



ADS 2008
January 2008
Customization and Configuration

Advanced Design System 2008

© Agilent Technologies, Inc. 2000-2008

395 Page Mill Road, Palo Alto, CA 94304 U.S.A.

No part of this manual may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Microsoft®, Windows®, MS Windows®, Windows NT®, and MS-DOS® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Java™ is a U.S. trademark of Sun Microsystems, Inc. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC.

Errata The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

Warranty The material contained in this document is provided "as is", and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this manual and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

Technology Licenses The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license. Portions of this product include the SystemC software licensed under Open Source terms, which are available for download at <http://systemc.org/>. This software is redistributed by Agilent. The Contributors of the SystemC software provide this software "as is" and offer no warranty of any kind, express or implied, including without limitation warranties or conditions or title and non-infringement, and implied warranties or conditions merchantability and fitness for a particular purpose. Contributors shall not be liable for any damages of any kind including without limitation direct, indirect, special, incidental and consequential damages, such as lost profits. Any provisions that differ from this disclaimer are offered by Agilent only.

Restricted Rights Legend If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

Contents

- Customizing the ADS Environment
 - Customizing Environment Variables
 - Customizing Configuration Variables
 - - Configuration File Format
 - - Configuration Variable Description
 - - Configuration Variable Expansion
 - How a Configuration is Determined
 - Special Variables
 - - Color Definitions, hpeecolor.cfg
 - - Fill Pattern Configuration, hpeefill.cfg
 - Viewing Details of the Current Configuration
 - Typical Variables
 - - Variables in the File de.cfg, de_sim.cfg
 - - Variables in the File hpeesof.cfg
 - - Variables in the File dfilter.cfg
 - - Variables in the File eeapi.cfg
 - - Variables in the File hpads.cfg
 - - Variables in the File hpeesofbrowser.cfg
 - - Variables in the File hpeesofdds.cfg
 - - Variables in the File hpeesofhelp.cfg
 - - Variables in the File hpeesofinstrio.cfg
 - - Variables in the File momentum.cfg
 - - Variables in the File smg_ui.cfg
 - - Variables in the File hpeesofsim.cfg
 - - Variables in the File hpeesofsess.cfg
- Setting Design Environment Preferences
 - Specifying Design Entry and Display Preferences
 - - Setting Select Options
 - - Setting Grid/Snap Options
 - - Setting Placement Options
 - - Setting Pin/Tee Options
 - - Setting Entry/Edit Options
 - - Setting Component Text/Wire Label Options (in Advance)
 - - Setting Text Options (in Advance)
 - - Setting Display Options
 - - Setting Units/Scale Options
 - - Setting Tuning Options
 - - Saving and Reading Preference Files
 - Specifying Layer Definitions
 - - Setting Colors and Fill Patterns
 - - Setting Shape Display Characteristics Layer-by-Layer
 - - Setting Line Style Characteristics Layer-by-Layer
 - - Setting the Visibility of Items Layer-by-Layer
 - - Setting the Selection Status of Items Layer-by-Layer
 - - Setting Layer Characteristics Globally
 - - Miscellaneous Layer Editor Features

Advanced Design System 2008

- Saving and Reading Layer Files
 - Changing the Current Entry Layer
- Customizing Keyboard Shortcuts
- Configuring Toolbars
 - - Customizing an Existing Toolbar
 - - Creating a New Toolbar
- Creating a Custom Component Palette
 - - Customizing a Component
 - - Adding Custom Symbols
 - - Deleting Custom Component Palettes
 - - Turning On/Off the Coordinate Readout Display
- Setting Layout Options
- ◦ Defining Layers
 - - Setting Layer Characteristics Globally
 - - Miscellaneous Layer Editor Features
 - - The Default Layer
 - - Determining the Layer for an Item
 - - Changing Layer Priority
 - - Changing the Entry Layer
 - - Using the Layers Window
 - - Defining Port Connections (Layer Binding)
 - - Changing the Visibility and Protected Status of Items
 - - Using IGES and GDSII Numbers
 - - Using DXF Names
 - - Assigning Layers for Transmission Line Components
 - - Layer Files and Library Components
 - - Layer Files and Design Files
 - - Layer File Format
- Preferences for Layout
 - - Changing Select Options
 - - Changing Grid and Snap Settings
 - - Selecting Placement Options
 - - Changing Options for Pins/Tees
 - - Changing Entry/Edit Attributes
 - - Changing Component Text Attributes
 - - Changing Typed-in Text Attributes
 - - Changing Display Colors
 - - Setting DRC Memory Use and Performance
 - - Changing Miscellaneous Display Options
 - - Setting Units/Scale Factors
 - - Changing Layout Units & Resolution
- Toggling the Coordinate Readout Display
- Saving a Layout Setup
 - - Saving Layout Preferences
 - - Saving Layer Information
- Using an Existing Layout Setup
 - - Reading in an Existing Preferences File
 - - Reading in an Existing Layer File
 - - Reading a File from a Different Project Directory
- Pick and Place Report

- - Viewing Pick and Place Report Configuration Options
- Configuring the Pick and Place Report
- DE_PARTS_SET_PICK_AND_PLACE_OPTIONS
- Configuring the Parts List Report
- Reformatting the Reports
- Customization Examples
 - Creating a Custom Menu
 - Creating Custom Libraries
 - Assisted Customization
 - - Site-Wide Customization (Assisted)
 - - User Customization (Assisted)
 - Manual Customization
 - - Site-Wide Customization (Manual)
 - - User Customization (Manual)
 - - Project Customization (Manual)
 - Advanced Library Customization Techniques
 - - Site-Wide Customization (Advanced)
 - - User Customization (Advanced)
 - Modifying the List of Available Symbol Names
 - Adding Online Help for User-Defined Items
 - - Optional Document Template
 - Adding Custom Documentation for Online Access

Customizing the ADS Environment

The Advanced Design System suite of tools uses a number of configuration files. The files contain definitions for configuration variables and environment variables.

Customizing Environment Variables

Environment variables are usually initialized in a shell start-up file such as `.cshrc` (C-shell) or `.profile` (Bourne-shell) in your home directory. The following table describes the most common environment variables used by the Advanced Design System. On the PC, these environment variables get set automatically at installation.

Name	Default Value	Description
HPEESOF_DIR	/hpeesof	The location where the software has been installed (also called the root location).
AGILEESOFD_LICENSE_FILE	\$HPEESOF_DIR/licenses/license.lic	Specifies the location of the FLEXlm security license files.


EESOF_64BIT	Not set by default.	Specifies whether 64-bit systems use the default 64-bit simulators or 32-bit simulators. Set EESOF_64BIT=0 to force 64-bit systems to use the 32-bit simulators.
PRINTER	lpr	Specifies the printer used by the hpeedit program.
EESOF_BROWSER	Not set by default. Example setting: EESOF_BROWSER=/usr/bin/firefox	Specifies the browser to use for viewing online help.
CDS_INST_DIR	Not set by default. The Connection Manager Client does not use the value. The presence or absence of this variable determines which online help to display.	Specifies whether the Connection Manager Client uses ADS or RFDE online help. If CDS_INST_DIR is not set, ADS online help is used. If CDS_INST_DIR is set, RFDE online help is used.

The environment variable HPEESOF_DIR is used extensively throughout Advanced Design System. It does not need to be set if Advanced Design System has been installed in the directory /hpeesof.

Customizing Configuration Variables

The default values of the configuration variables should get you up and running, but they can be customized to better reflect your work environment. By default, the program searches for these configuration files in the following order, and uses the first one found:

- Your current project directory
Define configuration variables here that apply only to this project.
- Your personal directory = %HOME%/hpeesof/config
Define configuration variables here that apply to all your projects.

 **Note**
On the PC, %HOME% represents the path you specified as the Home Folder during installation (by default, C:\users\default)

- Your site's customized site directory \$HPEESOF_DIR/custom/config
Define configuration variables here that apply to all projects of all users on a site-wide basis. Configuration variables defined here will not be overwritten by installation of subsequent program patches or updates.
- The ADS installation directory \$HPEESOF_DIR/config
Define the default installed configuration variables here that apply to all projects of all users. Configuration variables should not be customized here.

Each of the individual Advanced Design System tools has its own configuration file. The supplied configuration files can

Advanced Design System 2008

be found in the directory \$HPEESOF_DIR/config (where \$HPEESOF_DIR represents the complete installation path). The filenames for each of the tools are shown in the following table.

Product/Tool	Filename
Design Environment	de_sim.cfg
Digital Filter	dfilter.cfg
Digital Synthesis	dsynthesis.cfg
Data Displays	hpeesofdds.cfg
Data Sets	hpeesofdss.cfg
Instrument Server	hpeesofinstrio.cfg
Momentum	momentum.cfg
Browser (Vendor and component parts)	hpeesofbrowser.cfg
Simulation and Synthesis Message Window	hpeesofsess.cfg
Layered A.P.I.	eeapi.cfg
Ptolemy models and MATLAB	hpads.cfg
Colors	hpeecolor.cfg
Fill patterns	hpeefill.cfg
GUI search paths	hpeesof.cfg
Online help system	hpeesofhelp.cfg
AEL search paths	hpeesofsim.cfg
Library translator	hplibtrans.cfg
Ptolemy	hptolemy.cfg
SMG search path	smg_ui.cfg
Spice to IFF	spctoiff.cfg
Spice translator	spice_xlator.cfg
LineCalc	linecalc.cfg

Configuration File Format

Configuration variables are defined using standard assignment syntax:

```
<variable>=<value>
```

Advanced Design System 2008

where variable is the configuration variable name and value is the string that is assigned to the variable. This value can be a directory, search path, numeric value, or other value defined by the program. For example, the location of the preferences could be set by:

```
PREFERENCES_DIR=${$HPEESOF_DIR}/de/defaults
```

where \$HPEESOF_DIR is an environment variable representing the complete installation path.

Note the following details about file format, illustrated in the accompanying example:

- Any line starting with # is a comment line and is ignored
- Blank lines are ignored
- Blank spaces to the left or right of the equal sign are ignored
- Empty assignments are acceptable (as shown by env_var_2)

Example

```
#
# This is a sample configuration file
#
env_var_1 = first_value
env_var_1a = value-1a
#
env_var_2 =
env_var_3 = 17
env_var_4 = $SYSTEM/%HOME/ex4
env_var_4a = {$HPEESOF_DIR}/de/defaults/{%env_var_1a}.ex4a
```

References to environment variables \$name and ADS configuration variables %name can appear in any order; they are resolved inside each ADS program. A reference to an environment variable or configuration variable name may be enclosed in braces { } when their use is not followed by a punctuation character. The braces are not included in an expanded value.

Configuration Variable Description

ADS configuration variables are identified by a case-insensitive string of 1 to 31 alphanumeric characters (an underscore is allowed). Variables that are longer than 31 characters are automatically truncated when the configuration file is read. When a variable is set from a configuration file, the line

```
<variable>=<value>
```

is parsed and variable is set to the value.

A configuration variable can contain a reference. A reference may be enclosed in braces and the text is preceded by either a \$ or % (see [ADS Configuration Variables](#)). For example,

```
PREFERENCES_DIR={$HPEESOF_DIR}/de/defaults
```

assigns the string \$HPEESOF_DIR/de/defaults to the configuration variable PREFERENCES_DIR, and \$HPEESOF_DIR is defined in a start-up file.

ADS Configuration Variables

Configuration variable	Description
\$name	The configuration variable is replaced by the text string assigned to name in the environment.
%name	The configuration variable is replaced by the text string assigned to name in the ADS configuration file. Certain ADS variables have a pre-defined meaning in a given program. All of the special ADS configuration variables that are recognized are given in the shipped {\$HPEESOF_DIR}/config/<app-name>.cfg file.

Most ADS configuration variables are assigned a single value. This text can represent a number, string, file, or path as in the following examples.

```
STATUS_DISP = 0
EESTATUS_LOG_FILE = Off
HPEESOF_KEY = $HPEESOF_DIR/licenses/hpeesof.key
PREFERENCES_DIR={$HPEESOF_DIR}/de/defaults
```

Note that when the file is outside of a project configuration, it is necessary to include the full path with the filename to ensure that the file is found.

Some ADS configuration variables accept a path assignment. A path is a list of one or more directories, where each directory is separated by a colon.

```
variable=directory:directory: ....
```

These variables can represent search paths or load paths, depending on the variable and the program. Ordering of the list is significant. When treated as a search path, the list is only scanned until the item needed is found. For loading, the entire list is sequentially examined with the last directory usually taking precedence.

The documentation for each specific variable indicates the type of value allowed. Some variables may have a limited type, such as a range of numbers or list of specific strings.

Configuration Variable Expansion

Configuration variable expansion refers to the process of replacing all references and variables that make up the configuration variable with their text equivalents until the complete value of the configuration variable is known. (There are no limits to the levels of referencing you can use.) Note that this expansion is done internally by the program; the file that contains the variable assignments is not modified.

For example, assume that the environment variable HPEESOF_DIR is set to /hpeesof/mysite and the following two lines exist in the file de_sim.cfg.

```
PROJECT3 = de
SYSTEM_CUSTOM_DE_SYMBOLS = {$HPEESOF_DIR}/custom/{%PROJECT3}/symbols
```

After expansion within hpeesof tool:

```
SYSTEM_CUSTOM_DE_SYMBOLS = /hpeesof/mysite/custom/de/symbols
```

As in the environment, you may redefine and add your own ADS variables in any ADS configuration files to help manage system- and user-specific configurations. However, a given ADS program only uses certain expanded ADS configuration variables, but the unexpanded value could contain any number of user-defined and/or ADS variables.

How a Configuration is Determined

Each configuration is distinguished by its name. Each named configuration contains all of the variables needed for describing that particular setup. At the topmost level is the configuration file named hpeesof.cfg, which is associated with all ADS programs. Other named configurations are associated with specific programs.

Each named configuration is determined by the location and content of any related configuration files. For example, the complete hpeesof configuration could consist of:

Advanced Design System 2008

- The file hpeesof.cfg, located in the install directory \$HPEESOF_DIR/config
- An edited copy of this same file, located in the customized site directory \$HPEESOF_DIR/custom/config
- An edited copy of this same file located under your home directory \$HOME/hpeesof/config
- An edited copy of this same file located in a particular project directory

As noted earlier, configurations in the user directory take precedence over the installation directory, and configurations in the project directory take precedence over the user directory. This is because the search order for configuration information is: project directory, user directory, site directory, and installation directory.

Note that variables in each successive file override any previously set value. You can even put two lines in the same configuration that assign values to the same variable, as in:

```
MY_VAR1 = entry_a
MY_VAR1 = entry_b
```

When expanded, MY_VAR1 will always have the value of entry_b.

The following table shows a list of named configurations used in the programs.

Named Configurations

Activity	Configuration
Design entry	hpeesof
Fill patterns	hpeefill
Online Help	hpeesofhelp
Colors	hpeecolor
Simulation	hpeesof de de_sim comms_ckt (Communications Design Suite only)
Status/Synthesis Messages	hpeesof eestatus
Data Displays	hpeesof eehelp hpeefill hpeecolor eegraph

Special Variables

There are two configuration files that have a different syntax than the others: hpeecolor and hpeefill. Their formats are presented here. Note that the naming convention and loading of system-wide, site-wide, home directory, and project-specific configurations still apply. Only the internal file format differs.

Color Definitions, hpeecolor.cfg

The design windows (Schematic and Layout), Data Display windows, and plotters use the colors defined in the hpeecolor configuration. By default, these colors are read from `{ $HPPEESOF_DIR }/config/hpeecolor.cfg`, but can be overridden at the site, user, or project level (any of which can have an hpeecolor file).

The hpeecolor file format differs from the variable=value syntax. Each line contains these major fields:

- RGB color values
- Color name
- Plotter pen number

The syntax for each line is as follows:

```
<Red #> <Green #> <Blue #> : <Color name> : <Pen number>
```

The following example illustrates the format of a typical color definition file:

R	G	B	X11 color name	Plotter pen number
0	0	0	: black	: 1
255	0	0	: red	: 2
0	255	0	: green	: 3
0	0	255	: blue	: 4
255	255	255	: white	: 5
128	64	0	:	: 6
			: salmon	: 7

Colors are described in the first and/or second fields of this file. You can enter three RGB color values (in the range 0-255) in the first field and/or you can enter the X color name from the RGB color database (found in `/usr/lib/X11/rgb.txt` for Motif or `/usr/openwin/lib/rgb.txt` for Sun's Open Windows) in the second field. If both are entered, the RGB values take precedence, and the color name is ignored.

The last field specifies which pen number to use for hard-copy output when performing HPGL plotting. An entry in this field is required and pen numbers may be recycled and repeated.

By default, the current implementation uses shared server colors from the RGB color database and uses no privately allocated color cells for X Window displays. This is done to simplify the color specification and to promote sharing of

colors. Sharing of color cells can only happen if two clients allocate read-only color cells with the same X color name or RGB values.

i Important
On UNIX, we strongly recommend that you only use the names given in the RGB color database instead of explicit RGB values; on the PC, only RGB values are supported.

Because of differences in screen hardware, the same RGB values may generate different colors on different hardware. By using names from the color database, you are more likely to get colors close to the ones requested across different hardware platforms.

It is also important to note that values corresponding to color names are not fixed. Therefore, they may not be exactly the same across all platforms or correspond to an exact RGB value.

i Note
Some platforms have limited color resources and some platforms may use non-shared colors, resulting in color exhaustion problems. When ADS programs run into these types of color problems, the solution is to reduce the number of requested colors until color conflicts stop occurring. As an example, truncating the list of colors in `hpeecolor.cfg` to 64 colors may be a good start.

An example `hpeecolor` configuration file is shown:

```
# =====
# Example EEsof Color Definition File:
# =====
#   R   G   B   :   Color Name   :   Plotter Pen
#   --- --- ---   :   -----   :   -----
#   0   0   0   :   black       :       1
# 255  0   0   :   red         :       2
#   0 255  0   :   green      :       3
#   0   0 255  :   blue       :       4
# 255 255  0   :   yellow     :       5
# 255  0 255  :   magenta    :       6
#   0 255 255  :   cyan       :       7
# 255 255 255  :   white      :       8
# Either RGB values or standard X Color Names, or both,
# may be specified for color entries. Note, if both RGB
# and Color Name values are specified, the RGB values take
# precedence.
# =====
: black : 1
: red : 2
: yellow : 3
: green : 4
: cyan : 5
: blue : 6
: magenta : 7
: gray : 8
: white : 1
```

If the ADS configuration file cannot be found or the hpeesof configuration does not contain an HPEESOF_COLOR variable (which determines the location of the ADS color definition file), a set of ten basic default server colors are automatically loaded as follows:

1 = black	5 = cyan	9 = white
2 = red	6 = blue	10 = medium blue
3 = yellow	7 = magenta	
4 = green	8 = gray	

Fill Pattern Configuration, hpeefill.cfg

Besides colors, the design windows (Schematic and Layout), Data Display windows, and plotters also use the fill patterns defined by the hpeefill configuration. By default, these patterns are read from `{HPEESOF_DIR}/config/hpeefill.cfg`, but can be overwritten at the site, user, or project level (any of which can have

an hpeefill file).

The hpeefill file format differs from the variable=value syntax. Each line contains two major fields.

- Path and filename of the fill pattern
- HPGL/2-pattern-specification

The syntax for each line is as follows:

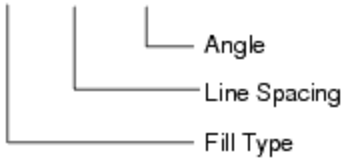
```
<Bitmap_file_location> : <Fill type> <Line spacing> <Angle>
```

where

- Fill type-is the pattern inside an object's border
- Line spacing-is the distance between the hatched lines in plotter units (where a plotter unit is typically 1/72-inch).
- Angle-is the angle of hatch lines in the fill pattern, expressed in degrees (for fill type 3 or 4 only).

The following example illustrates the format of a typical fill pattern definition file:

```
/fill/pat/path/pat1.pattern : 3 50 15
/flip/pat/path/pat2.pattern : 3 50 30
/flip/pat/path/pat3.pattern : 4 60 45
/flip/pat/path/pat4.pattern : 4 60 90
```



Fill patterns use standard X-bitmap files that can be easily created with the X bitmap utility. The first field specifies the path to the X-bitmap file, which is used for pattern fills, while the remaining fields are the HPGL/2 fill pattern specification.

Fill type description	Fill type number
Solid bidirectional	1
Solid unidirectional	2
Hatched	3
Cross hatched	4

Note
Solid bidirectional fill is faster than solid unidirectional fill, but is not supported by all plotters.

An hpeefill configuration file is shown in the following example:

```
# =====
# EEsof Fill Pattern Configuration File
# =====
# Screen Output      : Hardcopy Output
# X Bitmap Filename  : Type Spacing Angle
# -----
# path/pattern_file1 : 3      70      15
# path/pattern_file2 : 4      50      45
# path/pattern_file3 : 4      45      90
# =====
#
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat1.pattern : 1 0 0
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat1.pattern : 3 50 30
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat2.pattern : 3 50 45
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat3.pattern : 3 50 90
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat4.pattern : 4 50 15
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat5.pattern : 4 50 30
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat6.pattern : 4 50 45
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat7.pattern : 4 50 90
# {$HPEESOF_DIR}/lib/PDE/fill_patterns/pat8.pattern : 3 70 15
```

If the ADS configuration file cannot be found or the hpeesof configuration does not contain an HPEESOF_FILL variable (which determines the location of the ADS fill definition file), a single, unidirectional, solid fill pattern is automatically loaded.

Viewing Details of the Current Configuration

The Configuration Explorer enables you to search, view, save to file, and print the current settings of ADS configuration files. You can browse the configuration files in the following locations:

- Installation directory
- Site-wide customization directory
- User directory
- Current project

To launch the Configuration Explorer:

From the ADS Main window, choose Tools > Configuration Explorer.

Advanced Design System 2008

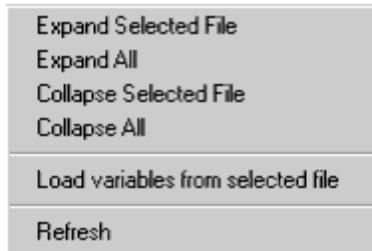
To view the properties of a given file:

1. Select the desired configuration directory from the drop-down list.
2. Click the desired filename. The file location, size, date, and permissions are displayed in the Status pane, as well as the location of any additional files by that same name.

```
File Name : C:/users/default/hpeesof/config/de_sim.cfg
File Size : 924
File Date : Jun 4 , 18:34
File Permissions : -rw-r--r--
Other de_sim.cfg files :
  C:\users\default\hpeesof\config\de_sim.cfg <<<
  C:\ADS2001\custom\config\de_sim.cfg
  C:\ADS2001\config\de_sim.cfg
```

To access the pop-up menu:

Position the pointer over any filename and click right.



- Use the uppermost group of commands to quickly expand and collapse the tree.
- Use Load variables from selected file to explicitly load into memory changes you have made to a configuration file since launching ADS.
- Use Refresh to view changes you have made to a configuration file since launching ADS.

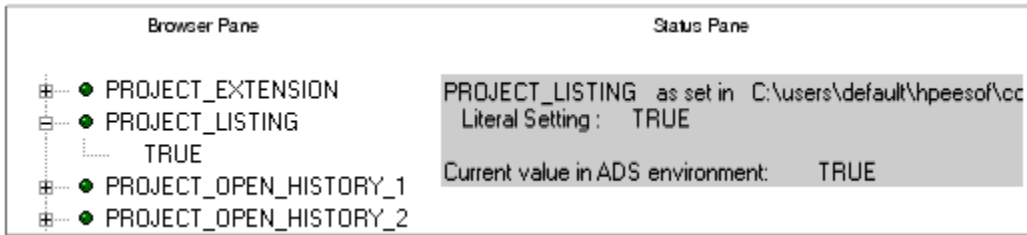
To view the list of variables in a given file:

Click once on the plus sign (expand) in front of the filename to display the list.

To view the current setting of a given variable:

- Click once on the variable name to display the value in the Status pane, or
- Click once on the plus sign in front of the variable to display its value in the browser pane

Advanced Design System 2008



To search for a specific variable name and/or value:

1. Select Variable in the Search pane.
2. Optionally, refine your search by selecting Variable Name and/or Variable Value.
3. Optionally, refine your search by selecting Exact Match.
4. Type the search string in the Search for field and click Search. The results are displayed in the Status pane.

To search for a specific filename:

1. Select File in the Search pane.
2. Optionally, refine your search by selecting Exact Match.
3. Type the search string in the Search for field and click Search. The results are displayed in the Status pane.

To retain the results of all listings and searches:

Select the Keep history option in the Status pane. Note that to retain the variable value listings, you must display the values in the Status pane, not the Browser pane. To clear the history click Clear.

To print the contents of the Status pane:

Click Print. The contents are sent to your default printer.


To save the contents of the Status panel to a file:

1. Click Save Status. A dialog box appears prompting you for a filename.
2. Change paths as desired, supply a filename, and click OK.

This information may be especially helpful when discussing configuration issues with Technical Support.

Typical Variables

The remaining variables use the variable=value syntax.


 **Note**
The Design Kit variables are documented in the Design Kit Installation and Setup manual.

Variables in the File de.cfg, de_sim.cfg

A number of resources related to the user interface and simulation can be customized in the file de_sim.cfg. This file can be customized for individual projects, all projects, etc., as described in, [Customizing Configuration Variables](#). You will find a default copy of this file, with a limited number of variables in it, in the following locations:

- In your /hpeesof/config directory under \$HOME (on UNIX) or c:\users\default (on PC, or whatever directory you chose instead during installation)
- Every project directory you create

You can add to this file (or modify, if they currently exist in the file) all variables described in the table below.

 **Note**
The variables described in the table are defined in either de.cfg or de_sim.cfg (under \$ HPEESOF_DIR/config), however when you customize any of these variables, it must be done in de_sim.cfg. To minimize typing and ensure accuracy of the variable names, copy the variables you want to customize from one or both of these files to the appropriate de_sim.cfg file and modify them there .

Variable, description, and example	Value(s)
FILE_FORMAT Specifies the format for writing design files. Use ASCII for portability and upgrades; binary is faster and uses less disk space. Example: FILE_FORMAT = ascii	ascii binary
LAYERS_PATH Specifies the layer file search path. Example: LAYERS_PATH = ./:{\$HPEESOF_DIR}/de/defaults	Path
PREFERENCES_PATH Specifies the preference file search path. Example: PREFERENCES_PATH = ./:{\$HPEESOF_DIR}/de/defaults	Path
LOCAL_AEL Specifies the project-specific AEL files to be automatically loaded each time when a project is opened. It should be set only	Filenames (without extension) or Path

<p>in the de_sim.cfg file located in the project directories. These AEL filenames may have absolute, relative, or no path specified, but since the files are generally located in the project directory, a relative path would be used beginning with a dot and slash (./). An AEL file that is already in the networks subdirectory will be automatically loaded. Any other AEL files that you would like included should be placed here. Example: LOCAL_AEL = my_file</p>	
<p>USER_AEL Specifies the user-specific AEL files to be automatically loaded once at startup. It should be set only in the de_sim.cfg file located in the user's personal customized directory \$HOME/hpeesof/config. If you specify a path, rather than filenames, it cannot be a relative path; it must be an absolute path, or you can use an environment variable. A typical use would include \$HOME to reference files in the user's home directory. Example: USER_AEL = \$HOME/hpeesof/MyLibrary/ael/</p>	<p>Filenames (optional extension) or Path</p>
<p>USER_DSN_PATH Specifies the directory (or directories) separated by a colon (:) where user-specific design files are located. It should be set only in the de_sim.cfg file located in the user's home directory. This variable cannot specify a relative path; it must be an absolute path, or you can use an environment variable. A typical use would include \$HOME to reference files in the user's home directory. Note: this variable should be used in conjunction with USER_AEL. Example: USER_DSN_PATH = \$HOME/hpeesof/MyLibrary/designs/</p>	<p>Path</p>
<p>SITE_AEL Specifies the site-specific AEL files to be automatically</p>	<p>Filenames (optional extension)</p>

Advanced Design System 2008

<p>loaded once at startup. It is typically set in the de_sim.cfg file located in \$HPEESOF_DIR/custom/config. If you specify a path, rather than filenames, it cannot be a relative path; it must be an absolute path, or you can use an environment variable. Example: SITE_AEL = {\$ADS_DESIGNS}/library1/ael/ Example: SITE_AEL = Q:\ads_designs\library1\ael\</p>	<p>or Path</p>
<p>SITE_DSN_PATH Specifies the directory (or directories) separated by a colon (:) where design files for site-wide use are located. It is typically set in the de_sim.cfg file located in \$HPEESOF_DIR/custom/config. This variable cannot specify a relative path; it must be an absolute path, or you can use an environment variable. Note: this variable should be used in conjunction with SITE_AEL. Example: SITE_DSN_PATH = {\$ADS_DESIGNS}/library1/designs/ Example: SITE_DSN_PATH = Q:\ads_designs\library1\designs\</p>	<p>Path</p>
<p>HPEESOF_LANGUAGE Specifies the language used by the PDE program. Example: HPEESOF_LANGUAGE = english</p>	<p>Language name</p>
<p>ENGLISH_MSG_EXT Specifies the language of the program messages. Example: english_MSG_EXT = eng</p>	<p>English = eng German = ger French = fra</p>
<p>ENGLISH_MSG_FONT Specifies the font supporting the selected local language. Example: ENGLISH_MSG_FONT = 9x15 bold</p>	<p>European = 9x15 Japanese = 8x16</p>
<p>PDE_MSG_FILE Specifies the full path to the PDE message directory file. Example: PDE_MSG_FILE = {\$HPEESOF_DIR}/msgs/english/PDE/PDE.msg</p>	<p>Path</p>
<p>NETLIST_FILE_NAME Specifies the netlist filename. Example: NETLIST_FILE_NAME = netlist.log</p>	<p>Filename</p>
<p>EGSLIST_FILE_NAME Specifies the filename for the EGS translator option. Example: EGSLIST_FILE_NAME = ./egslist.log</p>	<p>Filename</p>

Advanced Design System 2008

<p>EGS_OPTION_FILE: Specifies the path to the EGS translator option files. Example: EGS_OPTION_FILE = ./:{\$HPEESOF_DIR}/config</p>	Path
<p>GDSII_OPTION_FILE Specifies the path to the GDSII option files. Example: GDSII_OPTION_FILE = ./:{\$HPEESOF_DIR}/config</p>	Path
<p>MASK_OPTION_FILE Specifies the path to the mask translator option files. Example: MASK_OPTION_FILE = ./:{\$HPEESOF_DIR}/config</p>	Path
<p>IGES_OPTION_FILE Specifies the path to the IGES translator option files. Example: IGES_OPTION_FILE = ./:{\$HPEESOF_DIR}/config</p>	Path
<p>SPICE_OPTION_FILE Specifies the search path for the Spice files. Example: SPICE_OPTION_FILE = ./:{\$HPEESOF_DIR}/config</p>	Path
<p>IFF_OPTION_FILE Specifies the search path for the IFF option files. Example: IFF_OPTION_FILE = ./:{\$HPEESOF_DIR}/config</p>	Path
<p>DIALOG_TIME_OUT Specifies the time before the copyright or information dialog box is automatically dismissed. Example: DIALOG_TIME_OUT = 10000</p>	Time in milliseconds
<p>BALLOON_HELP_TIMEOUT Time taken for balloon help to appear when icon is selected. Example: BALLOON_HELP_TIMEOUT = 600</p>	Time in milliseconds
<p>BALLOON_HELP_DISMISSAL_TIMEOUT Time taken for balloon help to disappear when icon is selected. Example: BALLOON_HELP_DISMISSAL_TIMEOUT = 2500</p>	Time in milliseconds
<p>BALLOON_HELP Sets whether balloon help exists or not. Example: BALLOON_HELP = TRUE</p>	TRUE or FALSE
<p>DOUBLE_CLICKS_TIME_OUT Specifies the time interval in which a double-click is registered. Example: DOUBLE_CLICKS_TIME_OUT = 200</p>	Time in milliseconds

Advanced Design System 2008

<p>QUERY_PROCESS_STATUS Specifies how often PDE should query the child process status. Example: QUERY_PROCESS_STATUS =1</p>	Time in seconds
<p>MAX_PROCESS_READY_TIMEOUT Defines when PDE should quit waiting for the child process to be ready after it's spawned Example: MAX_PROCESS_READY_TIMEOUT = 300</p>	Time in seconds
<p>BOM_LIST Specifies the list of report types presented in the Bill of Materials dialog box. Example: BOM_LIST = Agilent EEsof (Agilent EEsof BOM format)/de_bom</p>	Report types
<p>PARTS_LIST Specifies the list of report types presented in the Parts List dialog box. Example: PARTS_LIST = Agilent EEsof (Agilent EEsof PL format)/de_parts:Agilent EEsof (netlist format)/de_net</p>	Report types
<p>BOM_ITEM Specifies the spacing allocated for item name field in the Bill of Materials dialog box. Example: BOM_ITEM = 30</p>	Integer
<p>BOM_QTY Specifies the spacing allocated for quantity field in the Bill of Materials dialog box. Example: BOM_QTY = 7</p>	Integer
<p>BOM_DESC Specifies the spacing allocated for description field in the Bill of Materials dialog box. Example: BOM_DESC = 50</p>	Integer
<p>TEXT_EDITOR_PROGRAM_NAME Text editor Example: TEXT_EDITOR_PROGRAM_NAME = write.exe</p>	Editor name
<p>INVOKE_DDS_ON_BOOTUP Program Preference. Example: INVOKE_DDS_ON_BOOTUP = FALSE</p>	TRUE or FALSE
<p>INVOKE_LIB_SERVER_ON_BOOTUP Program Preference.</p>	TRUE or FALSE

Advanced Design System 2008

Example: INVOKE_LIB_SERVER_ON_BOOTUP = FALSE	
<p>FILE_OPEN_HISTORY_COUNT Maximum number of files to be shown at the end of the File Menu in the Schematic and Layout windows. Example: FILE_OPEN_HISTORY_COUNT = 4</p>	Integer
<p>PROJECT_OPEN_HISTORY_COUNT Maximum number of projects to be listed at the end of the File Menu in the Schematic and Layout windows. Example: PROJECT_OPEN_HISTORY_COUNT = 4</p>	Integer
<p>DESIGN_LIST_COUNT Maximum number of designs to appear at the end of the Windows Menu in the Schematic and Layout windows. Example: DESIGN_LIST_COUNT = 9</p>	Integer
<p>SCHEMATIC_COMPONENT_PALETTE_LIST_COUNT Maximum number of palettes displayed in the component palette drop-down list on the Schematic window toolbar. Example: SCHEMATIC_COMPONENT_PALETTE_LIST_COUNT = 16</p>	Integer
<p>LAYOUT_COMPONENT_PALETTE_LIST_COUNT Maximum number of palettes displayed in the component palette drop-down list on the Layout window toolbar. Example: LAYOUT_COMPONENT_PALETTE_LIST_COUNT = 16</p>	Integer
<p>SCHEMATIC_COMPONENT_PALETTE_LIST_WIDTH Width of the component palette drop-down list on the Schematic window toolbar. Example: SCHEMATIC_COMPONENT_PALETTE_LIST_WIDTH = 16</p>	Integer
<p>LAYOUT_COMPONENT_PALETTE_LIST_WIDTH Width of the component palette drop-down list on the Layout window toolbar. Example: LAYOUT_COMPONENT_PALETTE_LIST_WIDTH = 16</p>	Integer
<p>SCHEMATIC_COMPONENT_HISTORY_LIST_COUNT Maximum number of components displayed in the component history drop-down list on the Schematic window toolbar. Example: SCHEMATIC_COMPONENT_HISTORY_LIST_COUNT = 10</p>	Integer

Advanced Design System 2008

<p>LAYOUT_COMPONENT_HISTORY_LIST_COUNT Maximum number of components displayed in the component history drop-down list on the Layout window toolbar. Example: LAYOUT_COMPONENT_HISTORY_LIST_COUNT = 10</p>	Integer
<p>SCHEMATIC_COMPONENT_HISTORY_LIST_WIDTH Width of the component history drop-down list on the Schematic window toolbar. Example: SCHEMATIC_COMPONENT_HISTORY_LIST_WIDTH = 7</p>	Integer
<p>LAYOUT_COMPONENT_HISTORY_LIST_WIDTH Width of the component history drop-down list on the Layout window toolbar. Example: LAYOUT_COMPONENT_HISTORY_LIST_WIDTH = 7</p>	Integer
<p>LAYER_LIST_COUNT Maximum number of layers to be shown in the layer list drop-down list on the Layout window toolbar. Example: LAYER_LIST_COUNT = 10</p>	Integer
<p>LAYER_LIST_WIDTH Width of the drop-down layer list on the Layout window toolbar. Example: LAYER_WIDTH = 14</p>	Integer
<p>SIMULATION_HOST_LIST Remote simulation host names Example: SIMULATION_HOST_LIST = machine.company.com</p>	Host names
<p>MAIN_WINDOW_X_LOC, MAIN_WINDOW_Y_LOC The default location of the Main window. The upper left corner of the window is positioned at the specified X,Y screen coordinates, where 0,0 is the upper left corner. Example: MAIN_WINDOW_X_LOC = 0 Example: MAIN_WINDOW_Y_LOC = 0</p>	Integer
<p>MAIN_WINDOW_WIDTH, MAIN_WINDOW_HEIGHT The default size of the Main window in pixels (window can still be re-sized manually). Example: MAIN_WINDOW_WIDTH = 365 Example: MAIN_WINDOW_HEIGHT = 340</p>	Integer Min. Width = 340 Min. Height = 300
<p>SCHEMATIC_WINDOW_X_LOC, SCHEMATIC_WINDOW_Y_LOC</p>	Integer

Advanced Design System 2008

<p>The default location of the Schematic window. The upper left corner of the window is positioned at the specified X,Y screen coordinates, where 0,0 is the upper left corner. Example: SCHEMATIC_WINDOW_X_LOC = 260 Example: SCHEMATIC_WINDOW_Y_LOC = 0</p>	
<p>SCHEMATIC_WINDOW_WIDTH, SCHEMATIC_WINDOW_HEIGHT The default size of the Schematic window in pixels (window can still be re-sized manually). Example: SCHEMATIC_WINDOW_WIDTH = 780 Example: SCHEMATIC_WINDOW_HEIGHT = 530</p>	<p>Integer Min. Width = 760 Min. Height = 500</p>
<p>LAYOUT_WINDOW_X_LOC, LAYOUT_WINDOW_Y_LOC The default location of the Layout window. The upper left corner of the window is positioned at the specified X,Y screen coordinates, where 0,0 is the upper left corner. Example: LAYOUT_WINDOW_X_LOC = 30 Example: LAYOUT_WINDOW_Y_LOC = 30</p>	<p>Integer</p>
<p>LAYOUT_WINDOW_WIDTH, LAYOUT_WINDOW_HEIGHT The default size of the Layout window in pixels (window can still be re-sized manually). Example: LAYOUT_WINDOW_WIDTH = 1012 Example: LAYOUT_WINDOW_HEIGHT = 500</p>	<p>Integer Min. Width = 760 Min. Height = 500</p>
<p>ELW_WINDOW_OPEN_WITH_SCHEMATIC When True, the Layers window will automatically be displayed when a schematic window is opened. Example: ELW_WINDOW_OPEN_WITH_SCHEMATIC = FALSE</p>	<p>TRUE or FALSE</p>
<p>ELW_WINDOW_OPEN_WITH_LAYOUT When True, the Layers window will automatically be displayed when a layout window is opened. Example: ELW_WINDOW_OPEN_WITH_LAYOUT = TRUE</p>	<p>TRUE or FALSE</p>
<p>ELW_WINDOW_AUTO_REDRAW_DESIGN When True, the current design representation's schematic or layout window will be redrawn whenever the Layers window changes the design representation's layers information. Example: ELW_WINDOW_AUTO_REDRAW_DESIGN = TRUE</p>	<p>TRUE or FALSE</p>

Advanced Design System 2008

<p>ELW_WINDOW_INITIAL_X_LOC, ELW_WINDOW_INITIAL_Y_LOC</p> <p>The default initial location of the Layers window. The upper left corner of the window is positioned at the specified X, Y screen coordinates, where 0,0 is the upper left corner. Example: ELW_WINDOW_INITIAL_X_LOC = 0 Example: ELW_WINDOW_INITIAL_Y_LOC = 100</p>	Integer
<p>ELW_WINDOW_INITIAL_WIDTH, ELW_WINDOW_INITIAL_HEIGHT</p> <p>The default initial size of the Layers window in pixels (window can still be resized manually). Example: ELW_WINDOW_INITIAL_WIDTH = 200 Example: ELW_WINDOW_INITIAL_HEIGHT = 450</p>	Integer
<p>SAVE_PROJECT_STATE</p> <p>Sets a default to be used for any project for which no save_project_state.ael file exists. This file is created/modified when you change projects or exit the program and the Save Project State on Exit option is on. When on (TRUE), the status of every currently open Schematic and Layout window will be saved and subsequently restored, the next time you open that project. Note that only windows with named designs are saved; untitled windows are not saved. Example: SAVE_PROJECT_STATE = TRUE</p>	TRUE, FALSE
<p>COMMAND_DIALOG_WIDTH</p> <p>Sets the width, in pixels, of the Command Line dialog box (Options > Command Line in the Main window). The default is 400; a wider setting may be helpful for viewing lengthy AEL functions. Example: COMMAND_DIALOG_WIDTH = 400</p>	Integer
<p>DATA_FILES</p> <p>Specifies the search path for data files. Example: DATA_FILES = ./data:{\$HOME}/hpeesof/my_datafiles</p>	Path
<p>OPEN_PROJECT_AUTO_SCHEMATIC</p>	TRUE, FALSE

Advanced Design System 2008

<p>Sets a default for all projects. When TRUE, an empty Schematic window is automatically displayed when you create a project. Example: OPEN_PROJECT_AUTO_SCHEMATIC = TRUE</p>	
<p>OPEN_PROJECT_AUTO_LAYOUT Sets a default for all projects. When TRUE, an empty Layout window is automatically displayed when you create a project. Example: OPEN_PROJECT_AUTO_LAYOUT = FALSE</p>	TRUE, FALSE
<p>SCHEMATIC_ANALOG_DEFAULT_PALETTE Sets the specified palette as the default palette for Analog/RF designs in the Schematic window. Example: SCHEMATIC_ANALOG_DEFAULT_PALETTE = Devices-MOS</p>	Standard or custom palette name, exactly as it appears in program.
<p>LAYOUT_ANALOG_DEFAULT_PALETTE Sets the specified palette as the default palette for Analog/RF designs in the Layout window. Example: LAYOUT_ANALOG_DEFAULT_PALETTE = TLines-Stripline</p>	Standard or custom palette name, exactly as it appears in program.
<p>SCHEMATIC_DSP_DEFAULT_PALETTE Sets the specified palette as the default palette for DSP designs in the Schematic window. Example: SCHEMATIC_DSP_DEFAULT_PALETTE = Timed Linear</p>	Standard or custom palette name, exactly as it appears in program.
<p>LAYOUT_DSP_DEFAULT_PALETTE Sets the specified palette as the default palette for DSP designs in the Layout window. Example: LAYOUT_DSP_DEFAULT_PALETTE = Block Text Fonts</p>	Standard or custom palette name, exactly as it appears in program.
<p>USER_MENU_FUNCTION_LIST Controls which functions are called every time a new window is opened. Example: USER_MENU_FUNCTION_LIST = app_add_user_menus</p>	Function names
<p>OPEN_DDS_AFTER_SIM Establishes a default state for whether or not to automatically</p>	TRUE or FALSE

<p>open a Data Display window upon completion of a simulation. This setting can be overridden for any given simulation using the Simulation Setup dialog box. Note: If this variable is not defined, or is defined but set to something other than TRUE or FALSE, the initial state for Analog/RF designs is TRUE and for DSP designs is FALSE. Example: OPEN_DDS_AFTER_SIM = TRUE</p>	
<p>CONFIG_EXPLORER_VARS Controls which variables are listed when you launch the Configuration Explorer. This list is delimited by semicolons. Each variable name is preceded by one of the following designations, as well as an underscore character: E_ For Environment variables Example: CONFIG_EXPLORER_VARS = E_HPEESOF_DIR C_ For Configuration variables Example: CONFIG_EXPLORER_VARS = C_SITE_AEL S_ For System variables (PC only) Example: CONFIG_EXPLORER_VARS = S_TEMP</p>	<p>E_<VariableName>, C_<VariableName>, S_<VariableName></p>
<p>CONFIG_EXPLORER_CMP_VARS Controls whether or not the Configuration Explorer evaluates the value of every variable and compares it against the current value in the environment. If the setting of a given variable is the same as the current value in the environment, it is preceded by a green dot; if the values are different, it is preceded by a yellow dot. (Note: Setting this variable to yes is helpful for troubleshooting configuration problems but takes your system considerable time to perform the evaluations.) The default value is no. Example: CONFIG_EXPLORER_CMP_VARS = yes</p>	<p>Yes or No</p>
<p>MAKELIB_USER_AEL_PATH Complete path for .ael files (associated with .dsn files) added to a user library using the custom library tool. Multiple libraries can be defined by specifying a semicolon-delimited list. Example: MAKELIB_USER_AEL_PATH = {\$HOME}/hpeesof/circuit/ael</p>	<p>Path</p>

Advanced Design System 2008

<p>MAKELIB_USER_DSN_PATH Complete path for .dsn files added to a user library using the custom library tool. Multiple libraries can be defined by specifying a semicolon-delimited list. Example: MAKELIB_USER_DSN_PATH = {\$HOME}/hpeesof/circuit/networks</p>	<p>Path</p>
<p>MAKELIB_USER_DATA_FILES Complete path for data files referenced by designs added to a user library using the custom library tool. Example: MAKELIB_USER_DATA_FILES = {\$HOME}/hpeesof/datafiles</p>	<p>Path</p>
<p>MAKELIB_USER_LIBRARY_NAMES The library name(s) that are used if no Library Name was specified in the Design Parameters dialog box. Multiple libraries can be defined by specifying a semicolon-delimited list. The members of this list must have a one-to-one correspondence with the lists set for the three aforementioned variables. Example: MAKELIB_USER_LIBRARY_NAMES = User Library</p>	<p>Default or custom library name</p>
<p>MAKELIB_SITE_AEL_PATH Complete path for .ael files (associated with .dsn files) added to a site library using the custom library tool. Multiple libraries can be defined by specifying a semicolon-delimited list. Example: MAKELIB_SITE_AEL_PATH = {\$HPEESOF_DIR}/custom/circuit/ael</p>	<p>Path</p>
<p>MAKELIB_SITE_DSN_PATH Complete path for .dsn files added to a site library using the custom library tool. Multiple libraries can be defined by specifying a semicolon-delimited list. Example: MAKELIB_SITE_DSN_PATH = {\$HPEESOF_DIR}/custom/circuit/networks</p>	<p>Path</p>
<p>MAKELIB_SITE_DATA_FILES Complete path for data files referenced by designs added to a</p>	<p>Path</p>

<p>site library using the custom library tool. Example: MAKELIB_SITE_DATA_FILES = {\$HPEESOF_DIR}/custom/datafiles</p>	
<p>MAKELIB_SITE_LIBRARY_NAMES The library name(s) that are used if no Library Name was specified in the Design Parameters dialog box. Multiple libraries can be defined by specifying a semicolon-delimited list. The members of this list must have a one-to-one correspondence with the lists set for the three aforementioned variables. Example: MAKELIB_SITE_LIBRARY_NAMES = Site Library</p>	<p>Default or custom library name</p>

Variables in the File hpeesof.cfg

All programs that are part of the ADS suite read the hpeesof configuration file.

Variable, description, and example	Value(s)
<p>HPEESOF_LM_LICENSE_FILE Complete path to FLEXlm license.lic file. Example: HPEESOF_LM_LICENSE_FILE = {\$HPEESOF_DIR}/license/license.lic</p>	<p>Filename</p>
<p>HPEESOF_COLOR Complete path to system-wide color configuration file. Example: HPEESOF_COLOR = {\$HPEESOF_DIR}/config/hpeecolor.cfg</p>	<p>Filename</p>
<p>HPEESOF_FILL Complete path to system-wide fill pattern configuration file. Example: HPEESOF_FILL = {\$HPEESOF_DIR}/config/hpeefill.cfg</p>	<p>Filename</p>
<p>HPEESOF_FONT_FILE Directory where TrueType raster fonts are located. Example: HPEESOF_FONT_FILE = {\$HPEESOF_DIR}/config</p>	<p>Directory</p>
<p>HPEESOF_OCCIDENTAL_FILE Complete path to European Hershey vector fonts. Example:</p>	<p>Filename</p>

HPEESOF_OCCIDENTAL_FILE = { \$HPEESOF_DIR }/config/hersh.oc	
HPEESOF_ORIENTAL_FILE Complete path to Asian Hershey vector fonts. Example: HPEESOF_ORIENTAL_FILE = { \$HPEESOF_DIR }/config/hersh.or	Filename
HPEESOF_LANGUAGE Specifies the default language. Example: HPEESOF_LANGUAGE = english	Language name
AGILEESOFD_LICPREF Controls bundle selection for all systems on a network with access to the license server. Example: AGILEESOFD_LICPREF = bundle_a	License bundle name
AGILEESOFD_LICPREF_<hostname> Controls bundle selection specifically for the system identified by <hostname>. Example: AGILEESOFD_LICPREF_alpha = bundle_b	License bundle name

Variables in the File dfilter.cfg

This configuration file is used by the Digital Filter tool.

Variable, description, and example	Value(s)
DF_AEL_PATH The AEL path for Digital Filter designs. Example: DF_AEL_PATH = ./:\$HOME/hpeesof/dfilter/ael: \$HPEESOF_DIR/custom/dfilter/ael:\$HPEESOF_DIR/dfilter/ael	Path
DFUI_DATA_FILE Data file, only for Digital Filter UI Example: DFUI_DATA_FILE = \$HPEESOF_DIR/dfilter/ael/firdemoui.ael	Filename
DFUI_SPEC_HEADING_COLOR Foreground color for spec heading data, only for Digital Filter UI Example: DFUI_SPEC_HEADING_COLOR = BLUE	Color name
DFUI_SPEC_EDITABLE_COLOR Foreground color for spec editable data, only for Digital Filter UI Example: DFUI_SPEC_EDITABLE_COLOR = BLACK	Color name

Advanced Design System 2008

DFUI_SPEC_NOEDITABLE_COLOR Foreground color for spec non-editable data, only for Digital Filter UI Example: DFUI_SPEC_NOEDITABLE_COLOR = GRAY	Color name
DFUI_SPEC_CELL_WIDTH Column width for spec data, only for Digital Filter UI Example: DFUI_SPEC_CELL_WIDTH = 7	Integer
DF_DDS_IDEAL_TRACE_COLOR Trace color index to be used in hpeesof data display server Example: DF_DDS_IDEAL_TRACE_COLOR = 1	Integer
DF_DDS_SCALED_TRACE_COLOR Trace color index to be used in hpeesof data display server Example: DF_DDS_SCALED_TRACE_COLOR = 3	Integer
DF_DDS_SPEC_TRACE_COLOR Trace color index to be used in hpeesof data display server Example: DF_DDS_SPEC_TRACE_COLOR = 4	Integer

Variables in the File eeapi.cfg

This configuration file is used by the layered Application Programming Interface (API).

Variable, description, and example	Value(s)
API_BALLOON_HELP_TIMEOUT Time taken for balloon help to appear when icon is selected. Example: API_BALLOON_HELP_TIMEOUT = 600	Time in milliseconds
API_BALLOON_HELP Sets whether balloon help exists or not. Example: API_BALLOON_HELP = TRUE	TRUE or FALSE
API_BALLOON_HELP_DISMISSAL_TIMEOUT Time taken for balloon help to disappear when icon is selected. Example: API_BALLOON_HELP_DISMISSAL_TIMEOUT = 6000	Time in milliseconds
API_DOUBLE_CLICKS_TIME_OUT The time between two clicks required for them to be recognized as a unit, or double-click. Example: API_DOUBLE_CLICKS_TIME_OUT = 400	Time in milliseconds

Advanced Design System 2008

<p>API_NO_TEAR_OFF_MENU Turn on/off Motif's tear-off menu Example: API_NO_TEAR_OFF_MENU = 0</p>	<p>0=allow tear-off menu 1= do not allow tear-off menu</p>
---	--

Variables in the File hpads.cfg

This configuration file is used by PTOLEMY for locating user-compiled models and the location of MATLAB software.

Variable, description, and example	Value(s)
<p>ADSPTOLEMY_MOD_PATH Controls where ADS Ptolemy will look for models. You must reference the ADSPTOLEMY_MODEL_PATH variable here so that user-compiled models are found. If ADS is started with the -b option, then \$HPEESOF_DIR/adsptolemy/beta will be appended to load beta models. Example: ADSPTOLEMY_MOD_PATH = \$HOME/hpeesof/adsptolemy: \$ADSPTOLEMY_MODEL_PATH:\$HPEESOF_DIR/adsptolemy</p>	<p>Path</p>
<p>MATLAB The MATLAB variable should point to the root of your Matlab 5 installation (directories bin and extern should be there). Example: MATLAB = /usr/local/matlab</p>	<p>Directory</p>
<p>MATLABCMD If the command to invoke Matlab is not matlab, you'll need to set the MATLABCMD variable. See the documentation for engOpen in the Matlab Application Programming Interface Guide for more details on what you can do here. This variable is ignored on Windows. Example: MATLABCMD = "matlab -c licensefile"</p>	<p>Command</p>

Variables in the File hpeesofbrowser.cfg

This configuration file is used by the Browse and Search dialog box.

Variable, description, and example	Value(s)
<p>HPEESOF_BROWSER_WINDOW_X_LOCATION</p>	<p>Integer</p>

Advanced Design System 2008

<p>The default X coordinate of the Component Library window relative to the upper left corner of the screen. Example: HPEESOF_BROWSER_WINDOW_X_LOCATION = 100</p>	
<p>HPEESOF_BROWSER_WINDOW_Y_LOCATION The default Y coordinate of the Component Library window relative to the upper left corner of the screen. Example: HPEESOF_BROWSER_WINDOW_Y_LOCATION = 100</p>	Integer
<p>HPEESOF_BROWSER_WINDOW_WIDTH The default width of the Component Library window. Example: HPEESOF_BROWSER_WINDOW_WIDTH = 600</p>	Integer
<p>HPEESOF_BROWSER_WINDOW_HEIGHT The default height of the Component Library window. Example: HPEESOF_BROWSER_WINDOW_HEIGHT = 400</p>	Integer
<p>HPEESOF_BROWSER_TIME_OUT The default time-out for warning/information messages. Example: HPEESOF_BROWSER_TIME_OUT = 10000</p>	Time in milliseconds
<p>HPEESOF_BROWSER_AEL_PATH Path for the browser AEL files. Example: HPEESOF_BROWSER_AEL_PATH = \$HOME/hpeesof/hpeesofbrowser/ael:\$HPEESOF_DIR/hpeesofbrowser/ael</p>	Path
<p>HPEESOF_BROWSER_AEL Top level AEL file Example: HPEESOF_BROWSER_AEL = lbb_main</p>	Filename
<p>HPEESOF_BROWSER_PATH Path for the control, records and Device Libraries Binary File. Path can only have System variables. Example: HPEESOF_BROWSER_PATH = \$HOME/hpeesof/hpeesofbrowser/records;\$COMPL_DIR/ComponentLibs/records</p>	Path
<p>HPTOLEMY_BROWSER_PATH ADS Ptolemy related control, record and IDF files. All the control files and the IDF files in these directories are read to create the ADS Ptolemy related libraries and sub-libraries Example: HPTOLEMY_BROWSER_PATH = \$HPTOLEMY_MODEL_AEL;\$HPEESOF_DIR/CustomEncoded/records</p>	Path
<p>HPEESOF_BROWSER_LIBRARIES_FIELD_WIDTH The default width of the component library tree</p>	Integer

Advanced Design System 2008

structure. Example: HPEESOF_BROWSER_LIBRARIES_FIELD_WIDTH = 30	
HPEESOF_BROWSER_PARTS_DISPLAY_WIDTH The default width of the Component column. (This column is always visible.) Example: HPEESOF_BROWSER_PARTS_DISPLAY_WIDTH = 15	Integer
HPEESOF_BROWSER_FIELD_NUM_ROWS The default number of rows of components listed for any selected sub-library. Example: HPEESOF_BROWSER_FIELD_NUM_ROWS = 20	Integer
HPEESOF_BROWSER_COMPONENT_DESCRIPTION_VISIBLE The default visibility status of the Description column. Example: HPEESOF_BROWSER_COMPONENT_DESCRIPTION_VISIBLE = 1	0 = False 1 = True
HPEESOF_BROWSER_PARTS_DESCRIPTION_WIDTH The default width of the Description column. Example: HPEESOF_BROWSER_PARTS_DESCRIPTION_WIDTH = 20	Integer
HPEESOF_BROWSER_COMPONENT_VENDOR_VISIBLE The default visibility status of the Vendor column. Example: HPEESOF_BROWSER_COMPONENT_VENDOR_VISIBLE = 0	0 = False 1 = True
HPEESOF_BROWSER_PARTS_VENDOR_WIDTH The default width of the Vendor column. Example: HPEESOF_BROWSER_PARTS_VENDOR_WIDTH = 10	Integer
HPEESOF_BROWSER_COMPONENT_LIBNAME_VISIBLE The default visibility status of the Library column. Example: HPEESOF_BROWSER_COMPONENT_LIBNAME_VISIBLE = 0	0 = False 1 = True
HPEESOF_BROWSER_PARTS_LIBNAME_WIDTH The default width of the Library column. Example: HPEESOF_BROWSER_PARTS_LIBNAME_WIDTH = 20	Integer
HPEESOF_BROWSER_COMPONENT_PLACEMENT_VISIBLE The default visibility status of the Placement column. Example: HPEESOF_BROWSER_COMPONENT_PLACEMENT_VISIBLE	0 = False 1 = True

Advanced Design System 2008

= 0	
<p>HPEESOF_BROWSER_PARTS_PLACEMENT_WIDTH</p> <p>The default width of the Placement column.</p> <p>Example:</p> <p>HPEESOF_BROWSER_PARTS_PLACEMENT_WIDTH = 15</p>	Integer
<p>HPEESOF_BROWSER_COMPONENT_WEBSITE_VISIBLE</p> <p>The default visibility status of the Website column.</p> <p>Example:</p> <p>HPEESOF_BROWSER_COMPONENT_WEBSITE_VISIBLE = 0</p>	<p>0 = False</p> <p>1= True</p>
<p>HPEESOF_BROWSER_PARTS_WEBSITE_WIDTH</p> <p>The default width of the Website column.</p> <p>Example: HPEESOF_BROWSER_PARTS_WEBSITE_WIDTH = 20</p>	Integer
<p>HPEESOF_BROWSER_COMPONENT_AVAILABILITY_VISIBLE</p> <p>The default visibility status of the Availability column.</p> <p>Example:</p> <p>HPEESOF_BROWSER_COMPONENT_AVAILABILITY_VISIBLE = 0</p>	<p>0 = False</p> <p>1= True</p>
<p>HPEESOF_BROWSER_PARTS_AVAILABILITY_WIDTH</p> <p>The default width of the Availability column.</p> <p>Example:</p> <p>HPEESOF_BROWSER_PARTS_AVAILABILITY_WIDTH = 10</p>	Integer
<p>HPEESOF_BROWSER_COMPONENT_LICENSE_VISIBLE</p> <p>The default visibility status of the License column.</p> <p>Example:</p> <p>HPEESOF_BROWSER_COMPONENT_LICENSE_VISIBLE = 0</p>	<p>0 = False</p> <p>1= True</p>
<p>HPEESOF_BROWSER_PARTS_LICENSE_WIDTH</p> <p>The default width of the License column.</p> <p>Example: HPEESOF_BROWSER_PARTS_LICENSE_WIDTH = 10</p>	Integer
<p>ADS_COMPONENT_SORT_ORDER</p> <p>The sort order for components.</p> <p>Example: ADS_COMPONENT_SORT_ORDER = Default</p>	<p>Default</p> <p>Ascending</p> <p>Descending</p>
<p>ADS_ANALOGRF_LIBRARY_SORT_ORDER</p> <p>The sort order for the analogRF library.</p> <p>Example: ADS_ANALOGRF_LIBRARY_SORT_ORDER = Default</p>	<p>Default</p> <p>Ascending</p> <p>Descending</p>
<p>ADS_HPTOLEMY_LIBRARY_SORT_ORDER</p> <p>The sort order for the Ptolemy library.</p> <p>Example: ADS_HPTOLEMY_LIBRARY_SORT_ORDER = Ascending</p>	<p>Default</p> <p>Ascending</p> <p>Descending</p>

Advanced Design System 2008

<p>HPEESOF_BROWSER_COMPONENT_HIDE_OBSOLETE Hides obsolete ADS library components from view in the ADS library browser. Example: HPEESOF_BROWSER_COMPONENT_HIDE_OBSOLETE = 1</p>	<p>0 = False 1= True</p>
<p>HPEESOF_BROWSER_COMPONENT_HIDE_UNLICENSED Hides unlicensed ADS library components from view in the ADS library browser. Example: HPEESOF_BROWSER_COMPONENT_HIDE_UNLICENSED = 1</p>	<p>0 = False 1= True</p>
<p>ADS_BROWSER_DEFAULT_WEBSITE Default website for ADS library records that do not have a website defined. Example: ADS_BROWSER_DEFAULT_WEBSITE = http://www.tm.agilent.com/tmo/hpeesof</p>	<p>URL String</p>
<p>HPANALOGRF_BROWSER_PATH ADS Analog/RF libraries related control, record and IDF files. All the control files and the IDF files in these directories are read to create the ADS Analog/RF related libraries and sub-libraries. Example: HPANALOGRF_BROWSER_PATH = \$HOME/hpeesof/circuit/records; \$HOME/hpeesof/esyn/records;\$HOME/hpeesof/rfsynthesis/records; \$HOME/hpeesof/CustomEncoded/records;\$HPEESOF_DIR/circuit/records; \$HPEESOF_DIR/esyn/records;\$HPEESOF_DIR/rfsynthesis/records; \$HPEESOF_DIR/CustomEncoded/records; \$HPEESOF_DIR/custom/circuit/records; \$HPEESOF_DIR/custom/esyn/records; \$HPEESOF_DIR/custom/rfsynthesis/records;{%DESIGN_KIT_BROWSER_PATH}</p>	<p>Path</p>
<p>HPVENDORLIB_BROWSER_PATH ADS Device libraries related control, record and IDF files. All the control files and the IDF files in these directories are read to create the ADS Vendor Component device related libraries and sub-libraries. Example: HPVENDORLIB_BROWSER_PATH = \$HOME/hpeesof/hpeesofbrowser/records; \$COMPL_DIR/ComponentLibs/records; \$HPEESOF_DIR/CustomEncoded/records</p>	<p>Path</p>
<p>HP_TEMPLATE_BROWSER_PATH Path information for the DDS template files. Each entry is in the form of</p>	<p>String Path</p>

<p><name> <path> where <name> is the top level node displayed in the browser and <path> is the path to the directory containing the templates. Example: HP_TEMPLATE_BROWSER_PATH = Product {\$HPEESOF_DIR}/circuit/templates; Customized {\$HPEESOF_DIR}/custom/circuit/templates; User {\$HOME}/hpeesof/circuit/templates; {%DESIGN_KIT_TEMPLATE_BROWSER_PATH}; {%MOMENTUM_TEMPLATE_BROWSER_PATH}</p>	
--	--

Variables in the File hpeesofdds.cfg

This configuration file is used to configure data displays.

Variable, description, and example	Value(s)
<p>AEL_PATH Top level AEL file Example: AEL_PATH = \$HOME/hpeesof/dds/ael/:\$HPEESOF_DIR/custom/dds/ael /:\$HPEESOF_DIR/dds/ael</p>	Path
<p>DDS_DEFAULTS_FILE Users default defaults file. This file gets read at start-up and written at shutdown with the current set of defaults. Example: DDS_DEFAULTS_FILE = \$HOME/hpeesof/config/ddsdefaults.ael</p>	Filename
<p>DDS_WIN_FOREGROUND Window foreground color Example: DDS_WIN_FOREGROUND = 0</p>	Integer
<p>DDS_WIN_BACKGROUND Window background color Example: DDS_WIN_BACKGROUND = 8</p>	Integer
<p>DDS_WIN_HEIGHT Window height Example: DDS_WIN_HEIGHT = 510</p>	Integer
<p>DDS_WIN_WIDTH Window width Example: DDS_WIN_WIDTH = 655</p>	Integer
<p>DDS_HIGHLIGHT_LINE_TYPE System-wide highlight line type</p>	Integer

Advanced Design System 2008

Example: DDS_HIGHLIGHT_LINE_TYPE = 0 [= solid]	
DDS_HIGHLIGHT_LINE_WIDTH System-wide highlight line width Example: DDS_HIGHLIGHT_LINE_WIDTH = 56 [= 4 points]	Integer
DDS_HIGHLIGHT_LINE_COLOR System-wide highlight line color Example: DDS_HIGHLIGHT_LINE_COLOR = 1 [= red]	Integer
DDS_DRAWING_UNITS_PER_INCH Default graphical object characteristic: Internal database resolution Example: DDS_DRAWING_UNITS_PER_INCH = 1000	Integer
DDS_PAGE_WIDTH Default graphical object characteristic: Plotting area width Example: DDS_PAGE_WIDTH = 33000	Integer
DDS_PAGE_HEIGHT Default graphical object characteristic: Plotting area is height Example: DDS_PAGE_HEIGHT = 21000	Integer
DDS_DEFAULT_LINE_WIDTH Default graphical object characteristic: Plotting line width Example: DDS_DEFAULT_LINE_WIDTH = 7	Integer
DDS_LEFT_CLICK_SENSITIVITY A left-mouse-button press-and-hold drag must move this many pixels or else it is considered a left click Example: DDS_LEFT_CLICK_SENSITIVITY= 2	Integer
DDS_DEFAULT_PLOT_WIDTH Default width of new plots Example: DDS_DEFAULT_PLOT_WIDTH = 3750	Integer
DDS_DEFAULT_PLOT_HEIGHT Default height of new plots Example: DDS_DEFAULT_PLOT_HEIGHT = 2318	Integer
DDS_DEFAULT_FONT Default font Example: DDS_DEFAULT_FONT = HersheyRomanNarrow	Font size, in points
DDS_NUMBER_OF_TRACE_COLORS Number of trace colors Example: DDS_NUMBER_OF_TRACE_COLORS = 16	Integer
DDS_TRACE n _COLOR n th color, where n = 1 to	Integer

Advanced Design System 2008

DDS_NUMBER_OF_TRACE_COLORS Example: DDS_TRACE16_COLOR = 90	
---	--

Variables in the File hpeesofhelp.cfg

This configuration file is used to locate the online documentation.

Variable, description, and example	Value(s)
TOPLEVEL_DOC Top level Index document location. Example: TOPLEVEL_DOC = \$HPEESOF_DIR/doc	Directory
LOCAL_DOC_PATH List of additional paths for the Help Server to search. Example: LOCAL_DOC_PATH = C:\newpath1;C:\newpath2;{% DESIGN_KIT_DOC_PATH}	List of Paths
LOCAL_BOOK_NAMES Optional list of new books. Example: LOCAL_BOOK_NAMES = newbook1;newbook2;{% DESIGN_KIT_BOOKNAMES}	List of names

Variables in the File hpeesofinstrio.cfg

This configuration file is used by the Instrument Server.

Variable, description, and example	Value(s)
AEL_PATH Top level AEL file Example: AEL_PATH = \$HOME/hpeesof/instrio/ael/: \$HPEESOF_DIR/custom/instrio/ael/:\$HPEESOF_DIR/instrio/ael:./	Path

Variables in the File momentum.cfg

This configuration file is used by Momentum.

Variable, description, and example	Value(s)
SUPL_GF_DATABASE	Directory

Advanced Design System 2008

Location of supplied Momentum Substrate files. Example: SUPL_GF_DATABASE = { \$HPEESOF_DIR }/momentum/lib	
SITE_GF_DATABASE Location of user Momentum Substrate files. Example: SITE_GF_DATABASE = { \$HOME }	Directory
LOCL_GF_DATABASE Location of project Momentum Substrate files. Example: LOCL_GF_DATABASE = ./	Directory
MOM_SLM_PATH Path to search for Momentum Substrate file that is saved with design files. Example: MOM_SLM_PATH = ./networks: { %SUPL_GF_DATABASE }	Path
MOMDDS_FILE On UNIX, location of default template file for Momentum Data Display. Example: MOMDDS_FILE = { \$HPEESOF_DIR }/sess/ael/momdds.ael	Filename
MOMMSH_REFACTET_ARC_RESOLUTION Maximum resolution for the automatic refacetting of arcs and circles in Momentum. Setting a lower value results in a higher resolution representation with more unknowns in the mesh, resulting in a longer simulation time. Example: MOMMSH_REFACTET_ARC_RESOLUTION = 45	Integer
MOM3D_USE_MATRIXSOLVER Sets solver algorithm to non-blocked or blocked. Example: MOM3D_USE_MATRIXSOLVER = 0	0 = non-blocked 1 = blocked
MOM3D_USE_MATRIXSOLVERBLOCKSIZE On UNIX, the block size used to tune the linear solver. Example: MOM3D_USE_MATRIXSOLVERBLOCKSIZE = 16	Integer

Variables in the File smg_ui.cfg

This configuration file is used by the SPICE Model Generator.

Variable, description, and example	Value(s)
SMG_AEL_PATH PATH to look for top level AEL file.	Path

Example: SMG_AEL_PATH={%HOME}/hpeesof/smg/ael: {%HPEESOF_DIR}/smg/ael: {%HPEESOF_DIR}/ael_smg_ui	
SMG_UI_AEL Top level AEL file. Example: SMG_UI_AEL = smg_main	Filename

Variables in the File hpeesofsim.cfg

Variable, description, and example	Value(s)
AEL_PATH Search path for locating AEL files loaded by the simulator Example: AEL_PATH = %HOME/hpeesof/circuit/ael: %HOME/hpeesof/expressions/ael:%HPEESOF_DIR/custom/circuit/ael: %HPEESOF_DIR/custom/expressions/ael:%HPEESOF_DIR/circuit/ael: %HPEESOF_DIR/expressions/ael:.	Path
USER_MODEL_PATH Path for User-Compiled models Example: USER_MODEL_PATH = /some/path/to/model/directory:/some/path/to/a/directory	Path
USER_SIM_FILE_PATH Path for user-defined simulation files Example: USER_SIM_FILE_PATH = /path/to/my/directory:/some/path/to/somewhere	Path
ADSLIBCONFIG_PATH Path for ADS library configuration file Example: ADSLIBCONFIG_PATH = %_HP_GEMINI_LIBPATH: % DKIT_ADSLIBCONFIG_PATH:%HOME/hpeesof/circuit/config: %HPEESOF_DIR/custom/circuit/config:%HPEESOF_DIR/circuit/config:.	Path

Variables in the File hpeesofsess.cfg

This configuration file is used by the Status window.

Variable, description, and example	Value(s)
STATUS_LINES_SHOWN Sets the maximum number of lines visible in the Status window.	Integer

<p>Example: STATUS_LINES_SHOWN =100</p>	
<p>SIMEXEC_MAP_TO_FRONT Sets the default behavior of the Status window, where on (the default) means the Status window pops to the front of the screen anytime new messages are displayed in it, and off means it does not pop forward automatically. Example: SIMEXEC_MAP_TO_FRONT = on</p>	<p>ON or OFF</p>
<p>HPEESOFSIM_BIN Path to the hpeesofsim executable. Relative pathnames are evaluated relative to the current project's top-level directory. Note that using HPEESOFSIM_BIN to point to a user-defined hpeesofsim executable will cause problems with statically-linked user-compiled models since the simulator containing the models is prevented from being used. If this variable is changed, ADS must be restarted for the changes to take effect. Example: HPEESOFSIM_BIN=hpeesofsim</p>	<p>Path</p>

Setting Design Environment Preferences

You can easily customize many aspects of design entry and display through the Preferences and Layer Editor dialog boxes accessed through the Options menu. Some of the options set here serve as defaults and can be changed on an individual basis through the Edit menu.

Some of the things you can customize are:

- The default settings for color of the grid, pins/tees, highlighted and selected objects, and the background and foreground of the design windows
- The sizes of pins/tees, the selection pick box, and the size of the marker drawn when you have the Vertices filter turned on and select the vertices of objects in the drawing area
- The display of the Component Parameters dialog box, coordinate readouts, pin names and numbers, and pins/tees

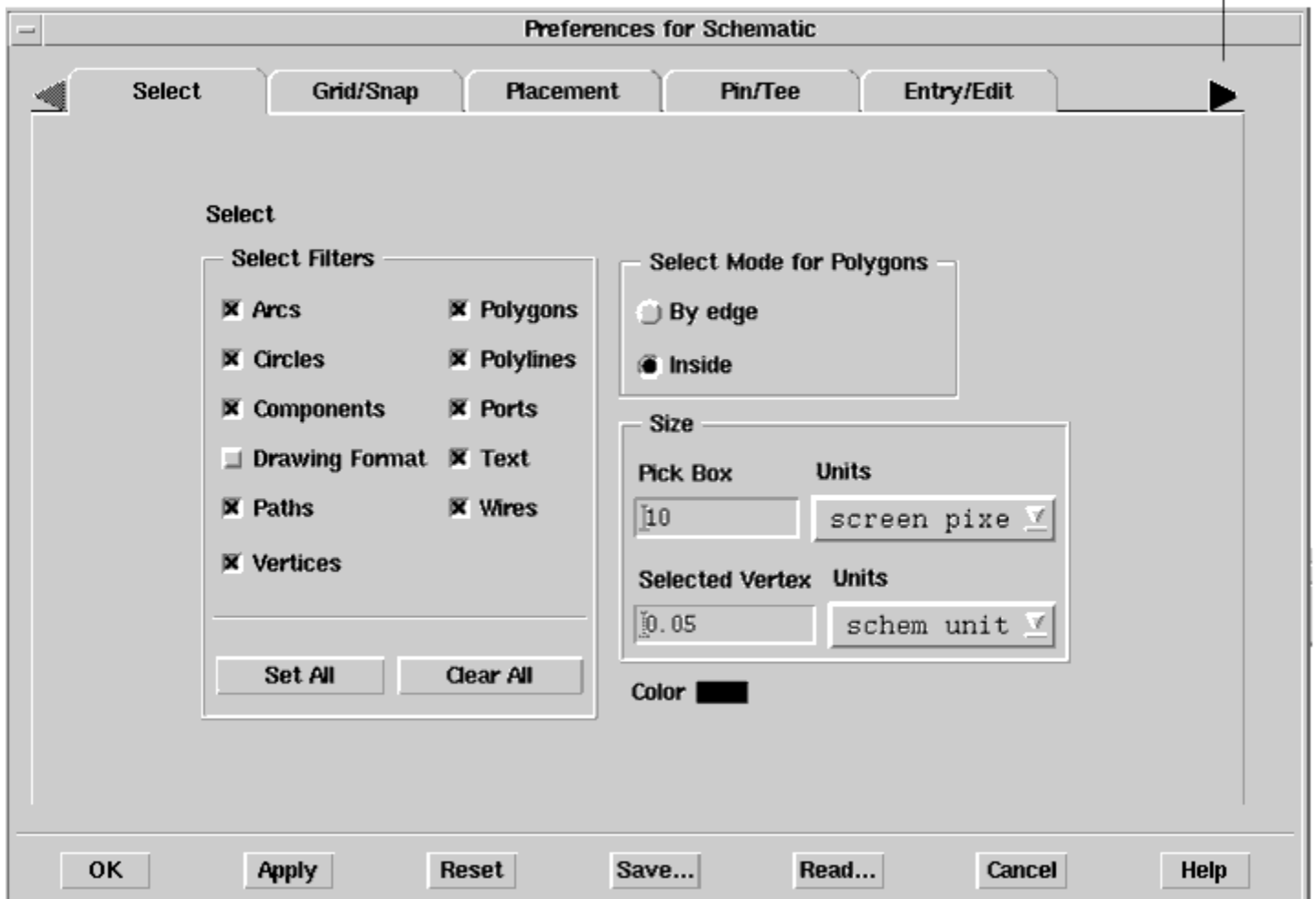
When you change the settings in this dialog box and click Apply, the design window is updated with the changes, and these changes will serve as defaults for all designs in this project. For details on saving changes to a file so that specific preferences can be associated with specific designs, refer to [Saving and Reading Preference Files](#).

Specifying Design Entry and Display Preferences

To change design entry and display preferences:

Choose Options > Preferences in any design window.

Click to see additional tabs



Many options relating to the size of an item displayed on the screen offer a choice of specifying the size in terms of screen pixels or schem units (or layout units, in the Layout window).

screen pixels—Use this setting to specify sizes in terms of pixels on the screen. For example, if you set 10 screen pixels for the Pick Box size, the pick region will be 10 pixels by 10 pixels.

schem units—Use this setting to specify sizes in terms of inches, in the Schematic window. In the Layout

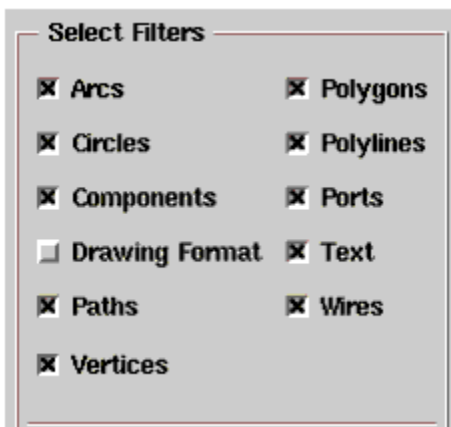
window, select layout units to specify a size with respect to the design units of the Layout window.

Setting Select Options

The Select options can assist you in editing your designs by modifying how items are selected.

Using Selection Filters

Selection filters enable you to specify types of items you want included in or excluded from your selections. For example, if you turn on only Components and Wires, none of the other types of items in the drawing area will be available for selection. By default, all types of items are turned on except Drawing Format.



To change the default settings:

Select the types of items you want available for selection, and deselect the types of items you want excluded from selection.

Hints:

- Any item type that is turned off will not be selected when you click it individually, attempt to enclose it in a selection window, or choose the Select All command.
- Only the Select By Name and Deselect By Name commands ignore the selection filters.
- To enable most filters, choose Set All to quickly select all filters, then deselect those you want excluded.
- To disable most filters, choose Clear All to quickly deselect all filters, then select those you want included.

Changing the Select Mode for Polygons

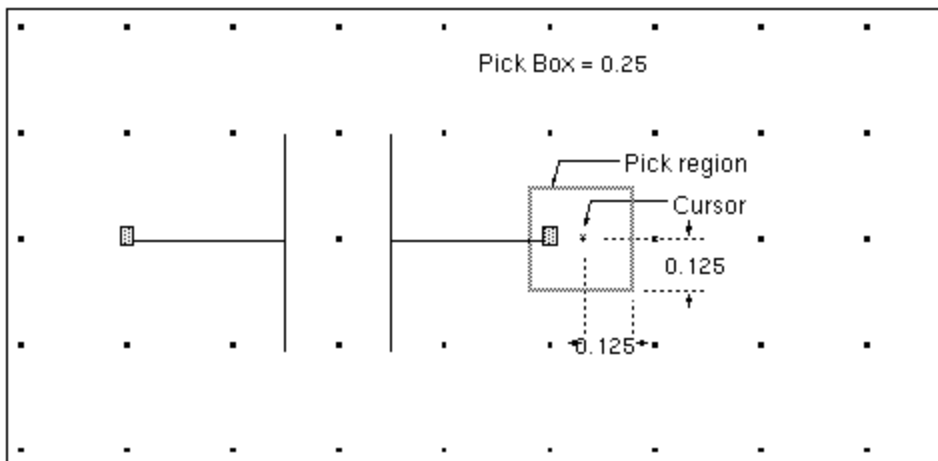
Advanced Design System 2008

- Choose By Edge to be able to select polygons by clicking on the outer edge.
- Choose Inside if you want to be able to select polygons by clicking anywhere inside the shape.

Changing the Pick Box Size

The pick box is a region you define that determines how close your cross cursor must be to an object before clicking will select that object. You can choose a size in schem/layout units, relative to the units of the design window, or you can choose a size in screen pixels.


The following figure shows an example of the region defined by a pick box specified in inches.



Pick region based on schem Units

In the previous figure, the pick box size was set to 0.25 inch (the same as the default grid display).

When you define the region using n screen pixels, the pick region is a square of n pixels \times n pixels, centered around the spot where you click. For example, if you specify 20, the pick region extends 10 pixels beyond the spot where you click, horizontally and vertically.

Size	
Pick Box	Units
10	screen pixels 



Hint

It may be necessary to use a very small number in designs where items are tightly spaced.

Changing the Selected Vertex Size

When the Vertices filter is enabled, and you select a vertex (or vertices), a marker appears identifying each selected vertex. You can change the size of the marker from the Select tab of the Preferences dialog box.

To change the size of the marker that identifies selected vertices:

1. Choose the desired units from the drop-down Units list box.
2. Change the value as desired.


Changing the Select Color

When objects in the drawing area are selected, a box is drawn around them identifying them as being selected. By default, this highlight color is black, but you can change it from the Select tab of the Preferences dialog box. This color is also the color of the marker that identifies selected vertices.


Setting Grid/Snap Options

The display grid and cursor snap features are provided to assist you in creating and editing your designs more quickly and accurately.

By default, snap mode is turned on and the cursor snaps to pins and to the grid defined by the snap spacing. The default snap spacing is 0.125 inch (in the Schematic window) with a display factor of 2. This means that although the cursor snaps every 0.125 inch, the dots only appear every 0.25 inch. If you set the display to anything smaller than 0.25 inch, the grid will be too dense to display without zooming in on it.

 **Hint**
The default component symbols have been created uniformly, in 0.125-inch increments. Thus, if you keep the default settings while creating your design, you should be able to connect all symbols with minimal effort.

Display

<input type="radio"/> Dots	Select Dots or Lines to display a visible grid made up of dots or lines
<input type="radio"/> Lines	Select None if you want no visible grid
<input checked="" type="radio"/> None	Click to access a palette for choosing a grid color
Color 	

Spacing

- Snap Grid Distance (in schem units) represents the snap spacing, where the number you specify determines the distance (in design units, or inches) between the points to which the cursor snaps.


Snap Grid Distance (in schem units)

X	<input type="text" value="0.125"/>	Y	<input type="text" value="0.125"/>
---	------------------------------------	---	------------------------------------

- Snap Grid per Display Grid represents the spacing between the dots (or lines) on the display grid in terms of a factor of the snap spacing. The smaller the number, the finer the grid.


Snap Grid per Display Grid

X	<input type="text" value="2"/>	Y	<input type="text" value="2"/>
---	--------------------------------	---	--------------------------------

 **Hint**
If the display factor you specify makes the grid too dense to display, it is invisible unless you zoom in. To see the grid without zooming, choose a larger display factor.

- Snap Distance - all other modes represents how close to an object the cursor must be before it will snap to that object.

Snap Distance - all other modes


Diameter	Units
<input type="text" value="15"/>	<input type="text" value="screen pixe"/> 

Active Snap Modes


Advanced Design System 2008

This section enables you to restrict the manner in which the cursor snaps. You can activate any combination of choices. If you select more than one, the cursor snaps to the nearest one. By default, Grid and Pin are turned on to assist you in quickly creating schematics.


- Enable Snap -Toggles snap mode on and off. You can also toggle snap mode on and off from the Options menu itself (Snap Enabled).
To select all or most snap modes:
Click Set All and then deselect those you want excluded.
To deselect all or most snap modes:
Click Clear All and then select those you want included.

 **Hint**
All snap modes (except Grid) rely on the cursor being within the distance specified as Snap Distance (Diameter).

- Pin -When the pin of an object you are positioning gets within the snap distance of a pin on an existing object, the pins are automatically connected. Pin snapping takes priority over all other snap modes.

 **Note**
Angle Snapping automatically occurs when only Pin snapping is enabled and you place a part so that the pin at the cursor connects to an existing part. The placed part rotates so that it properly aligns with the connected part.

- Vertex -When the object you are positioning gets within the snap distance of a vertex on an existing object, the object you are positioning is automatically placed at that vertex. This snap mode is especially helpful if something was originally drawn or placed on the grid and then moved off, or the grid spacing has been changed.
- Grid -The cursor snaps to points on the grid defined by snap spacing. All other snap modes have priority over grid snap mode.

 **Hint**
The Reset button returns settings to their defaults (if you have not yet clicked Apply).

Setting Placement Options

Set the following options-related to placing components in the drawing area-in a manner that suits the way you work:

Defining the Placement Mode for Schematic and Layout Representations

When you are working with both schematic and layout representations, you can select the simultaneous placement mode that works best for you. For schematic only design work, use the default option, Single Representation.

Advanced Design System 2008

- **Single Representation (schematic OR layout)**
When you place an item in one representation, nothing is placed automatically in the other representation.
- **Dual Representation (schematic AND layout)**
When you place an item in one representation and move the pointer into the window for the other representation, the equivalent component is already selected. Position the pointer as desired and click to place it. (If a window for the other representation-containing the same design-is not open, one will be opened automatically.)
- **Always Design Synchronize (schematic AND layout)**
Causes the program to fully synchronize both representations after each part is placed, ensuring all parts are fully interconnected. This takes more time than the Dual Representation mode and may move or rearrange the layout of the schematic to preserve connectivity.

Displaying the Component Parameter Dialog Box by Default

When you place a component in the drawing area, you can change parameters using the on-screen editor. Alternatively, you can make changes through the Component Parameter dialog box. By default, in the Schematic window, the option that controls the automatic display of the component parameter dialog box is turned off. You can turn this option on if you want the dialog box to be displayed every time you select a component (Options > Preferences > Placement).

- **Show Component Parameter Dialog Box**
Controls whether or not the Component Parameters dialog box appears every time you select an item to place in the drawing area.
- **Show Component Parameter Dialog Box for components without parameters**
Select this option if you want the Component Parameters dialog box to be displayed even for components that do not have parameters (for example, a design used as a subnetwork for which no parameters have been defined through File > Design Parameters). By default it is off and the dialog box does not appear.

Setting a Default for Component Swapping

- **Keep the original instance name(s) when swapping components**
Retains the component ID of the original component when using the Swap Components command. (Tip: this setting serves as a default; it can be changed for any given operation through the Swap Components dialog box.)

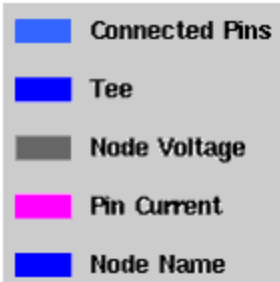
Setting Pin/Tee Options

You can change several options relating to pins/tees through Options > Preferences > Pin/Tee.

- **Pin Size**
Enables you to change the size of the pins drawn on all components.

Advanced Design System 2008


- Tee Size
Enables you to change the size of the Tee connections between interconnected wires.
- Color
Enables you to specify colors for the items shown here. Click each to display a color palette with the available selections.



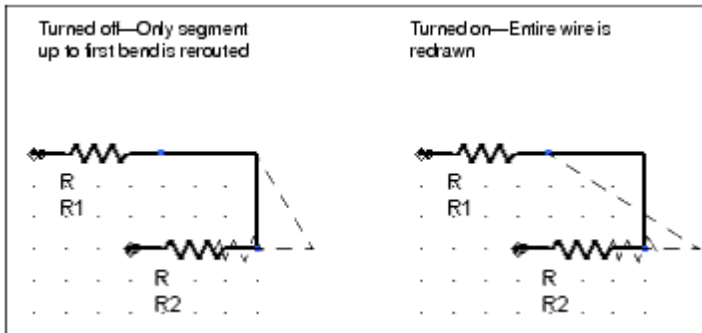
- Visibility
Enables you to change the visibility status of pin numbers and names, and whether or not pin connections are identified by markers.
Connected Pins -Select this option to display a marker identifying a pin connection. The marker is drawn using the size specified in the Pin/Tee tab of the Preferences dialog box.
Pin Numbers -Select this option to display pin numbers.
Pin Names -Select this option to display pin names.

Setting Entry/Edit Options

This group of options allows you to control several aspects of shape entry and editing including the angle at which lines and wires are drawn, the resolution of arcs and circles, and how wires are routed relative to component text.

 **Note**
Changes made to the settings in this dialog box exist only in memory unless you save them to a file. For details on saving to a file, see [Saving and Reading Preference Files](#).

- Reroute entire wire attached to moved component
When this option is selected, the wire connection is allowed to be completely redrawn and rerouted as needed. When this option is deselected, only the segment (up to the first bend) of the wire attached to the component you are moving is rerouted; the remainder of the wire is unaffected.

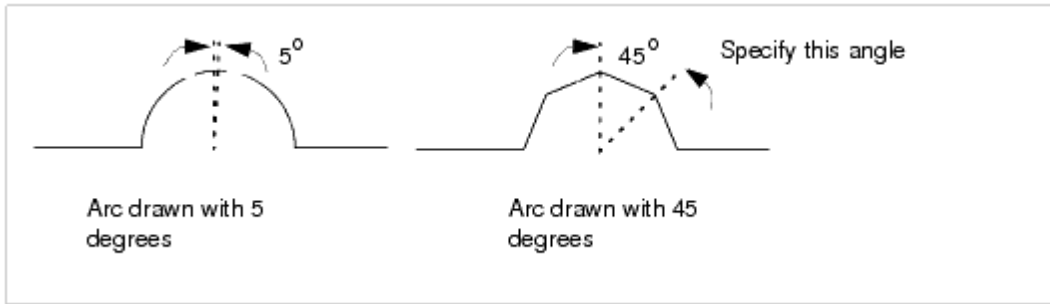


- Route around component text
By default, wires are routed through component text. If you want wires routed around component text, select this option. Note that routing around pins, wire endpoints, and collinear wires takes precedence. If the program cannot route wires around these items, as well as the component text, it will route wires through the component text.
- Route around component symbol
By default, wires are routed through component symbols. If you want wires routed around component symbols, select this option. Note that routing around pins, wire endpoints, and collinear wires takes precedence. If the program cannot route wires around these items, as well as the component symbols, it will route wires through the component symbols.
- Polygon Entry Mode: Any angle
Enables you to draw polylines, polygons, and wires using all angles.
- Polygon Entry Mode: 45 degree angle only
Restricts shape entry to 45 degree rotation increments.
- Polygon Entry Mode: 90 degree angle only
Restricts shape entry to horizontal or vertical.
- Show Coordinate Entry Dialog for Insert and Edit commands
Select this option to force the Coordinate Entry dialog box to be displayed when invoking the following commands:

Insert (Shape)—Polygon, Polyline, Rectangle, Circle, Arc (clockwise and counter-clockwise), Text, Construction Line, Symbol Pin, Path, Trace.

Edit—Move Wire Endpoint, Mirror X, Mirror Y, Move & Disconnect, Step And Repeat, Set Origin, Move Component Text.

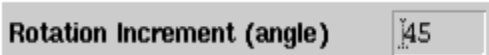
- Show Set Paste Origin Dialog for Copy Command
Select this option to force the Set Paste Origin dialog box to be displayed when you choose the Copy command. This dialog box enables you to specify X and Y coordinates to be used as a reference point when pasting.
- Polygon self-intersection checking
Prevents you from placing additional points on a polygon if overlapping lines will result.
- Maintain adjacent angles for Move Edge command
Restricts the Move Edge command to stretch an edge while maintaining the adjacent angles of the edge being stretched to other edges adjacent to that edge.
- Arc/Circle Resolution (degrees)
This setting determines how smoothly curves are drawn. The length of each line segment making up the arc is determined by the size of the angle drawn using the specified number of degrees.



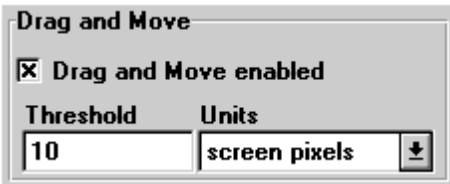
In general, the smaller the number of degrees, the smoother the shape, but the longer it will take to redraw the screen.

Note
This setting only affects circles in that the number specified here is used if you convert a circle to a polygon.

- **Auto-backup edit count**
Your file is automatically saved every time the number of edits you have performed reaches the number in this field.
- **Undo edit count**
This option represents the maximum number of commands held in the stack. Selecting Undo from the Edit menu or clicking the Undo button on the toolbar undoes the last editing command. A stack of edit commands is maintained for each window, thus the Undo command works independently from window to window. You can choose Undo repeatedly to return to an earlier state of your design. You can specify the number of commands you want the stack to hold using the Undo edit count option.
- **Rotation Increment (angle)**
This option forces objects you rotate to snap-during rotation-in n -degree increments, where n is the number you specify here.



- **Drag and Move**
This option is designed to prevent you from moving an item when you click to select it (for any purpose) and unintentionally move the pointer in the process. By default, a move less than 10 screen pixels is not recognized as a move. An intentional move must be more than the distance specified here for it to be recognized as a move.



Setting Component Text/Wire Label Options (in Advance)

Component text is the text associated with components selected from a Library or Palette. This text appears automatically when you place the component in the drawing area and consists of a name, a unique ID, and parameters (where applicable).

There are two ways to change component text attributes:

- In advance of placing components, through the Options menu (Options > Preferences > Component Text/Wire Label). This setting serves as a default, but attributes of component text for an individual component, or all components, can be changed later through the Edit menu.
- After placing components, through the Edit menu (Edit > Component > Component Text Attributes). Refer to "Changing Component Text Attributes" in the Schematic Capture and Layout manual.

You can specify the following display characteristics of component text: font, point size, maximum numbers of rows displayed in a single column, precision, and the layer on which each type of text is placed.

- **Font Definition**

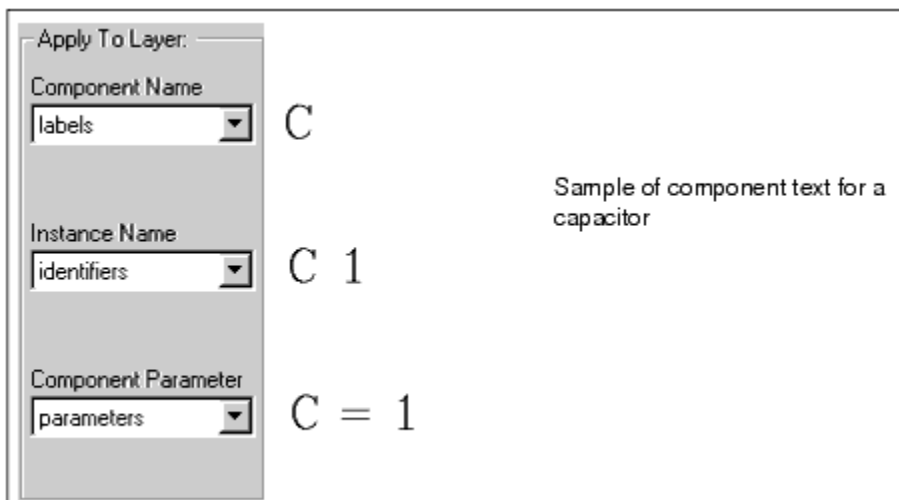
All TrueType fonts installed on your system are available. Select the desired font type from the drop-down list. When printing to an HP-GL/2 file, text information will not be saved if the font is a TrueType font. To preserve the text in your output file, convert it to HersheyRomanNarrow before saving to HP-GL/2.

Note
On UNIX, if you want to add additional TrueType fonts that were not supplied with ADS, copy them to \$HPEESOF_DIR/lib/fonts.

Point represents the size of text in traditional units used in printing.

- **Apply To Layer**

Each of the three types of component text resides on its own layer. This enables you to quickly change the appearance of the component text by changing the attributes of a layer, or making a layer invisible. For more information on changing layer attributes, refer to [Specifying Layer Definitions](#).



To change the layer for any given type of component text, click the arrow and select a new layer from the drop-down list.

- **Parameter Rows**

Represents the maximum number of rows of component text displayed in one column. Using a relatively small number here is helpful for large parameter sets so you can view the parameters in several short columns.

- **Format**

Enables you to abbreviate the displayed annotation on a schematic for Tune, Opt, Stat, and DOE syntax. This is useful when the standard annotation overlaps on the schematic. The format for Tune, Opt, Stat, and DOE can be set to one of the following three types:



- Full displays the standard syntax used in ADS 2004A and before (e.g., 50 Ohm tune{ 25 Ohm to 75 Ohm by 5 Ohm }).
- Short displays an abbreviated syntax (e.g., 50 Ohm {t}):
 - {t} for tune and {-t} for notune
 - {o} for opt and {-o} for noopt
 - {s} for stat and {-s} for nostat
 - {d} for doe and {-d} for nodoe
- None only displays the nominal values (e.g., 50 Ohm).
- Wire/Pin Label
 - Enables you to select defaults for the font, point size, and color of wire labels.

Setting Text Options (in Advance)

You can establish text attributes-prior to adding text to your design-that affect all subsequently added text. Establishing attributes in advance is done through the Options menu. Editing attributes of existing text is done through the Edit menu.

To establish text attributes:

1. Choose Options > Preferences > Text and set the text attributes as desired.
 - Font Type -All TrueType fonts installed on your system are available. Select the desired font from the drop-down list. When printing to an HP-GL/2 file, text information will not be saved if the font is a TrueType font. To preserve the text in your output file, convert it to HersheyRomanNarrow before saving to HP-GL/2.

Note
On UNIX, if you want to add additional TrueType fonts that were not supplied with ADS, copy them to \$HPEESOF_DIR/lib/fonts (where \$HPEESOF_DIR represents your complete installation path).

- Point -Represents the size of text in traditional units used in printing.
- Justification, Horizontal -This setting represents two types of justification: one is how individual lines of text in a block of text are aligned with one another; the second is how an individual line of text or block of text is positioned horizontally, relative to the reference point you specified to begin typing the text.
- Justification, Vertical -This setting aligns a string or block of text vertically, relative to the reference point you specified to begin typing the text.
- Placement Angle -The angle at which all text subsequently added to your design will be drawn.
- Non-rotating (when in hierarchy) -Select this option to prevent text on a symbol or design from being

rotated when the symbol is rotated.

2. Change any or all options as desired and click Apply (or OK if you are not changing any other preferences).

Setting Display Options

The Display tab enables you to change the Foreground, Background, and Highlight colors.

- **Foreground**
The color of the lines making up polygons, polylines, and arcs while they are being drawn.
- **Background**
The color of the drawing area background in the design window.
- **Highlight**
The color used to identify problem items (with respect to simulation), orphaned items in schematic and layout representations, and unconnected pins.
- **Fixed Component**
The color of fixed components.
- **Deactivated Component Text**
The color of component text for deactivated components.
- **Minimum Object Size To Display (in pixels)**
The minimum size—in pixels—an object must be before it is actually drawn in the Schematic window. Objects smaller than this are not visible.

Setting Units/Scale Options

With the exception of the Resistance setting, the settings in the Units/Scale tab of the Preferences dialog box serve as defaults only in the following situations:

- When a parameter of a supplied component does not have a default unit and you do not assign one (in the component parameter dialog)
- When you supply a default parameter value without units while creating a parametric subnetwork (File > Design Parameters)

The Resistance setting can be changed to serve as a default for all resistors (subsequently) placed in this project.

Setting Tuning Options

The Tuning settings in the Preferences dialog box serve as defaults and can be changed during tuning in the Tune Control and Tune Control Details dialog boxes.

Select the Tune Analysis mode that you want to serve as the default:

- Analysis Mode
 - Single—Perform analysis after each change.
 - Multiple—Perform analysis only after the Tune button is clicked. This is designed for tuning after multiple changes, but can be used for single changes.
 - Continuous—Perform analysis while the slider is moving.

Set the Tune State to the desired default:

- Data Displays
 - Restore data displays—Opens all the data displays that were saved for a design when tuning is launched.

Set the following Slider options to the desired defaults:

- Range Min and Max—Set the minimum and maximum range to the initial parameter value plus or minus the percent you enter here.
- Step Size—Enter a percentage of the initial parameter value.
- Slider Scaling—Choose Linear or Logarithmic.
- Snapping—Choose Snap to Step to increment Linear sliders by Step Size. When OFF, Linear sliders increment continuously.

Saving and Reading Preference Files

When you create a project, the files `schematic.prf` (for schematics) and `layout.prf` (for layouts) are copied to the new project from the installation directory. By default, all designs in a given project use the preference file by this name. You can customize these preference files, as well as create additional preference files by other names.

- To use the same set of customized preferences for every design in the project, customize the files with the default filenames (`schematic.prf` and `layout.prf`).
- To use a different preference file for any given design in a project, open that design, customize the preferences as needed, and save them to a preference file with a unique name. To associate this customized preference file with a another design, open that design, read the customized preference file, and save the design.

You can save any number of files containing customized preferences. Whenever you open a design, the last preference file associated with it is automatically read.



Note

When you save a preference file, all the current settings found in the Preferences dialog box are also saved, with the following exception: Options that may be set differently within individual designs, such as text height, are not saved in the preference file.


Advanced Design System 2008

To customize preferences while retaining the default filenames:

1. Choose Options > Preferences.
2. Change preferences as desired and click OK. The preference files with the default filenames are updated. Every design in the project will now use these preferences unless you explicitly associate a unique preference file with a given design.


To create customized preference files with unique filenames:

1. Choose Options > Preferences.
2. Change any desired settings and click Save. The Save Preference File dialog box appears displaying the default filename, schematic.prf.
3. Enter a name of your choosing (the .prf extension is added automatically) and click OK.

 **Hint**
If you have a design open when you create a customized preference file, that design will take on the preferences just saved when you click OK in the Preferences dialog box. However, this association is only in memory unless you save the design file.

To associate a previously saved preference file with a specific design:

1. Open that design.
2. Choose Options > Preferences and click Read. The Read Preference File dialog box appears.
3. Select the desired preference file from the list of files, and click OK. (You can read in preference files from other project directories.)
4. Save (File > Save) the design.

 **Hint**
The variable that defines the search path for these files is PREFERENCES_PATH. For details refer to [PREFERENCES_PATH](#).

Preference File Format and Descriptions

In a preference file, each preference must be on a separate line and must use the following format:
preference_name value

Preference Name	Description
autoRepeatableCompPlcmnt	Auto-repeatable component placement 0=Off 1=On
backgroundColor	Color of background represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0,

Advanced Design System 2008

	to the number of colors defined in eecolorRGB.cfg
backupCount	Number of edits before auto-backup of design
bboxColor	Color of bounding boxes below the hierarchical plotting depth represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
checkBinding	Wire/Trace check layer binding (Layout only) 0=Off 1=On
checkIntersection	Polygon self-intersection checking 0=Off 1=On
coordEntryPopup	Show Coordinate Entry dialog for Insert and Edit commands 0=Off 1=On
currentColor	Color of pin current represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
dispSubnetInstNames	Display instance names of components in subnetworks (Layout only) 0=Off 1=On
dragMove	Drag and Move enabled 0=Off 1=On
dragMoveThresholdSize	Drag and Move threshold size
dragMoveThresholdUnits	Drag and Move threshold units 0=User units 1=Screen pixels
DSEartDistanceX	Connecting wire length in X direction in layout units for Generate/Update Layout
DSEartDistanceY	Connecting wire length in Y direction in layout units for Generate/Update Layout
dseFixAllComponents	Default value for fixing all components in destination Schematic or Layout 0=Off 1=On
DSEL2Sreport	Show status report from Generate/Update Schematic
dsePrefLayoutLayer	Entry layer name for Generate/Update Layout command

Advanced Design System 2008

DSES2Lreport	Show status report from Generate/Update Layout
DSEsymbDistanceX	Connecting wire length in X direction in schematic units for Generate/Update Schematic
DSEsymbDistanceY	Connecting wire length in Y direction in schematic units for Generate/Update Schematic
dualPlacement	Control of simultaneous placement of components in Layout and Schematic 0=Single representation 1=Dual representation 2=Always Design Synchronize
dveBinWidth	Verification bin width
dveEpsilon	Verification epsilon
dveFringe	Verification fringe
dveRealMemory	Verification memory in MB
dveSortMode	Verification sort GEM layers 0=Off 1=On
dveStoragePerArea	Verification memory storage per area in bytes
editInPlaceBoxColor	Color of Edit In Place box represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
entryMode	Polygon entry mode 0=Any angle 1=45 degree angle only 2=90 degree angle only
fixedInstHighlightColor	Color of fixed components represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
forceSynchronize	Delete equivalent components in destination Schematic or Layout that have been deleted/deactivated in source Schematic or Layout
foregroundColor	Color of foreground represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
genericArtworkSize	Generic artwork size for Generate/Update Layout command (Layout only)
genericSizeUnits	Generic artwork size units for Generate/Update Layout command (Layout only)
globalArcResolution	Resolution of arcs in degrees

Advanced Design System 2008

gridColor	Color of grid dots/lines represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
gridDisplayMode	Grid display type 0=Dots 1=Lines
gridDisplayOn	Snap grid display 0=Off 1=On
gridDisplaySameXY	Automatically set Y=X when any X value is modified in Preferences>Grid/Snap>Spacing dialog box
gridDisplayX	Snap grid per display grid along X axis
gridDisplayY	Snap grid per display grid along Y axis
gridSnapMode	Bitwise value of items to be snapped to 1=Grid 2=Pin 4=Vertex 8=Edge 16=Midpoint 32=Arc center 64=Intersection
gridSnapOn	Enable snap 0=Off 1=On
gridSnapX	Snap grid distance along X axis in user units
gridSnapY	Snap grid distance along Y axis in user units
highlightColor	Color of highlighted objects represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
instanceRotate	Auto-rotate components being placed (only in layout) 0=Do not auto-rotate 1=Auto-rotate
instIDLayer	Layer number used for component text instance name (e.g. C1)
instNameLayer	Layer number used for component text name (e.g. C)
instParam1Layer	Layer number used for component text parameters (Schematic only)
instTextDoeFormat	Component text format for doe syntax where 0=Full

Advanced Design System 2008

	<p>1=Short 2=None</p>
instTextFontName	Component text placed in design will use the font given here
instTextOptFormat	<p>Component text format for opt syntax where</p> <p>0=Full 1=Short 2=None</p>
instTextPoint	Component text size in points
instTextRows	Number of rows of component text displayed in one column
instTextStatFormat	<p>Component text format for stat syntax where</p> <p>0=Full 1=Short 2=None</p>
instTextTuneFormat	<p>Component text format for tune syntax where</p> <p>0=Full 1=Short 2=None</p>
layoutIncr	<p>Layout increment is a multiplier against the precision. An increment of 2 and precision of -3 would make the conversion factor $.001 * 2 = .002$. In practice, an increment other than 1 is rarely used. Note: This preference is not used when a design is opened.</p>
layoutPrec	<p>Layout precision is an integer representing the conversion of user units to data base units. It is the power of 10 used to multiply against a data base unit to get a user unit.</p> <p>e.g., prec -3 = 10^{-3} or .001 If db unit = 1000, then $1000 * .001 = 1$ user unit</p> <p>Note: This preference is not used when a design is opened.</p>
layoutUnits	<p>Layout unit where</p> <p>1=mil 2=in 3=um 4=mm 5=cm 6=meter 7=ft</p>

Advanced Design System 2008

	Note: This preference is not used when a design is opened.
maintainAngle	Maintain adjacent angles for Move Edge command 0=Off 1=On
majorGridDisplayOn	Major snap grid display (Layout only) 0=Off 1=On
majorGridDisplayX	Minor grid per major display grid along X axis (Layout only)
majorGridDisplayY	Minor grid per major display grid along Y axis (Layout only)
minPixelDisplaySize	Minimum object size to display in pixels
minVertexDistance	Final minimum vertex distance for Merge/Boolean Logical/Create Clearance in Layout units (Layout only)
miterVertexLength	Desired length of the mitered edge in user units
moveVertexKeepRect	Drag rectangle vertex maintains rectangularity 0=Off 1=On
newRouteAroundInstSym	Wire avoidance routing around component symbol (Schematic only)
newRouteAroundInstText	Wire avoidance routing around component text (Schematic only)
nodeNameColor	Color of node names represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
onscreenCoordMode	On-screen coordinate display at cursor (Layout only) 0=None 1=Absolute 2=Relative
oversize	Default value in user units which the oversize command will use
oversizeMiter	Cutoff angle for mitering corners. Any angle of a polygon smaller than the specified cutoff angle is mitered. The default cutoff angle is 45 degrees.
pathBend	Bend type of paths and traces where 1=Mitered 2=Square 3=Curved

Advanced Design System 2008

pathMiterPercent	Mitered corner cutoff ratio of mitered paths or traces in percent
pathRadius	Curve radius of curved paths or traces in user units
pathWidth	Width of paths and traces in user units
pinColor	Color of component pins represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
pinSize	Size of pins
pinSizeUnits	0=User units 1=Screen pixels
pinSnapSize	Snap distance for all modes except grid
pinSnapUnits	Snap distance units 0=User units 1=Screen pixels
placePopUp	Show component parameter dialog box during placement
placePopupOnZeroParm	Show component parameter dialog box for components without parameters during placement
plotDepthForLessThanMinPixels	Maximum hierarchical depth at which to draw box for objects smaller than minimum object size
plotLessThanMinPixels	Display box for objects smaller than minimum object size
plotPinNames	Plot pin names next to component pins 0=Off 1=On
plotPinNumbers	Plot pin numbers next to component pins 0=Off 1=On
plotPins	Plot component pins 0=Off 1=On
plottingDepth	Hierarchical plotting depth
portColor	Color of Symbol pins represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
portOrientation	Default value of Symbol Pin angle
portSize	Port/Ground size (Layout only)
portSizeUnits	Port/Ground size units (Layout only) 0=User units

Advanced Design System 2008

	1=Screen pixels
rerouteTraces	Reroute entire trace attached to moved component (Layout only) 0=Off 1=On
rerouteWires	Reroute entire wire attached to moved component 0=Off 1=On
rotationIncrement	Rotation increment in degrees
scaleX	Default value of scale command in X direction
scaleY	Default value of scale command in Y direction
schemIncr	Schematic increment is a multiplier against the precision. An increment of 2 and precision of -3 would make the conversion factor $.001 * 2 = .002$. In practice, an increment other than 1 is rarely used. Note: This preference is not used when a design is opened.
schemPrec	Schematic precision is an integer representing the conversion of user units to data base units. It is the power of 10 used to multiply against a data base unit to get a user unit. e.g., prec -3 = 10^{-3} or .001 If db unit = 1000, then $1000 * .001 = 1$ user unit Note: This preference is not used when a design is opened.
schemUnits	Schematic unit where 1=mil 2=in 3=um 4=mm 5=cm 6=meter 7=ft Note: This preference is not used when a design is opened.
selectBoxSize	Size of cursor select pick box
selectBoxUnits	0=User units 1=Screen pixels
selectColor	Color of selected objects represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
selectFilter	Bitwise value of selectable items 0=None

Advanced Design System 2008

	<p>1=Components 2=Wires 4=Polygons 8=Polylines 16=Paths 32=Text 64=Arcs 128=Circles 256=Port Components 512=Drawing Formats 1024=Vertices 8192=Traces</p>
selectMode	<p>Select mode for polygons 0=By edge 1=Inside</p>
selectPointSize	<p>Size of selected vertex</p>
selectPointUnits	<p>0=User units 1=Screen pixels</p>
setPasteOriginPopup	<p>Show Set Paste Origin dialog for Copy command 0=Off 1=On</p>
showConnectedLay	<p>Show Connected Components in Layout 0=Off 1=On</p>
showConnectedSchem	<p>Show Connected Components in Schematic 0=Off 1=On</p>
showFixedLay	<p>Show Fixed Components in Layout 0=Off 1=On</p>
showFixedSchem	<p>Show Fixed Components in Schematic 0=Off 1=On</p>
stepRepeatConnect	<p>Automatically connect the pins of Step And Repeat items with one another 0=Off 1=On</p>
stepRepeatNumCols	<p>Number of columns for Step And Repeat command</p>
stepRepeatNumRows	<p>Number of rows for Step And Repeat command</p>
stepRepeatXspace	<p>X spacing between columns in user units for Step and Repeat command</p>
stepRepeatYspace	<p>Y spacing between rows in user units for Step and</p>

Advanced Design System 2008

	Repeat command
swapKeepInstName	Default value for keeping the original instance name for Swap Components command 0=Off 1=On
tapLength	Length of tee element for Tap Transmission Line command in user units
teeColor	Color of tees represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
teeSize	Size of tees
teeSizeUnits	0=User units 1=Screen pixels
textAbsolute	0=text will rotate when in hierarchy 1=text will not rotate when in hierarchy
textAngle	Angle of text placed in design in degrees
textFontName	Text placed in design will use the font given here
textJustification	Justification for text placed in design 9=Left Bottom (default) 10=Left Middle 12=Left Top 17=Center Bottom 18=Center Middle 20=Center Top 33=Right Bottom 34=Right Middle 36=Right Top
textOrigin	Display text origin marker (Layout only) 0=Off 1=On
textPoint	Text size in points
toArcRadius	Radius of arc in user units for Vertex to Arc command
traceSimMode	Type to convert trace to 0=Transmission line elements 1=Single Transmission line elements 2=Nodal Connection (short)
traceTech	Element set used during trace conversion 0=Microstrip 1=Strip line 2=Printed circuit board
traceTraverse	Insert Tee and Cross components during trace

Advanced Design System 2008

	conversion 0=Off 1=On
tuneRange	Tune dialog slider range as a percentage plus or minus of the value (Schematic only)
tuneScale	Tune dialog slider scaling 0=Linear 1=Logarithmic
tuneSimMode	Tune analysis mode (Schematic only) 0=Single-after each change 1=Multiple-after one or more changes 2=Continuous-while slider is moving
tuneSnap	Tune dialog slider snap to step 0=Off 1=On
tuneStepSize	Tune dialog slider step size as a percentage of the value (Schematic only)
undoEditCount	Maximum number of commands held in the undo stack
unitsAng	Default scale factor for angle parameters 0=deg 1=rad
unitsCap	Default scale factor for capacitance parameters 0=fF 1=pF 2=nF 3=uF 4=mF 5=F
unitsCond	Default scale factor for conductance parameters 0=pS 1=nS 2=uS 3=mS 4=S
unitsCur	Default scale factor for current parameters 0=fA 1=pA 2=nA 3=uA 4=mA 5=A 6=kA
unitsDist	Default scale factor for distance parameters

Advanced Design System 2008

	<p>0=meter 1=km 2=ft 3=mi 4=nmi</p>
unitsFreq	<p>Default scale factor for frequency parameters 0=Hz 1=kHz 2=MHz 3=GHz 4=THz</p>
unitsInd	<p>Default scale factor for inductance parameters 0=fH 1=pH 2=nH 3=uH 4=mH 5=H</p>
unitsLng	<p>Default scale factor for length parameters 0=um 1=mm 2=cm 3=meter 4=mil 5=in 6=ft</p>
unitsPower	<p>Default scale factor for power parameters 0=pW 1=nW 2=uW 3=mW 4=W 5=kW 6=dBm 7=dBW</p>
unitsRes	<p>Default scale factor for resistance parameters 0=mOhm 1=Ohm 2=kOhm 3=MOhm 4=GOhm 5=TOhm</p>
unitsTime	<p>Default scale factor for time parameters 0=fsec 1=psec 2=nsec</p>

	3=usec 4=msec 5=sec
unitsVolt	Default scale factor for voltage parameters 0=fV 1=pV 2=nV 3=uV 4=mV 5=V 6=kV
voltageColor	Color of node voltage represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
windowLLX	Default X coordinate of lower left corner of design area in user units
windowLLY	Default Y coordinate of lower left corner of design area in user units
windowURX	Default X coordinate of upper right corner of design area in user units
windowURY	Default Y coordinate of upper right corner of design area in user units
wireLabelColor	Color of wire labels represented as an index into the eecolorRGB.cfg. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg
wireLabelFontName	Wire label will use the font given here
wireLabelPoint	Wire label size in points

Specifying Layer Definitions

All shapes and text are entered on layers. By placing various groups of items on different layers, each of which may be assigned different characteristics, you can customize and easily alter the overall visual effect of the design in the viewing area. To access the Layer Editor, select Edit from the Layers window or select Options > Layers. The Layer Editor enables you to:

- Change display characteristics on a layer-by-layer basis
- Define new layers
- Change the plotting priority of layers (Layout window)
- Turn the display of items on and off on any specified layer. For example, you can turn on and off the display of any or all the parts making up component text-Name (labels), ID (identifiers), Parameters (parameters).
- Protect items on any given layer from being selected, that is, when clicking in the drawing area to select items,

those items on a protected layer cannot be selected as you edit your design. This is helpful when you have a lot of editing to do on certain kinds of items, but not others, in a crowded design.

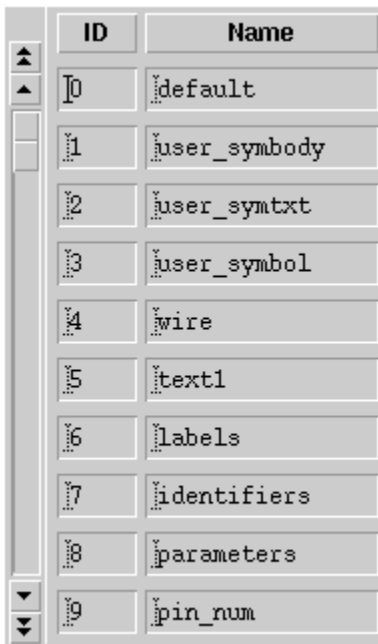
- Save the layer definitions to the default schematic.lay file or to a unique filename.lay file
At any time during the modification of layer definitions, if you have not yet clicked Apply, you can click Reset to return to the previous settings.

Note
The layer number-not the name-is the common identifier used in both the design file and the layer file, and should not be changed.

The order in which layers are plotted is determined differently in the Schematic window than it is in the Layout window.

- In the Schematic window, layers are plotted in numerical sequence-based on the layer number-beginning with zero and ending with the highest number. You can rearrange layers in the list for your convenience, but it will have no effect on the plotting order.
- In the Layout window, the position of the layers in the list, relative to each other, indicates their priority when plotted. Higher priority layers are plotted on top of lower priority layers-the lower in the list, the higher the priority.

The following figure shows the list of default defined layers. These are the default layers in the Schematic window; for details on layers in the Layout window, refer to [Defining Layers](#).



ID	Name
0	default
1	user_symbody
2	user_symtxt
3	user_symbol
4	wire
5	text1
6	labels
7	identifiers
8	parameters
9	pin_num

Important
Be sure to use compatible layer definitions for related designs, especially designs that are related hierarchically.

To move a layer within the layer list:

1. Click once to highlight the layer ID or Name.
2. Click Cut. The name and number disappear from the list.
3. Highlight the layer you want to paste the cut layer above, click Paste and click Apply.

To delete a layer:

Note
Do not delete the supplied, default layer definitions; the ability to delete is provided to enable you to redefine layer definitions you have created. Deleting supplied layer definitions will degrade the appearance of your schematic, as parts whose layer has been deleted will now be drawn on the default layer.

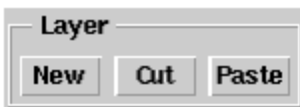
1. Click once to highlight the layer ID or Name.
2. Click Cut. The name and number are deleted from the list, and that number is now available for a new layer.
(Hint: If you change your mind, use Paste to add the layer back.)

To change a layer's name:

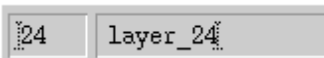
1. Click in the Name field and change the name as desired.
2. Click Apply.

To add a layer:

1. Click New from the group of buttons labeled Layer.



The layer list scrolls to the bottom and a new layer is added. The layer number is the next available, sequential number, and a default name appears that includes the layer number.



2. Rename the layer as desired and click Apply.

Setting Colors and Fill Patterns

The Color/Pattern columns of the Layer Editor dialog box enable you to choose colors and fill patterns of items, on a layer-by-layer basis.



To change color and/or pattern settings for any given layer:

1. Click to access the available colors or patterns (PC-click the arrow for the drop-down list; UNIX-click the color bar itself) from the color or pattern box of the layer you want to modify.
2. Click the desired color or pattern (UNIX-click OK).
3. The new color is displayed in the Layer Editor dialog box. To see the change in the drawing area, click Apply.

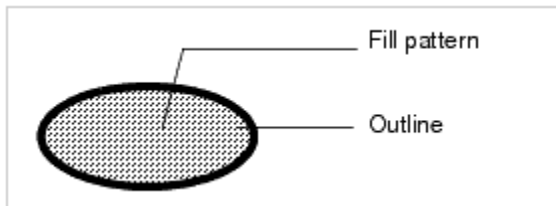
Note
When choosing Fill Patterns, keep in mind that the patterns produced by a Postscript printer are Postscript fill patterns and will vary somewhat from those on your screen.

Setting Shape Display Characteristics Layer-by-Layer

All shapes are drawn with one of the following display types:

- Both (both an outline and a fill pattern)
- Outline (an outline with no fill pattern)
- Filled (a fill pattern with no outline)

The following illustration shows the shape display Both, where you see both the fill pattern and the outline.



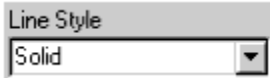
The Shape Display drop-down list for each layer enables you to choose the shape display of items, on a layer-by-layer basis.




Hint
For details on changing the shape display globally for all layers, refer to [Setting Layer Characteristics Globally](#).

Setting Line Style Characteristics Layer-by-Layer

When you include the Outline of shapes as part of Shape Display on any given layer, you can also choose a line style for that outline. Click to select a different line style from the drop-down list.




 Hint
For details on changing the line style globally for all layers, refer to [Setting Layer Characteristics Globally](#).

Setting the Visibility of Items Layer-by-Layer

The Vis (visible) column enables you to turn on and off the display of items on any given layer. For example, if you want to print or plot your schematic without component annotation, you could make the labels, identifiers, and parameters layers invisible. By default, all layers are visible.




 Hint
For details on changing the visibility globally for all layers, refer to [Setting Layer Characteristics Globally](#).

Setting the Selection Status of Items Layer-by-Layer

If you need to edit certain types of items (that reside on a given layer) and not others (that reside on other layers), and the selection filters do not meet your needs, you can prevent items from being selected by disabling the Sel (select) option for any layers as needed.

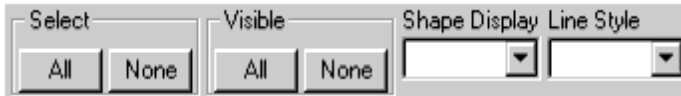


When you disable the select status for any given layer, items on that layer will not be selected as you edit your design.

 Hint
For details on changing the shape display globally for all layers, refer to [Setting Layer Characteristics Globally](#).

Setting Layer Characteristics Globally

This section of the Layer Editor dialog box enables you to change the Selection, Visibility, Shape Display, and Line Style status of all layers at once. For example, if you want to prevent selection on all layers except one or two of them, use Select None, then select the individual layers you want access to, and turn on the Select status for those layers.



Miscellaneous Layer Editor Features

- The Ins (insert) column enables you to change the current entry layer while working in the Layer Editor dialog box so that you can quickly see the effect of your changes.
- The Reverse button toggles the display of the layer list top-to-bottom, or vice versa.
- The Visibility tab enables you to reduce the size of the Layer Editor dialog box while keeping the most commonly used features of it available for editing.

Saving and Reading Layer Files

The default layer definitions are contained in the files `schematic.lay` (for schematics) and `layout.lay` (for layouts). These files are read automatically—from the installation directory—every time you open a design, unless you have explicitly associated another layer file with that design. To associate a customized layer file with a specific design, open the design, read the customized layer file, and save the design. Whenever you open a design, the last layer file associated with it is automatically read.


You can create any number of files containing customized layer definitions, and subsequently read in any of these files for any design.

To save customized layer files:

1. Make all the desired changes in the Layer Editor dialog box and click Save. The Save Layer File dialog box appears, displaying the default filename, `schematic.lay`.
2. To use the default filename for the current settings, click OK.
For another layer file, enter a name of your choosing (the `.lay` extension is added automatically) and click OK.

To read in a previously saved layer file:

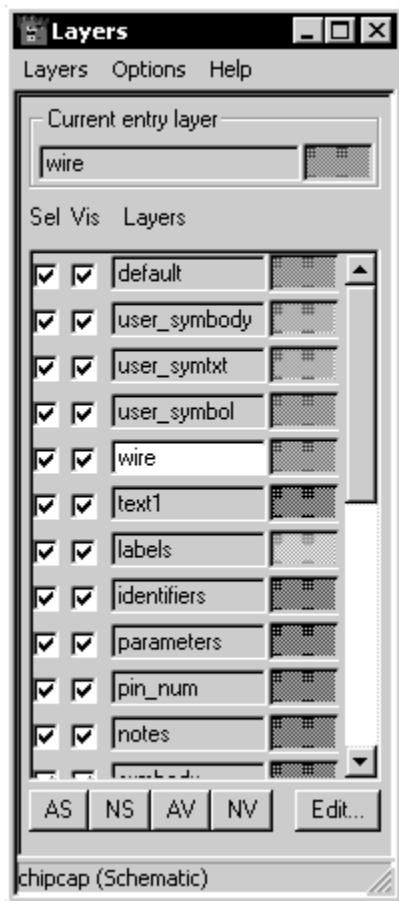
1. Click Read from the Layer Editor dialog box. The Read Layer File dialog box appears.
2. Select the desired file from the list of files, and click OK. (You can read in layer files from other project directories.)

 Hint
The variable that defines the search path for these files is [LAYERS_PATH](#).

Changing the Current Entry Layer

To change layers:

1. Choose Insert > Entry Layer and the Layers window appears listing all the currently defined layers.



2. Click a layer that is appropriate for the task at hand.
3. Click OK. Anything you draw now is drawn on this layer. The name of the layer is displayed in the status panel of the Schematic window. See [Using the Layers Window](#) for more information.

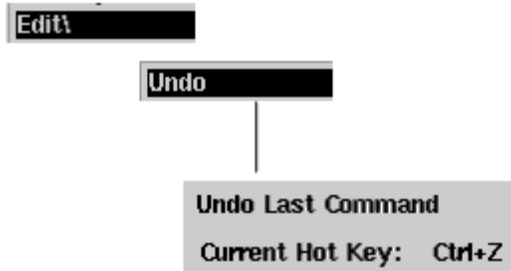
Customizing Keyboard Shortcuts

You can redefine default keyboard shortcuts as well as create new ones. These shortcuts are maintained individually

for the different ADS windows.

To change or add a keyboard shortcut:

1. Choose Tools > Hot Key/Toolbar Configuration and click the Hot Key tab in the dialog box that appears.
2. Select the menu name or menu/command sequence from the Category list box.
3. Select the command from the Item list box. If a shortcut currently exists for the item, the current assignment is displayed.



4. Select the modifier key(s)-Ctrl, Alt, Shift-and type the letter(s) you want to use in the Key field (UNIX is case-sensitive; the PC is not). If the combination you choose is currently assigned to another command sequence, you are warned and given the choice to proceed or to select another key sequence.

Note
If you use Alt as the modifier key, and a letter that is already assigned as an accelerator for a menu (see the underscored letters on the menu bar), the menu accelerator is replaced by your custom shortcut (with no warning).

5. To replace the assignment with your own choice, continue, otherwise choose a new key combination and click Apply. When you are through making all keyboard changes, click OK to dismiss the dialog box.

Configuring Toolbars

By default, the toolbar in each design window contains:

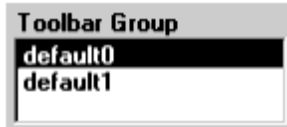
- A drop-down list for selecting the group of components you want to place on the palette (Palette List)
- A dynamically updated list of components you have placed in that window (Component History)
- In the Layout window, a drop-down list for selecting a different entry layer (Entry Layer List)
- A button for choosing the orientation of the component you are about to place
- Buttons representing frequently used commands

You can reconfigure these default toolbars and create your own to better meet your design needs (Tools > Hot Key/Toolbar Configuration).

Customizing an Existing Toolbar

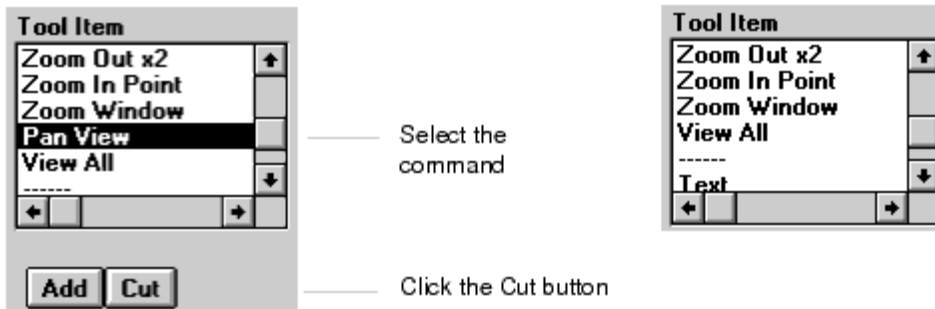
To reconfigure an existing toolbar:

1. Choose Tools > Hot Key/Toolbar Configuration and click the Toolbar tab in the dialog box that appears.
2. In the Toolbar Group list box, select the name of the toolbar you want to change.

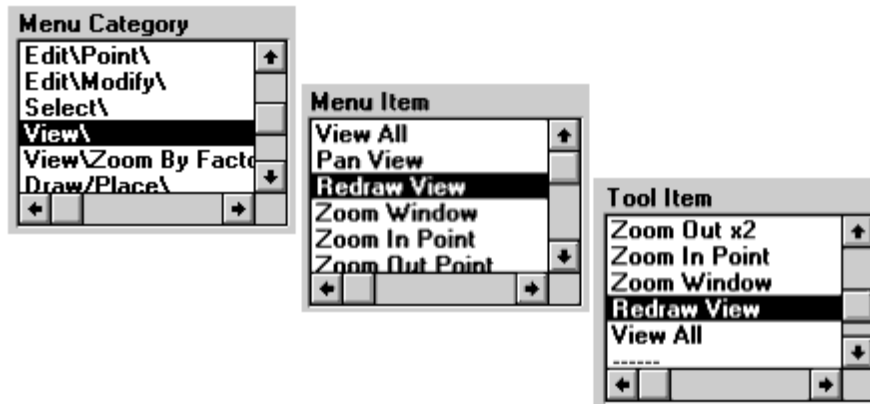


3. To display the selected toolbar, select the Display Toolbar check box. Unselecting the Display Toolbar check box will hide the toolbar.
4. To change the display order, click the Up or Down buttons to move the selected toolbar.
5. To add or delete icons, use one of the following methods:

- To delete an icon from the toolbar, select the associated command in the Tool Item list box, and click the Cut button.




- To add an icon to the toolbar, select the appropriate menu/command sequence from the Menu Category list box, select the command from the Menu Item list box, and click the Add button. The command is added to the Tool Item list box.



When you select a command, its default bitmap is displayed




 **Hint**
 When you add a button to the toolbar, its position relative to the other buttons is determined by its position in the Tool Item list box. Before you click the Add button, be sure to highlight the command that the new command should follow. In this example, Zoom In Point was highlighted before the Add button was clicked so that Redraw View would take the place of Pan View (deleted in the previous example).

6. If you want to edit another toolbar, click Apply to effect these changes and begin the process again. When you are through making changes to the toolbars, click OK.

Creating a New Toolbar

To create a custom toolbar:

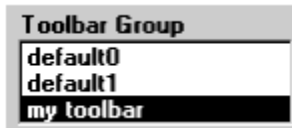
1. Choose Tools > Hot Key/Toolbar Configuration and click the Toolbar tab in the dialog box that appears.

 **Hint**
The position of the new toolbar, relative to the position of any existing toolbars, is determined by its position in the Toolbar Group list box; the new name is added below the name that is highlighted when you click the Add button. For example, if you keep both default toolbars and want to add a third one below them, highlight the bottom one before you click the Add button.

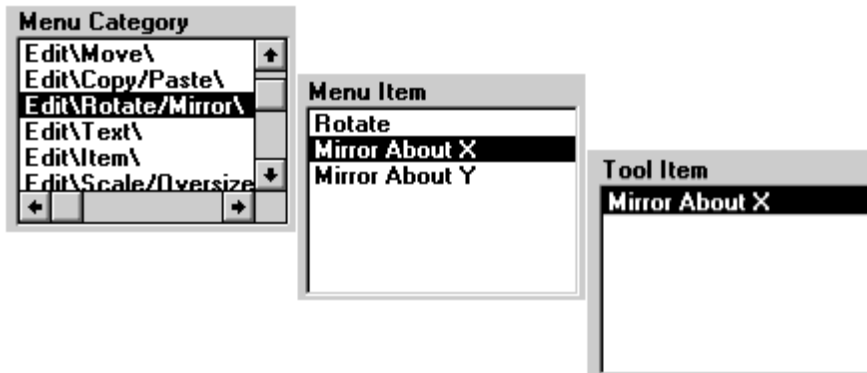
2. Supply a name in the Toolbar Name field and click Add.



The name you supply is added to the Toolbar Group list box.



3. Select the desired Menu Category, select the desired Menu Item and click Add. The command name is added to the Tool Item list box.




4. When you are through making changes to this toolbar, click Apply to effect the changes.
5. When you are through making changes to all toolbars, click OK.

Creating a Custom Component Palette

Creating a custom component palette can speed up the design creation process by grouping frequently used items in one or more palettes.

To create a custom palette:

1. Choose Tools > Component Palette Configuration to display the Create Component Palette dialog box.

 **Hint**
By default, the components currently listed in Component History are listed for inclusion in the new palette. If this list does not represent a significant number of components you want to include on the new palette, use the Clear Component History command (View > Component > Clear Component History) to avoid individually selecting and cutting components from the custom list.

2. Cut any components from the New Palette Group Components list box, as necessary.
3. Select a palette, from the List of Palette Groups, that contains components you want in the custom palette.
4. Select the desired component, from the Palette Group Components list box, and click Add.
5. Repeat as needed to include additional components from any palette.
6. Provide a name for the new palette in the New Palette Group Description field then click OK. This is the name that will appear in the drop-down Palette List enabling you to place the custom group on the palette.

Customizing a Component

Custom component palettes are defined by the `de_define_palette_group()` function in the `userpalette.ael` file located in your `$HOME\hpeesof\de\ael` directory.

To customize a component in the palette, for example to change the name of a component label, edit the `userpalette.ael` file. Refer to the `de_define_palette_group` function.

Adding Custom Symbols

To add your own custom symbols to a custom component palette:

1. Create a directory named `bitmaps` under the `$HOME\hpeesof\circuit` directory.
2. Create a bitmap image of the symbol and save it as `BMP_SymbolName.bmp` in the `bitmaps` directory, where `SymbolName` is the unique name of your symbol.
3. Edit the `userpalette.ael` file. Refer to the `de_define_palette_group` function.

Deleting Custom Component Palettes

You can either delete all or individual custom component palettes.

- To delete all custom component palettes, delete the userpalette.ael file located in your \$HOME\hpeesof\de\ael directory.
- To delete an individual custom component palette, open the userpalette.ael file located in your \$HOME\hpeesof\de\ael directory, then delete the de_define_palette_group() function that defines the palette.

Turning On/Off the Coordinate Readout Display

There are two types of coordinate readouts, positional and differential.

- Positional-the X,Y coordinates of the cursor position in relation to the total window. By default, the lower left corner is 0,0. This display also reflects the current precision setting (in this example, 1,000).
- Differential-the distance in X,Y the cursor has traveled since the last click. Set the starting point to 0,0 by clicking the left mouse button anywhere in the drawing area.

The X,Y coordinate readouts are displayed in the status bar of each window. By default, Positional readouts are turned on in every window.



You can turn the coordinate readout on or off from the View menu.

Setting Layout Options

This chapter provides details for setting layout options so that you can create a layout in an environment that is compatible with your design. Layout is shipped with preset options that can be modified on a project- or system-wide basis. Before you begin a layout, be sure that the environment is appropriate for the design, the program options, and the final output required.

Defining Layers

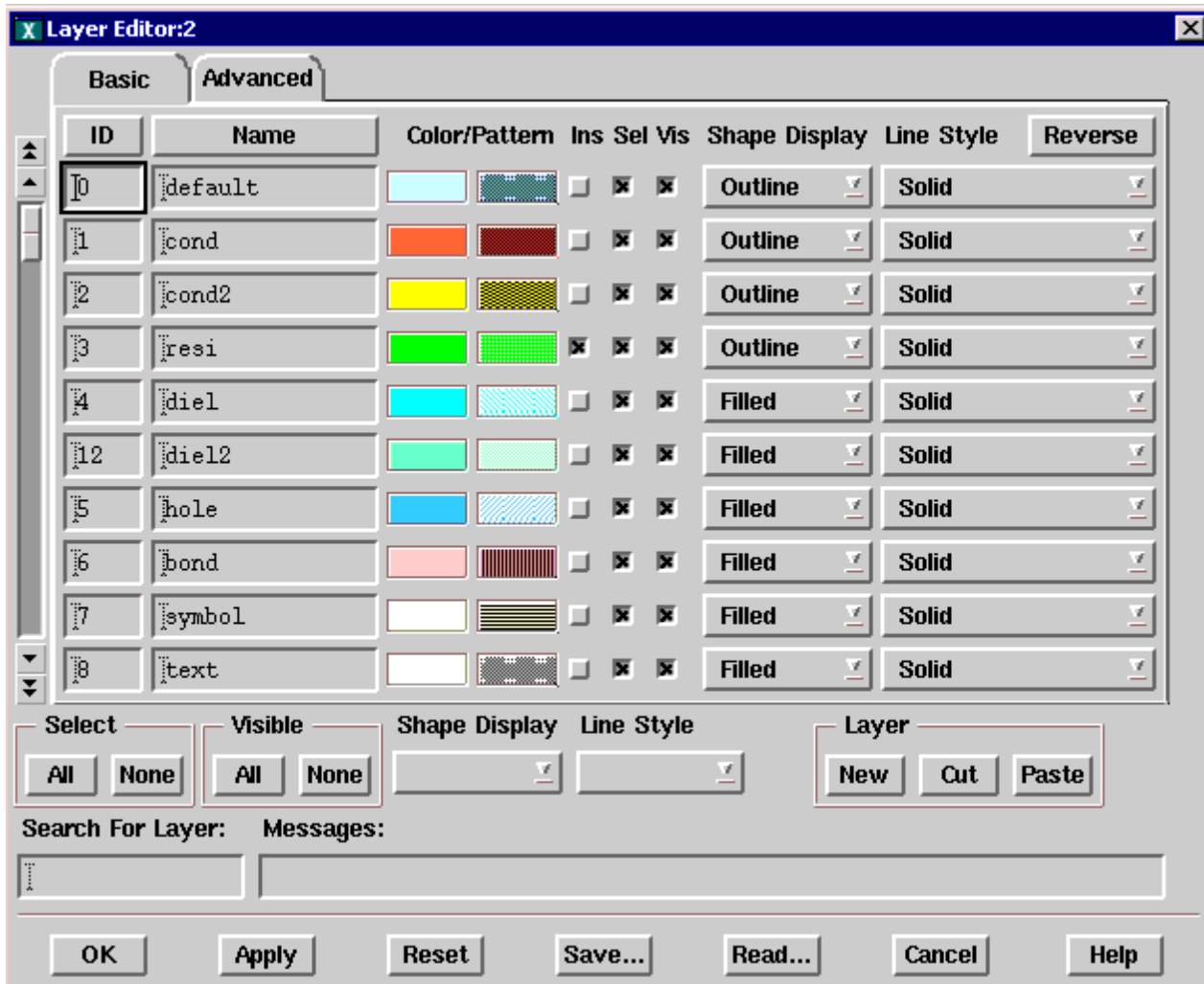
All shapes and text are entered on layers. Layout allows you to define any number of mask layers in a file, and to create any number of mask layer files. Each layer must have a unique name and number. Mask layers usually

correspond to the masks used in manufacturing a layout. However, mask layers can be used for a number of other purposes.

Often a single process mask layer is represented by a number of layers in a CAD program. For example, power and ground lines are often placed on the same mask for manufacturing, but on a different mask for CAD layout to distinguish them. Further, simple text notes and annotation can be placed on layers that are not output for manufacturing at all.

The program provides a default set of layer definitions, but you can define and save your own layer sets. (See [Saving a Layout Setup](#).) Using a standard set of layer definitions saves time because you do not have to define the layers each time you create a design. Note that it is important that hierarchically related designs use the same layer definitions. (See [Using an Existing Layout Setup](#).)

Layer definitions are modified through the Layer Editor. To display the Layer Editor, choose Options > Layers or select Edit from the Layout window.



Setting Layer Characteristics Globally

You can set attributes layer-by-layer or you can use Global Attributes Control to set the following attributes on all layers at once:

- Protection (against selection) of items on layers
- Visibility of items on layers
- How shapes are displayed
- The style of lines used

The buttons in this area of the Layer Editor dialog enable you to change at once the layer protection and visibility status, how shapes are displayed, and the line style used for all layers. This can be easier than making the same change layer-by-layer. It can also be faster to set an attribute the same for all layers, and then individually change that attribute on the few layers that are an exception. If you want to prevent selection on all layers except one or two of them, use Select None, then select the individual layers you want access to, and turn on the Select status for those layers.

Miscellaneous Layer Editor Features

- The Ins (insert) column enables you to change the current entry layer while working in the Layer Editor dialog box so that you can quickly see the effect of your changes.
- The Reverse button toggles the display of the layer list top-to-bottom, or vice versa.
- The Visibility tab enables you to reduce the size of the Layer Editor dialog box while keeping the most commonly used features of it available for editing.

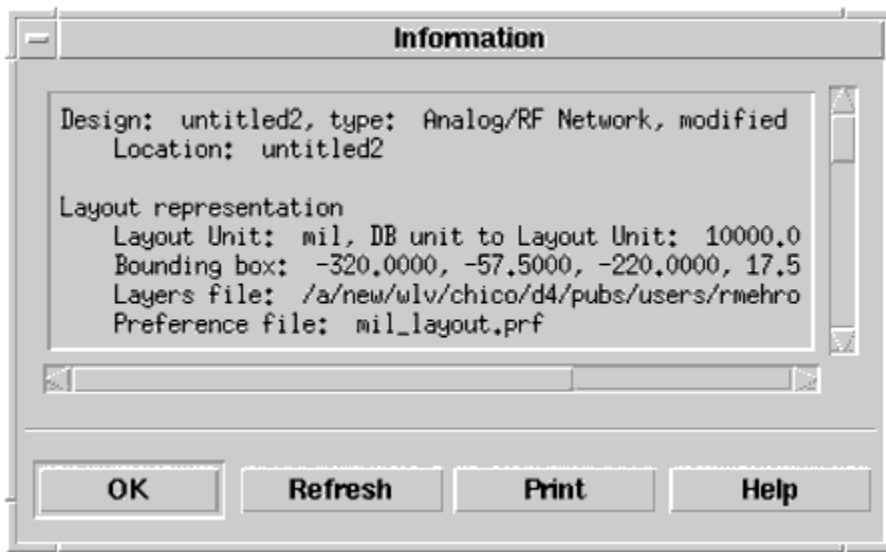
The Default Layer

Every layer set contains a default layer. If you end up with a shape whose layer number does not exist in the current layer set, the attributes of the default layer are assigned to the shape. For example, if you create a shape on a layer and then you delete that layer definition from the Layers list, the shape appears with the characteristics of the default layer. You cannot delete or change the name or number of the default layer, but you can modify all other attributes.

Determining the Layer for an Item

To determine the layer for a specific item:

- Select the item and choose Tools > Info.



Changing Layer Priority

Layer priority is determined by a layer's position in the layer list. In the Layout window, layers are drawn from lowest priority (at the top of the list) to highest priority (at the bottom of the list); higher priority layers are drawn on top of lower priority layers.

To change a layer's priority, change the position of that layer in the list:


1. In the Layers list, choose a layer name or ID.
2. Click Cut. The name and number are deleted from the Layers list.
3. Highlight the layer name that will follow the moved layer.
4. Click Paste to add the layer above the highlighted layer.

To add a layer:

1. Click New.
2. The layer list scrolls to the bottom and a new layer is added. The layer number is the next available, sequential number, and a default name appears that includes the layer number.
3. Rename the layer, if you want, and click Apply.

To delete a layer:

1. In the Layers list area, select a layer name or ID.
2. Click Cut. The name and number are deleted from the Layers list.

 **Note**
Do not delete the default layer definitions. The ability to delete is provided to enable you to redefine layer definitions you have created.

Changing the Entry Layer

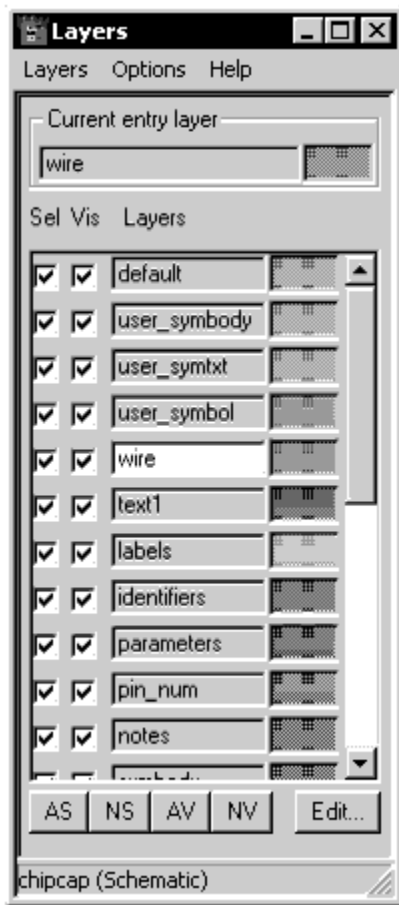
The name of the current entry layer is displayed in the status panel at the bottom of the window, in the toolbar at the top of the window, and in the Entry Layer dialog. Anything you draw is drawn on the layer you set.

To set the current entry layer, choose one:

- Choose Insert > Entry Layer. Select a layer from the entry layer list. See [Using the Layers Window](#) for more information.
- Choose Options > Layers. Select the layer from the layer editor list.
- Choose Insert > Change Entry Layer To. Click the object whose layer you wish to make current.
- Click the entry layer drop-down in the toolbar. Choose the layer from the list.

Using the Layers Window

The Layers window opens automatically when a Layout window is opened, but can also be accessed by selecting Insert > Entry Layer. The Layers window enables you to choose the design layer for each shape, set the visibility or selectability of any layer, and access the Layer Editor dialog.



To access the Layer Editor dialog, select Layers > Edit Layers... or click the Edit... button near the bottom of the window. See [Defining Layers](#) for information on the Layer Editor dialog.

The Options menu enables you to select the following:

- Open With Layout Window—If selected, the Layers window always opens with the layout window unless the Layers window is already open. This menu item is selected by default.
- Open With Schematic Window—If selected, the Layers window always opens with a schematic window unless the Layers window is already open. This menu item is deselected by default.
- Auto Redraw Design Window—If selected, refreshes the current layout or schematic window each time the visibility or selectability of any layer is changed within the Layers window. This menu item is selected by default.

The Current entry layer field displays the name, fill pattern, and color of the current design entry layer. In addition, the current design entry layer is highlighted in the Layers list. To choose a current entry layer, click a layer's name or fill pattern from the Layers list.

The Layers list displays the selectability status, visibility status, name, fill pattern, and color for each available layer in the current design. Select the check box in the layer's Sel column to enable editing. To protect a layer from unintentional editing, deselect its Sel check box. Select the check box in the layer's Vis column to display the layer. To hide the layer, deselect its Vis check box.

The four buttons below the Layers list enable you to set the selectability and visibility for all current design layers.

- Select the AS (All Selectable) button to set all current design layers to visible and selectable.
- Select the NS (Not Selectable) button to set all current design layers, except the default layer, to nonselectable.
- Select the AV (All Visible) button to set all current design layers to visible.
- Select the NV (Not Visible) button to set all current design layers, except the default layer, to nonselectable and invisible.

Defining Port Connections (Layer Binding)

Layer Binding enables you to control which layers will make a connection to other layers when pins and shapes overlap. Layer Binding is found under the Advanced tab of the Layer Editor dialog box. For ADS versions before 2004A, only pins could make connections. Starting with ADS 2004A, shapes (polygons, circles, paths, and rectangles) can also make connections between components. Also, components can have area and edge pins in addition to their point pins. Because of this, it is important to have your layer binding specified correctly.

In a design process, interconnects are made on metal layers or by holes in a via layer. To represent this in ADS, the Layer Binding should have values on metal or via layers and other layers should have empty Layer Bindings.

The layer binding field is a list of words separated by spaces. The words typically are layer names, but this is not required. When pins or shapes overlap but are on two different layers, they will connect if a word in the layer binding list of the first layer matches the same word in the layer binding list for the second layer.

A layer binding of "*" will match any word or "*" in another pin's layer binding list. Agilent Technologies recommends that you not use "*" because it can cause layers to be connected when you did not expect a connection. Connections with shapes will ignore the layer binding "*" and treat the Layer Binding as if it were empty.

Examples

Layer Name	Layer Binding List
Metal1	Metal1
Metal2	Metal2

Since neither layer has the same word in their layer binding list, they will not connect to each other.

Layer Name	Layer Binding List
Metal1	Metal1
Metal2	Metal1

Advanced Design System 2008

Since both layers have the same word (Metal1) in their layer binding list, overlapping pins and shapes on these layers will connect to each other.

Layer Name	Layer Binding List
Metal1	Metal1 Metal3
Metal2	Metal2
Metal3	Metal3 Metal1

The pins and shapes on layers Metal1 and Metal3 will connect, but those on the Metal2 layer will not connect.

Layer Name	Layer Binding List
Metal1	Metal1
Via1	Via1 Metal1 Metal2
Metal2	Metal2
Via2	Via2 Metal2 Metal3
Metal3	Metal3

This is a typical way to setup metal and via layers. Pins and shapes will connect from Metal1 to Metal2 if they overlay shapes on the Via1 layer. The same situation occurs for pins and shapes on Metal2 and Metal3 layers.

Layer Name	Layer Binding List
Metal1	conductors
Metal2	conductors
Metal3	conductors

The three layers have a common word, conductors, in their list and each will connect to the others. Using "conductors" this way enables you to refer to a group of layers that will all connect to each other.

Changing the Visibility and Protected Status of Items

Visible toggles the display of items on the selected layer. To display items on a given layer, enable the Vis option for that layer. By default, visibility is turned on for all layers, making all items visible.

Selected toggles the protection status of items on the selected layer. To protect a layer so that you can prevent items from being selected, disable the Sel option for that layer. This can be useful if you need to edit certain types of items, but not others, in a crowded design. By default, protection is turned off for all layers, making all items available for selection.

Using IGES and GDSII Numbers

These numbers do not have to be unique. The IGES number is used to set the IGES level number. The GDSII layer number is used for both reading and writing GDSII stream files; it must be a number between 0-255.

Using DXF Names

Use these names to map ADS layers to DXF layer names when exporting DXF hierarchical files. ADS layer numbers are mapped to DXF layer names. From the DXF layer name column in the Advanced tab in the layer editor, users can modify layer mappings and the DXF layer names.

Assigning Layers for Transmission Line Components

By default, transmission line components are placed on the cond layer, but you can specify different layers for multi-layer designs.

To assign the layer for transmission line components:

1. Double-click the appropriate substrate item (to open the Component Parameters dialog box):
 - For microstrip elements, this is the referenced MSUB.
 - For Stripline elements, use SSUB (Stripline Substrate).
 - For Suspended Substrate elements, use SSSUB (Suspended Substrate).
2. Select any of the following layer parameters to be re-mapped:

```
Cond1="cond"  
Cond2="cond2"  
Die1="diel"  
Die2="diel2"  
Hole="hole"  
Res="resi"
```

3. Select a different layer from the list of layers at the right and click Apply.
4. Regenerate the layout. All the microstrip components will be placed on the new layer.


Layer Files and Library Components

Library components rely on certain layers being defined. For example, by default, all top-level metallization for components in microstrip and stripline libraries is placed on layer 1 (cond). In general, layers 1-12 should be defined so that the layout libraries function properly. If a layer is missing, the artwork for an element can not be created.

If you remove layer 1 cond, you must change the element's corresponding substrate element to specify a different layer number. For example, if you remove layer 1, artwork for microstrip elements will not be generated unless you change the cond parameter of the MSUB_DEFAULT item referenced by these elements. Also, the packaged parts library uses a number of layers to display part-packaged outlines, leads and other information. If you plan to use this library, be sure to include layer definitions compatible with the defaults.

Layer Files and Design Files

Every design has an associated layer file. When you create a design, the program automatically looks for a layer file called layout.lay. By default, the program looks in the current project directory first. If it does not exist there, it looks for it in the directory HPEESOF_DIR\lib\ <program_name>\defaults.

 **Note**
The variable that defines this search order is LAYERS_PATH. For additional details, refer to [LAYERS_PATH](#).

All text and shapes are entered on layers, and each layer has a number assigned to it. The layer number for each part of a design is stored in the design file, but the attributes comprising the actual layer definition are stored in the layer file. For example, if your design contains a polygon, the program notes the layer number for the polygon in the design file and searches the current layer file for a matching layer number. It can then display the polygon with the appropriate color, fill pattern, etc.

Ensuring Compatible Layer Definitions

Always use the same (or compatible) layer definitions for related designs. This is especially important for designs that are related hierarchically. For example, if you use layer 1 for first layer metallization in one design, you should do the same for all related designs. If designs have incompatible layer numbering, you can change the layer number associated with a shape by moving that shape to the appropriate layer using the Edit > Move > Move to Layer command.

If you have made any changes to the layer definitions but have not yet clicked Apply, you can click Reset (in the Layer Editor dialog box) to return the layer definitions to the state they were in before you started making changes.

Different designs can have different layer sets associated with them by reading in different layer files.

Layer File Format

Each line in a layer file defines a layer. There is no limit to the number of layers that can be defined. The layer names and numbers must be unique. Any layer numbered 0, must have the name default. It can have its other characteristics set to any valid value. By default, when a new design is created, it becomes associated with the

Advanced Design System 2008

schematic.lay and layout.lay layer file found on the path. If none are found, the program uses internal defaults.

In the layer file, information for all the fields for each layer need to be on a separate line, separated by spaces or tabs:

```
default number = 0 stream = 1 iges = 1 color = 59 fill = 0 line = 0, 0 0 1, "**",  
type = 1, dxf = "cond"
```

layer_name Name of the layer. It must be unique within this file. The name default has special significance and must have layer number 0.

The string can be any length, but should only contain letters, numbers or the underscore.

layer_num This is the layer number. The number associates a layer's attributes (color, name, fill, etc.) with objects stored in the design. For example, stored with a rectangle is the information that it is on mask layer 4. The program searches the layer file by number to determine how to plot the layer.

All layer numbers are integers between 0 and the largest integer (approximately 2 billion). Layer number 0 is reserved for the default layer (see above). Each layer number in the file must be unique.


gds_num This is the number to use as the GDSII stream layer number when translating a layout to GDSII stream format with the GDSII export option.

This is an integer in the range of 0-255.

iges_num This is used as the IGES level number when exporting a layout to IGES format.

color An index into the eecolorRGB.cfg file that determines the color that an object is drawn. It can be any integer in the range of 0, to the number of colors defined in eecolorRGB.cfg.

fill An index into the hpeefill.cfg file that determines the fill pattern used when the layer plot mode is filled or both. It can be any integer in the range of 0, to the number of fill patterns defined in hpeefill.cfg.

 **Note**
The hpeefill.cfg file contains the names of X bitmap files that determine screen fill, and the HPGL fill pattern numbers when plotting to a HPGL hardcopy device.

line_type An integer representing the line style type. The available line styles are:

- 0 = solid
- 1 = dot
- 2 = double dot
- 3 = short dash

- 4 = short dot dash
- 5 = long dash
- 6 = long dot dash

plot_mode An integer representing how a closed object (circle or polygon) is plotted.

- 0 = outline
- 1 = filled
- 2 = both filled and outline

protect_flag Integer 0 or 1 representing whether a layer is protected or not (nothing can be selected on a protected layer).

- 0 = not protected
- 1 = protected

visible_flag The integer 0 or 1 representing whether a layer is visible or not.

- 0 = not visible (not plotted)
- 1 = visible

layer_binding Names of the layers a port must be on to connect to a port on this layer. An asterisk (*) = connection to any layer.

layer_type An integer representing the layer type.

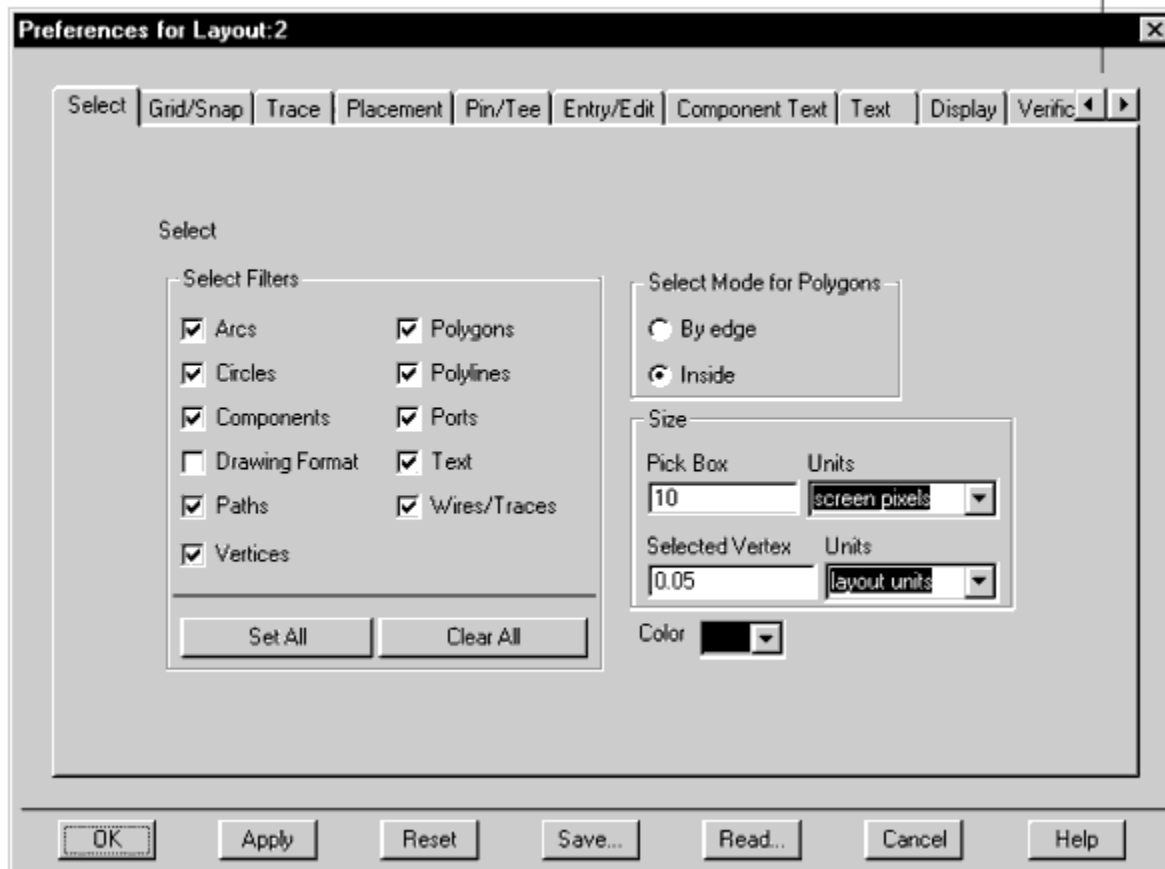
- 1 = Physical
- 2 = Notes
- 4 = DRC

DXF_layer_name Name of the DXF layer.

Preferences for Layout

To access Preferences for Layout, select Options > Preferences.

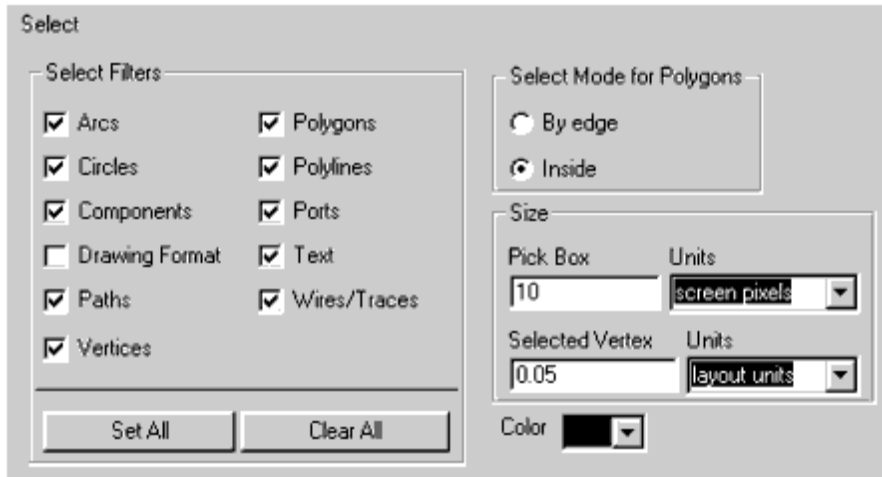
Use these to scroll among the various tabs



Changing Select Options

To change select options, select Options > Preferences > Select.

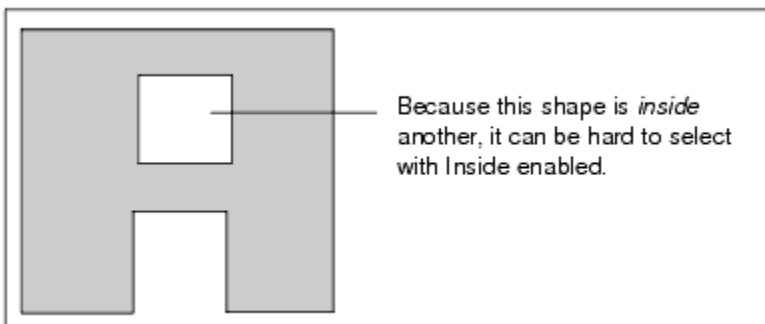
Note
To use the selection filters, see [Using Selection Filters](#).



Setting How Polygons are Selected

You can choose one of two select modes for closed shapes (polygons): clicking inside the shape or clicking near the edge of the shape.

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Select tab.
3. In the Select Mode for Polygons, choose the method:
 - By edge enables you to select a polygon by clicking on its outer edge.
 - Inside enables you to select a polygon by clicking anywhere inside the shape.



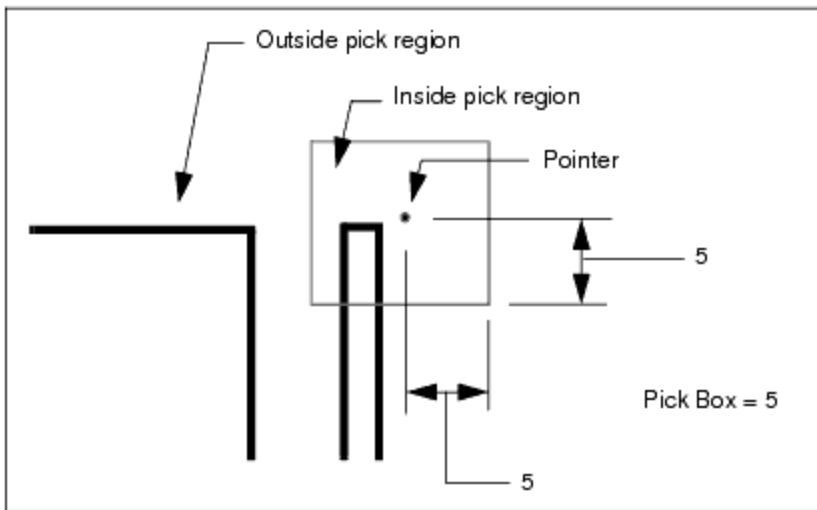
Setting Color for Selected Items

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Select tab.

- Click the colored box next to the word Color and select the color from the displayed palette. This sets the color for:
 - the color of the marker that identifies a selected vertex,
 - the box drawn around items identifying them as being selected.

Setting the Size of the Pick Region

The pick region defines how close the pointer must be to an item to select it.



- Choose the command Options > Preferences
- In the Preferences for Layout Dialog Box, choose the Select tab.
- In the Size area, locate the Pick Box field.
- Enter the size for the marker, and select the units.
 - Screen pixels specifies sizes in terms of pixels on the screen. For example, if you choose 5 screen pixels, an item must be within 5 pixels of the pointer to be selected.
 - Layout Units specifies sizes in terms of the current units of the window. For example, if you are using inches and choose 0.1 layout units, an item must be within 0.1 inch of the pointer to be selected.

Setting the Size of Vertex Markers

A Vertex marker identifies a selected vertex.

- Choose the menu command Options > Preferences.
- In the Preferences for Layout Dialog Box, choose the Select tab.
- In the Size area, locate the Selected Vertex field.
- Enter the size for the marker, and select the units.

- Screen pixels specifies sizes in terms of pixels on the screen. For example, if you choose 5 screen pixels, the size of the marker is 5 pixels.
- Layout Units specifies sizes in terms of the current units of the window. For example, if you are using inches and choose 0.1 layout units, the size of the marker is 0.1 inch.

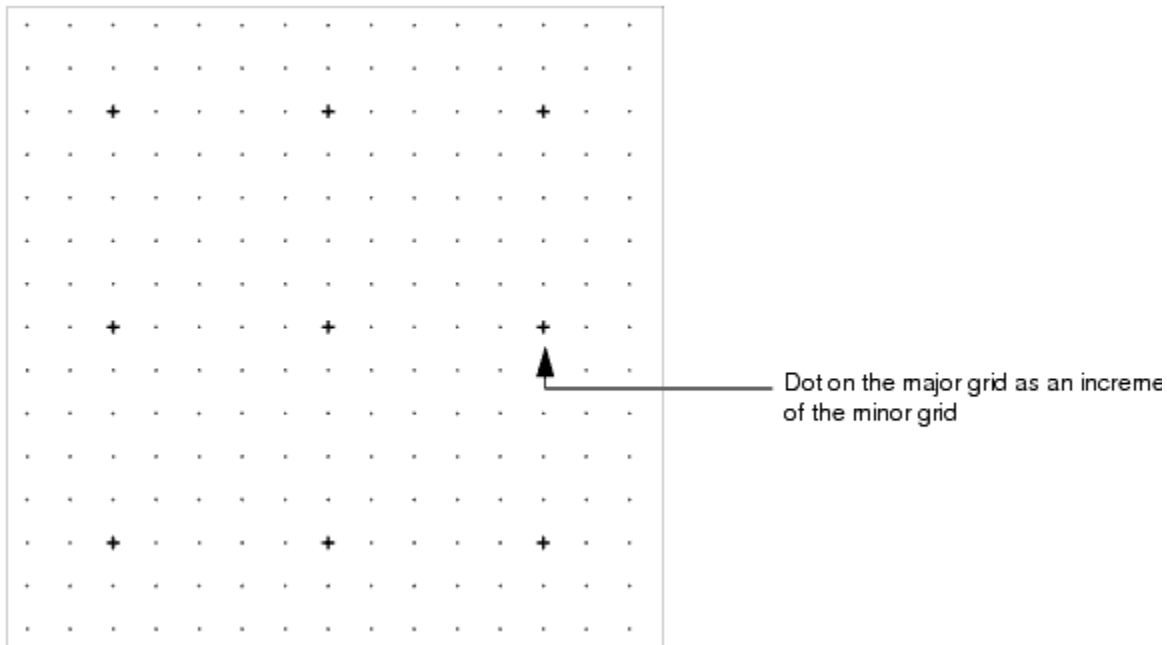
Changing Grid and Snap Settings

To change grid and snap settings, select Options > Preferences > Grid/Snap

The screenshot shows the 'Grid/Snap' preferences dialog box, which is organized into three main sections:

- Display:** Contains checkboxes for 'Minor Grid' (unchecked) and 'Major Grid' (checked). Below these are radio buttons for 'Type', with 'Dots' selected and 'Lines' unselected. At the bottom left is a 'Color' field with a small square icon.
- Spacing:** Contains three sets of input fields for X and Y values:
 - 'Snap Grid Distance (in layout units)': X = 5, Y = 5
 - 'Snap Grid Per Minor Display Grid': X = 2, Y = 2
 - 'Minor Grid Per Major Display Grid': X = 5, Y = 5Below these is a checked checkbox for 'Automatically set Y = X when any X value is modified'. At the bottom is a section for 'Snap Distance - all other modes' with a 'Diameter' field set to 20 and a 'Units' dropdown menu set to 'screen pixe'.
- Active Snap Modes:** Contains a checked checkbox for 'Enable Snap'. Below it is a 'Snap to:' section with a list of modes:
 - Pin
 - Vertex
 - Midpoint
 - Intersection
 - Arc/Circle Center
 - Edge
 - GridAt the bottom of this section are two buttons: 'Set All' and 'Clear All'.

You can establish settings for a snap grid and a display grid to assist you in creating a layout. The display grid appears on the screen as a series of vertical and horizontal lines or dots, but does not print. You use it to ensure exact alignment of pins and vertices as well as provide visual clues to spacing.



Setting Grid Visibility and Color

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Grid/Snap tab.
3. In the Display area, choose Major, Minor, or both.
4. Choose the Type of display (Dots or Lines). You may need to zoom in to see the grid display.
5. Click the colored rectangle next to the word Color, and choose the color for the grid. Click OK to dismiss the color palette.
6. Click Apply.

Setting Snap and Grid Spacing

The ability to display a major grid as an increment of the minor grid enables you to better gauge distances and align objects in layout.

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Grid/Snap tab.
3. In the Spacing area, enter Minor Grid display factors for both X and Y.
The larger the number, the wider the grid spacing.
4. Click Apply.
If the display factor you specify makes the grid too dense to display, it is invisible unless you zoom in. To see the grid without zooming, choose a larger display factor.
5. If the Major Grid requires changing, enter X and Y factors in those fields and click Apply.

Setting Pin/Vertex Snap Distance

Represents how close the cursor must be to a pin of a component or a vertex of a shape before the cursor will snap to it.

A large value makes it easier to place an object on a snap point when you are unsure of the snap point's exact location. A small value makes it easier to select a given snap point that has several other snap points very near it.

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Grid/Snap tab. In the Pin/Vertex Snap area, enter a number for the Diameter of the snap region.
3. Specify the Units.
 - Screen pixels specifies sizes in terms of pixels on the screen. For example, if you choose 15 screen pixels, the diameter of the snap region is 15 pixels.
 - Layout Units specifies sizes in terms of the current units of the window. For example, if you are using inches and choose 0.1 layout units, the diameter of the snap region is 0.1 inch.
4. Click Apply.

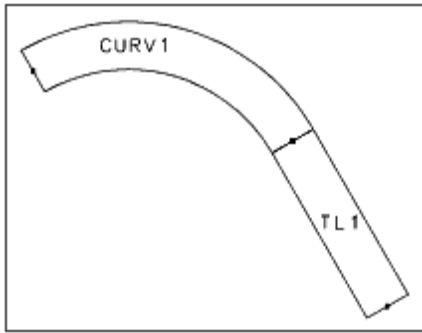
Setting Snap Modes

Snap modes control where the program places objects on the page when you insert, move, or stretch them; you can change snap modes when inserting, moving, or stretching an object, or drawing a shape. When snap is enabled, items are pulled to the snap grid. You can restrict or enhance the manner in which the cursor snaps by choosing any combination of snap modes. The following table lists the snap modes that you can set, and their priorities.

Setting Snap Modes

Snap Mode	Priority
Pin	1
Vertex Midpoint Intersect Arc/Circle Center	2
Edge	3
Grid	4

Angle Snapping automatically occurs when only Pin snapping is enabled and you place a part so that the pin at the cursor connects to an existing part. The placed part rotates so that it properly aligns with the connected part.



For example, if you have a microstrip curve at 30° and place a microstrip line so that it connects to it, the microstrip line will snap to 30° so that it properly abuts the curve.

Enable Snap toggles snap mode on and off. You can toggle snap mode on and off from the Options menu itself, and by default, there are snap mode buttons on the toolbar.

Except for pin snap, the pointer defines the point on the inserted object (the selected location).


When you set all snap modes OFF, you can insert objects exactly where you release them on the page. This is sometimes called raw snap mode. Like other snap modes, the raw snap mode also applies when you move or stretch objects.

Pin When a pin on an object you insert, move, or stretch is within the snap distance of a pin on an existing object, the program inserts the object with its pin connected to the pin of the existing object. Pin snapping takes priority over all other snapping modes.

Vertex When the selected location on an object you insert, move, or stretch is within the snap distance of a vertex on an existing object, the program inserts that object with its selected location on the vertex of the existing object. (Vertex refers to a control point or boundary corner on a primitive, or an intersection of construction lines.)

Midpoint When the selected location on an object you insert, move, or stretch is within the snap distance of the midpoint of an existing object, the program inserts that object with its selected location on the midpoint of the existing object.

Intersection When the selected location on an object you insert, move, or stretch is within the snap distance of the intersection of the edges of two existing objects, the program inserts that object with its selected location on the intersection of the existing objects.

 **Hint**
When working with a layout, if you notice a slowdown in the ADS environment, change your layout preferences to turn off the Snap to Intersection option (Options > Preferences > Grid/Snap). Doing so may help improve ADS responsiveness, especially if the layout or its subcircuits contain a large number of polygons.

Arc/Circle Center When the selected location on an object you insert, move, or stretch is within the snap distance of the center of an existing arc or circle, the program inserts that object with its selected location on the midpoint of the

existing arc or circle.

Edge When the selected location on an object you insert, move, or stretch is within the snap distance of the edge of an existing object, the program inserts that object with its selected location on the edge of the existing object. After a point snaps to an edge, it is captured by that edge, and will slide along the edge unless you move the pointer out of the snap distance.

Because edge snapping has a priority 3, if the cursor comes to within snap distance of anything with a priority 1 or 2 while sliding along an edge, it will snap the selected location to that.

Grid When the selected location on an object you insert, move, or stretch is within the snap distance of a grid point, the program inserts that object with its selected location on the grid point.

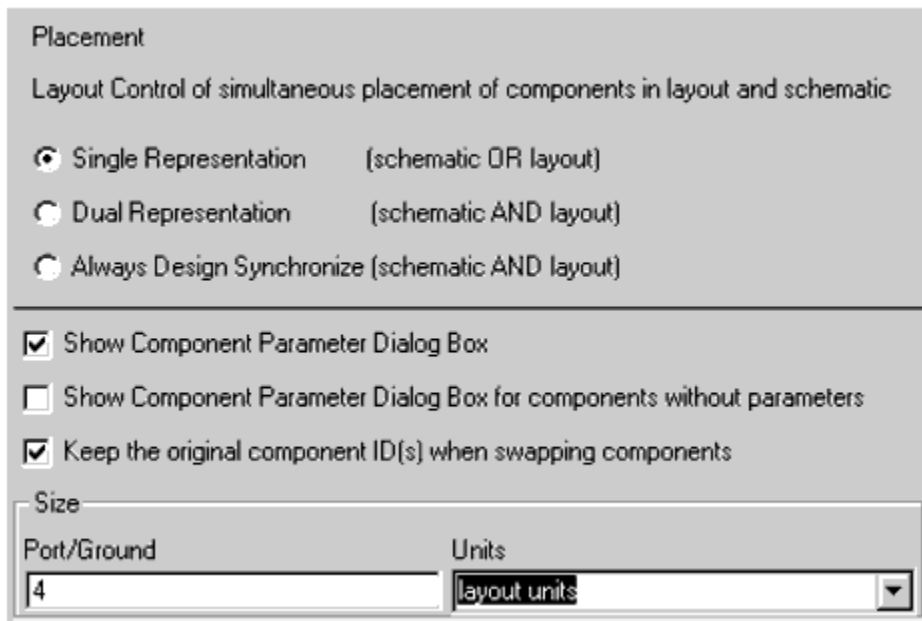
All other snap modes have priority over grid snap mode.

Hint

- Whenever possible, keep grid snapping on. After data is off grid, it is difficult to get it back on.
- Use 45- or 90-degree angles to ensure even alignment of data with less probability of small layout gaps due to round-off errors.
- Keep grid spacing set at increments of a base grid setting. When grid snapping is on, coordinates entered with the mouse are rounded off or snapped to the grid setting.

Selecting Placement Options

To change placement options, select Options > Preferences > Placement.



Single Representation When you place an item in one representation, nothing is automatically placed in the other representation.

Dual Representation When you place an item in one representation and move the pointer into the window for the other representation, the equivalent component is already selected. Position the pointer and click to place it. (If a window for the other representation—containing the same design—is not open, one is opened automatically.)

Always Design Synchronize Causes the program to fully synchronize both representations after each part is placed, ensuring all parts are fully interconnected. This takes more time than the Dual Representation mode and can move or rearrange the layout or the schematic to preserve connectivity.

toggling Display of the Component Parameter Dialog Box

The Component Parameter Dialog box displays the parameters for a selected component. Double-click a component to view this dialog box.

1. Choose the menu command Options > Preferences
2. In the Preferences for Layout dialog box, choose the Placement tab.
3. Toggle the options for the Component Parameter dialog box.
 - Component Parameter Dialog toggles the display of the Component Parameter dialog box. By default, when you click a component, a dialog box appears that displays the component's parameters. If you disable this feature, the dialog box appears only when you choose the command Edit > Item > Edit Component Parameters, or when you click the Edit Component Parameters button on the toolbar.
 - Show Component Parameter Dialog for components without parameters displays the Item Parameters dialog box even for components that do not have parameters (GROUND, for example). By default it is off

and the dialog box does not appear. Double-clicking the component symbol brings up the dialog box so that you can change the item ID.

4. Click Apply.

Toggling Repeatable Component Placement

By default, a component remains selected for placement until you deactivate it. This enables you to place more than one copy of a component without selecting it each time.

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout dialog box, choose the Placement tab.
3. Enable/disable the Auto-repeatable component placement option.
4. Click Apply.

Setting the Size of Ports and Grounds

Use the field in this panel to set the size (in layout units or screen pixels) of ports and grounds.

Changing Options for Pins/Tees

To change pins/tees options, select Options > Preferences > Pin/Tee.

Pin/Tee

Size

Pin Units
0.05 layout units

Tee Units
0.05 layout units

Color

Connected Pins

Tee

Node Voltage

Pin Current

Node Name

Unconnected Pins use Highlight
Color defined in Display page

Visibility (on/off)

Connected Pins

Pin Numbers


Pin Names

Setting the Size of Connection Markers

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Pin/Tee tab.
There are two types of connection markers:
 - Pin sets the size of the marker that identifies component pins.
 - Tee sets the size of the marker that identifies tee connections between interconnected wires.
3. Enter the size and select the units.
 - Screen pixels specifies sizes in terms of pixels on the screen. For example, if you choose 5 screen pixels, the size of the marker is 5 pixels.
 - Layout Units specifies sizes in terms of the current units of the window. For example, if you are using inches and choose 0.1 layout units, the size of the marker is 0.1 inch.


Setting the Color of Pin & Tee Connections

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Pin/Tee tab.
3. Use the selections in the Color area to specify the color of the markers that identify connected pins, tee connections between interconnected wires, pin numbers, pin names, node voltages, pin currents, and node names.

 **Note**
Unconnected pins appear in the color set for highlighted items (see [Changing Display Colors](#)).

Setting Visibility of Connected Pins, Pin Numbers & Names

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Pin/Tee tab.
3. Use the selections in the Visibility area to toggle the visibility status of connected pin markers, pin numbers, and pin names.

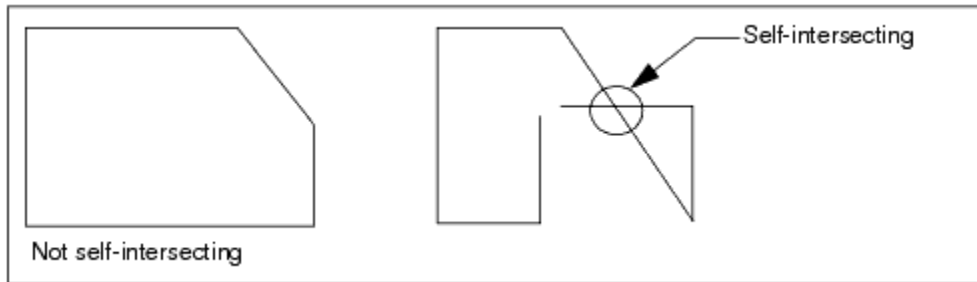
 **Note**
The Connected Pin selection in the Color area of this panel sets the color for the markers that identify connected pins, pin numbers, and pin names.

Changing Entry/Edit Attributes

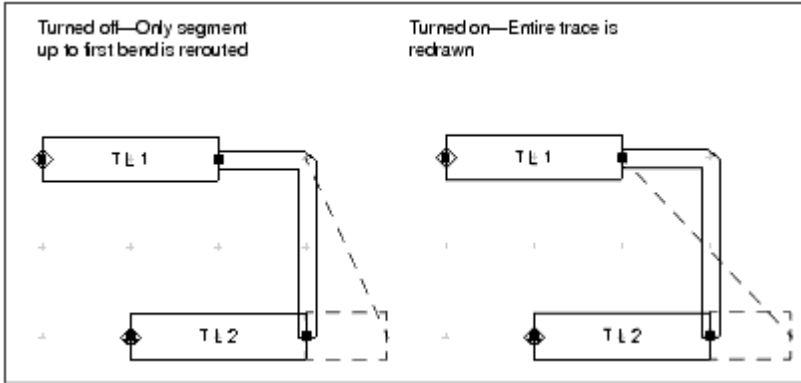
To change entry/edit options, choose Options > Preferences > Entry/Edit.

Advanced Design System 2008

- Polygon Entry Mode: Any angle
Enables you to draw polylines and polygons, using all angles.
- Polygon Entry Mode: 45 degree angle only
Restricts shape entry to 45 degree rotation increments.
- Polygon Entry Mode: 90 degree angle only
Restricts shape entry to horizontal or vertical.
- Show Coordinate Entry Dialog for Insert and Edit commands
Select this option to force the Coordinate Entry dialog box to be displayed when invoking the following commands:
 - Insert (Shape)—Polygon, Polyline, Rectangle, Circle, Arc (clockwise and counter-clockwise), Text, Construction Line, Symbol Pin, Path, Trace.
 - Edit—Move Wire Endpoint, Mirror X, Mirror Y, Move & Disconnect, Step And Repeat, Set Origin, Move Component Text.
- Show Set Paste Origin Dialog for Copy Command
Select this option to force the Set Paste Origin dialog box to be displayed when you choose the Copy command. This dialog box enables you to specify X and Y coordinates to be used as a reference point when pasting.
- Polygon self-intersection checking
Prevents you from placing additional points on a polygon if it will lead to overlapping lines.

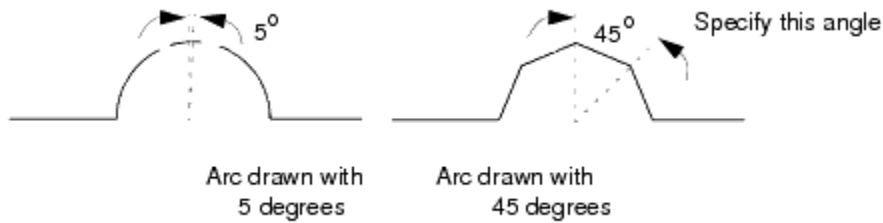


- Drag rectangle vertex maintains rectangularity
Insures that the dragging, the move relative, or the move using reference of one selected vertex of a single selected rectangle or of two adjacent selected vertices of a single selected rectangle will maintain its rectangularity.
- Maintain adjacent angles for Move Edge command
Restricts the Move Edge command to stretch an edge while maintaining the adjacent angles of the edge being stretched to other edges adjacent to that edge.
- Reroute entire wire attached to moved component
When this option is selected, the wire connection is allowed to be completely redrawn and rerouted as needed. When this option is deselected, only the segment (up to the first bend) of the wire attached to the component you are moving is rerouted; the remainder of the wire is unaffected.
- Reroute entire trace attached to moved component
When this option is selected, the trace connection is allowed to be completely redrawn and rerouted as needed. When this option is deselected, only the segment (up to the first bend) of the trace attached to the component you are moving is rerouted; the remainder of the trace is unaffected.



- Arc/Circle Radius (degrees)

Determines how smoothly curves are drawn. The number entered here defines when the program starts a new line segment. For example, an entry of 5 means that the program begins a new line every 5 degrees. In general, the fewer degrees, the smoother the shape, but the longer it takes to redraw the screen.

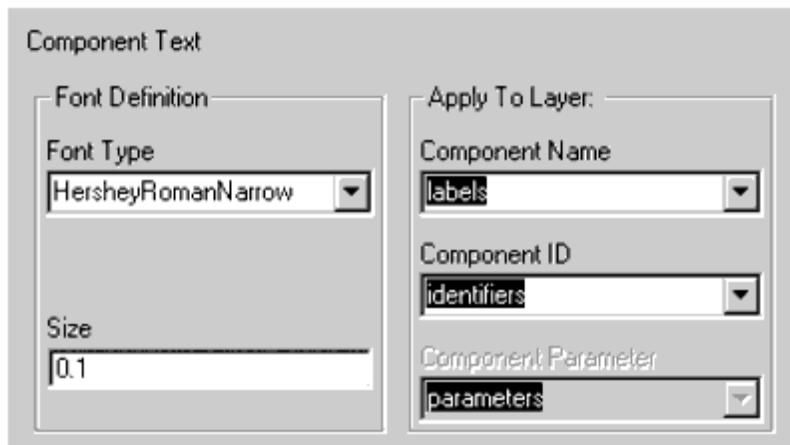


Note
This setting affects only circles in that the number specified here is used if you convert a circle to a polygon.

- Auto-backup edit count
Automatically saves a file each time the number of edits to that file reaches the number in this field.
- Undo edit count
This option represents the maximum number of commands held in the stack. Selecting Undo from the Edit menu or clicking the Undo button on the toolbar undoes the last editing command. A stack of edit commands is maintained for each window, thus the Undo command works independently from window to window. You can choose Undo repeatedly to return to an earlier state of your design. You can specify the number of commands you want the stack to hold using the Undo edit count option.
- Rotation Increment (angle)
This option forces objects you rotate to snap—during rotation—in n -degree increments, where n is the number you specify here.
- Drag and Move
This option is designed to prevent you from moving an item when you click to select it (for any purpose) and unintentionally move the pointer in the process. By default, a move less than 10 screen pixels is not recognized as a move. An intentional move must be more than the distance specified here for it to be recognized as a move.
- Merge/Boolean Logical/Create Clearance
Final Minimum Vertex Distance (in Layout Units)—Vertices that fall within the distance entered here are collapsed into one vertex. This eliminates the spikes or slivers created during merge operations that happen when vertices are too close together.

Changing Component Text Attributes

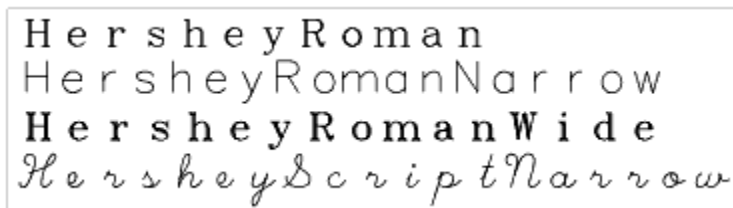
To change text attribute options, select Options > Preferences > Component Text.



Component text is the text associated with components selected from a library or palette. If the designated layer is visible, this text appears automatically when a component is placed in the Layout window.

Setting Component Text Font & Height

Font Use the drop-down list to choose a font. The default is HersheyRomanNarrow.



Height represents the text height with respect to the current units in a window (displayed in the status panel at the bottom of the window).

Setting the Layers for Component Text

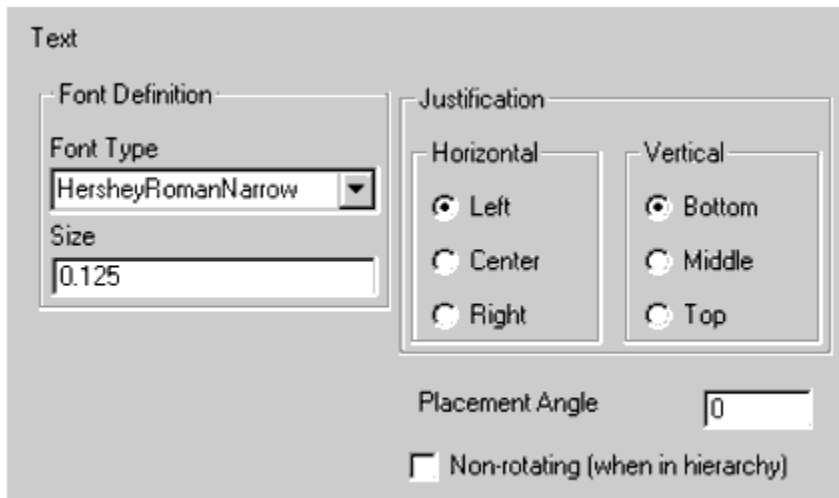
When a component is placed in layout, its name and reference designator (ID) are automatically placed with it on the silk screen layers. By default, the name is placed on the layer silk_screen2; the ID is placed on the layer silk_screen.

Name Use the drop-down list to define the layer for component names.

ID Use the drop-down list to define the layer for component IDs.

Changing Typed-in Text Attributes

To change text options, select Options > Preferences > Text.



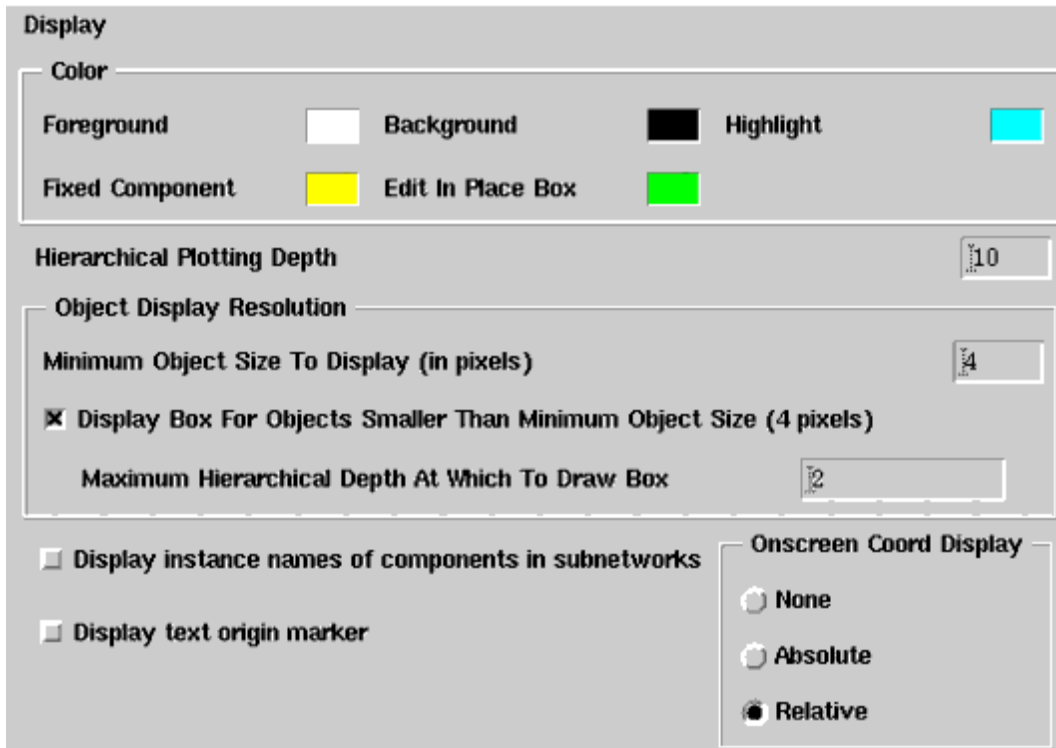
- Font Type—All TrueType fonts installed on your system are available. Select the desired font from the drop-down list. When printing to an HP-GL/2 file, text information will not be saved if the font is a TrueType font. To preserve the text in your output file, convert it to HersheyRomanNarrow before saving to HP-GL/2.

Note
On UNIX, if you want to add additional TrueType fonts that were not supplied with ADS, copy them to \$HPEESOF_DIR/lib/fonts (where \$HPEESOF_DIR represents your complete installation path).

- Size—Represents the size of text in traditional units used in printing.
- Justification, Horizontal—This setting represents two types of justification: one is how individual lines of text in a block of text are aligned with one another; the second is how an individual line of text or block of text is positioned horizontally, relative to the reference point you specified to begin typing the text.
- Justification, Vertical—This setting aligns a string or block of text vertically, relative to the reference point you specified to begin typing the text.
- Placement Angle—The angle at which all text subsequently added to your design will be drawn.
- Non-rotating (when in hierarchy)—Select this option to prevent text on a symbol or design from being rotated when the symbol is rotated.

Changing Display Colors

To change display color options, select Options > Preferences > Display.



Setting the Color of the Drawing Area

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Display tab.
3. In the Color area, click the color box to display a palette from which to choose a color.
 - Foreground defines the color of the lines making up polygons, polylines, and arcs.
 - Background defines the color of the Layout window background.
 - Highlight defines the color used to identify problem items, orphaned items in schematic and layout representation, and unconnected pins.
 - Fixed Component defines the color of fixed components.
 - Edit In Place Box defines the color of the box around the hierarchical design that is being edited during Edit in Place.
4. Click Apply.

Setting the Color of Unconnected Pins

1. Choose the menu command Options > Preferences.
2. In the Preferences for Layout Dialog Box, choose the Display tab.
3. In the Color area, click the colored box next to the word Highlight and select the color from the displayed palette.
4. Click OK to dismiss the palette.

5. Click Apply. This sets the color for:

- the marker that identifies an unconnected component pin, and
- the box that the program uses to highlight an item. This type of highlighting is used when you use choose one of the Layout (Schematic) > Show commands.

Setting DRC Memory Use and Performance

To set preferences for DRC memory usage and performance, refer to [Setting DRC Memory Use and Performance](#) in the Design Rule Checker manual.

Changing Miscellaneous Display Options

To change hierarchy display options, select Options > Preferences > Display.

The screenshot shows the 'Display' preferences dialog box. It is organized into several sections:

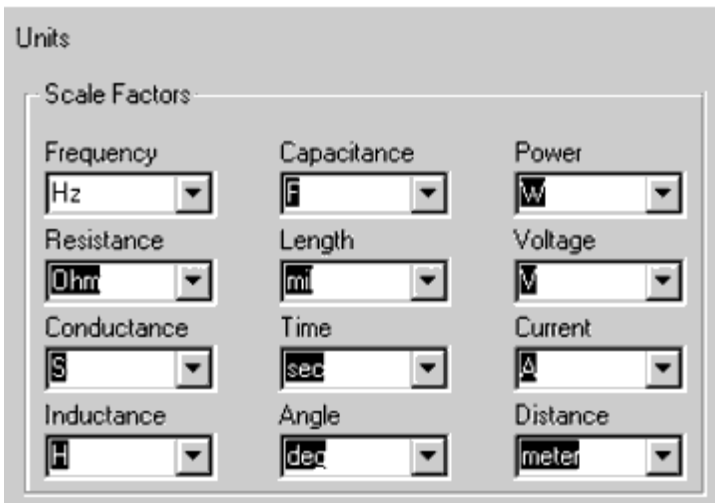
- Color:** A group box containing color selection swatches for 'Foreground' (white), 'Background' (black), 'Highlight' (cyan), 'Fixed Component' (yellow), and 'Edit In Place Box' (green).
- Hierarchical Plotting Depth:** A numeric input field set to 10.
- Object Display Resolution:** A group box containing:
 - Minimum Object Size To Display (in pixels):** A numeric input field set to 4.
 - Display Box For Objects Smaller Than Minimum Object Size (4 pixels)**
 - Maximum Hierarchical Depth At Which To Draw Box:** A numeric input field set to 2.
- Display instance names of components in subnetworks**
- Display text origin marker**
- Onscreen Coord Display:** A group box with three radio button options: 'None', 'Absolute', and 'Relative' (which is selected).

- **Foreground**—The color of the lines making up polygons, polylines, and arcs while they are being drawn.
- **Background**—The color of the drawing area background in the design window.
- **Highlight**—The color used to identify problem items (with respect to simulation), orphaned items in schematic and layout representations, and unconnected pins.
- **Hierarchical Plotting Depth**—The level of detail displayed in hierarchical designs. Any item nested below the plotting depth specified here is drawn as a bounding box, which can significantly increase the redraw speed of complex hierarchical designs. Plotting depth affects both screen and hardcopy output.

- Minimum Object Size To Display (in pixels)—The minimum size (in pixels) an object must be before it is actually drawn in the Schematic window. Objects smaller than this are not visible.
- Display Box For Objects Smaller Than Minimum Object Size (<current minimum size>)—Draws a box to represent any object that is smaller than the minimum object size (based on the setting in the field above).
 - Maximum Hierarchical Depth At Which to Draw Box—The number of levels in the hierarchical design for which you want a box drawn.
- Display text origin marker—Displays a cross mark at the point you clicked to begin typing text (the lower left corner of the text string).

Setting Units/Scale Factors

To change units and scale options, select Options > Preferences > Units/Scale.



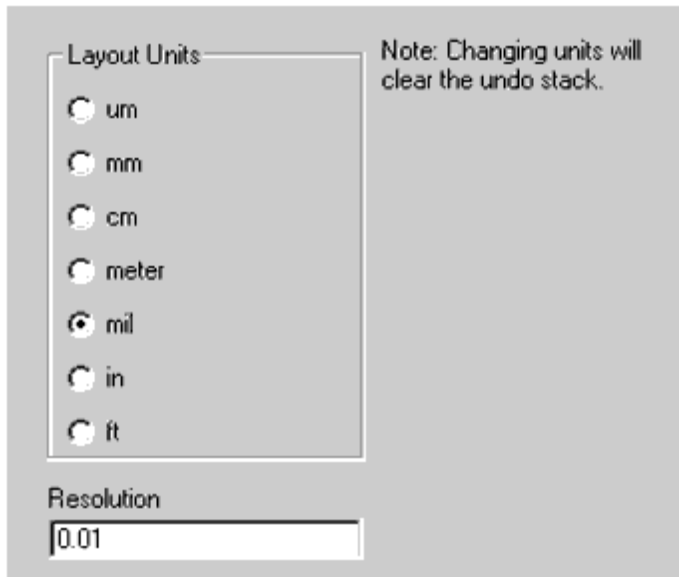
Scale factors are used in simulation and in generating artwork for parameterized artwork components.

Scale factors used in the layout should match those you want in the final output.

There are usually no problems associated with translating units that are in the same measurement system (mils to inches, or centimeters to millimeters), but round-off errors can occur when translating between metric and English units.

Changing Layout Units & Resolution

To change layout units and resolution options, choose Options > Preferences > Layout Units.



Layout units are used for any drawn item (such as a polygon, circle, or square).

- Note**
- Set the correct layout units at the beginning of a design. Changing units after a design is complete can result in the loss of information (due to round-off errors). Because of this, if you must change the units of an existing design, you should make a copy of the file before you change the units. Then you can compare the designs after the change to determine if any information was lost.
- When you change layout units, only the current design is rescaled to the new units

Setting Layout Resolution

Set the correct resolution (the smallest number allowed in layout) at the beginning of a design.

1. Choose the menu command Options > Preferences.
2. In the dialog box that appears, select the Layout Units tab.
3. Enter the resolution.
4. Click Apply. The resolution changes for the current design.

The smaller the number the more precise the data base, but because Layout uses a 32-bit integer data base, setting a very small resolution limits the largest user number that can be represented. This is usually not a problem for most designs until the resolution is greater than 0.0001.

The greater the resolution, the more difficult it is to ensure exact alignment of vertices in layout. Conversely, the greater the resolution, the smaller the gaps from round-off errors produced by non-orthogonal angles.

It is important to consider using a consistent resolution when creating related designs, or when using the packaged

parts library. Information can be lost when going from a higher to a lower resolution. The packaged parts library was created using the default resolution setting (100).

Angles are stored in the data base as integers, but they have a hard-coded resolution of 1000 data base units per degree. All angles are stored in degrees between -180 and 180 degrees. Angles specified with more than three decimal places are rounded off (24.7895 is stored as 24.790).

Toggleing the Coordinate Readout Display

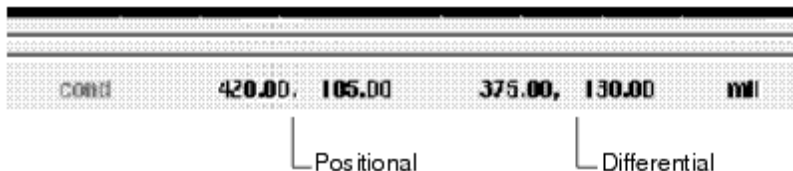
To change coordinate readout display options, select View > Coordinate Readout.

The X,Y coordinate display, which appears in the status bar at the bottom of the Layout window, displays two types of coordinates: positional and differential.

The coordinate readout is on by default.

Positional displays the X,Y coordinates of the cursor position in relation to the total window. By default, the large + in the center of the drawing area is 0,0.

Differential displays the distance in X,Y the cursor has traveled since the last click. Set the starting point to 0,0 by clicking anywhere in the drawing area.



To toggle the display:


1. Choose the menu command View.
2. Click Coordinate Readout.

Saving a Layout Setup

After you have the layout environment set optimally for your design, you can save these settings to be used for other

designs. A complete layout setup comprises two files:

- A preferences file that contains all settings under the Options menu except for layer information and options that can be set differently within a design (such as text height).
- A layer file that contains all of the layer information.

 **Note**
When you save a design file, the preference and layer files that are current at that time are read the next time you open that design file.

Saving Layout Preferences

You can save the settings in the Preferences for Layout dialog box to either the default preferences file (layout.prf), or a new preferences file.

1. In the Preferences for Layout dialog box, click Save.
2. In the Save Preferences File dialog box, add the file name to the end of the path displayed in the Selection field. If you use the default preferences filename (layout.prf), those preferences are read in each time you create a design in the current project directory.

Saving Layer Information

You can save the settings in the Layer Editor dialog box to either the default layer file (layout.lay), or a new layer file.

1. In the Layer Editor dialog box (Options > Layers), click Save.
2. In the Save Layer File dialog box, add the layer file name to the end of the path displayed in the Selection field. If you use the default layer filename (layout.lay), the default layer set is read in each time you create a design in that project directory.

Using an Existing Layout Setup

You can re-use existing layout information for a design, rather than setting up the layout environment each time you begin a design. A complete layout setup comprises two files:

- A preferences file that contains all settings under the Options menu except for layer information and options that can be set differently within a design (such as text height).
- A layer file that contains all of the layer information.

Note
When you save a design file, the preference and layer files that are current at that time are read the next time you open that design file.

Reading in an Existing Preferences File

1. In the Preferences for Layout dialog box, click Read.
2. In the Read Preferences File dialog box, double click the *.pref file you want to read.

Reading in an Existing Layer File

1. In the Layer Editor dialog box (Options > Layers), click Read.
2. In the Read Layer File dialog box, double click the *.lay file you want to read.

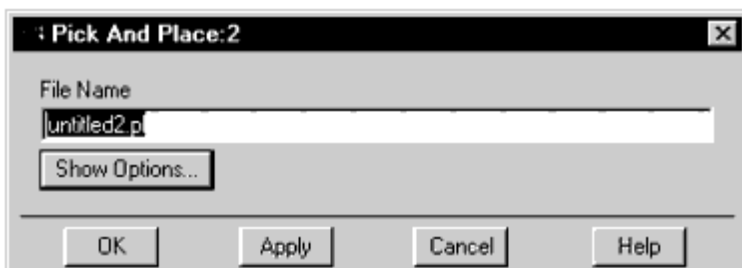
Reading a File from a Different Project Directory

1. In the Directories field, double click *_prj/...
2. Choose the project directory you want to read.
3. Double click the layer file.

Pick and Place Report

To generate a Pick and Place Report:

1. Select File > Reports > Pick And Place to open the dialog.

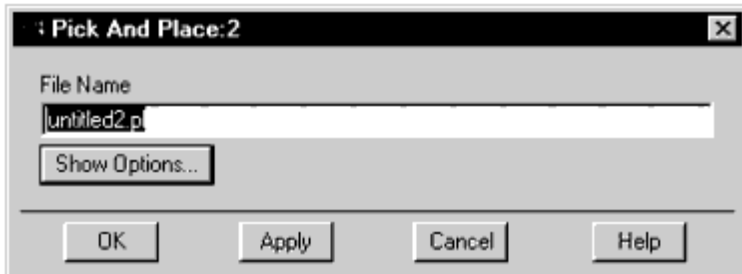


This dialog box allows you to edit the report name.

Viewing Pick and Place Report Configuration Options

To view the current pick and place report configuration options, do the following:

1. Select File > Reports > Pick and Place.
2. In the Pick And Place dialog box click the Show Options button.



3. The Report Options dialog box appears. This dialog box allows you to view and print the current report configuration options.



Configuring the Pick and Place Report

The Pick and Place Report is configured using the file `de_parts.ael`. This file is read during startup.

Note
Prior to ADS 1.3, the `de_parts.ael` file contained an AEL script that generated a Parts List Report. The report was generated when Parts List was executed. The `de_parts.ael` file is now used only to configure the Pick and Place and Parts List reports.

If you have customized the Parts List report, you can retain this functionality by concatenating your custom `de_parts.ael` file to the system `de_parts.ael` file. You will be redefining the `de_parts` function.

Advanced Design System 2008

You can customize the column data and the formatting of the report. You can add extra data columns in the report. The columns can be instance parameters, instance properties, or instance attributes.

The procedure `de_parts_set_pick_and_place_options` is called every time a Pick and Place Report is generated.

To modify the format of the Pick and Place Parts list, make a local copy of the system `de_parts.ael` file:

```
cp $HPPEESOF_DIR/de/ael/de_parts.ael $HOME/hpeesof/de/ael
```

To make your changes current without having to reboot ADS, reload the `de_parts.ael` file by typing the following line in the Command Line dialog box:

```
load ("de_parts.ael");
```

DE_PARTS_SET_PICK_AND_PLACE_OPTIONS

Routine: Configure the pick and place report

Method: This procedure is called every time a Pick and Place Report is generated.

```
defun de_parts_set_pick_and_place_options ()
{
  de_parts_option_initialize ();
  /* Reset to system defaults */
  /* Null out the exclusion and
  inclusion lists */
  de_parts_option_set_hierarchical (TRUE); /* Hierarchical report */
  de_parts_option_set_center_placement (TRUE); /* X,Y location at center */
  de_parts_option_check_bom (FALSE); /* Do not check BOM flag */
  /* Do not include simulation models */
  de_parts_option_add_exclusion_items (DePartsLumpedWithArtworkElements);
  de_parts_option_add_exclusion_items (DePartsMicrostripElements);
  de_parts_option_add_exclusion_items (DePartsPCBoardElements);
  de_parts_option_add_exclusion_items (DePartsStriplineElements);
  de_parts_option_add_exclusion_items (DePartsSuspSubElements);
  de_parts_option_add_exclusion_items (DePartsCoplanarElements);
  de_parts_option_add_exclusion_items (DePartsMultilayerElements);
  de_parts_option_add_exclusion_items (DePartsBlockTextFontsElements);
  /* Add an additional column to display the PART_NUM instance attribute */
  de_parts_option_set_attribute_columns (list ("PART_NUM"));
  de_parts_option_set_delimiter (NULL); /* Align columns */
  de_parts_option_include_header (TRUE); /* Include header */
  de_parts_option_sort_by_component (TRUE); /* Sort by component name */
}
```

Configuring the Parts List Report

Advanced Design System 2008

The procedure `de_parts_set_parts_list_options` is called every time a Parts List Report is generated.

```
//-----  
//          DE_PARTS_SET_PARTS_LIST_OPTIONS  
// Routine:   Configure the parts list report  
// Methods:   This procedure is called every time a Parts List Report  
//           is generated.  
//-----  
defun de_parts_set_parts_list_options ()  
{  
  de_parts_option_initialize ();          /* Reset to system defaults */  
                                          /* Null out the exclusion and  
                                          inclusion lists */  
  de_parts_option_set_hierarchical (TRUE); /* Hierarchical report */  
  de_parts_option_set_center_placement (TRUE); /* X,Y location at center */  
  de_parts_option_check_bom (FALSE);      /* Do not check BOM flag */  
  /* Do not include simulation models */  
  de_parts_option_add_exclusion_items (DePartsLumpedWithArtworkElements);  
  de_parts_option_add_exclusion_items (DePartsMicrostripElements);  
  de_parts_option_add_exclusion_items (DePartsPCBoardElements);  
  de_parts_option_add_exclusion_items (DePartsStriplineElements);  
  de_parts_option_add_exclusion_items (DePartsSuspSubElements);  
  de_parts_option_add_exclusion_items (DePartsCoplanarElements);  
  de_parts_option_add_exclusion_items (DePartsMultilayerElements);  
  de_parts_option_add_exclusion_items (DePartsBlockTextFontsElements);  
  de_parts_option_set_delimiter (NULL);   /* Align columns */  
  de_parts_option_include_header (TRUE);  /* Include header */  
  de_parts_option_sort_by_component (TRUE); /* Sort by component name */  
}
```

Reformatting the Reports

You can omit and reorder the columns in a Parts List or Pick and Place Report using AEL. Contact technical support if you wish to get a copy of the report formatting AEL script.

Check BOM Flag

Command: `de_parts_option_check_bom (TRUE|FALSE);`

TRUE Only include instances with attribute `INST_SPECIAL` set as `ITEM_BOM_ITEM`

FALSE Do not test for `ITEM_BOM_ITEM` (default)

Exclusion List

Command: `de_parts_option_add_exclusion_items (list ("MLIN"));`

Items in the list will not appear in the parts list.

This list is useful if parts have not been consistently flagged as BOM items. For this case, you wish to include everything except items in the exclusion list.

To include everything, do not check the BOM flag.

For example:

```
de_parts_option_check_bom (FALSE);  
de_parts_option_add_exclusion_items (DePartsLumpedWithArtworkElements);
```

Inclusion List

Command: `de_parts_option_add_inclusion_items (list ("res_smt"));`

Items in the list will appear in the parts list.

This list is useful if parts have not been consistently flagged as BOM items. For this case, specify to include only items flagged as BOM items, and add additional items in the inclusion list.

Inclusion items are treated as leaf-level parts and do not get flattened. For example, if an inclusion item is a hierarchical part, its sub-elements will not be included in the parts list.

For example:

```
de_parts_option_check_bom (TRUE);  
de_parts_option_add_inclusion_items (list ("res_smt"));
```

Hierarchical Reporting

Command: `de_parts_option_set_hierarchical (TRUE|FALSE);`

TRUE Produce a parts list containing instances from all levels of the hierarchy. (default)

FALSE Produces a parts list containing instances from only the top level of hierarchy.

- Enhancements:

Advanced Design System 2008

- Placement coordinates are in world space
- Arbitrary angles are supported
- Reference IDs are unique
- IDs contain the hierarchical path to the instance

Component Placement X,Y Coordinates

Command: `de_parts_option_set_center_placement (TRUE|FALSE);`

TRUE Coordinates represent the center point of the instance bounding box. The bounding box does not include the annotation text. (default)

FALSE Coordinates represent the location of pin one.

- Enhancement:
 - x,y placement coordinates can be either the center of the instance bounding box or the location of pin one

Component Placement X,Y Offset coordinates

Command: `de_parts_option_set_package_offset (packageName, packageName, xOffset, yOffset);`

For example:

```
de_parts_option_set_package_offset ("Package", "P1", 15, 0);
```

For each instance which has a user attribute named "Package", with attribute value "P1", the placement coordinate will be the origin offset by xOffset, yOffset.

- Enhancement:
 - The x,y placement coordinate for an asymmetrical instance is not the center of the instance bounding box, but rather a fixed offset from the origin. The offset can be incorporated in the report.

User Attribute Columns

Command: `de_parts_option_set_attribute_columns (list ("INST_SPECIAL", "PART_NUM", "Price"));`

Attributes in the list will appear as columns in the parts list. The attributes can be user properties, user parameters, or instance attributes. The following instance attributes can appear in the report:

INST_TYPE
INST_SPECIAL
INST_NAME
INST_DESIGN_NAME
INST_SYMBOL_NAME
INST_BBOX
INST_PROPERTY

Delimiter Character

Command: `de_parts_option_set_delimiter (delimiter);` where `delimiter` is used to separate column data (i.e. " ", ",")
Default is NULL.

If a NULL delimiter is specified, column widths will be determined by the longest data field and all data will be left justified.

For example:

```
/* Separate columns with commas */  
de_parts_option_set_delimiter (" ,");
```

For example:

```
/* Auto-format */  
de_parts_option_set_delimiter (NULL);
```

Include Header

Command: `de_parts_option_include_header (TRUE|FALSE);` TRUE = Output header information (default)

FALSE = Output part data only

- Enhancement:
 - User can specify whether to include the header information in the report. A report containing only part data will be easier to parse.

Sort by Component Name

Command: `de_parts_option_sort_by_component (TRUE|FALSE);`

TRUE = Sort the parts list by the component name (default)

FALSE = Parts are listed as they appear in the database

- Enhancement:
 - Default behavior is to sort the report by component name
 - Parts previously appeared in the order they were found in the database. Simple edits could cause the parts to be listed in a different order.

Customization Examples

The topics listed below represent some of the ways in which you can customize the Advanced Design System environment.


- [Creating a Custom Menu](#)
- [Creating Custom Libraries](#)
- [Advanced Library Customization Techniques](#)
- [Modifying the List of Available Symbol Names](#)
- [Adding Online Help for User-Defined Items](#)
- [Adding Custom Documentation for Online Access](#)

Many of the topics presented here refer to the variables `%HOME%` and `%HPEESOF_DIR%`. On the PC these variables have the following meaning:

- `%HOME%`—the path you specified as the Home folder during installation (`C:\users\default` by default)
- `%HPEESOF_DIR%`—the path you specified as your Program folder during installation (`C:\ADS2003` by default)

Creating a Custom Menu

You can add custom menus to the ADS Main window and Schematic and Layout windows by modifying the `.ael` file `usermenu.ael`.

 **Note**
This procedure applies only to user-level custom menus. The number of slots available for custom menus is limited; you must use care so that you do not wipe out another user's menu.

To add a custom menu:

1. Under \$HOME/hpeesof, create the additional directories /de/ael.
2. Copy the file usermenu.ael from \$HPEESOF_DIR/de/ael to the new directory \$HOME/hpeesof/de/ael and ensure you have write permission for this file.
3. Using any text editor, open the file usermenu.ael.
4. The file usermenu.ael contains sample code that can be uncommented to help you learn how to use the functions available for defining user menus. Before proceeding, review the descriptions of the following functions in the AEL manual: check_user_menu(), set_user_menu_label(), and add_menu(). After you understand how to use the sample code, replace it with your own code. The function app_add_user_menus() is the function in which you will add your custom code. The sample code provided includes the following function calls:

```
// to get the internal name of the first free user menu
decl freeMenuName = app_find_empty_user_menu(winType);
// uses the menu's internal name to set the label on a free user menu
set_user_menu_label("my first menu", freeMenuName);
// uses the menu's internal name to add a menu pick on a free user menu
add_menu( "menuPickA", "my_menu_cb", freeMenuName );
```

Uncomment these lines in your file by removing the two slashes at the beginning of the three executable lines, as shown above. Do not delete the slashes from the comment lines.

5. Save the file.
6. Restart ADS to see the change.

Note
If this code is used as is, it will initialize the first user menu, unless your system already has user menus defined. If your system does not have any user menus defined by other applications, and you want to see what would happen if it did, you can uncomment the test function prefill_menus() and the call to it, both provided in usermenu.ael. This will initialize the first two user menus, so that a call to app_find_empty_user_menu() will return "User3" as the first available user menu.

Creating Custom Libraries

As with other aspects of ADS customization, you can create custom libraries at several levels: for site-wide use, for all your projects, and for individual projects. The procedures are presented in three different groupings, based on the following use models:


- Assisted customization for the average user, done through the user interface (Tools > Custom Library), for a small number of designs
- Manual customization, typically done by a librarian, using the recommended directory structure, for a large number of designs
- Manual customization, typically done by a librarian, using directories other than the recommended directories (Advanced)

If creating libraries via the user interface method, before defining a Library Name-through the Design Parameters dialog box-for any given design, please review the following guidelines, as assigning a library name in this manner is not always recommended:

- Basic-Do not use the Library Name field in the Design Parameters dialog box. Create a library via the user interface method. Each process (Create > User Part or Create > Site Part) results in a single library. By default the library name is User Library or Site Library. Note that because all designs are in a single library, the entire library must be loaded or not loaded.
- Intermediate- Do use the Library Name field in the Design Parameters dialog box to categorize designs so that they appear in the library browser under those categories (for example, My Resistors, My Capacitors). Create a library via the user interface method. Each process (Create > User Part or Create > Site Part) results in a single library. Note that because all designs are in a single library, the entire library must be loaded or not loaded.
- Advanced-Do not use the Library Name field in the Design Parameters dialog box. Set the MAKELIB_* configuration variables to define multiple libraries. Create libraries via the user interface method (Create > User Part or Create > Site Part). Each time you add a part you will be prompted to select one of your pre-defined libraries. In this manner, the loading of individual libraries can be enabled or disabled.

Assisted Customization

The procedures in this section are geared toward individuals working with a small number of designs, because this method is based on adding designs to a library one at a time. Once the libraries have been created, either restart ADS to see the changes or use the Configuration Explorer to explicitly read the modified de_sim.cfg file for the current session.

 **Note**
Because this method is based on copying designs from their source location to a library location, if the source design is subsequently modified, you must repeat the process so that the design gets copied to the library again.

Keep in mind the following limitations when creating libraries in this manner:

- Only the current design is copied to the library when using the Create > User Part or Create > Site Part commands. If the design is hierarchical, you must open or push into each subnetwork and repeat the process.

 **Hint**
To easily see the designs comprising a hierarchical design, from the ADS Main window, choose View > Design Hierarchies.

- Some files required to successfully recreate the design in its library location are not copied to the library. Those files that are not automatically copied to the library must be explicitly copied to ensure the integrity of the design:

Files Copied to the Library

<design>.dsn -The design file
<design>.ael -The component definition AEL file
<design>.atf -The compiled component definition AEL file
<design>_art.ael -The layout artwork AEL file generated by the Graphical Cell Compiler, where applicable
<design>_art.atf -The compiled layout artwork (GCC) AEL file, where applicable
<datafile>.ext -Data files referenced by current design †
Files Not Copied to the Library
Custom symbols stored in a separate .dsn file. The workaround is to open the .dsn file containing the symbol and add it to the library using the appropriate create part command.
Referenced fixed artwork stored in a separate .dsn file. The workaround is to open the .dsn file containing the artwork and add it to the library using the appropriate create part command.
Artwork macros stored in a separate file, other than those created by the Graphical Cell Compiler. The workaround is to manually copy the .ael file containing the artwork to the library (\$HPEESOF_DIR/custom/circuit/ael for Site or \$HOME/hpeesof/circuit/ael for User).
†
<ul style="list-style-type: none"> • If a design being copied to the library references a data file that is not in the project's /data directory-but is specified using a path, absolute or relative—the file is not copied. Thus for site libraries, you must make sure all users on all workstations can access the file(s) in this location. • If your design contains a DAC that references a discrete MDIF file that lists multiple data files, only the directly referenced file will be copied to the library. The individual files listed in the MDIF file must be manually copied to the library (\$HPEESOF_DIR/custom/datafiles for Site or \$HOME/hpeesof/datafiles for User).

Site-Wide Customization (Assisted)

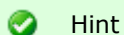
Copying designs to and removing designs from the site library requires write permission in that directory (by default, \$HPEESOF_DIR/custom/circuit).

To copy a specific design to a site library making it available for all users:

1. From the design you want to copy to the site library, choose Tools > Custom Library > Create > Site Part.
2. Repeat this process for each design you want to be part of a site library.

To remove a specific design from a library:

Choose Tools > Custom Library > Delete > Site Part and select the design you want to remove from the library.



Hint

If the design you select to remove from the library references any data files, you are prompted to choose to keep or remove the data files.

To prevent an entire library of designs from being loaded when you start ADS:

Choose Tools > Custom Library > Disable > Site Library, and where applicable, select the library.

To enable the loading of a library that you previously disabled:

Choose Tools > Custom Library > Enable > Site Library, and where applicable, select the library.

Notes:

- When you use Create > Site Part, the design and related files are copied to a number of default directories under \$HPEESOF_DIR/custom/circuit.
- When you use Enable > Site Library, the appropriate variable (in the file \$HPEESOF_DIR/custom/config/de_sim.cfg) is updated so that the library is loaded the next time you start ADS. This command is used to enable libraries previously disabled.
- Site libraries created using the default procedure just presented will always be loaded by all users. To allow users the freedom to enable/disable loading of these libraries, the following variables can be set in \$HPEESOF_DIR/custom/config/de_sim.cfg:

```
MAKELIB_USER_AEL_PATH  
MAKELIB_USER_DSN_PATH  
MAKELIB_USER_LIBRARY_NAMES *See note below  
MAKELIB_USER_DATA_FILES
```

*Note: This variable is only used if no Library Name was specified in the Design Parameters dialog box, for any given design. For details on using these variables, refer to [Variables in the Files de.cfg and de_sim.cfg](#) in [Customizing the ADS Environment](#).

User Customization (Assisted)

To copy a specific design to a local library making it available for all your projects:

1. From the design you want to copy to the library, choose Tools > Custom Library > Create > User Part.
2. Repeat this process for each design you want to be part of a user library.

To remove a specific design from a library:

Choose Tools > Custom Library > Delete > User Part, and select the design you want to remove from the library.

To prevent an entire library of designs from being loaded when you start ADS:


Choose Tools > Custom Library > Disable > User Library, and where applicable, select the library.

Manual Customization

This section is geared toward the librarian, or individual, working with a large number of designs, and working with the recommended directory structure.

For the two manual methods, each case involves the following basic steps, though the details vary:

- Creating the directories shown, if they do not exist
- Copying the .dsn and .ae I files (you want made available) to these directories
- Modifying the search paths of specific variables to look in those directories

 **Note**
Do not include line breaks of any kind when setting variables; let the lines break wherever your text editor breaks them.

When customizing for any of the scenarios described in the manual methods, wherever you see < product_area >, replace it with one of the following:

- de—if you are customizing both Analog/RF and ADS Ptolemy libraries
- circuit—if you are customizing Analog/RF libraries only
- adsptolemy—if you are customizing ADS Ptolemy libraries only

Site-Wide Customization (Manual)

To make a specific design available for all users:

1. Copy the .dsn files to \$HPEESOF_DIR/custom/<product_area>/symbols/ directory. Example:

Advanced Design System 2008


Copy mydesign1.dsn and mydesign2 .dsn to \$HPEESOF_DIR/custom/de/symbols/

2. Copy the .ael files to the \$HPEESOF_DIR/custom/<product_area>/ael/ directory. Example:

Copy mydesign1.ael and mydesign2.ael to \$HPEESOF_DIR/custom/de/ael/

3. Edit or create a file \$HPEESOF_DIR/custom/config/de_sim.cfg and set SITE_AEL equal to each of the .ael filenames (separated by semicolons or colons). Example:

```
SITE_AEL=mydesign1;mydesign2
```

 **Note**
If you want all .ael files in the directory to be loaded, a shortcut is to set the SITE_AEL variable to the directory. For example: SITE_AEL=\$HPEESOF_DIR/custom/de/ael/. Be sure to include the trailing slash.

User Customization (Manual)

To make designs available for all your projects:

1. Copy the .dsn files to \$HOME/hpeesof/<product_area>/symbols/ directory. Example:

Copy mydesign1.dsn and mydesign2.dsn to \$HOME/hpeesof/de/symbols/

2. Copy the .ael files to \$HOME/hpeesof/<product_area>/ael/ directory. Example:

Copy mydesign.ael and mydesign2.ael to \$HOME/hpeesof/de/ael/

3. Open \$HOME/hpeesof/config/de_sim.cfg and set USER_AEL equal to each of the .ael file names (separated by semicolons or colons). Example:

```
USER_AEL=mydesign1;mydesign2
```

Project Customization (Manual)

To make designs available in multiple-but not all-projects:

Advanced Design System 2008

1. Copy the .dsn files to \$HOME/hpeesof/<product_area>/symbols/ directory. Example:

Copy mydesign1.dsn and mydesign2.dsn to \$HOME/hpeesof/de/symbols/

2. Copy the .ael files to \$HOME/hpeesof/<product_area>/ael/ directory. Example:


Copy mydesign.ael and mydesign2.ael to \$HOME/hpeesof/de/ael/

3. For each project, open <project>/de_sim.cfg and set LOCAL_AEL to each of the .ael filenames (separated by semicolons or colons). Example:

```
LOCAL_AEL=mydesign1;mydesign2
```

Advanced Library Customization Techniques

We recommend storing library designs for site-wide use in the following locations:

 **Note**
The custom directory shown here is created during installation, but you must create the product-specific directories (beneath it) yourself.

	Analog/RF	Digital Signal Processing
.dsn files	\$HPEESOF_DIR/custom/circuit/symbols/ or \$HOME/hpeesof/circuit/symbols/	\$HPEESOF_DIR/custom/adsptolemy/symbols/ or \$HOME/hpeesof/adsptolemy/symbols/
.ael files	\$HPEESOF_DIR/custom/circuit/ael/ or \$HOME/hpeesof/circuit/ael/	\$HPEESOF_DIR/custom/adsptolemy/ael/ or \$HOME/hpeesof/adsptolemy/ael/

If for some reason you cannot or do not want to store your files in the these directories and would rather locate your files elsewhere, you can.

1. Copy the .dsn files to any directory. Example:


Q:\ads_designs\library1\designs

2. Copy the .ael files to any directory. Example:

Q:\ads_designs\library1\ael

Choose one of the following customization schemes:

- Site-Wide basis (all users at the site)
- User basis (all projects for a single user)

 Note

There is no way to do this for project-level customization at this time.

Site-Wide Customization (Advanced)

Edit the `$HPEESOF_DIR/custom/config/de_sim.cfg` file as follows:

1. The `SITE_AEL` variable needs to be set to load your `.ael` files. Typically you would set it to point to the directory containing your `.ael` files. Example:

```
SITE_AEL=Q:\ads_designs\library1\ael (Loads all .ael files in directory)
```

Be sure to include the trailing slash. Note that you cannot not use relative paths; you must use either an absolute path or an environment variable.


Alternatively, you can list specific files that you want to load. Example:

```
SITE_AEL=Q:\ads_designs\library1\ael\myfile
```

2. Set the `SITE_DSN_PATH` variable equal to the directory you copied the `.dsn` files to earlier. Example:

```
SITE_DSN_PATH=Q:\ads_designs\library1\designs (absolute path) or
```

```
SITE_DSN_PATH={$ADS_DESIGN}\library1\designs (environment variable)
```


 Important

If you set the `SITE_AEL` variable (in step 1) to a path, rather than a list of individual designs, you are done; if you listed designs individually, continue with the next two steps.

Edit the `$HPEESOF_DIR/custom/config/de_sim.cfg` file as follows:

1. Note that every user will already have the variable `AEL_PATH` set in their `$HOME/hpeesof/config/de_sim.cfg` file. It will be set to one of the following variables, based on the choice they made in the setup dialog box the first time they started ADS:

```
{%ANALOG_RF_AEL_PATH}  
{%DSP_AEL_PATH}  
{%BOTH_ANALOG_RF_FIRST_AEL_PATH}  
{%BOTH_DSP_FIRST_AEL_PATH}
```

 Note

The setting chosen on startup can be changed at any time from within ADS from the Main Window through Options > Advanced Design System Setup.

Advanced Design System 2008

The default definitions of these four variables can be found in `$HPEESOF_DIR/config/de_sim.cfg`. Copy all four variables from `$HPEESOF_DIR/config/de_sim.cfg` and add them to the `$HPEESOF_DIR/custom/config/de_sim.cfg` file.

2. Modify each variable to add the path-that you copied the .ael files to earlier-to the front of the existing path.
Example:

```
ANALOG_RF_AEL_PATH=Q:\ads_designs\library1\ael:{%HOME_CIRC...
DSP_AEL_PATH=Q:\ads_designs\library1\ael:{%HOME_HPTOL...
BOTH_ANALOG_RF_FIRST_AEL_PATH=Q:\ads_designs\library1\ael:{%HOME_...
BOTH_DSP_FIRST_AEL_PATH=Q:\ads_designs\library1\ael:{%HOME_HP...
```

Now, even if a user changes their setup (from within ADS) to start with a different design type-therby updating their local `AEL_PATH` variable-the designs can still be found because the search path is provided for all cases.

Example:

The `AEL_PATH` variable that is set based on your choice of design type:

`$HOME/hpeesof/config/de_sim.cfg` → `AEL_PATH={%ANALOG_RF_AEL_PATH}`

The variable your `AEL_PATH` variable points to, after being copied and modified:

`$HPEESOF_DIR/custom/config/de_sim.cfg`
↳ **`ANALOG_RF_AEL_PATH=Q:\ads_designs\library1\ael:{%HOME_CIRCUIT_AEL...`**

User Customization (Advanced)

Edit the `$HOME/hpeesof/config/de_sim.cfg` file as follows:

1. Set the `USER_AEL` variable equal to the design filenames (separated by semicolons or colons). Example:

```
USER_AEL=mydesign1;mydesign2
```

Note

If you want all .ael files in the directory to be loaded, a shortcut is to set the `USER_AEL` variable to the directory. For example: `USER_AEL=$HOME/hpeesof/MyLibrary/ael/`. Be sure to include the trailing slash.

2. Set the `USER_DSN_PATH` variable equal to the directory you copied the .dsn files to earlier. Example:

```
USER_DSN_PATH={ $HOME }/hpeesof/MyLibrary/Designs/
```

Note
If you set the USER_AEL variable (in step 1) to a path, rather than a list of individual designs, you are done; if you listed designs individually, continue with the next step.

3. Add the path-you copied your .ael files to earlier-to the front of the current AEL_PATH definition. Example:

```
AEL_PATH={ $HOME }/hpeesof/MyLibrary/ael;{%BOTH_ANALOG_RF_FIRST_AEL_PATH}
```

Note
If you have customized the AEL_PATH path variable and you change the design type startup setting through Options > Advanced Design System Setup, in the Main window and click OK, this variable will be overwritten and you will lose your customized path. If you need to change your setup, you will either have to change it manually in your de_sim.cfg file, or re-establish your customized path after changing it through the dialog box.

Example

The steps that follow can be used as a shortcut for creating a library from a single project.

1. Create a project, such as library_prj.
2. Place in library_prj/networks only those parts you are interested in using directly.
3. Modify the variables USER_AEL and USER_DSN_PATH, shown in the example below, in the file \$HOME/hpeesof/config/de_sim.cfg. Example:

```
USER_AEL = /eesof/users/kclarke/library_prj/networks/  
USER_DSN_PATH = /eesof/users/kclarke/library_prj/networks/
```

Modifying the List of Available Symbol Names

To assign a symbol to a schematic, you must supply a Symbol Name in the Design Definitions dialog box (File > Design Parameters). You can type a symbol name or you can select one from the default list of commonly used symbols. You can modify this list and add names of other supplied symbols as well as the names of custom symbols you have created.

To modify the list of names, you modify the AEL function set_design_choices in the file shown next, in accordance with its intended use (circuit vs. adsptolemy):

Advanced Design System 2008

- /circuit/ael/geminidsndef.ael
- /de/ael/spdsndef.ael

To modify this file for individual use, modify a local copy of it; to modify this file for site use, modify a copy of this file on a shared drive. We recommend that you maintain the established directory structure for storing and referring to .ael files. Note that the custom directory shown below is created during installation, but you must create the product-specific directories (beneath it) yourself.


Use this directory structure on UNIX:

Individual Use	\$HOME/hpeesof/circuit/ael \$HOME/hpeesof/adsptolemy/ael
Site Use	\$HPEESOF_DIR/custom/circuit/ael \$HPEESOF_DIR/custom/adsptolemy/ael

Use this directory structure on the PC:

Individual Use	%HOME%/hpeesof/circuit/ael %HOME%/hpeesof/adsptolemy/ael
Site Use	%HPEESOF_DIR%/custom/circuit/ael %HPEESOF_DIR%/custom/adsptolemy/ael

%HOME% represents the path you specified as the Home Folder during installation (C:\users\default by default); %HPEESOF_DIR% represents the path you specified as your Program Folder during installation (C:\AdvDesSys).

 **Note**
If you use a directory other than one of those shown in the tables for storing the modified file (gemini.ael or spdsndef.ael), then you must declare the variable USER_AEL and provide the search path. The custom directory shown here is created during installation, but you must create the product-specific directories (beneath it) yourself.

To take advantage of the modified version of the gemini.ael or spdsndef.ael file-if stored in a directory other than one of the defaults:

1. Using any text editor, open the file \$HOME/hpeesof/config/de_sim.cfg.
2. Add the variable USER_AEL and set it equal to the path you have chosen for your modified file.
3. Save the file.

Once you modify geminidsndef.ael or spdsndef.ael, you must also modify the variable USE_RELATIVE_AEL_PATHS (regardless of the location of the modified file). This should be done in the file \$HOME/hpeesof/config/de_sim.cfg and you must set the variable equal to TRUE (USE_RELATIVE_AEL_PATHS =TRUE). For information on modifying search paths, "Modifying Search Paths" in the Schematic Capture and Layout book.

Once you have created the directory structure, copy the geminidsndef.ael (circuit) or spdsndef.ael (adsptolemy) file from the installation directory to the newly created local / ael directory.

To modify the list of symbol names appearing for selection:

1. Using any text editor, open the copied `geminidsndef.ael` or `spdsndef.ael` file in the local / ael directory.
2. Locate the heading

```
/* symbol list: DMDESIGN_SYMBOL_CHOICES */
```

followed by the function

```
set_design_choices
```

Notice that the function name is followed by a list of symbol names, each surrounded by quotation marks, separated by commas (including sequential numbering and commas), all enclosed in parentheses.


3. Add the desired names to the list using the same format.
4. Save the file.


After you have completed this process, you will be able to select your custom symbol from the drop-down Symbol Name list (in the Design Definitions dialog box).

If the program is currently running, you need to restart the program to see these changes take effect.

Adding Online Help for User-Defined Items

When you create your own items, you can also create documentation for them and make that documentation available online. The starting point of this procedure assumes you have already created the items.

 **Note**
Refer to the [DesignGuide Developer Studio](#) manual for more information on how to create custom DesignGuides for the Advanced Design System.

 **Note**
This procedure requires the ability to write to the installation directory.

1. Create a directory, for example `my_items`, under `$HPEESOF_DIR/doc` for storing your new documentation files (where `$HPEESOF_DIR` represents your complete installation path).
2. Change the write permission to this directory as needed, based on the users who will create documentation for the new items.
3. Create documentation for the new items using the HTML format (one file for each item) and move (or copy) the files to the new directory. While filenames can be any name you choose-except for the required `.htm` extension-we recommend using `< item_name >.htm`. See [Optional Document Template](#) for a html template.

Advanced Design System 2008

4. Create a text file, for example `my_items_help.txt`, containing pointers to the new items, using the following syntax (note the spaces before and after the first exclamation point and before the final exclamation point):

```
item_name ! <path>/<filename> !
```

where

`item_name_` is the element name you used in the AEL `create_item()` function

`path_` is the directory you created below `$HPEESOF_DIR/doc` to store the HTML files you created (for example, `my_items`)

This file should contain one line for each new item you created. An example line is shown next.

```
PIPAD ! my_items/pipad.htm !
```

5. The file that contains the pointers for help on all components is named `hshpeesofsim_index` and can be found in `$HPEESOF_DIR/doc`. Create a copy of this file (as a safeguard), for example, `hshpeesofsim_index.sav`.
6. Using any text editor, add the contents of the help file you created (in this example, `my_items_help.txt`), to the bottom of the `hshpeesofsim_index` file.
Optionally, on UNIX, you can use the `cat` command (instead of a text editor) and concatenate the files:

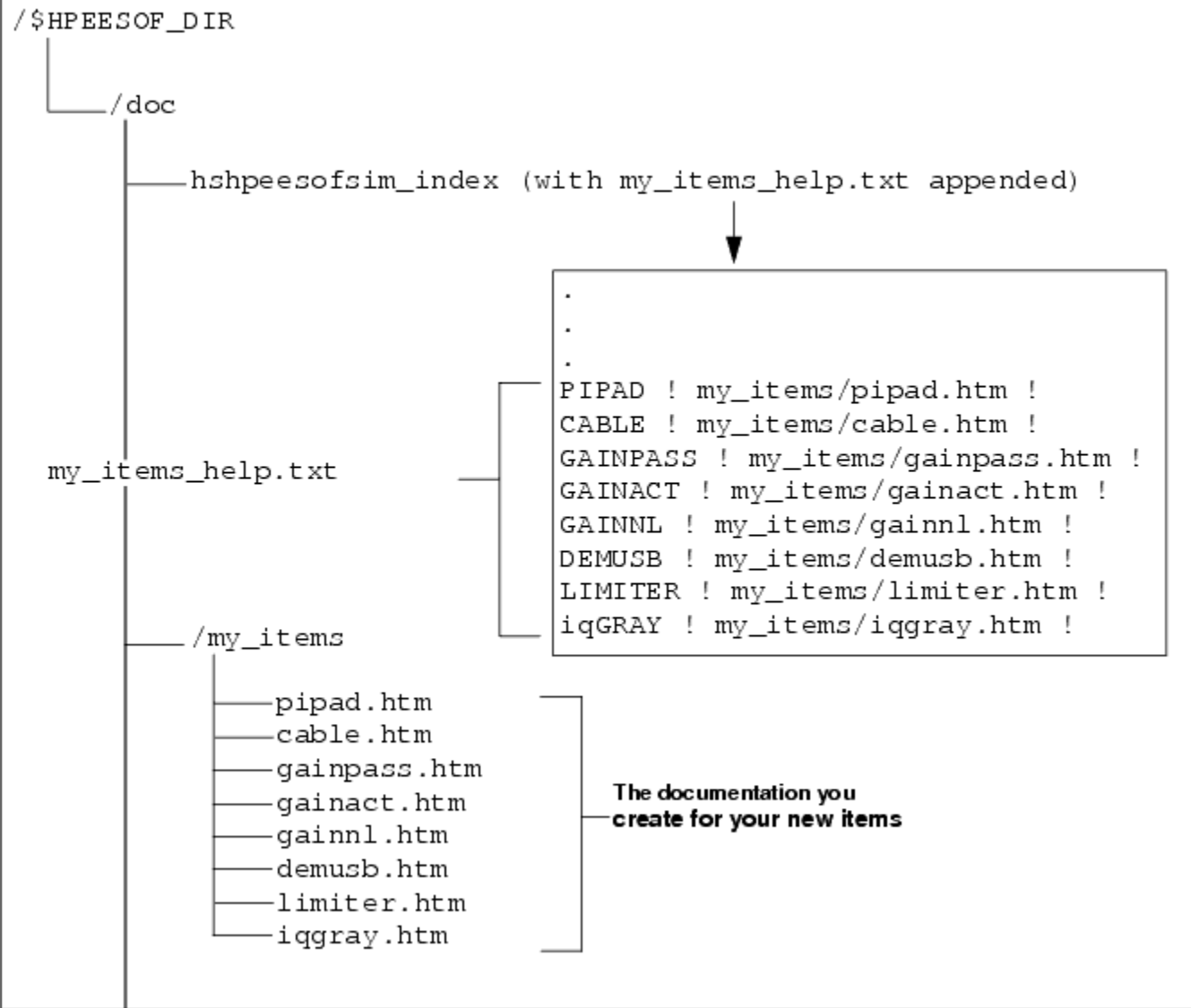
```
cat my_items_help.txt }} >> {{ hshpeesofsim_index
```

The help file now contains pointers to the documentation you created.

7. Save the file (no extension) and restart ADS to verify your changes.

The following figure illustrates the directory structure for the user-defined files.

Directory structure and files



Directory Structure for User-Defined Help

Optional Document Template

Use the following html code when you create your document.

Advanced Design System 2008

- Replace "ENTER YOUR DOCUMENT TITLE HERE" with the title you want to use for your document.
- Replace "PATH_TO_HPEESOF_DIR" with the relative path to your ADS install directory.
- Add your content after the "<!--Start Your Document Below -->" marker.

```
<html>
<head>
  <title>ENTER YOUR DOCUMENT TITLE HERE</title>
</head>
<body bgcolor="#ffffff">
  <p></p>
  <map name="navtop">
    <area shape="rect" coords="-7,1,326,20" href="manuals.htm" alt="ADS
Online Documentation" title="ADS Online Documentation"
target="_parent">
    <area shape="rect" coords="2,22,111,40" href="doc.html" target="_parent"
alt="ADS Online Manuals" title="ADS Online Manuals">
    <area shape="rect" coords="110,23,219,39" href="examples.html"
target="_parent" alt="ADS Examples" title="ADS Examples">
    <area shape="rect" coords="220,23,328,39" href="guides.html"
target="_parent" alt="ADS DesignGuides" title="ADS DesignGuides">
    <area shape="rect" coords="329,22,440,40" href="adstour/index.html"
target="_parent" alt="ADS Quick Start" title="ADS Quick Start">
    <area shape="rect" coords="440,23,549,40" href="search/search.html"
target="helpcontents" alt="Search ADS Documentation" title="Search ADS
Documentation">
  </map>
  <!--Start Your Document Below -->
</body>
</html>
```

Adding Custom Documentation for Online Access

It is possible to create your own custom documentation for online access. For instance, you can put custom documentation for a specific project in that project folder. You can also put documentation in other areas outside of \$HPEESOF_DIR, circumventing the issue of access permissions for \$HPEESOF_DIR.

1. Create a directory, for example custom_doc, for storing your new documentation files. The LOCAL_DOC_PATH, and LOCAL_BOOK_NAMES variables will need to be added to the hpeesohelp.cfg file, so that your custom doc will be accessed, instead of the standard ADS documentation. See [Customizing Configuration Variables](#) for more information on the location of .cfg files. See [Variables in the File hpeesohelp.cfg](#) for more information on the LOCAL_DOC_PATH and LOCAL_BOOK_NAMES variables.
2. Create documentation using the HTML format and move (or copy) the files to the new directory. While filenames can be any name you choose-except for the required .htm extension-we recommend using < item_name >. htm. See [Optional Document Template](#) for a html template.
3. Create a file, for example custom_doc_index, (no extension, _index required) __ containing pointers to the items that will be accessed in your documentation, using the following syntax (note the spaces before and after the first exclamation point and before the final exclamation point):

```
Index ! <filename>.html !
Topics and Index ! <filename>.htm !
item_name ! <path>/<filename> !
where
```

Advanced Design System 2008

Index	is the file that you would like the user to be directed to if a topic is not found. The standard for ADS documentation is \$HPPEESOF_DIR/doc/adshelp.html, if you choose to use your own file, make sure that it is located in your custom_doc directory. (See note below)
Topics and Index	is the file that is opened when the user selects Topics and Index from the Help Menu. The standard for ADS documentation is \$HPPEESOF_DIR/doc/manuals.htm, if you choose to use your own file, make sure that it is located in your custom_doc directory. (See note below)
item_name	is the element name you used in the AEL _create_item()_ function
path	is the directory you created to store the HTML files you created (for example, _custom_doc_

This file should contain one line for each new item you created. An example line is shown next.

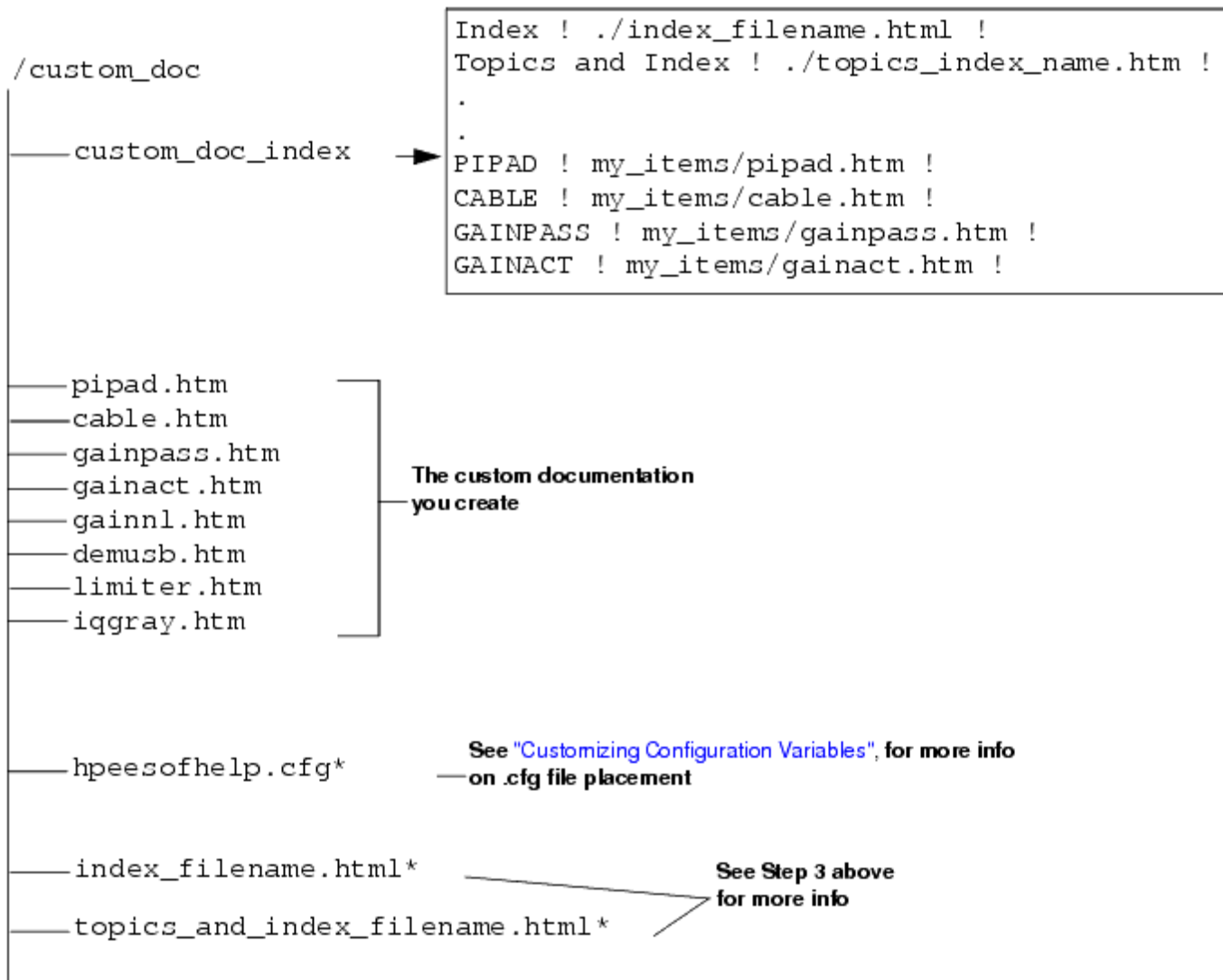
```
PIPAD ! custom_doc/pipad.htm !
```

i Note
If you point to custom Index and Topics and Index files within a specific project, and the project is moved, your paths will no longer be correct.

4. Save the file (no extension) and restart ADS to verify your changes.

The following figure illustrates the directory structure for custom documentation files.

Directory structure and files



Directory Structure for Custom Documentation