

`ANSYSLI_SERVERS=<`*`ansysliport`*`>@<`*`host`*`>`

`SERVER=<`*`flexnetport`*`>@<`*`host`*

For multiple single servers:

Each server should have its own `ANSYSLI_SERVERS=` and `SERVER=` lines.

On redundant (triad) servers on Windows platforms:

Windows uses semicolons (;) to separate servers:

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

```
SERVER=<flexnetport>@<host1>;<flexnetport>@<host2>;
<flexnetport>@<host3>
```

> **Important:**
>
> Do not use commas as separators. Use semicolons (;) on Windows machines, and colons (:) on Linux machines.

The Ansys Licensing Interconnect default port number is 2325 and the Ansys FlexNet default port number is 1055. The host is the license server hostname.

For example, if the license server name is alpha1:

```
ANSYSLI_SERVERS=2325@alpha1
```

```
SERVER=1055@alpha1
```

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

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

Overriding the ansyslmd.ini File:

If you want to override the server specification settings in the `ansyslmd.ini` file, you can do so by setting the **ANSYSLMD_LICENSE_FILE** and **ANSYSLI_SERVERS** environment variables on individual machines. These environment variables are useful if you want to temporarily point to a different license server machine without disrupting the machine's configuration.

- **ANSYSLI_SERVERS** – specifies the Licensing Interconnect port number.
- **ANSYSLMD_LICENSE_FILE** – specifies the FlexPort port number.

If the **ANSYSLMD_LICENSE_FILE** environment variable is set but the **ANSYSLI_SERVERS** environment variable is not set, the same server machines will be used to specify the Licensing Interconnect but the port number will be replaced by the Licensing Interconnect default port of 2325.

When both variables are set, **ANSYSLMD_LICENSE_FILE** explicitly defines the FlexNet servers while **ANSYSLI_SERVERS** explicitly defines the Licensing Interconnect servers.

If you set the **ANSYSLMD_LICENSE_FILE** or **ANSYSLI_SERVERS** environment variables on a three-server (redundant) system, specify all three systems in the same order as the SERVER lines are listed in the license file. If you specify only the primary server and it is down, you could see a “License Server Down” or “No License Found” message and the search for a license could fail. Join redundant or multiple single-server systems by separating the system names with semicolons on Windows systems.

5 - Message Passing Interface (MPI) Installation

The two supported Message Passing Interface (MPI) options in Ansys Electromagnetics Suite are Microsoft and Intel. MPI is not required if an analysis is not distributed, nor is it required for manual distribution of variations only. MPI is required for all other types of distribution, such as distribution of frequencies or domains and for auto distribution.

Intel MPI

For Intel MPI, user authorization must be configured to distribute an analysis using MPI. Intel MPI supports several authentication methods on Windows, including password-based authorization. Consult Intel MPI documentation on user authorization for additional information.

No additional MPI configuration is required for distribution using Intel MPI on the local host.

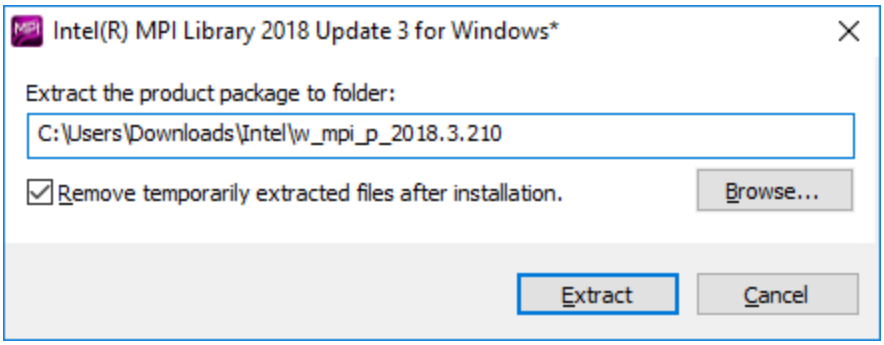
For distribution using Intel MPI to multiple hosts, Intel MPI must be installed on all of the hosts and the Hydra Service must be running on all of the hosts in addition to configuring user authorization on all of the hosts.

Installing Intel MPI

To install Intel MPI, perform the following steps:

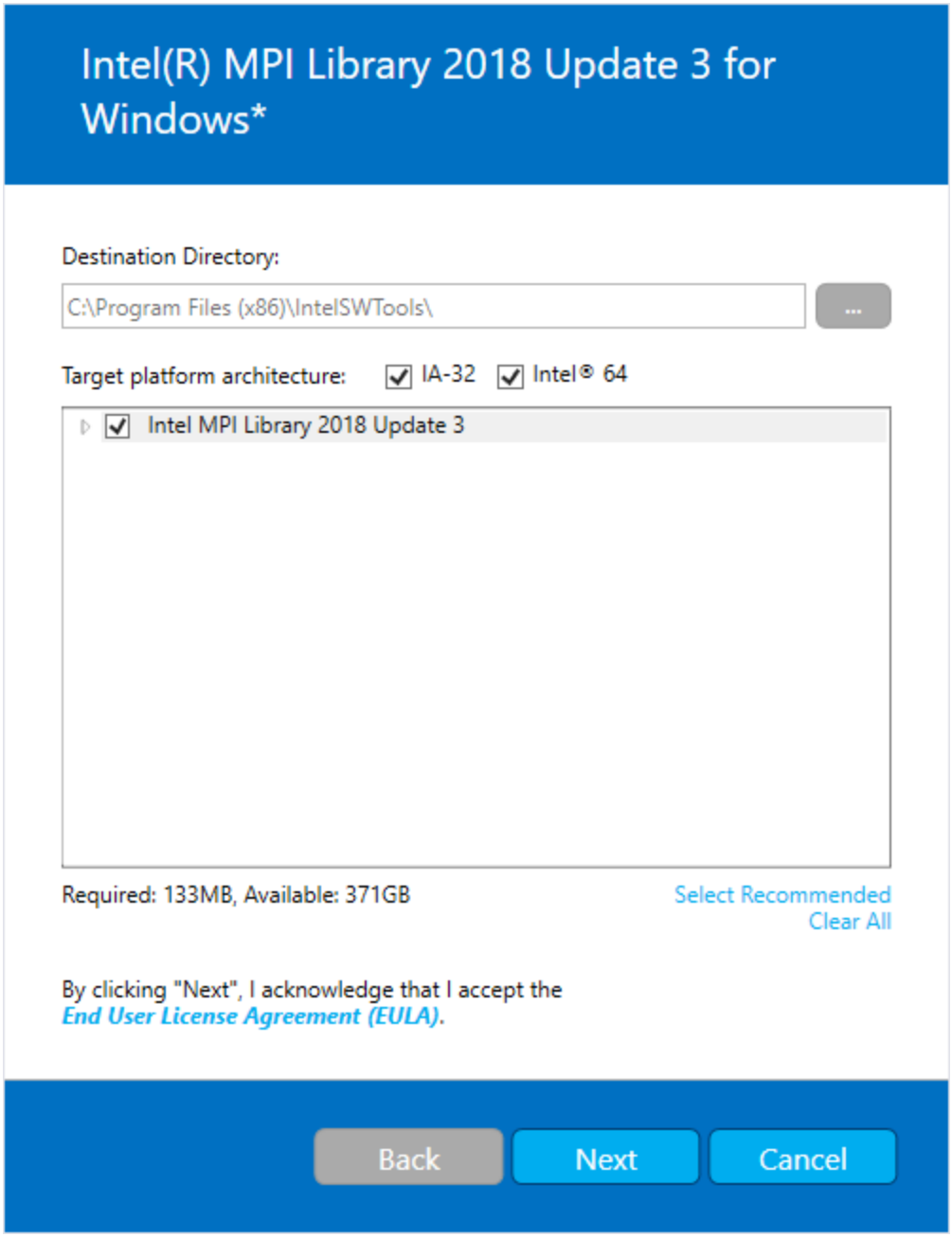
1. Launch the **Ansys Electromagnetics Installation Shell**:
   - If you have a physical disc, it will launch automatically. If it does not, locate the drive and launch autorun.exe.
   - If you downloaded a digital copy, unzip the download and launch autorun.exe.
2. Click **Install Distributed Simulation Technologies**.
3. Under **MPI**, click **Install Intel MPI**.

   The **Intel MPI Library...** dialog box appears.



4. Specify a directory, and click **Extract**.

   The **Intel MPI Library Update** window appears.



5. Click **Next** to install the software.

   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. Launch the **Intel MPI Library Runtime Environment for Windows** to set username and password.

> **Note:**
>
> You can work around the password registration for cases where accessing network resources is not needed by performing the Active Directory setup and setting the environment variable **I_MPI_AUTH_METHOD** to "delegate".
>
> Consult Intel documentation: https://software.intel.com/en-us/mpi-developer-reference-windows-user-authorization

6. Click **Finish**.

Microsoft MPI

For Microsoft MPI, no configuration of user authorization is required.

For Windows HPC jobs, no additional configuration is required for distribution using Microsoft MPI.

For distribution using Microsoft MPI on the local host (without Windows HPC), some additional configuration is required. Microsoft MPI must be installed on the local host and the MS-MPI Launch Service must be running. The Microsoft MPI redistributable package installs the MS-MPI Launch Service, with the startup type set to Manual, so the MS-MPI Launch Service must be started manually. This may be done using the Windows Services Manager or using the command line as described in the Microsoft MPI Release Notes. Distribution to multiple hosts using Microsoft MPI is only supported under Windows HPC.

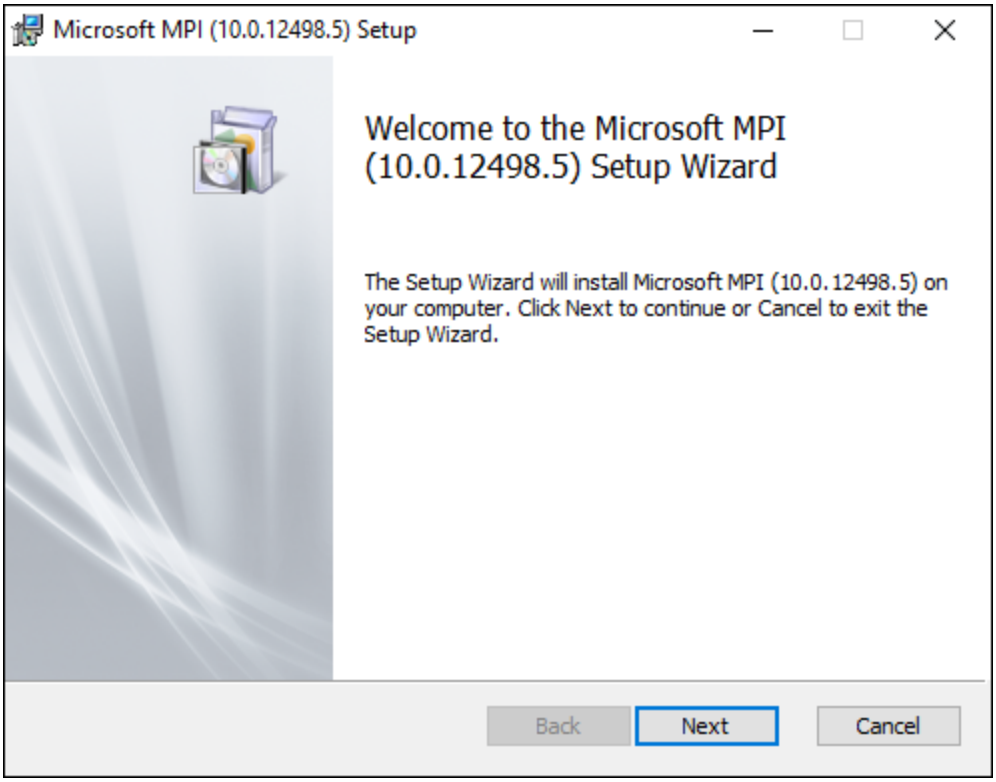
Installing Microsoft MPI

In order to use Microsoft MPI with Ansys EM Suite on the local host, Microsoft MPI must be installed on the local host and the MS-MPI Launch Service must be running.

To install Microsoft MPI:

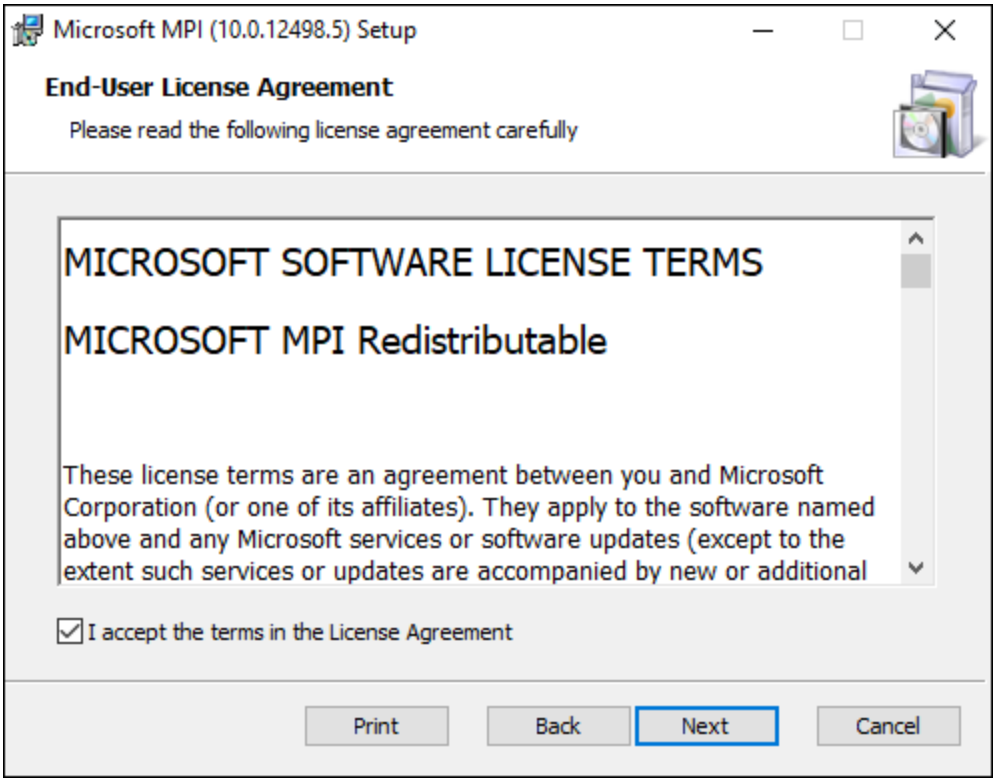
1. Launch the **Ansys Electromagnetics Installation Shell**:
   - If you have a physical disc, it will launch automatically. If it does not, locate the drive and launch autorun.exe.
   - If you downloaded a digital copy, unzip the download and launch autorun.exe.
2. Click **Install Distributed Simulation Technologies**.
3. Under **MPI**, click **Install Microsoft MPI**.

   The **Microsoft MPI Setup Wizard** launches.



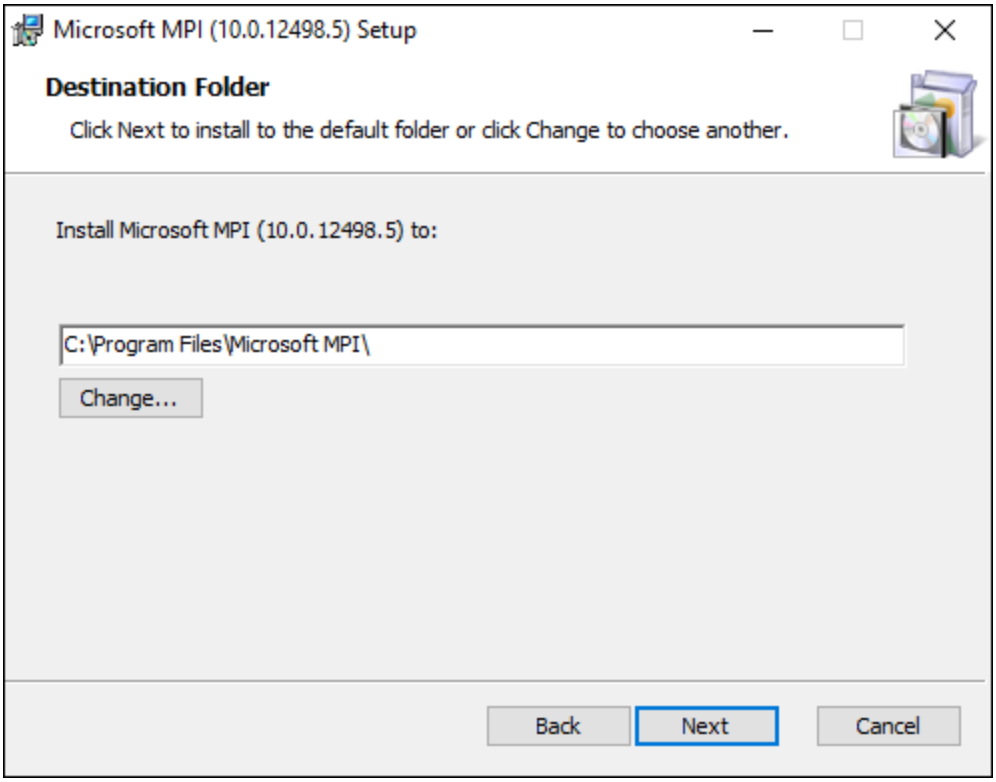
4. Click **Next**.

   The **End-User License Agreement** appears.



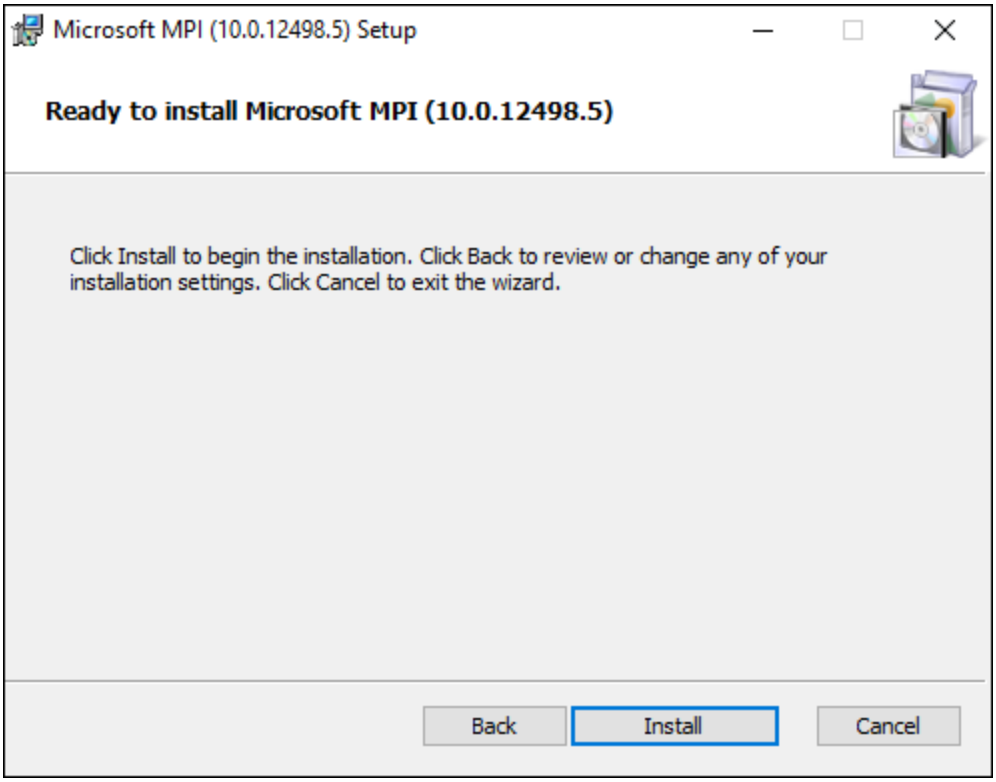
5. Select the check box to agree with the terms, then click **Next**.

   The **Destination Folder** selection screen appears.



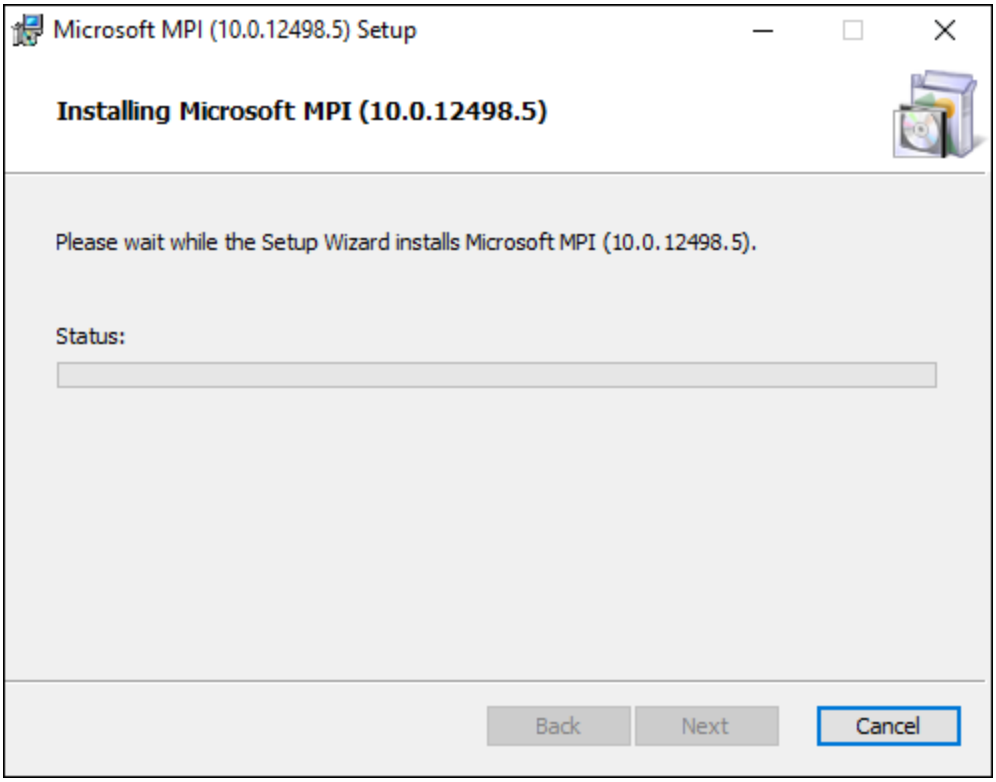
6. Select a destination folder, and click **Next**.

   The wizard is ready to install.

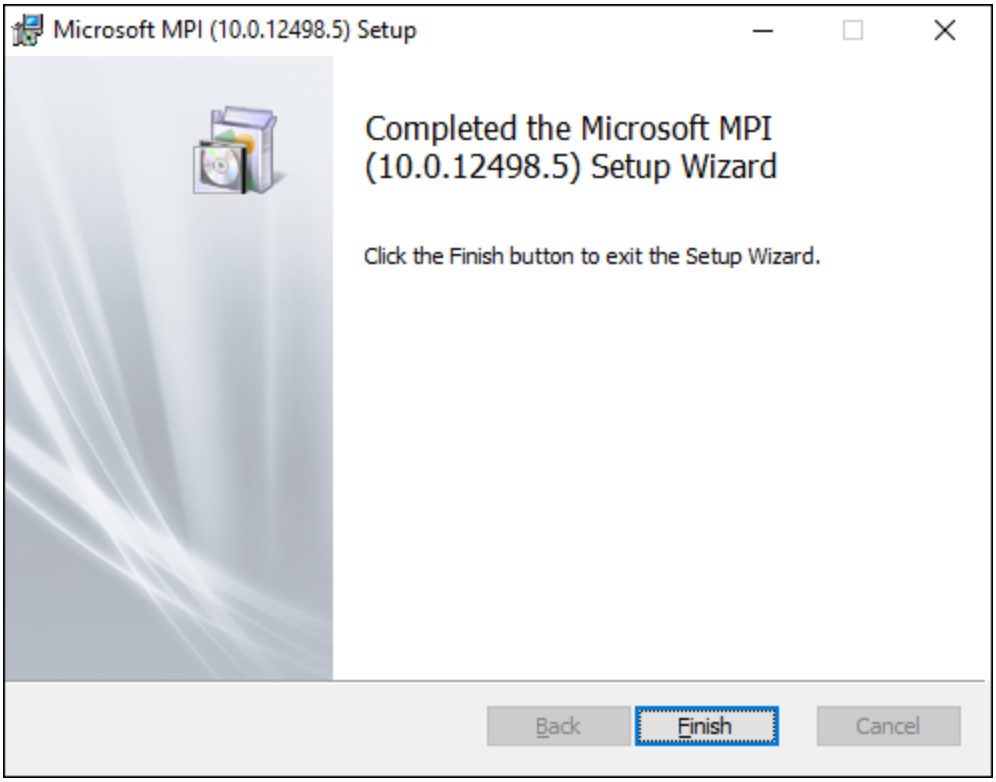


7. Click **Install**.

   A progress bar shows installation progress.



When installation completes, a success message appears.



8. Click **Finish** to close the wizard.

MS-MPI Launch Service

You can use MS-MPI Launch Service to launch MPI processes for local and remote machines.

For more information, refer to Microsoft MPI Release Notes for MS-MPI Launch Service.

Selecting an MPI Vendor in Ansys Electronics Desktop

To select which MPI to use:

1. Launch Ansys Electronics Desktop.
2. Click **Tools** > **Options** > **HPC and Analysis Options**.
3. On the **Options** tab, click the **MPI Vendor** field and select either **Intel** or **Microsoft**.

Important:

In order to use Intel MPI, you will first need to:

- Edit permissions for the temp directory on all machines so that the designated user is allowed full control.
- Disable the Windows firewall, or configure it as specified in the Firewall Configuration topic in the Ansys Electronics online help.

MPI Notes and Example Scenarios

Using network resources, such as shared drives, may require password registration on all machines for both Microsoft and Intel MPI.

It is easiest if share permissions are set to Full Control for the Everyone group for the temp directories.

Some example scenarios follow.

Note:

The scenarios described here assume that the user is part of a domain and is running on computers within the given domain.

Scenario #1: Local User Exists on All Machines

The user is an admin or standard user on the local machine and needs to run as an admin or standard user on one or more remote machines. 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: Local User Doesn't Exist on Remote Machine(s)

The user is an admin or standard user on the local machine, but is not defined as a user on the remote machine(s). 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 MPI jobs will run as a user that does not exist on the remote machine(s). *Ansys strongly recommends setting the temp directories to Full Control for the Everyone group to avoid issues with permissions*.

Scenario #3: Local User Only Runs Remote Machine(s)

The user is an admin or standard user on the local machine, but only needs to run jobs on remote machine(s). 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:

RSM Properties	Remote Simulation Options	Simulation runs on remote machine as	Notes
System	Service User	Local System user	Local System user will not have Ansys Electromagnetics Suite user preferences and may not have necessary privileges or access to resources.
System	Specified User = user_A	user_A	Runs as user_A
Specific User = user_A	Service User	user_A	Runs as user_A
Specific User = user_A	Specified User = user_B	May fail due to privileges	Not recommended

The preferred setup for this situation is to assign a Specific or Specified user, but not both.

Windows HPC Scheduler

When 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 a registered password expires or is changed for any other reason, then MPI failures may occur due to the incorrect password.

To use a command window to clear a previously registered password:

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 with Windows HPC

To use a network installation with the Windows HPC Scheduler:

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.

   Ensure that:

   - Password is registered with MPI on all nodes.
   - Active Directory is set up on all nodes.

   Consult Intel MPI online help:
   https://www.intel.com/content/www/us/en/developer/tools/oneapi/mpi-library.html

> **Note:**
>
> Password registration is not required for Windows HPC jobs.
>
> Before submitting a job, consult the online help for information on using MPI-based solves.

4. Install the following Microsoft Visual C++ Redistributable on all nodes:
   - Microsoft Visual C++ 2015-2019 (x86 and x64)

   The necessary installers are provided in the installation folder.
5. Verify installation was done correctly by launching Ansys Electronics Desktop on each node using the network installation.
6. Place the pjt file in a shared folder (one where Everyone has Full Control).
7. Launch Ansys Electronics Desktop on the head node.

   Set the head node in Electronics Desktop:

   - **Tools** > **Job Management** > **Select Scheduler**.
8. Open the Windows Scheduler GUI and submit the job.

   From Electronics Desktop:

   - **Tools** > **Job Management** > **Submit Job**.

   For more information on Windows HPC Schedulers, consult the Ansys Electronics online help.
9. Ensure you are using the UNC pathname for the product and project.
10. Create and set up a job, as desired.

    Ansys recommends setting the MPI version using batchoptions. For example:

    - `'HFSS/MPIVendor'=Intel`
    - `'HFSS 3D Layout/MPIVendor'=Intel`
    - `'Icepak/MPIVendor'=Intel`
    - `'Maxwell 2D/MPIVendor'=Microsoft`
    - `'Maxwell 3D/MPIVendor'=Microsoft`
    - `'Q3D Extractor/MPIVendor'=Microsoft`

After the job has been submitted, you can monitor it using **HPC Job Manager**.

Troubleshooting MPI

This section contains some frequently asked questions and error messages.

Frequently Asked Questions

Is there an easy way to test the computers in my Distributed Machine Configurations?

Yes. Perform the following steps:

1. Launch Ansys Electronics Desktop.
2. Select **Tools** > **Options** > **HPC and Analysis Options**.
3. Select your configuration from the list and click **Edit**.
4. Select one or more machines from the list and click **Test Machines**.

A set of tests will be run from the current machine to selected machine(s). These include pinging the computer(s) and checking RSM availability.

Why am I getting a password authentication failure?

This 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.

Error Messages

AnsoftRSMService COM Machine

You may receive the following error at simulation launch:

```
Unable to locate or start COM engine on <your_machine_name>:
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` is your local machine. Inability to start the COM engine 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` 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.

MPI Launch Failure

You may receive the following error during simulation:

```
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.
```

This error is often followed by:

```
Unable to create child process: mpirun
```

Possible causes include:

- 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.

Distributing Matrix Solutions Error

You may receive the following error during simulation:

```
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.

6 - Installing Layout Integrations for ECAD

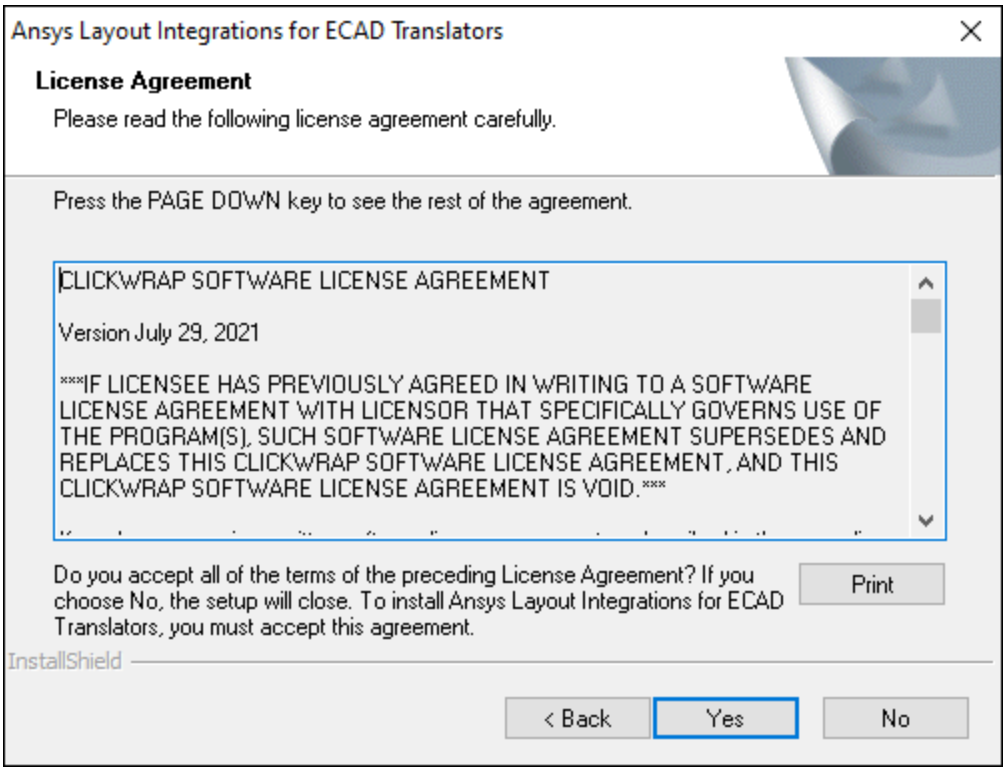
Layout Integrations for ECAD allow you to incorporate an ALinks module into third-party PCB/package layout tools. With these integrations installed, you can export salient design information—such as 2D geometry, layer stackup, and net names—directly into the SIwave user interface, or as Q3D Extractor or HFSS 3D Layout projects.

To install Layout Integrations for ECAD:

1. Log in as an administrator, or using an account with administrator privileges.
2. Launch the **Ansys Electromagnetics Installation Shell**:
   - If you have a physical disc, it will launch automatically. If it does not, locate the drive and launch autorun.exe.
   - If you downloaded a digital copy, unzip the download and launch autorun.exe.
3. Click **Install Layout Integrations for ECAD**.

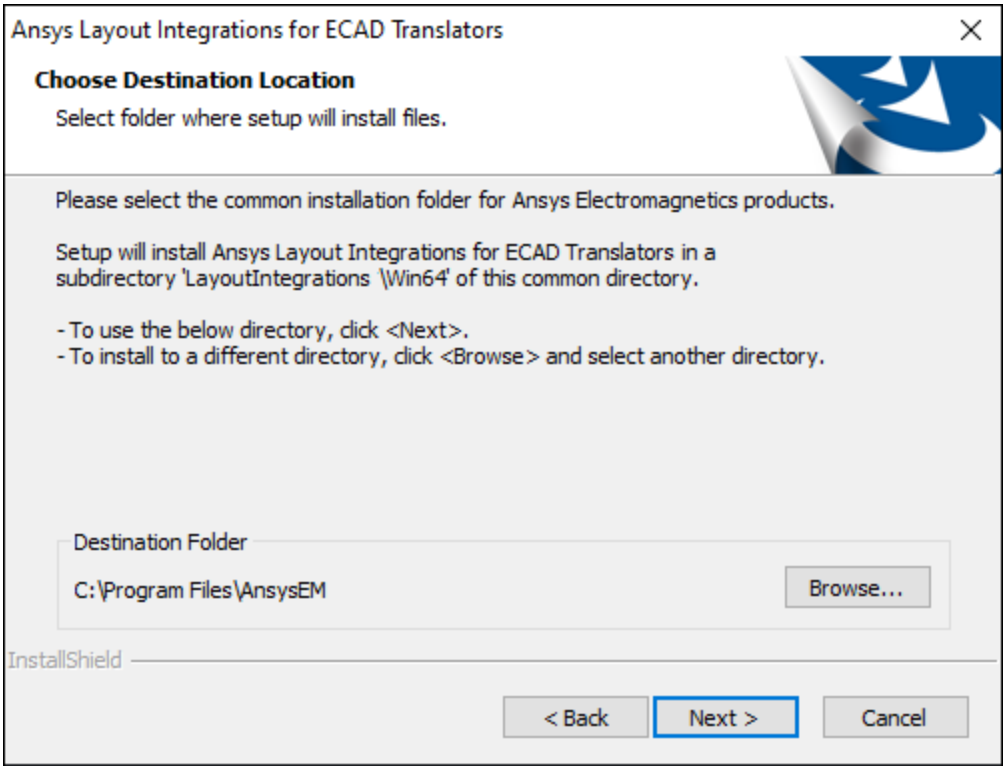
   The Installation Wizard appears.
4. Click **Next**.

   A **License Agreement** appears.



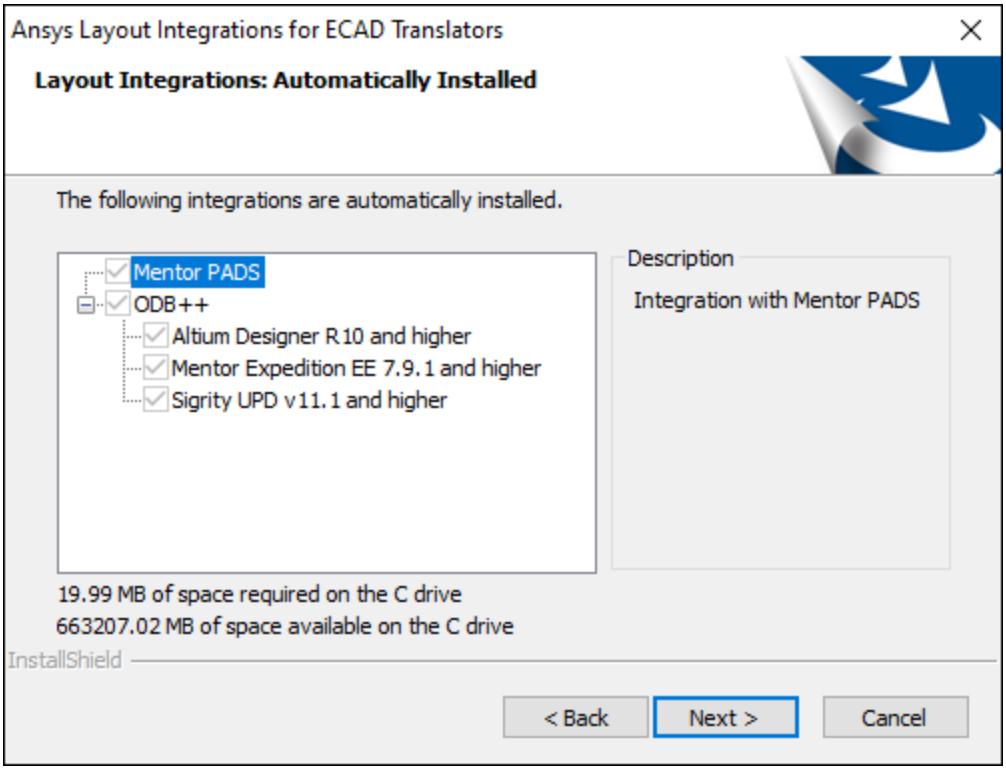
5. Review the agreement, and click **Yes** to agree.

   The **Choose Destination Location** window appears.



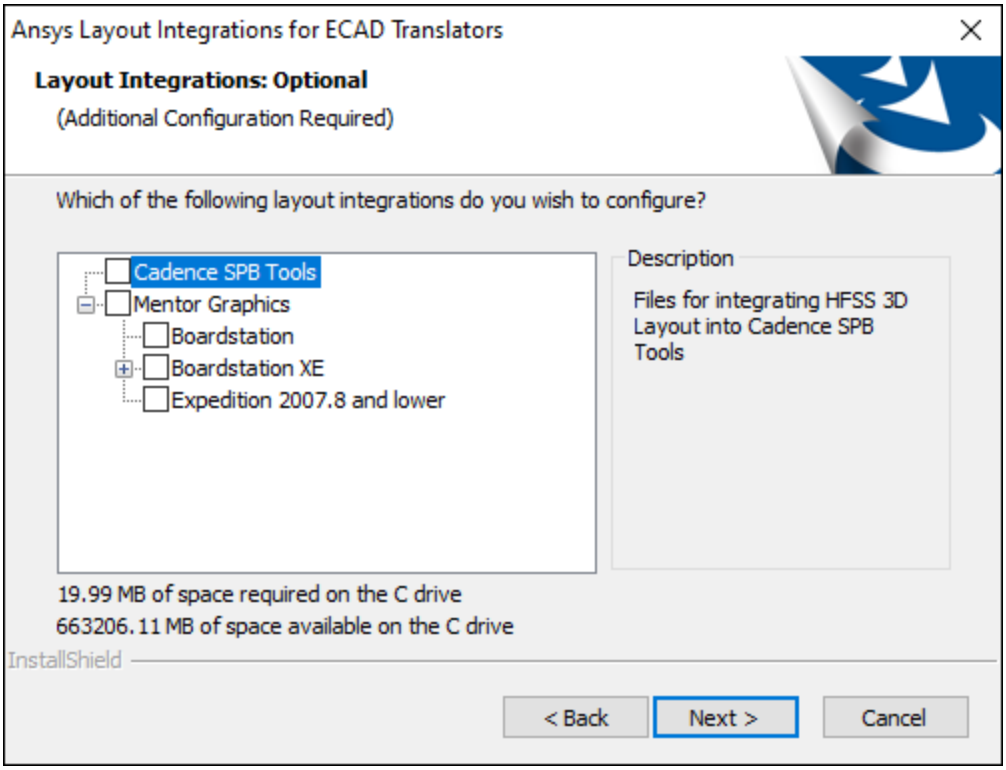
6. Select an installation directory, and click **Next**.

   A list of **Layout Integrations: Automatically Installed** appears. These integrations are pre-selected, and cannot be changed.



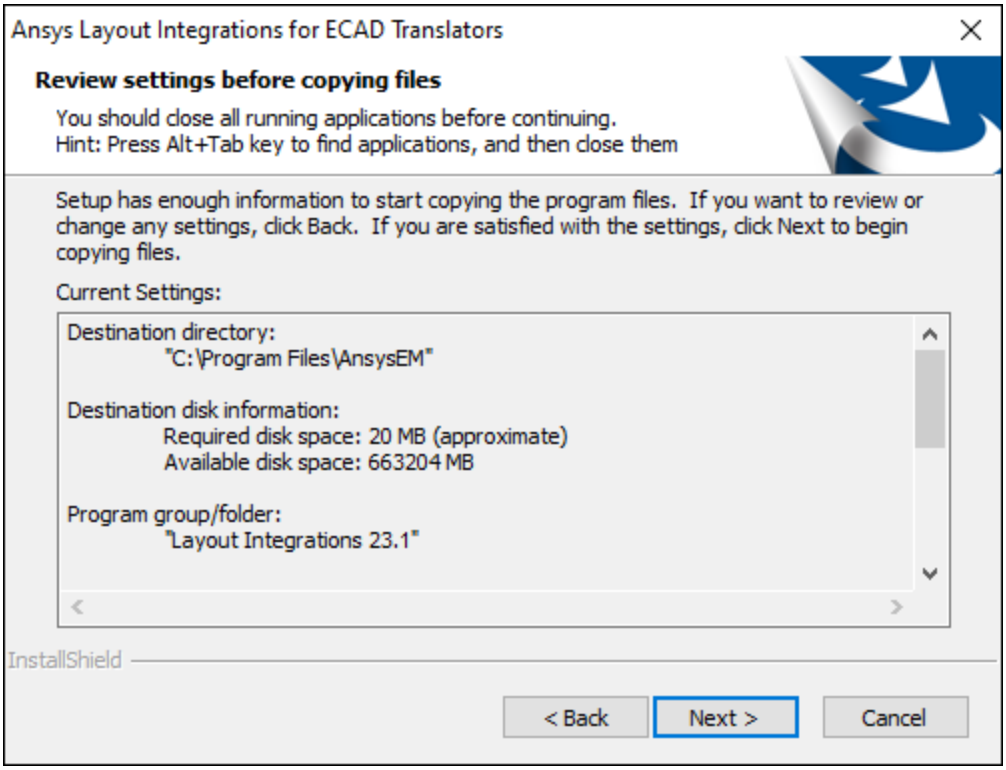
7. Click **Next**.

   A list of **Layout Integrations: Optional** appears.



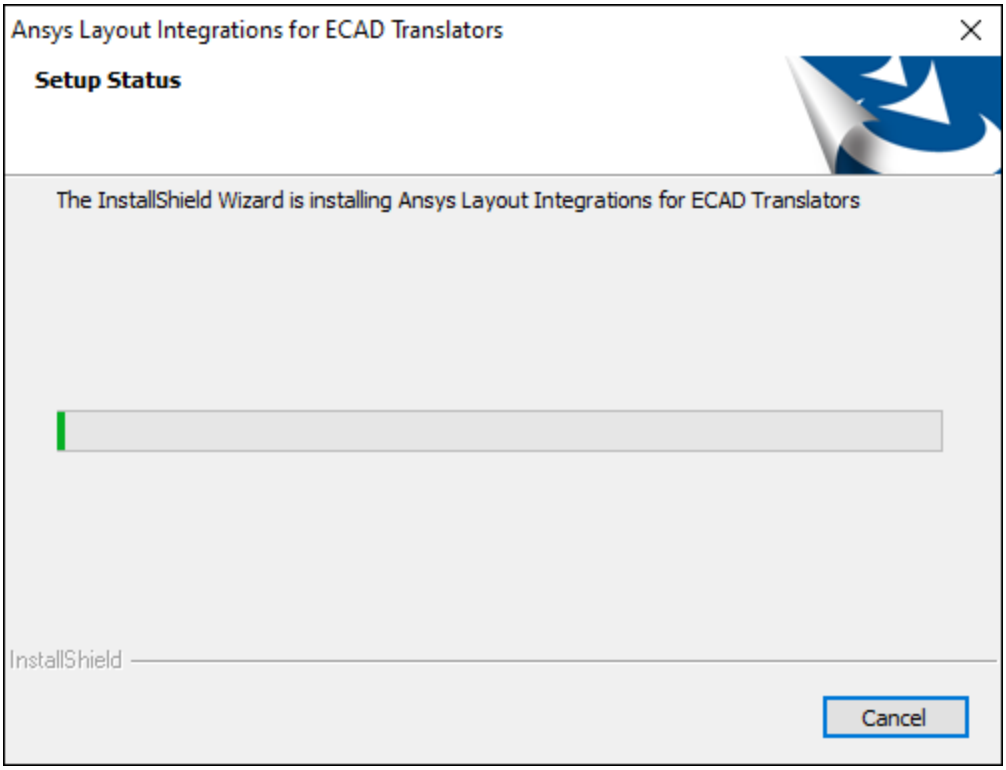
8. Select any integrations you wish to install, and click **Next**.

   The Installation Wizard prompts you to review settings.

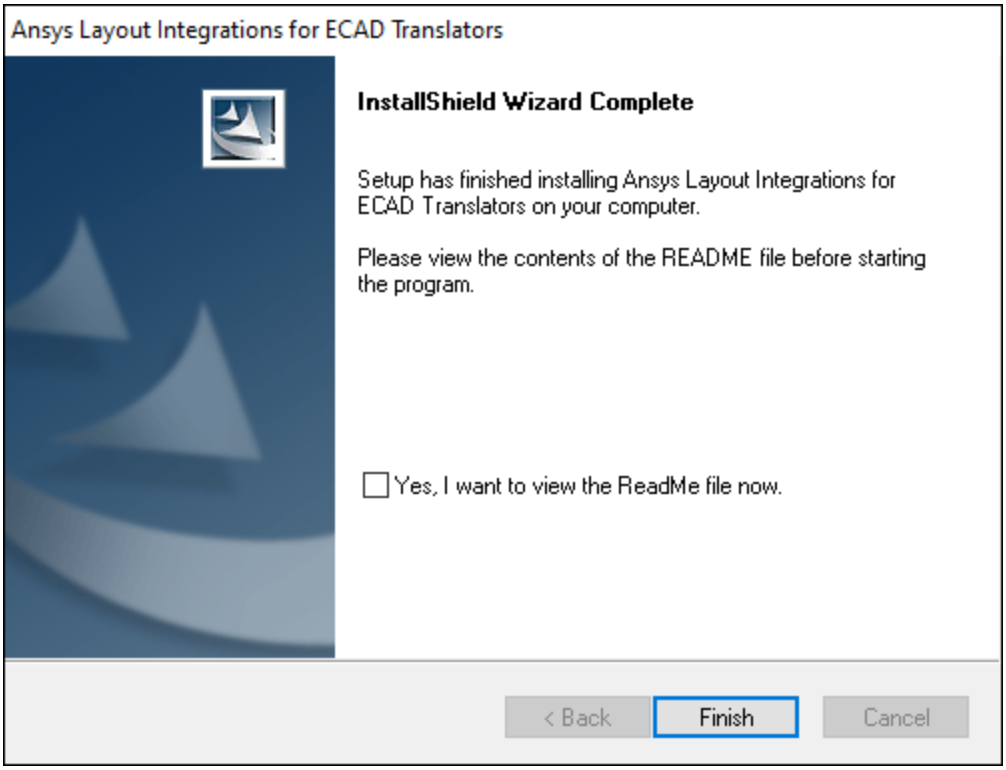


9. Review the settings, and click **Next**.

   A progress bar displays installation progress.



When installation has finished, a success screen appears.



If desired, select the check box to view the ReadMe file.

10. Click **Finish**.

7 - Uninstalling the Software

You can uninstall all of the Ansys and third party software discussed within this guide using the Windows Control Panel. Specifically, the following application are listed in the **Programs and Features** applet and can be uninstalled from there:

- Ansys Electromagnetics Remote Simulation Manager (RSM)
- Ansys Electromagnetics Suite
- Ansys Layout Integrations for ECAD Translators
- Microsoft MPI
- Intel MPI

Use the following procedure to uninstall any of these applications:

1. From the Windows Start Menu, select **Windows System** > **Control Panel**.

   The *Control Panel* window appears.

2. The next instruction depends on the display mode of your Windows Control Panel:
   - If the current "**View by**" selection is **Category**, then:

     Under *Programs*, click **Uninstall a program**.

   - If the current "**View by**" selection is either **Large icons** or **Small icons**, then:

     Click **Programs and Features**.

   In either case, the *Programs and Features* window appears.

3. Locate the application that you wish to uninstall and click to select it.
4. Just above the list of programs, click **Uninstall**.

   Alternatively, you can right-click the program to remove and choose **Uninstall** from the shortcut menu.

5. Follow the screen prompts to complete the uninstallation process.
6. Repeat steps 3 through 5 for each application you wish to uninstall.
7. When finished, close the *Programs and Features* and *Control Panel* windows.

A - Silent Installation

The Ansys Electromagnetics Suite silent installation feature installs the software on your computer with no windows appearing during the installation process. InstallShield suppresses all dialog boxes, including error messages and informational message boxes.

There are two ways to perform silent installations:

- **Method 1:** Silent Installation from the Command Line
- **Method 2:** Silent Installation Using a Response File

Method 1: Silent Installation from the Command Line

For this method, options for running the installation program (setup.exe) are included directly in the command line.

The applicable **options** for this method are as follows:

- **-help**: Displays a list of valid arguments for a silent installation.
- **-silent**: Run a completely silent installation.

  Do not confuse this option with the standard "-s" InstallShield option, which expects a response file for providing the necessary installation parameters. This custom "-silent" option does not look for a response file and expects all necessary parameters to be specified within the command line.

- **-install_dir**: Specify the path to the installation directory when not using the default location.

  Syntax: **`-install_dir <installation_directory_path>`**

  The default path, **%ProgramFiles%\AnsysEM**, is used if this option is omitted.

- **-server_info**: Specify the port number and name of the license server(s).

  Single Server Syntax: **`-server_info <port_number:hostname>`**

  Redundant Server Syntax: **`-server_info <port_number:hostname1,hostname2,hostname3>`**

  The default port and server, if not specified, is **1055:<*local_machine_name*>**, unless an existing license configuration is found. In that case, the server information is taken from the existing license file.

- **-temp_dir**: Specify the path to the temporary files directory when not using the default location.

  Syntax: **`-temp_dir <temp_directory_path>`**

The Temp directory is used during local and remote simulations for temporary workspace. The default path, **%temp%**, is used if this option is omitted.

- **-temp_permissions**: Specifies if Temp folder access is granted to all users or only the current user.

  Syntax: **`-temp_permissions [TRUE|FALSE]`**

  When TRUE (the default value), read/write access to the Temp directory is granted to EVERYONE. When FALSE, read/write access is only granted to the current user.

- **-commonlib_dir**: Specify the path to the common libraries directory if you will be using previously installed libraries.

  Syntax: **`-commonlib_dir <`*`common_libraries_directory_path`*`>`**

  The default action is to install local libraries. Specifying this option prevents their installation.

- **-prereqs**: Install only prerequisite software.

  Use this option by itself (all other command line options will be ignored). It limits the scope of the installation to the prerequisites portion. As such, setup.exe will install only .Net Framework and Visual C++ Redistributables (assuming the required versions are not already present on the system). An example of when this option is useful is for setting up remote computing nodes for High Performance Computing (HPC), where the full Ansys Electromagnetics Suite is not required on the host machines.

Note:

- All paths that contain one or more spaces must be enclosed in double quotes ("..."). You can also enclose paths without spaces in double quotes, but they are not required.
- Specify absolute paths for softare installation, temporary files, and common libraries locations.
- Absolute paths are recommended for the path to the installation command itself (setup.exe). This is an executable file that often exists in multiple locations, and a different one than intended may be included in your *Path* environment variable, leading to unpredictable results. However, if the current folder of the command prompt is already at the Ansys installer location, you could simply enter **`.\setup.exe`** followed by the required options. Appending ".\" is required to ensure that you access the setup.exe file in the currently active folder and not any other one. It is also required in Windows PowerShell and the Windows 11 Terminal window for the command to be found.

Command Line Examples:

- **Example 1:** Override the default installation and temporary file locations. The installer is located at E:\Installers\AnsysEM:

```
E:\Installers\AnsysEM\setup.exe -install_dir D:\AnsysEM -temp_
dir C:\Temp
```

- **Example 2:** Specify a shared common libraries location on the network. For this example, it is assumed that the command prompt is at the installer location:

```
.\setup.exe -silent -commonlib_dir
"\\LibraryServer\AnsysEM\Common Libraries"
```

- **Example 3:** Install prerequisite software only. For this example, it is assumed that the installer is located on a network folder:

```
\\Software_Installers\Ansys\Electronics_242_
winx64\AnsysEM\setup.exe -prereqs
```

Method 2: Silent Installation Using a Response File

For this method, you record a response file during an initial command line installation using the graphical user interface (GUI mode), not silently. For subsequent installations, run the installation silently, specifying the response file to use. Tthe response file ensures a duplicate configuration for all subsequent installations that reference it.

The applicable **options** for response file installations are as follows:

- **-r**: Run the installation in record mode, generating a response file.

  The response file is recorded from the initial GUI-mode installation. The path and filename of the response file are specified using the following -f1 option. The resultant file is then specified in the command line for subsequent silent installations.

- **-f1**: Used to specify an alternative path and filename for the response file (whether generating the file or using it to control a silent installation).

  Syntax: **-f1<response_file_path\name>**

  If this parameter is not specified, the default filename is **<*installer_name*>.iss** (for example, setup.iss) and its location is in the same folder where the installer (such as setup.exe) is located.

- **-f2**: Used to specify an alternative path and filename for the installation log file.

  Syntax: **-f2<log_file_path\name>**

If this parameter is not specified, the default filename is **<*installer_name*>.log** (for example, setup.log) and its location is in the same folder where the installer (such as setup.exe) is located.

- **-s**: Run the installation in silent mode using a response file and suppressing all dialog boxes, including error and informational messages.

  The installer looks for **<*installer_name*>.iss** (for example, **setup.iss**) in the same folder where it resides. Use the -f1 parameter to specify an alternative response file path and filename.

  Do not confuse the standard "-s" InstallShield option with the custom "-silent" option, which does not look for a response file and expects all necessary parameters to be specified within the command line.

- **-a**: Perform an administrative installation.

  An administrative installation copies all files to the destination path but does not create shortcuts, register COM servers, or create an uninstallation log.

Recording a Response File

1. Open the Windows command prompt.
2. Locate the installation folder containing setup.exe or a patch file.
3. Run one of the following commands:

   For a Full/New EM Suite Installation:

   `<path_to_setup.exe> -r -f1<path_to_.iss_response_file>`

   > **Note:**
   >
   > - If there are spaces in a file path, enclose the full path\filename in double quotation marks. For example:
   >
   >   `"E:\Ansys Installers\<version>\AnsysEM\setup.exe" -r -f1"E:\Response Files\AnsysEM\<version>\setup.iss"`
   >
   > - The -f1 parameter can be omitted if the response filename is **setup.iss** and its location is the same folder as setup.exe.

   For a Self-Extracting Patch:

   `<path_to_patch.exe> -a -r -f1<path_to_.iss_response_file>`

4. Follow the installation prompts.

   Once complete, the *.iss response file will contain information that can be used to install the same product with the same configuration on other Windows machines.

Using a Response File

To use the response file:

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

1. Open the Windows command prompt.
2. Locate the Installation directory containing setup.exe, or the directory containing a patch file.
3. Run one of the following commands:

   For a Full/New EM Suite Installation:

   `<path_to_setup.exe> -s -f1<path_to_.iss_response_file> -f2<path_to_.log_file>`

   > **Note:**
   >
   > - If there are spaces in a file path, enclose the full path\filename in double quotation marks. For example:
   >
   >   `"E:\Ansys Installers\<version>\AnsysEM\setup.exe" -s -f1"E:\Response Files\AnsysEM\<version>\setup.iss" -f2"E:\Installation Logs\AnsysEM\<version>\setup.log"`
   >
   > - The -f1 and -f2 parameters can be omitted if the default response and log filenames and locations are used. Specifically, the defalut filenames are **setup.iss** and **setup.log**, respectively, and both file locations default to the same folder where the installer (setup.exe) resides.

   For a Self-Extracting Patch:

   `<path_to_patch.exe> -s -f1<path_to_.iss_response_file> -f2<path_to_.log_file>`

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

4. Verify that the installation was successful using the .log files. Open the *.log file created during Step 3, and examine the `ResultCode` value in the [`ResponseResult`] section. If the value is zero, 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 reasons:

- An invalid path was specified.
- There were differences in the dialog box sequences between the recorded response file and the current silent installation. For example, an additional message appeared during the playback due to differences in the two computer systems.

Silent Mode Result Codes

Result Code	Description
0	Success
-1	General error
-2	Invalid mode
-3	Required data not found in the Setup.iss file
-4	Not enough memory available
-5	File does not exist
-6	Cannot write to the response file
-7	Unable to write to the log file
-8	Invalid path to the InstallShield Silent response file
-9	Not a valid list type (string or number)
-10	Data type is invalid
-11	Unknown error during setup
-12	Dialog boxes are out of order
-51	Cannot create the specified folder
-52	Cannot access the specified file or folder
-53	Invalid option selected

Performing a Silent Uninstall

You can perform a silent uninstall of the Ansys Electromagnetics Suite by entering the following at the Windows command prompt:

```
<path_to_setup.exe> -runfromtemp -z suppressconfirmation -removeonly
```

> **Note:**
>
> Make sure there is a space before each dash (-).

B - License hostids

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:

Operating System	Permitted Hostids
Windows	• The ethernet address of a network interface card (MAC address). • Flexera FLEXid USB port hardware key. • Disk serial number.
Linux	• The ethernet address of a network interface card (MAC address). • Flexera FLEXid USB port hardware key.

The following sections describe the process for obtaining valid Windows hostids.

Obtaining a Disk Serial Number

To obtain the disk serial number:

1. Launch the Windows command prompt.
2. Type the following:

   ```
   vol <DriveLetter>
   ```

   For example:

   ```
   vol C:
   ```

3. Press **Enter**.

Ansys also offers a utility that captures the disk id:

https://www.ansys.com/it-solutions/licensing/capture-license-server-information

Obtaining an Ethernet Address

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.

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

1. If you are using a docked laptop, disconnect it from the docking station.
2. From the Windows command prompt, type the following:

   ```
   ipconfig /all
   ```

3. Press **Enter**.

   A list of network interface cards appears.

4. Look for the **Physical Address** line.

   The ethernet address for the 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.

Using a Hardware Key

A Flexera FLEXid hardware key may be provided to you by Ansys. 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=`. Only a single device can be installed on each USB port.

To install a hardware key using a USB port:

1. Attach the hardware key to the USB port.
2. If a prompt appears asking you to install or specify a driver, click **Cancel**.

Consult Ansys License Manager documentation for instructions on using USB dongles:

https://www.ansys.com/it-solutions/licensing

C - Enabling System Coupling with Maxwell

To enable coupling with Maxwell, the environment variable **ANSYSEM_ROOT242** must be set to the location of your Ansys Electromagnetics installation.

> **Note:** The environment variable name is ANSYSEM_ROOT<*major*><*minor*>, where <*major*> is the two digit major version number (the last two digits of the year); and <*minor*> is the one digit minor version number. For example, for the Ansys Electronics 2024 R2 release, the environment variable name is ANSYSEM_ROOT242. For the Student Version of the software, the variable name is ANSYSEM_ ROOTSV<*major*><*minor*>.

- For Windows, this environment variable is automatically set during the installation of the Ansys Electromagnetics Suite (for example, C:\Program Files\AnsysEM\v242\Win64).

> **Note:** The environment variable must be set on all hosts if the analysis is distributed to multiple hosts. For Windows, this environment variable must generally be set as a System environment variable (not a user environment variable) for distributed analysis so that it is set for the Ansoft RSM Service.

Index

I

L

M

P

R

S

T

U