



Ansys Electromagnetics Suite – Legacy Linux Installation Guide (2024 R2 and Earlier)



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

ANSYS, Inc. and ANSYS
Europe, Ltd. are UL registered
ISO 9001:2015 companies.

Copyright and Trademark Information

© 1986-2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICM CFD is a trademark used by ANSYS, Inc. under license. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

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

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

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

Third-Party Software

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

Table of Contents

Table of Contents	Contents-1
1 - Getting Started	1-1
Installation Prerequisites	1-1
License File	1-1
Ansys License Manager	1-1
TCP/IP	1-2
OpenGL	1-2
Required Libraries for Linux	1-2
Additional Resources	1-3
2 - Installing Ansys Electromagnetics Suite Software	2-1
Verifying that Enough Disk Space is Available	2-1
Installing the Software	2-1
Integrating Ansys Electromagnetics Products with Ansys Workbench	2-5
3 - High Performance Computing	3-1
Options for Distributed Analysis	3-1
Semaphores on Linux	3-1
Recommended settings	3-2
Displaying semaphore settings	3-2
Changing semaphore settings	3-2
Getting Information on Semaphores	3-2
Removing Semaphore Arrays	3-3
Semaphore Errors	3-4
Remote Simulation Manager (RSM)	3-4
When Is RSM Needed?	3-4
Enabling and Disabling Remote Access	3-5
Installing RSM	3-6

Troubleshooting Installation	3-7
Network vs. Non-Network Installation	3-7
Configuring RSM	3-8
General RSM syntax	3-8
Determining Who Should Run RSM	3-9
Registering Ansys Electromagnetics Product Engines for use with RSM	3-9
Changing the RSM Listening Port	3-9
RSM Service Configuration	3-10
Ansoft Electromagnetics Desktop Configuration	3-11
Supported Job Schedulers	3-12
Custom Scheduler Integration	3-12
Limitations	3-13
Running HPC Test Jobs on a Cluster	3-13
Submitting and Running the Tests	3-13
Background: Selecting a Scheduler	3-14
Test #1: Submitting and Running a Non-distributed Job	3-14
Test #2: Submitting and Running a Non-MPI Job Distributed to Multiple Hosts	3-16
Test #3: Submitting and Running an MPI Job Distributed to Multiple Hosts	3-17
Scheduler-Specific Configuration and Testing	3-20
Altair PBS Professional	3-20
Cluster Configuration Guidelines	3-20
Running Test Jobs with Altair PBS Professional	3-21
Non-distributed Job	3-21
Non-MPI Job Distributed to Multiple Hosts	3-21
MPI Job Distributed to Multiple Hosts	3-21
IBM Spectrum LSF	3-21
Cluster Configuration Guidelines	3-22
Running Test Jobs with IBM Spectrum LSF	3-22

Non-distributed Job	3-22
Non-MPI Job Distributed to Multiple Hosts	3-22
MPI Job Distributed to Multiple Hosts	3-23
Univa Grid Engine (UGE)	3-23
Cluster Configuration Guidelines	3-23
qsh Command	3-23
Parallel Environments	3-24
Submitting Exclusive Jobs	3-24
Configuring an Exclusive Resource	3-25
Submitting an Exclusive Job from the Command Line	3-25
Submitting an Exclusive Job using the Job Submission GUI	3-26
Consumable Memory Limits	3-26
Running Test Jobs with Univa Grid Engine (UGE)	3-27
Non-distributed Job	3-27
Non-MPI Job Distributed to Multiple Hosts	3-28
MPI Job Distributed to Multiple Hosts	3-28
4 - Message Passing Interface (MPI) Installation	4-1
Configuring Intel MPI	4-1
Additional Information on RSM Settings	4-2
MPI Batchoptions	4-2
MPI Vendor	4-2
Remote Spawn Command	4-3
Setting up passwordless ssh	4-3
Troubleshooting MPI	4-4
Frequently Asked Questions	4-4
Error Messages	4-5
5 - Installing Layout Integrations for ECAD	5-1
6 - Uninstalling the Software	6-1

Uninstalling the Electromagnetics Suite	6-1
Uninstalling RSM	6-1
Uninstalling Layout Integrations for ECAD	6-2
A - Silent Installation	A-1
Setup.exe Command Line Options	A-1
Installer Properties	A-2
Silent Installation from the Command Line	A-3
Command Line Examples	A-3
Silent Installation Using a Response File	A-4
Recording a Response File	A-4
Response File Format	A-4
Using a Response File	A-5
Installer Result Codes	A-6
Example Response File	A-7
B - License hostids	B-1
Obtaining a Machine Hostid	B-1
Obtaining an Ethernet Address	B-1
Using a Hardware Key	B-2
C - Enabling System Coupling with Maxwell	C-1
Index	C-2

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 additional 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.
- 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.

OpenGL

Ansys Electromagnetics applications make use of OpenGL libraries. You will need to install OpenGL libraries if you experience sluggishness when interacting with the UI, or if you see any of the following errors:

- Unable to resolve function glXQueryExtension
- Failed to initialize an OpenGL view

Ansys recommends the use of OpenGL libraries provided by the manufacturer of the graphics card in your machine or by the PC or laptop manufacturer. For the latest information, please visit the following Ansys Help web pages:

- [Ansys Platform Support](#)

From this page, access the following documents:

- Ansys 2024 R2 – Graphics Cards Tested
- Ansys 2024 R2 – GPU Accelerator Capabilities
- [Additional Hardware and Software Requirements](#) (Ansys account login required)

Required Libraries for Linux

Information regarding library dependencies for various version of Linux are maintained in the online Ansys Help. Follow the link below for more information:

- [Required Linux Libraries and Packages by Product](#)

The following tables within this webpage are applicable to the Electromagnetics Suite products:

- Table 2.1: Packages Required for the Installer to Run
- Table 2.2: Common packages that may be required for all products
- Table 2.11: DCS (Distributed Compute Services)
- Table 2.19: Mechanical Products
- Table 2.22: optiSLang
- Table 2.23: Parasolid Geometry Interface

- Table 2.26: Remote Solve Manager Standalone Services
- Table 2.30: Ansys, Inc. License Manager

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 <i>What's New</i> 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
How-To Videos:	Video instructions for installation are available at https://www.youtube.com/channel/UCdymxOTZSP8RzRgFT8kpYpA

2 - Installing Ansys Electromagnetics Suite Software

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

1. Ensuring the machine meets installation prerequisites.

See: [Installation Prerequisites \(Page 1-1\)](#).

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

Verifying that Enough Disk Space is Available

The amount of disk space required by the Ansys Electromagnetics software depends on which packages you are installing. Refer to the hardware requirements for each package you have purchased for more details. During the setup of a particular installation, you are informed of the disk space requirements for that software.

You need sufficient disk space in the following directories:

- The directory where the software is installed.
- The directory where the software stores temporary files while solving a project.

Note:

On every machine where you are going to run the software, you also need enough swap space to accommodate at least the amount of RAM on the machine. Contact your system administrator if you have questions about swap space.

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** script:
 - If you have a physical disc, locate the drive and launch the **install** script.
 - If you downloaded a digital copy, unzip the archive and launch the **install** script.
3. Click **Install Electromagnetics Suite**.

The *InatallAnywhere* installation wizard appears.

Note:

If you receive an error that InstallAnywhere did not find enough temporary space to run:

- a. Set your IATEMPDIR environment variable to a different directory.
- b. Exit the installation.
- c. From a command window, type the following:

```
setenv IATEMPDIR <newdir> (csh)

export IATEMPDIR=<newdir> (bash)
```

Then, attempt installation again.

4. Click **Next**.

The *License Agreement* screen appears.

5. Read the license agreement carefully. Select **I accept the terms of the License Agreement** to proceed.
6. Click **Next**.

The *Choose Install Folder* screen appears.

7. Under **Where would you like to install?**, do one of the following to accept the default path or specify a different one:
 - Do nothing to accept the listed default installation folder.
 - To select a different folder, click **Choose**. The *Select a Folder* window appears. Navigate to the desired installation folder and click **Select**.

If you have other Ansys Electromagnetics software, Ansys recommends that you install it in the same folder.

- To create a new folder, click **Choose**. The *Select a Folder* window appears. Navigate to the parent folder where you want to create the new installation folder. Then, append the new folder name to the path in the **Folder Name** field. Alternatively, you can click the **Create New Folder** icon; rename the folder while the default name, *NewFolder*, is still selected; and then press **Enter**. Finally, click **Select** to accept the specified folder.

Note:

Multiple machines/nodes can run from the same installation tree. You do not have to install the application on each Linux machine/node. Ansys recommends installing to a network path.

8. Click **Next**.

Depending on the software and version being installed, you may see one or both of the following screens:

- *Select Temporary Folder*
- *Choose Library Option*

9. If the *Select Temporary Folder* 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 **Choose** button to display a folder browser window for navigating the folders on your machine and network.
- If desired, check the option to **Add read/write access for all users to the Temp directory**.

If the installation will be used for remote simulation by multiple users, all users must have permissions to access the temporary folder.

Then click **Next**.

10. If the **Choose Library Option** screen appears, do one of the following to specify the location of the 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.
- Click **The following common installation** to use libraries that are already installed (locally or on the network). Click **Choose**, select the appropriate path, and click **Select**.

Then click **Next**.

11. The *License Information* screen appears.

If an existing license configuration is found, it is shown near the top of the *License Information* screen.

12. Select one of the following two license options:

- **I want to specify the license configuration** – Select this option if you are using a single or multi-seat license and already know the machine on which the Ansys License Manager is or will be installed. Then:
 - a. Click **Next**.
 - b. Optionally, if you have redundant license servers, select the **Select redundant servers** option.
 - c. Enter the server name or names.

Note:

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.

- d. 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.
 - e. Click **Next**.
- **I want to keep the current license configuration** – Select this option if you already have a license configuration for a previous version of Ansys software and want to keep the same configuration. Then:
 - a. Click **Next**.

Note:

If you want to change the license server after installation, modify the `ansyslmd.ini` file in the **AnsysEM/shared_files/licensing** folder.

13. The *Pre-Installation Summary* screen appears next.

Review the displayed information and then do one of the following:

- If the information is correct and there is sufficient available disk space, click **Install** to start copying files.
- If the information is incorrect, click **Previous** 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 *Pre-Installation Summary* screen, and click **Install**.

The *Installing Ansys Electromagnetics* screen appears, displaying a progress bar and status messages as the installation progresses. 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.

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

15. When the installation is complete, the *Install Complete* screen appears.

Click **Done** to close the installer window and return to the *Ansys Electromagnetics Installation Shell* window.

16. Optionally, continue to the [Install Electromagnetics RSM](#) task or the [Install Layout Integrations for ECAD](#) task.
17. Click **Exit** when you are done installing software.

Integrating Ansys Electromagnetics Products with Ansys Workbench

After installation is complete, you can integrate Ansys Electromagnetics with Ansys Workbench by running the following command:

```
<install_dir>/ansys_inc/v242/AnsysEM/scripts/IntegrateWithAnsys24.2.pl.
```

Follow the prompts for integration.

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* (HPC_Admin.pdf) in the installation directory's Help folder.

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 daemon. An analysis may be distributed to any of the hosts where the Remote Simulation Manager is running. RSM is bundled with Ansys Electromagnetics products.

Alternatively, there are a number of job schedulers available from third parties, such as IBM Spectrum LSF, Univa Grid Engine, and Altair PBS Professional. Job schedulers (sometimes called distributed resource management systems, cluster managers, batch systems or distributed resource managers) provide job queuing, load balancing, resource assignment, and other management and monitoring capabilities. When a job is submitted to a scheduler, the product uses the resources allocated to the job to accomplish the analysis.

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

Semaphores on Linux

The Linux kernel provides a number of features to facilitate synchronization between threads and/or processes. Semaphores are one such mechanism. Linux supports two styles of semaphores, System V IPC semaphores and POSIX semaphores. On Linux, Ansys EM software

uses MainWin, which uses System V IPC semaphores. If there are not enough System V IPC semaphores available, the software may fail to start or terminate due to errors.

System V IPC semaphores are allocated as semaphore arrays. Each array may contain a few or many semaphores. The Linux kernel limits both how many semaphore arrays can be allocated, as well as the total number of semaphores that can be created.

Recommended settings

The following kernel semaphore settings are recommended:

- SEMMSL (maximum number of semaphores per semaphore set) = 256
- SEMMNS (system-wide limit on the number of semaphores in all semaphore sets) = 40000
- SEMOPM (maximum number of operations that may be specified in a semop() call) = 32
- SEMMNI (system-wide limit on the number of semaphore identifiers) = 32000

Displaying semaphore settings

The following Linux command will display the current semaphore settings:

```
cat /proc/sys/kernel/sem
```

The semaphore settings are displayed or set in the order: SEMMSL SEMMNS SEMOPM SEMMNI

Sample output:

```
% cat /proc/sys/kernel/sem  
250 32000 32 128
```

Changing semaphore settings

The following Linux command may be used to change the semaphore settings to the recommended values:

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

This command must be run as a privileged user, e.g., root. This command immediately changes the kernel limits. Rebooting the machine resets the semaphore limits to the default values.

Contact a system administrator to configure the semaphore settings at boot time.

Getting Information on Semaphores

Use the following Linux command to show information about the currently allocated System V semaphore arrays:

```
/usr/bin/ipcs -s
```

Sample output:

```
%/usr/bin/ipcs -s
```

```
----- Semaphore Arrays -----
```

key	semid	owner	perms	nsems
0x00000000	23887872	user1	664	2
0x00000000	19202049	user1	664	2
0x00000000	22413314	user1	664	2
0x00000000	3735555	user1	664	2
0x00000000	22446084	user2	664	2
0x00000000	22478853	user2	664	2

Use the following Linux command to show additional information about the semaphore array with semid id:

```
/usr/bin/ipcs -s -i id
```

Sample output:

```
%/usr/bin/ipcs -s -i 23887872
```

```
Semaphore Array semid=23887872
```

```
uid=789 gid=123 cuid=456 cgid=100
```

```
mode=0664, access_perms=0664
```

```
nsems = 2
```

```
otime = Wed Mar 11 18:48:16 2020
```

```
ctime = Wed Mar 12 18:48:16 2020
```

semnum	value	ncount	zcount	pid
0	0	0	0	0
1	1	0	0	31667

Removing Semaphore Arrays

System V IPC semaphore arrays are not automatically removed when a process terminates. Normally, processes that create semaphore arrays remove them on shutdown. If a process that creates some semaphore arrays terminates in an abnormal manner, such as due to a crash or a kill command, the semaphore arrays created by the process may not be removed. These sem-

aphore arrays will remain allocated, reducing the number of semaphore arrays available for other processes. On a reboot, all semaphore arrays are removed.

You can also remove semaphore arrays using the following Linux command, where *semid* is the id of the semaphore array to remove:

```
/usr/bin/ipcrm -s semid
```

Semaphore arrays should only be removed if they are no longer being used by any processes.

Semaphore Errors

Error messages due to running out of semaphores may look like the following:

```
*sem_lock->semop->op_op: Invalid argument
```

```
*sem_unlock->semop->op_op: Invalid argument
```

```
*sem_unlock->semctl: Invalid argument
```

If any errors similar to the above are seen, use the linux command `cat /proc/sys/kernel/sem` to display the semaphore limits, and use the `/usr/bin/ipcs -s` command to display the semaphore sets currently in use. To make more semaphore arrays available, use the commands shown above to change the semaphore limits, or to remove semaphore sets that are no longer in use.

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. See: [Supported Job Schedulers \(Page 3-12\)](#). 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 \(Page 3-12\)](#).

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.

Important:

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

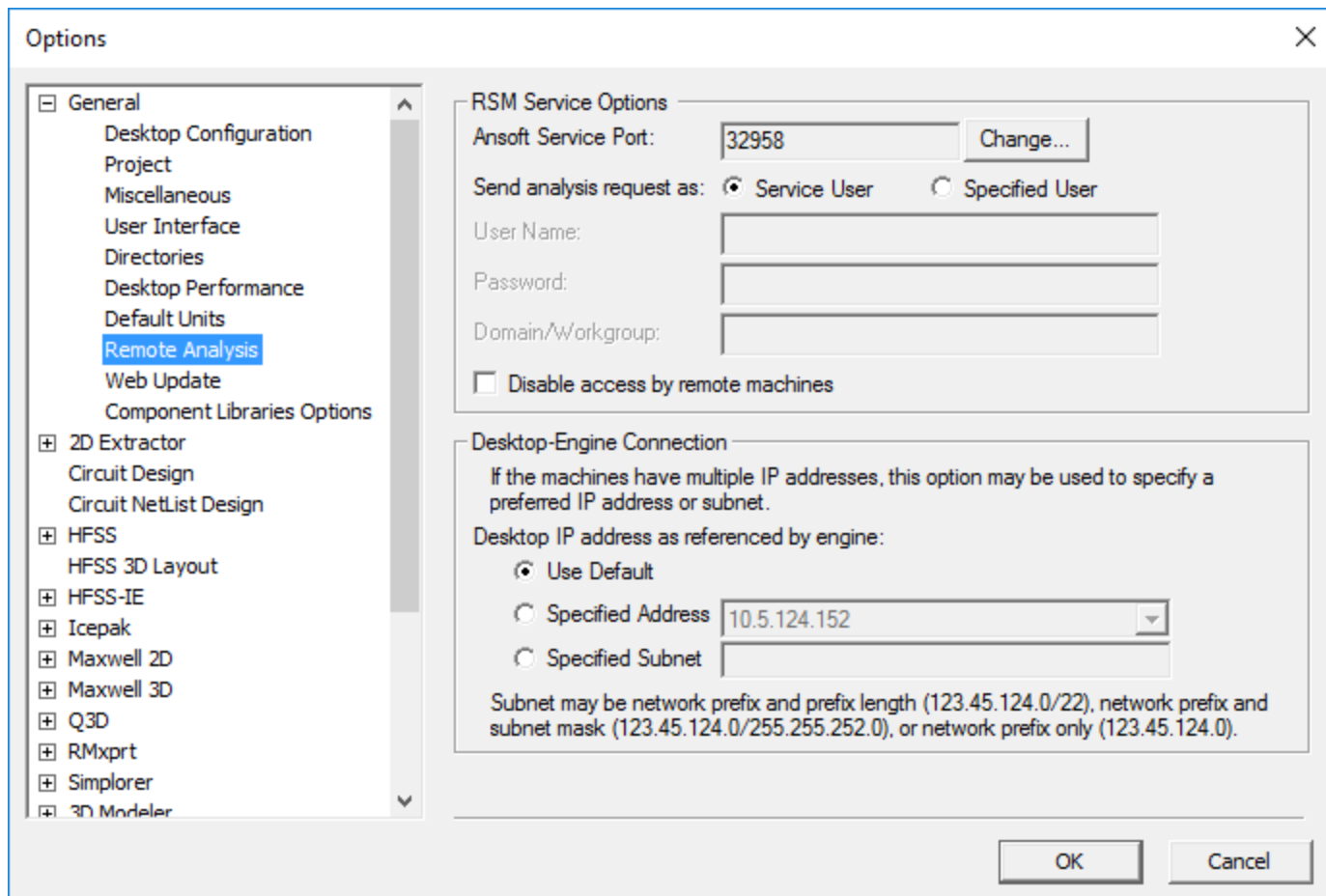
If you use RSM with MPI integration, you must set the environment variable ANSYS_EM_EXEC_DIR.

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.

Note:

If you receive an error that InstallShield did not find enough temporary space to run, set your ISHIELD_TMP environment variable to a different directory. :

1. Exit the installation.
2. From a command prompt, type the following:

```
setenv ISHIELD_TMP <newdir> (csh)
```

```
export ISHIELD_TMP=<newdir> (bash)
```

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 success 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.

Network vs. Non-Network Installation

Both RSM and Ansys Electromagnetics applications can be installed in one central location. You do not have to install the application on each machine/node. Ansys recommends using the network-based approach.

Network Installation

1. Install *all* Ansys Electromagnetics products in a network location accessible from all machines.
2. Install RSM in a network location accessible from all machines.
3. On every machine, configure and start RSM.
4. On one machine, register Ansys Electromagnetics product installations with RSM.

Non-Network Installation

1. Install Ansys Electromagnetics products on every machine.
2. Install RSM on every machine.
3. On every machine, configure and start RSM.
4. On every machine, register Ansys Electromagnetics product installations with RSM.

Configuring RSM

Configuring RSM on a Linux Job Management Host for submitting jobs to a cluster is separate from using RSM to launch solver processes. That is, a single RSM process cannot be used to do both functions. RSM may run as the root user when used to launch solver processes, but it should not be run as the root user when used for submission of jobs to a Linux cluster.

General RSM syntax

RSM's command-line interface uses an LSB-compliant syntax:

*<path to RSM installation>/rsm/ansoftrsmervice **command***

Where *command* can be any of:

- **help** – show full details of all RSM command options.
- **start** – start RSM.
- **status** – show status of RSM.
- **stop** – stop RSM.
- **condrestart** – restart RSM if it is already running.
- **restart** – restart RSM if it is already running; otherwise, start it. Same as reload.
- **reload** – restart RSM if it is already running; otherwise, start it. Same as restart.
- **startonboot** – configures RSM to start automatically on boot, as root. Command must be run as root.
- **startonboot -user** – configures RSM to start automatically on boot as a specific user. Command must be run as root.

Note: RSM should be configured to start automatically on boot on each node to which remote simulation requests will be sent. This will write the appropriate files to `etc/rc.d` and start the RSM service for the named user.

Determining Who Should Run RSM

The user who runs RSM determines who launches solver processes.

User running RSM	User who owns/launches solver processes
Non-root user	User running RSM.
root	User who requests to solve. If that user does not exist on this machine, the user who is running RSM.

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:

- Run `RegisterEngineswithRSM.pl`, located in the installation directory. The command should be run from the installation path to be registered with RSM. For example:

```
<installdirectory>/RegisterEngineswithRSM.pl add
```

The following commands are accepted:

- **add** – add a product installation's RSM engines to RSM.
- **help** – show full details of all command options.
- **remove** – remove a product installation's engines from RSM.
- **status** – show RSM registration status for each RSM engine.

Changing the RSM Listening Port

For Remote Analysis or Distributed Analysis, 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 `ansoftrsmervice.cfg` configuration file. This configuration file is located in the platform specific subdirectory of the RSM installation directory.

In Linux, the default location of this directory is `/opt/ansys_inc/[Primary.VersionNumber]AnsysEM/rsm/Linux64`.

To modify the RSM service configuration:

1. Stop the service.
2. Modify the `ansoftrsmervice.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 `ansoftrsmervice.cfg` file may not contain the lines:

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

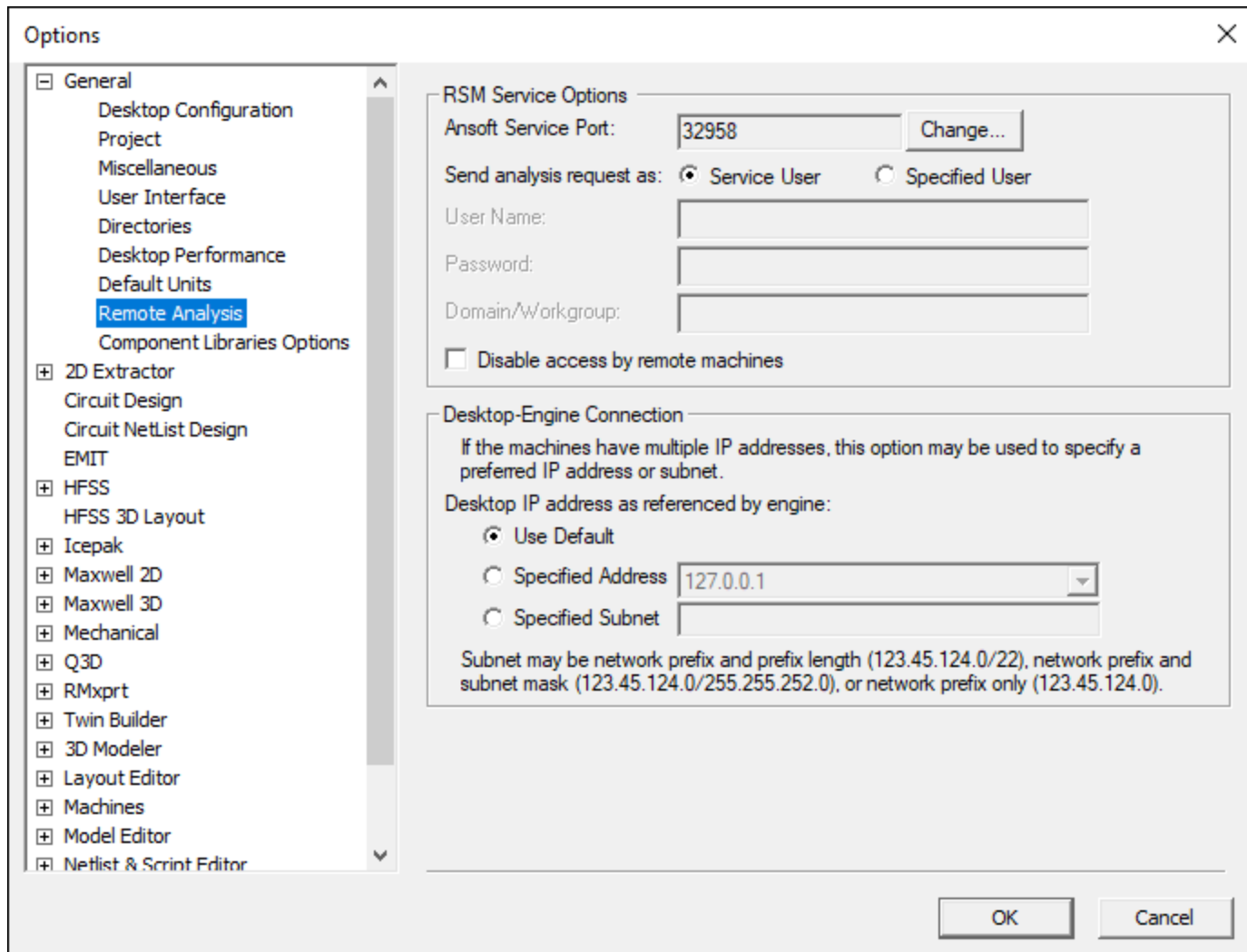
If these lines are not present, then add them to the `ansoftrsmervice.cfg` file after the first line of the file, and change the `ListenPort` to the desired port number. For example, to change the port

number to 32000, the beginning of the ansoftsrmservice.cfg file should look like the following after the changes:

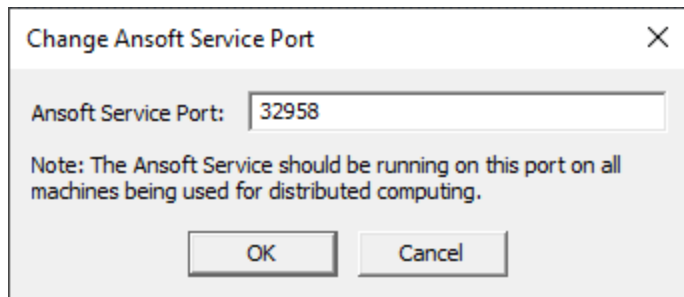
```
$begin 'AnsoftCOMDaemon'
$begin 'Default:CommDetails'
$begin 'CommDetails'
'ListenPort'='32958'
$end 'CommDetails'
$end 'Default:CommDetails'
```

Ansoft 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.

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.

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 \(Page 3-20\)](#).
- MPI should be installed and configured, as described in [MPI Installation \(Page 4-1\)](#).

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.

Submitting and Running the Tests

This section includes general instructions for submitting and running the jobs. For scheduler-specific information, see: [Scheduler-Specific Configuration and Testing \(Page 3-20\)](#).

The test jobs in this section may be submitted to a cluster using the command line or via the Electronics Desktop job submission GUI. When using the GUI, the software will create a job submission command line (the `bsub` command line) from the parameters specified. The options will be selected to effectively use the resources allocated to the job by the scheduler. Users may specify a specific queue and/or parallel environment for the job using the GUI. If no parallel environment is specified, one will be selected automatically. If no queue is specified, the job submission command will not contain a queue specification, and the scheduler will select a queue. The state of the **Use automatic settings** check box and other job submission GUI controls will affect the job submission command options.

Background: Selecting a Scheduler

To select a scheduler from Electronics Desktop:

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. Use the **Select scheduler** drop-down menu to select a scheduler. 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 additional information, 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 \(Page 4-1\)](#) 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 \(Page 4-1\)](#). 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 following Linux schedulers:

- [Altair PBS Professional](#)
- [IBM Spectrum LSF](#)
- [Univa Grid Engine](#)

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

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

Altair PBS Professional

This section describes troubleshooting HPC Test Jobs while using the Altair PBS Professional scheduler.

See: [Running HPC Test Jobs on a Cluster \(Page 3-13\)](#).

Cluster Configuration Guidelines

For cluster jobs, ensure the following environment variables are set correctly:

- PBS_CONF
- PBS_SERVER
- PBS_DEFAULT

If the PBSPro configuration file path has been changed from the default (/etc/pbs.conf), the PBS_CONF environment variable should be set to the correct path.

If the PBS_SERVER parameter in the PBSPro configuration file is not set to the default PBSPro server name, the PBS_DEFAULT environment variable should be set to the default server name.

Users who submit parallel jobs using the job submission GUI may select between several different methods for specifying the resources for the job. If the "Number of Nodes and Cores" option is selected, with the number of tasks determined automatically, Ansys Electromagnetics software will select nodes with the greatest number of cores per node that meet all job requirements. This may not be the best use of resources for non-homogenous clusters (those where nodes have different configurations).

In some cases, a job may be allocated one or more nodes with many cores per node, even if the job could run using the same number of nodes with fewer cores per node. To mitigate this issue,

create distinct queues for different job sizes, and specify the appropriate queue during job submission.

Running Test Jobs with Altair PBS Professional

If there is a problem connecting to the scheduler when the job submission node is the local node (where the GUI is running), ensure the following:

- 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 product path must be accessible by the user running the job from the cluster execution node(s) where the job runs.
- The project path 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 specified temp directory 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

If the job fails to complete successfully, check the items above and:

- Verify that the PBS `pbsdsh` command is able to start the `ansoftsm` process on the other host.

MPI Job Distributed to Multiple Hosts

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

- Verify that the MPI Vendor `batchoption` is set to "Intel".
- Verify that the `RemoteSpawnCommand` is specified via a `batchoption` setting when submitting the job; otherwise the default, SSH, will be used for the job.
- Verify that the specified MPI Vendor is installed and configured correctly for the user submitting the job. See: [MPI Installation \(Page 4-1\)](#).

IBM Spectrum LSF

This section describes troubleshooting HPC Test Jobs while using the IBM Spectrum LSF scheduler.

See: [Running HPC Test Jobs on a Cluster \(Page 3-13\)](#).

Cluster Configuration Guidelines

In order to submit exclusive jobs, users must have access to queues configured for such. If users need to submit exclusive jobs to a queue, set the queue's EXCLUSIVE property to Y.

Ansys Electromagnetics products allow users to submit jobs for which the user specifies the "Number of Cores and (Optional) RAM." However, the RAM per core may only be specified if the LSF cluster is configured with the parameter RESOURCE_RESERVE_PER_SLOT. This parameter should be set to allow user to specify the RAM per core for the job.

Running Test Jobs with IBM Spectrum LSF

If there is a problem connecting to the scheduler when the job submission node is the local node (where the GUI is running), ensure the following:

- 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, ansyedt.

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 product path must be accessible by the user running the job from the cluster execution node(s) where the job runs.
- The project path 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 specified temp directory 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

If the job fails to complete successfully, check the items above and:

- Verify that the LSF blaunch command is able to start the anssoftfrm process on the other host.

MPI Job Distributed to Multiple Hosts

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

- Verify that the MPI Vendor batchoption is set to "Intel".
- Verify that the specified MPI Vendor is installed and configured correctly for the user submitting the job. See: [MPI Installation \(Page 4-1\)](#).

Univa Grid Engine (UGE)

This section describes troubleshooting HPC Test Jobs while using the UGE scheduler.

See: [Running HPC Test Jobs on a Cluster \(Page 3-13\)](#).

Note:

The information in this section is generally applicable to previous iterations of UGE, including Sun Grid Engine (SGE) and Oracle Grid Engine (OGE).

Cluster Configuration Guidelines

qrsh Command

Ansys EM parallel batch jobs use the **qrsh** command to launch engine processes on remote hosts. If the qrsh command is not working correctly, the parallel job is unable to launch engine processes on remote hosts. If this problem occurs, the batch log for the job typically includes one or more error messages indicating that a COM engine was unable to be started on a remote host.

The **qrsh** command may be tested by running a simple command on a specified host, such as **qrsh -l hostname=host1 hostname** or **qrsh -l hostname=host1 ls /tmp**, where **host1** is the remote hostname. The first test should echo back the hostname of the remote machine. The second test should list the contents of the /tmp directory on the remote machine.

Failures are often associated with certain global configuration parameters, listed below with values that may cause the failures:

- qrsh_command /usr/bin/ssh -t
- rsh_command /usr/bin/ssh -t
- rlogin_command /usr/bin/ssh -t

If these parameter settings are removed, the UGE built-in mechanisms are used for qrsh, rsh, and rlogin. No problems with the built-in versions have been reported. The **qconf -sconf** global command may be used to view these parameter settings, and **qconf -mconf** may be used to modify them.

Parallel Environments

For these clusters, no parallel environment is needed to submit a job that is allocated only one core on one host. A parallel environment is required to submit a job that requires two or more cores, whether the cores are on the same host or on different hosts. Appropriate parallel environments should be configured for the cluster to run the types of parallel jobs that the users would like to run.

Important:

All parallel environments for Ansys Electromagnetics jobs must be configured with the `control_slaves` parameter set to TRUE.

If the **Use automatic settings** check box is not selected, you must also specify the total number of tasks, T , for the job, and the number of cores per task, C . The allocations must be integers. The software will then allocate the same number of tasks, N , on each node, where N is determined by the nodes available for the job and the user-specified queue and/or parallel environment. A minimum of $N * C$ cores is required per node, so the allocation rule must be an integer that is equal to or greater than $N * C$. Note that if the allocation rule is an integer greater than $N * C$, the extra cores allocated to the job are not used, leading to less than ideal resource usage. Ansys recommends configuring multiple parallel environments with different integral allocation rules to allow a good match of resources needed by the job to the resources allocated to the job.

If the **Use automatic settings** check box is selected, you can specify the **Number of Nodes and Cores** for the job or the **Number of Cores and (Optional) RAM** for the job.

- For **Number of Nodes and Cores**, the job is allocated the same number of cores on each node, and the parallel environment must be an integer. The parallel environment configurations above are recommended.
- For **Number of Cores and (Optional) RAM**, a parallel environment with any allocation rule may be used. Ansys recommends that at least one parallel environment with an allocation rule of `$fill_up` be provided, as this allows the job to be mostly run on hosts where no other job is running. An allocation rule of `$round_robin` may also be useful to allow such jobs to be distributed across all available hosts.

Submitting Exclusive Jobs

Some users require or desire the ability to submit jobs to a cluster that have exclusive access to each host allocated to the job. That is, any host running the job is not available for use by any other jobs. This may be needed to limit contention for resources that are not tracked by the cluster, such as network bandwidth. By default, a UGE cluster may not be configured for exclusive jobs. If the cluster configuration includes a complex of type `BOOL`, with `relop EXCL`, that is both consumable and requestable, then this complex may be used to submit exclusive jobs.

Configuring an Exclusive Resource

1. If there is no complex configured as per above, use the following command to add a new complex to the table of complexes:

```
qconf -mc
```

Recommended attributes are:

- name : exclusive
 - shortcut : excl
 - type : BOOL
 - relop : EXCL
 - requestable : YES
 - consumable : YES
 - default : 0
 - urgency : 0
2. After the complex is configured, the value of this complex for each execution host must be set. For each execution host, run the following command:

```
qconf -me hostname
```

This will display the host configuration settings for the specific host.

There may be multiple lines of settings, but the one you must configure is:

```
complex_values exclusive=TRUE
```

Submitting an Exclusive Job from the Command Line

When submitting a job using the `qsub` command, the job will have exclusive access to each host allocated to the job if the exclusive resource name ("exclusive") or exclusive resource shortcut ("excl") is included in the resource list specified by the `qsub -l` option. If the resource list does not include the exclusive resource name or the exclusive resource shortcut, then the job will not have exclusive access to each host allocated to the job; other jobs may run on the same host or hosts at the same time as this job.

Example **qsub** command line for exclusive serial job:

```
qsub -b y -l excl /opt/AnsysEM/v242/ansysedt -ng -BatchSolve -machinelist num=1 ~/Ansoft/OptimTee.aedt
```

Although serial jobs use only one slot, no other jobs will run on the host where this job is running, even if additional slots are present.

Example **qsub** command line for exclusive parallel job using eight engines, each using a single thread of execution:

```
qsub -b y -l excl -pe pe1 8 /opt/AnsysEM/v242/ansysedt -ng -BatchSolve -Distributed -  
machinelist num=8 ~/Ansoft/OptimTee.aedt
```

None of the hosts allocated to this job will be allowed to run other jobs while this job is running.

Submitting an Exclusive Job using the Job Submission GUI

For jobs submitted to a UGE cluster using the job submission dialog box, the state of the **Nodes are for exclusive usage by this job** check box controls whether the job is submitted as an exclusive job. This check box is located on the **Compute Resources** tab of the job submission dialog box. If this check box is selected, the job submission command line will include the exclusive resource option. The resource name does not need to be “exclusive”; any complex of type **BOOL**, with **relop EXCL**, that is both consumable and requestable may be used.

Consumable Memory Limits

When submitting memory intensive jobs on a UGE cluster, it may be useful to specify a consumable memory limit for the job. If the resource request for the job specifies a consumable limit, then, while the job is running, the specified resource is not available for other jobs. Jobs requiring the same resource will only be scheduled on a host if the total quantity of the resource on the host exceeds the sum of the consumable resource needs of the jobs. For non-consumable resources, the resource requirement for each job is independently compared to the quantity of the resource available on the host. Consumable resources may be configured as per-slot resources or as per-job resources.

Consumable memory limits are important for jobs submitted to a UGE cluster with the **Use automatic settings** check box selected and the user specifying the **Number of Cores and (Optional) RAM** for the job. If the RAM per core is specified by the user, then the job submission command includes a consumable memory resource requirement. If a complex of type **MEMORY** is configured to be requestable and consumable (per-slot, rather than per-job), then this complex will be used for the consumable memory resource requirement to meet the RAM per core requirement specified by the user. If no such complex is configured, then the RAM per core may not be specified by the user.

Some versions of Grid Engine come with "m_mem_free" and "mem_free" complexes already configured. If either one or both are configured as a requestable and consumable per-slot resource, there is no more configuration required. For other grid engine versions, it may be necessary to configure a complex to use as a memory resource requirement to allow users to use automatic settings and specify the Number of Cores and RAM for the job.

To configure this resource, add a complex with the following attributes:

- name: m_mem_free or mem_free or another name, if desired
- shortcut: mfree or mf or another alias, if desired
- type: MEMORY
- relop: <=
- requestable: YES

- consumable: YES
- default: 0
- urgency: 0

The **sconf -sc** command may be used to examine the complexes and their configurations.

The **sconf -mc** command may be used to modify the complexes or their configurations.

In addition to adding the complex to the cluster for the requestable and consumable memory resource, each host must be configured to specify the limit of this resource. The **sconf -sel** command may be used to list the execution hosts of the cluster. The **sconf -se hostname** command may be used to examine the configuration of the host with name **hostname**. The **sconf -me hostname** command may be used to modify the configuration of the host with name **hostname**. The memory limit should be added to the `complex_values` for the host. If there is no `complex_values`, then the `complex_values` entry should be added: the value of the `m_mem_free` complex must be the same as the amount of memory on the host. For example, if the complex name is `m_mem_free`, and the host has 32 GB of memory, then the `complex_values` of the host configuration should include the specification “`m_mem_free=32G`”. Different complex values are separated by commas in the host configuration.

Running Test Jobs with Univa Grid Engine (UGE)

If there is a problem connecting to the scheduler when the job submission node is the local node (where the GUI is running), ensure the following:

- 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 product path must be accessible by the user running the job from the cluster execution node(s) where the job runs.
- The project path 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 specified temp directory 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

If the job fails to complete successfully, check the items above and:

- Verify that the `qsh` command is able to start the `ansoftrsm` process on the other host.

MPI Job Distributed to Multiple Hosts

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

- Verify that the MPI Vendor `batchoption` is set to "Intel".
- Verify that the `RemoteSpawnCommand` is specified via a `batchoption` setting when submitting the job; otherwise the default, SSH, will be used for the job.
- Verify that the specified MPI Vendor is installed and configured correctly for the user submitting the job. See: [MPI Installation \(Page 4-1\)](#).

4 - Message Passing Interface (MPI) Installation

For Linux, the supported Message Passing Interface (MPI) option in Ansys Electromagnetics Suite is Intel. MPI is required only when using multiple nodes. MPI is never required when solving on a single node.

Important:

Intel MPI is provided in the Ansys Electromagnetics Suite installation. It is not necessary to install MPI on any machine.

Configuring Intel MPI

To configure Intel MPI, perform the following steps:

1. Install Ansys Electromagnetics Suite.
 - Ansys recommends installing in a single location on a networked drive that is available to all machines.
 - If installing on each machine individually, ensure the install location is the same on each machine (for example, /share/install/).
2. Install and configure the Remote Simulation Manager (RSM), if needed.
 - If the first machine in the distributed machine list is the local machine, you do not need to install or run RSM.
 - If you are running through a scheduler, the scheduler will launch the simulation; you do not need to install or run RSM.
 - If the first machine in the distributed machine list is not the local machine and you are not using a scheduler, you must use RSM. Install RSM on the remote machine that will host the simulation, and register Ansys EM Suite with the RSM on that machine. For more information, consult the online help.
3. Set up passwordless ssh to enable communication between machines. At a minimum, this must be set up from the simulation host machine to all remaining machines.
 - First verify that passwordless ssh is not already configured by running the following command on one node to access another:

```
ssh <remote node> echo hi
```
 - If you receive a response of 'hi', passwordless ssh is already configured between this node and the remote node. If you have already configured passwordless ssh and it is not working, run the following command to obtain information for diagnosing the problem:

```
ssh -v -v -v <remote node> echo hi
```

- See: [Passwordless SSH Setup \(Page 4-3\)](#).
4. Define the temp directory on all machines and assign Full Control to the designated user.
 5. If a firewall is installed, it should be off or configured as specified in the online help.
 6. To select Intel MPI:
 - Launch Electronics Desktop.
 - Click **Tools > Options > HPC and Analysis Options**.
 - On the **Options** tab, click the **MPI Vendor** field and select **Intel**.
 - Also select whether to use SSH (recommended) or RSH. If using an LSF or UGE scheduler, a third option (Scheduler) is available.

Additional Information on RSM Settings

If you choose a remote machine to host the simulation, RSM needs to be installed and running on that remote machine. There are two RSM settings that can affect the simulation. The first is the user running RSM. The second setting is in Electronics Desktop (**Tools > Options > General Options > Remote Analysis**). Choose to send the analysis request as a **Service User** or a **Specified User**. Here is additional information about these settings:

RSM User	Remote Simulation Options	Simulation runs on remote machine as	Notes
root	Service User	The user who is running Ansys Electronics Desktop.	RSM will try to run as the user, but may fail based on local privileges.
root	Specified User = user_A	user_A	Runs as user_A
User_A	Service User	user_A	Runs as user_A
User_A	Specified User = user_B	Will fail due to privileges.	Not recommended.

MPI Batchoptions

Several settings can be controlled via batchoptions. See below.

MPI Vendor

Control the MPI Vendor using the following settings.

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

Remote Spawn Command

Set the Remote Spawn Command to 'RSH', 'SSH', or 'Scheduler' using the following settings:

- 'HFSS/RemoteSpawnCommand'='RSH'
- 'HFSS 3D Layout/RemoteSpawnCommand'='RSH'
- 'Icepak/RemoteSpawnCommand'='SSH'
- 'Maxwell 2D/RemoteSpawnCommand'='SSH'
- 'Maxwell 3D/RemoteSpawnCommand'='Scheduler'
- 'Q3D Extractor/RemoteSpawnCommand'='Scheduler'

The Scheduler option can only be used for jobs running under an LSF or UGE scheduler and only if the MPI Vendor is Intel.

Setting up passwordless ssh

For initial setup, perform the following steps:

1. Generate an ssh key:

```
ssh-keygen -t rsa
```

2. Save the file to the default location (\$HOME/.ssh/id_rsa).
3. Press **Enter** for no passphrase.
4. Press **Enter** again to confirm no passphrase.
5. Ensure that permissions are correct for the following directories:
 - `chmod go-w ${HOME}`
 - `chmod 700 ${HOME}/.ssh`
 - `chmod 600 ${HOME}/.ssh/id_rsa`
6. Append your personal public key file to your personal authorized_keys list:

```
cat ${HOME}/.ssh/id_rsa.pub >> ${HOME}/.ssh/authorized_keys
```

```
cat ${HOME}/.ssh/id_rsa.pub >> ${HOME}/.ssh/authorized_keys2 (if required)
```

7. Test that passwordless ssh is working:

```
ssh <local node> echo hi
```

If you are prompted for a password, there is likely an access issue. Diagnose the problem by running the following:

```
ssh -v -v -v <local node>
```

8. If your HOME directory is not shared across machines:

- Copy the contents of your local id_rsa.pub file to the authorized_keys file on the remote machine:

```
cat ${HOME}/.ssh/id_rsa.pub | ssh <remote node> 'cat >>
```

```
${HOME}/.ssh/authorized_keys'; ssh <remote node> 'chmod 600
```

```
${HOME}/.ssh/authorized_keys'
```

9. Verify that you can log into the remote node from the local node:

```
ssh <remote node> echo hi
```

If the command returns 'hi', configuration is complete. If it fails, diagnose the problem by running:

```
ssh -v -v -v <remote node>
```

Troubleshooting MPI

This section contains some frequently asked questions and error messages.

Frequently Asked Questions

Why isn't the installation wizard displaying correctly on a dual monitor setup?

If there is a conflict with the RedHat gnome-shell window manager during installation, replace gnome-shell via the terminal emulator command shell with “metacity –replace &” and continue with the installation. After the installation is complete, change “metacity – replace &” back to “gnome-shell – replace &”.

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.
- 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.

MPI Job Initialization Error

During MPI initialization, you may receive an error like the following:

Unable to create child process.

Could not start memory inquiry.

This most often happens if there is an issue with the host key based authentication. Check the terminal where you launched the software for a warning like this one:

The authenticity of host '<hostname> (<ip address>)' can't be established.

Manually initiate an ssh connection from your current machine to this host using the exact host-name as listed in the warning. Answer yes at the prompt to save the host key.

5 - 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 Slwave 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**.

6 - Uninstalling the Software

On Linux platforms, you can simply remove all of the contents of the installation folder to uninstall the Electromagnetics Suite, RSM, or the Layout Integrations for ECAD. If you plan to upgrade to a newer version of the software, you will want to leave the **AnsysEM/Shared** subfolder, which contains the license configuration. If you have more than one version installed, only remove the folder containing the version you wish to remove.

You must have the necessary privileges in the installation path to be able to remove the files and folders contained there.

Uninstalling the Electromagnetics Suite

1. In a terminal window, use the **cd** command to change to the top-level folder where the Ansys Electromagnetics product is installed (typically opt/AnsysEM):

```
cd /<Installation_Path>/AnsysEM
```

Where *<Installation_Path>* is the the path to the parent folder of the AnsysEM folder.

2. Type the following command to remove the version-specific folder and all files and subfolders it contains:

```
rm -rf vxxx
```

Where *xxx* is the version number to uninstall. For example, v242 is the folder for version 2024 R2.

3. Press **Enter**.

It may take a few moments to delete all of the files and subfolders.

4. Optionally, remove any shortcut or script you might have created to launch the product just uninstalled.

Uninstalling RSM

The procedure for uninstalling RSM is the same as the procedure for uninstalling the Electromagnetics Suite, except for the name of the folder to remove (specifically, **rsm**).

1. In a terminal window, use the **cd** command to change to the top-level folder where the Ansys Electromagnetics product is installed (typically opt/AnsysEM):

```
cd /<Installation_Path>/AnsysEM
```

Where *<Installation_Path>* is the the path to the parent folder of the AnsysEM folder.

2. Type the following command to remove the rsm folder and all files and subfolders it contains:

```
rm -rf rsm
```

3. Press **Enter**.

It may take a few moments to delete all of the files and subfolders.

4. Optionally, remove any shortcut or script you might have created to launch RSM, unless you plan on reinstalling it.

Uninstalling Layout Integrations for ECAD

1. In a terminal window, use the **cd** command to change to the top-level folder where the Ansys Electromagnetics product is installed (typically opt/AnsysEM):

```
cd /<Installation_Path>/AnsysEM
```

Where *<Installation_Path>* is the the path to the parent folder of the AnsysEM folder.

2. Type the following command to remove the version-specific folder and all files and subfolders it contains:

```
rm -rf LayoutIntegrationsxx.x
```

Where *xx.x* is the version number to uninstall. For example, LayoutIntegrations24.2 is the folder for version 2024 R2.

3. Press **Enter**.

It may take a few moments to delete all of the files and subfolders.

A - Silent Installation

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

There are two ways to perform silent installations:

- [Run the installer from the command line](#) silently and include all necessary options.
- [Record a response file](#) during an initial command line installation using the GUI mode. For subsequent silent installations, specify the response file to use, ensuring duplicate configurations on all subsequent workstations.

Setup.exe Command Line Options

Options for running the installation program (setup.exe) can be included directly in the command line or indirectly from an installer properties file. The syntax is as follows:

```
setup.exe [-f <path_to_installer_properties_file> | -options]
```

Where the available **options** are:

- **-?**, **-h**, **-help**, or **--help**: Display the setup.exe command help information.
- **-i [gui | silent]**: Choose the installation mode (graphical user interface or silent; **-i gui** is the default mode).
- **-D<name>=<value>**: Specify [installer properties](#).
- **-r <path/name_to_generate_response_file>**: Generates a response file with the specified path and name.

Note:

- The path to the installer properties file may be either absolute, or relative to the directory in which the installer resides.
- If an installer properties file is specified and exists, all other command line options will be ignored.
- If a properties file named **installer.properties** or **setup.exe.properties** is in the same folder as the installer, it is used automatically (overriding all other command line options), unless **-f** is used to point to another valid properties file.
- If an installer properties file is specified but does not exist, the default properties file is used, if present. Otherwise, any specified command line options will be used or, if omitted, the installer runs using default settings.
- The setup.exe command help displays additional options beyond those listed here. This list has been condensed to the options that are relevant to installation of the Ansys Electromagnetics Suite.

Installer Properties

Regardless of which silent installation methods you choose, the following installer properties are applicable. Specify installer properties only when you wish to deviate from the default value.

Name	Description	Default
USER_INSTALL_DIR	Installation folder path	/opt/AnsysEM
TMP_DIR	Temp folder path	/tmp
TMP_RW	Add read/write access to the Temp folder for all users?	1
LIBRARY_LOCAL_INSTALL	Libraries will be accessed from this local installation by default	1
LIBRARY_COMMON_INSTALL	Libraries will be accessed from a common installation by default	0
LIBRARY_COMMON_DIR	Common libraries folder path	<USER_INSTALL_DIR>/ansys_inc/v242/AnsysEM
SPECIFY_LIC_CFG	I want to specify the license configuration	0 (see Note a) or 1 (see Note b)
KEEP_LIC_CFG	I want to keep the current license configuration	1 (see Note a) or 0 (see Note b)
LICENSE_SERVER1	Name of 1st or sole license server	<local machine> (see Note c)
REDUNDANT_SERVERS	Specify redundant servers?	0 (see Note b)
LICENSE_SERVER2	Name of 2nd license server	<blank> (see Note c)
LICENSE_SERVER3	Name of 3rd license server	<blank> (see Note c)
SPECIFY_PORT	Specify license server TCP/IP port number?	0 (see Note c)
LICENSE_PORT	License server TCP/IP port number	1055 (see Note c)

Notes:

- This is the default value if an existing license configuration is found on the machine.
- This is the default value if no license configuration is found on the machine.
- The listed default value applies only if no license configuration is found on the machine. Otherwise, the default is the value from the existing configuration.

Silent Installation from the Command Line

The command line method of installation is a convenient way to specify any overrides to default options in a single command line, which can be run unattended on multiple computers to produce duplicate configurations. In a silent installation, the graphical user interface (GUI) does not appear and all messages and prompts are suppressed.

Refer to the supported [setup.exe command line options](#) and available [installer properties](#). Proceed as follows:

1. Open the command prompt.
2. Use the **cd** command to navigate to the installation directory containing setup.exe (Linux/AnsysEM/Disk1/InstData).
3. Run **setup.exe** using the **-i silent** option and append any [installer properties](#) for which you want to deviate from the default value. Each property is prefixed with **-D** (with no space between the D and the property name). Several examples follow:

Command Line Examples

- **Example 1:** Override the default installation directory and specify a single license server:

```
setup.exe -i silent -DUSER_INSTALL_DIR=<install_dir> -DSPECIFY_LIC_CFG=1 -DLICENSE_SERVER1=<license_server>
```

- **Example 2:** Force keeping the existing license configuration – if none currently exists, this option will create a new configuration (specifying the local machine as the license server):

```
setup.exe -i silent -DKEEP_LIC_CFG=1
```

- **Example 3:** Configure a common library directory:

```
setup.exe -i silent -DLIBRARY_COMMON_INSTALL=1 -DLIBRARY_COMMON_DIR=<common_library_dir>
```

- **Example 4:** Configure a custom temporary file directory and do *not* add read/write access to this folder for all users:

```
setup.exe -i silent -DTMP_DIR=<temp_dir> -DTMP_RW=0
```

- **Example 5:** Configure redundant license servers:

```
setup.exe -i silent -DSPECIFY_LIC_CFG=1 -DLICENSE_SERVER1=<license_server_1> -DREDUNDANT_SERVERS=1 -DLICENSE_SERVER2=<license_server_2> -DLICENSE_SERVER3=<license_server_3>
```

Silent Installation Using a Response File

Install the Ansys Electromagnetics Suite using the installer's graphical user interface (GUI). Launch the installer from the command line one time and record a response file. Then use that response file to install the software on subsequent computers.

The advantage of this method is that you can automate the installation process but don't have to be concerned about the [installer properties](#) and their syntax. The GUI guides you through the installation options, and the correct properties are recorded to the response file. However, if you choose to edit the response file after recording it, you will have to maintain the correct property names and syntax.

Recording a Response File

To record a response file:

1. Open the command prompt.
2. Use the `cd` command to navigate to the installation directory containing `setup.exe` (Linux/AnsysEM/Disk1/InstData).
3. Run the following command:

```
setup.exe -r "<path/name_to_generate_response_file>"
```

Specify **.properties** as the response file extension.

Note:

The **-i gui** option has been omitted because *graphical user interface* is the default installation mode.

4. Follow the installation prompts.

Once completed, the **.properties* response file will contain information you can use to install the same product with the same configuration on other machines.

Response File Format

Any line in the response file beginning with the `#` symbol is a comment. Blank lines are ignored.

You can change values only to the right of the `=` sign. See: [Example Response File \(Page A-7\)](#).

Using a Response File

To use the response file:

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

1. Open the command prompt.
2. Use the `cd` command to navigate to the installation directory containing `setup.exe` (Linux/AnsysEM/Disk1/InstData).
3. Run the following command:

```
setup.exe -f "<response_file_path/name>" -silent
```

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

4. A zero (0) exit code is returned to the command window after a successful installation. If a code other than 0 is encountered, consult the [Installer Result Codes](#) table for information concerning the cause of the failure. Contact Ansys technical support for additional assistance if needed.

Note:

Installations typically fail for one of the following reasons:

- An invalid path was specified.
- Free space is insufficient on the installation drive.
- There were differences in the property sequences between the recorded response file and the current silent installation. For example, an old response file is being used for a newer software version (in which properties have been added or removed), or an additional prompt must be satisfied due to differences in the two computer systems.

Installer Result Codes

The following table lists all response codes produced by InstallAnywhere. Not all of them are necessarily applicable to the Linux 64-bit platform or the Ansys Electromagnetics Suite:

Code	Description
0	Success: The installation completed successfully without any warnings or errors.
1	The installation completed successfully, but one or more of the actions from the installation sequence caused a warning or a non-fatal error.
-1	One or more of the actions from the installation sequence caused a fatal error.
1000	The installation was canceled by the end user.
1001	The installation includes an invalid command-line option.
2000	Unhandled error.
2001	The installation failed the authorization check, may indicate an expired version.
2002	The installation failed a rules check. A rule placed on the installer itself failed.
2003	An unresolved dependency in silent mode caused the installer to exit.
2004	The installation failed because not enough disk space was detected during the execution of the Install action.
2005	The installation failed while trying to install on a 64-bit Windows system, but installation did not include support for 64-bit Windows systems.
2006	The installation failed because it was launched in a UI mode that is not supported by this installer.
2009	Indicates that the user attempted to launch multiple instances of an installer at the same time even though the installer was configured to prevent multiple launches.
3000	Unhandled error specific to a launcher.
3001	The installation failed due to an error specific to the lax.main.class property.
3002	The installation failed due to an error specific to the lax.main.method property.
3003	The installation was unable to access the method specified in the lax.main.method property.
3004	The installation failed due to an exception error caused by the lax.main.method property.

Code	Description
3005	The installation failed because no value was assigned to the lax.application.name property.
3006	The installation was unable to access the value assigned to the lax.nl.java.launcher.main.class property.
3007	The installation failed due to an error specific to the lax.nl.java.launcher.main.class property.
3008	The installation failed due to an error specific to the lax.nl.java.launcher.main.method property.
3009	The installation was unable to access the method specified in the lax.nl.launcher.java.main.method property.
4000	A Java executable file could not be found at the directory that was specified by the java.home system property.
4001	An incorrect path to the installer jar caused the relauncher to launch incorrectly.
5000	Modification of existing instance failed because the instance has not been uninstalled properly or because the product registry has been corrupted.
7000	The installation was rolled back due to a fatal exception.
8000	The upgrade was canceled because a newer version of the product is already installed on the target system.
8001	The end user canceled the upgrade.
8002	The upgrade exited because the earlier version of the product could not be successfully uninstalled.

Example Response File

```
#####
#
# InstallAnywhere Options File
#
# -----
# This file was built by the Replay feature of InstallAnywhere.
# It contains variables that were set by Panels, Consoles or Custom Code.
```

ANSYS Electromagnetics Suite – Legacy Linux Installation Guide (2024 R2 and Earlier)

```
#####
#
# ANSYS Electromagnetics Install Location
#
# Specify a valid directory into which the product should be installed.
# If the directory contains spaces, enclose it in double-quotes.
# Example: To install the product to /home/user/My Product, use:
#
#     USER_INSTALL_DIR="/home/user/My Product"
#
# Choose Install Folder
# -----

USER_INSTALL_DIR=/tmp/AnsysEM

#####
#
# Temporary Path
#
# Specify directory to be used by default for temporary files. The
# Temp directory is used during local and remote simulations for temporary
# workspace. Users may configure a different temp directory from this
# installation default. This folder will be used during local and/or
# remote simulations as temporary work space. If the path contains
# spaces, enclose it in double-quotes.
#
# Select Temporary Directory
# -----

TMP_DIR=/scratch

#####
#
# Temporary Folder Permissions
#
# If multiple users will run simulations from this installation, they need
# read/write permissions on the Temp directory.
# Changes permissions to 777 on temporary directory by setting TMP_RW=1.
# Changes permissions to 755 on temporary directory by setting TMP_RW=0.
#

TMP_RW=1
```

```
#####  
#  
# Library Option  
#  
# Select whether new users should use a common library location by default.  
# This product supports the common libraries installation, a centralized  
# installation of materials, components, and other libraries used in Ansys  
# Electromagnetics products. You may also add custom materials that are  
# automatically accessible to all users and products that use this common  
# library installation.  
#  
# This is the state of the library option radio buttons.  
# Selecting Install Libraries:  
# LIBRARY_LOCAL_INSTALL = 1 and LIBRARY_COMMON_INSTALL=0  
# To Use Existing Libraries:  
# LIBRARY_LOCAL_INSTALL = 0 and LIBRARY_COMMON_INSTALL=1  
#  
# Choose Library Option  
# -----  
  
LIBRARY_LOCAL_INSTALL=1  
LIBRARY_COMMON_INSTALL=0  
  
#####  
#  
# Library Path  
#  
# This is the path for which the client will retrieve material libraries.  
# This path should point to the default libraries installed with the client,  
# or pointto a location of libraries installed in a common shared network  
# location. If the path contains spaces, enclose it in double-quotes.  
#  
  
LIBRARY_COMMON_DIR=  
  
#####  
#  
# License Configuration Option  
#  
# Here the option is specified where licenses for this software will be  
# served. This is the state of the license option radio buttons. Valid  
# entries are:
```

```
#
# To specify license server:
#
# SPECIFY_LIC_CFG=1 and KEEP_LIC_CFG=0
# To keep the current license configuration:
# SPECIFY_LIC_CFG=0 and KEEP_LIC_CFG=1
#
# License Information
# -----

SPECIFY_LIC_CFG=1
KEEP_LIC_CFG=0

#####
#
# Client license server name (1)
#
# This is the machine name for the client license server. There can only be
# one single server, or three redundant servers.
#

LICENSE_SERVER1=pitrh6licsrv1

#####
#
# REDUNDANT SERVERS
#
# If SPECIFY_LIC_CFG=1 and you also want to specify the redundant license
# servers, set: REDUNDANT_SERVERS=1
#

REDUNDANT_SERVERS=0

#####
#
# Client license server name (2)
#
# This is the machine name for the second of three client license servers.
# There can only be one single server, or three redundant servers.
#

LICENSE_SERVER2=
```

```
#####  
#  
# Client license server name (3)  
#  
# This is the machine name for the third of three client license servers.  
# Therec an only be one single server, or three redundant servers.  
#  
  
LICENSE_SERVER3=  
  
#####  
#  
# TCP/IP Port  
#  
# This is the TCP/IP port used to communicate with the license server.  
# To specify port: SPECIFY_PORT=1 and modify LICENSE_PORT according to  
# the desired value. By default, LICENSE_PORT is set to 1055  
#  
  
SPECIFY_PORT=0  
LICENSE_PORT=1055  
  
#####  
#  
# The value "Yes" in this Install part should not be modified and left as  
# it is for a successful install. The path of ".CreateSymlinks.bash" must  
# be modified to the valid installation path on the target machine.  
# For example: /tmp/AnsysEM will be replaced by the value as specified  
# for USER_INSTALL_DIR  
#  
# Install  
# -----  
  
-fileOverwrite_/tmp/AnsysEM/v242/scripts/.CreateSymlinks.bash=Yes
```


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	<ul style="list-style-type: none"> The ethernet address of a network interface card (MAC address). Flexera FLEXid USB port hardware key. Disk serial number.
Linux	<ul style="list-style-type: none"> Machine hostid 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 hostids.

Obtaining a Machine Hostid

A machine hostid is present on most Linux machines. To determine the hostid for a supported Linux machine, enter the following command:

```
hostid
```

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 terminal, type the following:

```
ifconfig
```

3. Press **Enter**.

A list of network interface cards appears.

Devices with an ethernet address show a **Link encap** value of **Ethernet**. The ethernet address for each device is the value in the **HWaddr** field, without colons.

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 Linux, you must set this environment variable manually. For example:

`/opt/ansys_inc/v242/AnsysEM`

Note: The environment variable must be set on all hosts if the analysis is distributed to multiple hosts. For Linux, the user must ensure that the environment variable is set for the AnsoftRSM Service daemon processes.

Index

A

Ansys Electromagnetics Suite

installing 2-1

Ansys Workbench

integration with 2-5

D

distributed analysis

options 3-1

H

high performance computing 3-1

running test jobs 3-13

hostid B-1

obtaining ethernet address B-1

obtaining hardware key B-2

obtaining machine hostid B-1

I

installation

installing the software 2-1

prerequisites 1-1

L

license hostid B-1

licensing

hostid B-1

license file 1-1

license manager 1-1

license server 1-1

M

Maxwell

enabling system coupling C-1

Message Passing Interface (MPI) 4-1

notes 4-2

MPI installation 4-1

P

prerequisites to installation 1-1

R

remote simulation manager (RSM) 3-4

changing listening port 3-9

command syntax 3-8

configuring 3-8

installing 3-6

network installation 3-7

registering product engines 3-9

troubleshooting installation 3-7

user roles 3-9

S

schedulers

Altair PBS Professional 3-20

configuring 3-20

custom 3-12

IBM Spectrum LSF 3-21

list of supported 3-12

Oracle Grid Engine (OGE) 3-23

Sun Grid Engine (SGE) 3-23

testing 3-20

Univa Grid Engine (UGE) 3-23

silent installation A-1

software

installing 2-1

silent installation A-1

uninstalling 6-1

system coupling with Maxwell C-1

T

TCP/IP 1-2

U

uninstalling the software 6-1