



**Feburary 2011
Customization and Configuration**

© Agilent Technologies, Inc. 2000-2011

5301 Stevens Creek Blvd., Santa Clara, CA 95052 USA

No part of this documentation 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. Mentor products and processes are registered trademarks of Mentor Graphics Corporation. * Calibre is a trademark of Mentor Graphics Corporation in the US and other countries. "Microsoft®, Windows®, MS Windows®, Windows NT®, Windows 2000® and Windows Internet Explorer® 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. Oracle and Java and registered trademarks of Oracle and/or its affiliates. Other names may be trademarks of their respective owners. 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. FLEXIm is a trademark of Globetrotter Software, Incorporated. Layout Boolean Engine by Klaas Holwerda, v1.7 <http://www.xs4all.nl/~kholwerd/bool.html> . FreeType Project, Copyright (c) 1996-1999 by David Turner, Robert Wilhelm, and Werner Lemberg. QuestAgent search engine (c) 2000-2002, JObjects. Motif is a trademark of the Open Software Foundation. Netscape is a trademark of Netscape Communications Corporation. Netscape Portable Runtime (NSPR), Copyright (c) 1998-2003 The Mozilla Organization. A copy of the Mozilla Public License is at <http://www.mozilla.org/MPL/> . FFTW, The Fastest Fourier Transform in the West, Copyright (c) 1997-1999 Massachusetts Institute of Technology. All rights reserved.

The following third-party libraries are used by the NlogN Momentum solver:

"This program includes Metis 4.0, Copyright © 1998, Regents of the University of Minnesota", <http://www.cs.umn.edu/~metis> , METIS was written by George Karypis (karypis@cs.umn.edu).

Intel® Math Kernel Library, <http://www.intel.com/software/products/mkl>

SuperLU_MT version 2.0 - Copyright © 2003, The Regents of the University of California, through Lawrence Berkeley National Laboratory (subject to receipt of any required approvals from U.S. Dept. of Energy). All rights reserved. SuperLU Disclaimer: THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF

SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

7-zip - 7-Zip Copyright: Copyright (C) 1999-2009 Igor Pavlov. Licenses for files are: 7z.dll: GNU LGPL + unRAR restriction, All other files: GNU LGPL. 7-zip License: This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA. unRAR copyright: The decompression engine for RAR archives was developed using source code of unRAR program. All copyrights to original unRAR code are owned by Alexander Roshal. unRAR License: The unRAR sources cannot be used to re-create the RAR compression algorithm, which is proprietary. Distribution of modified unRAR sources in separate form or as a part of other software is permitted, provided that it is clearly stated in the documentation and source comments that the code may not be used to develop a RAR (WinRAR) compatible archiver. 7-zip Availability: <http://www.7-zip.org/>

AMD Version 2.2 - AMD Notice: The AMD code was modified. Used by permission. AMD copyright: AMD Version 2.2, Copyright © 2007 by Timothy A. Davis, Patrick R. Amestoy, and Iain S. Duff. All Rights Reserved. AMD License: Your use or distribution of AMD or any modified version of AMD implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA. Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. AMD Availability: <http://www.cise.ufl.edu/research/sparse/amd>

UMFPACK 5.0.2 - UMFPACK Notice: The UMFPACK code was modified. Used by permission. UMFPACK Copyright: UMFPACK Copyright © 1995-2006 by Timothy A. Davis. All Rights Reserved. UMFPACK License: Your use or distribution of UMFPACK or any modified version of UMFPACK implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License

as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included.

UMFPACK Availability: <http://www.cise.ufl.edu/research/sparse/umfpack> UMFPACK (including versions 2.2.1 and earlier, in FORTRAN) is available at <http://www.cise.ufl.edu/research/sparse> . MA38 is available in the Harwell Subroutine Library. This version of UMFPACK includes a modified form of COLAMD Version 2.0, originally released on Jan. 31, 2000, also available at <http://www.cise.ufl.edu/research/sparse> . COLAMD V2.0 is also incorporated as a built-in function in MATLAB version 6.1, by The MathWorks, Inc. <http://www.mathworks.com> . COLAMD V1.0 appears as a column-preordering in SuperLU (SuperLU is available at <http://www.netlib.org>). UMFPACK v4.0 is a built-in routine in MATLAB 6.5. UMFPACK v4.3 is a built-in routine in MATLAB 7.1.

Qt Version 4.6.3 - Qt Notice: The Qt code was modified. Used by permission. Qt copyright: Qt Version 4.6.3, Copyright (c) 2010 by Nokia Corporation. All Rights Reserved. Qt License: Your use or distribution of Qt or any modified version of Qt implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the

terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission."

Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. Qt Availability: <http://www.qtsoftware.com/downloads> Patches Applied to Qt can be found in the installation at:

\$HPEESOF_DIR/prod/licenses/thirdparty/qt/patches. You may also contact Brian Buchanan at Agilent Inc. at brian_buchanan@agilent.com for more information.

The HiSIM_HV source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code, is owned by Hiroshima University and/or 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 documentation 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 U.S. Government Restricted Rights. Software and technical data rights granted to the federal government include only those rights customarily provided to end user customers. Agilent provides this customary commercial license in Software and technical data pursuant to FAR 12.211 (Technical Data) and 12.212 (Computer Software) and, for the Department of Defense, DFARS 252.227-7015 (Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial Computer Software or Computer Software Documentation).

Customizing the ADS Environment	7
Customizing Environment Variables	7
Customizing Configuration Variables	7
How a Configuration is Determined	11
Special Variables	12
Viewing Details of the Current Configuration	16
Typical Variables	18
Setting Design Environment Preferences	35
Workspace Preferences	35
Library-Specific Preferences	35
Specifying Design Entry and Display Preferences	35
Specifying Layer Definitions	59
Customizing Keyboard Shortcuts	60
Configuring Toolbars	61
Creating a Custom Component Palette	63
Setting Layout Options	66
Defining Layers	66
Preferences for Layout	70
Saving and Reading Preference Files	87
Toggling the Coordinate Readout Display	88
Pick and Place Report	89
Customization Examples	96
Creating a Custom Menu	96
Adding Online Help for User-Defined Items	97
Adding Custom Documentation for Online Access	100

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).
ADS_LICENSE_FILE	\$HPEESOF_DIR/licenses/license.lic	Specifies the location of the FLEXIm 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 <i>hpeedit</i> program.
EESOF_BROWSER	Not set by default. Example setting: EESOF_BROWSER=/usr/bin/firefox	Specifies the browser to use for viewing online help.
EESOF_REMOTE_DISPLAY	AUTO	Specifies if Remote Display mode is set to automatic, on, or off. Setting to <code>AUTO</code> provides the default behavior. Setting to <code>YES</code> (or <code>1</code>) turns on Remote Display mode. Any other value turns off Remote Display mode. Remote Display mode is only available on UNIX. For details, see <i>Using a Remote Display (install)</i> .

 **Note**

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

Customizing Configuration Variables

The default values of the configuration variables can help you to get started and you can customize them 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 workspace directory - Define configuration variables here that apply only to this workspace.

- Your personal directory = `$HOME/hpeesof/config`
Define configuration variables here that apply to all your workspaces.

**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 `$HPEESOF_DIR/custom/config`
Define configuration variables here that apply to all workspaces of all users on a site-wide basis. Configuration variables defined here will not be overwritten by installation of subsequent program patches or updates.
- The ADS installation directory `$HPEESOF_DIR/config`
Define the default installed configuration variables here that apply to all workspaces 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 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
Datasets	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>
```

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 <i>name</i> in the environment.
%name	The configuration variable is replaced by the text string assigned to <i>name</i> 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 workspace 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:

- 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 workspace directory

As noted earlier, configurations in the user directory take precedence over the installation directory, and configurations in the workspace directory take precedence over the user directory. This is because the search order for configuration information is: workspace 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 workspace-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 *{\$HPEESOF_DIR}/config/hpeecolor.cfg*, but can be overridden at the site, user, or workspace 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	: salmon	: 6
				: 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.

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.

**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:

```
# =====
#   R   G   B   :   Color Name   :   Plotter Pen #
#   --- --- ---   -----   -----
#   0   0   0   :   black   :   1
# 255   0   0   :   red   :   2
#   0   255   0   :   greeen   :   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
#
# For Unix X Windows systems either RGB values or standard
# X Color Names, or both, may be specified for color entries.
# For Windows95/NT only RGB values are supported.
# Note, if both RGB and Color Name values are specified,
# the RGB values take precedence.
# =====
#
0   0   0   :   black   : 0
255   0   0   :   red   : 1
255 255   0   :   yellow   : 2
```

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 worspace 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
/fill/pat/path/pat2.pattern : 3 50 30
/fill/pat/path/pat3.pattern : 4 60 45
/fill/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:

```

# =====
# EEs of 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 workspace

To launch the Configuration Explorer:

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

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.

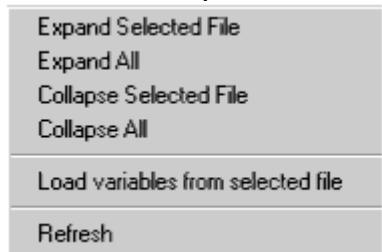
```

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.



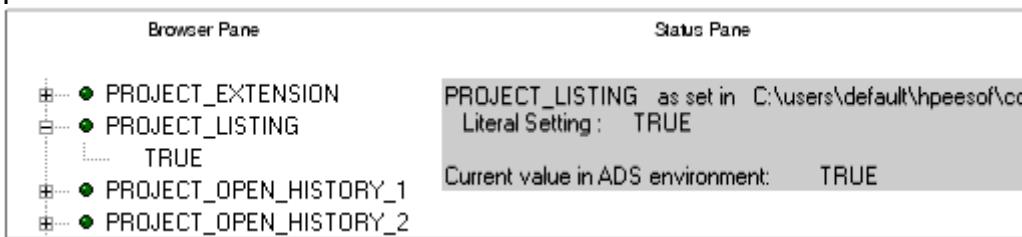
- Use the uppermost group of commands to quickly expand and collapse the tree.
- Use *Load variables from selected file* to explicitly load into memory changes you have made to a configuration file since launching ADS.
- Use *Refresh* to view changes you have made to a configuration file since launching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



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.

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 workspaces, all workspaces, 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 workspace 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)
PREFERENCES_PATH Specifies the preference file search path. Example: PREFERENCES_PATH = ./:{\$HPEESOF_DIR}/de/defaults	Path
LOCAL_AEL Specifies the workspace-specific AEL files to be automatically loaded each time when a workspace is opened. It should be set only in the <i>de_sim.cfg</i> file located in the workspace directories. These AEL filenames may have absolute, relative, or no path specified, but since the files are generally located in the workspace directory, a relative path would be used beginning with a dot and slash (./). An AEL file that is already in the <i>networks</i> subdirectory will be automatically loaded. Any other AEL files that you would like included should be placed here. Example: LOCAL_AEL = my_file	Filenames (without extension) or Path
USER_AEL Specifies the user-specific AEL files to be automatically loaded once at startup. It should be set only in the <i>de_sim.cfg</i> file located in the user's personal customized directory <code>\$HOME/hpeesof/config</code> . 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 <code>\$HOME</code> to reference files in the user's home directory. Example: USER_AEL = \$HOME/hpeesof/MyLibrary/ael/	Filenames (optional extension) or Path
SITE_AEL Specifies the site-specific AEL files to be automatically loaded once at startup. It is typically set in the <i>de_sim.cfg</i> file located in <code>\$HPEESOF_DIR/custom/config</code> . 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\	Filenames (optional extension) or Path
HPEESOF_LANGUAGE Specifies the language used by the PDE program. Example: HPEESOF_LANGUAGE = english	Language name
ENGLISH_MSG_EXT Specifies the language of the program messages. Example: english_MSG_EXT = eng	English = eng German = ger French = fra
ENGLISH_MSG_FONT Specifies the font supporting the selected local language. Example: ENGLISH_MSG_FONT = 9x15 bold	European = 9x15 Japanese = 8x16
PDE_MSG_FILE Specifies the full path to the PDE message directory file. Example: PDE_MSG_FILE = {\$HPEESOF_DIR}/msgs/english/PDE/PDE.msg	Path
NETLIST_FILE_NAME Specifies the netlist filename. Example: NETLIST_FILE_NAME = netlist.log	Filename
EGSLIST_FILE_NAME	Filename

Specifies the filename for the EGS translator option. Example: EGSLIST_FILE_NAME = ./egslist.log	
EGS_OPTION_FILE: Specifies the path to the EGS translator option files. Example: EGS_OPTION_FILE = ./:{\$HPEESOF_DIR}/config	Path
GDSII_OPTION_FILE Specifies the path to the GDSII option files. Example: GDSII_OPTION_FILE = ./:{\$HPEESOF_DIR}/config	Path
MASK_OPTION_FILE Specifies the path to the mask translator option files. Example: MASK_OPTION_FILE = ./:{\$HPEESOF_DIR}/config	Path
IGES_OPTION_FILE Specifies the path to the IGES translator option files. Example: IGES_OPTION_FILE = ./:{\$HPEESOF_DIR}/config	Path
SPICE_OPTION_FILE Specifies the search path for the Spice files. Example: SPICE_OPTION_FILE = ./:{\$HPEESOF_DIR}/config	Path
IFF_OPTION_FILE Specifies the search path for the IFF option files. Example: IFF_OPTION_FILE = ./:{\$HPEESOF_DIR}/config	Path
DIALOG_TIME_OUT Specifies the time before the copyright or information dialog box is automatically dismissed. Example: DIALOG_TIME_OUT = 10000	Time in milliseconds
DOUBLE_CLICKS_TIME_OUT Specifies the time interval in which a double-click is registered. Example: DOUBLE_CLICKS_TIME_OUT = 200	Time in milliseconds
QUERY_PROCESS_STATUS Specifies how often PDE should query the child process status. Example: QUERY_PROCESS_STATUS = 1	Time in seconds
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	Time in seconds
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	Report types
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	Report types
BOM_ITEM Specifies the spacing allocated for item name field in the Bill of Materials dialog box. Example: BOM_ITEM = 30	Integer
BOM_QTY Specifies the spacing allocated for quantity field in the Bill of Materials dialog box. Example: BOM_QTY = 7	Integer
BOM_DESC	Integer

Specifies the spacing allocated for description field in the Bill of Materials dialog box. Example: BOM_DESC = 50	
TEXT_EDITOR_PROGRAM_NAME Text editor Example: TEXT_EDITOR_PROGRAM_NAME = write.exe	Editor name
INVOKE_DDS_ON_BOOTUP Program Preference. Example: INVOKE_DDS_ON_BOOTUP = FALSE	TRUE or FALSE
INVOKE_LIB_SERVER_ON_BOOTUP Program Preference. Example: INVOKE_LIB_SERVER_ON_BOOTUP = FALSE	TRUE or FALSE
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	Integer
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	Integer
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	Integer
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	Integer
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	Integer
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	Integer
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	Integer
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	Integer
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	Integer
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	Integer
LAYOUT_COMPONENT_HISTORY_LIST_WIDTH Width of the component history drop-down list on the Layout	Integer

window toolbar. Example: LAYOUT_COMPONENT_HISTORY_LIST_WIDTH = 7	
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	Integer
LAYER_LIST_WIDTH Width of the drop-down layer list on the Layout window toolbar. Example: LAYER_WIDTH = 14	Integer
SIMULATION_HOST_LIST Remote simulation host names Example: SIMULATION_HOST_LIST = machine.company.com	Host names
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	Integer
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	Integer Min. Width = 340 Min. Height = 300
SCHEMATIC_WINDOW_X_LOC, SCHEMATIC_WINDOW_Y_LOC 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	Integer
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	Integer Min. Width = 760 Min. Height = 500
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	Integer
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	Integer Min. Width = 760 Min. Height = 500
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	TRUE or FALSE
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	TRUE or FALSE
ELW_WINDOW_AUTO_REDRAW DESIGN When True, the current design representation's schematic or	TRUE or FALSE

layout window will be redrawn whenever the Layers window changes the design representation's layers information. Example: ELW_WINDOW_AUTO_REDRAW DESIGN = TRUE	
ELW_WINDOW_INITIAL_X_LOC, ELW_WINDOW_INITIAL_Y_LOC 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	Integer
ELW_WINDOW_INITIAL_WIDTH, ELW_WINDOW_INITIAL_HEIGHT 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	Integer
SAVE_PROJECT_STATE Sets a default to be used for any project for which no <i>save_project_state.ael</i> file exists. This file is created/modified when you change worspaces or exit the program and the <i>Save Workspace State on Exit</i> 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 worspace. Note that only windows with named designs are saved; untitled windows are not saved. Example: SAVE_PROJECT_STATE = TRUE	TRUE, FALSE
COMMAND_DIALOG_WIDTH Sets the width, in pixels, of the Command Line dialog box (<i>Options > Command Line</i> in the Main window). The default is 400; a wider setting may be helpful for viewing lengthy AEL functions. Example: COMMAND_DIALOG_WIDTH = 400	Integer
DATA_FILES Specifies the search path for data files. Example: DATA_FILES = ./data:{\$HOME}/hpeesof/my_datafiles	Path
OPEN_PROJECT_AUTO_SCHEMATIC Sets a default for all worspaces. When TRUE, an empty Schematic window is automatically displayed when you create a workspace. Example: OPEN_PROJECT_AUTO_SCHEMATIC = TRUE	TRUE, FALSE
OPEN_PROJECT_AUTO_LAYOUT Sets a default for all worspaces. When TRUE, an empty Layout window is automatically displayed when you create a workspace. Example: OPEN_PROJECT_AUTO_LAYOUT = FALSE	TRUE, FALSE
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	Standard or custom palette name, exactly as it appears in program.
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	Standard or custom palette name, exactly as it appears in program.
SCHEMATIC_DSP_DEFAULT_PALETTE Sets the specified palette as the default palette for DSP designs in the Schematic window.	Standard or custom palette name, exactly as it appears

Example: SCHEMATIC_DSP_DEFAULT_PALETTE = Timed Linear	in program.
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	Standard or custom palette name, exactly as it appears in program.
OPEN_DDS_AFTER_SIM Establishes a default state for whether or not to automatically 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	TRUE or FALSE
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 <i>Environment</i> variables Example: CONFIG_EXPLORER_VARS = E_HPEESOF_DIR C_ For <i>Configuration</i> variables Example: CONFIG_EXPLORER_VARS = C_SITE_AEL S_ For <i>System</i> variables (PC only) Example: CONFIG_EXPLORER_VARS = S_TEMP	E_<VariableName>, C_<VariableName>, S_<VariableName>
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	Yes or No
DEFAULT_HIERARCHY_POLICY Change the system's default hierarchy policy. The hierarchy policy should be of the form library:policy without the .pol extension Example: DEFAULT_HIERARCHY_POLICY=MyLibrary:CustomPolicy	Custom policy name

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)
HPEESOF_LM_LICENSE_FILE Complete path to FLEXIm license.lic file. Example: HPEESOF_LM_LICENSE_FILE = {\$HPEESOF_DIR}/license/license.lic	Filename
HPEESOF_COLOR Complete path to system-wide color configuration file. Example: HPEESOF_COLOR = {\$HPEESOF_DIR}/config/hpeecolor.cfg	Filename
HPEESOF_FILL Complete path to system-wide fill pattern configuration file. Example: HPEESOF_FILL = {\$HPEESOF_DIR}/config/hpeefill.cfg	Filename
HPEESOF_FONT_FILE Directory where TrueType raster fonts are located. Example: HPEESOF_FONT_FILE = {\$HPEESOF_DIR}/config	Directory
HPEESOF_OCCIDENTAL_FILE Complete path to European Hershey vector fonts. Example: HPEESOF_OCCIDENTAL_FILE = {\$HPEESOF_DIR}/config/hersh.oc	Filename
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
AGILEESOFT_LICPREF Controls bundle selection for all systems on a network with access to the license server. Example: AGILEESOFT_LICPREF = bundle_a	License bundle name
AGILEESOFT_LICPREF_<hostname> Controls bundle selection specifically for the system identified by <hostname>. Example: AGILEESOFT_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
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_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
API_NO_TEAR_OFF_MENU Turn on/off Motif's tear-off menu Example: API_NO_TEAR_OFF_MENU = 0	0=allow tear-off menu 1= do not allow tear-off menu

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)
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/adspptolemy/beta will be appended to load beta models. Example: ADSPTOLEMY_MOD_PATH = \$HOME/hpeesof/adspptolemy: \$ADSPTOLEMY_MODEL_PATH:\$HPEESOF_DIR/adspptolemy	Path
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	Directory
MATLABCMD If the command to invoke Matlab is not <i>matlab</i> , 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"	Command

Variables in the File hpeesofbrowser.cfg

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

Variable, description, and example	Value(s)
HPEESOF_BROWSER_WINDOW_X_LOCATION 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	Integer
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	Integer
HPEESOF_BROWSER_WINDOW_WIDTH The default width of the Component Library window. Example: HPEESOF_BROWSER_WINDOW_WIDTH = 600	Integer
HPEESOF_BROWSER_WINDOW_HEIGHT The default height of the Component Library window. Example: HPEESOF_BROWSER_WINDOW_HEIGHT = 400	Integer
HPEESOF_BROWSER_TIME_OUT The default time-out for warning/information messages. Example: HPEESOF_BROWSER_TIME_OUT = 10000	Time in milliseconds
HPEESOF_BROWSER_AEL_PATH Path for the browser AEL files. Example: HPEESOF_BROWSER_AEL_PATH = \$HOME/hpeesof/hpeesofbrowser/ael:\$HPEESOF_DIR/hpeesofbrowser/ael	Path
HPEESOF_BROWSER_AEL Top level AEL file Example: HPEESOF_BROWSER_AEL = lbb_main	Filename
HPEESOF_BROWSER_PATH Path for the control, records and Device Libraries Binary File. Path can only have System variables. Example: HPEESOF_BROWSER_PATH =	Path

\$HOME/hpeesof/hpeesofbrowser/records; \$COMPL_DIR/ComponentLibs/records	
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	Path
HPEESOF_BROWSER_LIBRARIES_FIELD_WIDTH The default width of the component library tree structure. Example: HPEESOF_BROWSER_LIBRARIES_FIELD_WIDTH = 30	Integer
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	0 = False 1 = True
HPEESOF_BROWSER_PARTS_PLACEMENT_WIDTH The default width of the Placement column. Example: HPEESOF_BROWSER_PARTS_PLACEMENT_WIDTH = 15	Integer
HPEESOF_BROWSER_COMPONENT_WEBSITE_VISIBLE The default visibility status of the Website column. Example: HPEESOF_BROWSER_COMPONENT_WEBSITE_VISIBLE = 0	0 = False 1 = True
HPEESOF_BROWSER_PARTS_WEBSITE_WIDTH The default width of the Website column. Example: HPEESOF_BROWSER_PARTS_WEBSITE_WIDTH = 20	Integer
HPEESOF_BROWSER_COMPONENT_AVAILABILITY_VISIBLE The default visibility status of the Availability column. Example: HPEESOF_BROWSER_COMPONENT_AVAILABILITY_VISIBLE = 0	0 = False 1 = True
HPEESOF_BROWSER_PARTS_AVAILABILITY_WIDTH The default width of the Availability column. Example: HPEESOF_BROWSER_PARTS_AVAILABILITY_WIDTH = 10	Integer

HPEESOF_BROWSER_COMPONENT_LICENSE_VISIBLE The default visibility status of the License column. Example: HPEESOF_BROWSER_COMPONENT_LICENSE_VISIBLE = 0	0 = False 1 = True
HPEESOF_BROWSER_PARTS_LICENSE_WIDTH The default width of the License column. Example: HPEESOF_BROWSER_PARTS_LICENSE_WIDTH = 10	Integer
ADS_COMPONENT_SORT_ORDER The sort order for components. Example: ADS_COMPONENT_SORT_ORDER = Default	Default Ascending Descending
ADS_ANALOGRF_LIBRARY_SORT_ORDER The sort order for the analogRF library. Example: ADS_ANALOGRF_LIBRARY_SORT_ORDER = Default	Default Ascending Descending
ADS_HPTOLEMY_LIBRARY_SORT_ORDER The sort order for the Ptolemy library. Example: ADS_HPTOLEMY_LIBRARY_SORT_ORDER = Ascending	Default Ascending Descending
HPEESOF_BROWSER_COMPONENT_HIDE_OBSOLETE Hides obsolete ADS library components from view in the ADS library browser. Example: HPEESOF_BROWSER_COMPONENT_HIDE_OBSOLETE = 1	0 = False 1 = True
HPEESOF_BROWSER_COMPONENT_HIDE_UNLICENSED Hides unlicensed ADS library components from view in the ADS library browser. Example: HPEESOF_BROWSER_COMPONENT_HIDE_UNLICENSED = 1	0 = False 1 = True
ADS_BROWSER_DEFAULT_WEBSITE Default website for ADS library records that do not have a website defined. Example: ADS_BROWSER_DEFAULT_WEBSITE = http://eesof.tm.agilent.com/partners/vendor_libraries.html	URL String
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}	Path
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	Path
HP_TEMPLATE_BROWSER_PATH Path information for the DDS template files. Each entry is in the form of <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};	String Path

Variables in the File hpeesofdds.cfg

This configuration file is used to configure data displays.

Variable, description, and example	Value(s)
AEL_PATH Top level AEL file Example: AEL_PATH = \$HOME/hpeesof/dds/ael/:\$HPEESOF_DIR/custom/dds/ael /:\$HPEESOF_DIR/dds/ael	Path
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	Filename
DDS_WIN_FOREGROUND Window foreground color Example: DDS_WIN_FOREGROUND = 0	Integer
DDS_WIN_BACKGROUND Window background color Example: DDS_WIN_BACKGROUND = 8	Integer
DDS_WIN_HEIGHT Window height Example: DDS_WIN_HEIGHT = 510	Integer
DDS_WIN_WIDTH Window width Example: DDS_WIN_WIDTH = 655	Integer
DDS_HIGHLIGHT_LINE_TYPE System-wide highlight line type Example: DDS_HIGHLIGHT_LINE_TYPE = 0 [= solid]	Integer
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	Integer

Example: DDS_DEFAULT_LINE_WIDTH = 7	
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 <i>n</i> _COLOR <i>n</i> th color, where <i>n</i> = 1 to DDS_NUMBER_OF_TRACE_COLORS Example: DDS_TRACE16_COLOR = 90	Integer

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 Location of supplied Momentum Substrate files. Example: SUPL_GF_DATABASE = {\$HPEESOF_DIR}/momentum/lib	Directory
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/momddsael	Filename
MOMMSH_REFACET_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_REFACET_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. Example: SMG_AEL_PATH={\$HOME}/hpeesof/smg/ael: {\$HPEESOF_DIR}/smg/ael: {\$HPEESOF_DIR}/ael_smg_ui	Path
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
DS_min_variables_for_grouping This is the required minimum number of dependent output variables before a more efficient dataset format will be used. Data in such datasets are read faster by the data display, but the simulator will take longer to create these datasets. Also, this dataset format is only efficient for large numbers of dependent variables; it is inefficient for small numbers, which is why this parameter exists. Setting this variable to zero will fully disable the creation of this new dataset format. The default is a minimum of 1000 dependent output variables. Example: DS_min_variables_for_grouping=1000	Nonnegative integer
rf_cw_priority This variable controls the order of license checking. By default, the simulator first looks for the "sim_linear" codeword first, and then, if that was not found, "sim_syslinear" would be used. Setting the value of this parameter to "y" will make the simulator look for "sim_syslinear" first and "sim_linear" second.	y n

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. Example: STATUS_LINES_SHOWN =100	Integer
SIMEXEC_MAP_TO_FRONT Sets the default behavior of the Status window, where <i>on</i> (the default) means the Status window pops to the front of the screen anytime new messages are displayed in it, and <i>off</i> means it does not pop forward automatically. Example: SIMEXEC_MAP_TO_FRONT = on	ON or OFF
HPEESOFSIM_BIN Path to the hpeesofsim executable. Relative pathnames are evaluated relative to the current workspace'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	Path

Setting Design Environment Preferences

You can easily customize many aspects of design entry and display through the **Preferences**, **Layer Preferences**, and **Technology** dialog boxes accessed through the **Options** menu of Schematic or Layout window. 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 workspace.

Workspace Preferences

Each workspace must contain a `schematic.prf` file (for the Schematic and Symbol window) and `layout.prf` file (for the Layout window). These `*.prf` files define the preferences for all designs in the workspace.

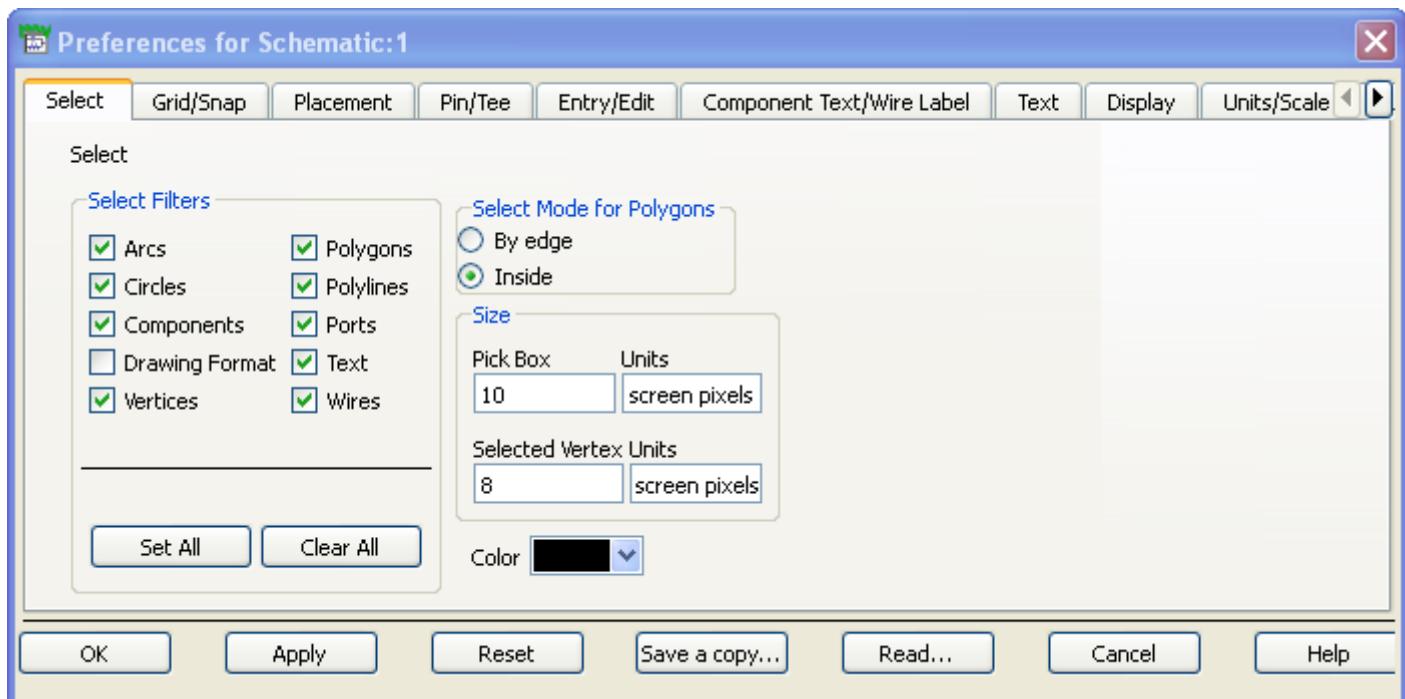
Library-Specific Preferences

Preferences that should be specific to libraries are contained in the `schematic.prf` or `layout.prf` located in the library directory. Preferences that are related to the units of the library or the technologies used are considered library-specific. Any value that is library-specific takes precedence over the value which is defined in the workspace. Those library-specific preferences are marked with an asterisk (*) in the Preferences dialog box. For example, if Snap Grid Distance is defined by the current library's `schematic.prf` or `layout.prf`, its value will take precedence over a value defined in the workspace's `schematic.prf` or `layout.prf` and will be displayed in the Preferences dialog box.

Specifying Design Entry and Display Preferences

To change design entry and display preferences:

Choose **Options > Preferences** in any design window.



Some options relating to the size of an item displayed on the screen may 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.

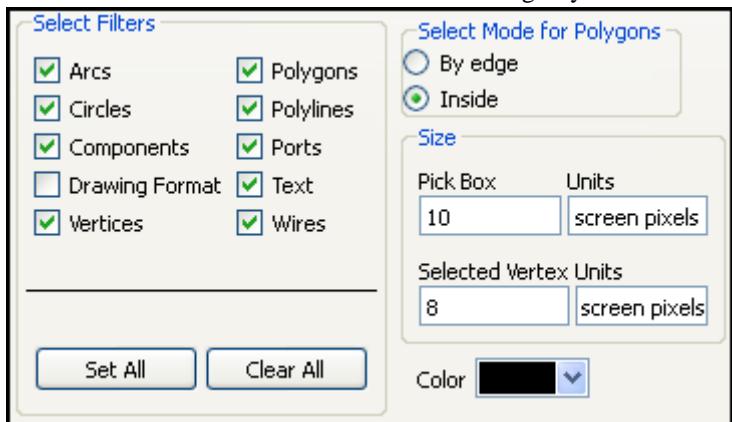
schem units—Use this setting to specify sizes in terms of inches, in the Schematic window. In the Layout window, select layout units to specify a size with respect to the design units of the Layout window.

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 to include or exclude 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.

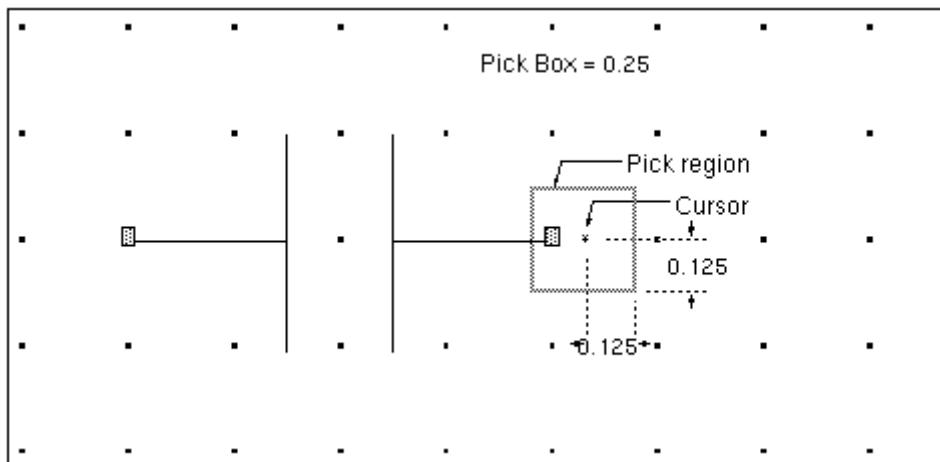
Changing the Select Mode for Polygons

- Choose *By Edge* to select polygons by clicking on the outer edge.
- Choose *Inside* if you want 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.

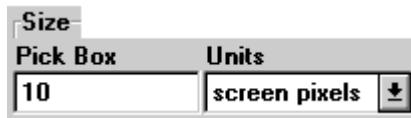
The following figure shows an example of the region defined by a pick box specified in inches.



Pick region based on schem Units

In the above figure, the pick box size is 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.



 **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

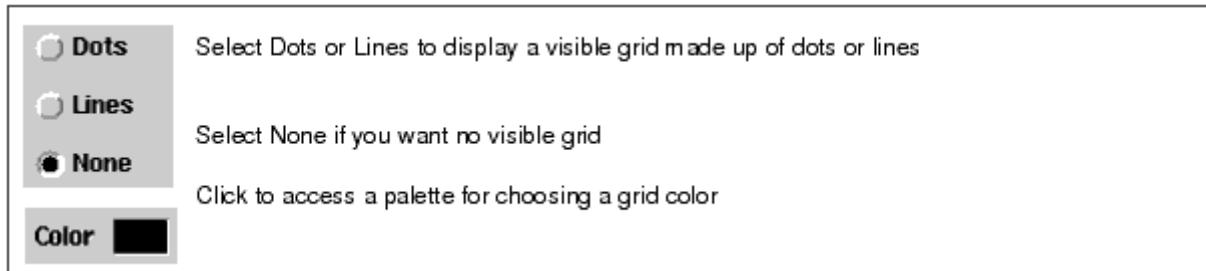
The display grid and cursor snap features are provided to assist you in creating and editing your designs more quickly and accurately.

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

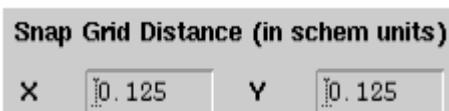
The default component symbols have been created uniformly, in 0.125-inch increments. Thus, if you keep the default settings while creating your design, you should be able to connect all symbols with minimal effort.

Display

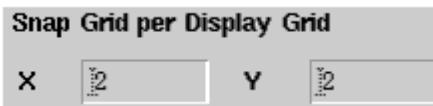


Spacing

- **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.



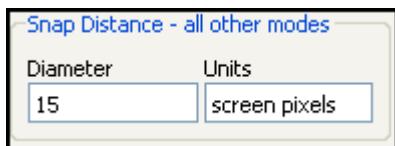
- **Snap Grid per Display Grid** represents the spacing between the dots (or lines) on the display grid in terms of a factor of the snap spacing. The smaller the number, the finer the grid.



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.



Active Snap Modes

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.

- **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.)

Toggling Repeatable Component Placement

- **Auto-repeatable component placement**

Enable/Disable the Auto-repeatable component placement option.

Setting Pin/Tee Options

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

- **Pin Size**

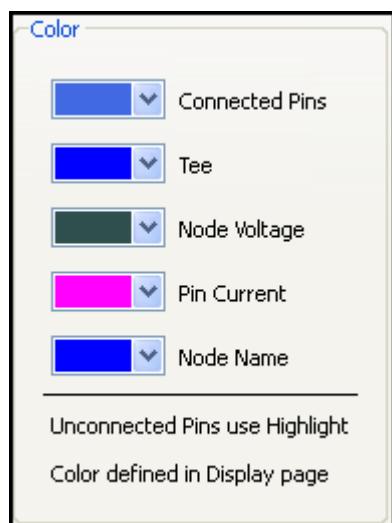
Enables you to change the size of the pins drawn on all components.

- **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

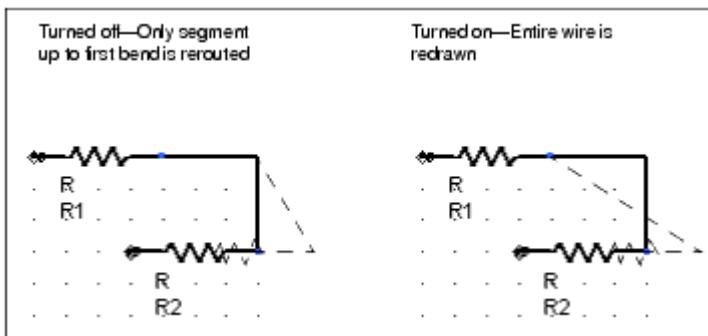
This group of options allows you to control several aspects of shape entry and editing including the angle at which lines and wires are drawn, the resolution of arcs and circles, and how wires are routed relative to component text.

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**

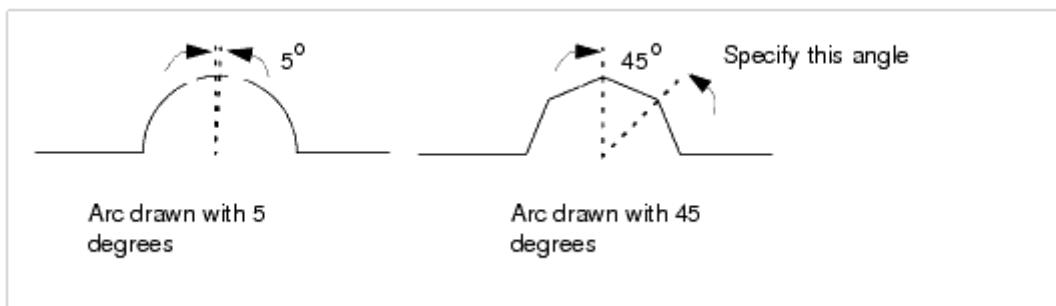
Prevents you from placing additional points on a polygon if overlapping lines will result.

- **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**

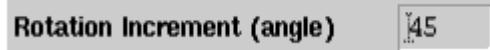
This option represents the maximum number of commands held in the stack.

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.

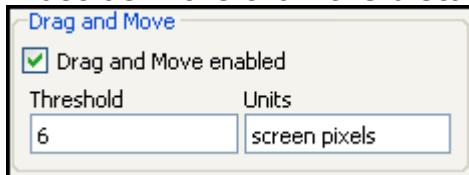


- **Move handles enabled**

This option toggles display of a move handle on objects. Move handles on an object are easy-to-spot locations which can be grabbed so that the object can be moved. They appear when the object is selected. If Drag and Move is not enabled, this feature still allows the object to be moved by the move handle.

- **Drag and Move enabled**

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* (usrguide).

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.



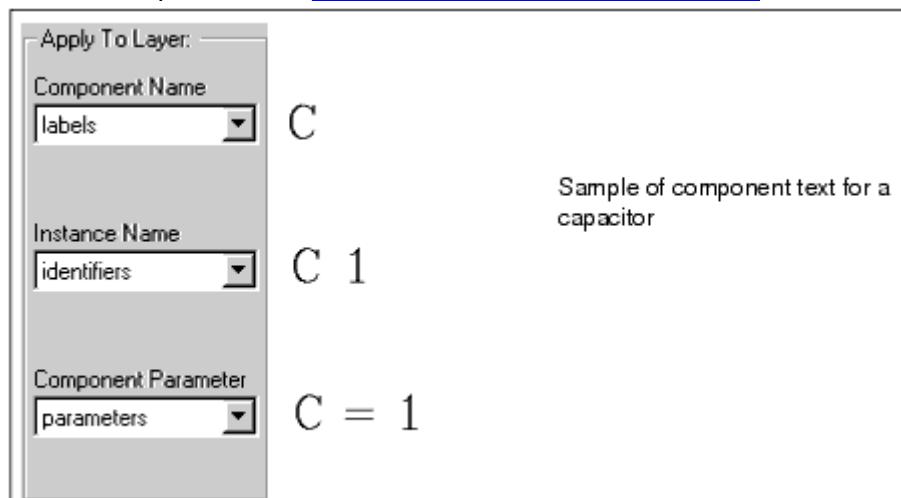
Note

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](#).



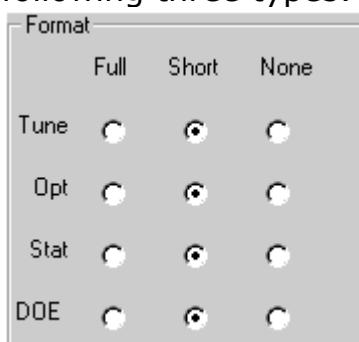
To change the layer for any given type of component text, click the arrow and select a new layer from the drop-down list.

- **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).

2. Choose **Options > Preferences > Text** and set the text attributes as desired.
 - **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 workspace.

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 workspace, the files `schematic.prf` (for schematics) and `layout.prf` (for layouts) are copied to the new workspace from the ADS installation directory. All designs in a given workspace use the preference files by these names.

The preference files are saved in following ways:

- When the workspace is closed, the files with the default filenames (`schematic.prf` and `layout.prf`) are saved. The values of these preference files apply to every design in the workspace.
- To save a copy of the current preferences, customize the preferences as needed, and save them to a preference file with a unique name.

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.

To customize preferences to the default filenames:

1. Choose **Options > Preferences**.
2. Change preferences as desired and click **OK**. The preference are updated and every design in the workspace will now use these preferences. The preference files with the default names will be saved when the workspace is closed.

To save a copy of the current preferences files to unique filenames:

1. Choose **Options > Preferences**.
2. Change any desired settings and click **Save a copy....** The Save Preferences File dialog box appears.
3. Enter a name of your choosing (the `.prf` extension is added automatically) and click **OK**.

Hint

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

To read a preference file:

1. Choose **Options > Preferences**.
2. Click **Read...** and choose a `.prf` file. The preferences from that `.prf` file are read and made current. When the workspace is closed, the values are saved to `schematic.prf`

or `layout.prf`.

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, 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
dseFixAllComponents	Default value for fixing all components in destination Schematic or Layout 0=Off 1=On
DSEL2Sreport	Show status report from Generate/Update Schematic

Advanced Design System 2011.01 - Customization and Configuration	
dsePrefLayoutLayer	Entry layer name for Generate/Update Layout command
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)
globalArcResolution	Resolution of arcs in degrees
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 1=Short 2=None
instTextFontName	Component text placed in design will use the font given here
instTextOptFormat	Component text format for opt syntax where 0=Full 1=Short 2=None
instTextPoint	Component text size in points
instTextRows	Number of rows of component text displayed in one column
instTextStatFormat	Component text format for stat syntax where 0=Full 1=Short 2=None
instTextTuneFormat	Component text format for tune syntax where 0=Full 1=Short 2=None
layoutIncr	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.

layoutPrec	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. 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.
layoutUnits	Layout 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.
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
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	Pin/Ground size (Layout only)
portSizeUnits	Pin/Ground size units (Layout only) 0=User units 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 1=Components 2=Wires 4=Polygons 8=Polylines 16=Paths 32=Text 64=Arcs 128=Circles 256=Pins 512=Drawing Formats 1024=Vertices 8192=Traces
selectMode	Select mode for polygons 0=By edge 1=Inside
selectPointSize	Size of selected vertex

selectPointUnits	0=User units 1=Screen pixels
setPasteOriginPopup	Show Set Paste Origin dialog for Copy command 0=Off 1=On
showConnectedLay	Show Connected Components in Layout 0=Off 1=On
showConnectedSchem	Show Connected Components in Schematic 0=Off 1=On
showFixedLay	Show Fixed Components in Layout 0=Off 1=On
showFixedSchem	Show Fixed Components in Schematic 0=Off 1=On
stepRepeatNumCols	Number of columns for Step And Repeat command
stepRepeatNumRows	Number of rows for Step And Repeat command
stepRepeatXspace	X spacing between columns in user units for Step and Repeat command
stepRepeatYspace	Y spacing between rows in user units for Step and 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)
traceTraverse	Insert Tee and Cross components during trace 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 0=meter 1=km 2=ft

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

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

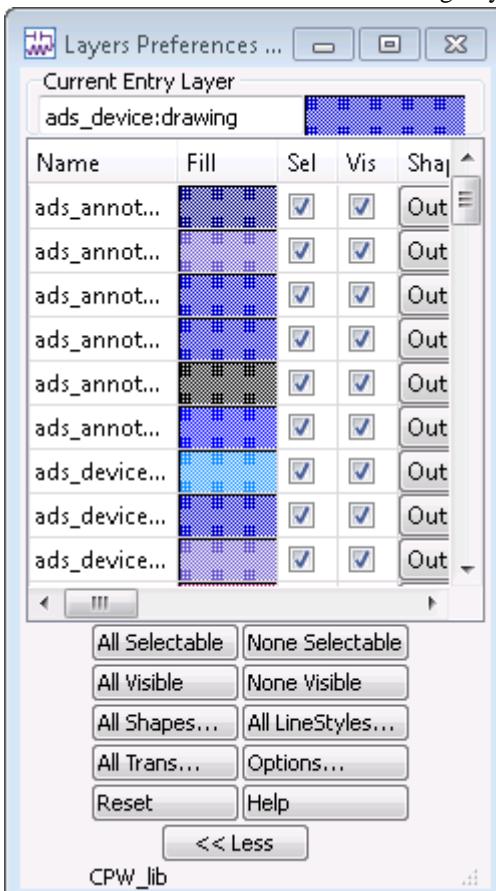
Specifying Layer Definitions

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 modify layer definitions, refer to *Layer Definitions*. (usrguide)

Changing the Current Entry Layer

To change layers:

1. Choose **Options > Layer Preferences** and the *Layer Preferences* (usrguide) window appears listing all the currently defined layers.



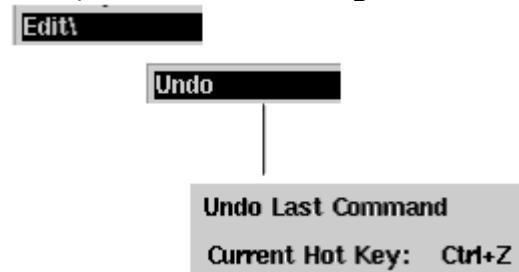
2. Click a layer that is appropriate for the task at hand.
3. Anything you draw now is drawn on this layer. The name of the layer is displayed in the status panel of the Schematic window.

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:

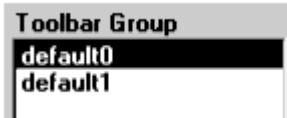
- A drop-down list for selecting the group of components you want to place on the palette (Palette List)
- A dynamically updated list of components you have placed in that window (Component History)
- In the Layout window, a drop-down list for selecting a different entry layer (Entry Layer List)
- Buttons 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*).

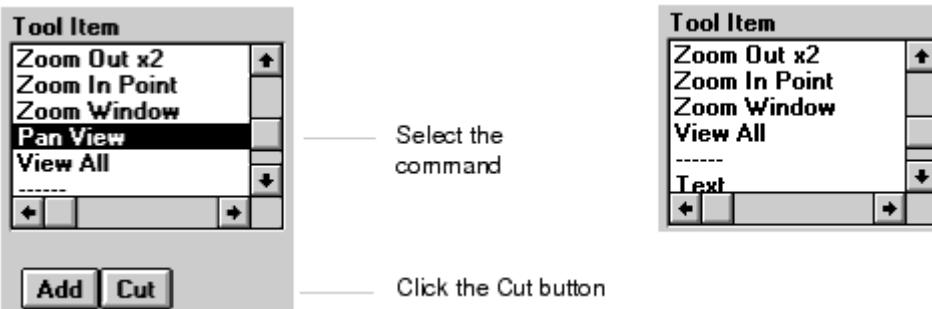
Customizing an Existing Toolbar

To reconfigure an existing toolbar:

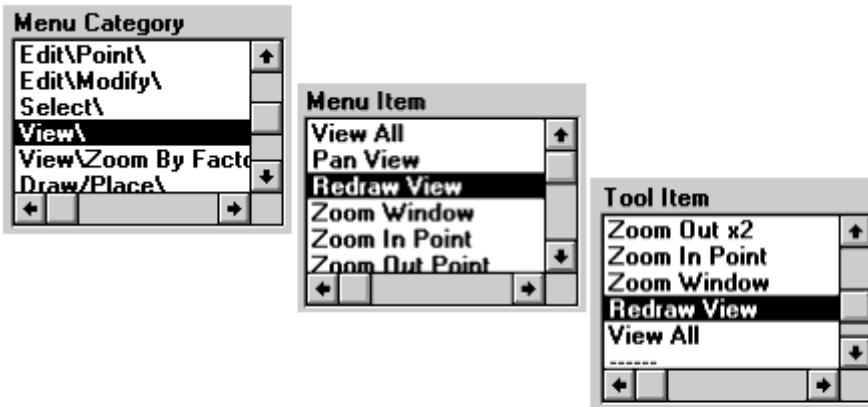
1. Choose **Tools > Hot Key/Toolbar Configuration** and click the Toolbar tab in the dialog box that appears.
2. In the Toolbar Group list box, select the name of the toolbar you want to change.



3. To change the display order, click the **Up** or **Down** buttons to move the selected toolbar.
4. 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.



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).

- 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:

- 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.

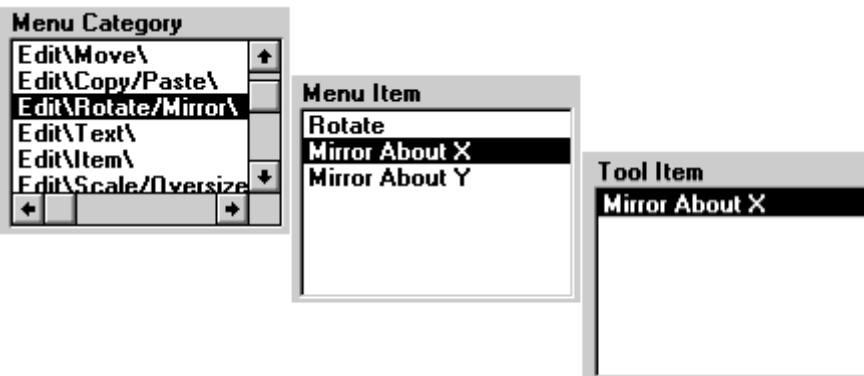
- Type a name in the Toolbar Name field and click **Add**.



The given name 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.



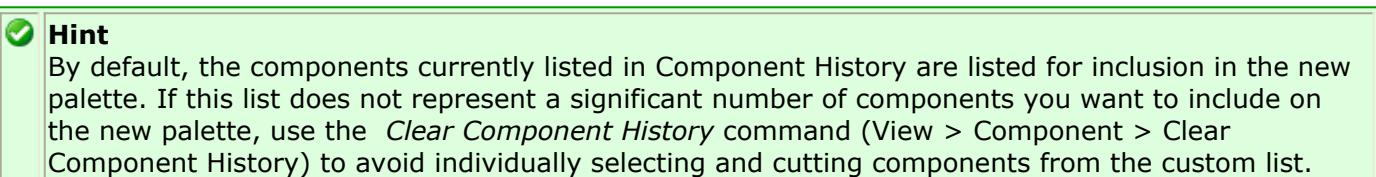
4. Click **Apply** to implement made changes to this toolbar.
5. Click **OK** to save changes to all toolbars.

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.



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_library_palette()` 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_library_palette` 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_library_palette` 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 statement which contains the `de_define_library_palette()` 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.

Positional  Differential

The coordinate readout bar consists of three segments: 'Positional' on the left, a central box containing '4.375, 4.000' and '1.125, 0.500' separated by a vertical line, and 'Differential' on the right.

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

Setting Layout Options

This section 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 workspace- 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.

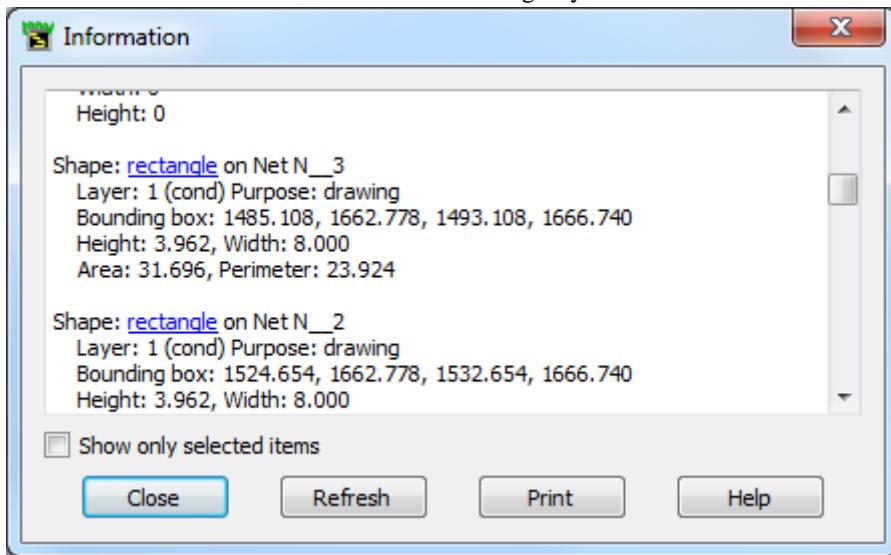
Often a single process mask layer is represented by a number of layers in a CAD program. For example, power and ground lines are often placed on the same mask for manufacturing, but on a different mask for CAD layout to distinguish them. Further, simple text notes and annotation can be placed on layers 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. 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 *Layer Definitions*. (usrguide))

Determining the Layer for an Item

To determine the layer for a specific item:

- Select the item and choose **Tools > Info**.
- Layer that a shape is on and layers used by instances are displayed.



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.

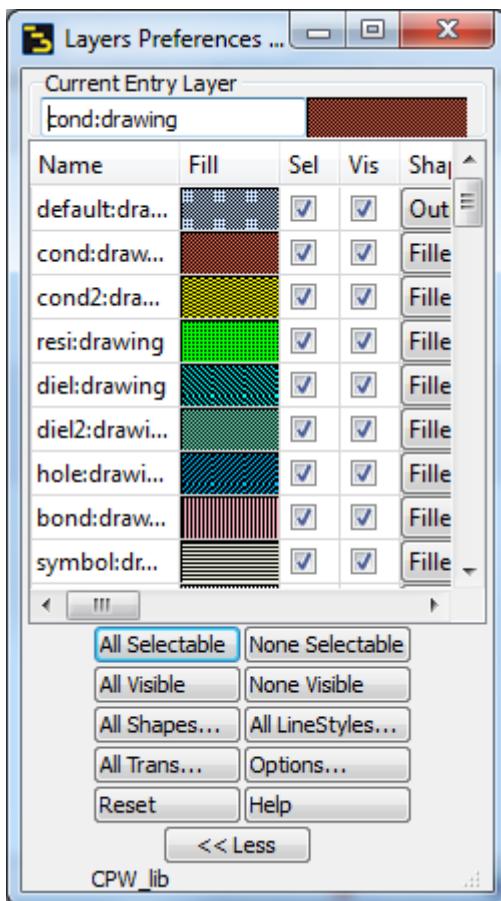
To set the current entry layer, choose one:

- Choose **Options > Layer Preferences**. Select a layer from the entry layer 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 Preferences Window

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

Any modifications made in the *Layers Preferences* window apply only to the current workspace. These modifications are stored in a .layerprf file with the format <libraryName>.layerprf and resides in the workspace directory. See *Setting Layer Preferences* (usrguide) for more information. To modify layers, purposes, and layer display properties of a library, use the *Layer Definitions* (usrguide) dialog box.



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.

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 Layers tab of the *Layer Definitions* (usrguide) 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 ports in addition to their point pins. Because of this, it is important to have your layer binding specified correctly.

For more details and examples, see *Layer Binding* (usrguide)

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 *Se/* 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.

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"
Diel1="diel"
Diel2="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.

Layers and Technologies

Each library should reference at least one technology. By default, "ads_standard_layers" is used as a standard technology for layout so that the layout libraries function properly. Inside this technology is the layer information. If a layer is missing, the artwork for an element can not be created. See *Technology Setup* (usrguide) for more information.

If layer *cond* is removed, the element's corresponding substrate element must be changed to specify a different layer number. For example, if layer *cond* is removed, 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.

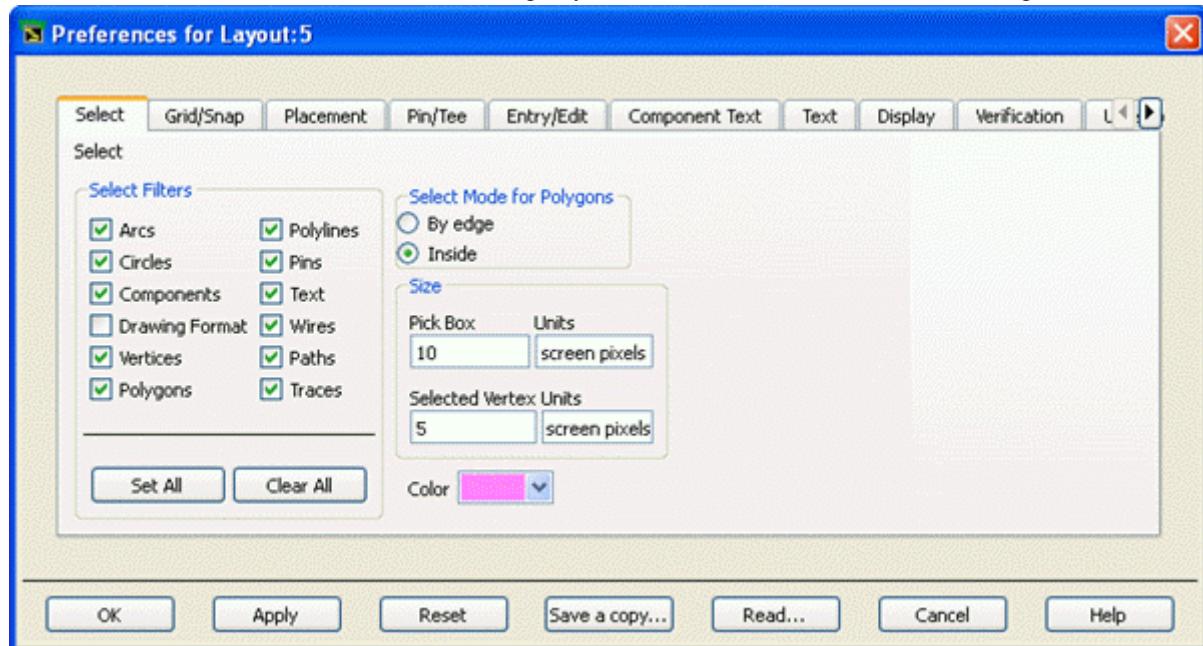
If placing a layout that uses a different technology is desired, refer to the section *Nested Technology* (usrguide).

Layout Units & Resolution

Layout units and resolution are contained in the technology of a library. See *Layout Units* (usrguide) for more information.

Preferences for Layout

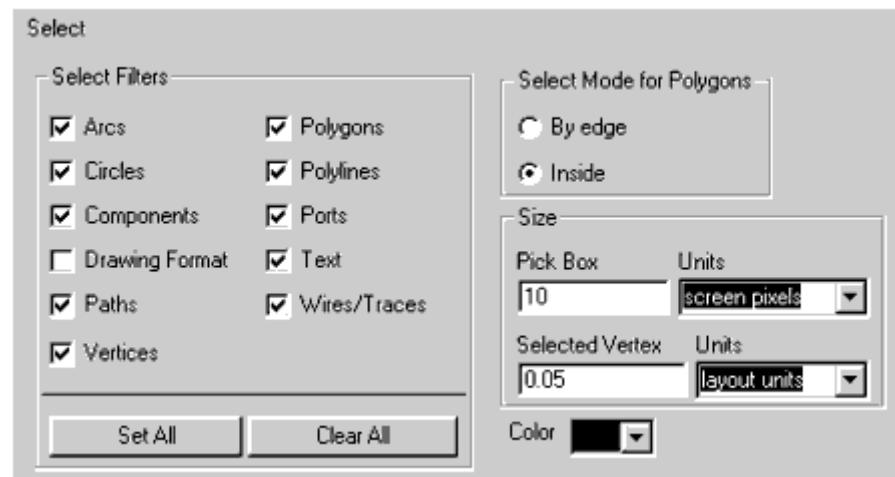
To access Preferences for Layout, select **Options > Preferences**.



Changing Select Options

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

Note
To use the selection filters, see *Using Selection Filters (custom)*.

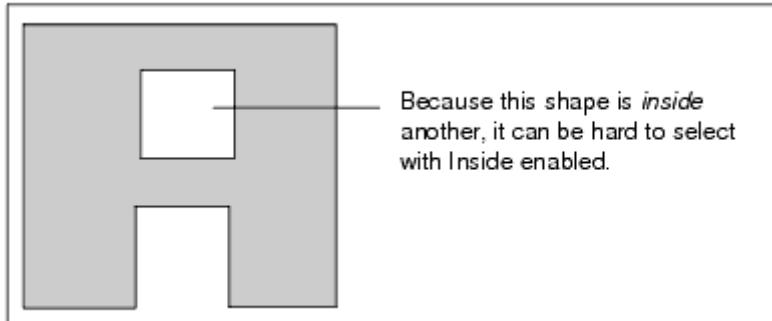


Setting How Polygons are Selected

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.

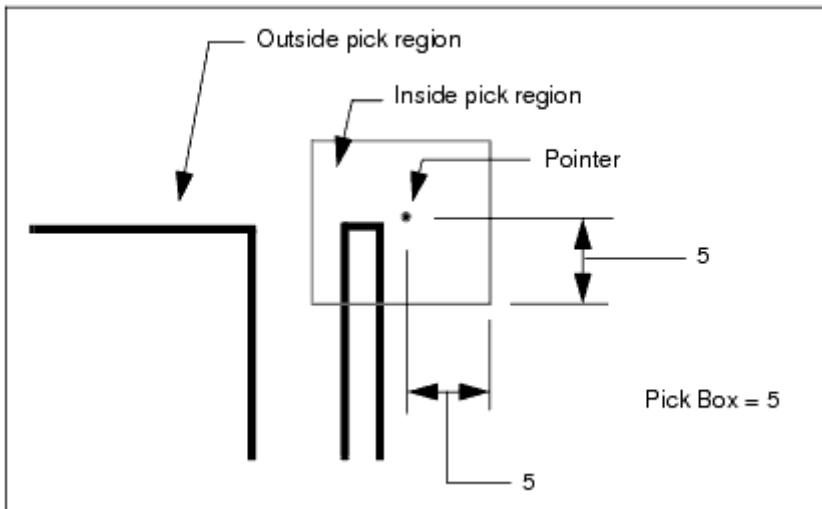


Setting Color for Selected Items

1. Choose the menu command **Options > Preferences**.
2. In the Preferences for Layout Dialog Box, choose the **Select** tab.
3. 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.



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

3. In the Size area, locate the Pick Box field.
4. Enter the size for the marker, and select the units.
 - *Screen pixel s* 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 Unit s* 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.

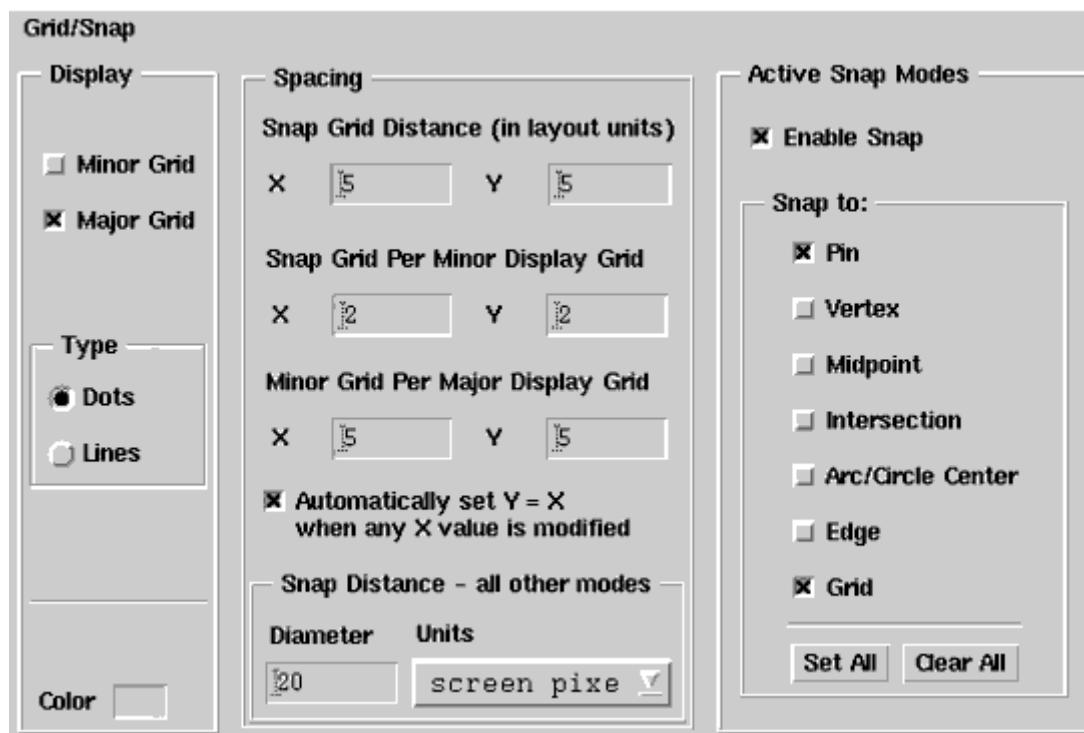
Setting the Size of Vertex Markers

A Vertex marker identifies a selected vertex.

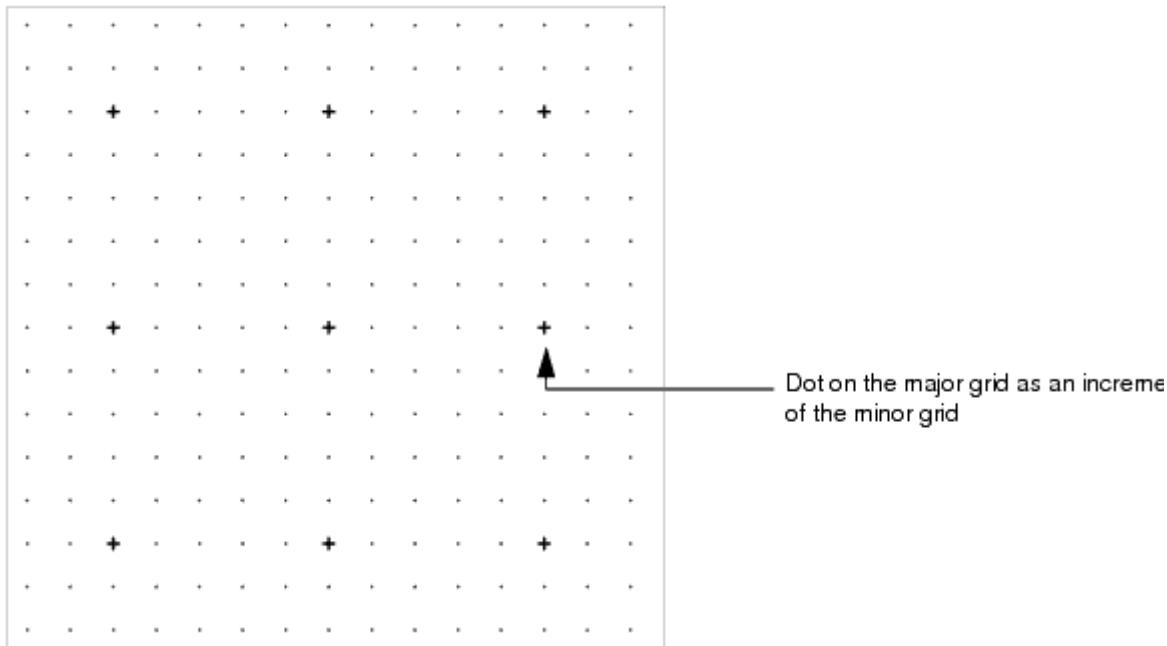
1. Choose the menu command **Options > Preferences**.
2. In the Preferences for Layout Dialog Box, choose the Select tab.
3. In the Size area, locate the Selected Vertex field.
4. Enter the size for the marker, and select the units.
 - *Screen pixel ls* specifies sizes in terms of pixels on the screen. 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*



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**.

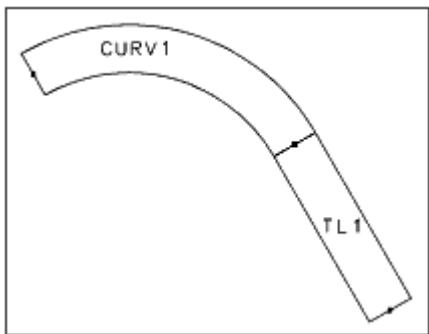
Setting Snap Modes

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	2
Midpoint	
Intersect	
Arc/Circle Center	
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 .

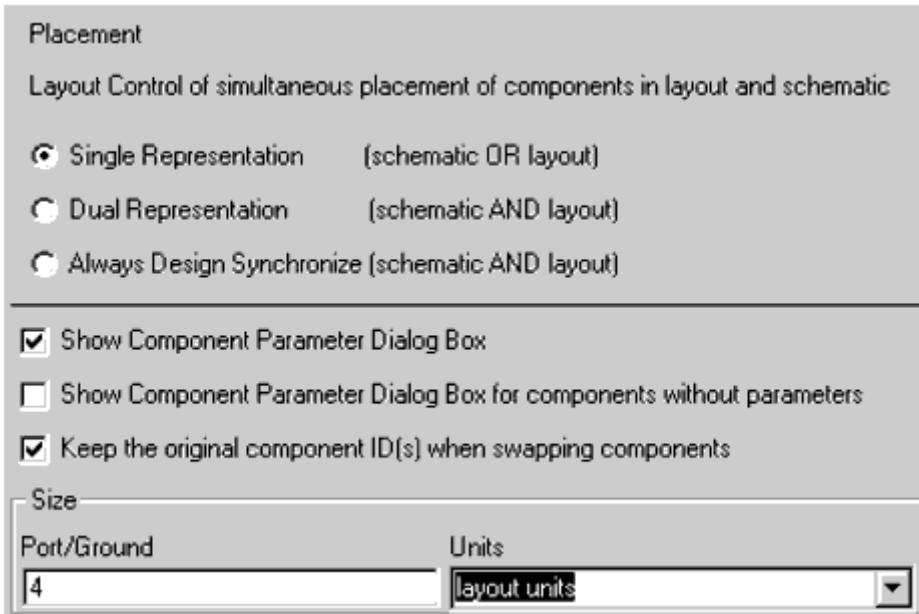


Hint

- 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.

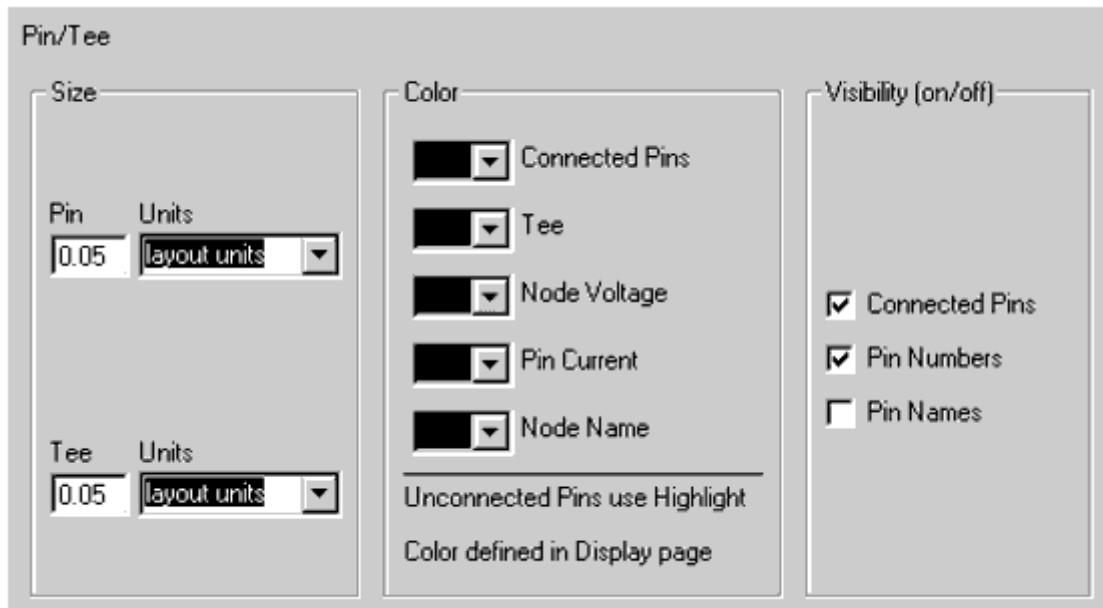
1. Choose the menu command **Options > Preferences**.
2. In the Preferences for Layout dialog box, choose the Placement tab.
3. Enable/disable the Auto-repeatable component placement option.
4. Click **Apply**.

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*.



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*.

- **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:

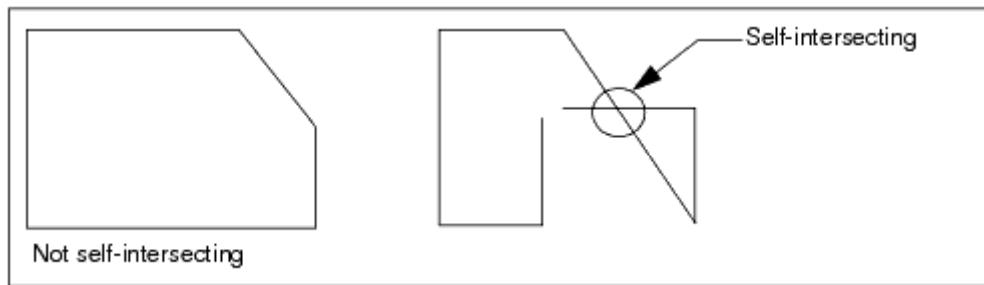
- Insert (Shape)—Polygon, Polyline, Rectangle, Circle, Arc (clockwise and counter-clockwise), Text, Construction Line, Symbol Pin, Path, Trace.
- Edit—Move Wire Endpoint, Mirror X, Mirror Y, Move & Disconnect, Step And Repeat, Set Origin, Move Component Text.

- **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**

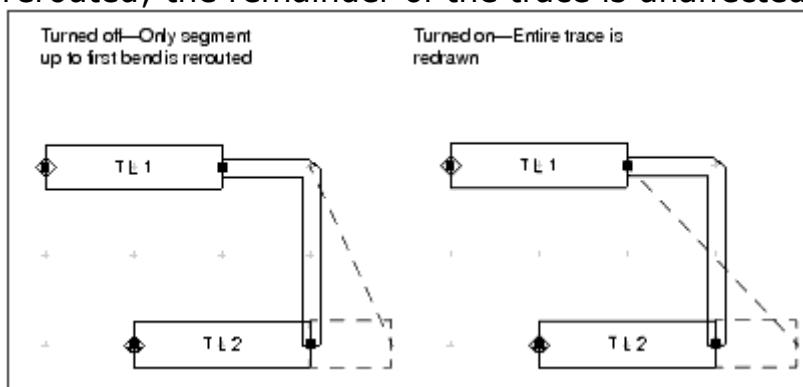
Restricts the Move Edge command to stretch an edge while maintaining the adjacent angles of the edge being stretched to other edges adjacent to that edge.

- **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.

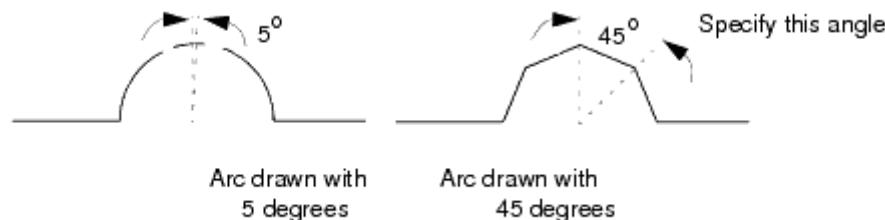
- **Reroute entire trace attached to moved component**

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.

- **Move handles enabled**

This option toggles display of a move handle on objects. Move handles on an object are easy-to-spot locations which can be grabbed so that the object can be moved. They appear when the object is selected. If Drag and Move is not enabled, this feature still allows the object to be moved by the move handle.

- **Drag and Move**

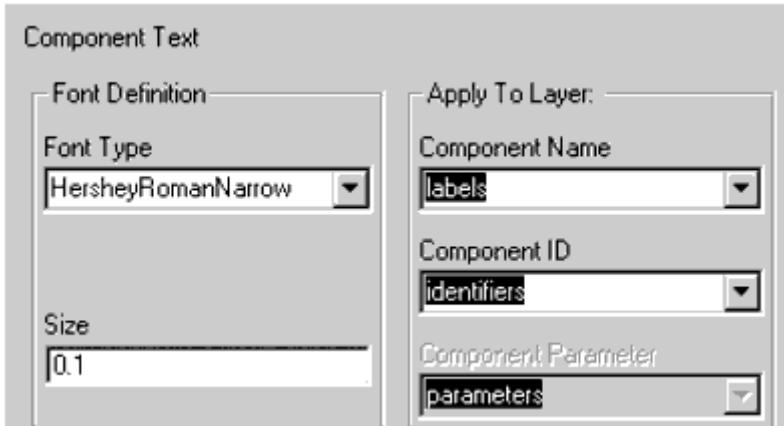
This option is designed to prevent you from moving an item when you click to select it (for any purpose) and unintentionally move the pointer in the process. By default, a move less than 10 screen pixels is not recognized as a move. An intentional move must be more than the distance specified here for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

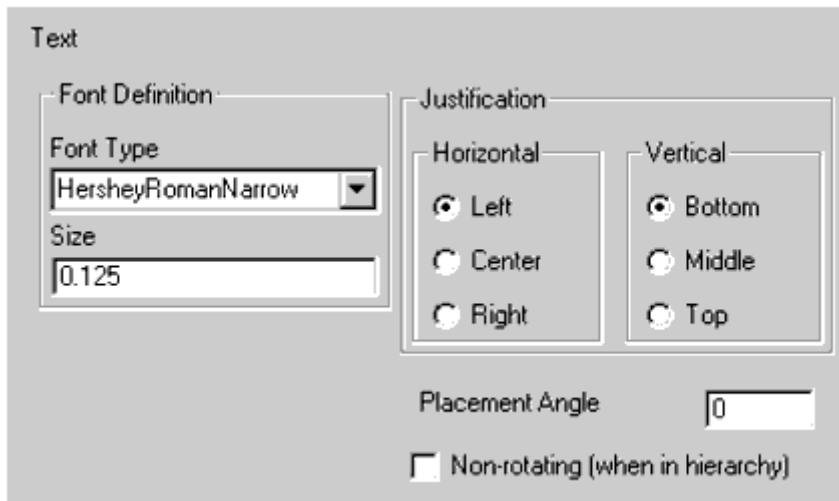
When a component is placed in layout, its name and reference designator (ID) are automatically placed with it on the silk screen layers. By default, the name is placed on the layer *silk_screen2*; the ID is placed on the layer *silk_screen*.

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.



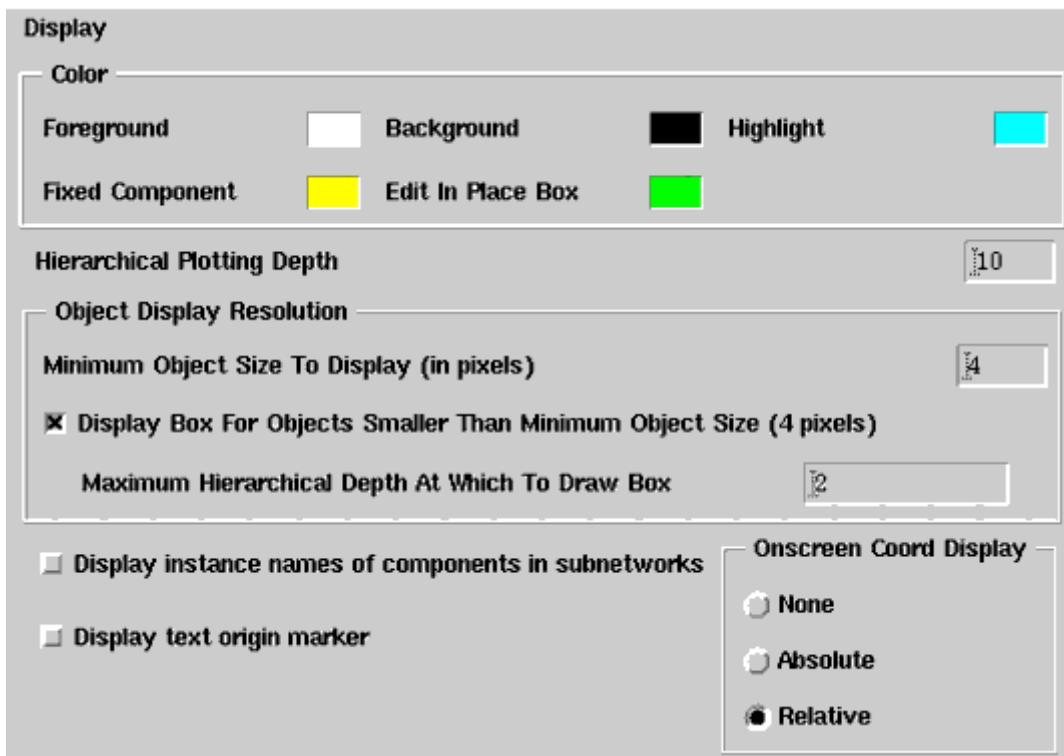
Note

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

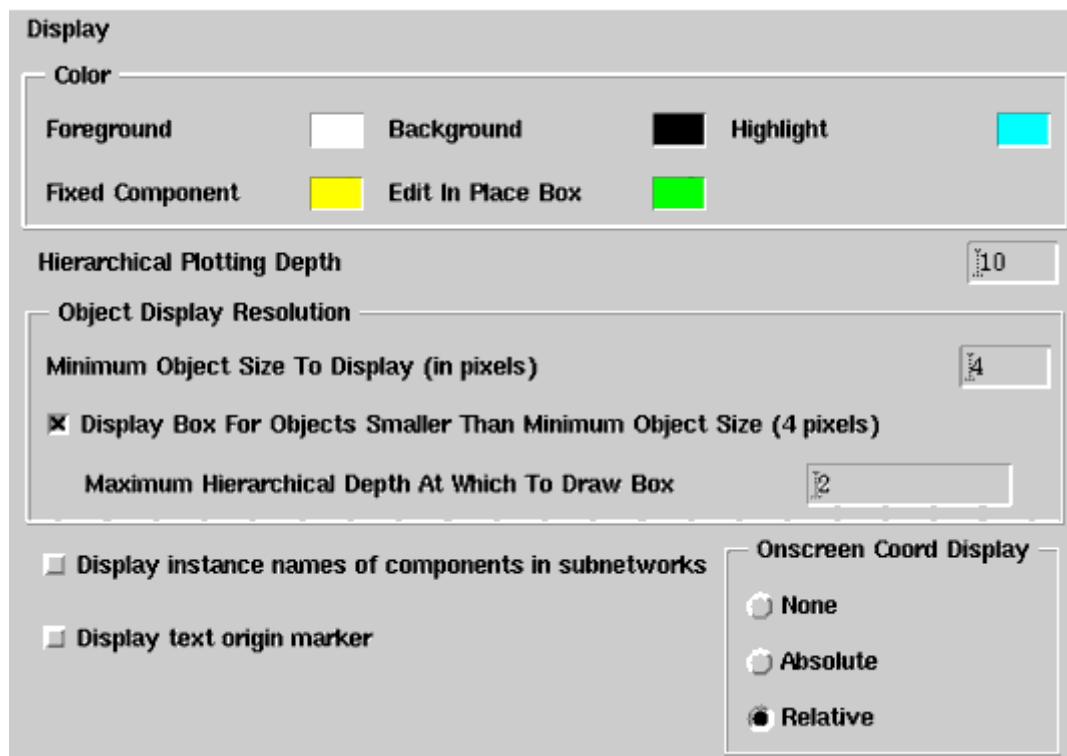
- the box that the *program* uses to highlight an item. This type of highlighting is used when you use choose one of the *Layout (Schematic) > Show* commands.

Setting DRC Memory Use and Performance

To set preferences for DRC memory usage and performance, refer to *Setting DRC Memory Use and Performance* (drc) in the *ADS Desktop Design Rule Checker* (drc).

Changing Miscellaneous Display Options

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



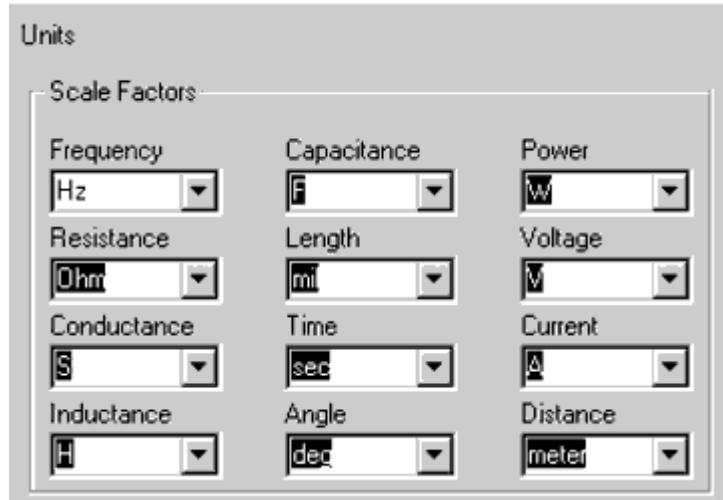
- Foreground**—The color of the lines making up polygons, polylines, and arcs while they are being drawn.
- Background**—The color of the drawing area background in the design window.
- Highlight**—The color used to identify problem items (with respect to simulation), orphaned items in schematic and 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.

Saving and Reading Preference Files

When you create a workspace, the files *schematic.prf* (for schematics) and *layout.prf* (for layouts) are copied to the new workspace from the installation directory. All designs in a given workspace use the preference files by these names.

Preference files are saved in the following ways:

- When the workspace is closed, the files with the default filenames (*schematic.prf* and *layout.prf*) are saved. The values of these preference files apply to every design in the workspace.
- To save a copy of the current preferences, customize the preferences as needed, and save them to a preference file with a unique name.

**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.

To customize preferences to the default filenames:

1. Choose **Options > Preferences**.
2. Change preferences as desired and click **OK**. The preference are updated and every design in the workspace will now use these preferences. The preference files with the default names will be saved when the workspace is closed.

To save a copy of the current preferences files to unique filenames:

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

**Hint**

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

To read a preference file:

1. Choose **Options > Preferences**.
2. Click **Read...** and choose a *.prf* file. The preferences from that *.prf* file are read and made current. When the workspace is closed, the values are saved to *schematic.prf* or *layout.prf*.

Toggling the Coordinate Readout Display

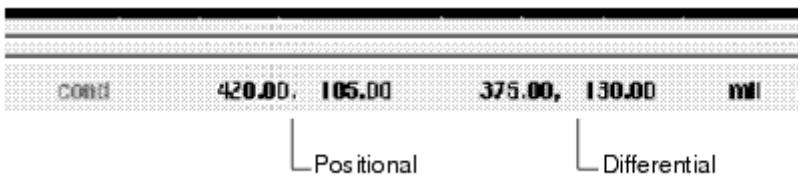
To change coordinate readout display options, select *View > Coordinate Readout*.

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**.

Pick and Place Report

To generate a Pick and Place Report:

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

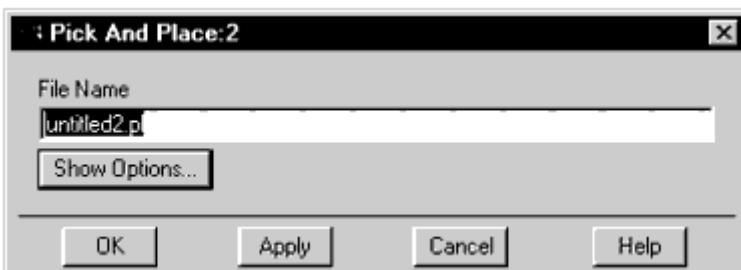


This dialog box allows you to edit the report name.

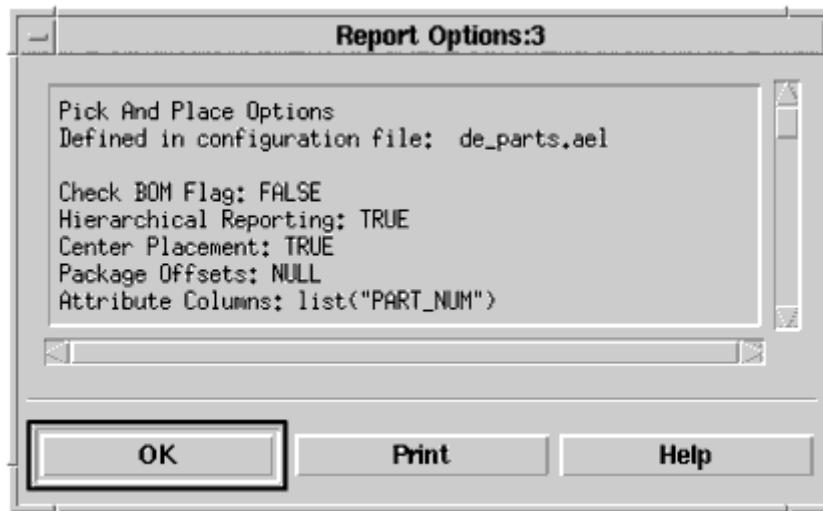
Viewing Pick and Place Report Configuration Options

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.

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 $HPEESOF_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

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:
 - 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* (packageAttributeName, 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](#)
- [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` is the default location)
- `%HPEESOF_DIR%`—the path you specified as your *Program* folder during installation (`c:\agilent\ADS2011_01` is the default location)

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:
 - `check_user_menu()` (ael)
 - `set_user_menu_label()` (ael)
 - `add_menu()` (ael)

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.

1. Save the file.
2. 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.

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 *DesignGuide Developer Studio* (dgstudio) 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.
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:

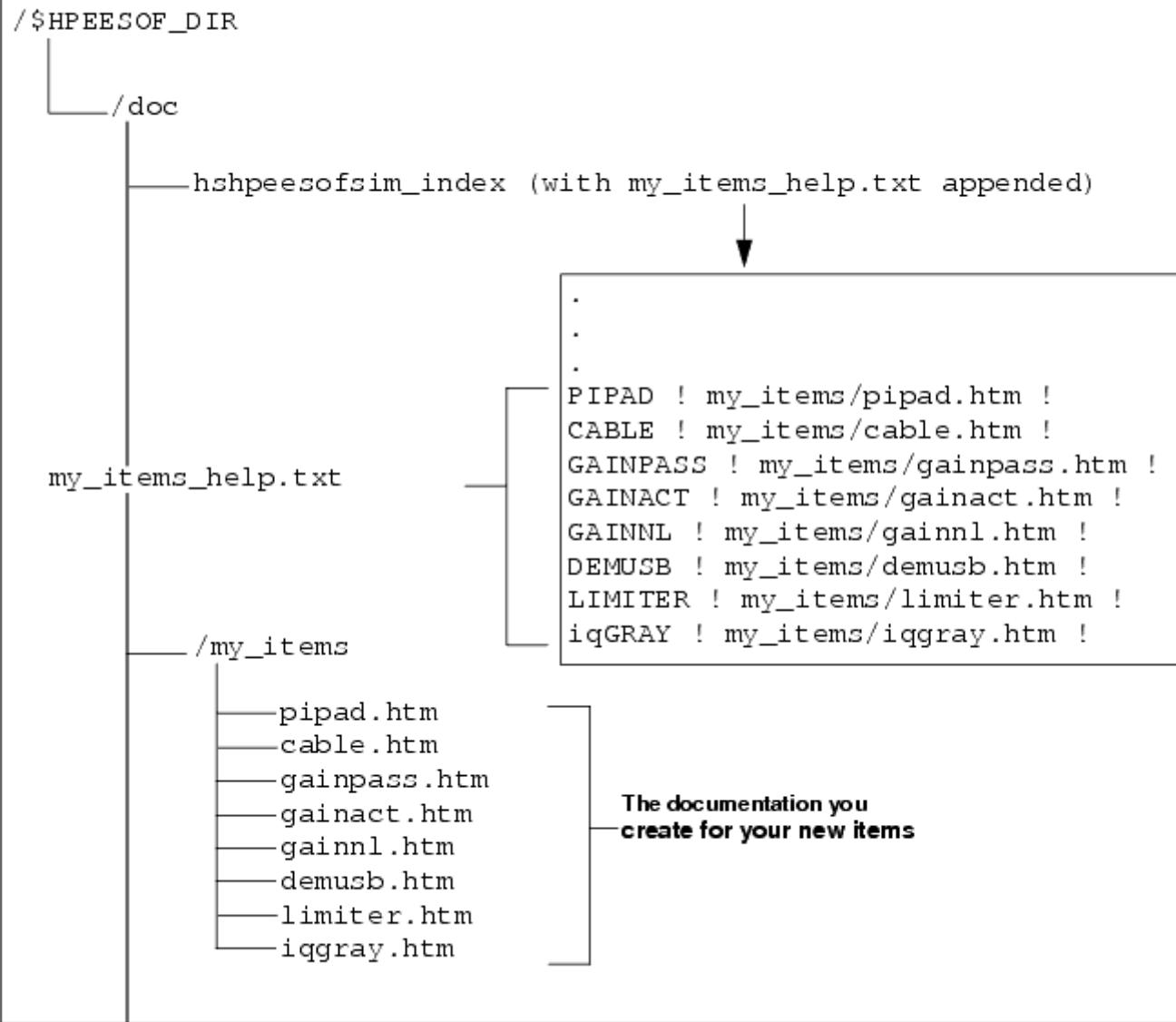
```
cat my_items_help.txt }} >> {{ hshpeesofsim_index
```

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.

- 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 workspace in that workspace 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 `hpeesofhelp.cfg` file, so that your custom doc will be accessed, instead of the standard ADS documentation. See *Customizing Configuration Variables* (custom) for more information on the location of .cfg files. See *Variables in the File hpeesofhelp.cfg* (custom) 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

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 \$HPEESOF_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 \$HPEESOF_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 !
```



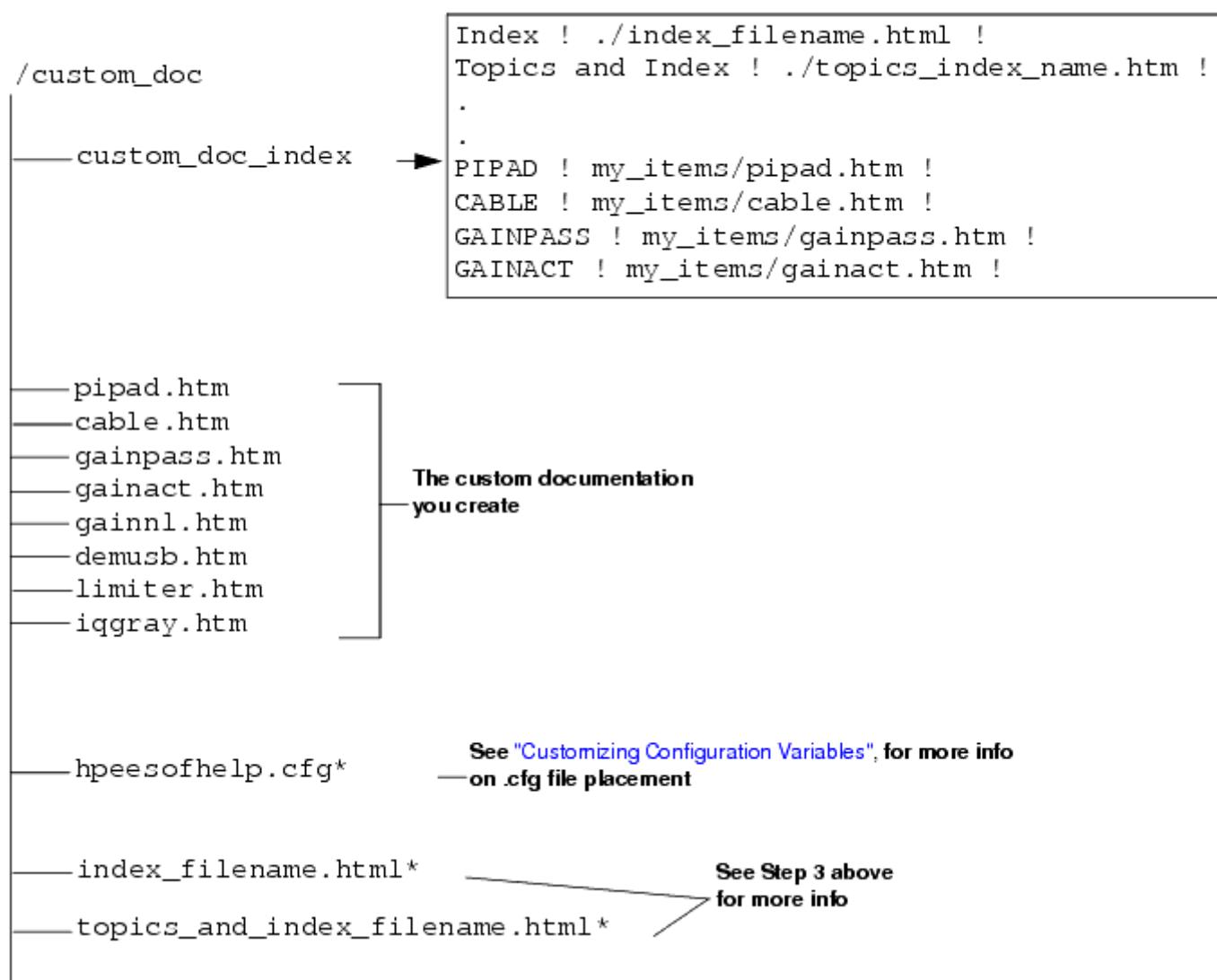
Note

If you point to custom *Index* and *Topics and Index* files within a specific workspace, and the workspace 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