

Installing HSPICE on UNIX, Linux or Windows Platforms

This document describes how to install the HSPICE product.

Note:

The installation instructions in this document are the most up-to-date available at the time of production. However, changes might have occurred. For the latest installation information, see the product release notes or documentation.

This document provides instructions for UNIX, Linux and Windows platforms.

This document includes the following sections:

- [Media Availability and Supported Platforms](#)
- [Disk Space Requirements](#)
- [Installing the Software on UNIX or Linux Platforms](#)
- [Configuring HSPICE and AvanWaves for UNIX and Linux](#)
- [Setting Up the User Environment on UNIX and Linux](#)
- [Verifying the Installation](#)
- [Installing the Software on Windows Platforms](#)
- [HSPICE Integration to Cadence™ Virtuoso® Analog Design Environment](#)
- [Related Documentation and Customer Support](#)

Important:

You must set the `DISPLAY` environment variable before you install the software. Because the HSPICE postinstallation script is GUI based, the tool installation will fail if this variable is not set. (See [Configuring HSPICE and AvanWaves for UNIX and Linux on page 5.](#))

Media Availability and Supported Platforms

HSPICE, Discovery AMS Simulation Interface, AvanWaves, and MetaEncrypt are available on CD and by EST on all platforms, with the following exceptions:

- Discovery AMS Simulation Interface is not available on Windows platforms.
- AvanWaves is not available on the Linux platform.

For additional information about product availability, see *Quickstart Installation Guide*, available at http://www.synopsys.com/support/installation/new_install_guide072.html.

Obtain the appropriate binary executable files for your operating system. Table 1 lists the supported platforms for this release.

Table 1 *Platforms and Keywords*

Platform	Operating system	Synopsys platform keyword	Window environment
IA-32 (X86) ¹ (AMD Opteron) ¹	Red Hat Enterprise Linux v4, 5 ²	linux (32-bit mode) (64-bit mode) ³	GNOME
IBMRS/6000	AIX 5.1, 5.3	rs6000 (32-bit mode)	CDE
Sun SPARC	Solaris 9, 10 ²	sparcOS5 (32-bit mode) (64-bit mode) ²	CDE
EM64T	SUSE Enterprise Linux 9, 10	suse32 (32-bit mode) suse64 (64-bit mode) ²	KDE or GNOME
IA-32 (X86)	Windows 2000, XP	win_setup.exe (32-bit mode)	

1. IA-32 (X86) supports only Pentium 4 and later compatible processors. Older processors, such as 386, 486, 586, Pentium, Pentium 2, and Pentium 3, are not supported.

2. Binary-compatible operating system. Note, however, that binary compatibility is not guaranteed.

3. The 64-bit binary files for HSPICE and HSPICE RF are packaged with the 32-bit files. (To invoke the 64-bit executable files, use `hspice64` or `hspicerf64`.)

Note:

The 32-bit binary files for HSPICE on Windows platforms are binary compatible with the Windows X64 operating system.

Disk Space Requirements

Make sure you have enough disk space for the installation. For a full installation on all platforms, 708 MB is recommended. For a single platform installation, approximately 280 MB is recommended.

Installing the Software on UNIX or Linux Platforms

HSPICE uses the Synopsys Installer tool, which allows you to use a graphical user interface (GUI) or a text script. For information about downloading the Synopsys Installer and HSPICE, see *Quickstart Installation Guide* at http://www.synopsys.com/support/installation/new_install_guide072.html.

To install HSPICE by EST or from the CD, follow the procedures described in *Quickstart Installation Guide*.

Quickstart Installation Guide shows an example Synopsys media installation script for the synthesis tools. HSPICE is installed in a similar manner.

Important:

The HSPICE postinstallation script is GUI based and will terminate abnormally unless the `DISPLAY` environment variable is set correctly.

HSPICE is a stand-alone product and cannot be installed over an existing Synopsys product, including a prior version of HSPICE. You must create a new directory for HSPICE.

HSPICE installs with other tools, including AvanWaves.

Configuring HSPICE and AvanWaves for UNIX and Linux

The configuration program for HSPICE is automatically invoked after installation has finished. If the program does not start, you can invoke it manually by setting the `DISPLAY` variable to the correct value and invoking `config`. For example,

```
% setenv DISPLAY my_display:0.0  
% install_dir/hspice/bin/config
```

This command opens the Configure Products dialog box. Follow these steps:

Installing HSPICE on UNIX, Linux or Windows Platforms

Setting Up the User Environment on UNIX and Linux

1. To configure AvanWaves, select Configure AvanWaves. No option is required.

Note:

AvanWaves is not available on the Linux platform.

2. To configure HSPICE, select Configure HSPICE. The default settings are applied unless you set one of the following configuration options: “versions file,” “meta.cfg file...”, or “hspice.ini file...”.
3. After selecting your options, click the Configure button. The configuration log is shown in the middle of the dialog box.
4. Click Exit when the configuration has finished.

Setting Up the User Environment on UNIX and Linux

To set up the user environment, you must specify the location of the executable file and set the license environment variable.

Specifying the Executable File Location (2 Methods)

Preferred

Source the `cshrc.meta` file in the `bin` directory (or `kshrc.meta`).

Alternatively

To set up a new HSPICE tool user, add the directory containing the HSPICE executable file to the `PATH` environment variable.

- If you are using the C shell, add the following line to the `.cshrc` file:

```
set path=(install_dir/hspice/bin $path)
```
- If you are using the Bourne, Korn, or Bash shell, add the following line to the `.profile`, `.bashrc`, or `.kshrc` file:

```
PATH=install_dir/hspice/bin:$PATH
export PATH
```

Setting the SNPSLMD_LICENSE_FILE or LM_LICENSE_FILE Environment Variable

You must install the SCL software and define the SNPSLMD_LICENSE_FILE, LM_LICENSE_FILE, or another tool-specific variable before you can verify the HSPICE installation.

For information about downloading and installing SCL and on setting the license variable, see http://www.synopsys.com/support/installation/new_install_guide072.html.

Setting Up the Discovery AMS Simulation Interface Environment

To set up a new Discovery AMS Simulation Interface,

- If you are using the C shell, source the CSHRC_simif file located in the install directory.

```
% source install_dir/CSHRC_simif
```

The CSHRC_simif file sets the path for Discovery AMS Simulation Interface as follows:

```
setenv SNPS_SIMIF install_dir  
set path=(${SNPS_SIMIF}/bin $path)
```

where *install_dir* is the directory where the tool has been installed.

If you do not source the CSHRC_simif file, copy the preceding line and set the path from that file.

- If you are using the Bourne, Korn, or Bash shell, add the following line to the .profile, .kshrc, or .bashrc file:

```
SNPS_SIMIF=install_dir  
export SNPS_SIMIF  
  
PATH=${SNPS_SIMIF}/bin:$PATH  
export PATH
```

Verifying the Installation

After you set up the licensing and HSPICE software packages, do the following:

- For C shell users, enter

```
% source install_dir/hspice/bin/cshrc.meta
```
- For Bourne or Korn shell users, enter

```
$ install_dir/hspice/bin/kshrc.meta
```

If this step fails, or if the cshrc.meta file does not exist, the installation program failed. To find the problem, review the installation log file:

```
install_dir/hspice/bin/synopsys_config.log_pid
```

Verifying the HSPICE Installation

To verify the HSPICE installation,

1. Make sure you are in a directory where you have read/write privileges.

```
% cd $HOME
```

The `$installdir` environment variable refers to the HSPICE installation directory. This variable is created when you source the cshrc.meta file.

2. To run a demonstration simulation, enter

```
% hspice $installdir/hspice/demo/hspice/bench/demo.sp \  
-o demo.lis
```

3. To view the simulation output, enter

```
% vi demo.lis
```

If you are able to get a license, you will see a message about licenses near the top of the listing.

```
lic: Checkout hspice; Encryption code: xxx
```

If you get a `***** job concluded` statement near the bottom of the listing, the simulation was successful.

Verifying the AvanWaves Installation

To verify the AvanWaves installation,

1. Change to the installation directory.

```
% cd $installdir/hspice/bin/
```

The `$installdir` environment variable refers to the AvanWaves installation directory. This variable is created when you source the `cshrc.meta` file.

2. To start AvanWaves, enter

```
% awaves &
```

Installing the Software on Windows Platforms

The following sections describe how to install and set up the license server for HSPICE on a PC:

- [Installation Requirements](#)
- [Installing With Windows Explorer](#)
- [Installing With Windows DOS Shell](#)
- [Using Dongle-Based Licenses](#)
- [Setting Up the User Environment on Windows](#)
- [Setting Up the Environment for Verilog-A Simulation](#)
- [Running HSPICE on Windows](#)
- [Running AvanWaves on Windows](#)
- [Running Older Versions of HSPICE](#)
- [The hspice.ini File](#)

Installation Requirements

To run HSPICE, your system must have

- A Pentium or compatible processor or later
- At least 64 MB of memory (128 MB or more is recommended)

Installing HSPICE on UNIX, Linux or Windows Platforms

Installing the Software on Windows Platforms

- A parallel port for the security key (if you are using a dongle-based license)
- Windows 2000 or Windows XP operating system
- Approximately 180 MB of free disk space for a full installation of HSPICE, AvanWaves, and MetaEncrypt

Installing With Windows Explorer

To install HSPICE by using Windows Explorer,

1. Insert the HSPICE CD into the CD drive.
2. In Windows Explorer, double-click the CD drive icon.
3. Double-click the pc_hspice folder icon.
4. Double-click the Setup application icon.
5. To install the software, follow the screen prompts.

Installing With Windows DOS Shell

To install HSPICE by using the Windows DOS shell,

1. Insert the HSPICE CD into the CD drive.
2. At the DOS shell command prompt, enter

```
c:\> CD-ROM drive letter:\pchspice\hspice_version_platform_setup.exe
```

3. To install the software, follow the screen prompts.

Using Dongle-Based Licenses

The FLEXid drivers are no longer distributed with HSPICE. HSPICE now requires Synopsys Common Licensing (SCL) 10.9.1 or later to run dongle-based licenses.

After installing SCL 10.9.1 or later, follow these steps to verify that the Windows-based computer correctly identifies the dongle ID:

1. Make sure the dongle is securely attached to the parallel or USB port of your system. (The dongle must be directly attached to the computer and must be the first device attached; it cannot be piggybacked to another device.)
2. Open your Synopsys license file and locate the dongle ID. The ID is on the SERVER line or INCREMENT line of your license file. For example,
SERVER hostname1 FLEXID=8-5E700192562A 27000
3. Go to Start > Programs > Synopsys and click SCL`version` > Launch `lmttools.exe`, where `version` is 10.9.1 or later.
4. In the LMTOOLS window, click the System Settings tab. In the FLEXID box at the lower left, verify that the correct dongle ID appears. The dongle ID must match the dongle ID specified in the license file (see step 2).
5. If no dongle ID appears, choose Start > Settings > Control Panel > Add or Remove Programs and uninstall any Globetrotter, Macrovision, or Sentinel FLEXid drivers, and install the latest driver provided by Synopsys.
 - a. Choose Start > Programs and click SCL`version` > `flexid.exe`. (In some SCL versions, the executable file appears instead as `LaunchIDInstaller.exe`.)
 - b. Follow the onscreen instructions to install the correct FLEXID=7, FLEXID=8, or FLEXID=9 driver. Restart the system if instructed to do so.
 - c. Repeat steps 3 and 4. If this fails to resolve the problem, contact your local Synopsys Support Center.
6. Start SCL. (See http://www.synopsys.com/support/installation/new_install_guide072.html.)
7. Set `SNPSLMD_LICENSE_FILE`, `LM_LICENSE_FILE`, or another tool-specific license variable. See “[Setting Up the User Environment on Windows](#),” next.

Setting Up the User Environment on Windows

Set the `SNPSLMD_LICENSE_FILE`, `LM_LICENSE_FILE`, or another tool-specific license variable.

1. For Windows 2000, Windows XP, or later versions of Windows, choose Control Panel > System > Advanced > Environment Variables.
2. Under User variables to affect only the current user account or System Variables to affect all users, click New to create a new variable or Edit to edit an existing variable.
3. In the New (or Edit) User Variable dialog box, enter `SNPSLMD_LICENSE_FILE`, `LM_LICENSE_FILE`, or another tool-specific license variable in the “Variable name” box.
4. In the “Variable value” box, enter the `port@hostname` variable value. For example,
`27000@license_server`
where the *port* and *hostname* variables correspond to the TCP port and license server host name specified in the `SERVER` line of the Synopsys license file (27000 is the default TCP port).
5. To save your changes, click OK (Windows 2000 or later).

Note:

If you have multiple dongles connected to each other, the Synopsys dongle must be connected directly to the PC.

Setting Up the Environment for Verilog-A Simulation

Verilog-A is supported by both HSPICE and HSPICE RF on all platforms, including 64-bit and multithreading versions.

- To run Verilog-A on UNIX and Linux, source the `cshrc.meta` file located in `installdir/bin` or set the following environment variables:

```
setenv HSP_HOME installdir
set path=($HSP_HOME/bin \
          $HSP_HOME/$ARCH/verilog_utils/veriloga/include \
          $HSP_HOME/$ARCH/verilog_utils/tools/bin $path)
```

- To run Verilog-A on Windows platforms,
 - If you invoke HSPICE by using the GUI (HSPUI), Verilog-A is automatically invoked.
 - If you invoke HSPICE from the command prompt, set the following environment variables:

```
set HSP_HOME+%installdir_version%
set path=%installdir_version\bin; %path%
```

Running HSPICE on Windows

You can run HSPICE in any of the following ways:

- Double-click the HSPUI icon and then click the Simulate button.
- Double-click the HSPICE icon. You are prompted to enter names for the input netlist file and output list file. If you do not name an output file, all of the `.lis`, `.st0`, and `.tr0` files will be written in the directory in which you run HSPICE or `hspice.exe`.
- At the DOS prompt in Windows, enter

```
c:\> cd work_directory
c:\> %installdir%\BIN\HSPICE -i netlist -o listfile
```

Running AvanWaves on Windows

Start AvanWaves in any of the following ways:

- Double-click the HSPUI icon and then click the AvanWaves button.
- Double-click the AvanWaves icon.
- At the DOS prompt in Windows, enter

```
c:\> %installdir%\BIN\AWAVES
```

Important:

If you start AvanWaves by using a method other than selecting the HSPUI icon and then you open online Help, you must delete the Galaxy Help Server task from the Task Manager after you exit AvanWaves. Otherwise the next invocation of HSPICE can have a memory conflict with the AvanWaves Galaxy Help server.

Running Older Versions of HSPICE

You can use the HSPUI utilities to run different versions of HSPICE that are already installed on your computer. In the installation directory (%installdir%), the versions.txt file contains all the information about different versions of HSPICE.

An example of a versions.txt file follows:

```
<BOF>
  c:\synopsys\Hspiceversion\BIN\hspice.exe   HSPICE version
  c:\synopsys\Hspiceversion\BIN\hspice_mt.exe HSPICE \
      version
<EOF>
```

You can add different HSPICE executable file full paths to subsequent lines in the versions.txt file. The second column contains comments as a version reminder and is ignored by the HSPUI utility. After invoking HSPUI, you can select the HSPICE version in the Version list on the HSPUI window. HSPICE will run according to the path selected in the Version list.

The hspice.ini File

On Windows platforms, the hspice.ini file is the configuration file that contains HSPICE options such as the search path. For example,

```
.Option Post =1  
.Op  
.Option Post_version = 9601  
.....
```

If you have installed more than one version of HSPICE on the same PC, more than one hspice.ini file will exist, but only one takes effect when you start the tool.

The %installdir%\meta.cfg file points to the current hspice.ini file.

HSPICE Integration to Cadence™ Virtuoso® Analog Design Environment

The following sections describe the installation processes for the HSPICE integration into the Cadence environment.

Platforms Supported

sparcOS5 and linux

Software/Licenses Required

- HSPICE
- Cadence Virtuoso Analog Design Environment
- OASIS_Simulation_Interface license

1. Download the Installation Package

Unpack the tarfile:

```
tar xvf aa_integ_v<release>.tar
```

2. Create/Update the Cadence Virtuoso Analog Design Environment Hierarchy With the HSPICE Integration

Choose one of the two options presented below for installing the HSPICE Integration.

Installing HSPICE on UNIX, Linux or Windows Platforms

HSPICE Integration to Cadence™ Virtuoso® Analog Design Environment

Option 1: Create an Alternate Cadence Installation Hierarchy

If you prefer not to disturb your Cadence DFII installation directory, or if you don't have write permissions to it, you can create an alternate Cadence installation hierarchy. This new hierarchy will contain soft-links to the original Cadence installation.

1. Make sure that you are in the “interface” directory from the unpacked tarfile.
2. Type `./mknew_HSPICECDN_install <new_install_directory>`
3. Follow the instructions as you are prompted. The script will ask you to either confirm the location of the existing Cadence hierarchy if it finds one in your path or to specify the location of the existing Cadence hierarchy that you want to use for the soft-links.

Option 2: Install Directly into an Existing Cadence Hierarchy

If you prefer to install the HSPICE integration directly into an existing Cadence hierarchy, you may use this script.

1. Make sure that you are in the “interface” directory from the unpacked tarfile.
2. Type `./install_HSPICECDN`
3. Follow the instructions as you are prompted. The script will ask you to either confirm the location of the existing Cadence hierarchy if it finds one in your path, or to specify the location of the existing Cadence hierarchy that you want to update with the HSPICE Integration code.

3. Setup to Run HSPICE in Cadence™ Virtuoso® Analog Design Environment

You must perform the following steps in order to run `icms` with the HSPICE Integration:

1. Update your path to point to the latest version of HSPICE.

For example, if you unpacked the tarfile using the steps above while you were in the directory where you unpacked the latest HSPICE installation (see [Installing the Software on UNIX or Linux Platforms](#)), you will have a directory called something similar to: `/remote/HSPICE/A-2008.03/hspice/bin`. You need to add this directory to your search path:

```
set path=(/remote/HSPICE/A-2008.03/hspice/bin $path)
```

2. Update your path to point to the Cadence installation hierarchy that contains the HSPICE Integration.

For example, if you created an alternate installation hierarchy in /remote/HSPICE/A-2008.03/installDir, you will have the directories called /remote/HSPICE/A-2008.03/installDir/tools/bin and /remote/HSPICE/A-2008.03/installDir/tools/dfl/bin. You need to add these to your search path:

set path=(/remote/HSPICE/A-2008.03/installDir/tools/bin /remote/HSPICE/A-2008.03/installDir/tools/dfl/bin \$path)

3. Add the following to your effective .cdsinit file:

load(prependInstallPath("local/HSPICE/HSPICE.ini"))

The effective .cdsinit file that should be edited is the one which is loaded when you launch the Cadence tools. This may be in the local directory in which you launch the Cadence tools, the Cadence installation or other location. This command causes the HSPICE integration code to be loaded into your `icms` session when you start that tool.

Result: You are now finished with the installation and user setup.

Updating Libraries With HSPICE SimInfo

In order to update your libraries with HSPICE SimInfo, you must be set up to run `icms` with the HSPICE Integration. Follow the steps in the section above (Setup to Run HSPICE in Virtuoso Analog Design Environment) before proceeding with updating your libraries. Note that updating a library only needs to be done once, and multiple users can point to the same library through their `cds.lib` files.

See Chapter 2 in the *HSPICE Integration User Guide* on updating your libraries for more information on the actual library update process.

Adding HSPICE simInfo to your libraries

In order to run HSPICE within the Analog Design Environment, your simulation primitive libraries (such as `analogLib`) must have the necessary HSPICE simulator information. Use the following steps to update all libraries from which you are instantiating simulation primitives in your designs.

Installing HSPICE on UNIX, Linux or Windows Platforms

HSPICE Integration to Cadence™ Virtuoso® Analog Design Environment

Note:

This update utility updates recognized components with hspiceD simInfo. Most of the components in analogLib are recognized. If you have components that don't fit this category or don't have hspiceD simInfo, they will have to be updated manually in order to be netlisted.

Proceed as follows:

1. Start `icms` and in the CIW, select Tools->HSPICE Library Update Utility.

Note:

If you don't have this item in the Tools pulldown, your `.cdsinit` file may not be properly updated.

2. Select the name of the Library that you wish to update in the Library combo box.
3. You can instruct the Library Update Utility to make a copy of your library rather than altering your existing library. To do this, enable "Copy to New Library Before Updating" and specify the new library name. Your library will be copied to the new library name and updates will be applied to the new library.

Note:

Since the Cadence tool does not allow same-named libraries, you need to specify a name other than your source library name. If you do this, and want to continue to use your original library name in your schematic instantiations, you need to change the new library logical name in your `cds.lib` so that it matches the source library name.

For example, the Update Utility may add this to your `cds.lib`:

```
DEFINE analogLib_new /remote/v-XXX/analogLib_new
```

You must change this to:

```
DEFINE analogLib /remote/v-XXX/analogLib_new
```

4. Click **OK** or **Apply** to run the Update Utility.
5. Check the CDS.log file for unrecognized cells. If any were encountered, you will see a message such as:
Cell: TTL_a2d not updated as its type was not recognized
You may need to manually update these cells. Contact your Synopsys support person for information as to how this is done.
6. Once the Library Update Utility has completed its run, you may continue to use `icms` or you may exit.

Adding the HSPICE Simulator to the Tool Filter

If you need to access parameters that apply only to HSPICE, it is necessary to have the Tool Filter recognize the HSPICE integration. Although this is not necessary to run the HSPICE Integration or to run the Library Update Utility, it is highly likely that you will need to set or modify HSPICE-only parameters on components like the vsource. Once you perform this step in the installation hierarchy, all users pointing to the hierarchy will benefit from the Tool Filter.

1. Change directories to:
`<cadence_install_dir>/tools/dfl/etl/tools/auCore`

Note:

The `cadence_install_dir` is the one that you either created or updated in the steps above (**Option 1** or **Option 2**).

2. Edit the `.cdsenv` file and add HSPICE to the list of simulators. For example:
`auCore.toolFilter toolList string "spectre spectreS ams auCdl hspiceD HSPICE" nil`
3. In the schematic window, you can manually change the Tool Filter to show parameters for any specific simulator in the list above. Follow these steps to see the HSPICE parameters:
 - From the schematic window, select Tool Filter... from the Options menu.
 - Disable all selections for other simulators.
 - Enable the selection for HSPICE.
4. You can also set up your environment to automatically change the filter when you change the target simulator. To do this, add the following line to your `.cdsenv` file:
`auCore.toolFilter autoUpdate boolean t`

Optional: Making HSPICE Your Default Simulator Choice

If you want the environment to be invoked with HSPICE set as your default simulator, add the following line to your `.cdsinit` file:

`envSetVal("asimenv.startup" "simulator" 'string "HSPICE")`

Troubleshooting

If you don't see HSPICE listed in the Simulator choices pulldown, your .cdsinit file may not be properly updated

Related Documentation and Customer Support

The HSPICE and HSPICE RF documentation is available through Documentation on the Web. For information about accessing the documentation, see h.

Note:

To view and print HSPICE documentation in Portable Document Format (PDF), you must have Adobe Acrobat Reader installed on your machine. To acquire the latest version of the Adobe Reader software, free of charge, go to <http://www.adobe.com> and click the Get Adobe Reader button. Follow the instructions on the Web page.

For information about contacting Customer Support, see http://www.synopsys.com/support/installation/new_install_guide072.html

If you cannot solve a problem, use an editor to review the following files for system error messages:

- */tmp/pid/synopsys_install.log_pid* (which is generated during installation)
- */tmp/pid/synopsys_config.log_pid* (which is generated during configuration)