Agilent Technologies

Advanced Design System 2002

User's Guide

February 2002

Notice

The information contained in this document is subject to change without notice.

Agilent Technologies makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

Warranty

A copy of the specific warranty terms that apply to this software product is available upon request from your Agilent Technologies representative.

Restricted Rights Legend

Use, duplication or disclosure by the U. S. Government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 for DoD agencies, and subparagraphs (c) (1) and (c) (2) of the Commercial Computer Software Restricted Rights clause at FAR 52.227-19 for other agencies.

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

Acknowledgements

Netscape is a U.S. trademark of Netscape Communications Corporation

PostScript is a registered trademark of Adobe Systems, Inc.

Contents

Chapter 1: Program Basics

The Advanced Design System design environment includes all the tools you need to manage your projects, create and edit schematics, and simulate your designs easily and efficiently. The program basics described here include:

Note For installation instructions, refer to the Advanced Design System *Installation* manual for your platform.

Documentation Conventions

To help you locate and interpret information easily, Advanced Design System manuals employ consistent visual cues, a few standard text formats, and some special terminology. Table 1-1 describes the typographic conventions used throughout the manual set.

Table 1-1. Typographic conventions

Type style	Used for
italic	Signals a new term; identifies directory names, filenames, and program names; highlights commands used within text.

Table 1-1. Typographic conventions (continued)

Type style	Used for
ALL CAPITALS	Acronyms
bold	Menu names, command names, items from a list, filenames or project names that you choose when following a procedure. Also used to denote the names of buttons you press or anything you type, including equations. Keys on the keyboard that are pressed are shown in bold.

Basic Terminology

Your familiarity with the terms described in Table 1-2 will make the concepts and procedures presented here easier to understand.

Table 1-2. Basic terminology

Term	Meaning
Click	To quickly press and release the left mouse button
Choose	To click a menu or command name
Directory	A collection of files
Double-click	To click the mouse button twice in rapid succession
Drag	To hold down the mouse button while you slide the mouse
Icon	(1) A picture (on a button) that represents the action that will be carried out or the component that will be chosen when you press that button; (2) the small box on your screen that represents an entire window; (3) a graphic symbol used on the PC to start a program.
Menu	A collection of related commands.
Press	To hold a mouse button down, usually combined with dragging, before releasing to complete some action
Project name	A designation for the location of files or directories on your disk.
Select	To mark an item by clicking on or near it with a mouse. For example, you might select an item and then choose the Delete command to remove the item from the drawing area.

Starting the Program

To start the program:

- UNIX, from the terminal window type:

  hpads

  Note Starting ADS in this manner assumes you have established a path to the ADS installation directory. If you have not, type <*installation_dir*>/*bin*/*hpads*, where <*installation_dir*> represents your complete installation path. For details on establishing a path statement, refer to Chapter 2 of the *Installation on UNIX Systems* manual.

- Windows, from the Start menu choose:

  Programs > **Advanced Design System 2002** > **Advanced Design System**

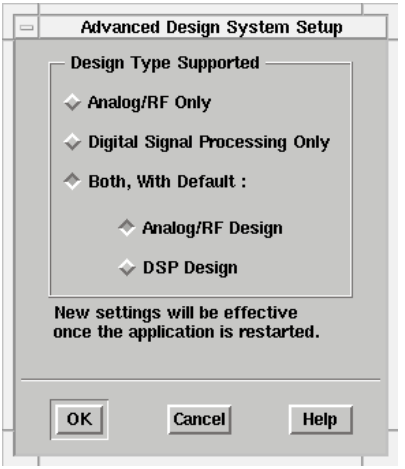
  to bring up the Advanced Design System Main window, which provides access to all features of the Advanced Design System. Alternatively, choose one of the other commands to access these specific features of the Advanced Design System:

ADS Tools	Displays a list of ADS tools such as Digital Filter Designer, DSP Synthesis, Library Translator, or LineCalc. Choose the tool you want to launch.
ADS Documentation	Launches the online documentation viewer and displays the primary window for accessing documentation from: Disk, if you chose to install it during Setup or CD-ROM, with the documentation CD-ROM in place
RF Designer	Launches the ADS Main window for use with the RF Designer product (the base ADS product for RF design)

Hint The first time you start the program, a window appears displaying information on new features in this release.

Selecting a Design Type

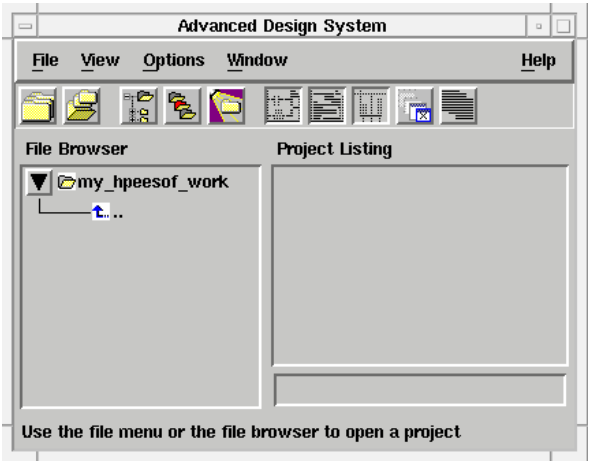
The first time you launch the application, you are prompted to select which type of components you want loaded on start-up: *Analog/RF Only, Digital Signal Processing Only,* or *Both*. Choosing either of the first two categories limits your choice of components, for the current session, to the selected category. Choosing Both allows both types of design work in the same session.



If you select Both (*Both, With Default:*), you must also specify a default design type. This default design type serves the following purposes: defines the components available by default in a design window and defines the default design type that appears in the New Design dialog box. (This choice only serves as a default and can be changed any time you start a new design (*File > New*).

Note You can change these options at any time through *Options > Advanced Design System Setup* in the Main window. When you change these options, you are changing them for the subsequent session and you are prompted to exit the application and restart it to effect the change. The selections remain valid until you explicitly change them again.

When you dismiss the setup dialog box, the Advanced Design System Main window appears.



The Design Environment

The design environment is made up of windows. All operations take place within the framework of windows.

Main window. Contains a title bar with the name of the window (Advanced Design System), a menu bar, and a toolbar for quick access to common operations. In addition it provides a tree-like view of directories, and where applicable, the designs comprising a hierarchical design.

From the Main window you can:

- Create and manage projects and designs
- Set program preferences
- Change toolbar configuration and keyboard shortcuts for the Main window
- Change the type of components loaded on start-up
- Playback macros created through Applications Extension Language (AEL)
- Issue AEL commands
- Launch a text editor
- Open a data display window
- Display all types of files and open as desired through context-sensitive menu

Design windows. The Schematic and Layout windows have the same basic appearance: title bar, menu bar, toolbar, palette, prompt panel, and drawing area.

From the Schematic window you can:

- Create and edit schematics
- Create variables and equations
- Simulate and optimize
- Open a data display window
- Generate a layout from the schematic

From the Layout window you can:

- Create and edit layouts
- Generate a schematic from the layout
- Create and edit parameterized artwork macros
- Define design rules and verify the design meets the established rules
- Launch Momentum
- Open a data display window

Data Display window. Contains a title bar, a menu bar, a toolbar, a palette, and a graphics area. This window enables you to view and compare simulation data.

Message/Status window. Appears whenever a simulator is launched and displays messages about the status of the current process as well as warning messages.

Hint As you work with multiple windows, you may find it helpful to minimize the Status window to clear additional space on your screen. To quickly restore it to the screen, choose *Window > Simulation Status* from any program window.

Working in the Main Window

The Main window enables you to create and manage projects. Projects are central to the operation of all the simulators and allow you to organize your related designs.

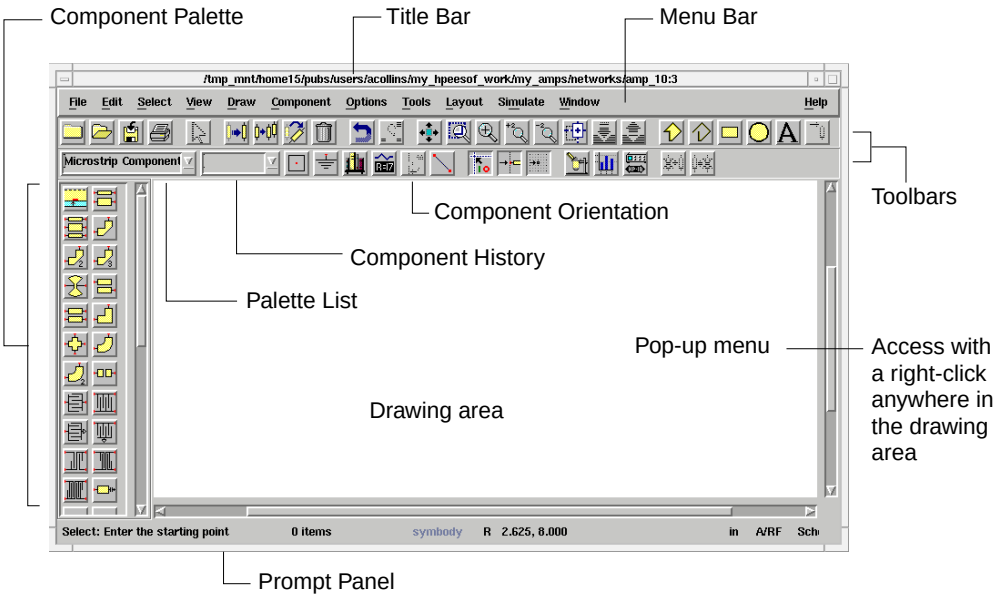
From the Main window you can:

- Create and manage projects and designs
- Quickly open example projects (*File > Example Project*)
- Set program preferences
- Change toolbar configuration and keyboard shortcuts
- Change the type of components loaded on start-up
- Playback macros created through Application Extension Language (AEL)
- Issue AEL commands
- Launch the text editor
- Open a data display window
- Pop obscured windows to the top (from the Window menu)
- Display all types of files and open as desired through context-sensitive menu (*View > Show All Files*). Click right to open different types of files in the appropriate type of window (including a text editor for *.ael*, *.cfg*, etc.).

For a detailed discussion of projects, refer to Chapter 2, Managing Projects and Designs.

Working in Design Windows

A design window is where you create and edit all of your designs. You can resize and move these windows in the workspace. You can enlarge one window to fill the entire workspace and you can shrink each window to an icon. The following illustration shows the parts of a design window.



- The *Title bar* displays the window type, design type, filename, and a number identifying which window of that type it is
- The *Menu bar* displays the menus available in that window
- The *Toolbar* contains buttons for frequently used commands and for choosing the appropriate orientation for components. The collection of buttons on the toolbar is configurable (*Options* > *Menu/Toolbar Configuration*) and can be toggled on and off (*View* > *Toolbar*).

  When you move your pointer slowly over the buttons on the toolbar, a *balloon* appears with a label identifying the function of that button. By default, the option that controls the display of this label is turned on. To turn this option off, choose *Options* > *Preferences* in the Main window and turn off *Balloon Help*.

Hint To change the timing for the display of the balloon, set the variable BALLOON_HELP_TIMEOUT in the file *de_sim.cfg*.

- The *Palette List* enables you to choose a category of components to place on the *Component Palette*
- The *Component History* drop-down list is continuously updated to reflect the components you have placed in your design. It provides a quick method of placing another instance of a component in your design and can be used as a starting point for creating a custom palette.
- The *Drawing area* is where you create your designs
- The *Component Palette* contains buttons for placing components
- The *Prompt panel* provides messages to assist you during the execution of most commands, as well as various pieces of information to assist you in creating a design.
- The *Pop-up menu* lets you access many common commands with a minimum of mouse movement. You access the pop-up menu by pressing the right mouse button in the drawing area of any design window. The context-sensitive commands appear on the pop-up menu when the pointer is positioned over certain shapes or text and you click right.

  Table 1-3 describes the actions of the pop-up menu commands. Not all windows support all of these commands. Most of the commands can be given at any time without interrupting other commands in progress.

Table 1-3. Pop-up menu shortcuts

Command	Action
End Command	Cancels the active command
Deselect All	Deselects anything in the window that is selected
Redraw View	Refreshes the screen without affecting window magnification
View All	Scales the view of your drawing so that all of it fits inside the current drawing area
Pan View	Moves a point you specify to the center of the window
Zoom Area	Enables you to define a window of the drawing to fill the drawing area
Zoom In x2	Zoom in by a factor of 2 around a point you specify

Table 1-3. Pop-up menu shortcuts (continued)

Command	Action
Zoom Out x2	Zoom out by a factor of 2 from a point you specify
Delete	Deletes the selected object(s)
Arc (clockwise)	Enables you to draw a clockwise arc
Arc (counterclockwise)	Enables you to draw a counterclockwise arc
Undo Vertex	Removes the last arc or vertex entered
Context-sensitive commands	
Polygon Layer	Enables you to select a different layer for the polygon
Polyline Layer	Enables you to select a different layer for the polyline
Rectangle Layer	Enables you to select a different layer for the rectangle
Edit Component Artwork	Enables access to selecting different artwork for the selected component
Wire Layer	Enables you to select a different layer for the wire
Edit Circle	Enables you to select a different layer for the circle, change its radius and its resolution
Edit Arc	Enables you to select a different layer for the arc and change its resolution
Edit Text	Enables you to select a different layer for the text, edit the text itself, change the font, size, justification, and angle
Set As Text Default Values	Uses the new settings as defaults (same as Options > Preferences > Text)
Edit Path/Trace	Enables you to modify Path/Trace characteristics such as Width, Layer, and Corner Type
Set as Path/Trace Default Values	Uses the new settings as defaults
Symbol Pin Edit (Symbol view only)	Enables you to edit a pin's name, number, angle, and type

Hint The display of the following options can be toggled on and off through these View menu commands by these names:

Toolbar, Component Palette, Status Bar, Coordinate Readout

Opening Design Windows

There are several ways to open design windows and the method you use is based on what you want to do in that window.

- New design—To open a Schematic or Layout window for creating a new design or editing an existing design not currently in memory, click the *New Schematic* or *New Layout* button or choose the command (by the same name) from the Window menu in the Main window.
- Additional window—To open an additional Schematic or Layout window for a design that is already open, choose the *Schematic* or *Layout* command from the Window menu in that window.

Hint You can also open a new Data Display window from the Main window.

Note that Schematic and Layout windows are numbered sequentially as they are opened throughout a session. If Schematic window number three is open (the title bar reflects *(Schematic):3*) and you open a dialog box from that window, the title bar of the dialog box will also reflect *:3*. This is to assist you in identifying which design window you are about to make changes to.

Opening Multiple Design Windows for the Same Design

The design environment enables you to use multiple Schematic and Layout windows at the same time. For example, you can open two Schematic windows with different designs making it easy to copy and paste parts from one design to another. Or the windows can contain the same design making it easier to accomplish certain design tasks.

Hint When you want to open a window for creating a new design or editing an existing design, use the Window menu in the Main window; when you want to open an additional window for the current design, use the Window menu in that window (Schematic or Layout).

Figure 1-1 illustrates how you can connect components in a large schematic when the components are far apart and the pins difficult to see. You can open an additional window containing the same design. In the first window, zoom in on one of the components. In the second window, zoom in on the other component. Choose the

desired *Wire* command and draw the wire by clicking the appropriate pin in one window, moving the pointer to the second window and clicking the appropriate pin there.

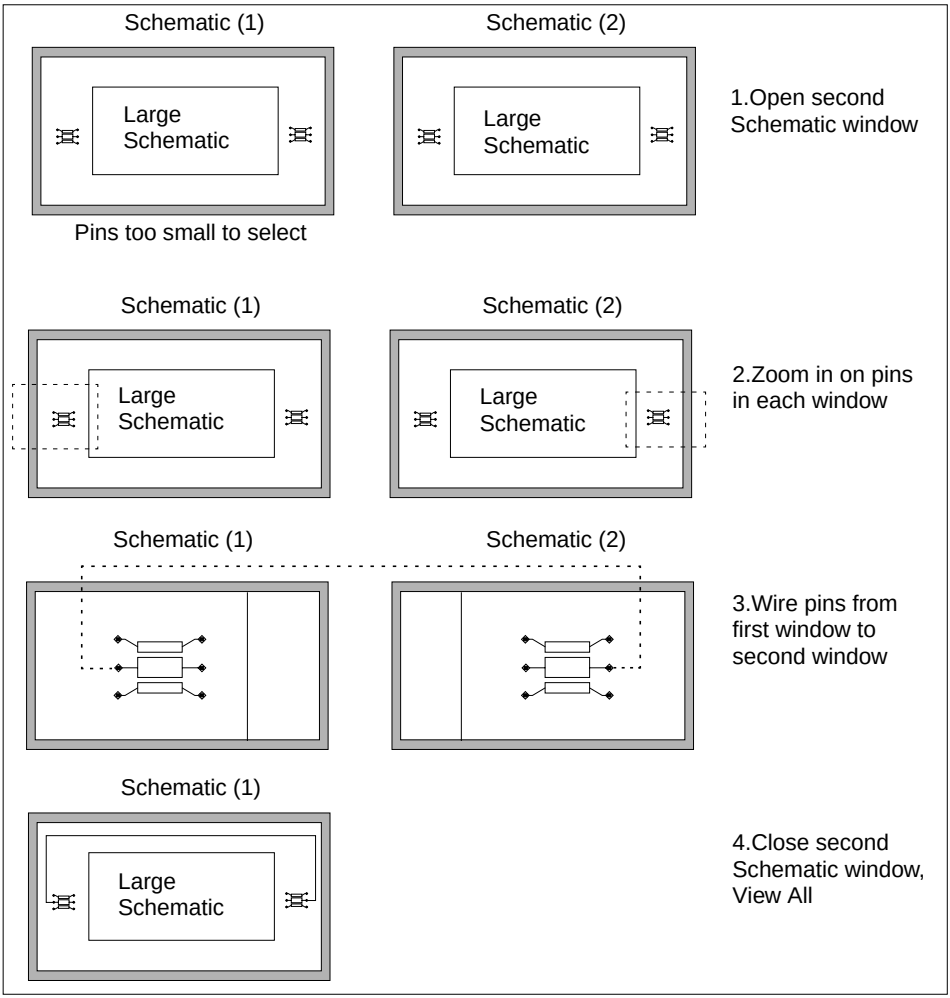
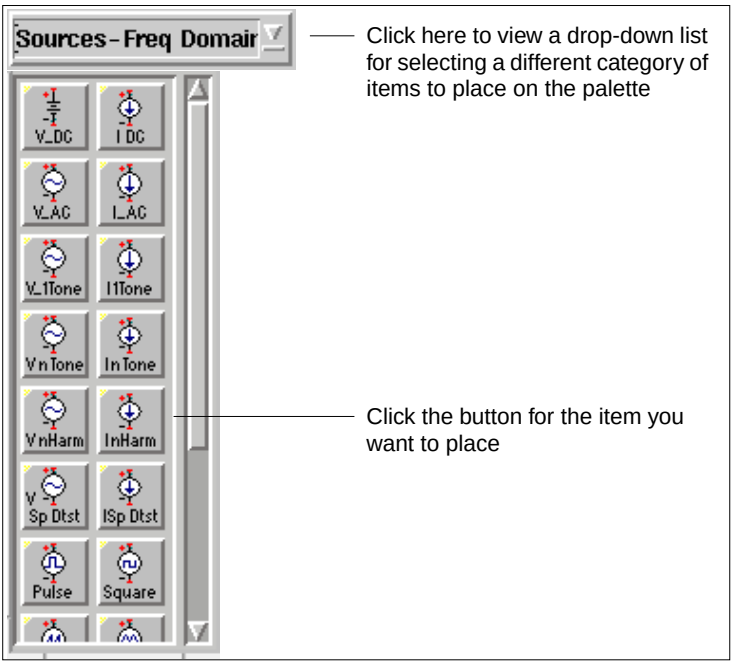


Figure 1-1. Connecting components using multiple windows

Using the Component Palette

The *Component Palette* contains buttons that provide a quick method of placing items to create your design.



Hint All palette items can also be placed through the Library. Some items are only available through the Library.

Detaching the Component Palette

The component palette can be detached from the window and moved anywhere on the screen. You may find this helpful in temporarily providing additional space in the drawing area.

To detach the component palette:

Choose **View** > **Component** > **Detach Component Palette**. Window borders and a title bar appear. Where applicable, scroll bars also appear. This window can now be moved around and manipulated like any other window.

To attach the palette to the window again:

Choose **View** > **Component** > **Attach Component Palette**. The palette is once again integrated with the design window.

Hint On the PC only, you can detach and re-attach using the mouse. Position the pointer over a blank area between buttons on the palette and press the left mouse button. A border appears. Drag the palette to the desired location and release. To re-attach, position the pointer in the title bar of the detached palette and press the left mouse button. Drag the palette toward its original location, positioning the title bar just under the bottom of the palette drop-down list and release.

Moving Toolbars (PC Only)

The toolbars can be repositioned anywhere on the screen. You can move them away from the window and use them like floating palettes or you can dock them along the window's edges.

Hint When the title bar of a toolbar is visible, positioning your pointer within the title bar for the drag operation simplifies the docking process. If a title bar is not visible, move the toolbar away from the window's edge and release; when it is not docked, a title bar appears.

To float a toolbar away from the window:

1. Position the pointer over a blank area between icons on the toolbar and press the left mouse button.
2. Drag the toolbar to the desired location and release. When you release the toolbar, a title bar appears at the top of it.

To dock a toolbar on a window border:

1. Position the pointer over a blank area between icons and press the left mouse button.
2. Drag the toolbar toward the desired window border and notice that the ghost image of the toolbar changes as needed to fit in a vertical or horizontal space.

3. When the ghost image reflects the proper orientation, release the mouse button and refine the toolbar's position by dragging as necessary.

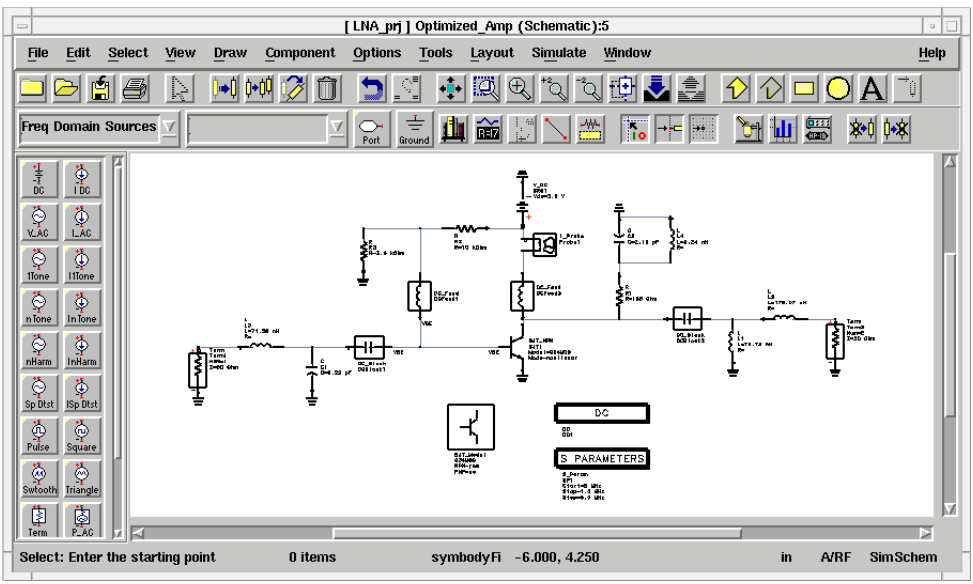
To reattach a toolbar near the top of the window:

1. Position the pointer in the title bar of the toolbar and press the left mouse button.
2. Drag the toolbar toward the top of the window and when your pointer is overlapping the menu bar, or another toolbar, release.

For details on customizing and creating toolbars, refer to the section, "Configuring Toolbars" on page 9-31.

Schematic Window

The Schematic window is where you create your schematic designs. You create your design by placing components, ports, data items, units, variables, equations, etc.



The program is shipped with a set of standard defaults and parts libraries. These differ depending on program options. However, all defaults can be modified on a system-wide, or project basis. Before beginning any serious design effort, you can customize these defaults to better match the typical designs done at your site.

The most important thing you can do before starting your design is to configure your setup correctly. There are many ways to configure the program defaults. The best configuration for you depends on the type of designs you create, the options you have, and the type of final output you require.

Layout Window

Although the Layout window has many things in common with the Schematic window, it also has many important differences. For details on the Layout option, refer to the *Layout* manual.

Closing Design Windows

To close an individual design window, but keep the design in memory:

From the Window menu in that window, choose **Close**.

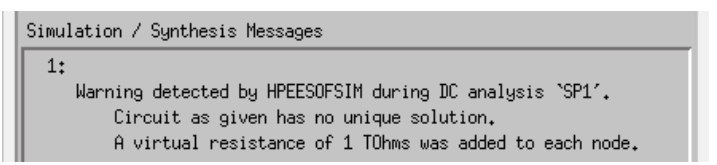
Status Message Window

The Simulation/Synthesis Message window appears whenever a simulator is launched and displays messages about the status of the current process, as well as warning messages. Each simulation generates its own set of messages which are stored in memory during the current session. These sets of messages can be distinguished from one another by the number displayed in the title bar of the window.

The window contains two information panels:

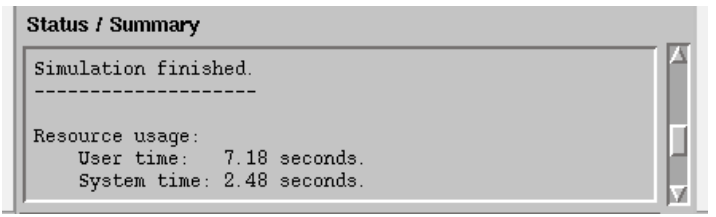
- Simulation/Synthesis Messages
- Status/Summary

The *Simulation/Synthesis Messages* portion of the window displays detailed messages about problems encountered during a simulation or synthesis, and where possible, what you can do to solve the problem.



Hint Watch for a message that prompts you to click to view the source of the problem. Clicking this message highlights the component(s)—in the Schematic window—causing the problem.

The *Status/Summary* portion of the window displays a *Simulation finished* message, statistics such as how long the simulation or synthesis took, and the system resources used.



Viewing Simulation Status and Error Messages

When the simulation/synthesis is finished, you can save the displayed information to file or you can send it directly to the printer.

To save the currently displayed information to file with a default filename:

Choose **File** > **Save Design** and click **OK**. The default filename consists of the simulation process number (from the title bar of the window), with a prefix of the string *sessloghpeesofsim* and a file extension of *.txt*. The file is saved to the current project directory.

To save the currently displayed information to file with a filename of your choosing:

Choose **File** > **Save As**. Supply a filename and click **OK**. The file is saved to the current project directory.

Note If you have changed projects during the current session, the file may be written to the initial project opened in this session.

To send the information directly to the printer:

1. If needed, choose **File** > **Print Setup** to establish the desired setup and click **OK**.
2. Choose **File** > **Print**. The displayed information is sent to the printer.

   For details on print setup, refer to Chapter 11, Printing and Plotting.

Because each simulation generates a set of messages identified by unique names, you can view any messages generated during the current session. You can view these one at a time in the same window, or you can open multiple windows and display different ones all at the same time.

To view messages generated by another simulation:

1. Optionally, choose **Window** > **New Window**.
2. From the Window menu, select the simulation information you want to view, as identified by the unique number displayed in the title bar associated with each simulation. The display changes to reflect your selection.

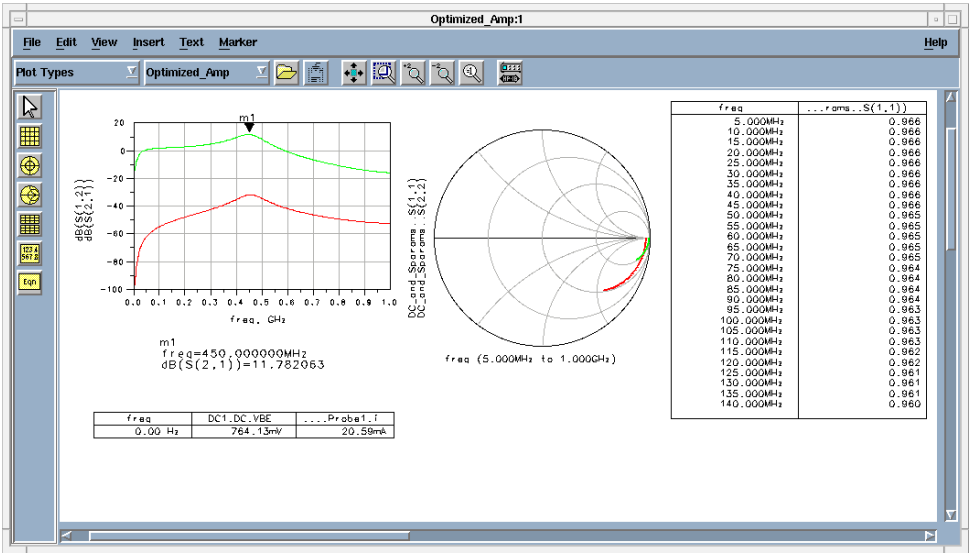
To close any individual status window:

Choose **Window** > **Close Window**.

Using the Data Display Window

You can open a Data Display window to see the results of your simulation analysis. To view a graph, choose *Window > New Data Display* from the Main or Schematic window. After you open a window, you can select an independent swept variable, select dependent measurements, scale the data, and add captions to your graph. Then you can print or plot the graph.

You can open one or more Data Display windows at a time inside the same work space. For example, you can view the same data on a graph and in a tabular format at the same time. Each graph appears in a separate window.



For detailed information on working with data displays, refer to the *Data Display* manual.

Using the File/Instrument Window

The Instrument window enables you to read data from sources such as a network analyzer or a Touchstone file into Advanced Design System as a dataset. The Instrument Server can also write data from a dataset out to an instrument or file. To display the Instrument Server window, choose *Window > File/Instrument Server*. For details on using the Instrument Server, refer to the *Using Instruments with ADS* manual.

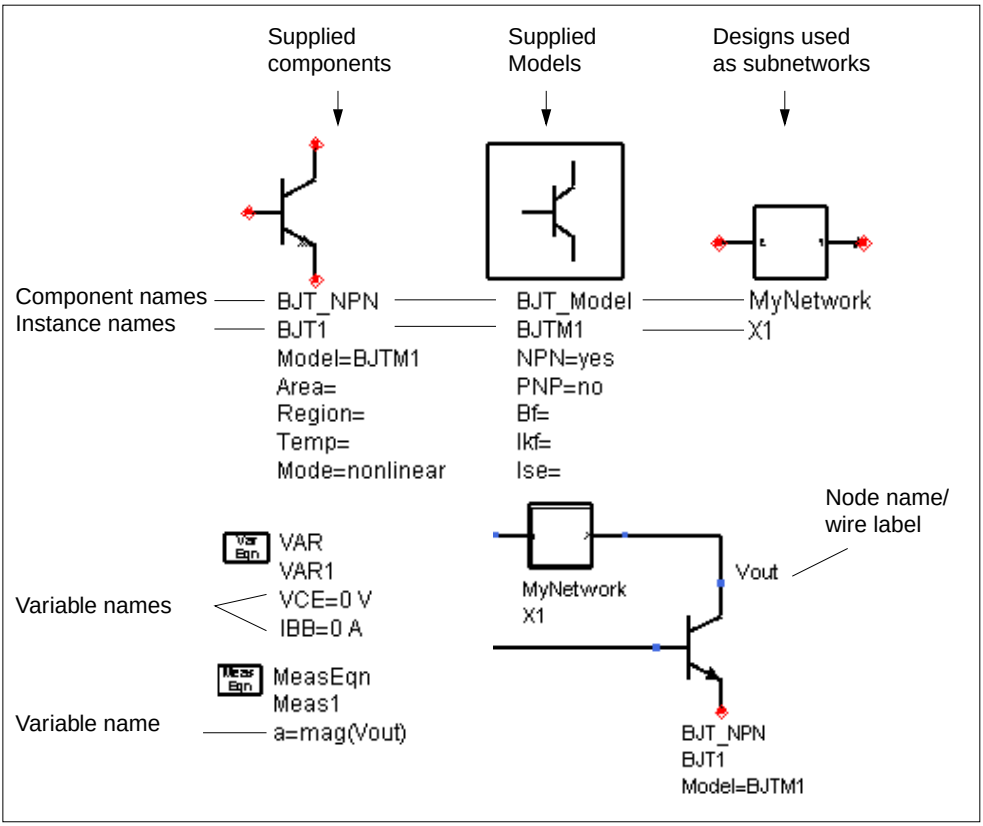
Naming Conventions

Prior to Advanced Design System 1.5, user-supplied names throughout the software were restricted in numerous ways. Not only were names restricted with respect to the allowable set of characters, but you could not have any duplication of names among certain types of items, such as node names and instance names. Using the same name for the following items is now allowed:

- Variable names (created in VarEqn and MeasEqn components)
- Node names/Wire labels
- Instance Names
- Component names (includes supplied components and models, and your designs used as subnetworks)

Note The names of supplied components cannot be used as design names. If you receive an error message stating the supplied design name is reserved for Advanced Design System, you have most likely used a name that is reserved for a component. To review these names, see the component libraries or the component manuals.

The following illustration shows how these terms are used.



User-supplied names that were previously restricted to alphanumeric and underscore (_) characters (hereafter referred to as the *standard* character set), can now take advantage of an *extended* character set that incorporates additional special characters. The *extended* character set (a superset) consists of the following characters:

alphanumeric _ + - = ^ ` @ # & $ %

In addition, you can now use a numeral as the first character in many names.

The following table denotes where the extended set and numeric prefixes can be used, as well as several exceptions:

Name	Character Set	Exceptions	Numeric Prefix
Variable (VarEqn, MeasEqn)	Standard		No
Node/wire label	Extended		Yes
Instance	Extended	An underscore cannot be used as the first character. The following components are restricted to the standard character set for their Instance Name: SweepPlan, ParamSweep, DC, AC, S_Param, HB, LSSP, P2D XDB, Envelope, Transient, Options, YieldSpec, Goal, Yield, Optim, YieldOptim, MeasEqn, VarEqn, DataAccessComponent	Yes
Component	Extended		Yes
Design	Extended	Design names may not contain $ or %. These characters are reserved for ADS environment variable substitution. Note: If a design name includes special characters or starts with a numeral (such as "00a" or "@"), optimization results are not updated Note: If a design name consists solely of numerals, The transistor device operating points dialog box cannot be displayed.	Yes
Project	Standard		Yes

Limitations

- When the extended character set is used to define a name, and that name becomes part of a variable name in a dataset, the data cannot be accessed directly with respect to either data displays or measurement equations. A special access function called var(), must be used. For example, if you name a node V++ then to display its magnitude you must use the expression:

  mag(var("V++"))

  where *var()* is passed the name of the variable as a string. The *var()* function interprets the passed string as the name of a variable, which prevents any expression processing of the string. Note that using this function limits you to accessing datasets in the data directory of the current project. Data Display automatically adds *var()* when required to access names defined using the extended character set.

- Substrate Instance Names—If a leading number is used as a substrate ID, then that substrate cannot be referenced by its corresponding set of distributed models.

- Model Instance Names—Currently, the characters $ and % are not allowed in Instance Names for model items, such as BJT_Model, R_Model, etc.

- Dataset Names—Special characters from the *extended* set are not allowed in dataset names (*standard* set, with non-numeric first character, is required). Thus, if you have two or more designs whose names are distinguishable from one another only with respect to the use of special characters, you should supply unique dataset names prior to simulating. If you do not, and you perform successive simulations of these designs, the dataset resulting from the first simulation will be overwritten by the next (because the special characters will be dropped from the automatically derived dataset names).

  Note: If your design name begins with a number, the default dataset name will be the design name with an underscore character (_) added to the front.

- Optimization Goals—Special characters currently cannot be referenced by an Optimization Goal or Yield spec. If a design name includes special characters or starts with a numeral (such as "00a" or "@"), optimization results are not updated.

- Operating Point Annotation—The transistor device operating points dialog box does not come up when a design has a numeric name such as "0000" or "1234".

- Spaces are not allowed in project and design names

Note An invalid character is changed to "_" (underscore).

Global Node Naming Conventions

If you use an exclamation point (!) as the last character in a node name, it denotes that the node is a *global* node.

Creating an Electronic Notebook

The Electronic Notebook enables you to generate a portable *notebook* containing screen captures of schematic and layout designs, as well as data displays, for a given project. You can add descriptions for every page in the notebook, and you can include text and graphics from other sources. The annotated body of design work can then be viewed in a browser, enabling you to share your designs with others, without running ADS. You can also *zip* the set of files generated by the notebook to facilitate transferring them.

To create an electronic notebook:

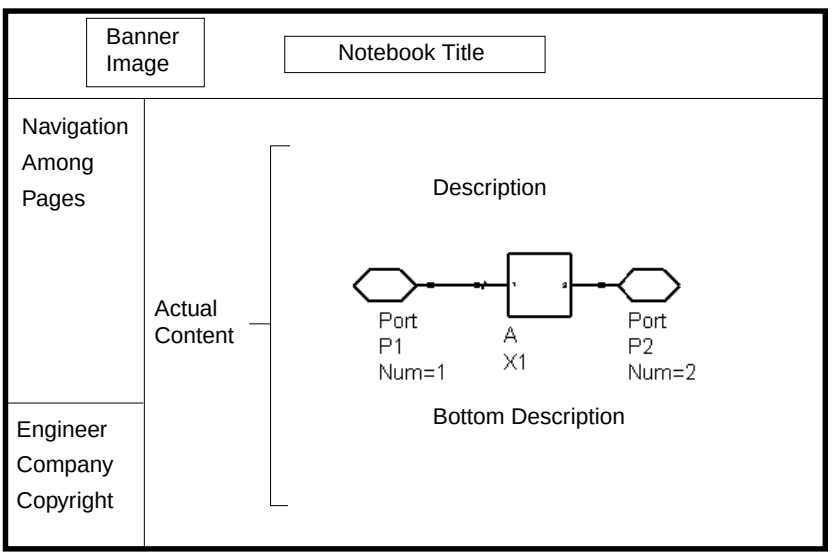
1. From a Schematic window in the project of interest, choose **Tools** > **Electronic Notebook**. A dialog box appears providing a brief description of the notebook basics.
2. Click **OK** and the main Electronic Notebook Editor dialog box appears. The notebook displays the default notebook structure, which includes every design and data display in the current project, as well as a single *Description* page for the notebook.

   At this point, you can click *Generate* and generate the actual HTML, based on the default structure and default options, but the notebook editor contains several features that enable you to customize it. For details, refer to the following topics of interest:

   - “Deleting Pages from the Notebook” on page 1-26
   - “Adding Descriptions to the Notebook” on page 1-26
   - “Adding Pages to an Existing Notebook” on page 1-27

- “Reorganizing Pages in the Notebook” on page 1-28
- “Adding External Images to the Notebook” on page 1-28
- “Changing Image Capture Settings in the Notebook” on page 1-29

3. To generate a new notebook using the defaults, click **Generate.** An information message appears with important guidelines regarding the status of ADS windows during the HTML generation process. After reading these messages, click **OK** and the HTML generation process begins. When the generation is complete, the first page of the notebook is displayed in your browser. The following illustration identifies the basic layout of the notebook using an individual Schematic page as an example.



Refer to the following topics for descriptions of additional features:

- “Saving Changes to the Notebook” on page 1-31
- “Viewing an Existing Notebook” on page 1-31
- “Updating an Existing Notebook” on page 1-32
- “Zipping the Files of a Notebook” on page 1-32

Deleting Pages from the Notebook

By default, when you create a new notebook, it includes pages for every design and data display in the project, but you can easily delete designs from the notebook.

To delete individual designs from the notebook:

Select the design you want to delete and click **Delete Page**. If you change your mind, click *Add Page*, select the appropriate Page Type, select the design from the drop-down list, and click *OK* to add it back.

Adding Descriptions to the Notebook

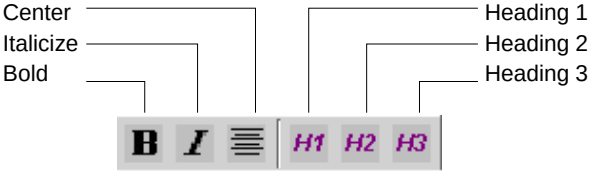
To add descriptive text to a specific page of the notebook:

1. Select the page from the list of pages on the left and enter the desired text in the *Description* area(s) on the right. For individual design pages, you are provided with two text boxes: *Description* and *Bottom Description*, which are added above and below, respectively, the captured image.

Hint To include existing text from another source: on UNIX, highlight that text and use the middle mouse button to paste it, or on the PC, copy the desired text and use the pop-up menu available from within the Description text boxes to paste it. On the PC only, you can copy and paste text among the various description text boxes using this pop-up menu.

---

2. Optionally, you can use HTML in any of the description text boxes, enabling you to format it as you please. Some basic HTML shortcuts are provided, but you can enter most standard HTML tags directly in the text boxes.



To use the shortcut HTML tags, highlight the desired text and click the desired shortcut icon. The associated HTML tags appear.

Note Due to differences in individual browsers and the fonts installed on your operating system, these formats may not always produce the expected results.

3. When you are through making changes, click **Generate** and these descriptions will be incorporated in the notebook.

Adding Pages to an Existing Notebook

You can add a number of page types to the notebook. You can add new (since the notebook was generated) or previously deleted Schematic, Layout, and Data Display pages. You can also add a page that combines a Schematic and a Data Display on the same page. And you can add an *Image* page for displaying a graphic from a source other than ADS.

To add a page to the notebook:

1. Click the **Add Page** button.
2. Select the desired Page Type. The dialog box changes, based on the selected Page Type.
   - If adding a *Description* page, no additional action is needed yet.
   - If adding a *Schematic*, *Layout*, or *Data Display* page, select the name of the design (to appear on that page) from the drop-down list.
   - If adding a combined *Schematic/Data Display* page, select both the schematic design name and the data display name from the respective drop-down lists.
   - If adding an *Image* page, enter the filename or use the browser to select it (refer to "Adding External Images to the Notebook" on page 1-28).
3. Click **OK** and the page appears among the list of other notebook pages.

Hint If an individual design is highlighted when you add the page, the new page appears under that design. If a group is highlighted, the new page appears at the bottom of that group. You can then move it up and down, as well as left and right, to position where you want it.

Reorganizing Pages in the Notebook

To reorganize the designs in the notebook to reflect the design hierarchy:

1. Select the top-level design and click the Left arrow to make it a *folder*.
2. Use the Up and Down arrows to move the subnetwork designs under it, in the desired order.

Moving a design/display from one group to another (such as, moving a data display into the schematic group) requires making it a *folder* temporarily. To group related schematic/layout designs and data displays together:

1. Select the design and click the Left arrow button (to make it a folder).
2. Use the Up and Down arrow buttons to move it above or below the design you want to group it with, and click the Right arrow button to move it in again.

Adding External Images to the Notebook

You can include an image from an external source by adding an *Image* page, and you can add descriptive text above and below it, just as you can with other types of pages. You can also replace the Agilent logo with an image of your own.

To add an *Image* page to your notebook:

1. Click **Add Page** and select **Image** as the Page Type. A field for a filename appears.
2. Type the path and filename or use the browser to select the file.
3. Optionally, click **View** to verify you have the image you want.
4. Click **OK**.

Hint The image will be copied to the notebook directory, so if you change this image in its original location, and want those changes to be part of the notebook, be sure to import it to a new image page, copy and paste descriptions, etc., and then delete the image page containing the older image.

---

5. Optionally, reposition the image page within the notebook using the Up and Down arrows, and add any desired descriptive text.

6. When you are through making changes, click **Generate** and the new image page will be incorporated in the notebook.

To replace the Agilent logo with your own image:

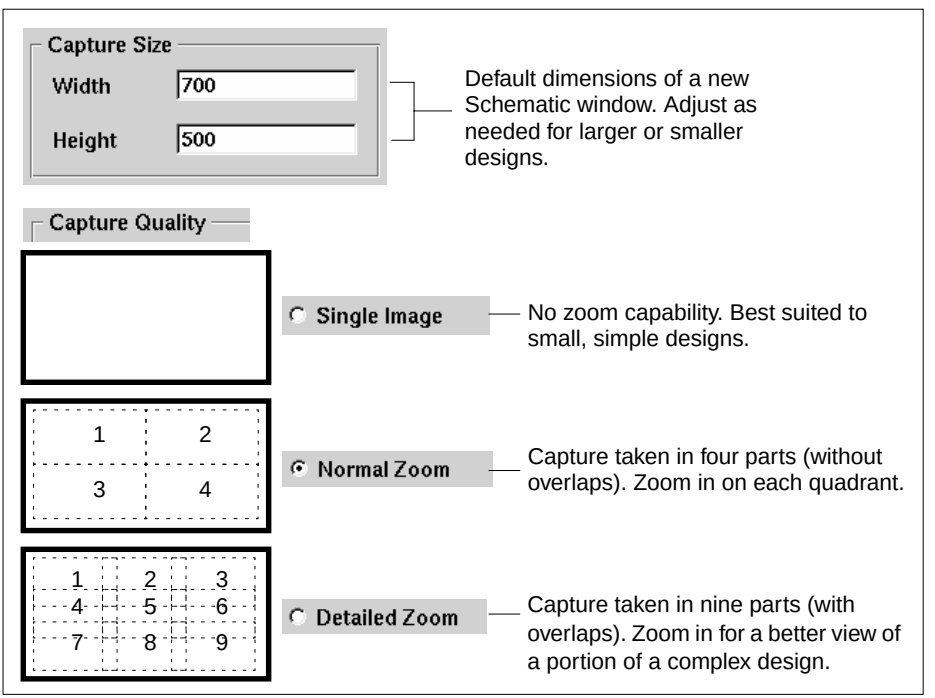
In the Banner Image field, type the path and filename, or use the browser to select one. Click Generate when you have made all other desired changes.

Changing Image Capture Settings in the Notebook

During the HTML generation, screen captures are taken of all schematics, layouts, and data displays that are part of the notebook. By default, these screen captures will be 700 pixels wide x 500 pixels high, with a normal zoom (see illustration that follows). You can establish default settings for all new screen captures, and override these defaults for individual designs.

To set a default capture size and zoom for all designs in the project:

1. From the Notebook Properties pane, click **Preferences**.



2. Change the capture dimensions and zoom setting as desired and click **OK**.

To override the default capture size and/or zoom for an individual design:

1. Select the design of interest and click **Capture Options**.
2. Change the settings as desired.

Hint If after experimenting you decide you want to go back to the original settings, click *Restore Defaults*.

- For a new notebook, click **OK** in the Capture Options dialog box. When you generate the notebook as a whole, these settings will be used for this design.
- If you are updating an existing notebook, click **Recapture Image**. A message appears explaining that if you proceed, the currently saved image will be replaced by a new capture, using the new capture settings. If this is what you want to do, click **Yes** to continue and click **OK** in the Capture Options dialog box. The next time you generate the notebook, this design will be recaptured using the new settings.

Note The *Recapture Image* capability only works if the option *Update images automatically* is enabled (the default state).

Saving Changes to the Notebook

Whenever you *Generate* the notebook, the information is saved automatically. But any time you have made changes to the notebook (such as modifying descriptions or reorganizing pages) that you want to keep, and you have not regenerated, click *Save* to explicitly save the changes.

Viewing an Existing Notebook

Whenever you want to view an existing notebook in the browser, launch the notebook (*Tools > Electronic Notebook*) and click *View*. Note that if you have made changes to any of the designs, or to the notebook itself, you must *Generate* to see those changes.

Updating an Existing Notebook

To modify or update an existing notebook:

1. From a Schematic window in the project of interest, choose **Tools** > **Electronic Notebook**.
2. When the notebook appears, make the desired changes.
   - By default, any design that has changed will be updated when you *Generate*. This behavior is controlled by the option *Update images automatically*, which is set individually for every page in the notebook.

     ☑ Update images automatically

     If you do not want a given image to be updated, disable this option.
   - If you have made changes to the individual image capture settings of several designs, you can click *Recapture All Images* (from the Notebook Properties pane) to enable the notebook to recapture these images when the notebook is regenerated.

   Note The *Recapture All Images* capability only works for designs where the option *Update images automatically* is enabled (the default state).

   - Modify any descriptions or rearrange pages as desired.
3. Click **Generate**. The notebook is regenerated to incorporate the changes, and is displayed in your browser.

Zipping the Files of a Notebook

To zip the files comprising the notebook so that they can be easily shared:

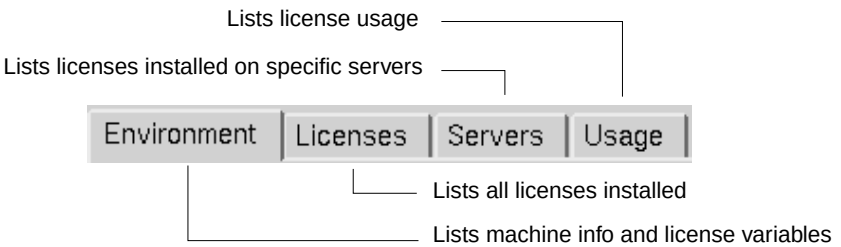
1. Click **Zip HTML**. You are prompted for a filename and location.
2. Changes paths as desired, supply a filename, and click **Save**.

   The recipient of the file can view the notebook by unzipping it and opening *index.html*.

Verifying License Status

The license information tool enables you to view the current status of your ADS licenses. There are two ways to launch the viewer:

- From the ADS Main window, through **Help** > **License Information**
- UNIX—From a terminal window, type **$HPEESOF_DIR/bin/aglmtool**, where *$HPEESOF_DIR* represents your complete installation path.
- PC—From Windows Explorer, locate <*install_dir*>/*bin* and double-click **aglmtool.exe** where <*install_dir*> represents your complete installation path.



Environment Lists environment/license variables and machine information that affect your ADS license configuration. This information can be very helpful when debugging license trouble. Use the *Compact View* option to wrap the lines making it easier to see where multiple path statements start and stop.

Licenses Lists all licenses found in the *license.dat* file installed on your computer, or in the case of a network installation, the license server. Use the *Compact View* option to wrap the lines making it easier to view complete license statements. Click *Refresh* if the *license.dat* file has been modified while viewing license information.

Servers Lists all license servers serving Agilent EEsof licenses on your network. You can expand any given server to see the licenses served by that server.

Usage This pane enables you to view the current status of all installed licenses. You can sort by *Licenses* (multiple licenses for a single feature) or by *Users* (select the *User Info* option first). Select All to view a complete list of installed licenses. Select *Available* to view only those licenses that are currently available. Select *In Use* to view only those licenses that are currently in use.

Using Keyboard Shortcuts and Accelerator Keys

Keyboard shortcuts are available for every command and are identified by the underlined character in every command on every menu. Accelerator keys are available for many menu commands and are identified by a key combination listed to the right of the command, where applicable.

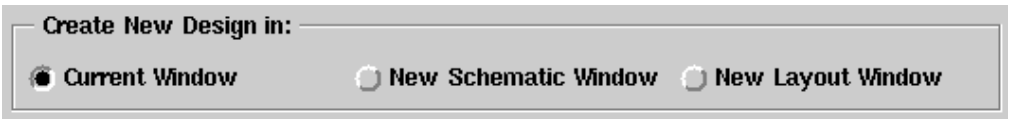
- For a table listing the default assignments, refer to Appendix B, Shortcut Keys.
- For details on changing the default assignments, refer to the section "Customizing Keyboard Shortcuts" on page 9-30.

Setting Preferences for Miscellaneous Options

The Preferences dialog box, accessed through the Options menu in the Main window, enables you to establish preferences for a variety of features that affect you throughout the design environment.

- **Warning Bell**—The system beeps anytime you receive a pop-up window with a warning message.
- **Error Bell**—The system beeps anytime you receive a pop-up window with an error message.
- **Balloon Help**—As you move your pointer over the toolbar and palette buttons, a small balloon appears with text describing that button's purpose (or a component's name).
- **Design Synchronization Checking**—You are warned if you attempt to simulate a design that is not fully synchronized.
- **Large Toolbar Bitmap**—A set of large bitmaps is placed on the toolbar. Turn this option off to place a set of small bitmaps on the toolbar (better for monitors with lower screen resolution). This change will be evident in any subsequently opened windows. To see the change take effect in a currently open window, open the Menu/Toolbar Configuration dialog box, click the Toolbar tab, and click OK.
- **Display Project Listing**—Filters the contents of the selected directory to display only project directories under the Project Listing heading.
- **Save Project State on Exit**—The setup of the project you are exiting is saved, including all design windows. The group of windows, and their positions on the screen, are restored the next time you open the project.

- **Create Initial Schematic Window**—A Schematic window opens automatically each time you create a project.
- **Create Initial Layout Window**—A Layout window opens automatically each time you create a project.
- **New/Open Design in New Window**—Sets a default in the New Design dialog box (*File > New Design*) that determines whether to open a new window for the new design or use the currently open window.



  Changing this setting does not affect the default setting in currently open Schematic/Layout windows, but will take affect in any subsequently opened Schematic/Layout windows.

- **Add Project Extension**—The extension you want appended to project names to clearly identify them as projects (default is *_prj*).
- **External Text Editor**—Specifies the text editor to be launched when you choose *Options > Text Editor* in the Main window.
- **Wire Thickness**—The thickness (Thin, Medium, Thick) of all wires drawn in a Schematic window.

To change any of these settings:

1. Choose **Options** > **Preferences**, in the Main window, and a dialog box appears.
2. Change any or all options as desired, and click **OK**. All changes take effect immediately, except as noted in the descriptions.

Exiting the Program

You can exit the program from the design windows or the Main window.

To close your project and exit the program:

Choose **File** > **Exit Advanced Design System** in any window.

- Click Yes to exit Advanced Design System
- Click No if you do not want to exit Advanced Design System

To save all designs in all windows, choose **File** > **Save All** in the Main window.

If any files with unsaved changes exist, a dialog box appears listing one of the files and offering the following choices:

- Yes—Click this to save changes to the named file and to be prompted individually for any additional files with unsaved changes
- No—Click this to disregard changes to the named file and to be prompted individually for any additional files with unsaved changes
- Yes To All—Click this to save changes to all files without being prompted individually
- No To All—Click this to disregard changes to all files without being prompted individually
- Cancel—Click this to cancel the command

Chapter 2: Managing Projects and Designs

This chapter describes the project directory concept, managing projects and design files, and the basics of importing and exporting designs. For information on these topics, review the following sections:

Working in Projects

All design work must be done in a project directory. Working in project directories enables you to organize related files within a predetermined file structure. This predetermined file structure consists of a set of subdirectories. These subdirectories are used in the following manner:

- *networks* contains schematic and layout information, as well as information needed for simulating
- *data* is the default directory location for input and output data files used or generated by the simulator
- *mom_dsn* contains designs created with the Agilent EEsof planar electromagnetic simulator, Momentum
- *synthesis* contains designs created with DSP filter and synthesis tools
- *verification* contains files generated by the Design Rule Checker (DRC), used with Layout

Creating a Project

You can create any number of project directories.

To create a project directory:

1. Choose **File** > **New Project** and a dialog box appears. By default, the path is set to your start-up directory.

Note Spaces are not allowed in project paths or project names.

2. Type a new path directly in the Name field or use the Browser to specify the location for the new project.
3. Enter a project name in the Name field.

Note By default, the suffix *_prj* is automatically added to the project name you supply. This behavior is defined by the option *Add Project Extension*. You can change this option through *Options> Preferences* in the Main window.

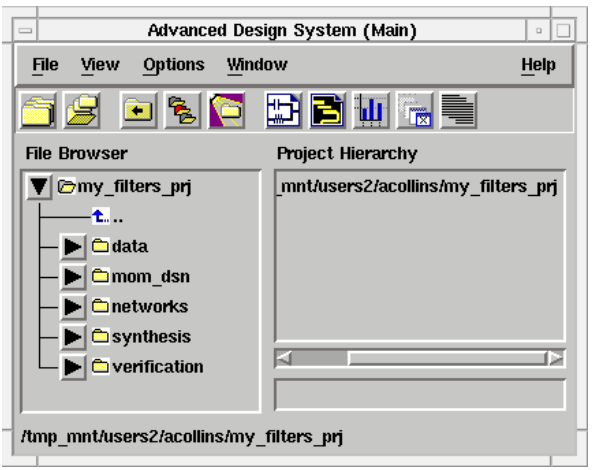
4. Click **Length Units** and select the appropriate setting.

This unit setting serves as a default for all designs in this project and is both:

- The unit of measure for parameters with physical length (in both Schematic and Layout windows)
- The design unit (grid display and cursor snapping) in the Layout window

Hint The design unit (grid display and cursor snapping) in the Schematic window is inches.

5. Click **OK** to establish Length Units.
6. Click **OK** to create the specified project. When the directory structure is complete, the path and the project name appear at the bottom of the Main window.



Hint By default, a Schematic window opens on creation of a project. This behavior is defined by the option *Create Initial Schematic Window*. You can turn this option off through *Options> Preferences* in the Main window.

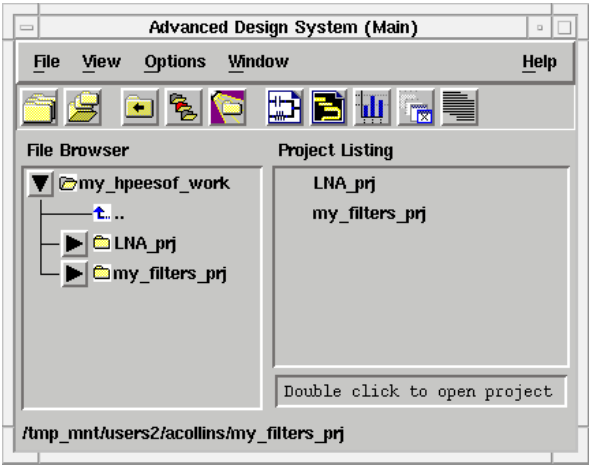
Opening a Project

The directory that appears as the current directory when you start the program varies by platform:

- UNIX—the directory from which you started the program. Once you have created project directories, you can start the program from a project directory, if desired.
- PC—the path you specified as the Work Directory during installation (by default, *c:\AdvDesSys*). You can set a different work directory through file *Properties*. (Right-click a program's shortcut icon and adjust the path in the *Start* field.) If you want to open a specific project directory while starting the program, use that project directory's name in the *Start in* field.

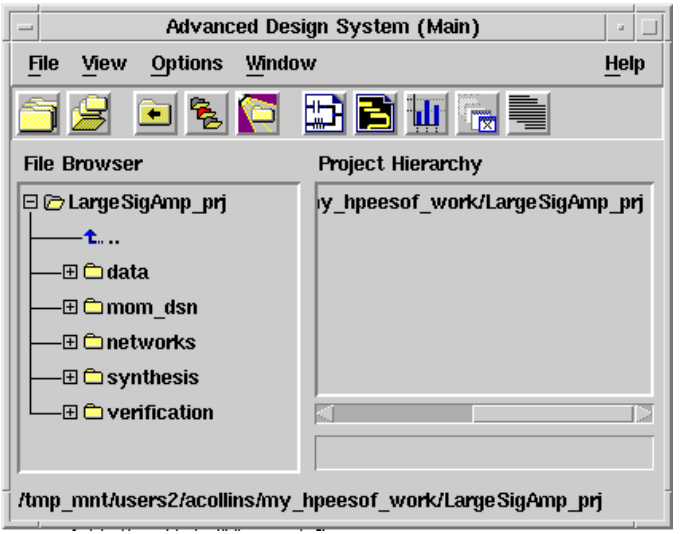
Once the Main window appears, there are two ways to open a project directory:

- Use the *File* > *Open Project* command
- Double-click the project name in the Project Listing pane of the Main window



To open a project using the *Open Project* command:

1. Choose **File** > **Open Project** and a dialog box appears. All projects in the current directory are listed in the Files list box.
2. Change directories as needed to find the directory containing the project.
3. Select the project name and click **OK**. Once the project is open, the right-hand group of toolbar buttons is activated and the path and project appear in the status panel at the bottom of the window.



To open a project using the *File Browser*:

1. Change directories as needed in the File Browser pane to locate the directory containing the project.
2. Choose **View** > **Project Listing**. All projects in the current directory are listed under the Project Listing pane. (For information on the Project Listing preference, refer to the section, "Setting Preferences for Miscellaneous Options" on page 1-34.)
3. Double-click to open the desired project. Once the project is open, the right-hand group of toolbar buttons is activated and the path and project appear in the status panel at the bottom of the window.

The windows that open when you open a project vary based on the following options:

- If the *Save Project State on Exit* option was enabled for this project, any design windows that were open when you last exited the project are restored
- If the *Save Project State on Exit* option was not enabled for this project, but the option *Create Initial Schematic window* was enabled (the default), a blank Schematic window appears.
- If neither of the aforementioned options was enabled, you must manually open a Schematic window. (Click the *New Schematic Window* toolbar button or choose *Window* > *New Schematic*.)

Hint For descriptions of these options, refer to "Setting Preferences for Miscellaneous Options" on page 1-34.

Copying a Project

The *Copy Project* command copies a project directory and its contents, to a new project directory with a name you specify.

Note Copying projects should only be done through the program, as described here. Copying projects outside the program may result in invalid projects.

To copy a project:

1. Choose **File** > **Copy Project.**
2. Locate the project you want to copy.

   Hint To copy an example project, click the *Example Directory* button to quickly set the path to the examples directory.

   Click **Browse** next to the From Project field.
3. In the dialog box that appears, change directories as needed to locate the project.
4. Select the project you want to copy and click **OK**.
5. Specify the destination directory for the copied project.

   Hint To copy the project to your start-up directory, click the *Startup Directory* button to quickly set the path to your start-up directory.

   Click **Browse** next to the To Project field.
6. In the dialog box that appears, select the destination path, and click **OK**.
7. Supply a project name, if desired, in the To Project field (following the path).

Hint If you want the copy to use the same name as the original project, you do not need to specify a name in the To Project field.

8. If the project you are copying is hierarchical, and you want to preserve the hierarchy, leave the *Copy Project Hierarchy* option enabled.

Note When copying a hierarchical project, you are prompted for each included project to confirm whether or not you want to copy that project. If you do not copy an included project (*Skip*), a reference to its source location is created in the copied project hierarchy. You are also prompted to supply a path and name for each copied project in the hierarchy. Click *Browse* to adjust the path without typing.

9. Click **OK**.

Deleting a Project

The *Delete* command enables you to delete a project directory and all its contents.

Note Deleting projects should only be done through the program, as described here. Deleting projects outside the program may result in program errors.

To delete a project:

1. Choose **File** > **Delete Project** and a dialog box appears.
2. Change the path as needed to locate the project directory you want to delete.

Note You cannot delete the current project directory.

3. Select the project and click **OK**. You are prompted to confirm that you want to delete that project directory.
4. Click *Yes* to delete it; click *No* to keep it.

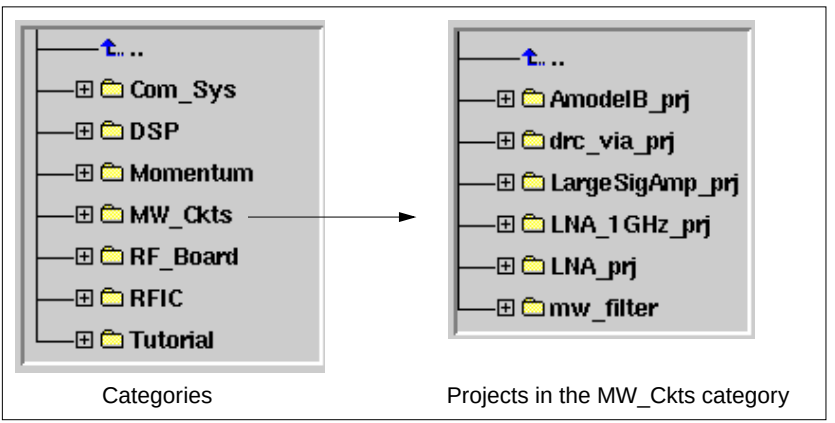
Using an Example Project

An extensive set of example projects is provided to demonstrate designing for various technologies.

To view the list of example projects:

1. In the Main window, click the **Examples** button on the toolbar. The File Browser pane changes to display the categories of examples available.

2. Click the category of interest to view the projects in that category.



To open an example project:

1. Use any of these methods:
   - Double-click the project name listed in the File Browser pane on the left.
   - Double-click the project name listed in the Project Listing pane on the right. (Refer to the Project Listing option described in "Setting Preferences for Miscellaneous Options" on page 1-34.)
   - Choose **File** > **Example Project**. Select the appropriate category and then the desired example.
2. Example projects are saved in a particular state and one or more designs will open automatically.
   - UNIX-Notice that when the Schematic window appears, the title bar reflects that the example is READ-ONLY. To simulate or modify designs in this project, make a copy of the project.

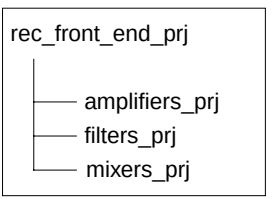
- PC-To preserve the example designs, make a copy of the project before modifying any of the designs in it.

Creating a Hierarchical Project

You can create hierarchical projects using the *Include/Remove Projects* command. This command creates a reference, or link, to another project. Hierarchical projects offer several benefits, including:

- The ability to create hierarchical designs by referencing designs from other projects
- The ability to maintain a single source of a design referenced by other users. Other users can *Include* the project containing the design of interest, and benefit from updates to the original design, since they are only *linking* to it, not copying it.
- Reduced disk space required for shared designs.

An example of a hierarchical project is shown next.



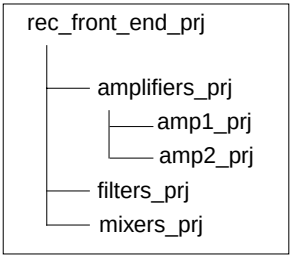
To include a project:

1. Open the project directory under which you want to include other projects.
2. Choose **File** > **Include/Remove Projects**.
3. Use the File Browser to locate and select the project you want to include.
4. Click the **Include** button. The project is added to the Project Hierarchy listing.
5. Repeat as needed, then click **OK**.

To remove an included project:

1. Open the project that includes the project to be removed.
2. Choose **File** > **Include/Remove Projects**.
3. Under Project Hierarchy, select the project you want to remove from the hierarchy and click the **Remove** button. The Project Hierarchy display is updated.

   If your hierarchical project includes other hierarchical projects, projects at lower levels cannot be removed unless the project in which they were originally *included* is the current project. For example, in the illustration shown next, you cannot remove the project *amp1_prj* from the hierarchy when *rec_front_end_prj* is the current project; you must make *amplifiers_prj* the current project first.



4. Repeat as needed, then click **OK**.

Working with Hierarchical Projects

Observing the following tips may simplify working with hierarchical projects:

- The title bar of the Schematic window displays the source of a project/design. This tip may help orient you when working with hierarchical designs within hierarchical projects. If you *Push Into Hierarchy* to view a design being referenced, the title bar displays the source of the design.
- If the project name in the title bar of the Schematic window does not match the current project name shown in the prompt panel of the Main window, simulation of that design is not allowed. This situation would occur, for example, if you place an instance of a design (as a subnetwork) from a project that is included in another project, and push into that subnetwork and attempt to simulate.

Archiving a Project

You can create a single file for a project, making it easy to transfer the project to another file system or another location on the same file system.

To archive a project:

1. In the Main window, choose **File** > **Archive Project**.
2. Use the Archive Project Browser and select the project you want to archive.
3. Use the To File Browser and select a path for the archived file. (Hint: You cannot archive directly to a floppy disk; you need more space for the process.)
4. Supply a name for the archived file. The extension *.zap* is automatically appended to the filename you supply.
5. If the project is hierarchical and you want to preserve the hierarchy, select the *Archive Project Hierarchy* option.
6. Click **OK**.

Hint If transferring projects back and forth between UNIX and PC, keep in mind that UNIX is case-sensitive and the PC is not. The PC does not always preserve case, so filenames going back and forth should be unique.

Unarchiving a Project

To unarchive a project:

1. In the Main window, choose **File** > **Unarchive Project**. (Hint: When you unarchive it, the project will be restored in the directory containing the archived file.) The file listing displays all files in the current directory with the extension *.zap*.
2. Change directories as needed to locate the archived project file.
3. Select the file and click **OK**. If the project is hierarchical, all projects are restored with their original name(s).

Hint The directory you use to unarchive the project cannot contain any subdirectory or filename of the same name as the archived project(s).

Importing and Exporting

The *Import* and *Export* commands enable you to import and export HP IFF files, as well as files in a variety of formats produced by other software. You can import files through the Main, Schematic, and Layout windows; exporting is done from Schematic and Layout. The listing below shows the available formats.

- For details on these formats and descriptions of the options associated with each, refer to the *Importing and Exporting Designs* manual
- For details on importing and exporting SPICE files, refer to the *Spice Netlist Translator* manual
- For details on importing Series IV designs, refer to the *Series IV Migration* manual

Table 2-1. Program Windows and Available Import/Export Formats

	Main	**Schematic**	**Layout**
Import	HP IFF	EGS Generate	DXF (hierarchical)
	Netlist	HPGL/2	EGS Archive
		HP IFF	EGS Generate
		Mask File (.msk)	GDSII Stream Format
		Netlist File	HPGL/2
			HP IFF
			IGES
			Mask File (.msk)
Export		HP IFF	DXF (hierarchical), (flattened)
			EGS Archive
			EGS Generate
			GDSII Stream Format
			Gerber
			Gerber Viewer
			HPGL/2
			HP IFF
			IGES

Table 2-1. Program Windows and Available Import/Export Formats (continued)

	Main	Schematic	Layout
			Mask File (.msk)
			MGC/PCB

Importing a Design

To import a design:

Note You must open a project before importing a design.

---

1. Choose **File** > **Import**.
2. In the dialog box that appears, select the appropriate file format from the File Type drop-down list.
3. To define options for the imported file, click **More Options** and dialog box appears.

   Note The program translators are controlled by translator options files. A system-wide options file exists for each translator. These files can be found in the *$HPEESOF_DIR/config* directory. The default system file is automatically read when you click *More Options* in the Import dialog box (unless a local options file already exists in the current project directory). When you make changes in the options dialog box and click OK, a local copy of the options file is written to the current project directory.

   ---

4. To specify the path and filename of the file you want to import, click **Browse**.
5. Double-click as needed to locate the directory containing the design. By default, all files are listed that have the file suffix appropriate for the chosen file format.
6. Select the design you want to import and click **OK**. You are returned to the Import dialog box and the selected filename appears in the field labeled Import File Name (Source).
7. Where applicable, type a new name for your imported design in the New Design Name (Destination) field. Note: For certain file types, the translator uses the existing filename to determine the new design name.

8. Click **OK** to import the design and dismiss the Import dialog box.

Exporting a Design

To export a file:

1. Choose **File** > **Export**.
2. In the dialog box that appears, select a file format from the File Type drop-down list.
3. To set export options, click **More Options** and a dialog box appears.

Note The program translators are controlled by translator options files. A system-wide options file exists for each translator. These files can be found in the *$HPEESOF_DIR/config* directory. The default system file is automatically read when you click *More Options* in the Export dialog box (unless a local options file already exists in the current project directory). When you make changes in the options dialog box and click OK, a local copy of the options file is written to the current project directory.

---

4. Change options as needed and click **OK**.
5. To specify a path for the exported file, click **Browse**.
6. Double-click as needed to locate the directory for the exported design. By default, all files are listed that have the file suffix appropriate for the chosen file format.
7. Click **OK**.
8. Type a new filename in the Export dialog box, following the path, and click **OK**. The file is written to the specified directory.

Managing Design Files

Before you begin your design work, you should understand the basics of file management within the design environment.

Creating a Design File

You can begin your design work in an untitled design window or supply a filename before you begin. When selecting a filename, keep in mind the guidelines described in the section, "Naming Conventions" on page 1-20.

To create a design file:

1. In a Schematic (or Layout) window, choose **File** > **New Design** and a dialog box appears.
2. Enter a design name in the Name field. The program automatically adds the extension *.dsn* to your filename.

Note If a project created on UNIX contains two or more designs whose names are only distinguishable from one another by differences in case, do not archive and transfer this project to a PC without renaming the designs such that they all have unique names. This requirement is due to the fact that the PC is case insensitive.

---

3. Where applicable (you selected *Both* in the initial or setup dialog box), select the design type.

   Analog/RF Network or *Digital Signal Processing Network*

   This affects the type of components available for design work.

Hint The choices presented here reflect the setting in the Advanced Design System Setup dialog box (Main window, Options menu).

---

4. Select the *New Window* option to create the new design in its own window. (This step is only required if you chose *File* > *New* in a window containing a design and you want to keep that design open in addition to creating a new design.)

5. Optionally, select a *Design Template* as a starting point for your design. (For details, refer to the section, "Using a Template" on page 2-16.)
6. Click **OK**. The design name is reflected in the title bar of the window.

Using a Template

Several simulation templates are provided as a convenience to help you create designs more quickly. You can turn any of your own designs into a template using the *Save Design As Template* command.

To start a new design with an existing template:

Choose **File** > **New Design**, select a template from the list, and click **OK**.

To add an existing template to an existing design:

Choose **Insert** > **Template**, select a template from the list, and click **OK**.

To create a new design for use as a template:

1. Create the design just as you would any other design.
2. Choose **File** > **Save Design As Template**. (The program automatically adds the extension *.tpl* to your filename.) The design is saved to your local templates directory and will now appear in the list of available templates (*File > New Design* and *Insert > Template*) when you choose to use one.

To modify a supplied template or one you have created:

1. Choose **Insert** > **Template**, select that template from the list, and click **OK**.
2. Make the desired changes and choose **File** > **Save Design As Template**.

Note To associate an AEL macro file with a template, give it the same name as the template file, add underscore *tpl (_tpl)*, and an *.ael* extension. Example: For a template filename of *my_amp.tpl*, name the AEL macro file *my_amp_tpl.ael*. Place this file in *home/hpeesof/circuit/templates* (or *home/hpeesof/hptolemy/templates*), along with the template file, and it will be loaded when you insert the template in another design.

Saving a Design File

There are three commands related to saving files: *Save Design*, *Save Design As*, and *Save Design As Template*.

- The *Save Design* command enables you to save changes to an existing file. (If you choose Save in an *untitled* window, the Save Design As dialog box appears.)
- The *Save Design As* command enables you to save an existing file with a new name. For example, to make a copy of an existing design so that you can edit it while preserving the original, use the *Save Design As* command to create a copy of the file with another name.
- Use the *Save Design As Template* command to save a design, at any stage, for use as a template. (Select a template for use through *File* > *New Design* or *Insert* > *Template*.) The program automatically adds the extension *.tpl* to your filename.

If you have made changes to a design and want to discard those changes, but continue working with the previously saved version of the design, choose **File** > **Revert to Saved Design**.

To save changes to an existing file:

Choose **File** > **Save Design**. If the file was previously saved (it resides on the disk), a dialog box appears.

- If you want to overwrite the old version with the new version, click **Yes**.
- If you do not want to overwrite the old version, click **No** and use the *Save Design As* command to save the design to another name.

To save an existing file with a new name, or to save a new file you have not yet named:

1. Choose **File** > **Save Design As** and a dialog box appears prompting you for a New File Name.
2. Enter a name for the design and click **OK**. The file is saved and automatically assigned an extension, which varies depending on the window (Schematic/Layout uses *.dsn*; Data Display uses *.dds*).

Note The *Save Design As* command names or renames all files associated with the design (*.dsn*, *.ael*, etc.) and is therefore the preferred method for saving a design to a new name. Therefore we recommend you use the design environment for all file management operations.

Saving All Designs in Memory

To save any designs currently in memory, including data displays, that have not been saved since changes were made:

In the Main window, choose **File** > **Save All**.

Opening an Existing Design

There are several ways to open existing designs:

- Double-click it in the Main window (from the Design Information pane)
- Double-click it from the Design Hierarchy dialog box (*View* > *Design Hierarchies*)
- Use the *File* > *Open Design* command in the appropriate design window
- If it is one of the last four designs opened, it appears on the file list at the bottom of the File menu and you can click there to open it
- If the design is currently open (in memory), it appears on the list at the bottom of the Window menu and you can click there to open it. You can also open it using the *Designs Open* command (on the Window menu).

To open an existing design from the browser in the Main window:

1. In the File Browser pane, click once to expand the networks directory and a list of all designs in that directory appears.
2. Double-click to select a design name. If the design is hierarchical, the hierarchy is listed under Design Information on the right side of the window; if the design is not hierarchical, its complete path and name appear under Design Information.
3. Double-click the design name from the right pane to open it.

To open an existing design from the File menu in a design window:

1. Choose **File** > **Open Design** and a dialog box appears.

   By default, the files displayed in this dialog box are located in the current project directory and have a *.dsn* extension.

2. Select the design you want to open from the Files list box, and click **OK**. The design appears in the window.

Important To make a complete set of files for a design in another project directory, use the *File* > *Copy Design* command in the Main window.

To view a design currently open (in memory) from the Window menu:

Choose **Window** (in the Schematic or Layout window) and select the design of interest from:

- The list of files at the bottom of the Window menu

  or

- The dialog box that appears when you choose *Designs Open*

  Note that when the number of designs in memory exceeds the number of designs displayed at the bottom of the Window menu, an additional menu choice, *More*, appears. Choose *More* to see a complete list of designs in memory.

Hint The number of designs displayed on the Window menu (by default, nine) can be customized by setting the variable DESIGN_LIST_COUNT equal to the desired value. For details, refer to Chapter 1, Customizing the ADS Environment, in the *Customization* manual.

Copying a Design

The *Copy Design* command copies all files associated with the design and is therefore the preferred method for copying designs.

Note Copying designs should only be done through the program, as described here. Copying designs outside the program may result in invalid designs.

To copy a design:

1. Choose **File** > **Copy Design** and click **Browse** (From Design).
2. Change directories as needed to locate the directory containing the design you want to copy.
3. Select the design and click **OK**.
4. Click **Browse** (To Path) and change directories as needed to specify the path for the copied design. Click **OK**.
5. Supply a name, following the path, for the copied design.
6. If the design is hierarchical, and you want to preserve the design in its entirety, select the *Copy Design Hierarchy* option.
7. When the From and To fields in the Copy Design dialog box reflect the appropriate paths and filenames, click **OK**.

Deleting a Design

To delete a design:

1. Choose **File** > **Delete Design** and a dialog box appears. By default, all *.dsn* files in the current project are listed.

   Note Deleting designs should only be done through the program, as described here.

   ---

2. Change directories as needed to locate the project containing the designs/files you want to delete.
3. Change the File Type as needed to locate the designs/files you want to delete. You can select a different file type from the drop-down list, or type a file extension and press **Enter** or click **Filter**.
4. Select the design/file you want to delete and click **Apply** (or *OK* if this is the only file you want to delete). You are prompted to confirm you want to delete that file. Click *Yes* to delete it; click *No* to keep it.

   Important If you delete a design that serves as a subnetwork in other designs, remember to delete all occurrences of that subnetwork in those designs.

Clearing a Design from Memory

Every design you open is stored in memory until you explicitly clear it from memory or exit the program. There are two ways to clear designs from memory: *Close Design*, *Delete All*. This distinction enables you to clear an entire design (both schematic and layout information, which are stored in the same file) from memory, or clear only schematic information or only layout information from memory.

- *Close Design*—File menu—clears the entire design file from memory

  If any unsaved changes are detected, you are prompted, *Design not saved, clear anyway?*

  - If you want to clear the design without saving, click **Yes**.
  - If you do not want to clear the design without saving, click **No**, and then select **Save Design** from the File menu.

- *Delete All*—Edit menu—clears the current schematic information, if issued from the Schematic window, or the layout information, if issued from the Layout window

---

Hint As with other commands found on the Edit menu, the *Undo* command works on *Delete All*.

Clearing All Designs from Memory

To clear all designs, including data displays, currently in memory:

In the Main window, choose **File** > **Close All**.

If any files with unsaved changes exist, a dialog box appears listing one of the files and offering the following choices:

- Yes—Saves changes to the named file and prompts you individually for any additional files with unsaved changes
- No—Discards changes to the named file and prompts you individually for any additional files with unsaved changes
- Yes To All—Saves changes to all files without prompting you for each one
- No To All—Discards changes to all files without prompting you for each one
- Cancel—Stops the *Close All* command

Chapter 3: Creating Designs

This chapter describes creating designs in the Advanced Design System environment. The starting point of this chapter assumes you have started the program, as described in Chapter 1, Program Basics, and that you are familiar with the project directory concept and design file management, as described in Chapter 2, Managing Projects and Designs.

Although you can perform a wide variety of editing operations on your design as you create it or once the design is complete, you can set numerous options before you begin your design work to minimize the need for editing. These options can be set through the Preferences dialog box (*Options* > *Preferences*) from a design window. Listed below are some of the options that can be set in advance to assist you in creating your designs:

- "Setting Placement Options" on page 9-9
- "Setting Entry/Edit Options" on page 9-12
- "Setting Grid/Snap Options" on page 9-6

These options, as well as others, are described in Chapter 9, Setting Design Environment Preferences.

Because all shapes and text are entered on layers, you may also want to modify layer definitions before beginning your design work. For details refer to the section, "Specifying Layer Definitions" on page 9-22.

For details on adding a drawing sheet to the design window, refer to the section, "Adding a Drawing Sheet" on page 7-1.

Defining Units for a Design

The design environment uses *Units* in a number of ways, differentiated as follows:

- *Layout Units*—the unit used for grid and snap features in the drawing area of the Layout window (*Options* > *Preferences* > *Layout Units*)

  Note The unit used for grid and snap features in the drawing area of the Schematic window is inches, and cannot be changed. When you see choices in various dialog boxes for *screen pixels* or *schem units*, the *schem units* are inches.

- *Length Unit*—the unit of measure for parameters with physical length in both the Schematic and Layout windows, and by default, the *Layout Units* of the Layout window

  When you create a project directory, you choose the *Length Unit* to be used as a default for all designs in that project. When working with the Layout option, we recommend keeping the *Length Unit* and the *Layout Units* the same, thus the *Length Unit* serves both purposes, by default.

  Note You can choose a different *Length Unit* for an individual design (*Options* > *Preferences* > *Units/Scale*) but a change made in this manner exists only in memory unless you save the preferences to a file. For details, refer to the section "Saving and Reading Preference Files" on page 9-20.

- *Units/Scale*—The scale factors shown in the *Units/Scale* tab of the Preferences dialog box serves as defaults in a limited number of situations. For details refer to the section, "Units/Scale Factors" on page 3-21.

Placing Components

You create designs by placing items (such as, components, data items, measurements, sources, simulation controls, etc.) in a design window. There are several ways to access these items:

- *Browsing*—in the Component Library window
- *Searching*—in the Component Library window
- *Component Palette*—on the left side of the design window
- *Component History*—on the toolbar or as a dialog box
- *Hot Keys*—once you have added components to the Component submenu

Although there are several methods of access, the basic steps for placing a component are the same.

To place a component in the design window:

1. Locate and click to select the component.
2. Move the pointer into the drawing area and click the orientation button to rotate the symbol as necessary.
3. Click to place the symbol in the desired location.

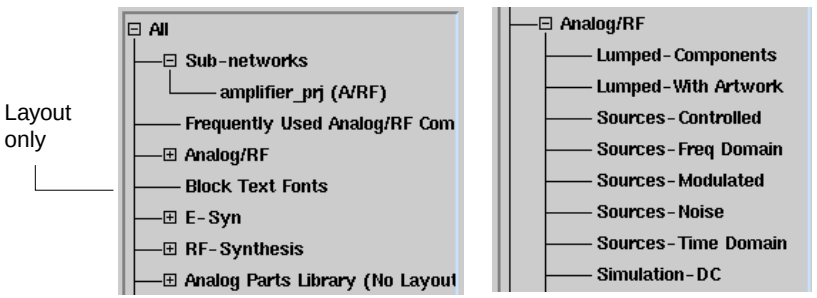
For additional information on working with components, refer to the following sources:

- For details on changing the orientation of a component, refer to the section, "Rotating Components" on page 3-19.
- For details on connecting components, refer to the section, "Connecting Components" on page 3-26.
- For details on changing component parameters, refer to the section, "Editing Component Parameters" on page 6-2.
- For details on changing attributes of component text:
  - Prior to placing components, refer to the section, "Setting Component Text/Wire Label Options (in Advance)" on page 9-15.
  - After placing components, refer to the section, "Changing Component Text Attributes" on page 6-10.

Browsing for Components

You can view the components that make up any individual library by selecting the library name in the Component Library window. The libraries listed vary with the current design type—*Analog/RF* versus. *Digital Signal Processing*.

In the following figure, the left side shows the top-level libraries (for the Analog/RF design type) collapsed; the right side shows a partial listing of the Analog/RF sub-libraries as it appears expanded.

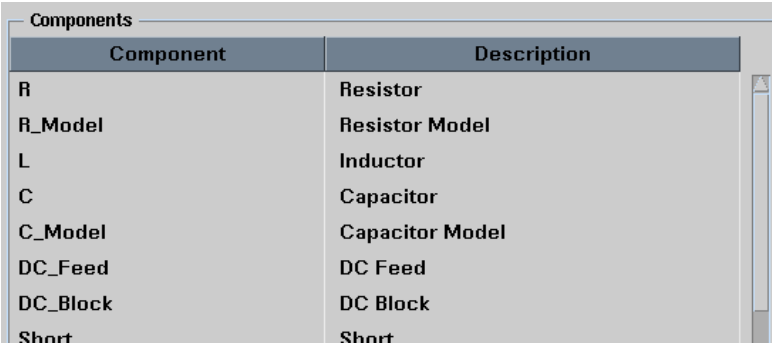


To browse for a component:

1. From any design window, click the **Library** button on the toolbar (or choose *Insert > Component > Component Library*).
2. Click the plus sign in front of a library name to expand it and view its sub-libraries.

   In addition to these libraries, a library named *Frequently Used Components* is filled, as you work, with the components you place, enabling you to quickly place additional instances of those components. Note that the *Frequently Used Components* library is project-specific; the library that is created in any given project directory will remain until you explicitly clear it.

   The *Subnetworks* library provides access to all the designs in the current project for use as subnetworks in other designs in that project. (To enable access to designs in other projects, create a hierarchical project as described in the section "Creating a Hierarchical Project" on page 2-9.)
3. Select a sub-library and its components are displayed in the Components section of the window. By default, a brief description for each component is also displayed.



Additional component information is available once you select a library. For details, refer to the section, "Customizing the Component Library Display" on page 3-8.

To place components from any library:

1. Select the library to display the list of components.
2. Select the component you want to place, move the pointer into the drawing area, and click to place it.

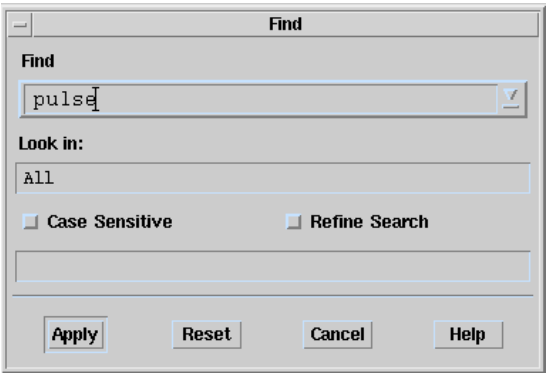
Hint To clear the current list of frequently used components, select the *Frequently Used Components* library and choose *Edit > Clear Frequently Used Components*.

Searching for Components

You can search for a component (or group of components with something in common) based upon text that is part of the component name or description.

To search for a component:

1. From any design window, click the **Library** button on the toolbar (or choose *Insert* > *Component* > *Component Library*).
2. In the Component Library window, choose **Tools** > **Find**.
3. Type the word or phrase you want to search for.



Note Only the library currently highlighted in the Libraries pane will be searched. By default, *All* libraries are searched. To narrow the search, select a specific library or sub-library. Your choice is reflected in the *Look in* field.

When the search is complete, the components and/or descriptions matching the search criteria are displayed.

The following figure shows the results of a search for the word *pulse* in all libraries.

Components

Component	Description
VtPulse	Voltage Source: Pulse with Edge Shape Bei
ItPulse	Current Source: Pulse with Edge Shape Be
VtBitSeq	Voltage Source: Pseudo Random Pulse Trai
VtLFSR_DT	Voltage Source: Pseudo-Random Pulse Tra
VtPulseDT	Voltage Source: Pulse Train Defined at Disc
VtImpulseDT	Voltage Source: Impulse Train Defined at D
VtOneShot	Voltage Source: Retriggerable Pulse Train
VtRF_Pulse	Voltage Source: RF Pulse
PtRF_Pulse	Power Source: RF Pulse Train
Vf_Pulse	Voltage Source: Fourier Series Expansion

To limit your initial search using case sensitivity:

1. Select the **Case Sensitive** option.
2. Type the search text matching the case of the component name(s) as it appears in the program.

   Example: The component you are looking for contains the word BEND, as part of its name, in uppercase and you want to search for all components that include BEND as part of their name, but exclude components whose descriptions contain the word Bend (mixed case).

To perform a secondary search on your initial search results to view some subset of those results:

1. Select the **Refine Search** option.
2. Change the search text in the Find field as needed and click **Apply**. The initial results are searched for anything meeting the revised search criteria.

   Example: Initiate a search through all libraries on the word amplifier. As you scan the resulting list, you notice several amplifiers with drain current of 17 mA scattered throughout. To view only those amplifiers, type id=17ma in the Find field, select the Refine Search option, and click Apply. The resulting subset of amplifiers is displayed.

Customizing the Component Library Display

Several features are provided to enable you to customize the Component Library display. You can:

- Collapse/expand the display of libraries and sub-libraries to minimize scrolling
- Add new libraries
- Rearrange the libraries and sub-libraries using Cut, Copy, and Paste
- Show or hide the display of several types of component information
- Set defaults for the widths of the various component information displays
- Alphabetically sort the display of libraries and components

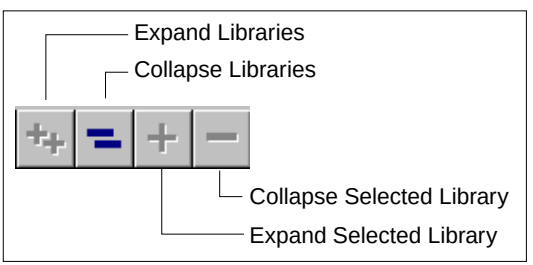
Once you customize the display to meet your needs, you can save the settings to file and use those settings in any subsequent session. For details refer to, "Saving Customized Library Displays" on page 3-12.

Note Several aspects of the browser display can be set for all projects (user) or on a site-wide basis. For details, refer to the *hpeesofbrowser.cfg* file described in Chapter 1, Customizing the ADS Environment, of the *Customization and Configuration* manual.

To expand a library so that its sub-libraries are displayed:

Click the plus sign in front of the library name. To collapse the library, click the minus sign.

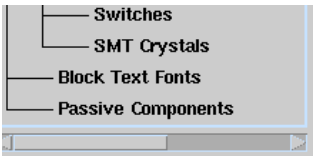
The expanding and collapsing of libraries can also be accomplished via toolbar buttons or the View menu (*View > Libraries*).



- Expand Libraries—Expands the view of all top-level libraries
- Collapse Libraries—Collapses the view of all top-level libraries
- Expand Selected Library—Expands the view of the currently selected library
- Collapse Selected Library—Collapses the view of the currently selected library

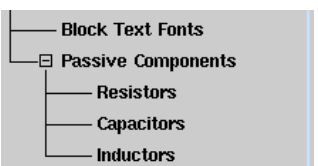
To add a new library:

1. Choose **File** > **New** > **Library**.
2. Supply a library name and click **Apply** (or OK if only creating one library at this time). The new library is added to the bottom of the current list of libraries.



To add a new sub-library:

1. Choose **File** > **New** > **Sub-Library**.
2. Select from the drop-down list the library you want the new sub-library to be part of.
3. Supply a sub-library name and click **Apply** (or OK if only creating one sub-library at this time). The new sub-library is added to the bottom of the current list of sub-libraries.



Rearranging Libraries

You can rearrange and delete libraries to make the library display better meet your design needs:

- Use *Cut* and *Paste* to rearrange the order of libraries—relative to other libraries, or the order of sub-libraries—relative to other sub-libraries. (Use *Copy* and *Paste* to copy a sub-library to another location).

  You can also turn a sub-library into a library by pasting it at the library level—highlight *All* and paste.

- Use *Cut* to delete libraries (or sub-libraries) you do not use for a specific project or are not licensed for, to help minimize scrolling.

Note that when you paste, the affected library or sub-library is pasted at the bottom of the list. Once you make your changes, you need to save your customized views. ("Saving Customized Library Displays" on page 3-12.)

To rearrange or delete libraries using Cut/Copy/Paste:

1. Highlight the library you want to move (or delete) and choose **Edit** > **Cut**.
2. To paste it at the bottom of the list of *All* libraries, highlight **All** and choose **Edit** > **Paste**.

To rearrange sub-libraries using Cut/Copy/Paste:

1. Highlight the sub-library you want to move, copy, or delete.

   Note Sub-libraries cannot be pasted within another sub-library.

2. Choose **Edit** > **Cut** (or **Copy**), and:
   - To paste the sub-library at the bottom of an existing library, highlight that library.
   - To paste the sub-library as a library (at the bottom of the list of all libraries), highlight *All*.
3. Choose **Edit** > **Paste**.

Setting Display Preferences

Several aspects of the Component Library display can be customized through *Options > Preferences*.

Component Tree Width

To modify the width of the tree in the Library pane:

Enter the desired value (in characters). Note that the change will not take effect until your next session of ADS.

Field Width

To set default widths (in characters) for the columns of component information:

In the Field Width section, set each field as desired. Note that these settings serve as defaults, but the columns can be resized manually by dragging the cell border one direction or the other. Setting the Field Width has no effect if the column status is hidden (deselected). To make it visible, select it in the *Show Columns* section of the dialog box.

Show Columns

The Component column is always visible. The Description column is the only optional column that is visible by default.

To show or hide optional columns of component information:

1. In the Show Columns section select or deselect any of the following:
   - Vendor—Displays the name of the manufacturer for the vendor parts in the selected sub-library.
   - Description—Provides a brief description of each component.
   - Library Name—Lists the name of the library for each component listed. This can be helpful if you find a component by searching, and want to know which library it is from.
   - Placement Status—Identifies for each component listed whether or not it is used in layout
   - Availability (Available/Obsolete)—Reserved for future use.
   - Web-site Address—Reserved for future use.
   - License Information—Reserved for future use.

2. Click **Apply**.

Show Components

To change the Show/Hide status of obsolete or unlicensed components:

1. In the Show Components section of the dialog box, set these options as desired:
   - Hide Obsolete Components
   - Hide Unlicensed Components
2. Click **Apply**.

Sorting Libraries and Components

To alphabetically sort the list of libraries:

1. Highlight any library under *All*.
2. In the Library Sort section of the Preferences dialog box, select the desired sorting method—Ascending or Descending (Unsorted is the default state) and click **Apply**.

To alphabetically sort the list of components for a specific sub-library:

1. Highlight the sub-library whose components you want to sort.
2. In the Component Sort section of the Preferences dialog box, select the desired sorting method—Ascending or Descending (Unsorted is the default state) and click **Apply**.

Saving Customized Library Displays

When you first open the Component Library—in any given session of ADS—the view of libraries is the *Default* view. This is the complete set of libraries as shipped. Changes you make to this view will persist during the current session, however, the next time you restart ADS, the *Default* view is restored. You can save any number of customized views and restore them later. By default, customized view files are saved to the current project directory, but you can save them in and restore them from any other project directory.

To save a customized view in the component library:

Choose **View** > **Save View As**. Supply a filename and click **Save**.

To open a previously saved customized view:

Choose **View** > **Open View**. Select the filename of the view you want to restore and click **Open**.

To close any currently open customized view:

Choose **View** > **Close View**. If no other customized views are open, the *Default* view is restored.

Resetting and Updating the Library Display

If during any given session, you cut libraries from the *Default* view and you want to restore them (without exiting and restarting ADS), you can do so with the *Reset/Update* command. This command will also update the library display with any libraries that were created through AEL.

To restore libraries cut from the Default view and/or update the display with new libraries:

Choose **View** > **Reset/Update**.

Note Some additional information is available for libraries and components through the Edit menu (*Library Properties* and *Component Properties*). These features will be enhanced in a future release.

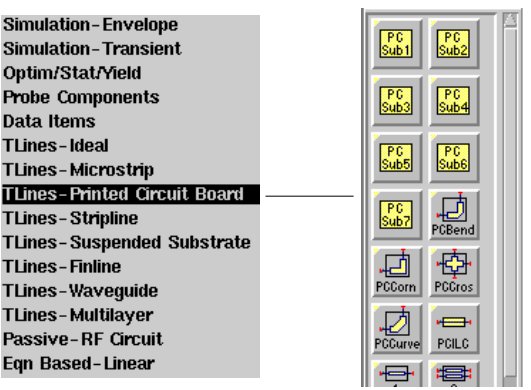
Using the Component Palette

The palette, on the left side of each design window, is a quick method of placing components if you know the name of the library containing the component. To change the library of components on the palette, select a new one from the drop-down *Palette List*.

Hint You can also bring the palette selection list up as a dialog box by selecting *View > Component > Select Component Palette*. Leave it on the screen if you want to change the palette frequently.

To use the *Palette* to place items:

1. Display the desired library on the palette by selecting it from the drop-down list (in this example, *TLines-Printed Circuit Board*).



2. If necessary, scroll the new palette to locate the button representing the component you want to place.

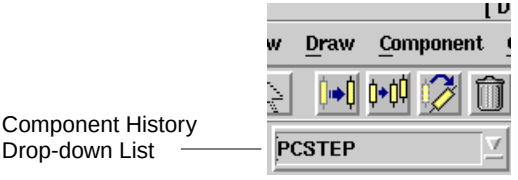
Using Component History

As you place components in your design, a history of these components is created. This dynamically created history enables you to quickly place additional instances of components placed in the current session.

Hint This history serves as the starting point for creating a custom component palette. For details refer to, "Creating a Custom Component Palette" on page 9-34.

To place items from Component History:

1. Click to access the drop-down Component History list or bring up the dialog box (*View > Component > Component History*).



2. If necessary, scroll the list to locate the component you want to place.

Hint Once vendor components are placed, they are part of Component History which can be used to create a custom palette containing these vendor parts.

Using Hot Keys to Place Components

You can place frequently used components via hot keys, by adding them to the Component submenu (*Insert* > *Component*) and then assigning hot keys to them. Components added to the menu in this manner apply to all projects. Customization of this Component menu is done separately for the Schematic and Layout windows.

To add components to the *Component* submenu:

1. Choose **Options** > **Hot Key/Toolbar Configuration** and in the dialog box that appears, select the **Component Menu/Hot Key** tab.
2. Select the components you want to add to the *Component* submenu from any of the following lists:
   - Available Components displays all the components contained in the current design.
   - Select any of the standard component libraries from the drop-down list.
   - Select *History* (for a list of components placed in the current session) or *Selected Design Components* (for a list of all currently selected components in the active design).

   Hint You can use the PC method of Shift+click to select a contiguous group of components, or Ctrl+click to select components that are not contiguous.

3. Click **Add** to add the selected component(s) to the Component Menu/Hot Key list box.

Alternatively, you can select one or more components currently placed in the drawing area of the desired (Schematic or Layout) window and choose **Insert** > **Component** > **Add Selected to Component Menu**. All selected components are added to the Component submenu.

To create hot keys for the menu-based components:

1. Choose **Options** > **Hot Key/Toolbar Configuration** and in the dialog box that appears, select the **Component Menu/Hot Key** tab.
2. Select a component from the Component Menu/Hot Key list box.
3. 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 or component, 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).

---

4. Click **Apply**. The shortcut appears next to the component.
5. Repeat as needed for each component.

To remove components from the *Component* submenu:

1. Choose **Options** > **Hot Key/Toolbar Configuration**.
2. In the dialog box that appears, select the **Component Menu/Hot Key** tab.
3. Select any component you want to delete from the *Component* submenu and click **Delete**. (To delete all but the standard components—Port, GROUND, VAR—click *Delete All*.)

Hint To remove a shortcut for a given component, select that component and deselect the modifier key(s) and erase the letter(s), then click Apply.

Placing Components at Specific Coordinates

The *Coordinate Entry* command enables you to place a component at specific coordinates. Pin 1 of the symbol is placed at the coordinates you specify.

To use the coordinate entry method for placing components:

1. Choose **Insert** > **Coordinate Entry** and a dialog box appears. Move the dialog box so that the desired design window is visible.
2. Select the component you want to place.
3. Select the desired orientation for this component.
4. Enter the desired X and Y coordinates in the dialog box.

   Enter coordinates by typing them directly in the fields labeled *X* and *Y*, or use the up and down arrows of the X Increment and Y Increment fields to increase or decrease the values of the *X* and *Y* fields. By default, the X and Y Increment fields are set to the current snap spacing, but you can use any increment that meets your design needs.
5. Click **Apply** and the component appears in the drawing area at the specified location.

Rotating Components

You can specify the orientation, or rotation, of components in a number of ways. To change the orientation during or after placing it in the drawing area, use any of the following methods.

- Click the **Rotate By Increment** button on the toolbar.
- Press **Ctrl+r**.
- Choose **Edit** > **Rotate**.

Each of these actions rotates the component *n* degrees clockwise, where *n* is the increment specified in *Options* > *Preferences* > *Entry/Edit* > *Rotation Increment (angle)*. The default is 90 degrees.

In addition to the above methods of rotation, you can rotate a component during the insertion process using any of the following commands:

- Choose **Insert** > **Component** > **Set Component Orientation UP**
- Choose **Insert** > **Component** > **Set Component Orientation DOWN**
- Choose **Insert** > **Component** > **Set Component Orientation LEFT**
- Choose **Insert** > **Component** > **Set Component Orientation RIGHT**

These directions refer to the direction the symbol is drawn relative to pin 1.

Hint Place these individual commands on the toolbar to have individual toolbar buttons for each orientation. For details refer to "Configuring Toolbars" on page 9-31.

Defining Parameters

As you place components in the drawing area, you will notice that some parameters have default values and that other parameters are followed by an equal sign (=) and nothing else. The lone equal sign indicates that a default value is defined elsewhere for this parameter. Some of these parameter values are defined by the simulator, while others are defined in various Simulation Control items. For details on where the default value is defined and to review the guidelines for choosing other values for that parameter, refer to the *Circuit Components* manual or the *Signal Processing Components* manual.

Note The *at* symbol (@) must be used to suppress quotes when specifying a variable as a parameter value, for example, use *@freq1*, where *freq1* is a variable declared in a VAR item.

You can change parameter values using the on-screen editor or through the Component Parameters dialog box. To display the Component Parameters dialog box for editing parameters after placing components:

- Position the pointer over the component you want to edit and double-click

or

- Select the component and choose *Edit > Component > Edit Component Parameters*

Alternatively, you can specify parameters for each component as you create your design. By default, the option controlling the automatic display of the Component Parameters dialog box is turned off (in the Schematic window). To display the dialog box automatically when you select a component, select the option *Show Component Parameter Dialog Box* through *Options > Preferences > Placement*.

Hint To keep the dialog box on your screen for editing parameters of many components, use the *Apply* button to effect changes for each component.

For details on the Component Parameters dialog box and using the on-screen editor, refer to "Editing Component Parameters" on page 6-2.

Units/Scale Factors

The fundamental units for ADS are shown in Table 3-1. An ADS parameter with a given dimension is evaluated based on the corresponding units. For example, for a resistance R=10, 10 is assumed to be 10 Ohms.

Table 3-1. Fundamental units in ADS

Dimension	Fundamental unit
Frequency	Hertz
Resistance	Ohms
Conductance	Siemens
Capacitance	Farads
Inductance	Henries
Length	meters
Time	seconds
Voltage	Volts
Current	Amperes
Power	Watts
Distance	meters
Temperature	Celsius

Variations on these fundamental units are referred to as *scale factors*. A scale factor is a single word/abbreviation that begins with a letter or an underscore character (_). The remaining characters, if any, consist of letters, digits, and underscores. The value of a given scale factor is resolved using the following rules, in the order shown:

1. If the scale factor exactly matches one of the predefined *scale-factor words* (see Table 3-2), then use its numerical equivalent

   else

2. If a scale factor exactly matches one of the *scale-factor units* (see Table 3-3) with the exception of *m*, then use its numerical equivalent

   else

3. If the first character of the scale factor is one of the *scale-factor prefixes* (see Table 3-4), then use its numerical equivalent

else

4. The scale factor is not recognized. When ADS does not recognize a scale factor it issues a warning and uses a scale-factor value of 1.0.

Table 3-2 lists the ADS scale-factor words and their numerical equivalents.

Table 3-2. Predefined scale-factor words

Scale Factor Words	Numerical Equivalent
mil	2.54e-5
mils	2.54e-5
cm	1.0e-2
in	2.54e-2
ft	12*2.54e-2
mi	5280*12*2.54e-2
nmi	1852
PHz	1.0e15
dB	1.0

Table 3-3 lists the ADS scale-factor units and their numerical equivalents.

Table 3-3. Scale-factor units

Scale Factor Unit	Meaning	Numerical Equivalent
A	Amperes	1.0
F	Farads	1.0
H	Henries	1.0
Hz	Hertz	1.0
meter	meters	1.0
meters		
metre		
metres		
Ohm	Ohms	1.0
Ohms		
S	Siemens	1.0

Table 3-3. Scale-factor units (continued)

Scale Factor Unit	Meaning	Numerical Equivalent
sec	seconds	1.0
V	Volts	1.0
W	Watts	1.0

Table 3-4 lists the ADS scale-factor prefixes and their numerical equivalents.

Table 3-4. ADS scale-factor prefixes

Scale Factor Prefixes	Meaning	Numerical Equivalent
a	atto	1e-18
f	femto	1e-15
p	pico	1e-12
n	nano	1e-9
u	micro	1e-6
m	milli	1e-3
_ (underscore)	no scale	1
k, K	kilo	1e3
M	Mega	1e6
G	Giga	1e9
T	Tera	1e12

Notes:

- Scale factors are case sensitive. Note the different meanings for *f* and *F*, and *a* and *A* in the preceding tables.
- The imperial units (mils, in, ft, mi, nmi) do not accept prefixes.
- Scale factors can be used anywhere in an expression (e.g., 1 GHz + 1 MHz).

When you select a parameter—with which a fundamental unit is associated—in the Component Parameter dialog box, a drop-down list appears containing all available scale factors for that particular parameter.

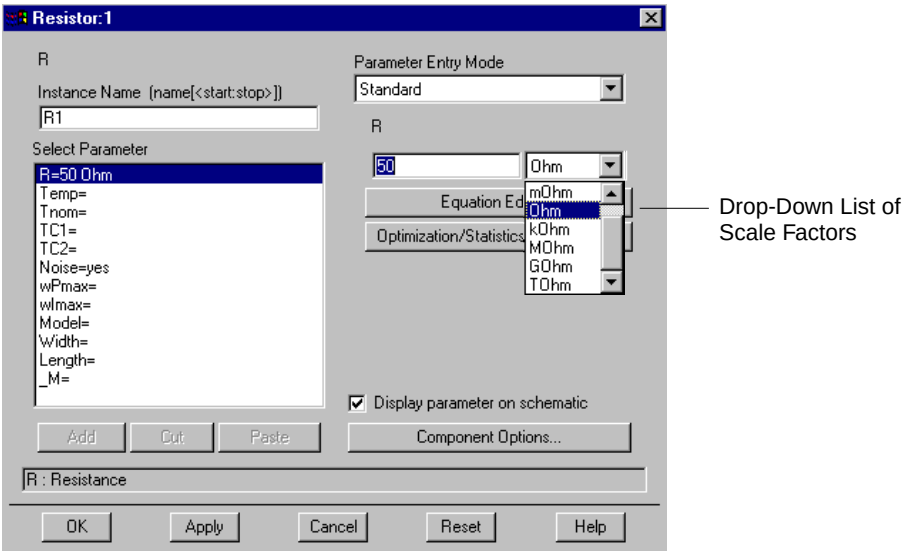


Table 3-5 lists the available scale factors for parameters with which a fundamental unit is associated.

Table 3-5. Available scale factors

Frequency	none	Hz	kHz	MHz	GHz	THz			
Resistance	none	mOhm	Ohm	kOhm	MOhm	GOhm	TOhm		
Conductance	none	pS	nS	uS	mS	S			
Capacitance	none	fF	pF	nF	uF	mF	F		
Inductance	none	fH	pH	nH	uH	mH	H		
Voltage	none	fV	pV	nV	uV	mV	V	kV	
Current	none	fA	pA	nA	uA	mA	A	kA	
Time	none	fsec	psec	nsec	usec	msec	sec		
Length	none	um	mm	cm	meter	mil	in	ft	
Distance	none	meter	km	ft	mi	nmi			
Power	none	pW	nW	uW	mW	W	kW	dBm	dBW

Notes:

- The option *none* means no scale factor is applied to the value.
- Although dBm and dBW appear on the drop-down lists, they are not valid scale factors; built-in functions convert these values to Watts and Celsius.
- There is no scale-factor option for temperature.

This same set of scale factors (with the exception of *none*) appears in the Preferences dialog box (*Options* > *Preferences* > *Units/Scale*). Note that these default settings are only used 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 box)
- When you supply a default parameter value without units while creating a parametric subnetwork (*File* > *Design Parameters*)

Hint You will see both *Length* and *Distance* scale factors in the Preferences dialog box. *Length* typically applies to parameters such as transmission line length, and *Distance* is a much larger scale factor that applies to things like antenna path length.

For details on how preference settings are associated with designs, refer to the section "Saving and Reading Preference Files" on page 9-20.

Measuring Distance and Angle

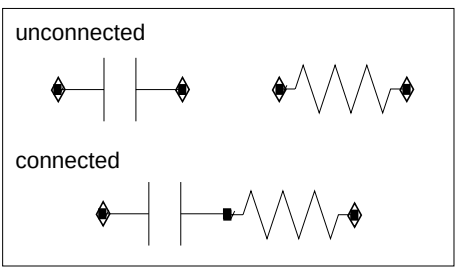
The *Measure* command produces a dialog box that displays the cumulative distance between points you specify in the design window. In addition, it displays the angle from the first point to the current point.

To use the Measure command:

1. Choose **Measure** from the Insert menu or the pop-up menu and a dialog box appears.
2. Click on the desired points in the design window and observe the information displayed in the dialog box.
   - To clear the information but keep the command active, click **Clear** or double click in the design window.
   - To stop the command and dismiss the dialog box, click **Cancel**.

Connecting Components

When you place a component in the design area, you will notice that each of the pins is highlighted by a diamond-shaped outline. This shape identifies unconnected pins, and disappears when a connection is made.



There are three ways to connect components:

- Directly, pin-to-pin
- With wires
- Without wires (with names)

Hint When a connection is made, the pin color changes to the color designated as the *Connected Pin* color (*Options > Preferences > Pins/Tees*). The color of the pin and the diamond-shaped outline prior to connection is the color designated as the *Highlight* color (*Options > Preferences > Display*).

Connecting Components Directly

To connect components directly:

1. Select the desired rotation for the component you are about to connect.
2. Position the pointer directly over the pin of the component you are connecting to, and click.

Connecting Components with Wires

To connect components with wires:

1. Click the **Insert Wire** button or choose **Wire** from the Insert menu. You are prompted to enter the starting point.
2. Position the pointer at the desired location and click. You are prompted to enter the next point.
3. Position the pointer at the desired location and click. A wire is drawn between the specified points.

Tips:

- To specify an endpoint of an unconnected wire, double-click or press the space bar.
- To move an unconnected endpoint of a wire, use *Edit > Move > Move Wire Endpoint*.
- Wires are always routed on the grid unless items connected to them were placed or moved off the grid. In this case, the shortest wire segment needed to make the connection is drawn off the grid as required. This means the maximum number of wires that connect to a single point is four.

This is especially important to understand when using multiple input components (such as *Add* and *Mpy)* in Signal Processing designs, because to connect more than four wires to the input, you will need to connect additional wires to one of the existing wires rather than to the input pin itself.

- Context-sensitive editing is available for wires with respect to changing the layer on which the wire is drawn. Position the pointer over the wire, right click, and select *Wire Layer* from the pop-up menu.

Connecting Components Without Wires

Connecting components without wires is accomplished by adding wire labels. Adding wire labels to your schematic enables you to:

- Indicate connectivity without wires. Two or more pins in the same network, with the same wire label, are connected as if they were wired.
- Identify node voltages— in an Analog/RF network—that you want output to the dataset after simulation. Note that if you assign a wire label to a pin that is connected to a wire, all pins connected to that wire will display that name.
- Create buses and bundles. For details, refer to the section that follows, "Creating Buses" on page 3-29.

For information on naming conventions, refer to "Naming Conventions" on page 1-20.

Hint Using the same node name in nodes across subnetworks does not result in connectivity unless a *Global Node* data item is placed in the design declaring that node name global. A *Global Node* data item with the specified node name will connect all nodes with that name across all subnetworks. This is useful for distribution of signals such as supply voltages and clocks.

To connect without wires or identify a node voltage for output:

1. Choose **Insert** > **Wire/Pin Label** and a dialog box appears.
2. Enter a Name and click all pins you want to connect with this name. As you click each pin, the name you have supplied appears near the pin.

   To produce output from a single pin, assign a unique node name to that pin.
3. To name another node, supply a different name, and click each pin, as described above. To dismiss the dialog box, click **Close**.

To clear a node name:

1. Choose **Edit** > **Wire/Pin Label** > **Remove Wire/Pin Label**.
2. Click each pin or wire whose name you want to remove.

   Alternatively, you can use the on-screen editor and the Backspace key.

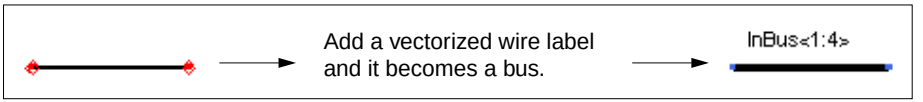
Creating Buses

Using buses and iterated instances can greatly simplify the schematic representation of your design. The following terms are used in describing how to create buses in ADS:

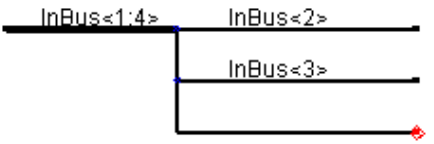
- Net—In the broadest sense, any connection between the pins of different electrical components
- Bus—A set of wires or a single wire carrying a set of signals. A bus is a kind of net or collection of nets where all members share the same *base name* (Data<1:4>⇒Data<1>, Data<2>,Data<3>,Data<4>).
- Wire Label—An identifying label used to name a net that enables referencing that net by name. A *simple* wire label (A) merely labels a wire for identification or connection purposes; a *vectorized* wire label (A<1:4>) identifies that wire as a bus and enables connection by *tapping*, using the base name and an index (A<2>). The vectors identify the bus width.
- Bundle—A collection of wires (or buses) that do not share the same base name (Data<1>,A,B).
- Iterated Instance—A single instance on the schematic that represents multiple instances. To iterate an instance you modify the Instance Name to include a vectorized label (*InstanceName*<1:4>).

Note See the example project, *Examples/Tutorials/wire_bus_prj*.

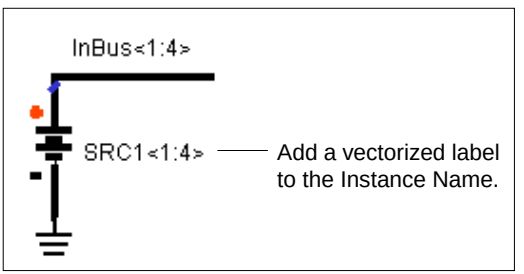
To represent a bus on an ADS schematic, add a *vectorized* wire label by supplying a base name and vectors that identify the bus width.



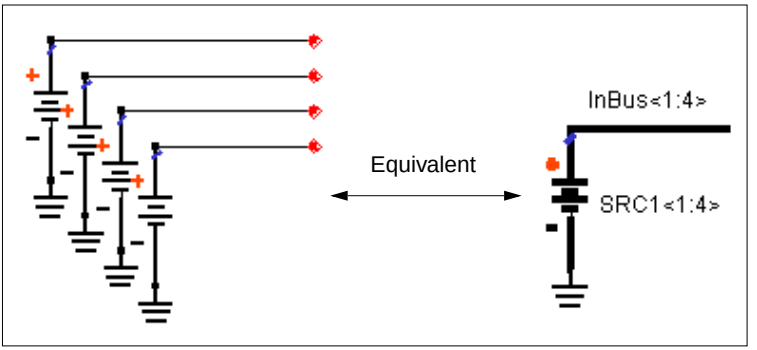
To tap the individual bus wires, use the bus base name and an index.



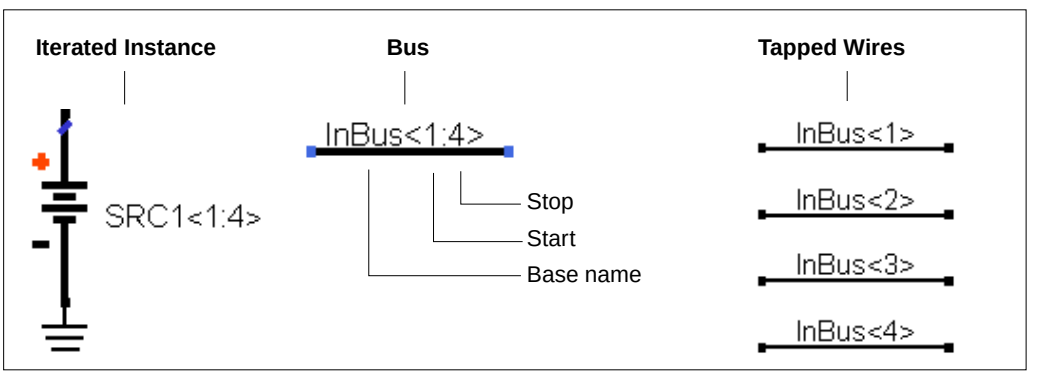
To iterate an instance (connected to the bus), modify the *Instance Name* by adding the vectorized portion of the bus wire label.



In the following illustration, the bus and iterated instance on the right are equivalent to the four grounded sources connected to individual wires on the left.



The labels you use to identify the bus and everything connected to it must be added using the appropriate syntax. The terminology shown in the following illustration identifies the terms used to define this syntax.



Syntax:

Bus

base name<start:stop>

Example: InBus<1:4>

or

base name<start:stop:increment> for an increment other than 1. Use this to create patterns such as {2,4,6,8} and {1,3,5,7}.

Example: InBus<2:8:2>

Tapped Wires

base name<index>

Example: InBus<1>

Iterated Instance

instance_name<start:stop>

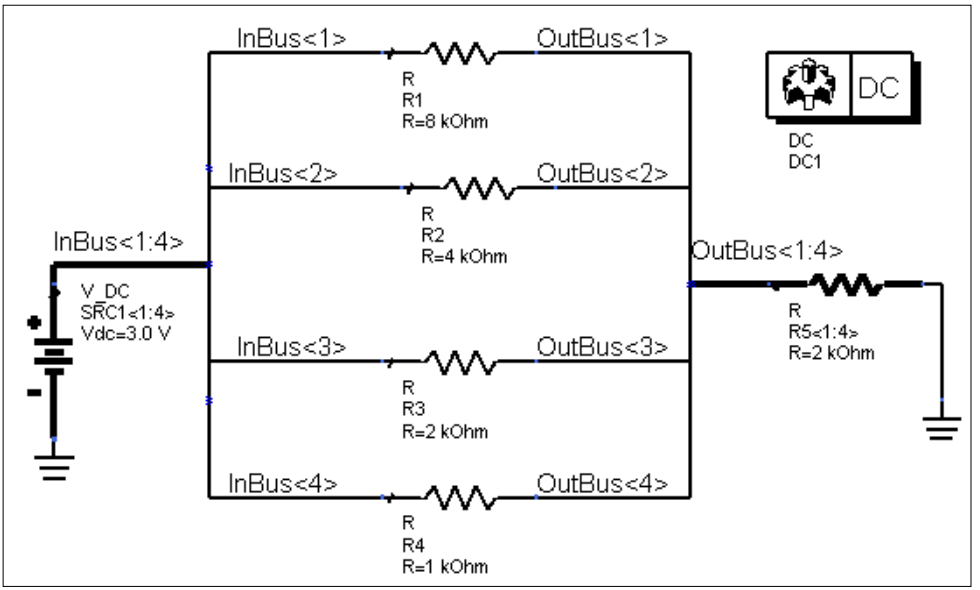
where *instance_name* is the actual instance name you are using for this component, and the vector information tracks with the bus vectors, including an increment, where applicable.

Example: SRC1<1:4>

Note that pins and ports can be iterated in the same manner. For details refer to, "Bus Pins and Iterated Ports" on page 3-37.

Important Be sure to use the correct syntax for all related parts of a bus; syntax errors produce connectivity errors.

The following illustration is a simple example using input and output buses consisting of four wires each.



The basic steps to include this type of bus in your schematic are as follows:

- Designate the desired wire as a bus by adding a *vectorized wire label*, for example, *InBus<1:4>*.
- Label the tapped wires of the bus by using the same *base name* you used for the bus (in this example, *InBus*), and supplying an index to indicate which bus wire it is.
- Add vectors (in this example, <1:4>) to the instance name of the component connected to the bus (in this example, V_DC) to signify that it represents multiple instances or an *iterated* instance.

Notes:

- Occasionally the visual rendering of a bus wire might not appear as expected. But as long as it is named correctly, connectivity will be correct.
- When a port is connected to an iterated instance, the port must be iterated as well.
- Setting the value of a parameter of an iterated instance sets that parameter to that value for all instances created by the iteration.
- Iterated instances cannot be tuned, swept, or optimized directly. To perform any of these operations on an iterated instance, you must create an intermediate variable representing the parameter of interest. You can then set the instance parameter to this variable, and sweep/tune/optimize the variable, indirectly sweeping/tuning/optimizing the instance parameter.
- It is not necessary to create a bus on the schematic that includes all of the array indices. The simulator will collect all bus references with the same base name into a single array (vector) with the required range of indices. For example a schematic with A<0>, A<1>, A<2>, A<3> and A<4:15> will create a vector A<0:15>.
- Negative indices are not allowed, although a negative increment is. You can effect a countdown using indices such as A<4:1> which contains the nets A<4>, A<3>, A<2>, A<1>, or use a negative increment if the increment is other than 1, such as <8:2:-2> which contains the nets A<8>, A<6>, A<4>, A<2>.
- When choosing a bus *base name*, keep in mind that the name must adhere to the same rules as node names. For details, refer to "Naming Conventions" on page 1-20 in the *User's Guide*.

To create a bus or bundle:

1. Draw the wire and choose **Insert** > **Wire/Pin Label** and a dialog box appears.
   - Bus—Enter the bus base name using the appropriate syntax and click the wire that you want to designate as a bus.
   - Bundle—Enter the names of the wires (or buses) that should be part of the bundle and click the wire that you want to designate as a bundle.
2. The label appears and the rendering of the wire changes to a thicker line indicating that the wire now represents a bus/bundle. Click **Close**.

To tap wires off the bus:

1. Draw the bus tap wire, choose **Insert** > **Wire/Pin Label** and a dialog box appears.
2. Enter the bus base name and the index for the bus wire you want to tap, and click that wire. Increment the index in the dialog box as needed and click the next wire. Continue in the same manner until you have tapped all desired wires of the bus, then click **Close**.

To iterate an instance of a component/port or a symbol pin:

1. Double-click the instance or pin to display the dialog box for editing that item.
2. Add the appropriate iteration syntax to the instance name and click **OK**.

   For components and ports, you can add labels using the on-screen editor; for pins you must use the *Edit* > *Symbol Pin* dialog box (*View* > *Create/Edit Schematic Symbol*).

Wire labels can be edited in the following ways:

- To move a wire label—Position the pointer over the bus/bundle label you want to move. Press the left mouse button, drag the label (you will see a ghosted image) to the desired location, and release. Note that moving a label a very small amount is affected by whether the *Snap Enabled* option is on or off (*Options* menu toggle), and the size of the *Drag and Move* threshold option (*Options* > *Preferences Entry/Edit*). You can also use *Edit* > *Move* > *Move Wire/Pin Label*.
- To set the color, size, and font of wire labels in advance—Use *Options* > *Preferences* > *Component Text/Wire Label*.

- To change the color, size, and font of existing wire labels—Choose one of the following methods:
  - Use *Edit > Wire/Pin Label > Wire/Pin Label Attributes*
  - Double-click the wire label to display the dialog box
  - Right-click and select *Wire/Pin Label Attributes* from the pop-up menu
- To delete a wire label, use one of the following methods:
  - Invoke the on-screen editor and use the Backspace key (followed by *Esc*)
  - Choose *Edit > Wire/Pin Label > Remove Wire Label* and click the wire whose label you want to delete.
- To modify the label itself:
  - Double-click the wire to display the Wire/Pin Label dialog box

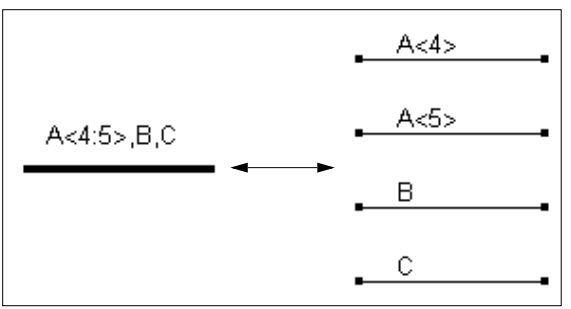
Creating Bundles

A bundle is a collection of wires or buses that do not share the same base name. To indicate a bundle on your schematic, add a wire label consisting of a list of the wire labels and/or buses to be included in the bundle, separated by commas.

Syntax:

<wire_label/bus>, <wire_label/bus>, ...

Any wire/bus in the bundle can be accessed by name. In the following example, if you want to access A<4:5>, you can name a net A<4:5>.

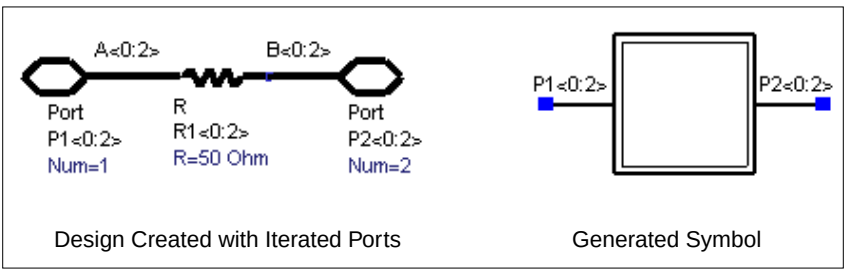


Bus Pins and Iterated Ports

A *bus pin* enables you to connect a bus or bundle to it. If you generate a symbol for a design containing iterated ports, the symbol generator will create bus pins for you. If you create a custom symbol for a design containing iterated ports, you must explicitly assign an iteration to the symbol pins on your custom symbol.

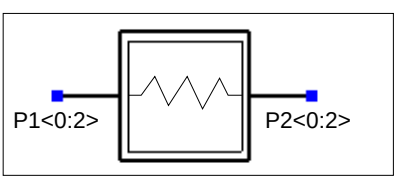
Creating a Bus Pin via Iterated Ports and a Generated Symbol

When you iterate one or more ports on your schematic (by modifying the Instance Name), then generate a symbol for it, the generated symbol is created with bus pins.



Creating a Bus Pin on a Custom Symbol

To explicitly create bus pins for a custom symbol representing a design containing iterated ports, modify the pin name (*Edit > Symbol Pin*), using the same syntax as for a *bus* or *bundle* (e.g., P1<1:4> or A,B) in symbol view (*View > Create/Edit Schematic Symbol*). The number of iterated ports in the schematic must always equal the number of symbol pins. This equivalence can be checked using *Options > Check Representation > Rep port vs Symbol port mismatch*.

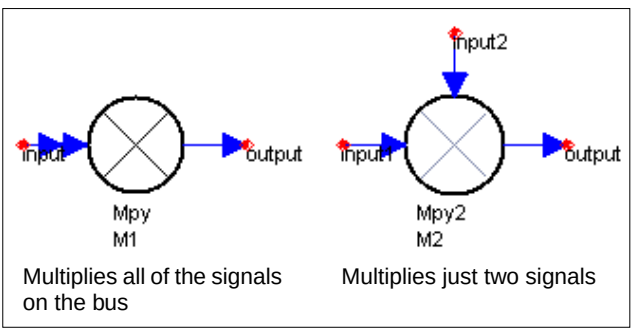


Buses in Ptolemy

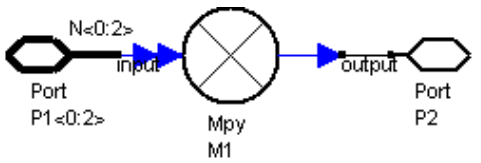
Agilent Ptolemy provides a different use model for buses, and there are two types: *MultiPortHoles* and the *fixed-point* data type (which uses components—not wires—for bus manipulations).

A MultiPortHole is an ordered set of PortHoles that can be expanded dynamically. A PortHole is equivalent to a pin in ADS. Pins that have double arrows are MultiPortHoles that initially have zero PortHoles. As each connection is made on the bus, a new PortHole is added to the MultiPortHole. Thus in this configuration, the MultiPortHole pin represents a bus input.

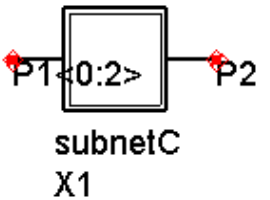
A MultiPortHole can also be expanded (using netlist syntax) to a predefined size. These MultiPortHoles are represented in the design environment as *n* separate pins where *n* is the number of PortHoles in the MultiPortHole. All of the current PortHoles and MultiPortHoles are directional, in contrast to the non-directional pins of analog components. The following illustration shows the Agilent Ptolemy multiplier components.



A MultiPortHole that is represented as a double arrow is equivalent to a *bus pin* with unspecified size. To fix the size, add wire labels (in the schematic) just as you would for an Analog/RF design, generate a symbol for it, and place it as a subnetwork, as shown next.



Design Created with an Iterated Port



Design Placed as a Subnetwork

Checking Connectivity

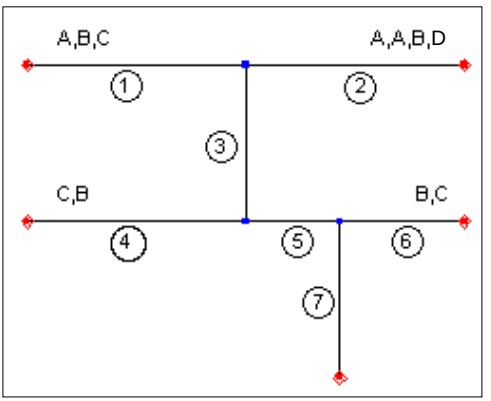
Although a number of operations automatically perform a connection check, you can explicitly test the bus-to-instance connections in a schematic through *Options > Check Representation > Bus Connectivity*. The connection check attempts to resolve all bus-instance connections, and highlights (on the schematic) any connections it is unable to resolve. Connections that are part of a bus or bundle are verified using a set of rules. The following terms are used in describing these rules:

- Connection Width—The total number of pins at a given port of an *iterated instance*. Computed by multiplying the pin width at the port by the *instance width*.
- Instance Width—The number of instances in an iterated instance (I<0:3> represents 4 instances)
- Net Width—The number of individual wires in a bus/bundle (N<0:7> represents 8 wires)
- Pin Width—The number of pins at the given port of a component

Resolving Connections Related to Names

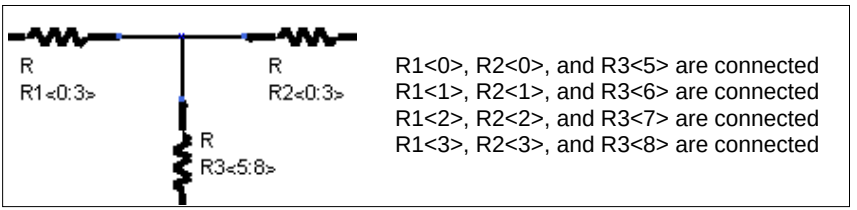
When a net includes both named and unnamed wire segments, the unnamed segments are considered to include every name that appears on all named segments. In other words, the name implicitly assigned to an unnamed segment is the intersection of all names on named segments.

In the following illustration, the segments 3, 5, and 7 are unnamed. A, B, C, D is the intersection of all names on this net, and this is the name that is used for all three unnamed segments.

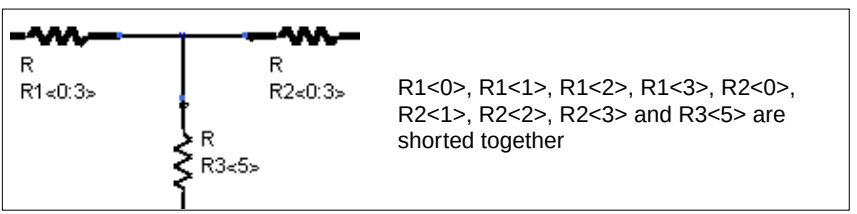


In an unnamed net (one with no named wires at all):

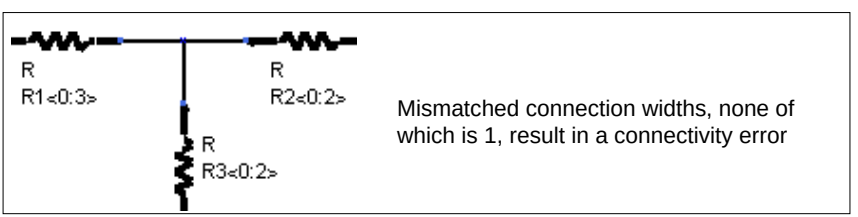
- If all instances to which this wire network is connected have a connection width of N, then each wire in the network is given width N.



- If *any* instance to which this wire network is connected has a connection width of 1, then the entire wire network is given width 1. Instances with connection widths greater than 1 will be shorted together.



- If the instances to which this wire network are connected have mismatched connection widths, and none of them have a connection width of 1, then there is a connectivity error.



Resolving Connections Related to Widths

To verify a connection, the net width of the bus is checked against the pin width of the component, the instance width, and the connection width (where the connection width represents the total number of pins: connection width = pin width * instance width).

Agilent Ptolemy incorporates the concept of pins that can accept buses as inputs. These pins are of arbitrary size and are annotated by double arrows on the pin stem. For the Ptolemy MultiPortHole examples:

- Wires without names—connected to MultiPortHole pins—have unspecified width.
- MultiPortHoles without a pin width have an unspecified pin width.
- Instances without an explicit width have an instance width of 1 (in other words, the instance width can never be unspecified because it defaults to 1).

In the following descriptions of how various connection types are resolved, some connection types apply only to Digital Signal Processing (DSP) designs while others apply to both Analog/RF (A/RF) and DSP designs:

- "Unspecified net width, multiple input or multiple output connections (DSP only)" on page 3-43
- "Unspecified input or output pin width, specified net width (DSP only)" on page 3-45
- "Unspecified pin width, unspecified net width, specified instance width" on page 3-45
- "Unspecified net width, pin width specified, instance width specified" on page 3-45
- "Net width = connection width" on page 3-46
- "Net width = pin width" on page 3-47
- "Net width = 1" on page 3-47

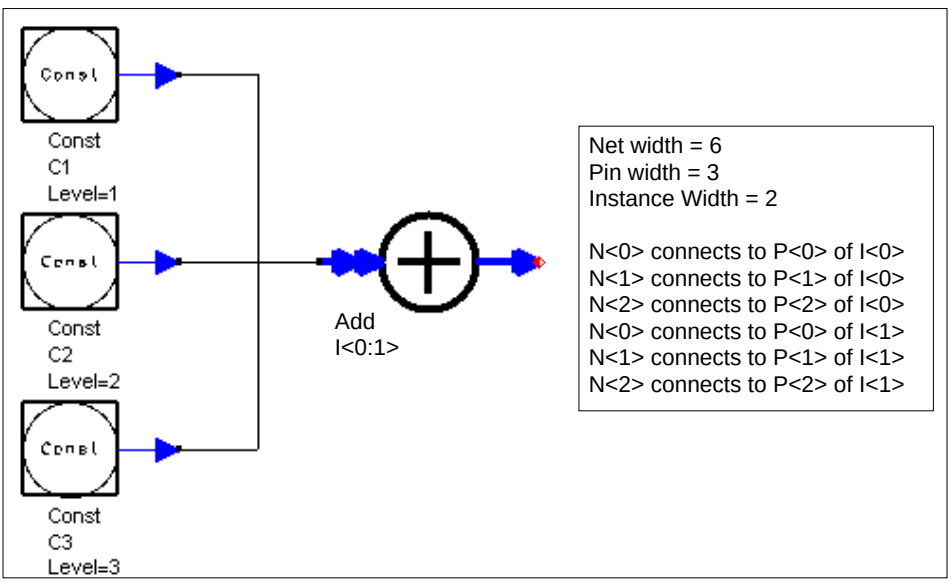
If none of the conditions above apply, then the connection is not valid, and will be flagged as an error. When an invalid connection is found during *connection checking*, that invalid connection will be highlighted on the schematic.

Unspecified net width, multiple input or multiple output connections (DSP only)

Agilent Ptolemy currently supports this connection type for instance widths equal to one. Only the following two cases are supported: multiple outputs feeding a multiport input and a single output feeding multiple inputs.

- Multiple outputs feeding a multiport input
  - Net width = number of outputs
  - Pin width of input = net width
  - Pin width of any unspecified output pin = 1

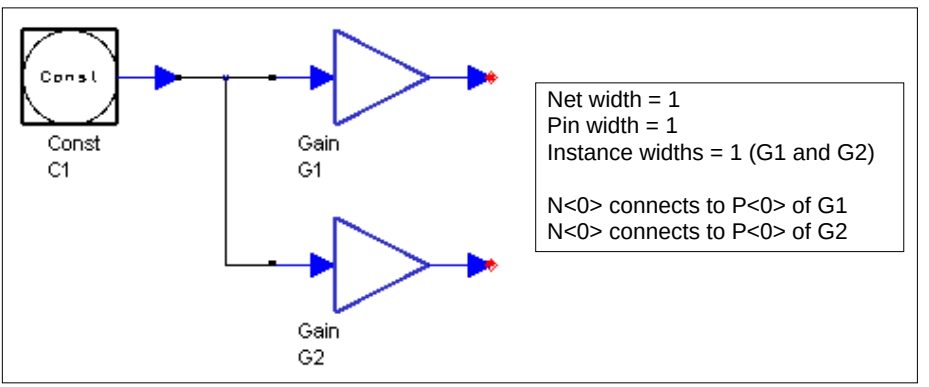
This connection only works for commutative operators because it does not guarantee the ordering of the bus.



To specify the exact ordering of the net, you must add wire labels or use the *BusSplit* or *BusMerge* operators.

- Output feeding multiple inputs:
  - Net width = 1
  - Pin width of output = 1
  - Pin width of any unspecified input pin = 1

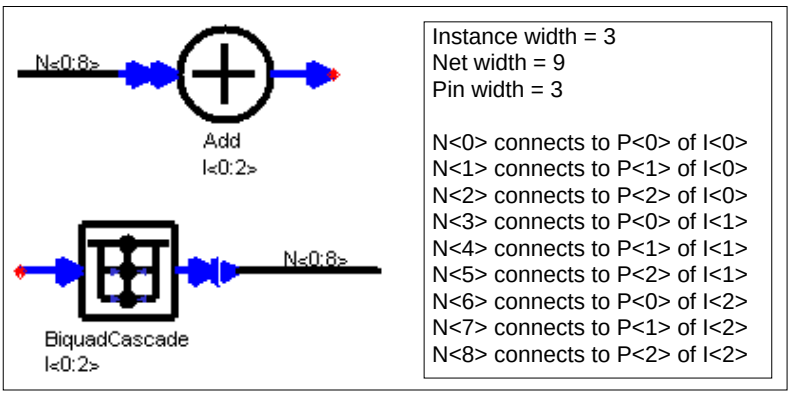
The problem with named nodes seen in the previous example does not occur in the next example because the net width is resolved to 1. Ptolemy interprets this as a single wire and replicates the output. The output here can be either a uniport or multiport pin.



Unspecified input or output pin width, specified net width (DSP only)

If the pin width is unspecified and the net width is an integer multiple of the instance width then:

- Pin width = net width/instance width



For the *Add* example above, the Ptolemy post-processor will need to recognize that the input/output pin is connected to N<0>,N<1>,N<2> for I<0>, N<3>,N<4>,N<5> for I<1> and N<6>,N<7>,N<8> for I<2>. An example netlist follows:

```
Options ResourceUsage=yes
UseNutmegFormat=no
TopDesignName="C:\test_prj\networks\buses"
_vAgilentEEsof_dSDF_nAdd_lsdfstars:I<0>  N<0>,N<1>,N<2>  _net1
_vAgilentEEsof_dSDF_nAdd_lsdfstars:I<1>  N<3>,N<4>,N<5>  _net2
_vAgilentEEsof_dSDF_nAdd_lsdfstars:I<2>  N<6>,N<7>,N<8>  _net3
```

Unspecified pin width, unspecified net width, specified instance width

Pin width is set to 1 and net width is set to the instance width.

Unspecified net width, pin width specified, instance width specified

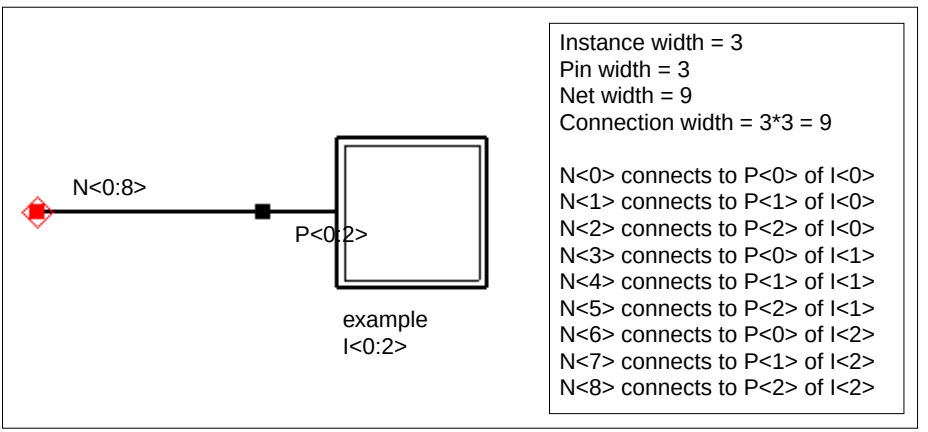
Net width inherits connection width (pin width * instance width).

For the next three cases, everything (pin, instance and net width) is specified. The first case that applies (based on the order shown next) is the correct case.

- *Net width = connection width*
- *Net width = pin width*
- *Net width = 1*

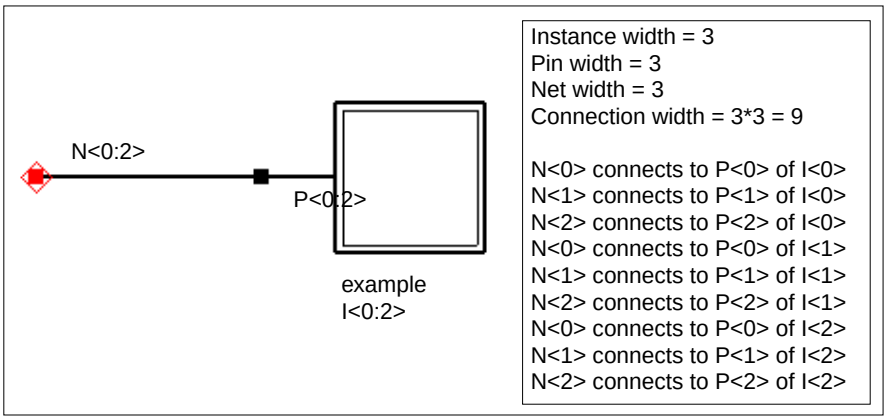
Net width = connection width

The net width of the bus/bundle is the same as the connection width. This connection is resolved and verified because the total number of pins (connection width) matches the individual wires available.



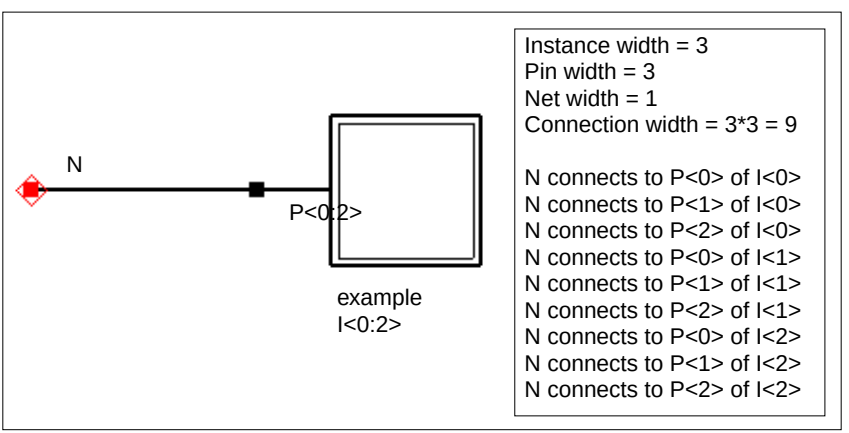
Net width = pin width

If the net width and the pin width are the same, but the connection width is different, then the connection is valid, but the wires making up the bus/bundle must be repeated *M* times, where *M* is the instance width.

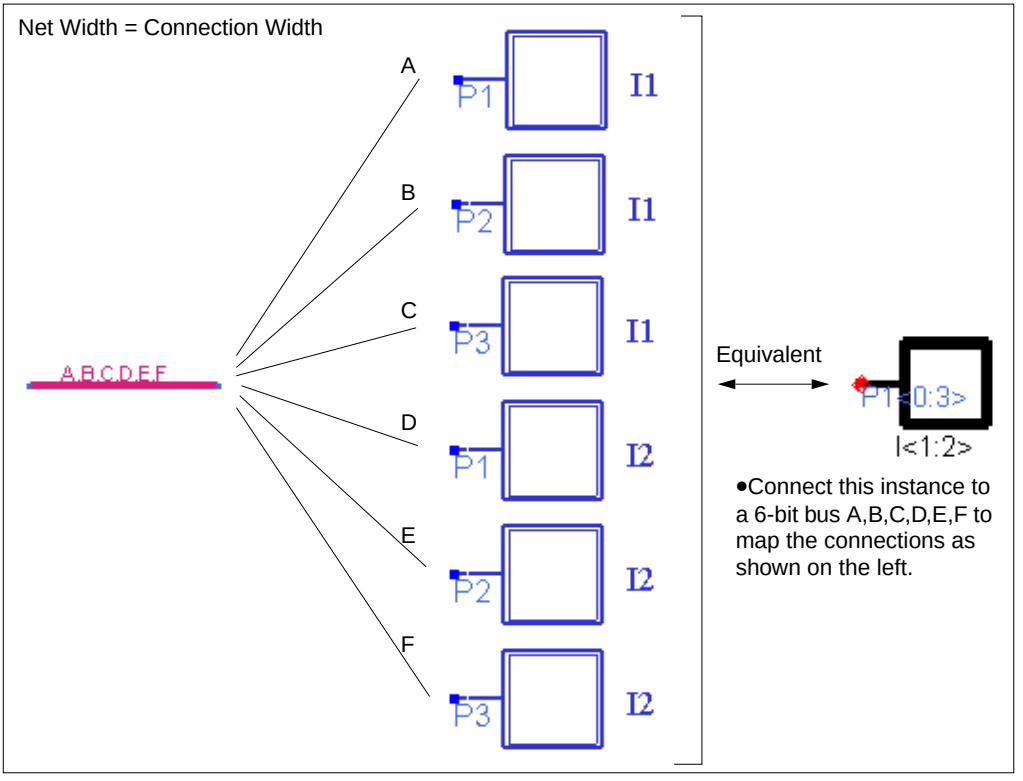


Net width = 1

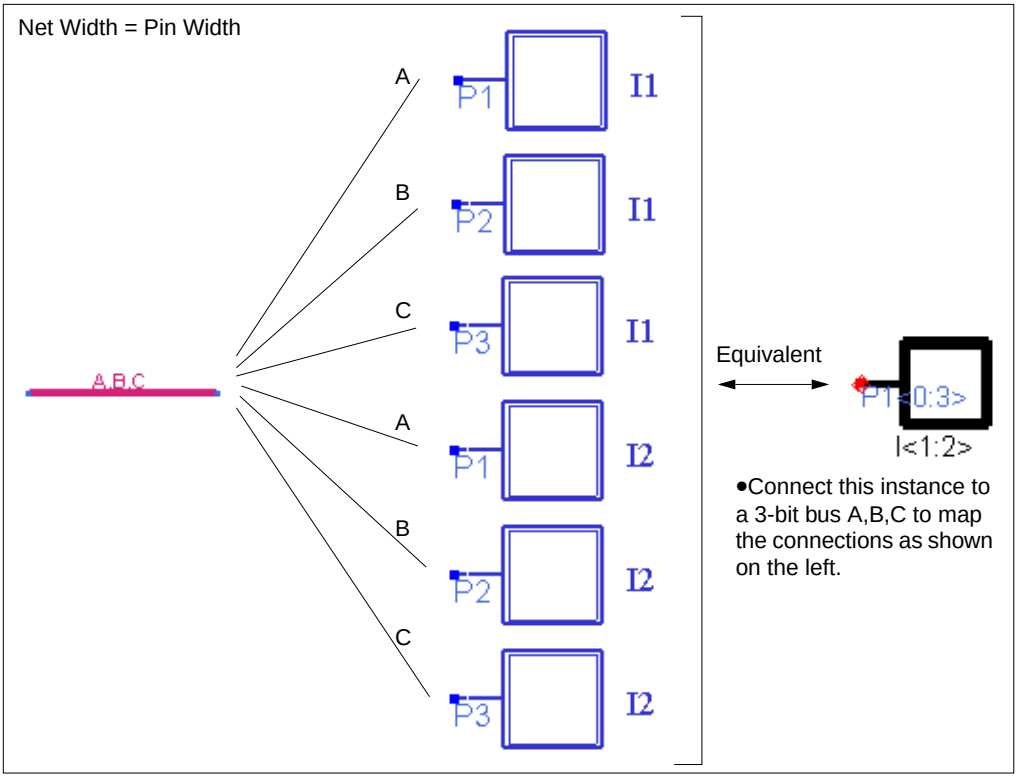
If the net width is 1, and does not satisfy the previous two conditions, then the bus (really a single wire) is repeated M times, where M is the connection width. Thus this is the case when the pin width is greater than 1 and the net width is 1. This case will issue a warning because although this is valid, we assume that a pin with width connected to a single wire (shorted) may not be what the user intended.



The following figure illustrates how the connections of a six-bit bus are mapped to which pin of which instance, where there are—in essence—two instances, each with three pins.



The next figure illustrates how the connections are made if you connect the aforementioned instance to a three-bit bus instead, which is allowed because the iterated instance has a three-bit pin.



Adding Ports to a Design

1. Click the port symbol on the toolbar (or choose *Insert* > *Port*).
2. Select the appropriate rotation by clicking the toolbar button (Rotate By -90) as needed.
3. Move the pointer into the drawing area, position the symbol as needed, and click to place it there.

Hint Do not use the *Move To Layer* command to move ports to a different layer; set the *Layer* parameter of the port to the desired layer.

Using Special Components

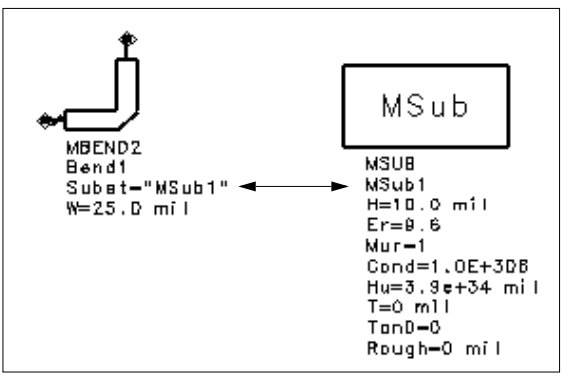
There are a number of commonly used components in ADS whose special features must be understood to be used successfully. Please review the following topics, and where applicable, the additional referenced topics:

- “Using Substrates” on page 3-50
- “Using Nonlinear Models” on page 3-51
- “Components that Allow File-Based Parameters” on page 3-52

Using Substrates

Substrates (such as microstrip, stripline, etc.) are specified by placing the required substrate component in your design and then setting the substrate parameter (*Subst*) of the associated circuit component(s) equal to the instance name of the substrate component.

In the following example, the substrate parameters of the MSub1 instance are associated with the Bend1 instance by assigning the instance name “MSub1” (of the MSUB substrate component) to the *Subst* parameter of the MBEND2 component.

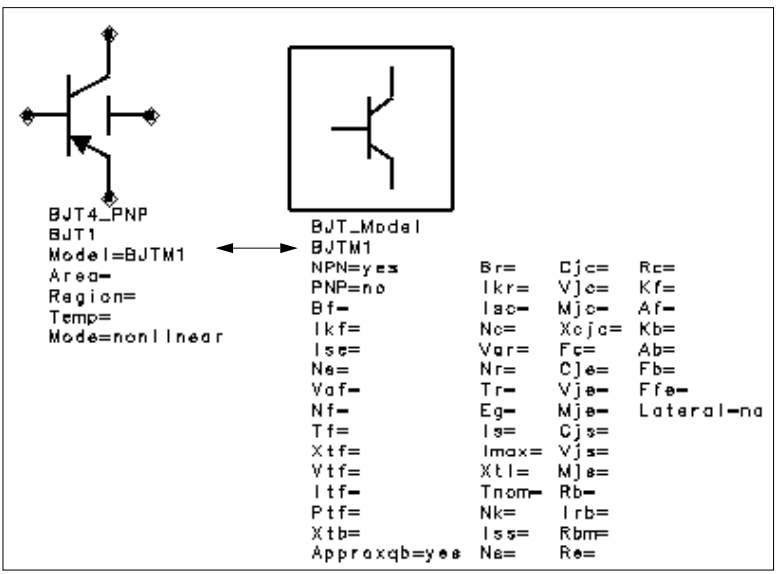


Note that the default value of the substrate parameter *Subst*, as well as the default substrate instance name (in this example, *MSub1*), can be edited.

Using Nonlinear Models

A nonlinear model can be associated with a nonlinear device instance by placing the required nonlinear model item in your design and then setting the parameter *Model* of the nonlinear device equal to the instance name of the nonlinear model item. This is especially useful in a hierarchical design that contains multiple subnetworks, each of which should reference the same model. In this case, place the _Model item in the top-level design, and place an instance of the device in each of the subnetworks, setting their Model parameter equal to the Instance Name of the _Model item.

In the following example, the model parameters of the BJTM1 instance are associated with the BJT1 instance by assigning the instance name “BJTM1” (of the BJT_Model item) to the parameter *Model* of the BJT4_PNP device.



Note The default value of the *Model* parameter—for any given device—is the same as the default Instance Name of the related model (in this example, *BJTM1*). The Instance Name of the model can be changed, like any other Instance Name. If you do, be sure to change the *Model* parameter (of instances referencing that model) accordingly.

Components that Allow File-Based Parameters

Several components, such as DataAccessComponent (DAC), Deembed, and the S*n*P components, enable you to set a parameter to reference a file-based set of values. For details on the file formats of the data file types, refer to Chapter 20, Data File Formats, in the *Circuit Simulation* manual.

To specify file-based parameters for these components:

1. Select the **File** parameter.
2. Select the appropriate Parameter Entry Mode: **Data filename** (DAC) or **Network parameter filename** (S*n*P, Deembed1, Deembed2).
3. Type the data filename, or *Browse* to select it. By default, only the files listed in the current */data* directory are displayed.

   Alternatively, click *Data files list* to select a data file. The files listed are the files found in the set of search paths assigned to the DATA_FILES variable (in *de_sim.cfg*). The data directory of the current project is usually the first path. For more information on setting variables refer to the section, *Variables in de.cfg, de_sim.cfg* in the *Customization* manual.
4. Optionally, click **Edit** to display the file (in your default text editor) for modification.
5. Optionally, click **Copy Template** to copy a data file of a specific format for use as a template. This template can be a supplied template or one you have created. By default, the program looks in the following location:

   $HPEESOF_DIR/circuit/templates (for Analog/RF designs)

   or

   $HPEESOF_DIR/hptolemy/templates (for Digitial Signal Processing designs)

Using the DataAccessComponent

Many component parameters, as well as variables, can be assigned a value from a data file by using the DataAccessComponent (DAC). The basics of accessing file-based parameters are as follows:

- Add a DAC component to your design
- Set the *File* parameter of the DAC equal to the data file of interest
- Set the parameter of the component of interest to reference the DAC (by its Instance Name)

For details on setting the DAC's parameters and some basic examples, refer to *DataAccessComponent* in Chapter 1, Introduction, in the *Circuit Components* manual.

Using Macros to Automate Tasks

You can record a macro (a series of AEL commands) by performing a sequence of commands using the mouse, and then play it back later to repeat the sequence.

Note Not all commands/actions are supported in macro recording; therefore, we do not recommend relying on this for recording complicated command sequences, but rather as a tool for learning how to use AEL.

To record a macro:

1. In the Main window choose **Options** > **Start Recording Macro** and a dialog box appears.
2. Enter a name for your macro—the file extension *.dem* will be added automatically—and click **OK**.
3. Perform the desired sequence of commands.
4. When you are finished with the desired sequence, choose **Options** > **Stop Recording Macro**. The file is saved to your current project directory.

To play back a macro:

1. Choose **Options** > **Playback Macro** and a dialog box appears displaying the macro files in the current project directory (that match the filter).
2. Select the desired macro and click **OK**. (To run a macro stored in another project directory, adjust the path as necessary, select the macro from the list, and click OK.) The selected macro is executed.

Viewing and Entering AEL Commands

The AEL commands that are issued in response to your activity in the Main window and the design windows are displayed in the Command Line dialog box. This command summary is updated continuously as you work. You can view this summary any time and you can issue previously executed commands from this list.

To view the command summary:

Choose **Options** > **Command Line** in the Main window and the dialog box appears. As you execute commands, the corresponding AEL functions are displayed.

To execute AEL commands from this command summary:

- Type the command(s) in the *Command* >> field and press **Return** or click **Apply** after each to execute. (Note: All commands entered in the *Command* >> field must be in AEL format.)

  or

- Select a previously typed command from the list and press **Return** or click **Apply** to execute.

  or

- Double click any command in the list and the command is executed.

Click *Cancel* to dismiss the dialog box.

Note For configuration details on using AEL, refer to the *AEL* manual. For layout artwork and usage, refer to the *Layout* manual.

Creating a Netlist

By default, every time you simulate a design, you generate a netlist file, *netlist.log*, in the current project directory, but it is possible to generate the netlist file without simulating the design.

To generate a netlist without simulating:

1. Open the desired Schematic.
2. Choose **Options** > **Command Line** in the Main window and the Command Line dialog box appears.

Hint Replace the number *1*, shown in parentheses in the next step, with the number of the window from which you want to generate a netlist. The number of the window is displayed in the title bar.

---

3. Click in the Command >> field and type **de_set_window(1);** and press **Return**. Notice that what you type is echoed in the list above.
4. Click in the Command >> field and type **de_netlist( );** and press **Return** or click **Apply**. After a short time, the *netlist.log* file is written to the project directory.

To use a different filename for any given project, you can insert the following line in the *de_sim.cfg* file in that project directory and specify a filename other than *netlist.log*.

```
NETLIST_FILE_NAME=netlist.log
```

Important If you edit your schematic after generating a netlist, you will need to generate the netlist again, since nodes may be renumbered as you edit.

Generating Reports

The *Reports* command enables you to generate a Bill of Materials (BOM) and a Parts List. Examples of a BOM and a Parts List are shown below.

Note By default, you are supplied with one format for the BOM and two for the Parts List, but these formats can be customized, and additional formats added, through AEL. For details, refer to the *AEL* manual.

To generate a Bill of Materials:

1. Choose **File** > **Reports** > **Bill of Materials**.
2. By default, the design name and a *.rpt.bom* extension appear as the filename for the generated file. Accept this name or supply another and click **OK**. The file is displayed in a window on the screen.

```
Bill of Materials for: tee_atten    Tue Mar 31 14:22:19 1998



Item         Qty    Description                    Designators
=========================================================


RES          3      Resistor                        R2 R3 R1
```

Figure 3-1. Bill of Materials Example

3. To save it to file with the default filename, click **OK**; to save it to file with a filename of your choosing, click **Print**. Supply a filename and click **OK**.

To generate a Parts List:

1. Choose **File** > **Reports** > **Parts List**.
2. Choose the desired Report Type.
   - HP EEsof (HP EEsof PL format)—component name, ID, coordinates, angle, and side
   - HP EEsof (netlist format)—component names and parameters

3. By default, a *.rpt.pl* (parts list) or *.rpt.net* (netlist) extension is automatically added to the filename. Accept this name or supply another and click **OK**. The file is displayed in a window on the screen. Figure 3-2 shows an example of the parts list format.

```
Parts list for: /networks/tee_atten    Tue Mar 31 11:28:45 1998



Component     Ref ID      X, Y,            ANG      SIDE
=====================================================


R             R1         1, 0.5             0     top
R             R2         3.5, 0.5           0     top
R             R3         2.75, -0.125     -90     top
```

Figure 3-2. Parts List Format Example

Figure 3-3 shows an example of the netlist format.

```
Net list  Tue Mar 31 11:49:33 1998

TOP LEVEL DESIGN:  tee_atten

Design: tee_atten

Port P1 1
  Num=1
Port P2 4
  Num=2
GROUND G1 0

R R1 1 2
  R=10 Ohm
R R2 2 4
  R=10 Ohm
R R3 2 0
```

Figure 3-3. Netlist Format Example

4. Click **OK** to save it to file with the default filename. Click **Print** to print to file or to the printer, based on your current Print Setup. If the current Print Setup is set to print to file, a dialog box appears prompting you for a filename. Supply a filename and click **OK**.

Chapter 4: Creating Hierarchical Designs

You can use any network as a subnetwork within another network to create a hierarchical design. There are two ways to create a subnetwork:

- Use the *Create Hierarchy* command and specify a portion of an existing design to be copied to its own design file for use as a subnetwork
- Create a new design consisting of a network you want to use as a subnetwork

---

Hint To view design hierarchy in the current project, choose *View > Design Hierarchies* from the Main window.

To access a design in one project for use as a subnetwork in a design in another project, create a hierarchical project (*File > Include/Remove*, in the Main window). For details, refer to "Creating a Hierarchical Project" on page 2-9.

Note The *Update Component Definitions* command (*Edit > Component*) enables you to explicitly update component definitions (that you have changed) throughout the current design. If you select the *Update Component Definitions in Subnetwork* option, the design hierarchy will be traversed (downward) and components will be updated throughout the hierarchy.)

Creating a Subnetwork from an Existing Design

The *Create Hierarchy* command copies the selected portion of your design to another file, saves that new file, deletes the selected items in the original file and replaces them with a default symbol representing the deleted items.

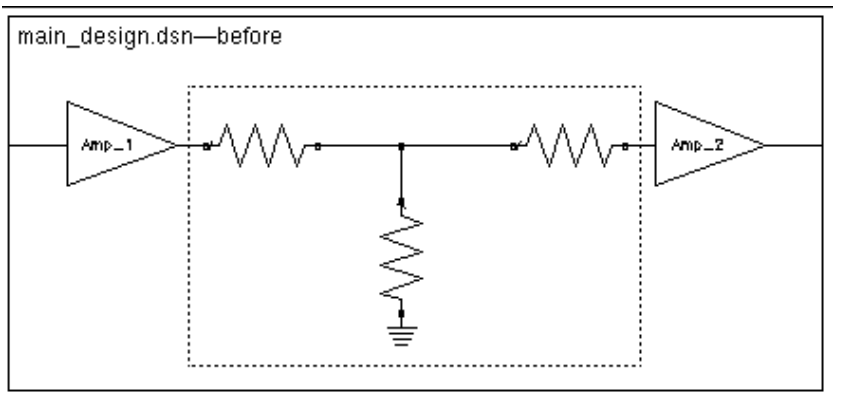
Hint You can create a custom symbol to use in place of the default symbol. For details on symbols, refer to Chapter 10, Working with Symbols.

The example used to illustrate this command is based on a simple, 3-resistor attenuator that is part of a larger design. In this example, the main design is named

main_design and the file created using the *Create Hierarchy* command (containing the 3-resistor attenuator) is named *my_atten*.

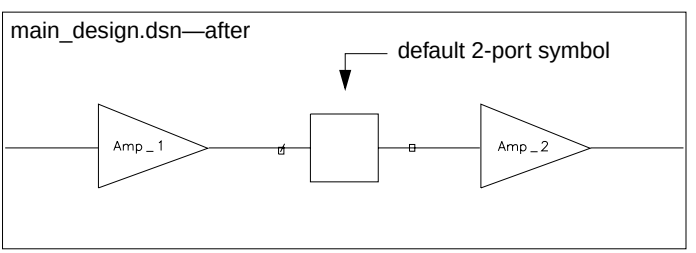
To create a subnetwork from an existing design:

1. Select the items you want to include in the subnetwork (represented here inside the box drawn with a dashed line).



2. Choose **Edit** > **Component** > **Create Hierarchy** and a dialog box appears.
3. Provide a name for the new file, in this example, **my_atten,** and click **OK**. (The name you supply is the filename for the subnetwork as well as part of the annotation for the symbol when you place it in a design—however, no annotation is displayed in this example.)

   The selected items disappear from your original design and are replaced by a default symbol (in this example, a 2-port symbol). Wires are redrawn to reconnect the remainder of the design to the symbol.
4. **Save** the file. The design used in this example appears as shown next.



If you want to use this network as a parametric subnetwork in a hierarchical design, you must open this newly created file (*my_atten.dsn*) and define the parameters that you want to be passed from the subnetwork to the network.

To define parameters for an existing subnetwork:

1. Open the file containing the subnetwork design.
2. Choose **File** > **Design Parameters** and a dialog box appears.
3. Supply a parameter name (not to exceed 8 characters), and select the appropriate characteristics for that parameter.
4. When you are through assigning characteristics to that parameter, click **Add** and the newly defined parameter is added to the Parameters list box.
5. Continue in this manner until you have assigned all the desired parameters for this network and click **OK**.

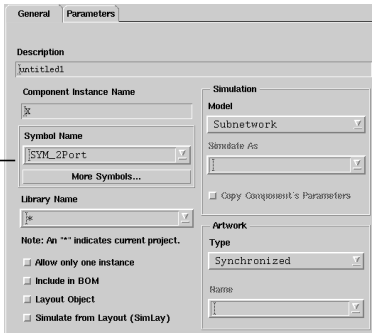
For a more detailed discussion on defining parameters, refer to the section titled, "Defining Parameters" on page 4-9.

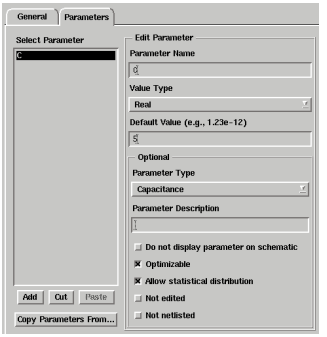
Creating a Parametric Subnetwork

Any network can serve as a *parametric* subnetwork. A parametric subnetwork is any network for which you define the control parameters that pass through to the network into which you place the subnetwork. Once you have defined the parameters, your subnetwork can serve as a template enabling you to assign parameter values each time you place it in a design. You can construct whole libraries of re-usable subnetworks in this manner.

The details of this process are presented using a simple design consisting of a capacitor and a resistor in series and two ports. This subnetwork is called *series_r_c* and is represented by the default 2-port symbol.

File > Design Parameters





Create the subnetwork

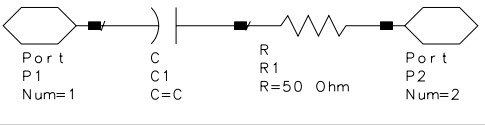
Define the design characteristics

Define the parameters

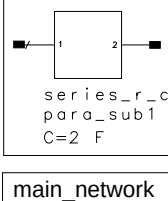
Accept the default symbol or assign a custom symbol

Place subnetwork within desired design

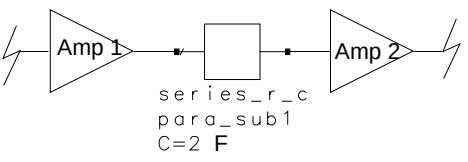
series_r_c



default 2-port symbol



main_network



When you place the subnetwork within another design, you have the opportunity to assign values to any parameters you defined.

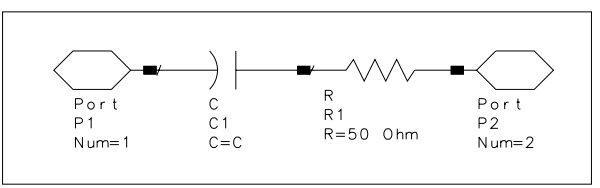
Creating the Subnetwork

You can create the subnetwork first, or define parameters and then create the subnetwork. In this example, we create the subnetwork first.

To create the subnetwork:

1. Choose **File** > **New Design** in the Schematic window and assign a name, in this example, **series_r_c**. This name becomes part of the annotation for the symbol representing the subnetwork when you place it within another design.
2. Select **Analog/RF Network** as the Type of Network and click **OK**.
3. Click the **Library** button and select a category, in this example, **Lumped Components.**
4. Select a component, in this example, **C (Capacitor)**.
5. Place the capacitor in the Schematic window. Note the component parameter C is set equal to the default value of 1.0 pF. Click the **End Command** button.

6. Click the capacitance value to invoke the on-screen editor and use the *Back Space* key to erase the *1.0 pF*.
7. Type **C** and press **Return**. The parameter now reads C=C. (This C parameter will be defined in the Design Parameters dialog box in a later step and serves as a variable here.) The parameter C (capacitance) of this component will now pass through to the network in which it is placed.
8. Place the next element, in this example, **R (Resistor)**, accepting the default values. Connect it to pin 2 of the capacitor symbol.
9. Add ports and click the **End Command** button. The subnetwork should resemble the illustration shown next.



Defining Design Characteristics

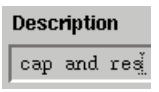
Design characteristics include things such as the name of the symbol used to represent the subnetwork and a library from which the subnetwork can be accessed. In some cases you may find the default design characteristics acceptable (default symbol and default current project as the library, for example). If this is the case, proceed to the next section, "Defining Parameters" on page 4-9.

To alter the default characteristics:

1. Choose **File** > **Design Parameters** and a dialog box appears. In the *General* tab, the current design name is reflected in the Name field at the top of the dialog box.

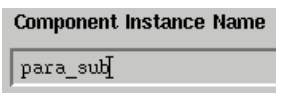
2. The Description field also displays the current design name by default. You can change this to a more helpful label defining the purpose of the network design. The label you provide here will be displayed, together with the design name, as a component to place from the designated library (Library Name field).

   Optionally, add a description, in this example, **cap and res**.



3. The Component Instance Name default is X. The text in this field is used as a prefix in building a unique name (ID) for every item. This prefix becomes part of the annotation displayed with the symbol representing the parametric subnetwork when you place it in a design.

   Optionally, assign a unique name, in this example, **para_sub**.



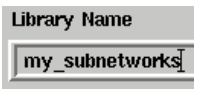
4. Notice that the Symbol Name field reads *SYM_2Port*. This is the default symbol for a 2-port design. In this example, we are using the default symbol, but you can select one of the other symbols from the drop-down list, or click *More Symbols* to select one by clicking an icon from the appropriate category. For details refer to Chapter 10, Working with Symbols.

5. In the Library Name field, specify the name of the library in which you want the subnetwork stored. This library name is the name that appears in the

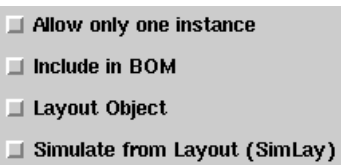
Component Library enabling you to select the subnetwork and place it in a design. There are several ways to specify a library name:

- Accept the default Library Name—an asterisk (*). This means the design will be available in the Component Library through the current project
- Type any name to create a new library
- Enter the name of any of the supplied libraries
- Select the name of any library you created previously

Create a new library name, in this example, **my_subnetworks**.



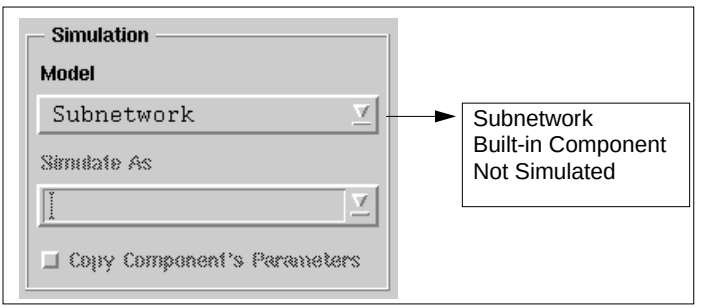
6. Turn on or off the following options, as required by your design.



- *Allow Only One Instance*—Specifies whether or not the subnetwork can be placed more than once in a design. The default is off, meaning the subnetwork can appear more than once in a design. Change to on if you want to restrict placement of the subnetwork to once per design (seldom done).
- *Include in BOM*—Specifies whether or not the details of this design are included when a BOM is generated. Without this option, only the top level design information is included in the BOM.
- *Layout Object*—Analog/RF designs only. Defines the design as an object. Layout objects have no parameters and are used to assign artwork to new elements or designs with no default artwork. For details, refer to the *Layout* manual.
- *Simulate from Layout (SimLay)*—Analog/RF designs only. The netlist required for simulation is generated from either the Schematic or the Layout. Select this option to generate the netlist from the Layout.

The *SimLay* portion of this label will appear in the Status panel of the Schematic and Layout windows if you select this option; the default is to simulate from the Schematic, and the Status panel reflects *SimSchem*.

7. Layout only—select the appropriate Simulation method:

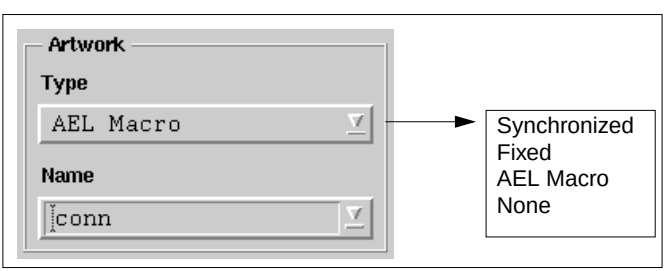


- Select *Subnetwork*, to use a schematic symbol you have created
- Select *Built-in Component*, to use a supplied simulator item.

  Simulate As—Select a component name from the drop-down list, or type the name of any component

  Copy Component's Parameters—Select this option to copy the parameters of the selected component to use as a starting point
- Select *Not Simulated*, to create layout- or schematic-only non-simulated items

8. Analog/RF layout designs only—select the appropriate artwork *Type*. For details refer to Chapter 7, Artwork in the *Layout* manual.



9. Click **Save AEL file** to save the information defined so far. (This is done automatically when you save the design file.)

Defining Parameters

When you define parameters for a network, the network serves as a template, enabling you to assign new parameter values each time you use the network. This is useful when a portion of the network is used several times in a design with certain element value differences, or in constructing libraries of reusable networks.

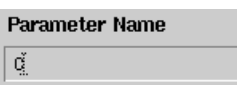
Parameters of the network are generally referenced as variables by the elements of the network. You can define the parameters before or after creating these variable references.

Each parameter has characteristics that determine how it is handled when the network is reused. These include the name and label displayed in the Item Parameters dialog box, the unit of the parameter, the type of value assigned to the parameter, the default value, and certain control attributes.

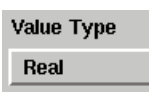
To define the parameters that should be passed to the upper-level network:

1. Click the **Parameters** tab.
2. Optionally (Analog/RF designs only), you can click **Copy Parameters From** as a shortcut for defining parameters, if one of the supplied components has a number of parameters in common. You can then cut any unwanted parameters, as well as modify the characteristics of the remaining parameters.
3. The Parameter Name field contains the parameter name that will be referenced in the subnetwork schematic. Parameter names become part of the annotation of the symbol representing the parametric subnetwork when you place it in a design.

   Supply a Parameter Name (not to exceed 8 characters), in this example, **C**.



4. Select a Value Type from the drop-down list, in this example, **Real.**

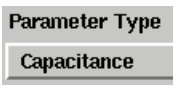


5. Specify a Default Value, in this example, 5. This value can be changed at the time you place the subnetwork.

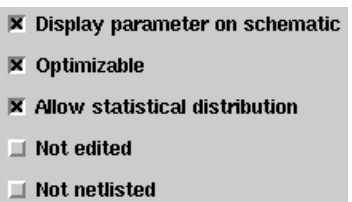
Hint If you do not specify a scale factor along with the default value, the current setting in the Preferences dialog box is used (*Options* > *Preferences* > *Unit/Scale*), based on the Parameter Type.

---

6. Optionally, select a Parameter Type for this parameter, in this example, **Capacitance**. This can be a dimensional unit or a string for the parameter, if one is needed. (*String* is used for assigning SMT artwork in layout.)



7. Optionally, supply a Parameter Description.
8. Select any of all of the following options, as they apply. Note that some options are desensitized depending on the current Value Type.



- *Display parameter on schematic*—Select this option to display, on the schematic, the parameter being defined.
- *Optimizable*—Select this option to allow this parameter to be optimized.
- *Allow Statistical Distribution*—Select this option to allow post-production tuning for this parameter during yield analysis.
- *Not edited*—Select this option to prevent this parameter from appearing in the Component Parameters dialog box for editing and always use the default value assigned here instead.
- *Not netlisted*—Select this option to prevent a parameter from being considered in simulation, but still be recognized for artwork generation

9. Click **Add** to add the newly defined parameter to the Select Parameter list box. The parameter C is added to the Parameters list box.

- Use *Add* to add a new parameter to the list
- Use *Cut* to delete parameters
- Use *Cut* and *Paste* buttons to rearrange the order of the parameters.

Hint You can position these user-defined parameters to display somewhere other than the default location of supplied parameters. For details refer to "Positioning Parameters for Your Symbol" on page 10-14.

---

10. Optionally (Analog/RF designs only), you can click **Add Multiplicity Factor (_M)** to enable simulation of this subnetwork as though it were *x* number of these subnetworks—connected in parallel—where *x* is the value you assign to the parameter _M.
11. Click **OK** to dismiss the dialog box and choose **File** > **Save Design**.

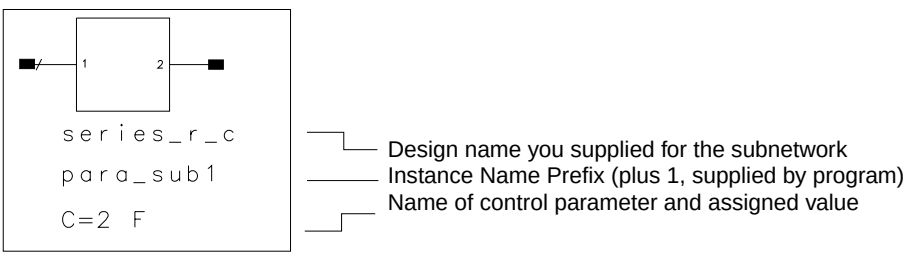
Placing the Subnetwork in a Design

Now you can place the subnetwork you just created into another network. The parameter you assigned (capacitance in this example) will pass through to the network.

To place a subnetwork:

1. Open the file containing the network in which you want to place the subnetwork (in this example, *main_network*—not shown).
2. Click the **Library** button (or choose *Insert* > *Component* > *Component Library*).
3. In the Component Library window, locate and select the library containing the design, in this example, **my_subnetworks**.
4. Select the subnetwork, in this example, **series_r_c**, and place it in the drawing area. Click **End Command**.

   Notice that C = 5 F. This 5 is the value from the Default Value field (for the parameter C) in the Design Definition dialog box. Because we did not specify a scale factor, the default F (Farads), from the Preferences dialog box is used (*Options* > *Preference* > *Unit/Scale*).
5. Using the on-screen editor, change the default capacitance value to the desired value for the current design, in this example, **2**. Press **Return**.



Modifying a Design that Serves as a Subnetwork

If you make changes to a subnetwork that affect the component definition (any changes in the Design Parameters dialog box—information that is saved in the *.ael* file), that serves as a subnetwork in a higher-level design, you need to explicitly update the higher-level design to recognize those changes.

To update a higher-level design that contains a modified subnetwork:

1. From the higher-level design, choose **Edit** > **Component** > **Update Component Definitions**.
2. To traverse the hierarchy (downward) in search of any subnetwork designs whose component definitions have been changed, select the option **Update Component Definitions in Subnetwork** and click **OK**. Changes to any subnetwork designs are now reflected in the instance(s) in the main design.

Viewing the Network Represented by a Symbol

Whenever your design contains a symbol that represents a network, you can view the actual network being represented by the symbol by using the *Push Into Hierarchy* command.

To push into and then pop out of an item:

1. Select the item.
2. Choose **View** > **Push Into Hierarchy**. The network represented by the symbol is displayed.

   The *Pop Out of Hierarchy* command is the reverse of pushing, and only works if a design has been pushed into.
3. When you are through viewing the network, choose **Pop Out of Hierarchy** and you are brought back to the item (or design containing the item).

Chapter 5: Viewing Designs

The View menu commands enable you to change the current view of the drawing area to aid you in working with the image in the window. Additionally, there are a number of commands on the Options menu that display various aspects of design information.

Zooming In and Out

The *Zoom* commands enable you to enlarge or shrink the area being viewed. *Zoom Window* enables you to specify your own view window for zooming, if the other zoom commands do not meet your needs.

To zoom in on a specified point in the window:

Choose **Zoom In Point**.

Click to specify a point and the current view is magnified by a factor of two, moving the point you specify to the center of the window.

To zoom out from a specified point in the window:

Choose **Zoom Out Point**.

Click to specify a point and the current view is decreased by a factor of two, moving the point you specify to the center of the window.

To specify a particular factor by which to zoom:

Choose **Zoom By Factor** and choose the appropriate command.

- *Zoom In x2* (zooms in by a factor of 2)
- *Zoom Out x2* (zooms out by a factor of 2)
- *Zoom in x5* (zooms in by a factor of 5)

- *Zoom Out x5* (zooms out by a factor of 5)
- *Zoom In by . . .* (and specify the desired factor)
- *Zoom Out by . . .* (and specify the desired factor)

To specify a particular portion of the view for zooming:

1. Choose **Zoom Area**. You are prompted to enter the first corner.
2. Move the cursor to the point representing the upper-left corner of the desired view window and click left. You are prompted to enter the second corner. As you move the mouse, a flexible box, representing the view window, moves with it.
3. Move the cursor to the point representing the lower-right corner of the desired view window and click left. The portion of your drawing enclosed by the view window is magnified. (The magnification amount is determined by the size of the view window you specified.)

To zoom to a selected object(s):

Select the object(s) and choose **View** > **Zoom** > **Zoom To Selected**.

Repositioning a Design to Fit the Window

To rescale and reposition your design so that it all fits in the window:

Choose **View All**. Your design is scaled as needed and repositioned to fit it all, plus a five-percent border, in the viewing area.

Moving the Center Point of a Window

The *Pan* command moves a point you specify, to the center of your window. Alternatively, you can use the scroll bars to move a different part of the window to the center.

To change the center point:

1. Choose **Pan View** from the View menu or the pop-up menu. You are prompted to enter the new window center.
2. Click once and the selected point becomes the new center point of the window. Your design is redrawn accordingly.

Redrawing the View in a Window

The *Redraw View* command refreshes the image in your window without changing anything. Choose *Redraw View* from the View menu anytime you make changes and see that an image is not completely drawn.

Saving and Restoring Views

It is possible to save multiple views of your design at various zoom settings.

- *Save View* enables you to save the current zoomed or panned view with a name
- *Restore View* enables you to retrieve a saved view
- *Delete View* enables you to delete a saved view
- *Restore Last View* enables you to restore the view that was in the window the last time you issued a *Pan* or *Zoom* command

To save the current view:

1. Choose **View** > **Save View**. A dialog box appears.
2. Supply a name for the view and click **OK**.

To restore a view that has been saved:

1. Choose **View** > **Restore View**. A dialog box appears.
2. Supply the name of the view you want and click **OK**.

To delete a view that has been saved:

1. Choose **View** > **Delete View**. A dialog box appears.
2. Supply the name of the view you want and click **OK**.

To restore the view that was in the window the last time you issued a *Pan* or *Zoom* command:

Choose **View** > **Restore Last View**.

Viewing Design Information

You can display a variety of design information using commands found on the Options menu:

- *Hierarchy* displays a listing of the hierarchical information of the current design in the Schematic and Layout windows
- *Info* displays a detailed listing of design information including current units, preference and layer file associated with design, instances by name and ID, mask layer information, and a summary
- *Identify* lists detailed data for selected instances, shapes, or text
- *Check Representation* provides information on unconnected pins, port vs. pin mismatch, and nodal mismatch

Viewing Detailed Design Information

The details on closed shapes include area, perimeter, and layer; for text, it includes the string and font attributes; for polylines it includes length. For components, the Name, ID, X, Y location, fixed/free status and equivalent element are displayed. A short summary is included that shows the number of items and shapes, the layers used, and the total selected area, length and perimeter.

To display detailed information for selected items:

1. Choose **Options** > **Info** and the Information dialog box appears.
2. To view details on items not selected initially, select and click **Refresh**. The information is updated to display details of the newly selected item(s).
3. Optionally, click **Print** to send the information to your default printer.
4. Click **OK** to dismiss the Information dialog box.

Viewing Detailed Instance Information

To display detailed information for selected instances:

1. Choose **Options** > **Identify** and a dialog box appears.
2. To print the information, click **Print** and it is sent to the default printer.
3. Click **OK** to dismiss the dialog box.

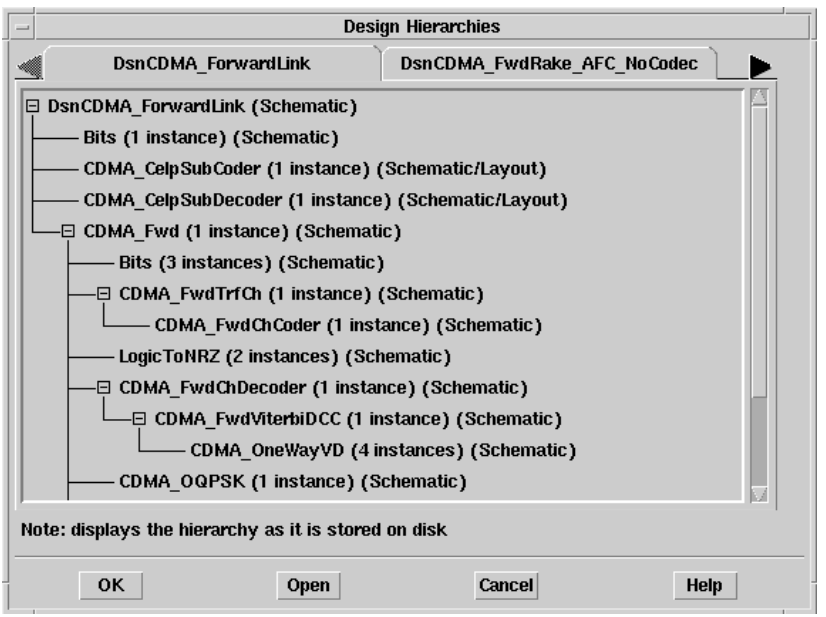
Viewing Hierarchical Design Information

There are two different ways to view hierarchical design information:

- At the *design* level, for the current project (*View* > *Design Hierarchies*, in the Main window)
- At the *component* level, for the current design (*Options* > *Hierarchy*, in the Schematic and Layout windows)

To view design hierarchies for the current project:

1. In the Main window, choose **View** > **Design Hierarchies**.



   If the project contains more top-level designs than can be displayed at one time, the arrows on either side of the tabs enable you to cycle through the remaining top-level designs to select the one you want.

2. Double-click any design to open it.

To view the component hierarchy information for the current design:

1. Choose **Options** > **Hierarchy** and the Hierarchy dialog box appears.

   Hierarchical levels are indicated by the level of indentation in the list. Top level items are not indented; each nested level is indented with two spaces.

2. Click **Print** to send the information to your default printer.
3. Click **OK** to dismiss the Hierarchy dialog box.

Viewing Connectivity Information

The *Check Representation* command provides access to information about any of the following characteristics of your design:

- *Open Connections*—Displays the total number of unconnected pins and wires. For each item with an unconnected pin, it lists the component name and ID, the pin number and the coordinates of the unconnected pin. For each wire with an open end, it displays the coordinates of the wire segment. The affected items are highlighted in the design window.
- *Nodal mismatches (layout vs schematic)*—Reports items that are connected differently in one representation than they are in the other. The report lists the name of the item, the pin that is connected differently and what the pin is connected to. The affected items are highlighted in the design window.
- *Pin mismatches (instance vs symbol)*—Schematic only. Checks every instance in the design (against the source design it represents) and reports any discrepancy between the number of pins on the symbol in the current design and the number of ports on the schematic in the source design.
- *Port/Pin mismatch (schematic vs symbol)*—Schematic only. Compares the number of pins on a given symbol to the number of ports on the schematic the symbol represents and reports any discrepancy.
- *Wires in layout*—Layout only. Reports all items connected to pins that are interconnected with a wire, or a zero-width trace.
- *Overlaid components*—Reports the IDs of any overlapping items where the items contain the same number of pins and pin 1 of each item is placed in the same location.
- *Overlap wires* (*Overlap wire/traces* in layout)—Highlights any wires (or traces) that overlap, that are not part of the same node.

- *Bus connectivity*—Reports failed connections listing the bus or bundle, its width, and the Instance Name of the component to which you have attempted to connect the bus or bundle.

To view this information:

1. Choose **Options** > **Check Representation** and a dialog box appears.
2. Select the desired information category (or categories) and click **OK**. A dialog box appears displaying the requested information.
3. Click **Print** to print the report, if desired.
4. Click **OK** to dismiss the report dialog box.

Chapter 6: Editing Designs

While editing, keep in mind that most edit commands allow you to select the item(s) before you select the edit command, or vice versa. Take note also that many editing commands are *repeating* commands, that is, once the command is chosen, it remains active until another command is chosen or until it is explicitly canceled (*Edit > End Command*).

You will find it easier to edit your designs if you understand some of the features provided to assist you in selecting the item(s) you want to edit. While there are several features available to assist you in selecting objects for editing, there are three distinct features you should know about:

- Using Selection Filters
- Changing the Pick Box Size
- Setting the Selection Status of Items Layer-by-Layer

These and other design entry and display preferences are described in Chapter 9, Setting Design Environment Preferences.

Using the Undo Command

Selecting *Undo* undoes the last editing command. A *stack* of edit commands is created enabling you to choose *Undo* repeatedly to return to an earlier state of your design. A stack is maintained for each window, thus the *Undo* command works independently from window to window. You can specify the number of commands you want the stack to hold through *Options > Preferences > Entry/Edit, Undo edit count*.

Hint You can also press <*Ctrl*> *u* to issue the Undo command.

Deleting Items

To delete selected items, click the *Delete* button on the toolbar, or press the Delete key on the keyboard, or choose *Delete* from the Edit menu. Deleted items can be restored using the *Undo* command.

Editing Component Parameters

There are several methods for changing component parameters. The simplest methods involve editing parameters for individual components:

- Using the on-screen editor
- Using the component parameter dialog box

Additional methods that accommodate editing parameters in specific situations are:

- Searching for a particular parameter—a parameter whose value is stated as a reference to another component (*Search / Replace Reference*)—and then replacing that parameter throughout the design
- Changing the value of a particular parameter common to components throughout the design (*Group Edit Parameter Value*)

Note The *at* symbol (@) must be used to suppress quotes when specifying a variable, for example, *@freq1*, where *freq1* is a variable declared in a VAR item.

Editing Component Parameters On-screen

You can use on-screen parameter editing to change parameter values. In addition, you can change the Value Type from nominal to variable, and vice-versa. If you need to change a parameter's Value Type to anything other than nominal or variable, refer to the next section, "Editing Component Parameters Through the Dialog Box" on page 6-3.

Hint When you click the component name, you initiate the *Swap Components* command.

To edit one or more parameters for a component using the on-screen method:

1. Position the pointer over the parameter you want to change and click. The editable portion of the parameter takes on the current *Highlight* color (*Options > Preferences > Display*). You will also see that a vertical bar (|), representing a text insertion cursor, appears in the parameter line.

2. Use the mouse, arrow keys, and backspace key as necessary, to change the parameters.
3. To end the parameter editing command, move the pointer away from the component and click once. (If the parameter is the last one in a list of parameters or is the only parameter for this component, pressing Return also ends the command.)

Hint When editing several parameters for one component, you can click each individual parameter you want to edit, or you can press Return as many times as needed to get to the next parameter you want to edit. Pressing Return for the last parameter in the list ends the parameter editing command.

Editing Component Parameters Through the Dialog Box

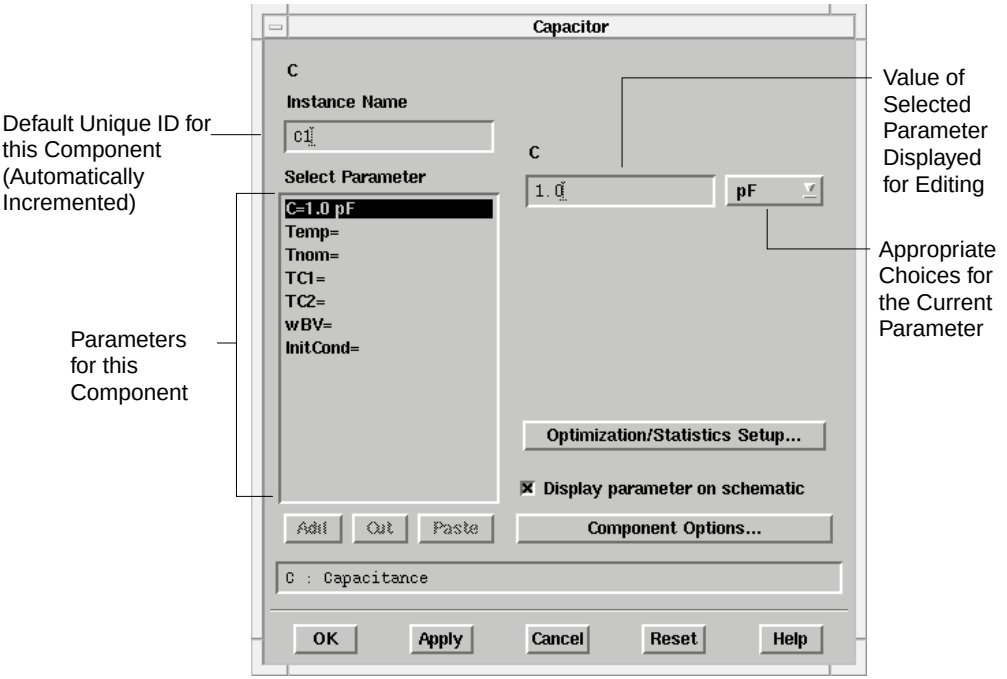
Despite variations for some components, certain basic guidelines apply to completing the component parameters dialog box for most components. Note the following features of this dialog box:

- The input fields change based on the individual parameter selected.
- A default ID for a given instance of a given component appears in the *Instance Name* field. This ID is unique for every component in the design. You can use the default name or supply any name of your choosing.
- Some parameters offer the ability to use data from a referenced data file. For details refer to the section, "Components that Allow File-Based Parameters" on page 3-52.
- You can turn the display of individual parameters on or off. You can also use *Component Options* to set or clear the display of all parameters for the component at once. For details, refer to the section, "Changing the Visibility of Component Parameters on a Schematic" on page 6-5.
- For details on Optimization/Statistics Setup, refer to the section *Specifying Component Parameters for Optimization* in Chapter 2, Performing Nominal Optimization, in the *Tuning, Optimization and Statistical Design* manual.

To edit one or more parameters for a component through the dialog box:

1. Choose one of the following methods for displaying the dialog box:
   - Choose *Edit > Component > Edit Component Parameters* and click the component symbol
   - Click the *Edit Component Parameters* button on the toolbar and click the component symbol

   - Double-click the component symbol



Hint Use the following shortcut to edit parameters for most or all components: select everything in the drawing area (*Select > Select All*) and choose *Edit > Component > Edit Component Parameters*. Click *Apply* to accept parameter changes for one component, and another component is displayed for editing.

---

2. Select the parameter you want to change from the Select Parameter list box.

3. Type the new value in the parameter value editing field.
4. Press **Return**. The Parameters list box is updated to reflect the new value and the value of the next parameter is displayed for editing.
5. When you are through editing parameters, click **OK** to dismiss the dialog box.

Changing the Visibility of Component Parameters on a Schematic

You can change the visibility status of all parameters of a given component through the Component Options dialog box (accessed through the component parameters dialog box).

- Set All—Displays all parameters for this component on the schematic. Use this option to display all, or almost all, parameters for this component. To display most—but not all—parameters, select *Set All* and then go back and turn off the display of individual parameters as desired.
- Clear All—Clears the display of all parameters for this component from the schematic. Use this option to turn off the display of all, or almost all, parameters for this component. To display a small subset of parameters, select *Clear All* and then go back and turn on the display of individual parameters as desired.

Referencing VAR Data Items and Model Items in Hierarchical Designs

The *Scope* option applies to the VAR (Variables and Equations) data item and most model items (such as R_Model, BJT_Model, BSIM3_Model). Exception: it does not apply to multilayer models. Scope indicates the levels, from a hierarchical standpoint, that recognize the expressions defined in the VAR data item or model item.

- Nested—VAR or model item expressions are recognized within the design containing the VAR or model item, as well as within any subnetworks (designs at lower levels) referenced by the design containing the VAR or model item.
- Global—VAR or model item expressions are recognized throughout the entire design, no matter what level in the design hierarchy the VAR or model item is placed.

Editing Common Parameters for a Group of Items

The *Group Edit Parameter Value* command enables you to select a group of components with one or more common parameters, select a parameter that applies to them all, and change that parameter's value for all selected components.

To edit every occurrence of a parameter for selected components:

1. Select all components containing the parameter you want to edit.
2. Choose **Edit** > **Component** > **Group Edit Parameter Value** and a dialog box appears.
3. Click **Name Options** and a dialog box appears listing all parameters in your design by name.
4. Select the desired parameter from this list and click **OK**. The selected parameter appears in the Parameter Name field.
5. If the value type of the selected parameter is numeric, type the new value in the Parameter Value field; otherwise, click **Value Options** and a dialog box appears. The contents of the dialog box vary depending on the value type of the chosen parameter.

   Note You can only change the nominal value of the parameter, not the Value Type.

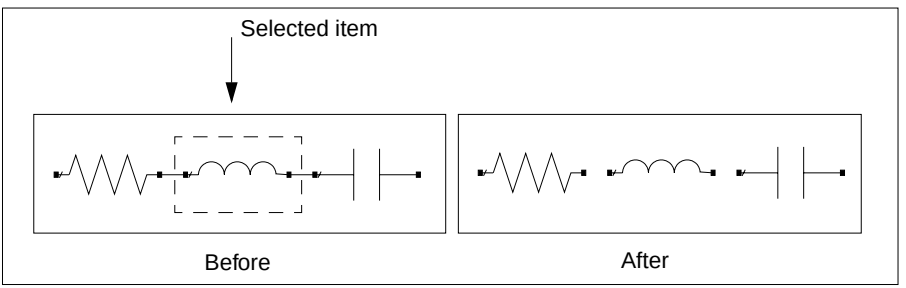
   ---

6. Select the desired parameter value and click **OK**.
7. Click **Apply** in the Group Edit Parameter Value dialog box (or click *OK* if you are through with this dialog box) and the design is updated accordingly.

Breaking Wire Connections Between Components

To break connections:

1. Select the component(s) whose connections you want to break.
2. Choose **Edit** > **Component** > **Break Connections**. The interconnections of selected components are deleted.



Swapping Components

The *Swap Components* command enables you to select any number of components, with any number of ports, and replace them all with another component.

To swap components:

1. Select all components you want to replace.

Hint Use the *Select* > *Select By Name* command to quickly select every component of a particular type.

---

2. Choose **Edit** > **Component** > **Swap Components** and a dialog box appears.
3. Click **Select** and the Component Library window appears.
4. Select the appropriate library and desired component. The component name appears in the New Component Name field of the dialog box.
5. Click **Edit Parameters** and the component parameters dialog box appears.
6. Change any parameters as desired and click **OK**.

7. To keep the IDs of the original components, select the option *Keep the original component ID(s)*; to let the program replace the IDs with new ones, deselect this option. Click **OK**. All selected components in the design window are replaced by the new component with the specified parameters.

Searching and Replacing References

The *Search / Replace Reference* command enables you to replace every occurrence of a parameter value where that parameter value is a *reference*, such as a reference to a data item. For example, if your design contains two MSUB data items, MSUB1 and MSUB2, you can replace every reference to MSUB1 with a reference to MSUB2 instead. You can replace references to data items or variables.

To search and replace references to data items or variables in your design:

1. Choose **Edit** > **Component** > **Search/Replace Reference** and a dialog box appears.
2. Select the desired Reference Type, **Component** or **Variable**.
3. Under the heading *Search For*, click **Select** and a dialog box appears. The listing varies according to the selected reference type.

Hint To highlight these references prior to replacing any, use the *Search Only* option and click *Apply*.

---

4. Select the Instance Name or Variable name representing the item you want to search for and click **OK**. The selected reference appears in the *Search For* field.
5. Under the heading *Replace With*, click **Select**.
6. Select the Instance Name or Variable name representing the desired replacement and click **OK**. The Replace With field is updated.
7. Click **Apply** (or *OK* if you are through with the Search/Replace dialog box). All occurrences of the selected reference are updated in your design.

Moving Component Text

You can reposition the component text, collectively, of any component with the *Move Component Text* command. In addition, you can change the layer assignment of individual pieces (Name, ID, Parameters) with the *Change Component Text Layer* command.

To move component text:

1. Choose **Edit** > **Move** > **Move Component Text.** You are prompted to enter a reference location.
2. Click the component whose component text you want to move. You are prompted to enter an offset location.
3. As you move the pointer, a ghost image representing the component text moves with it. Click again to reposition the component text in the new location.

Hint The function key F5 initiates the Move Component Text command. Press F5, click the component symbol, move the pointer and a ghost image of the component text moves with it. Position the image in the desired location and click again to place it there.

To change the component text layer:

1. Choose **Insert** > **Entry Layer** and select the desired destination layer from the list.
2. Choose **Edit** > **Component** > **Change Component Text Layer**.
3. Click any individual piece (Name, ID, Parameters) of component text. Its layer is changed to the current entry layer and the component text immediately takes on the characteristics of the current entry layer.

Note To change the layer on which component text is placed in advance of placing components, use *Options* > *Preferences* > *Component Text/Wire Label* to specify the desired layer for each type of component text. For details, refer to the section, "Setting Component Text/Wire Label Options (in Advance)" on page 9-15.

Changing Component Text Attributes

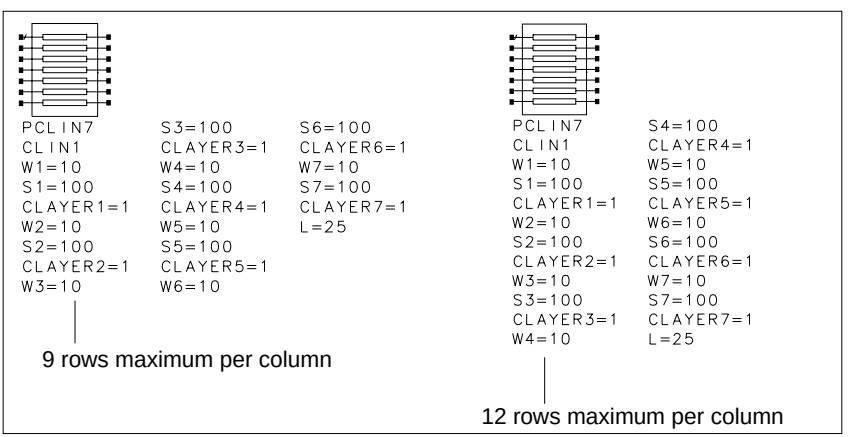
You can change the attributes of existing component text through the Edit menu. This command only affects existing component text. To establish attributes for all subsequent component text, specify the desired settings through *Options* > *Preferences* > *Component Text/Wire Label*. For details on changing text you have added to your design, refer to "Editing Existing Text and Text Attributes" on page 6-34.

To change component text attributes:

1. Select the components whose component text you want to edit.
2. Choose **Edit** > **Component** > **Component Text Attributes**.
3. Make any desired changes to the text attributes.
   - Font Type—All True Type fonts (Schematic only) installed on your system are available. Select the desired font from the drop-down list.

Note On UNIX, if you want to add additional True Type 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.
   - Parameter Rows—The maximum number of rows of parameters before another column is created.



4. Click **OK** and the component text is immediately updated to reflect the changes.

Editing Symbol Pins

The *Symbol Pin* command enables you to change the name, number, and orientation angle of existing pins.

To edit characteristics of existing pins:

1. Select a pin for editing and click **Edit** > **Symbol Pin**. A dialog box appears.
2. Change any of the characteristics as desired and click **Apply** (or click *OK* to accept the changes and dismiss the dialog box).

For details on pin characteristics, refer to, "Adding Pins to Your Symbol" on page 10-9.

Selecting and Deselecting Items

While you can always use the mouse to select and deselect items, several commands found on the Select menu can assist you in selecting and deselecting items more quickly.

Note Only objects on *selectable* layers can be selected for editing. If the select status (*Sel*) of a given layer is disabled (through the Layer Editor dialog box), the select commands have no effect on items on that layer.

Selecting/Deselecting All Items in the Drawing Area

To select all items in the drawing area:

Choose **Select** > **Select All**. Boxes are drawn around all items (that match the filter selection) showing that they are currently selected.

To deselect all selected items that match the filter selection:

Choose **Select** > **Deselect All**

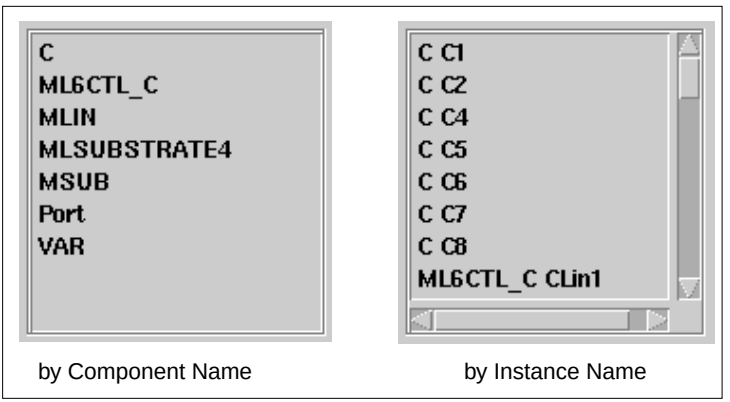
or click anywhere inside the window, away from the selected objects.

Selecting/Deselecting Items by Name

The commands *Select By Name* and *Deselect By Name* ignore the selection filters.

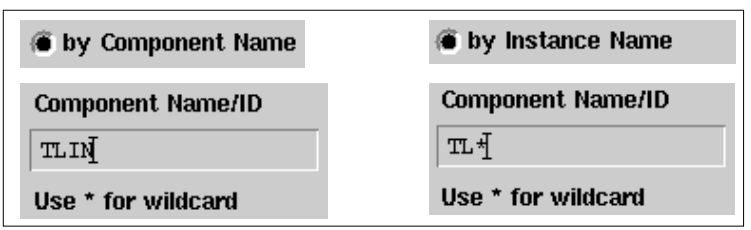
To select items by name:

1. Choose **Select** > **Select By Name**. A dialog box appears. By default, a list of each type of item in the design is displayed *by Component Name*. You can also view a list of individual items by selecting *by Instance Name*. The following illustration shows the listings of a simple network.



2. Select the desired list type and click the item(s) from the list that you want to select. Click **Apply**. The specified items are selected in the design window.

   If you have a very long list of components, you can use the wildcard field to filter the list for items of the same type. For example, to list all transmission line elements in the design, type TLIN (for a listing by *Component Name*) or TL* (for a listing by *Instance Name*), and click *Apply*.



To deselect items by name:

1. Choose **Select** > **Deselect By Name** and a dialog box appears. By default, a list of each type of item selected in the design is displayed *by Component Name*. You can also view a list of individual selected items by selecting *by Instance Name*.
2. Select the desired list type and click the item(s) from the list that you want to deselect. Click **Apply**. The specified items are deselected in the design window.

Hint Alternatively, you can use the wildcard field to filter the list for selected items of the same type. For example, to list all selected transmission lines in the design, type TLIN (for a listing by *Component Name*) or TL* (for a listing by *Component ID*), and click *Apply*.

Selecting/Deselecting With a Selection Window

You can include several objects at once for selection/deselection by enclosing them in a selection window.

To select items using a selection window:

1. Position the pointer at one corner of a window that will enclose the desired items, and press the left mouse button.
2. As you move the mouse, keeping the button depressed, a ghost image of the selection window is drawn. Release the button to specify the opposite corner of the window. All items totally enclosed within the selection window, that match the filter selection, are now selected and are identified by taking on the *Select* color chosen under *Option* > *Preferences*.

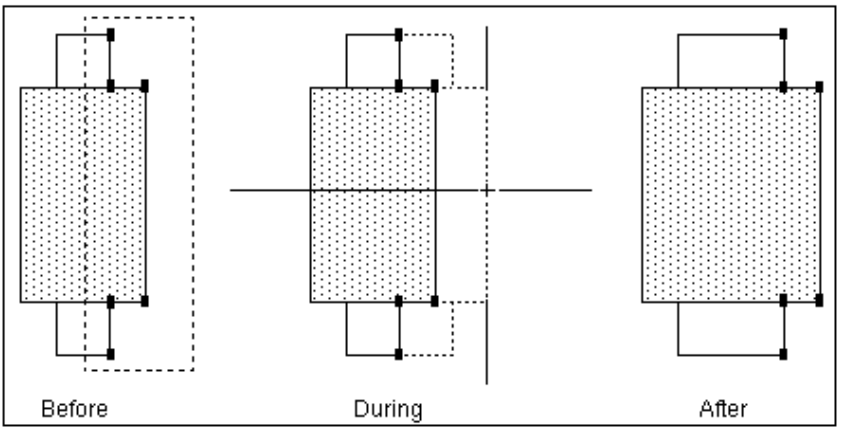
To deselect items using a selection window:

1. Choose **Select** > **Deselect Area**.
2. Draw a selection window enclosing the items you want to deselect. All items in the selection window (that match the filter selection) are deselected.

Using the Vertices Filter

By default, the *Vertices* filter is turned on. This means that when you use a selection window to encompass a portion of a given shape, the shape itself is not selected, but rather only the vertices that fall within the selection window. To modify this behavior so that individual vertices are not selected when you click an individual vertex or use a selection window, turn off the *Vertices* filter through *Options* > *Preferences* > *Select*.

The illustration that follows shows what happens when the Vertices filter is on, you draw a selection window enclosing parts of shapes on two different layers, and then choose *Edit* > *Move* > *Move Using Reference*.



Hint When the Vertices filter is turned on, all selected vertices are identified by a marker. You can change the size of this marker with the *Selected Vertex* option through *Options* > *Preferences* > *Select*.

Copying and Pasting Items

There are several Copy commands that enable you to copy and paste items in different ways:

- *Cut*—Enables you to delete one or more items from one window, and paste in another window.
- *Copy*—Enables you to copy items in a given design window and then paste those items within the same design window or another design window. On the PC only, it also copies the items to the Windows clipboard enabling you to paste the item(s) as a graphic in a Windows application.

Hint To enable pasting the item using coordinates as the reference point, choose *Options > Preferences > Entry/Edit* and select the option *Show Set Paste Origin Dialog for Copy command*.

- *Paste*—Enables you to paste items that you previously *cut* or *copied*

  Advanced Copy/Paste

  - *Copy Using Reference*—Enables you to copy selected items, prompting for a reference point and a destination point. The copied items can then be placed anywhere within the same design window.
  - *Copy Relative*—Enables you to copy items a specified distance from the original items
  - *Copy To Layer*—Enables you to copy items from one layer to another, within the same design window
  - *Step And Repeat*—Enables you to create a copy in the form of an array, with the number of rows and columns you specify

To copy an item and paste it on the same layer:

1. Select the item(s).
2. Choose **Edit** > **Copy** (or click the **Copy** button on the toolbar).

   If you enabled the aforementioned option, *Show Set Paste Origin Dialog for Copy command*, a dialog box appears enabling you to specify a reference point for pasting. (The default reference point for pasting a component, or group of components, is the first unconnected pin; when pasting shapes, the default

reference point is the lower left corner.) Specify the X and Y coordinates. (These are the *positional* coordinates identifying the position of the pointer in relation to the total window.) Click **OK**.

3. Choose **Edit** > **Paste** from the desired design window. As you move the pointer, a ghost image of the copied item moves with it.
4. Click again to specify the destination point.

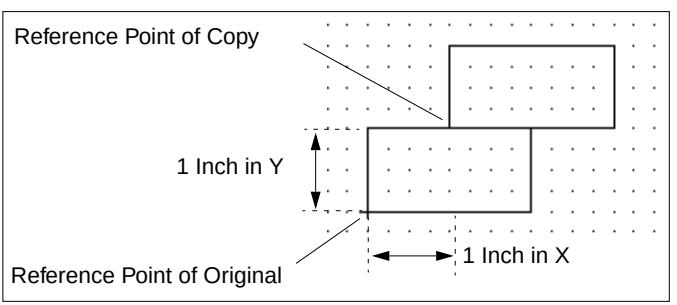
To copy items using a reference point:

1. Select the item(s).
2. Choose **Edit** > **Advanced Copy/Paste** > **Copy Using Reference**.
3. You are prompted to enter a reference point. Click the point on the item (or group of items) you want to use as a reference point for positioning the copy.
4. You are prompted to enter the offset location. Click to place the item(s) in the desired location.

To create a copy in a specific position, relative to the selected item:

1. Select the item you want to copy.
2. Choose **Edit** > **Advanced Copy/Paste** > **Copy Relative**. A dialog box appears enabling you to specify the distance from the original that the copy should be placed.

   The values you specify here are with respect to the units of the window, *schem* or *lay*. For example, using the default *inches* of the Schematic window, if you supply 1.0 for both X and Y, the reference point of the copied item will be placed 1 inch in the direction of X and 1 inch in the direction of Y from the reference point of the original item.



Hint The reference point on symbols is the first unconnected pin; the reference point on shapes is the lower left corner.

3. Specify the desired units in X and Y and click **Apply**. A copy appears at the specified location.

To copy from one layer and paste to another:

1. Select the item(s).
2. Choose **Edit** > **Advanced Copy/Paste** > **Copy To Layer**.
3. Select the desired destination layer from the dialog box that appears and click **Apply**.

Important Clicking *Apply* enables you to continue copying items from the current layer to any other layer. If this is the only layer you want to copy the selected shape to, then click *Cancel*; if you click OK, you will paste an additional copy on the last layer selected.

The *Step and Repeat* command enables you to select an item or items you would like multiple copies of and then specify how many copies and how many rows and/or columns. This command also enables you to specify the distance between the items in X and Y coordinates.

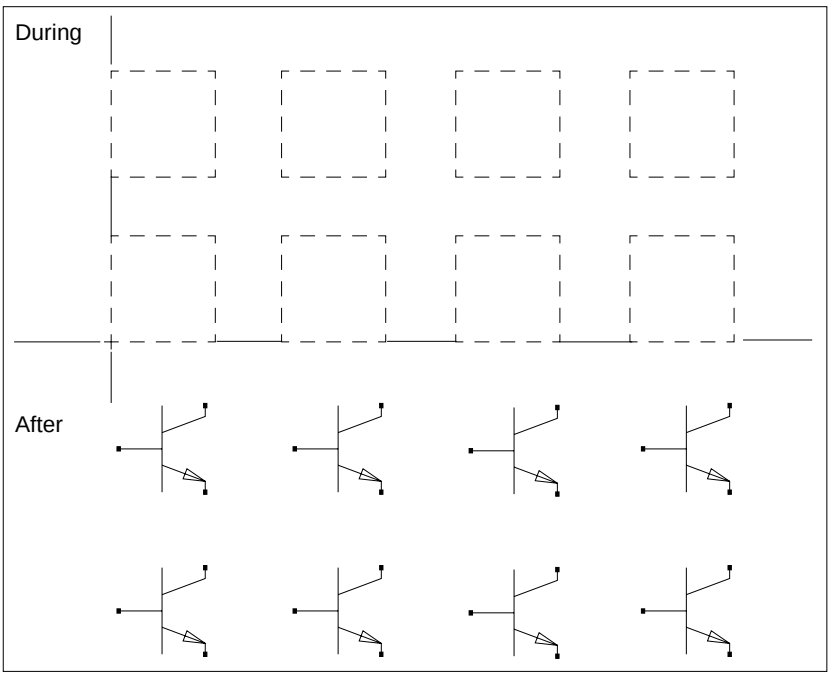
To copy items using the *Step and Repeat* command:

1. Select the item or items you want to copy.
2. Choose **Edit** > **Advanced Copy/Paste** > **Step and Repeat**. A dialog box appears.
3. By default, the current snap spacing is displayed as the X and Y spacing in this dialog box. You can either accept the default spacing or change these numbers by typing or clicking the up and down arrows.
4. Specify the number of rows and columns by typing or by clicking the up and down arrows.
5. To automatically connect the pins of these items with one another, select the option *Connect overlapping pins* and change the X or Y spacing field to *0*, based upon the component and the desired configuration.

6. Click **Apply**. As you move the pointer into the drawing area, a ghost image of the items moves with it. The lower left corner of the group serves as the reference point for placement.

Hint The *Step and Repeat* command is a repeating command; you can place this same configuration as many times as you like. Or you can change the configuration each time and choose *Apply* before placing.

An illustration using a 2 × 4 array of BJTs is shown next.



Moving Items

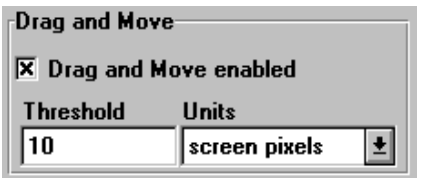
The quickest way to move objects is with the mouse:

1. Position the pointer over the object(s) you want to move.

Hint To move several items at once, select them using a selection window. To add additional items to (or delete items from) a selected group, use Shift+Click.

---

2. Press the left mouse button, drag to the new location, and release. When moving items using the drag method, the move must be more than the distance specified as the *Threshold* for it to be recognized as a move.



This option (*Options* > *Preferences* > *Entry/Edit*) protects you from moving an item unintentionally if you click to select it and accidentally move the pointer.

There are several additional *Move* commands that enable you to move components and shapes in specific ways:

- *Move Using Reference*—Enables you to move selected items, prompting for a reference point and a destination point
- *Move Edge*—Enables you to stretch the edge (between two vertices) of an existing shape

---

Hint This movement can be restricted by setting the option *Maintain adjacent angles for Move Edge command*. Refer to "Setting Entry/Edit Options" on page 9-12.

- *Move Relative*—Enables you to move items by specifying coordinates, relative to 0,0
- *Move & Disconnect*—Breaks connections as you move selected components
- *Move To Layer*—Enables you to move items from one layer to another

- *Move Wire Endpoint*—Enables you to manipulate an unconnected end of a wire

To move items specifying the reference and destination locations:

1. Select the item(s) you want to move.
2. Choose **Edit** > **Move** > **Move Using Reference**. You are prompted to enter a reference location.
3. Specify a reference point by clicking the point on the item (or group of items) you want to use when specifying the destination. If you are moving component symbols, clicking a pin as a reference point will help you align and connect the items.

   As you move the mouse, a ghost image of the selected objects follows. You are prompted to enter an offset location.
4. Click again to place the items in the new location. Where applicable, wires are redrawn maintaining connections.

Hint The manner in which wires are redrawn is controlled by the option *Reroute entire wire attached to moved component*. For more information, refer to the section, "Setting Entry/Edit Options" on page 9-12.

To move an item a specific amount, relative to the coordinates 0,0:

1. Select the item you want to move.
2. Choose **Edit** > **Move** > **Move Relative**.
3. Specify the amount in X and the amount in Y (in inches) you want to move the selected object and click **Apply**. The item is moved, using the default reference point, by the specified amount.

Hint The default reference point of an item varies depending on the nature of the item—for a component, it is pin 1; for a shape, it is the lower left corner.

To move and disconnect:

1. Select the items you want to move.
2. Choose **Edit** > **Move** > **Move & Disconnect**. You are prompted to enter a reference location.
3. Specify a reference point by clicking on or near the selected items. If you are moving component symbols, clicking on a pin as the reference point will help you align and connect the items.

   As you move the mouse, a ghost image of the selected items follows. You are prompted to enter an offset location.
4. Click again to place the items in the new location. The connections of the items moved and the items left behind, are deleted. The interconnections among the items moved, remain intact.

To move shapes or text to another layer:

1. Select the object you want to move.

   Note Do not use the *Move To Layer* command to move ports to a different layer; set the *Layer* parameter of the port to the desired layer.

   ---

2. Choose **Edit** > **Move** > **Move To Layer**. A dialog box appears with a list of currently defined layers. Select the desired layer and click **OK**. The selected object immediately takes on the color and other display characteristics of the selected layer.

   Note The following items can be moved to another layer using the context-sensitive menu that appears when you right-click with the pointer positioned over any of these items: Polygon, Polyline, Rectangle, Circle, Arc, Text, Arrow, Wire, Construction Line, Path, Trace.

Rotating Items

There are several commands to assist you in rotating components and shapes. The *Advanced Rotate* commands are more often used for shapes than components, but they do operate on components. For details on basic component rotation, refer to "Rotating Components" on page 3-19.

Use any of the following methods to rotate components and shapes by a specified increment:

- Click the **Rotate By Increment** button on the toolbar.
- Press **Ctrl+r**.
- Choose **Edit** > **Rotate**.

Each of these actions rotates the component *n* degrees clockwise, where *n* is the increment specified in *Options* > *Preferences* > *Entry/Edit* > *Rotation Increment (angle)*. The default is 90 degrees.

The *Mirror About X* and *Mirror About Y* commands enable you to rotate objects across an axis you specify.

Advanced Rotate

- *Rotate Around Reference* enables you to rotate components and shapes using the mouse, specifying the reference point and the destination point.
- *Rotate Relative* enables you to rotate components and shapes by a specific number of degrees, relative to the 0,0 coordinates of the drawing area.
- *Set Rotation Angle* rotates the selected item by the number of degrees specified, in conjunction with the Rotate command. Note that setting this angle resets the Rotation Increment (angle) in *Options* > *Preferences* > *Entry/Edit*.

Hint The *Rotate* and *Rotate Around Reference* commands can both be used on either components or shapes, but the *Rotate* command is typically better for working with components (the reference point is specified by the program) whereas the *Rotate Around Reference* command is typically better for working with shapes (you are prompted to specify the reference point around which to rotate).

Rotating Items Around a Specified Point

To rotate a selected object around a specified point:

1. Select the object.
2. Choose **Edit** > **Advanced Rotate**> **Rotate Around Reference**. You are prompted to enter a reference location.
3. Click once on the object at the point around which you want to rotate it. You are prompted to enter the offset location. As you move the mouse, a ghost image of the object moves with it. The pointer snaps in increments of the number of degrees specified in the *Rotation Increment (angle)* field in the Preferences dialog box (*Options* > *Preferences* > *Entry/Edit*).
4. Move the pointer until you are satisfied with the angle of rotation and click to place it.

Rotating Items in Degrees, Relative to 0,0

To rotate a selected object by a specific number of degrees, relative to 0,0:

1. Select the object.
2. Choose **Edit** > **Advanced Rotate** > **Rotate Relative**.
3. In the dialog box that appears, enter the number of degrees by which you want the object rotated.

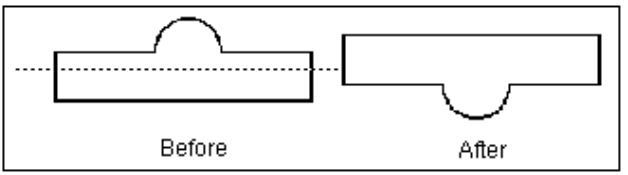
   Hints:

   - Positive values rotate the object in a counterclockwise direction; negative values rotate the object clockwise.
   - The reference point for rotating shapes is the lower left corner.
   - The reference point for rotating components is the left-most pin 1.
   - Values entered here are rounded up or down to the nearest incremental value in accordance with the number of degrees specified in the *Rotation Increment (angle)* field in the Preferences dialog box (*Options* > *Preferences* > *Entry/Edit*).

Rotating Objects Across a Specified X- or Y-axis

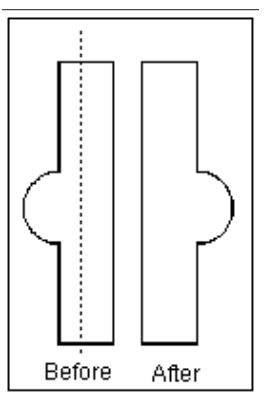
To rotate selected objects across a specified X-axis (mirror):

1. Select the object.
2. Choose **Edit** > **Mirror About X**. You are prompted to enter a point on the X-axis.
3. Click to specify the X-axis over which you want the object rotated. The selected object is rotated.



To rotate selected objects across a specified Y-axis:

1. Select the object.
2. Choose **Edit** > **Mirror About Y**. You are prompted to enter a point on the Y-axis.
3. Click to specify the Y-axis over which you want the object rotated. The selected object is rotated.



Rotating Objects Using an Absolute Angle

You can specify an absolute angle of rotation using the *Set Rotation Angle* command. This command is used in conjunction with the *Rotate* command.

Note The angle you specify here becomes the new Rotation Increment (angle) in *Options > Preferences > Entry/Edit*.

To rotate an object using an absolute rotation angle:

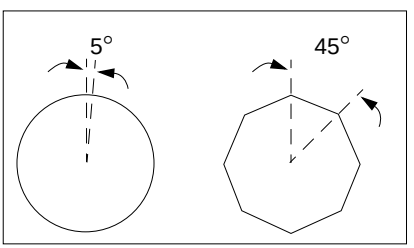
1. Select the object you want to rotate.
2. Choose **Edit** > **Advanced Rotate** > **Set Rotation Angle**. The Set Rotation Angle dialog box appears.
3. Specify the desired rotation angle in degrees.
4. Click **Apply** and click the **Rotate** button. The object is rotated by that amount.

Editing Shapes

There are several ways you can edit or manipulate various shapes after you have added them to your design:

Converting Circles/Arcs to Simple Polygons

Circles and arcs can be converted to simple polygons allowing vertex editing. The smoothness of the converted circle or arc is determined globally by the setting *Arc/Circle Resolution (degrees)* in *Options* > *Preferences* > *Entry/Edit*. You can use different settings for individual shapes using *Edit* > *Modify* > *Arc* or *Edit* > *Modify* > *Circle*.



To convert a selected circle to a polygon:

Choose **Edit** > **Modify** > **Convert To Polygon**.

To edit the characteristics of a selected circle:

1. Choose **Edit** > **Modify** > **Circle**.
2. Select the desired mode for changing the radius:
   - Absolute Radius—Select this option to scale a circle by an absolute amount. For example, if your circle has a radius of .75 inches and you specify a Radius of 1, the circle is scaled such that the radius is 1 inch.
   - Delta Radius—Select this option to scale a circle by a relative amount. For example, if your circle has a radius of .75 inches and you specify a Radius of 1, the circle is scaled such that the radius is 1.75 inches.
3. Specify the Radius, the amount by which you want the circle scaled.
4. Specify a new number for resolution, if desired. (See the illustration at the beginning of this section.) Note: This changes the resolution for the selected circle only, independent of the global setting made through the Preferences dialog box, *Options > Preferences > Entry/Edit*.
5. Specify a different layer for the circle, if desired.
6. Click **Apply** (or *OK* if you are done editing circles).

To edit the characteristics of a selected arc:

1. Specify a new number for resolution, if desired. (See the illustration at the beginning of this section.) Note: This changes the resolution for the selected arc only, independent of the global setting made through the Preferences dialog box, *Options > Preferences > Entry/Edit*.
2. Specify a different layer for the arc, if desired.
3. Click **Apply** (or *OK* if you are done editing arcs).

Editing Polygons and Polylines

Several commands found on the *Edit* > *Modify* menu can assist you in editing polygons and polylines.

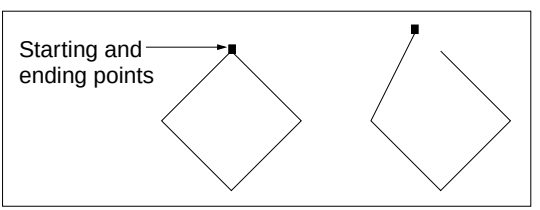
- The *Join* command allows selected polylines with coincident endpoints to be joined into a single polyline. If a closed shape results, the joined polylines are converted to a polygon.
- The *Break* command converts a selected polygon into a single polyline.
- The *Explode* command converts selected polygons or polylines into two-point polylines.

To join multiple polylines into a single polyline:

1. Select the individual polylines you want to join.
2. Choose **Edit** > **Modify** > **Join**. All coincident endpoints are joined. You can verify what has been joined by clicking on the shape to select it and observing whether or not the entire shape is selected.

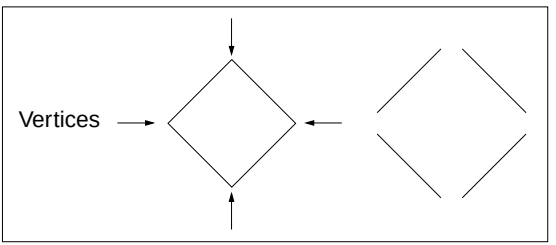
To convert a polygon into a single polyline:

1. Select the polygon.
2. Choose **Edit** > **Modify** > **Break**. The starting and ending points of the polygon are broken, identified by a marker, and you can now manipulate the shape as a polyline.



To convert a polygon or polyline to individual, two-point line segments:

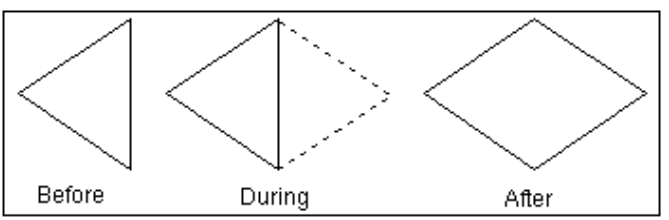
1. Select the polygon or polyline.
2. Choose **Edit** > **Modify** > **Explode**. All vertices are disconnected leaving you with individual line segments that you can edit as needed.



Adding a New Vertex

To add a new vertex to a polygon or polyline:

1. Choose **Edit** > **Vertex** > **Add**.
2. Click a point between two existing vertices, and move the pointer. A flexible line is drawn between the vertices and the pointer.
3. Click again to specify the new vertex and the shape is redrawn.



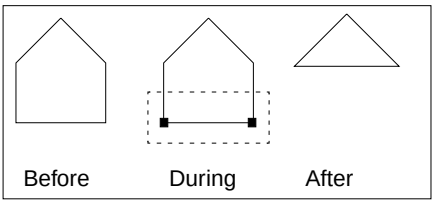
Moving a Vertex

To move a vertex on a polygon or polyline to change its shape:

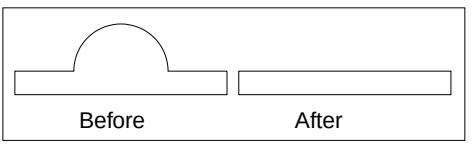
1. Click to select the vertex and drag the pointer in the desired direction. A flexible line is drawn from the affected vertex to the pointer.
2. Click again to specify the new location of the vertex and the shape is redrawn.

Deleting a Vertex

To delete a vertex on a shape you have drawn, be sure the *Vertices* filter is turned on (*Options* > *Preferences* > *Select*) and draw a selection window enclosing all vertices you want to delete. Click the *Delete* button on the toolbar and the shape is redrawn without those vertices.



Clicking anywhere on an arc deletes the arc and connects the former endpoints of the arc with a straight line.



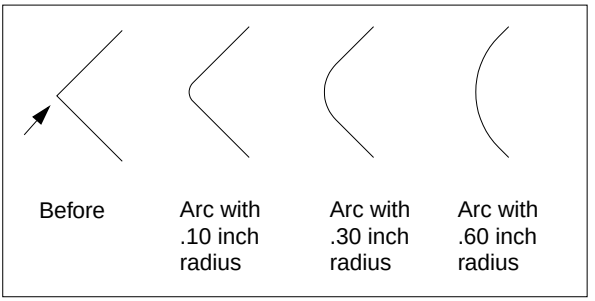
Converting a Vertex to an Arc

You can convert any vertex to an arc and specify the desired radius of the arc, with respect to the units of the window.

To convert a vertex to an arc:

1. Choose **Edit** > **Vertex** > **To Arc**. You are prompted *enter location of the vertex* and a dialog box appears.
2. Change the radius as desired and click **Apply**.
3. Click any vertex you want to convert to an arc. The vertex is redrawn accordingly.

You can continue converting vertices in this manner using a different radius each time if desired, but you must click *Apply* each time you change the radius. When you are through making these changes, click *OK* to dismiss the dialog box.



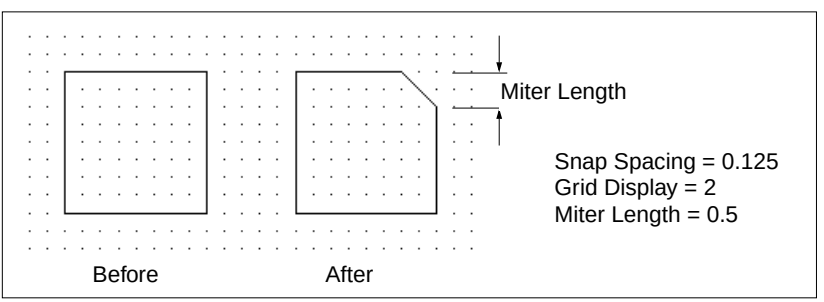
Converting a Vertex to a Mitered Edge

You can convert any vertex to a mitered edge and specify the desired length of the mitered edge, with respect to the units of the window.

To convert a vertex to a mitered edge:

1. Choose **Edit** > **Vertex** > **Miter**. You are prompted *enter location of the vertex* and a dialog box appears.
2. Change the miter length as desired and click **Apply**.
3. Click any vertex you want to convert to a mitered edge. The vertex is redrawn accordingly.

You can continue converting vertices in this manner using a different miter length each time if desired, but you must click *Apply* each time you change the length. When you are through making these changes, click *OK* to dismiss the dialog box.

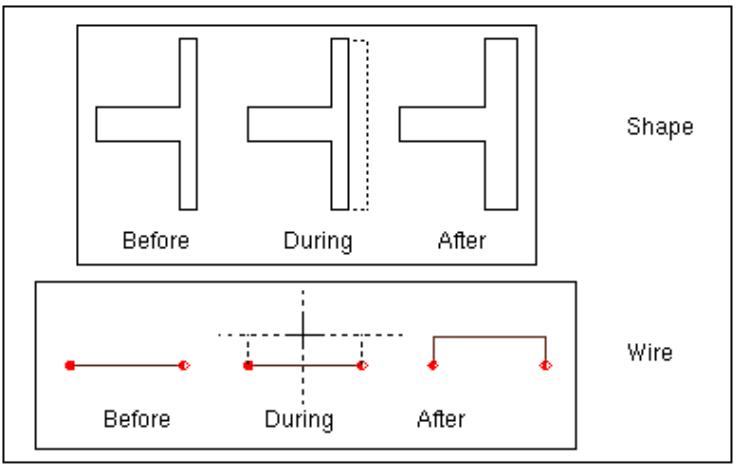


Stretching a Wire or an Edge of a Shape

The *Move* > *Move Edge* command enables you to change the shape of an existing wire, or to redefine a shape by stretching an *edge* (a segment between two vertices).

To stretch an edge:

1. Choose **Edit** > **Move** > **Move Edge**. You are prompted to enter the location of the line.
2. Click the edge you want to stretch. As you move the pointer, a ghost image moves with it and showing how the shape will be redrawn.
3. Click again to define the new shape.



Scaling an Object Using a Scaling Factor

To scale an object using a scaling factor:

1. Choose **Edit** > **Scale/Oversize** > **Scale** and the Scale dialog box appears.
2. Enter scaling factors for both X and Y.

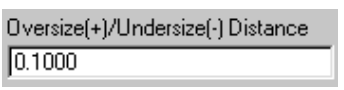
   Scaling factors must be positive. Scaling factors greater than 1.0 increase the size of objects, while factors less than 1.0 decrease the size of objects. To scale the objects uniformly, enter the same scaling factor for both X and Y.
3. Click **OK** and you are prompted to enter a reference point on the object around which to scale.
4. Click to specify the reference point and the object is scaled.

Scaling an Object Relative to the Design Window Units

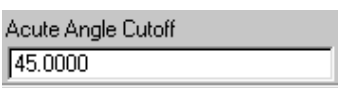
There are two commands, *Oversize* and *Copy & Oversize*, that enable you to scale an object with respect to the design units, for example, inches or mils. The *Oversize* command replaces the original image with a scaled image. The *Copy & Oversize* command places a copy of the selected object, using the size you specify, on the current entry layer, preserving the original object.

To scale the object itself, in Schem or Layout units:

1. Select the object.
2. Choose **Edit** > **Scale/Oversize** > **Oversize** and a dialog box appears.

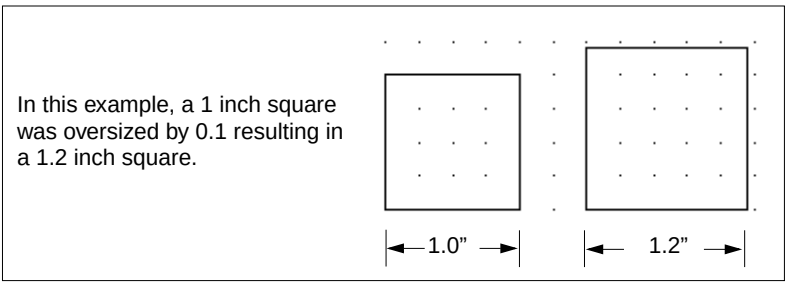


   This field enables you to specify an amount by which the selected object should be scaled (in *all* directions). A positive number increases the size of the object by that amount; a negative number decreases the size of the object by that amount.



This field enables you to specify a 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.

3. Make any necessary changes in the dialog box, and click **OK**.



To make a scaled copy of an object, in Schem or Layout units:

1. Select the object.
2. Choose **Edit** > **Scale/Oversize** > **Copy & Oversize** and a dialog box appears.
3. Specify the desired angle and click **OK**.

Editing Existing Text and Text Attributes

There are two methods of editing text you have added to your design: you can use the on-screen editor or edit through a dialog box. Note: To edit text attributes, you must use the dialog box method.

To edit text using the on-screen method:

1. Position the pointer over the text you want to edit and click. Notice the following changes:
   - The text takes on the color currently defined for *Highlight* in the Preferences dialog box (*Options* > *Preferences* > *Display*).
   - The status panel prompt changes to read, *On-screen Text Editor: in progress*
   - A vertical bar (|) representing a text insertion cursor appears in the line of text.

2. Use the arrow keys or the mouse to reposition the cursor, as needed, near the text you want to change. Use the backspace key, as necessary, to make your changes.
3. When you are through, move the pointer away from the text and click once to end the text editing command.

To edit text using the dialog box method:

1. Choose **Edit** > **Edit Text**. The dialog box appears and you are prompted to enter the location of the text.
2. Click to select the text you want to edit. The selected text appears in the dialog box for editing. (If you type multiple lines as a block of text, that block appears.)
3. Make any desired changes to the text.
4. Make any desired changes to the text attributes.
   - Font Type—Schematic only—All True Type fonts installed on your system are available. Select the desired font from the drop-down list.

Note On UNIX, if you want to add additional True Type 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.
   - Layer—Enables you to select a different layer for the text.
   - Placement Angle—Enables you to rotate existing text by specifying an angle in degrees. Positive values rotate the text counterclockwise; negative values rotate it clockwise.
   - Non-rotating (when in hierarchy)—Select this option to prevent text on a symbol or design from being rotated when the symbol is rotated.
   - 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.

5. Click **Apply**. Changes to the selected text are reflected immediately in the drawing area.

   You can continue to select text and make changes, but you must click *Apply* for each set of changes for those changes to take effect.

6. When you are through making changes to text, click **OK** to dismiss the dialog box.

Editing Wire/Pin Label Attributes

To change the color, size, and font of existing wire labels, choose one of the following methods:

- Use *Edit* > *Wire/Pin Label* > *Wire/Pin Label Attributes*
- Double-click the wire label to display the dialog box
- Right-click and select *Wire/Pin Label Attributes* from the pop-up menu

Forcing Objects Back onto the Grid

If an object is offset from the current grid spacing, you can force it back to the nearest grid point with the *Modify* > *Force to Grid* command. If the selected object is an item with pins, pin 1 is forced to the nearest grid point.

To force an object back onto the grid:

1. Select the object.
2. Choose **Edit** > **Modify** > **Force to Grid**. The selected object snaps to the grid.

Chapter 7: Annotating Designs

You can annotate your schematic by adding a drawing sheet, inserting a variety of shapes, and adding text (including system variables such as date, time, etc.). For details, refer to the following topics:

Adding a Drawing Sheet

If you want to use a drawing sheet that contains any constant information or graphics, such as a company logo, you will probably want to save it as a template enabling you to insert it in any design where it is needed.

To create a drawing sheet:

1. Choose **File** > **New Design**.
2. Supply a name for the file.
3. Choose **Insert** > **Component** > **Component Library**.
4. Select the **Drawing Formats** category.
5. Select the appropriate drawing sheet size, for example, **FORMATA**.
6. Click **OK** and move the pointer into the drawing area.
7. Note that the lower left corner serves as the reference point. Position the image of the drawing sheet as required and click to place it there.

To move the drawing sheet after placing it, you must be able to select it, and by default, the *Drawing Format* filter is turned off. To turn it on, choose *Options* > *Preferences* > *Select* and enable the *Drawing Format* option. If you do enable this filter, you will probably want to disable it again after moving the drawing sheet, before beginning your design work.

Hint If you want to reposition the lower left corner of the drawing sheet at the coordinates 0,0 (to assist in placing items using exact measurements), you can use the *Set Origin* command. Choose *Edit* > *Modify* > *Set Origin* and click the lower left corner of the drawing sheet. Use *View All* if necessary to bring the entire sheet into view.

---

8. Add all desired information.
9. Choose **Save Design As Template** to make it available for insertion in any design.

Adding Text

Once you have established the desired text attributes, you can add text to your design using the *Text* command on the Insert menu or the Text button on the toolbar. Prior to adding text, you should change the current entry layer to *text1* (or a text layer you have created) and place all text on that layer. For details on which type of objects are placed on which layers, refer to the section, "Specifying Layer Definitions" on page 9-22.

To change the current entry layer:

Choose **Insert** > **Entry Layer** > **text1**.

Any text you add, or object you draw, will now be placed on the *text1* layer until you change to another entry layer. Text will take on the characteristics defined in the Preferences dialog box (except for color, which is defined through the Layer Editor). For details on changing attributes of text:

- Prior to adding it to the drawing area, refer to the section, "Setting Text Options (in Advance)" on page 9-17.
- After adding it to the drawing area, refer to the section, "Editing Existing Text and Text Attributes" on page 6-34.

To add text to your design:

1. Choose **Insert**> **Text**. The status panel prompt changes to read, *New_Text: Enter location for new text*.
2. Position your cursor in the desired location, click once, and begin typing.

If necessary, use the arrow keys, the backspace and spacebar to make changes. (You can drag the mouse across text to highlight it and type over it or use the spacebar or backspace to delete it.) To continue the text on the next line, press Enter and continue typing.

- To type text in another location, click in that location and begin typing.
- To stop the text command, press **Esc** or move the pointer away from the new text and click once—to signify the end of that text block—then click the **End Command** button.

Hint Context-sensitive editing is available for text. Position the pointer over the text, right click, and select *Edit Text* from the pop-up menu.

Using Variables to Display Design and System Information

You can add any of the following variables to your design to display the type of information indicated:

@dname	= Design name
@pname	= Full path to design
@ddate	= Date design was last saved
@dtime	= Time design was last saved
@cdate	= Current date
@ctime	= Current time
@SIM_HOST	= Name of machine hosting the simulation
@SIM_DDS	= Name of the simulation data set

To add variables to your text:

1. Choose **Insert** > **Text**.
2. Click to begin typing text and include any of the variables shown above as desired. For example:

```
This version of @dname was last
saved on @ddate at @dtime.
```

3. Press **Esc** to stop the *Text* command and the variables are replaced by their equivalents.

This version of spar_sim was last
saved on Jan 12, 2001 at 10:11 AM.

Drawing Shapes

The Insert menu contains commands that enable you to draw a variety of shapes and lines to help annotate your schematic. Select *End Command* from the pop-up menu (or click the *End Command* toolbar button, or press *Esc*) during execution of any draw command, to terminate the command and remove the partial shape.

Hints:

- Several shapes that require you to specify multiple points (such as polygons, polylines, arcs) allow you to specify the last point by pressing the space bar.
- Context-sensitive editing is available for several shapes (rectangle, polygons, polylines) with respect to changing the layer on which the shape is drawn. Position the pointer over the shape, right click, and select the layer command from the pop-up menu.
- You can change the thickness of the lines of the following shapes drawn in the Schematic window (schematic view): polygons, polylines, rectangles, circles, arrows, and all arcs. (*Edit* > *Modify* > *Line Thickness*)
- You can set (in advance) the thickness of the lines of the following shapes drawn in the Schematic window (symbol view): polygon, polyline, rectangle, circle. To edit line thickness subsequently, choose *Edit* > *Modify* > *Line Thickness*.

To draw a rectangle (or square):

1. Choose **Insert** > **Shape** > **Rectangle** (or click the **Rectangle** button). You are prompted to enter the first corner.
2. Click to specify the first corner. You are prompted to enter the second corner.
3. As you move the mouse, a flexible box stretches from the anchor point to the position of the mouse, expanding and contracting as you move. When you are satisfied with the size, click to finish.

To draw a polygon:

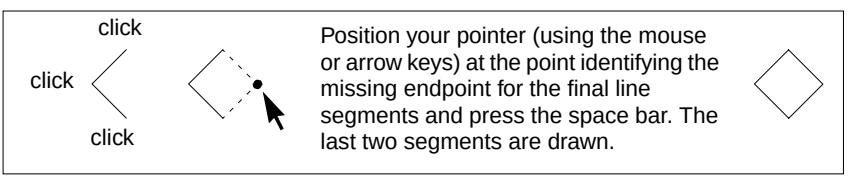
1. Choose **Insert** > **Shape** > **Polygon** (or click on the Polygon button). You are prompted to enter a vertex.
2. Click to anchor the starting point of the polygon.
3. Move the cursor to the position where you want the first line segment to end, and click. A line appears between the two points.
   - At any time during entry of a polygon, you can choose one of the *Arc* commands to include an arc in your polygon.
   - You can use the *Undo Vertex* command to backtrack to the previous point anytime you want to erase the segment or arc you have just drawn.

Hint If you want all line segments to be drawn perfectly horizontal or vertical, choose *Options* > *Preferences* > *Entry/Edit Mode* and check *Orthogonal*.

---

4. Continue in this manner until you are ready to draw the final line segment(s) completing your polygon.
   - To draw all but the last line segment, double-click *or* press the space bar while the pointer is still positioned over the last vertex.
   - To draw all but the last two line segments, move the pointer so that it is positioned over the next vertex, and double-click *or* press the space bar.

     The polygon is automatically closed. The illustration demonstrates the latter method.



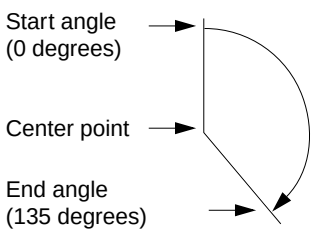
To draw a polyline (a series of connected line segments):

1. Choose **Insert** > **Shape** > **Polyline** (or click on the Polyline button). You are prompted to enter a vertex.
2. Click to anchor the starting point of the polyline.

3. Move the cursor to the position where you want the first line segment to end, and click. A line appears between the two points.
   - At any time during entry of a polyline, you can choose one of the *Arc* commands to include an arc in your polyline.
   - You can use the *Undo Vertex* command to backtrack to the previous point anytime you want to erase the segment or arc you have just drawn.
4. When you have drawn the final line segment, double-click or press the space bar to end the *Polyline* command.

To draw an Arc:

1. Choose **Insert** > **Shape** > **Arc (clockwise** or **counterclockwise)**. You are prompted *Enter the start point of the arc.*
2. Move the cursor into the drawing area and click left to specify the start point of the arc. You are prompted *Enter the arc center.*
3. Click to specify the center point of the arc. A flexible arc appears as you move your cursor. You are prompted *Enter the ending point of the arc.*
4. Click to specify the end point. If you are through drawing arcs, click **End Command**.



Hint Context-sensitive editing is available for arcs. Position the pointer over the arc, right click, and select *Edit Arc* from the pop-up menu.

You can also draw an arc by specifying the start point, end point, and circumference. To draw an arc in this manner:

1. Choose **Insert** > **Shape** > **Arc (start, end, circumference)**. You are prompted *Start pt of arc.*

2. Move the pointer into the drawing area and click left to specify the start point of the arc. You are prompted *End pt of arc*.
3. Click to specify the end point of the arc. You are prompted *Circ point of arc*. A flexible arc appears as you move the pointer.
4. Click to specify the final size of the arc.

To draw a circle:

1. Choose **Circle** from the toolbar or from the Insert menu. You are prompted to enter the circle center.
2. Click to specify the center point.
3. Drag the mouse until the circle is the desired size and click.

Hint Context-sensitive editing is available for circles. Position the pointer over the circle, right click, and select *Edit Circle* from the pop-up menu.

To draw an arrow:

1. Choose **Insert** > **Arrow**. You are prompted to enter the first point. In the dialog box that appears, set the following options as needed:

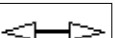
   - Number of Arrowheads

     One  Two

   - Polygon arrowhead—select to draw arrowhead as a polygon

     Polyline arrowhead  Polygon arrowhead

   - Arrowhead width in schematic units:

     Arrowhead width increased (from default of 0.08) to 0.125 

   - Arrowhead length in schematic units:

     Arrowhead length decreased (from default of 0.25) to 0.125 

Hint To restore the default values/options, click *Default*.

2. Click **Apply**.
3. Click to specify the first point of the arrow. You are prompted to enter the second point.
4. Move the pointer as needed to identify the endpoint of the line segment and click.

To change the line thickness of an existing shape:

1. Select the shape.
2. Choose **Edit** > **Modify** > **Line Thickness**.
3. Select the desired thickness and click **OK**.

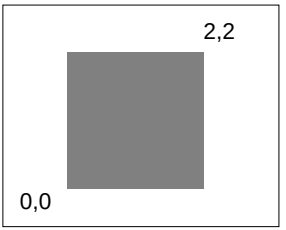
Drawing Shapes Using Specific Coordinates

The *Coordinate Entry* command enables you to draw a variety of shapes by specifying coordinates for each vertex.

To use the coordinate entry method for drawing:

1. Choose **Insert** > **Coordinate Entry** and a dialog box appears. Move the dialog box so that the desired design window is visible.
2. Choose the required drawing or editing command.
3. Enter the X and Y coordinates for the first point.

   You can enter coordinates by typing them directly in the fields labeled *X* and *Y*, or you can use the up and down arrows of the X Increment and Y Increment fields to increase or decrease the values of the *X* and *Y* fields. By default, the X and Y Increment fields are set to the current snap spacing, but you can use any increment that meets your design needs.



Hint If using the *Rectangle* command, specify coordinates for two corners opposite each other.

4. Click **Apply**.
5. Continue specifying all desired points, clicking *Apply* for each.

Hint When drawing polygons with coordinate entry, click Apply for the final segment of the shape to be drawn automatically.

Chapter 8: Simulating and Viewing Results

The information presented here explains only the basics of simulating and viewing results in the Advanced Design System. Simulation is described in detail in the *Circuit Simulation* and *Agilent Ptolemy Simulation* manuals. Viewing results is described in detail in the *Data Display* manual.

All simulations are performed from the Schematic window. If you are simulating typical microwave/RF circuits, you may want to set up your simulation using the *Smart Simulation Wizard*; for other types of simulation you will probably need to set them up manually. Refer to one of the following sections, based on your needs:

- "Using the Smart Simulation Wizard" on page 8-1
- "Setting up a Simulation Manually" on page 8-6

Using the Smart Simulation Wizard

The *Smart Simulation Wizard* is provided to assist new ADS users, as well as those who use it infrequently, in setting up simulations for typical microwave/RF circuits. The wizard will guide you through the process of:

- Selecting an application-specific design (or your own design)
- Selecting predefined simulation setups
- Specifying simulation settings (frequency, bias, etc.)

The wizard then configures the sources and simulation controls and begins the simulation(s). When multiple simulations—requiring different configurations—are requested, the wizard automatically reconfigures the subnetwork for the appropriate sources, terminations, and simulation controls. When the simulation is finished, simply click to display the results. Note that although basic simulation setups are provided with the various simulator licenses, additional simulation setups require specific DesignGuide licenses. These differences are identified in the wizard.

To invoke the *Smart Simulation Wizard*:

From the Schematic window in the project of interest, choose **Simulate** > **Smart Simulation Wizard**.

Step 1 prompts you to select one of several different application types.

Device Characterization

BJT Characterization

FET Characterization

MOSFET Characterization

Amplifier

Amplifier

Mixer

Single-Ended Mixer

Linear Circuit

Linear 2-port

Linear 4-port

Step 2 prompts you to select one of the following design types:

- A sample design provided by the *Smart Simulation Wizard*
- An existing ADS subnetwork design
- A new subnetwork design

Step 3 varies based on the choice made in Step 2. You are prompted to select an existing design, enter a name for a new design, or select one of the following application-specific designs.

Device Characterization			
	BJT Characterization		
		NPN BJT	NPN BJT model, biased with IBB = 60 uA, VCE = 2.7V.
		PNP BJT	PNP BJT model, biased with IBB = -60 uA, VCE = -2.7V.
	FET Characterization		
		GaAs MESFET Statz Model	Statz FET model for device FLC301XP.
		EEFET Model	EEFET3 FET model for device FLC081XP.
		GaAs MESFET Model	Basic MESFET model.
		HEMT Model	Basic HEMT model.
		JFET Model	Basic JFET model.
	MOSFET Characterization		
		NMOSFET Model	Basic BSIM3 model for NMOSFET. Width = 1e-5, Length = 2.5e-7.
		PMOSFET Model	Basic BSIM3 model for PMOSFET. Width = 1e-5, Length = 2.5e-7.
Amplifier			
	Amplifier		
		MOSFET Power Amplifier	Power Amplifier with a single MOSFET, 14 dB gain between 750 - 800 MHz.
		BJT Power Amplifier	Power amplifier with 8 BJTs, 12 dB gain at 2 GHz.
		Behavioral Model Amplifier	Ideal amplifier with Behavioral model. Gain, S-parameters and noise figure can be specified directly.
Mixer			
	Single-Ended Mixer		
		MESFET Gilbert Cell Mixer	MESFET Gilbert Cell Mixer internally matched to 50 ohm at 900 MHz.
		FET Mixer	Single-ended MOSFET Mixer.
		BJT Gilbert Cell Mixer	Single-ended BJT Gilbert Cell Mixer.
		Behavioral Model Mixer	Ideal Mixer Behavioral model.

Linear Circuit			
	Linear 2-port		
		Simple Lowpass Filter	Simple LC lowpass filter with cut-off frequency at 10 MHz.
		Microstrip Bandpass Filter	Simple bandpass filter composed of two concatenated microstrip subnetworks. Center frequency: 12 GHz. 10% bandwidth.
		S-Parameter Data File	Two-port subcircuit defined by an S-parameter file *nec71000.dat*.
		Linear FET	Linear FET model for small-signal modeling.
	Linear 4-port		
		Linear FET Modeling	Matching a linear FET model to measured S-parameters. Measured data file *nec71000.s2p*.

Step 4/Step 5 varies based on your previous choices. For an existing ADS design, you are prompted to identify the port type for each port in your design (input, output, base, collector, etc.). For all design types, the wizard then describes how to view the network associated with the schematic symbol and how to access the simulation setup portion of the wizard.

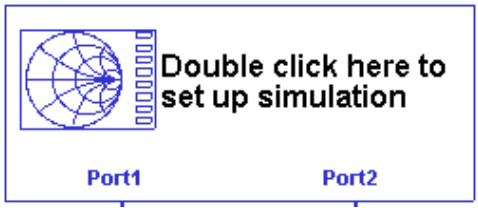
When you click *Finish*, the top-level design appears, and you will see that it consists of two main parts: a *schematic symbol* representing the subnetwork to be simulated and a *simulation setup symbol*.

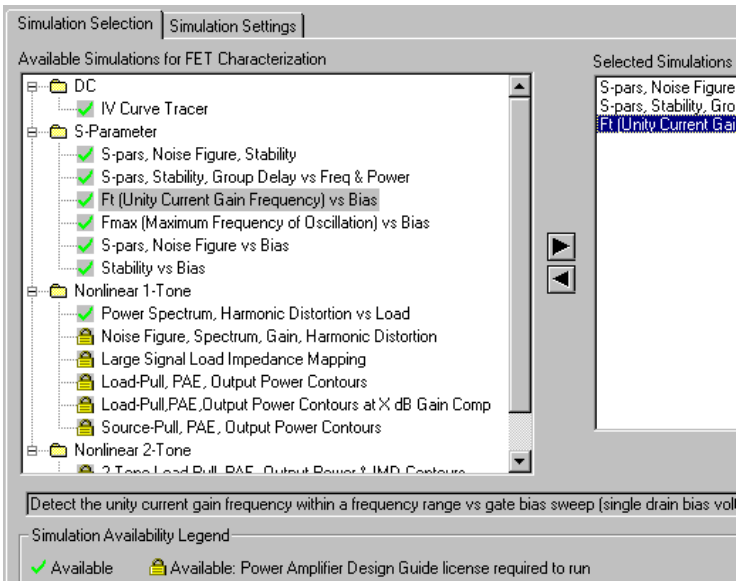
Note If working with a sample design, the top-level and subnetwork designs, as well as the related data displays, are copied to the current project. If you select an existing design from a different project (via an *Included* project), that design is copied to the current project.

- *Schematic symbol*—A schematic symbol representing the subnetwork to be simulated, is visually connected to the simulation setup symbol. *Push* into the symbol to view, edit, or create the subnetwork.

  When you push into most of the schematic symbols, you will notice that the subnetwork designs contain cautions against deleting or renumbering of ports.

- *Simulation setup symbol*—The simulation setup symbol is similar to the one shown next. Double-click (or right-click and select the first choice from the pop-up menu) to specify the simulation setup details.





Each simulation is marked with one of two icons, as shown next.

 —— Only the appropriate simulator license is required for this simulation

 —— A specific DesignGuide license is required, in addition to the appropriate simulator license, for this simulation

You can highlight any selected simulation (from the list box on the right) and click *Show Schematic* to view the design containing the simulation setup.

From the *Simulation Settings* tab you can specify the desired settings for the simulation parameters such as frequency, power, bias, etc. When you have selected all the desired simulations and specified the desired settings, click *Simulate* to proceed. The progress window appears and is dynamically updated

to indicate which simulations have completed and which remain. When all simulations are complete click *Display Results* to view the data displays. Note that the results for each simulation are displayed on separate pages, which can be accessed individually from the Page menu.

Hint After simulating a given design once, you can display the results from the previous simulation via the pop-up menu. Position the pointer over the simulation setup symbol, click right, and select *Display Data from Last Simulation*.

Setting up a Simulation Manually

These are the basic steps for setting up a simulation:

- Add the required circuit sources (Analog/RF designs only)
- Specify points for collecting data
- Place the appropriate simulation control item(s) in your design
- Specify a name for the dataset generated by your simulation

Optionally, you can establish a name for the resulting data display. You can also select a remote machine for simulating on.

If using the *Load Sharing Facility* (LSF) utility, you can break up a sweep and run the simulation on multiple machines, in parallel. You can also use this utility to select the fastest available machine.

Placing Simulation Control Items in Your Design

Simulating with the Advanced Design System is accomplished by placing simulation control items in the Schematic window along with your design (no connection required). The control items available vary with the current design type. Some of the categories of control items for each design type are listed next.

Analog/RF Designs

- Frequency-Domain Sources
- Time-Domain Sources
- Noise Controlled Sources
- Modulated Sources
- DC Simulation
- AC Simulation
- S-Parameter Simulation
- Harmonic Balance Simulation
- Large-Signal S-parameter (LSSP) Simulation
- Gain Compression (XDB) Simulation
- Envelope Simulation
- Transient Simulation

Digital Signal Processing Designs

- Data Flow Controller
- Sweep Plan
- Parameter Sweep
- Performance Optimization Controller

For Digital Signal Processing designs, you must also connect a *Sink* component or an *Interactive Controls and Displays* component to the output of your schematic. A Sink collects data or performs a measurement. The data can then be displayed via the Data Display window.

Controlling Simulation Data

All data produced by a given simulation is stored collectively as a *dataset*. Every dataset has a name associated with it. You can assign a name prior to simulating or you can accept the default dataset name. The default name is the name of the current design, with the extension *.ds* automatically appended.

Note If you accept the default dataset name and perform multiple simulations, the dataset will be overwritten each time. To collect separate datasets for each simulation, specify a unique name (for the data set you are about to create) prior to each simulation.

For details on selectively sending data to the dataset, refer to "Selectively Saving and Controlling Simulation Data" on page 1-15 in the *Circuit Simulation* manual.

For information regarding the naming of datasets, refer to "Limitations" on page 1-23 (under *Naming Conventions*).

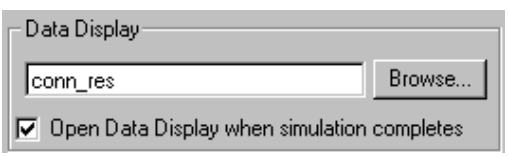
To specify a dataset name prior to simulating:

1. Choose **Simulate** > **Simulation Setup**.
2. Supply a name in the *Dataset* field. (Click Browse to view existing dataset names in the current project.)

   Hint Datasets are stored in the */data* subdirectory of a project. This means that you should provide unique names for the datasets that will be generated from the different designs in the same project directory.

   ---

3. Supply a name in the Data Display field, or accept the default name.



Data Display—The name you specify here will be the title of the Data Display window that is opened, and the default filename should you choose *Save As* (from the Data Display window). The default name shown is based on the current design name.

Open Data Display when simulation completes—If enabled, a Data Display window will open automatically when the simulation is complete. For details refer to the section, "Automatically Displaying Simulation Data" on page 8-10.

4. Select the desired Simulation Mode:

   - To simulate on a single machine, select **Single Host**. Select the **Single** tab and select the desired Simulation Host Type, *Local* or *Remote*. If you select *Remote*, you can then select *Specify* to choose a specific server by name, or select *Find Fastest* and let the *Load Sharing Facility* (LSF) choose the most suitable remote host.

   - To break up your sweep and simulate individual parts simultaneously on remote machines, select **Parallel Hosts**. Select the desired Sweep Variable from the drop-down list and set the desired Start, Stop and Step values. Individual sweep points are run on each machine and the results are combined into a single dataset on the local machine.

   - To break up a signal processing BER simulation over multiple hosts, select the **Parallel BER** option from the Parallel tab and, in the Number of Partitions field, specify the number of hosts to be used. This feature is only available when *berMC* or *berMC4* components are used. For more information, refer to the documentation of these components in the *Signal Processing Components* manual.

   Taking advantage of the LSF utility requires the installation and configuration of that software on the necessary files/machines. For details on setting up your remote and local machines for remote processing, refer to the appropriate appendices:

   - Appendix E, *Using Remote Simulation* in the *Installation on PC Systems* manual

   - Appendix D, *Using Remote Simulation* in the *Installation on UNIX Systems* manual.

5. Click **Simulate**.

Automatically Displaying Simulation Data

When setting up your simulation (*Simulate > Simulation Setup*, in the Schematic window), you can enable an automatic display of your results.

Select the *Open Data Display when simulation completes* option to force a Data Display window to open automatically when the simulation is complete. The data display that appears in that window depends on the simulation setup and the status of display templates:

- If you specify the name of an existing display to open, that display is opened.
- If you specify a new name, then a blank Data Display window opens.
- If one or more data display templates are associated with the current design, then the Data Display window opens and inserts each template on its own page. A data display template can be associated with a design in one of two ways:
  - By default, if your design includes a supplied schematic template, and a data display template is associated with that schematic template.
  - Explicitly, if you create your own data display template (*File > Save As Template* in the Data Display window) and associate that template with your design via the *DisplayTemplate* component. (Refer to the next section, "Using a DisplayTemplate Component" on page 8-11.)
- If none of the above criteria are met, but you select the option, then a blank Data Display window opens.

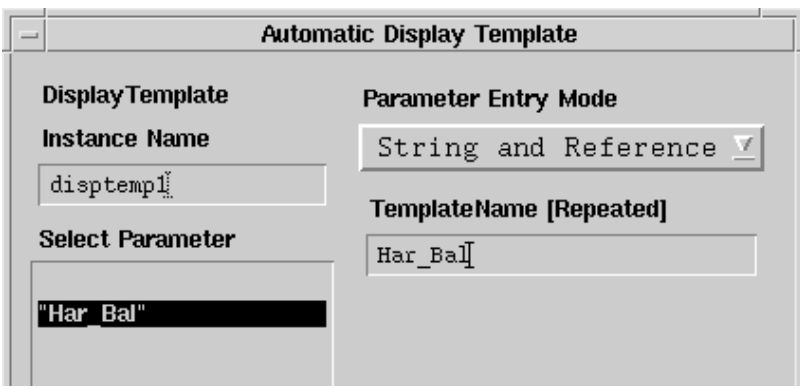
Using a DisplayTemplate Component

The *DisplayTemplate* component (available from most simulation libraries) enables you to associate one or more data display templates with a given design. (If you include one of the supplied schematic templates in your design, it most likely has a data display template associated with it.)

The starting point of this procedure assumes you have already created a data display file for use as a template, by setting up the Data Display window as desired and choosing *File > Save As Template*.

To associate a data display template with the current design:

1. Place a *DisplayTemplate* component in the Schematic window.



2. Select **String and Reference** as the Parameter Entry Mode.
3. Enter the name of the template (the filename you supplied in the Data Display window) and click **Add**.

Hint You can specify multiple templates for the same data display and subsequently access them from the *Page* menu).

---

4. When you are through specifying display templates for the current design, click **OK**.

Manually Displaying Simulation Data

If you do no want the Data Display window to open automatically, disable the option *Open Data Display when simulation completes* in the Simulation Setup dialog box.

To open a new window for displaying and manipulating data:

Choose **Window** > **New Data Display** from the Main, Schematic, or Layout windows.

To save a graph for later viewing/manipulating:

Choose **File** > **Save**.

To save a graph for use as a template:

Choose **File** > **Save As Template**.

To open a previously saved data display:

1. Choose **Window** > **Open Data Display** from the Main, Schematic, or Layout windows. In the dialog box that appears, the path is automatically set to the current project directory and the filter displays all saved graphs (*.dds).
2. Double-click the graph you want to open or select it and click **OK**.

For details on working with simulation data, refer to the *Data Display* manual.

Simulating

To simulate using current data set name:

Choose **Simulate** > **Simulate**. The Status window appears and displays messages about the simulation as it progresses. When the simulation is complete, it will display the message, *Simulation finished*.

To highlight a specific node on your schematic:

Choose **Simulate** > **Highlight Node**. The list box that appears displays a list of all nodes on the schematic. Select any node name from the list and that node is highlighted on the schematic. (The highlight color can be changed through *Options* > *Preferences* > *Display* > *Highlight*).

To clear one or more highlights, choose **Simulate** > **Clear Highlighting**.

Excluding Individual Items from the Simulation

You can *deactivate* items in the Schematic window, excluding them from a simulation without deleting them from your design.

To deactivate an item:

1. Click the **Deactivate** button on the toolbar or choose **Edit** > **Component** > **Deactivate**.
2. Click the item or items you want to deactivate. A box with an X drawn through it appears over each item.

Hint The color of this identifying box is the Highlight color defined through *Options > Preferences > Display.*

To activate a deactivated item:

1. Click the **Activate** button on the toolbar or choose **Edit** > **Item** > **Activate**.
2. Click the item or items you want to activate. The box disappears and theses items are considered in a subsequent simulation.

Clearing Highlights from Items Causing Simulation Errors

When an error occurs during simulation, a box is drawn around each item causing an error. To clear all highlights, choose the *Clear Highlighting* command from the View menu in the appropriate window.

Hint The color of this identifying box is the Highlight color defined through *Options > Preferences > Display.*

Tuning

Advanced Design System's tuning capability enables you to change one or more design parameter values and quickly see the effect on the output without resimulating the entire design. Multiple plots generated from various tuning trials can be overlaid in the Data Display window. This can help you find the best results and the most sensitive components or parameters more easily.

Basic tuning consists of the following steps:

1. Build the design you want to tune.
2. Simulate your design.
3. Set up the Data Display window to display the results you want from your design.
4. Choose **Simulate** > **Tuning** or click the **Tuning** icon (tuning fork) on the toolbar. When the initial analysis is complete, the Tune Control dialog box appears.

5. Select the parameter you want to tune by clicking it on the schematic.
6. Select one of the following tune analysis modes from the Perform Analysis drop-down list:
   - After each change
   - After pressing Tune
   - While slider is moving
7. Change the tunable parameter by moving the slider, clicking the left/right arrows, or typing the value in the box. Depending on the selected tuning mode, your results will be updated as soon as the new simulation is completed.

Chapter 9: Setting Design Environment Preferences

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

Some of the things you can customize are:

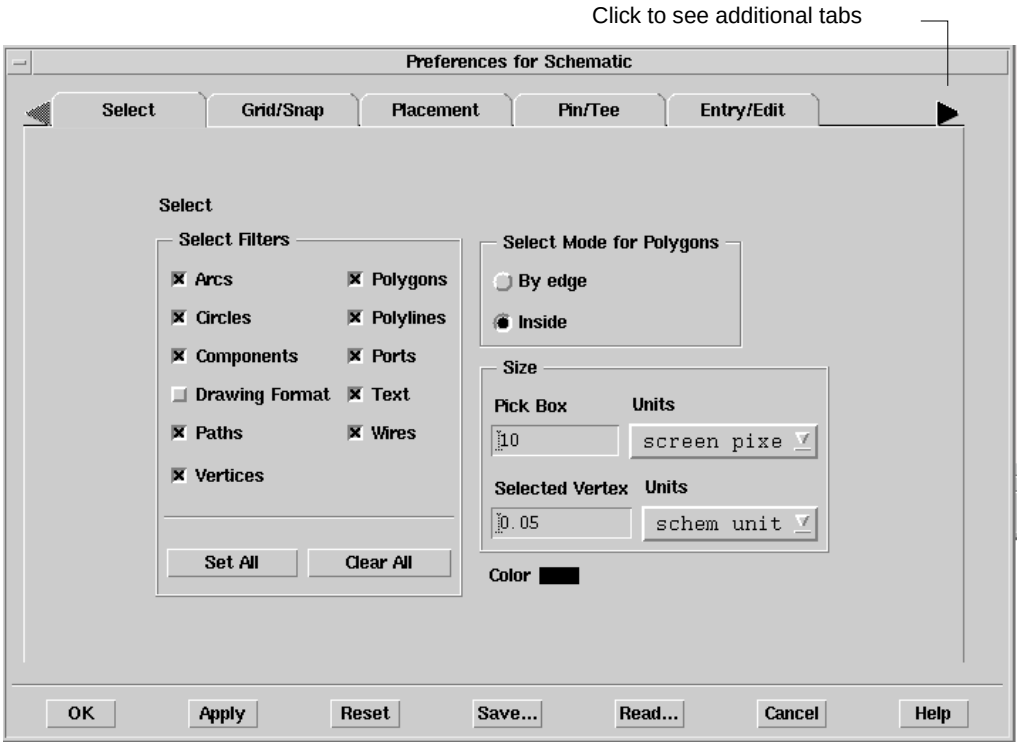
- The default settings for color of the grid, pins/tees, highlighted and selected objects, and the background and foreground of the design windows
- The sizes of pins/tees, the selection pick box, and the size of the marker drawn when you have the *Vertices* filter turned on and select the vertices of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

When you change the settings in this dialog box and click *Apply*, the design window is updated with the changes, and these changes will serve as defaults for all designs in this project. For details on saving changes to a file so that specific preferences can be associated with specific designs, refer to, "Saving and Reading Preference Files" on page 9-20.

Specifying Design Entry and Display Preferences

To change design entry and display preferences:

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



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

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

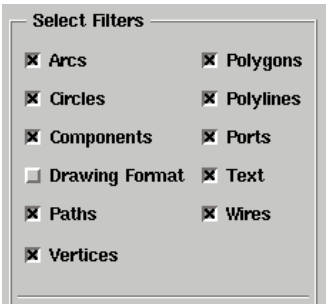
schem units—Use this setting to specify sizes in terms of inches, in the Schematic window. In the Layout window, select *layout units* to specify a size with respect to the design units of the Layout window.

Setting Select Options

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

Using Selection Filters

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



To change the default settings:

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

Hints:

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

Changing the Select Mode for Polygons

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

Changing the Pick Box Size

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

Figure 9-1 shows an example of the region defined by a pick box specified in inches.

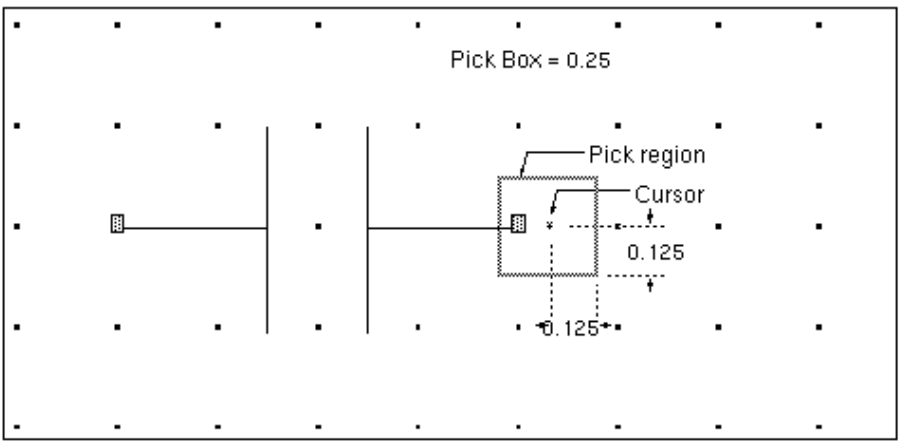
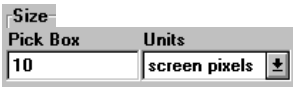


Figure 9-1. Pick region based on schem Units

In Figure 9-1, the pick box size was set to 0.25 inch (the same as the default grid display).

When you define the region using *n* screen pixels, the pick region is a square of *n* pixels × *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.

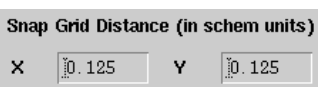
Hint The default component symbols have been created uniformly, in 0.125-inch increments. Thus, if you keep the default settings while creating your design, you should be able to connect all symbols with minimal effort.

Display

Dots	Select Dots or Lines to display a visible grid made up of dots or lines
Lines	
None	Select None if you want no visible grid
Color	Click to access a palette for choosing a grid color

Spacing

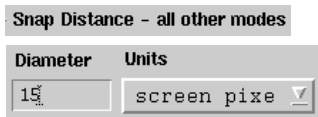
Snap Grid Distance (in schem units) represents the snap spacing, where the number you specify determines the distance (in design units, or inches) between the points to which the cursor snaps.



Snap Grid 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.)

Setting Pin/Tee Options

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

- **Pin Size**

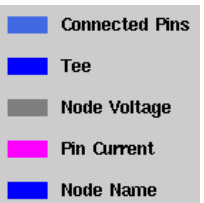
  Enables you to change the size of the pins drawn on all components.

- **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.

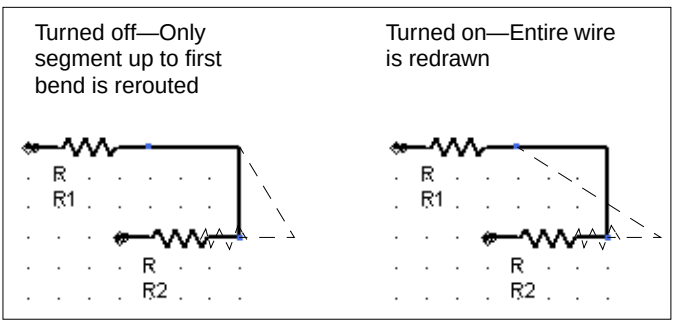
Setting Entry/Edit Options

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, refer to the section, "Saving and Reading Preference Files" on page 9-20.

- **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.

- **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**

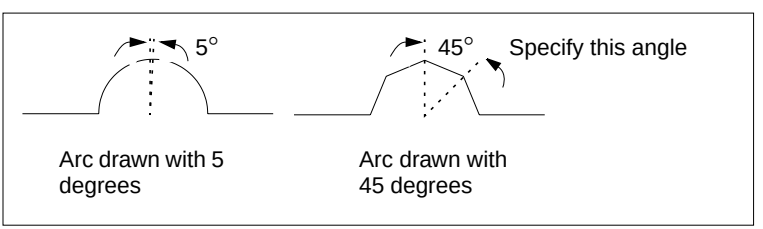
  Prevents you from placing additional points on a polygon if overlapping lines will result.

- **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**

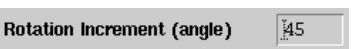
  Your file is automatically saved every time the number of edits you have performed reaches the number in this field.

- **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.

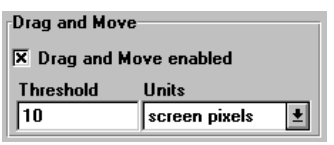
- **Rotation Increment (angle)**

  This option forces objects you rotate to snap—during rotation—in *n*-degree increments, where *n* is the number you specify here. For details on how this default is used, refer to the section "Rotating Items" on page 6-22.



- **Drag and Move**

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



Setting Component Text/Wire Label Options (in Advance)

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

There are two ways to change component text attributes:

- In advance of placing components, through the Options menu (*Options > Preferences > Component Text/Wire Label*). This setting serves as a default, but attributes of component text for an individual component, or all components, can be changed later through the Edit menu.
- After placing components, through the Edit menu (*Edit > Component > Component Text Attributes*). Refer to "Changing Component Text Attributes" on page 6-10.

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 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*.

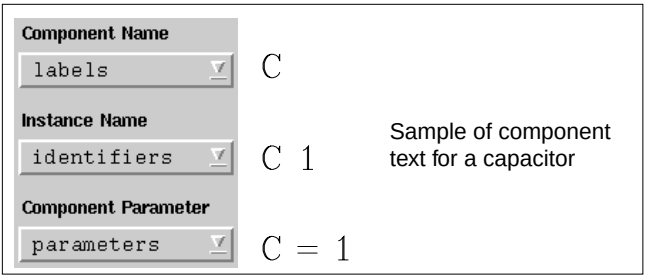
- **Point**

  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 the section, “Specifying Layer Definitions” on page 9-22.



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.

- **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 (refer to "Editing Existing Text and Text Attributes" on page 6-34).

To establish text attributes:

1. Choose **Options** > **Preferences** > **Text**. and set the text attributes as desired.

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

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

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

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

Setting Display Options

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

- **Foreground**

  The color of the lines making up polygons, polylines, and arcs while they are being drawn.

- **Background**

  The color of the drawing area background in the design window.

- **Highlight**

  The color used to identify problem items (with respect to simulation), orphaned items in schematic and layout representations, and unconnected pins.

- **Minimum Object Size To Display (in pixels)**

  The minimum size—in pixels—an object must be before it is actually drawn in the Schematic window. Objects smaller than this are not visible.

Setting Units/Scale Options

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

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

The Resistance setting can be changed to serve as a default for all resistors (subsequently) placed in this project. For details on how units/scale factors are handled by ADS simulators, refer to "Units/Scale Factors" on page 3-21.

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

- **Trace History**

  Number of traces to display—Set the default number of traces to be displayed on your plots

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

Saving and Reading Preference Files

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

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

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

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

To customize preferences while retaining the default filenames:

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

To create customized preference files with unique filenames:

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

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

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

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

Hint The variable that defines the search path for these files is PREFERENCES_PATH. For details refer to Appendix D, Customizing the ADS Environment

---

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. The Layer Editor dialog box enables you to:

- Change display characteristics on a layer-by-layer basis
- Define new layers
- Change the plotting priority of layers (Layout window)
- Turn the display of items on and off on any specified layer. For example, you can turn on and off the display of any or all the parts making up component text—Name (labels), ID (identifiers), Parameters (parameters).
- Protect items on any given layer from being selected, that is, when clicking in the drawing area to select items, those items on a protected layer cannot be selected as you edit your design. This is helpful when you have a lot of editing to do on certain kinds of items, but not others, in a crowded design.
- Save the layer definitions to the default *schematic.lay* file or to a unique *filename.lay* file

At any time during the modification of layer definitions, if you have not yet clicked *Apply*, you can click *Reset* to return to the previous settings.

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

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

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

The following figure shows the list of default defined layers. These are the default layers in the Schematic window; for details on layers in the Layout window, refer to the *Layout* manual.

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

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

To move a layer within the layer list:

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

To delete a layer:

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

---

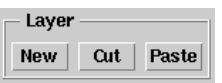
1. Click once to highlight the layer ID or Name.
2. Click **Cut**. The name and number are deleted from the list, and that number is now available for a new layer. (Hint: If you change your mind, use *Paste* to add the layer back.)

To change a layer's name:

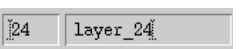
1. Click in the Name field and change the name as desired.
2. Click **Apply**.

To add a layer:

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



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



2. Rename the layer as desired and click **Apply**.

Setting Colors and Fill Patterns

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

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

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

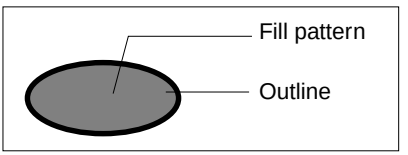
Note When choosing Fill Patterns, keep in mind that the patterns produced by a Postscript printer are Postscript fill patterns and will vary somewhat from those on your screen.

Setting Shape Display Characteristics Layer-by-Layer

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

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

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

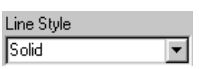


The Shape Display drop-down list for each layer enables you to choose the shape display of items, on a layer-by-layer basis.

Hint For details on changing the shape display globally for all layers, refer to the section, "Setting Layer Characteristics Globally" on page 9-28.

Setting Line Style Characteristics Layer-by-Layer

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



Hint For details on changing the line style globally for all layers, refer to the section, "Setting Layer Characteristics Globally" on page 9-28.

Setting the Visibility of Items Layer-by-Layer

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

Hint For details on changing the visibility globally for all layers, refer to the section, "Setting Layer Characteristics Globally" on page 9-28.

Setting the Selection Status of Items Layer-by-Layer

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

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

Hint For details on changing the shape display globally for all layers, refer to the section, "Setting Layer Characteristics Globally" on page 9-28.

Setting Layer Characteristics Globally

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

Miscellaneous Layer Editor Features

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

Saving and Reading Layer Files

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

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

To save customized layer files:

1. Make all the desired changes in the Layer Editor dialog box and click **Save**. The Save Layer File dialog box appears, displaying the default filename, *schematic.lay*.
2. To use the default filename for the current settings, click **OK**.

   For another layer file, enter a name of your choosing (the *.lay* extension is added automatically) and click **OK**.

To read in a previously saved layer file:

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

Hint The variable that defines the search path for these files is LAYERS_PATH. For details refer to Chapter 1, Customizing the ADS Environment, in the *Customization* manual.

Changing the Current Entry Layer

To change layers:

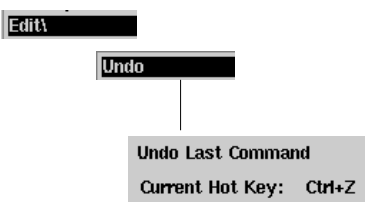
1. Choose **Insert** > **Entry Layer** and a dialog box appears listing all the currently defined layers.
2. Select a layer that is appropriate for the task at hand.
3. Click **OK**. Anything you draw now is drawn on this layer. The name of the layer is displayed in the status panel of the Schematic window.

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 **Options** > **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.

For a list of default keyboard shortcuts, refer to Appendix B, Shortcut Keys.

Configuring Toolbars

By default, the toolbar in each design window contains:

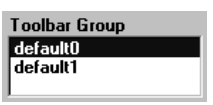
- A drop-down list for selecting the group of components you want to place on the palette (Palette List)
- A dynamically updated list of components you have placed in that window (Component History)
- In the Layout window, a drop-down list for selecting a different entry layer (Entry Layer List)
- A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

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

Customizing an Existing Toolbar

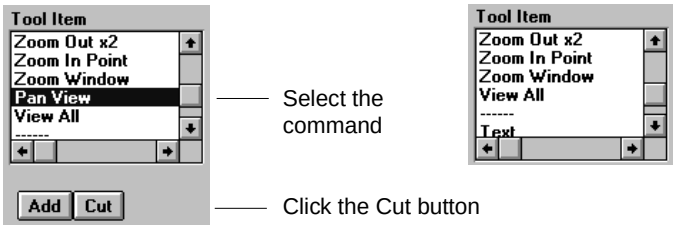
To reconfigure an existing toolbar:

1. Choose **Options** > **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.

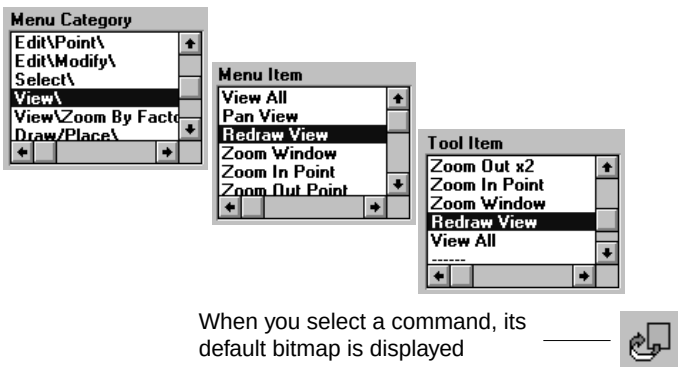


3. To add or delete icons, use one of the following methods:

- 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).

---

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

Creating a New Toolbar

To create a custom toolbar:

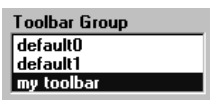
1. Choose **Options** > **Hot Key/Toolbar Configuration** and click the **Toolbar** tab in the dialog box that appears.

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

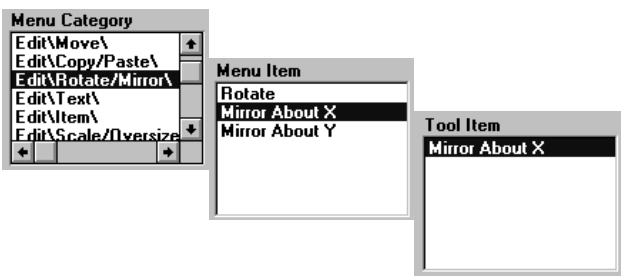
---

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

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



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



4. When you are through making changes to this toolbar, click **Apply** to effect the changes.

5. When you are through making changes to all toolbars, click **OK**.

Creating a Custom Component Palette

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

To create a custom palette:

1. Choose **View** > **Component** > **Create Component Palette** and a dialog box appears.

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

---

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

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

Chapter 10: Working with Symbols

Symbols are used to represent individual components in a schematic and subnetworks in hierarchical designs. Default symbols exist for schematics with 0 to 99 ports, but you might want to draw your own symbol for any of the following reasons:

- You want a symbol you feel better represents the schematic
- You want to replace the default symbol of a simulator component with one of your own
- You create a new simulator component (using the model builder) and need to create a schematic symbol for it

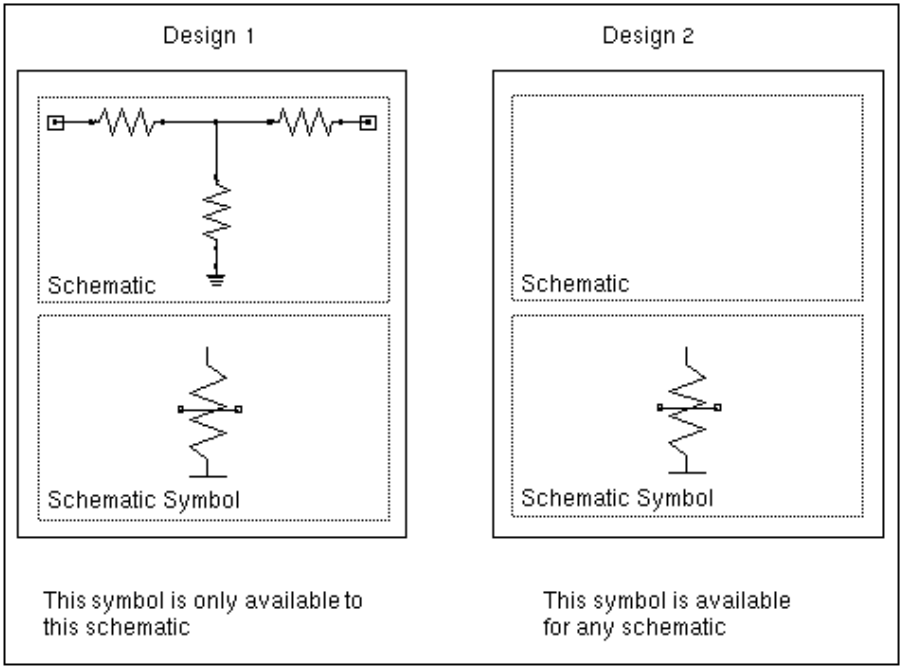
Schematic symbols consist of a symbol body and, optionally, symbol pins (interconnect points). If you are drawing a symbol to represent an electrical component or a subnetwork for a hierarchical design, you must add pins to the symbol body.

The default symbols have been created in a uniform manner. Custom symbols should be created in a similar manner, that is, their overall length, from pin to pin, should be a multiple of 0.125 inch. This ensures the custom symbol will connect easily to the set of supplied symbols. In addition, pin 1 should be located at the coordinates 0,0 which serve as the reference point when you place the symbol in a design.

A symbol can be defined in one of two ways:

- As part of a design file containing a schematic—when you need the symbol to represent only that schematic
- In a design file containing nothing but the symbol—when you want the symbol to be available to represent any schematic

An illustration representing these differences is shown next.



For information on assigning a symbol to a schematic, refer to the section "Assigning a Symbol to a Schematic" on page 10-15.

Switching Between Schematic and Symbol Views

When you are creating a schematic, the *View* menu displays the command *Create/Edit Schematic Symbol*. When you choose this command, you display the symbol view of the current design. If you look at the *View* menu again, you will see that the command name has changed to read *Create/Edit Schematic*. This enables you to switch back and forth between these views.

If you place an instance of your schematic design as a subnetwork—without creating a special symbol—a default symbol is used, based on the number of ports in your design. If you want to create a custom symbol for your design, you can switch to symbol view and:

- Accept the default generated symbol and then modify it
- Select one of the supplied symbols, which you can use as is or modify
- Click Cancel in the dialog box that appears and create your symbol from scratch

Refer to the following sections for details:

- "Generating a Symbol" on page 10-4
- "Using One of the Supplied Symbols" on page 10-5
- "Drawing a Custom Symbol" on page 10-6

Creating a Symbol for use with any Design

To create a symbol for use with any design:

1. Create a new (empty) design file.
2. Choose **View** > **Create/Edit Schematic Symbol**.
   - For details on how to copy and then modify one of the supplied symbols, refer to "Using One of the Supplied Symbols" on page 10-5
   - For details on creating a symbol from scratch, refer to "Drawing a Custom Symbol" on page 10-6
3. When the symbol is complete, you can associate any schematic with this symbol. For details refer to "Assigning a Symbol to a Schematic" on page 10-15.

Generating a Symbol

You can generate a symbol for your completed schematic providing minimal specifications.

To generate a symbol:

1. From the completed schematic, choose **View** > **Create/Edit Schematic Symbol**.
2. In the dialog box that appears, select the **Auto-Generate** tab and specify the desired symbol characteristics (or accept the defaults). The symbol characteristics are defined as follows:

   Symbol Type

   Dual—Restricts pins to two sides of symbol body

   Quad—Allows pins on all four sides of symbol body

   Order Pins by

   Location—Numbers symbol pins in the same relative order as the ports on the schematic

   Number—Numbers symbol pins sequentially in a left-right, top-down order

   Replace existing symbol—Replaces the current symbol, if one exists, with the one you are about to generate

   Lead Length—The length of any line drawn between the symbol body and a pin

   Distance Between Pins—Distance between pins drawn on the same side of the symbol body
3. Click **OK**. A symbol is drawn consisting of a symbol body, connecting lines, and pins.

Hint To regenerate the symbol specifying different symbol characteristics, choose *Insert > Generate Symbol*.

You can edit the generated symbol as desired at any time.

For details on drawing and editing shapes, refer to:

- "Drawing Shapes" on page 7-4
- "Editing Shapes" on page 6-26

For details on symbol-specific editing, refer to:

- "Establishing Pin Characteristics" on page 10-12
- "Positioning Parameters for Your Symbol" on page 10-14

Using One of the Supplied Symbols

You can use any of the symbols we supply and modify them or use them as is.

To associate a supplied symbol with your network:

1. From the completed schematic, choose **View** > **Create/Edit Schematic Symbol**.
2. In the dialog box that appears, select the **Copy/Modify** tab.
3. Select the desired symbol category from the Symbol Category drop-down list and the icons for that category are displayed. These categories represent the basic component libraries, but note that not all individual components will be displayed; if multiple components share the same symbol, only one of those components is displayed.
4. Click the desired symbol icon and its name is displayed in the Symbol name field.

   Alternatively, you can type a name directly in the Symbol name field:

   - Type an actual component name whose symbol you want to use
   - Type the name of a design file containing a symbol (or select it using the browser)
5. Click **Apply** to view the symbol in the design window and click **OK** when you are satisfied with your symbol selection. The program compares the number of pins on the symbol with the number of ports in the schematic view and reports any discrepancies.

You can edit the supplied symbol as desired at any time.

For details on drawing and editing shapes, refer to:

- "Drawing Shapes" on page 7-4
- "Editing Shapes" on page 6-26

Drawing a Custom Symbol

When you draw a custom symbol, you can draw it in a file containing a schematic, where it will be dedicated to that schematic, or you can draw it in an empty file and use it to represent any schematic.

Drawing Setup

Before you begin, you may find it helpful to review a number of the program's defaults, and you will most likely need to make a few changes.

Setting Snap and Grid Spacing

By displaying a grid and drawing with *snap* enabled, you can quickly draw objects with great accuracy. To view or change these settings, use *Options > Preferences > Grid/Snap*. Snap mode offers options for snapping to many different types of objects. These options are all described in the section, "Setting Grid/Snap Options" on page 9-6. When drawing a custom symbol, the most important option will probably be Grid, however if you are drawing complex shapes, several other options may be helpful.

The display grid can be made visible or invisible. When you turn on the grid display, you should specify a sufficiently large factor of the snap spacing so that a grid is displayed even when the snap spacing is very fine. For example, the default snap spacing (in the Schematic window) is 0.125 which means the cursor snaps every 1/8 inch. The default display factor is 2 which means that the dots appear every 1/4 inch (0.125 x 2).

Before you begin drawing, check to see if the current grid spacing is set to something other than the default of 0.125 inch, and if it is, change it back to the default and make sure *Enable Snap* is turned on. This step ensures that your custom symbol will easily connect to the set of supplied symbols.

Hint Once you have drawn the symbol body and pins, you can change these settings if desired. For example, you might want to turn snap off before adding text to your design to give you more flexibility in the positioning of the text.

Specifying the Drawing Layer

All shapes and text are entered on layers. The color and visibility of any shape is controlled by the layer on which it is drawn. Before you begin drawing any part of the symbol, change the current entry layer in accordance with the part you are about to draw. For example, if you are creating a custom symbol, you should draw the symbol body, pins, and lead lines on the *symbody* layer so that it will be on the same layer as the supplied symbols, and subject to changes you make to that layer.

To change layers:

1. Choose **Insert** > **Entry Layer** and a dialog box appears listing all the currently defined layers.
2. Select a layer that is appropriate for the task at hand.
3. Click **OK**. Anything you draw now is drawn on this layer. The name of the layer is displayed in the status panel of the Schematic window.

Table 10-1 lists the default layers and the types of items placed on those layers (by default).

Table 10-1. Layer Names, Numbers, and Descriptions

Layer name	Layer number	Description
default	0	Objects drawn on an undefined layer are plotted on this layer. Also, if you delete a layer, any objects residing on that layer are moved to this layer.
user_symbody	1	User-created schematic symbol
user_symtxt	2	User-created schematic symbol text
user_symbol	3	User-created schematic symbol
wire	4	Wires
text1	5	Available layer name for text
labels	6	Component or item names
identifiers	7	Unique ID of components or items
parameters	8	Parameters of components or items
pin_num	9	Pin numbers and names for symbols†
notes	10	Any additional descriptive information you want to add
symbody	11	Schematic symbol body outline
symdesign	12	Portion of schematic symbol contained within symbol body

Table 10-1. Layer Names, Numbers, and Descriptions (continued)

Layer name	Layer number	Description
SPint	13	Signal Processing integer data††
SPfix	14	Signal Processing fixed point data††
SPfloat	15	Signal Processing floating point data††
SPcomplex	16	Signal Processing complex data††
SPtimed	17	Signal Processing timed data††
SPany	18	Signal Processing any data type††
SPother	19	Signal Processing user-defined data††
symtext	20	Text contained within schematic symbol body
symbodyFilled	21	Schematic symbol body with fill pattern
symbodySlash	22	The slash that identifies pin 1

† The pin_num layer is visible by default, but you must turn on Pin Numbers and/or Pin Names in the Visibility (on/off) section of the Pin/Tee tab of the Preferences dialog box to make pin numbers and names appear.

†† For details, refer to Chapter 15, *Data Types for Model Builders*, in the Agilent Ptolemy Simulation manual.

Drawing the Symbol Body

The commands you need to draw your symbol can be found on the Insert menu. In addition, many of them are available on the default toolbar. For details on using these commands, refer to the section, "Drawing Shapes" on page 7-4.

Note The Layer Editor dialog box enables you to determine the color and fill pattern, shape display, and line style for objects drawn on each layer. For details, refer to the section, "Specifying Layer Definitions" on page 9-22.

Adding Pins to Your Symbol

There are two kinds of pins:

- Symbol Pin—A pin that represents a port of a network, that is needed to connect that network as a subnetwork in another design.
- Power Pin—A pin that also represents a port of a network, but does not appear in the schematic. The connection created via a power pin is an implied connection.

Symbol Pins

Remember that symbol pins should be located at 0.125-inch intervals so that your custom symbol will connect easily to the set of supplied symbols. In addition, pin 1 should be placed at the coordinates 0,0. Choose any of the methods shown below to accomplish this.

To specify 0,0 in the process of drawing pin 1:

1. Choose **Insert** > **Coordinate Entry** and a dialog box appears.
2. Choose **Insert** > **Symbol Pin** and a dialog box appears. Define the pin characteristics as described in the section, "Establishing Pin Characteristics" on page 10-12, and click **Apply**.
3. Enter the coordinates 0,0 in the Coordinate Entry dialog box and click **Apply**. Pin 1 is placed at 0,0. You can now use the *View All* command to reposition the image in your design window.

To move 0,0 to the center of the window before you begin:

1. Choose **Pan** from the View menu or the pop-up menu.
2. Click the small cross identifying the 0,0 coordinates (lower left corner of the drawing area). The 0,0 coordinates move to the center of your design window.
3. Choose **Insert** > **Symbol Pin** and a dialog box appears. Define the pin characteristics as described in the section, "Establishing Pin Characteristics" on page 10-12.

To move pin 1 to 0,0 after you have placed it:

1. Choose **Insert** > **Symbol Pin** and a dialog box appears. Define the pin characteristics as described in the section, "Establishing Pin Characteristics" on page 10-12, and click **Apply.**
2. Choose **Edit** > **Modify** > **Set Origin**.
3. Click the pin. The pin moves to 0,0. You can now use the *View All* command to reposition the image in your design window.

Power Pins

Power pins, also called *inherited pins*, are added in symbol view, but do not appear in the schematic. They create an implied connection. This connection is inherited from subnetworks throughout the hierarchy.

Note There is no support for power pins in the IFF translator in the current release.

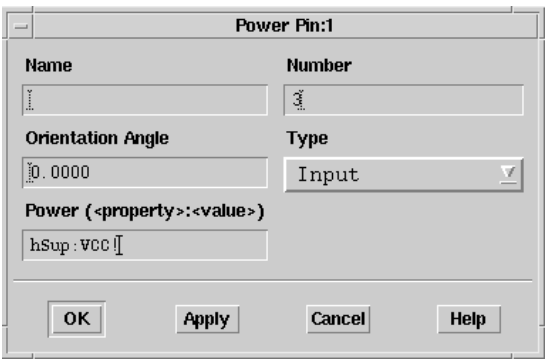
To add a power pin to your symbol:

1. From the subnetwork design of interest, choose **View** > **Create/Edit Schematic Symbol**.
2. If a symbol does not yet exist for the current design, the Symbol Generator appears. Set the options as desired (refer to "Generating a Symbol" on page 10-4) and click **OK**. A symbol is generated that includes one pin for each port of the network.

   If a symbol already exists for the current design, that symbol appears.
3. Delete the symbol pin you want to replace with a power pin.
4. Choose **Insert** > **Power Pin**.

Note The numbering sequence for all pins of a given design must follow this rule: the symbol pin numbering sequence comes first, and the power pin numbering sequence follows.

5. In the dialog box that appears, set the pin characteristics as needed (refer to "Establishing Pin Characteristics" on page 10-12), including renumbering, where applicable, and click **Apply**.



6. Position the pointer in the drawing area and click to position the power pin.
7. Define additional power pins for this design as needed, and click **Cancel** to close the dialog box.

To override the default value of a power pin for a given instance:

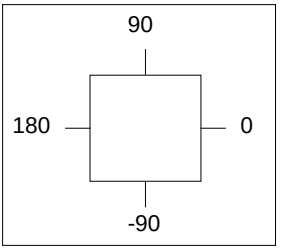
1. Select the instance and choose **Edit** > **Properties**.
2. If the instance has more than one property, select the desired property. All power properties are inherited from instances throughout the hierarchy. All properties of this type will be identified as *inherited*.
3. Supply the desired Value (the name of the node or pin to which the *invisible* power pin should be connected).

4. Retain the default Value Type setting of *String* and click **OK**.

Establishing Pin Characteristics

The pin characteristics are defined as follows:

- *Name*—Optional. The pin name is not displayed on your schematic unless you select the *Pin Names* option in *Options* > *Preferences* > *Pin/Tee*.
- *Number*—Optional. A default pin number is displayed. This number is automatically incremented as each pin is added. To display pin numbers on the schematic, select *Pin Numbers* in *Options* > *Preferences* > *Pin/Tee*.
- *Orientation Angle*—Optional. This feature is only used by the program when generating a schematic from a layout. It enables you to specify an orientation angle for each pin that determines the orientation of an item connected *directly* to that pin. Any angle from –90 to 180 degrees is allowed, but we recommend 90-degree increments. The recommended angles for schematic symbol pins are shown in the following illustration.



- *Type*
  - *Input*—Identifies the pin as an input port
  - *Output*—Identifies the pin as an output port
  - *Input/Output*—Identifies the pin as an input or output port
- *Power (<property>:<value>)*—Power pins only. Supply a *property* or name (a unique identifier) and a default value, such as the name of a global node contained in the hierarchy, separated by a colon.

To edit existing pin characteristics:

1. From symbol view (*View* > *Create/Edit Schematic Symbol*), choose **Edit** > **Symbol Pin** or for power pins, **Edit** > **Power Pin**.
2. Click the pin of interest.
3. Change the settings as desired and click **OK**.

Note When you create a symbol for use with any design, it is only available in the project directory in which you create it, unless you modify certain configuration files (and for model builder usage, modify AEL files according to stated guidelines). If you want the symbol to be available for all design work, refer to the section, "Making Symbols Available Globally" on page 10-16.

Positioning Parameters for Your Symbol

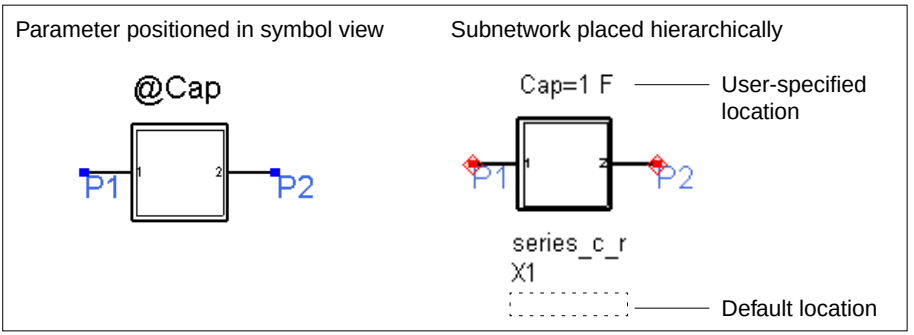
By default, when you define parameters for your subnetwork (*File > Design Parameters*), those parameters are positioned in the same manner as the parameters of supplied components. You can position these parameters by adding references to them using the onscreen editor in symbol view.

To position user-defined parameters for a subnetwork symbol:

1. Open the design for which you want to customize parameter positions.
2. In symbol view (*Edit > Create/Edit Schematic Symbol*), choose **Insert** > **Text** (or click the Text icon on the toolbar).
3. Position the cursor where you want a given parameter to appear (relative to the symbol) and click once.
4. Type the parameter name—as defined in the Design Parameters dialog box—beginning with the *at* symbol (@). For example, if your subnetwork has a parameter *width*, type *@width*. (Parameters names are case sensitive on UNIX.)
5. To end the text string, do one of the following:
   - Move the cursor away from the text and click once. (That string is done, but the Text command is still in effect and you can click in a new location to add another string.)
   - Choose *End Command*. (That string is done and the Text command ends.)

Hint When using the onscreen editor to position individual parameters, do not press *Enter* to move from one parameter to another; you must treat each parameter as an individual text string.

When you place the subnetwork hierarchically, the parameters you positioned in symbol view appear in the same position, relative to the network symbol.



User-defined parameters positioned in this manner:

- Take on the color of the layer on which the @<*parameter_name*> string was placed
- Take on the font and size defined for other parameters (in advance, through *Options* > *Preferences* > *Component Text/Wire Label* or with editing, through *Edit* > *Component* > *Component Text Attributes*)

Assigning a Symbol to a Schematic

Symbols are stored in design (*.dsn*) files. If you generate a symbol for a particular schematic, or create one in a design file containing a schematic, then that symbol is stored in the same design file as the schematic and that symbol is automatically associated with that schematic, unless you explicitly specify the name of another symbol (design file). You can verify the symbol being used for any given schematic by opening that schematic and choosing *File* > *Design Parameters*, clicking the *General* tab and noting the name in the *Symbol Name* field. If it reflects the current design name (minus the *.dsn* extension), then the program will look in this file for the symbol.

If you create a symbol in its own design file—which makes it available to represent any schematic—you must assign that symbol to the desired schematic.

To assign a symbol to the schematic:

1. Open the schematic design.
2. Choose **File** > **Design Parameters** and the Design Definition dialog box appears.

3. In the Symbol Name field, type the name of the file (minus the *.dsn* extension) that contains the desired symbol.

Hint The drop-down list of symbols can be modified to include any supplied symbols as well as custom symbols. For details refer to the section, *Modifying the List of Available Symbol Names* in the *Customization* manual.

---

4. Click **OK**.

Hint If you generate a symbol and then decide to use a custom one instead, after you type the symbol filename in the *Symbol Name* field, click the *Save AEL* button in the dialog box to save the change.

---

5. Choose **File** > **Save Design**.

The next time you place this design within another design, the new symbol will appear.

Making Symbols Available Globally

Custom symbols are only available in the project directory where you create them unless you modify the search path of a particular variable in a configuration file. In addition, if you want the names of these symbols to appear for selection within the related dialog box, you must modify a certain AEL file.

While it is possible to leave all files containing custom symbols in the project directories in which they were created, the aforementioned configuration changes will be greatly simplified if you move all custom symbols to central locations, one location for individual use and one for site use. (If necessary, see your system administrator to create the directory for site use.) We recommend maintaining the same directory structure used by Advanced Design System. Thus, the symbols should be placed in the following locations, in accordance with their use (*circuit* vs. *hptolemy*):

Use these directories on UNIX:

UNIX	
Individual Use	$HOME/hpeesof/circuit/symbols $HOME/hpeesof/hptolemy/symbols
Site Use	$HPEESOF_DIR/custom/circuit/symbols $HPEESOF_DIR/custom/hptolemy/symbols

Use these directories on the PC:

PC*	
Individual Use	*%HOME%/hpeesof/circuit/symbols* *%HOME%/hpeesof/hptolemy/symbols*
Site Use	*%HPEESOF_DIR%/custom/circuit/symbols* *%HPEESOF_DIR%/custom/hptolemy/symbols*
* *%HOME%* represents the path you specified as the *Home Folder* during installation (*C:\users\default* by default); *%HPEESOF_DIR%* represents the path you specified as your *Program Folder* during installation (*C:\ADS2002* by default).	

Note If you use a directory other than one of those shown in the tables, then you must declare the variable USR_DSN_PATH and provide the search path.

To access custom symbols—if stored in a directory other than one of the defaults:

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

Once you have created the symbols directory (regardless of where), move the files containing the custom symbols from their current project directory locations into the appropriate, newly created directory.

Modifying Search Paths

Search paths that control the order of directories searched are defined by certain program variables. These variables should be modified in a local copy of the *de_sim.cfg* file. This file is written to the */config* directory during installation and should not be modified there. But you can open this file and copy the default variable definition (where applicable), then paste it in your own local copy and modify it. (This simplifies typing a lengthy search path.)

The program creates a *de_sim.cfg* file automatically in *$HOME/hpeesof/config*. Settings modified here apply to all projects.

To modify the search path of the variable:

1. Using any text editor, open the **de_sim.cfg** file in $*HPEESOF_DIR/config*.
2. Locate the variable definition, copy it, and close the file.
3. Open the *de_sim.cfg* file in your *$HOME/hpeesof/config* directory and paste the variable definition.
4. Edit the variable definition as needed.
5. Save the file.

Chapter 11: Printing and Plotting

This chapter describes printing and plotting from Advanced Design System. You can send output to printers and plotters as well as to file in a variety of formats. When printing to file, the format of the file is determined by the current output device and the file is saved in the current project directory.

You can connect any output device that is supported by your operating system. To connect additional printers and plotters, and select a default printer, choose the appropriate method for your platform:

- Windows NT—Print Manager
- Windows 98 and 2000—Start menu > Settings > Printers
- UNIX—Choose File > Print Setup > Install > Add Printer

The basic Print commands are summarized next:

- Use the *Print* command to print the contents of the drawing area of the current window
- Use the *Print Area* command to print a region of the drawing area
- Use the *Print Setup* command to establish a default printing configuration, although you can modify it at the time of printing

---

Note For detailed information on printing and print setup options on the PC, refer to your Windows documentation.

Because the print methods vary significantly between UNIX and the PC, this chapter describes printing from these platforms separately. Refer to the appropriate section for your platform:

- "Printing from UNIX" on page 11-2
- "Printing from the PC" on page 11-12

Printing from UNIX

Printing and plotting from Advanced Design System on UNIX is accomplished by establishing the desired print setup and then choosing *File > Print*. The Print Setup and related dialog boxes enable you to:

- Choose to send output to a printer/plotter or print to file
- Install additional printers
- Select a printer (if sending to printer) other than the default
- Select a file format (if printing to file)
- Select Portrait or Landscape orientation
- Scale the output
- Specify the number of copies

When you select a printer, you can change the following default printer-specific options:

- Resolution
- Page Size
- Paper Tray

When you choose *File > Print*, you can select from the following additional options:

Convert to HP-GL/2 file	Select this option to print to file using the HP-GL/2 format.
Color output	Select this option to print in color (on a color printer) or in grayscale, rather than black and white (on a monochrome printer)
Scale to fit page	Select this option to fill the printed page. If selected, this option overrides any scaling factor you have set.

Hint Click *Options* in the Print dialog box as a shortcut back to the Print Setup dialog box if you decide to make changes to the current setup.

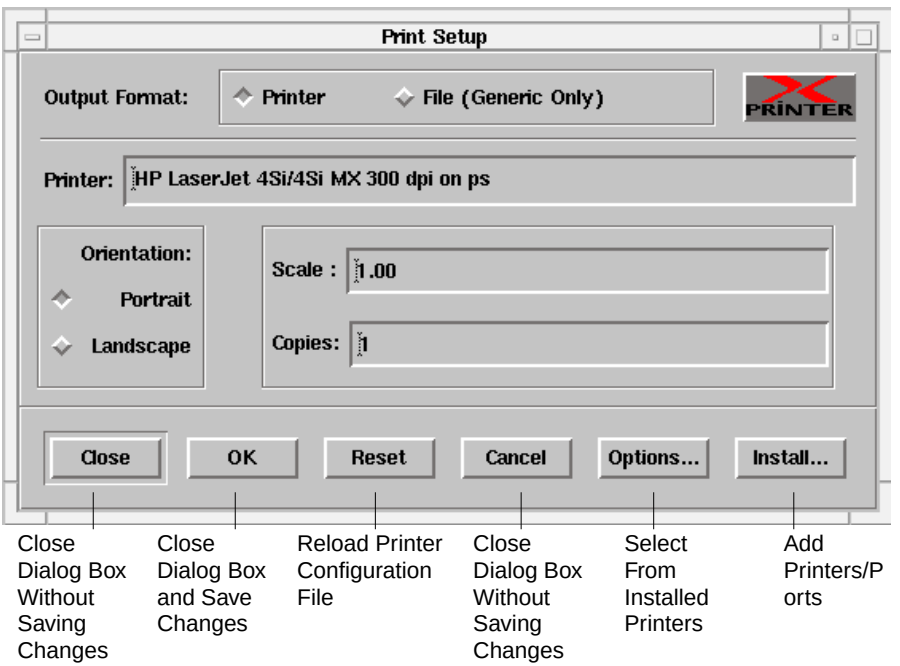
Your print setup is saved in *$HOME/.Xprinterdefaults*. If you do not have a local copy of this file, or the file *.Xpdefaults* (from a previous release), the default file is read from the *$HPEESOF_DIR/xprinter* directory. When you change your print setup, the changes are saved (as new defaults) to *$HOME/.Xprinterdefaults*. Note: If you do have a file *.Xpdefaults* (from a previous release), the settings of this file are copied to the new filename to serve as the starting point for your print setup. Both files are valid, depending on which release of ADS you are using. The old file is maintained for running an earlier version of ADS, but the new file is used when you run ADS 1.5 (or later).

Adding a Printer

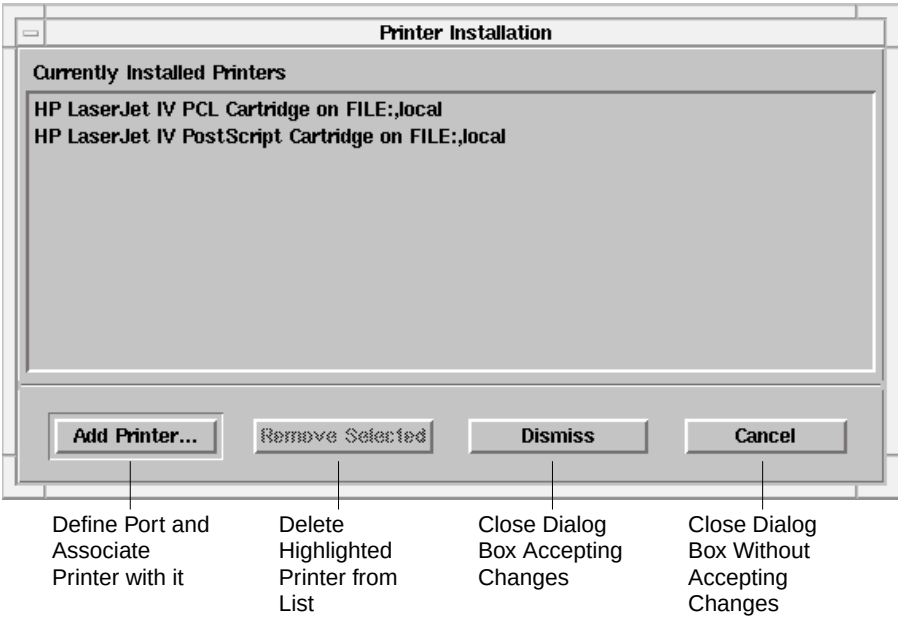
The basic steps required for adding a printer through the Print Setup dialog box are: defining a port and associating a printer with that port.

To define a port and add a printer:

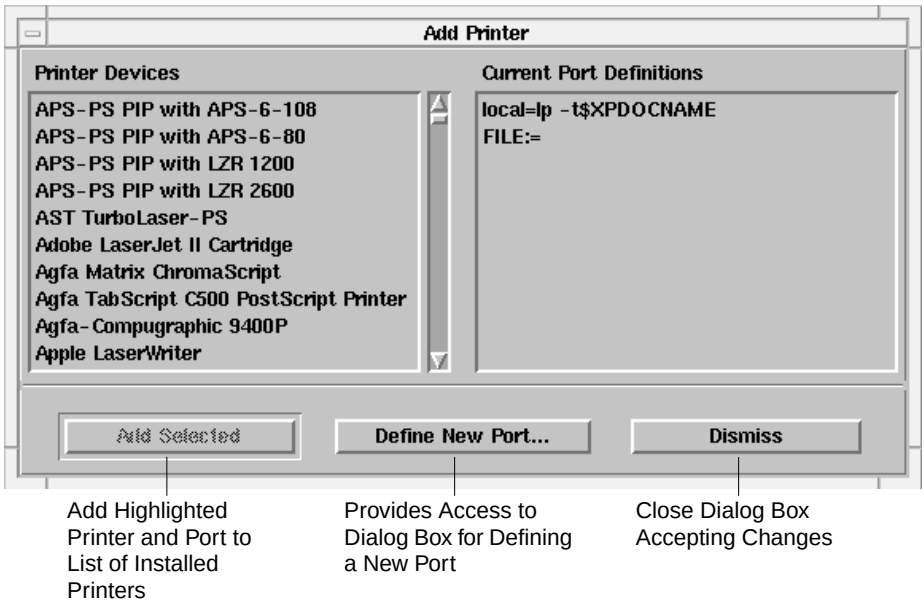
1. Choose **File** > **Print Setup** and a dialog box appears.



2. Click **Install** and a dialog box appears listing all currently installed printers.



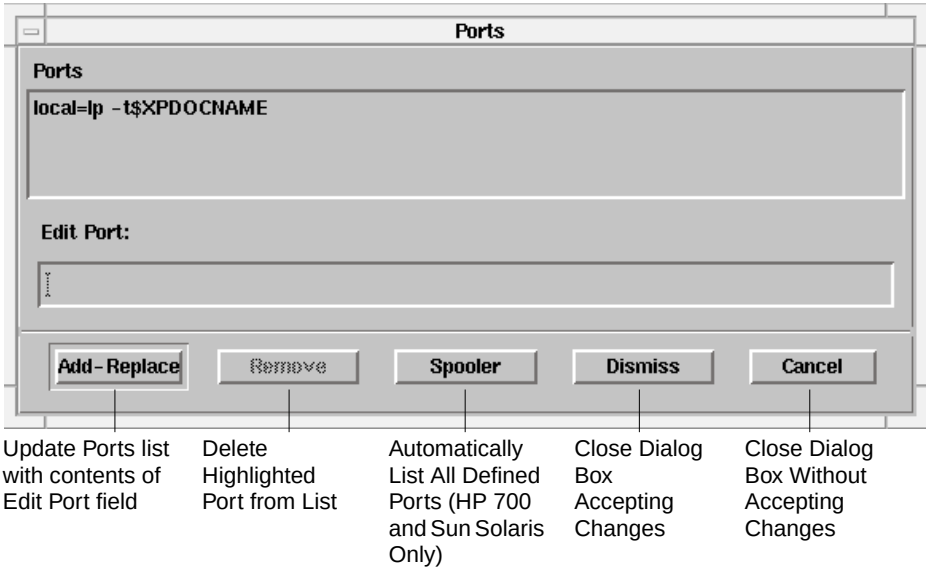
3. Click **Add Printer** and a dialog box appears listing all available printer devices and all currently defined ports.



Note For a list of supported printers, see *Supported_Printers_XPV331.html* in *$HPEESOF_DIR/xprinter*.

---

4. Click **Define New Port** and a dialog box appears listing all currently defined ports.

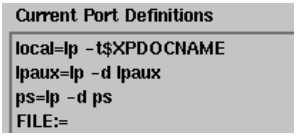


5. Add all ports you want to access for printing:
   - On HP 700 and Sun Solaris workstations, click **Spooler** and the list of ports is automatically generated (based on your *printcap* file).
   - On all other workstations, type the port definition in the Edit Port field using the following syntax: *printer name=print command* (no spaces around equal sign), where *print command* is the print alias, just as you would type it in the terminal window. Click **Add-Replace**. Repeat for each desired port.

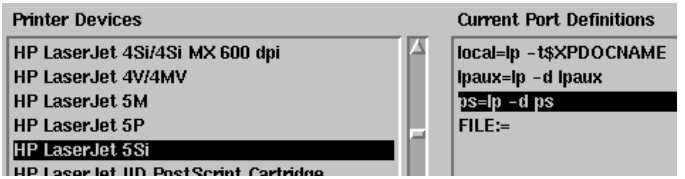
Note Port names can be any names you choose with the exception of *FILE:* which is a reserved port name.

---

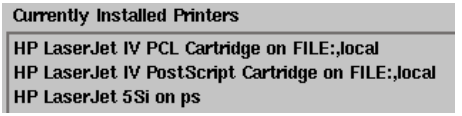
6. Click **Dismiss** to accept the new port definitions and return to the Add Printer dialog box. The Current Port Definitions list box is updated.



7. Select the desired printer from the list of Printer Devices.
8. Select the port you want to associate with this printer.



9. Click **Add Selected**. The Printer Installation dialog box is updated.



If you defined multiple ports, you can associate a printer with each, as just described.

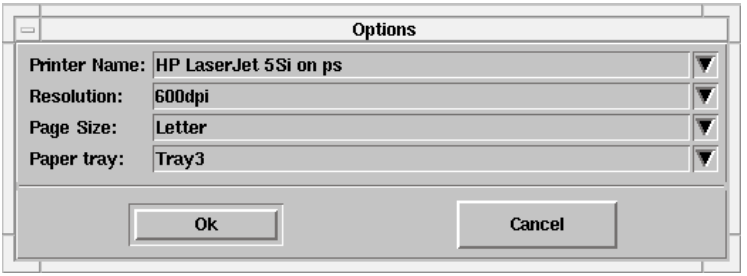
10. **Dismiss** the Add Printer and Printer Installation dialog boxes. You will now be able to select any of the installed printers, as needed.

Selecting a Printer

To select a printer other than the current default:

1. Choose **File** > **Print Setup**.
2. Select **Printer** as the Output Format in the Print Setup dialog box.

3. Click **Options** and a dialog box appears.



4. Select the desired printer from the Printer Name drop-down list. The printer-specific options are updated to reflect the default options for the selected printer.
5. Change any or all of these options as needed and click **OK**.
6. Set the following options, in the Print Setup dialog box, as desired for the current printer:
   - Orientation
   - Scale (the value 2.0 would double the size; 0.5 would reduce it by half)
   - Copies

Hint Like all other information in the setup-related dialog boxes, settings made here become the new defaults for this printer.

---

7. Click **OK** and the current printer configuration information is saved.

Sending Output to the Printer

To send the entire contents of the window to the printer:

1. Choose **File** > **Print** and a dialog box appears.
2. Select any print options as needed, and click **OK**.

Hint The *Scale to fit page* option will fill the printed page, regardless of the current value in the Scale field of the Print Setup dialog box

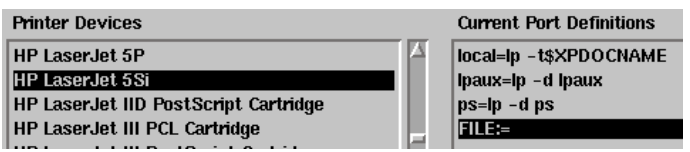
---

Creating a Printer-specific Print File

You can create a print file in the format used by a particular printer by associating that printer with the *FILE:* port. This port appears by default in the Add Printer dialog box. Once you make the association, you can select this printer as the current printer and create a print file for it.

To associate the desired printer with the *FILE:* port:

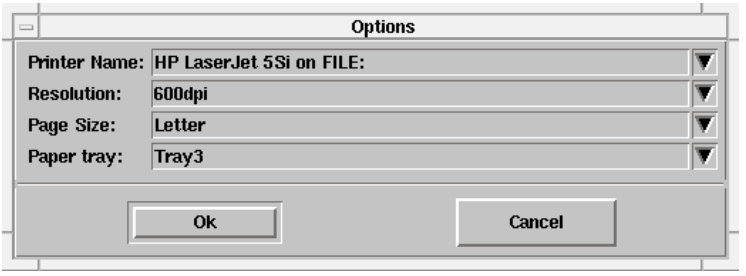
1. Choose **File** > **Print Setup**.
2. Click **Install** > **Add Printer**.
3. Select the desired printer and the **FILE:=** port definition.



4. Click **Add Selected.** The Printer Installation dialog box is updated.
5. **Dismiss** the Add Printer and Printer Installation dialog boxes.

To make this printer the current printer:

1. Click **Options** and select the printer associated with *FILE:* from the Printer Name drop-down list.



2. Change any options as needed and click **OK**.
3. Click **OK** in the Print Setup dialog box.

To create the print file:

1. Choose **File** > **Print**.
2. Select any print options as needed, and click **OK**. A dialog box appears prompting you for a filename.
3. Change directories if desired—the file is written to the current project by default—and supply a name in the *Output To File* field. Click **OK**. The file is written to the specified directory.

Printing to File in a Generic Format

You can send your output to file, in a limited number of generic formats, which can then be imported in a variety of applications. The available formats are:

- Encapsulated Postscript®
- PCL4
- PCL5

Note You have the option of converting to an HPGL/2 file when you choose *File* > *Print*.

To establish a print setup for printing to file:

1. Choose **File** > **Print Setup**.
2. Select **File (Generic Only)** as the Output Format.
3. Select a different file type from the drop-down list as needed. The default File Name is updated to reflect the selected file type.

4. Set the following options as desired:
   - Orientation
   - Scale
   - Copies
5. Click **OK** in the Print Setup dialog box.

To print to file:

1. Choose **File** > **Print** and a dialog box appears offering additional options.
2. Select any or all of these options, as needed, and click **OK**.
3. The file is written to the current project directory, using the filename from the Print Setup dialog box.

Printing from the PC

This section describes some of the actual printing features available on the PC. For information on basics (such as adding a printer), refer to your Windows documentation.

Printing from Advanced Design System on the PC is accomplished by establishing the desired print setup and then choosing *File* > *Print* (or *Print Area*). Listed below are some of the more common options you can set through the Print Setup and related dialog boxes:

Note The options available vary based on the printer/printer driver you select.

- Printer (select any installed printer)
- Paper size and source
- Orientation (Landscape is generally recommended for schematics and layouts)
- Number of copies
- Single- or two-sided printing
- Scaling

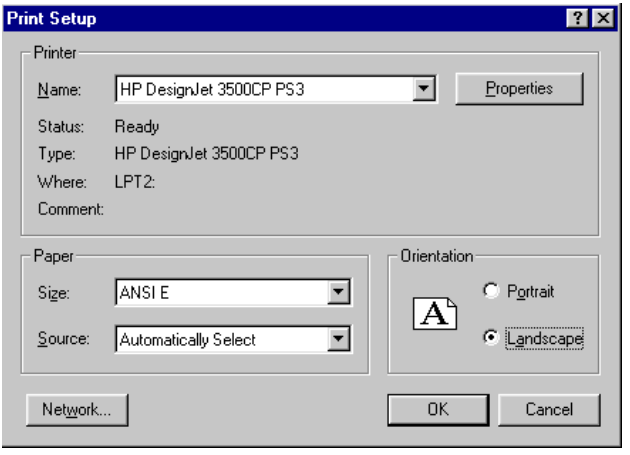
When you choose *File* > *Print*, you can select from the following additional options:

☑ Print to file	Select this option to send output to file for printing at a later time. Select Enhanced Metafile, Windows Metafile, or HP-GL/2 as the file format when the Print to File dialog box appears.
☑ Color output	Select this option to print in color (on a color printer) or in grayscale, rather than black and white (on a monochrome printer).
☑ Copy to clipboard	Select this option to place the image on the Windows clipboard (Bitmap) for pasting in any Windows application.
☑ Fit to page	Select this option to fill the printed page. If selected, this option overrides any scaling percentage you have set.

Establishing a Print Setup

To establish a print setup:

1. Choose **File** > **Print Setup**.
2. Select the desired printer from the drop-down list.



3. Change any of the options here as desired, or click **Properties** to set additional options, such as *Scaling*. Note that the appearance of the Properties dialog box varies depending on the selected printer.
4. Change any other options as desired and click **OK** to dismiss the Properties dialog box.
5. Click **OK** in the Print Setup dialog box and you are ready to print.

Basic Printing

To send the entire contents of the window to the printer:

1. Choose **File** > **Print** and a dialog box appears.

Note The *Fit to page* option overrides any scaling percentage set. Disable this option if you have intentionally set a scaling percentage.

---

2. Change any print options as needed, and click **OK**.

To print a specific region of your schematic or layout:

1. Choose **File** > **Print Area**. As you move your pointer into the drawing area, it changes to a cross-hair cursor, and the Status panel prompt changes to read, *PrintArea: Enter the starting point*.
2. Position the pointer at one corner of a window that will enclose the area you want to print, and click. The Status panel prompt changes to read, *PrintArea: Enter the next point*.
3. As you move the mouse, a ghost image of the window is drawn. Click again to specify the opposite corner of the window and the Print dialog box appears.
4. Change any options as needed and click **OK** to print the specified region.

Printing a Scaled Layout

To scale a layout for printing:

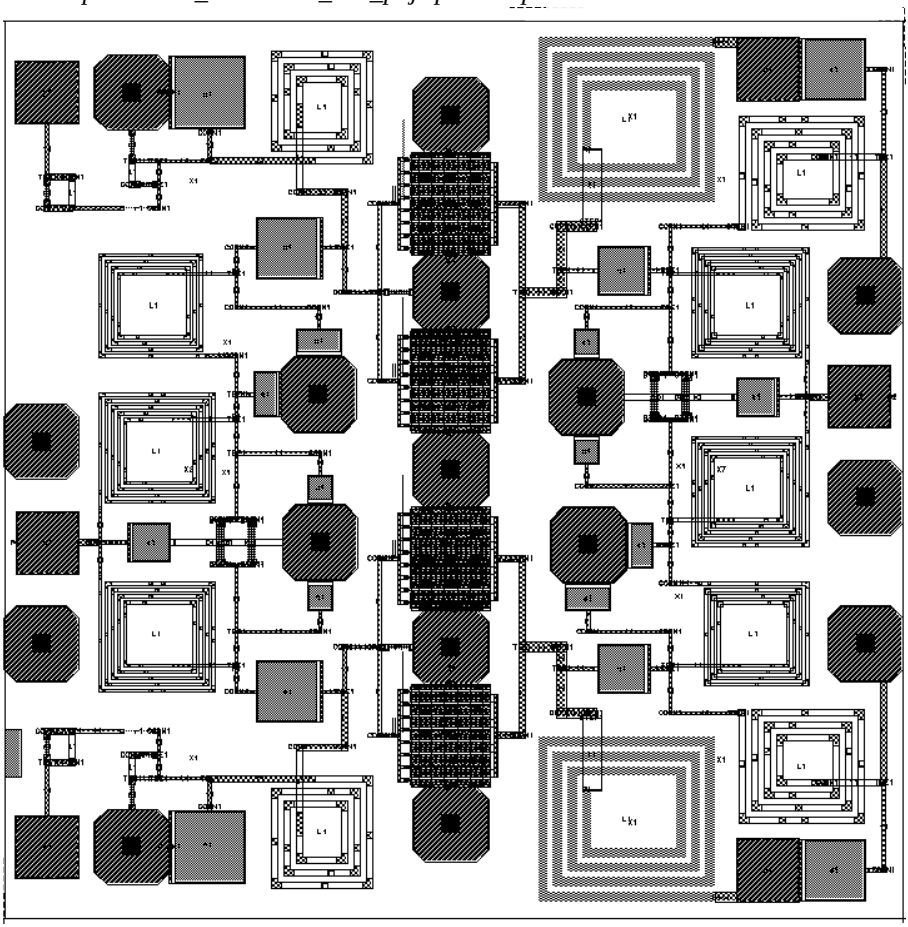
Note This process involves flattening and, potentially, scaling your layout in ADS. You can do this to your actual design—and then close it without saving changes—or make a copy of it before printing and make the changes to the copy.

---

1. Choose **File** > **Save Design As** to make a copy of the layout.
2. Choose **Edit** > **Component** > **Flatten** to convert it to shapes-only information. (Components are not scalable.)
3. Choose **Insert** > **Measure** and establish the size of the design.
4. Determine the appropriate scaling factor based on the actual size of the design and the paper size you want to use.

   Note The ability to scale a design for printing is a function of the printer driver associated with the selected printer. In general, PostScript printer drivers enable scaling while PCL printer drivers do not. Many printer drivers can be downloaded from the Internet. If your printer driver does not allow a sufficiently large scaling factor, you can also scale the design in ADS to reach a combined scale factor appropriate for your layout.

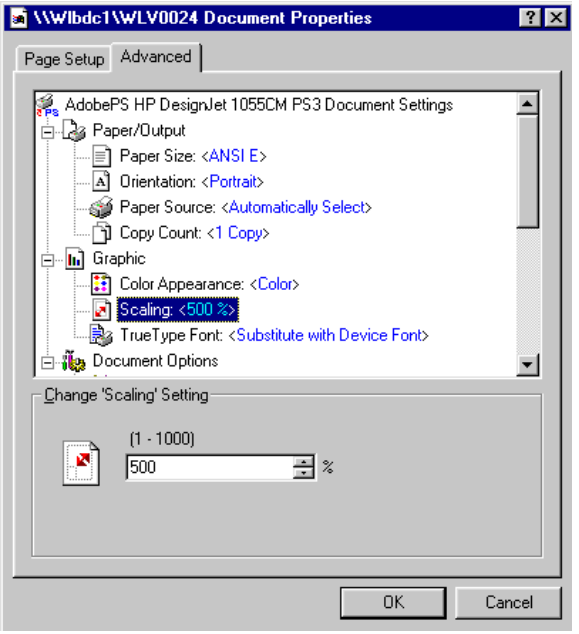
The example shown here is based on a design from the examples directory: */examples/MW_Ckts/drc_via_prj/pwramp*.



This design measures 1525 microns (or 0.1525 cm) across, and is close enough to being square to use that measurement as the basis for determining a scaling factor. In this example we want to basically fill ANSI E paper (34" x 44") and so we convert the metric units to English units 0.1525 cm x 0.39370 = ~0.06 inches. To simplify this example, we settle on a 30 inch image, yielding a required scaling factor of 500 (30 / 0.06 = 500).

5. To verify the maximum scaling factor your printer driver allows, choose **File** > **Print Setup** and click **Properties**.

6. Locate the *Scaling* option. It may appear on the tab displayed by default, or in an *Advanced* tab, as shown in this example.



7. Select the **Scaling** option and supply the desired scaling percentage in the field provided.

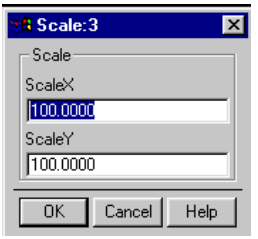
   In this example, we opted for 500% (factor of 5) scaling through the printer driver, and a scale factor of 100 (for a total of 500) through ADS.

8. Select the **Paper Size** option and select the appropriate size.

9. Change any other options as desired and click **OK** to dismiss the Properties dialog box.

10. Click **OK** in the Print Setup dialog box.

11. If necessary, scale the design in ADS. Choose **Select** > **Select All**., then choose **Edit** > **Scale/Oversize** > **Scale**. Supply the desired scaling factor, in this example, 100, and click **OK**.



12. Choose **View** > **View All**.
13. Choose **File** > **Print Area** and draw a border around the layout. (This is to ensure best results, since printing otherwise begins at the coordinates 0,0 and may include empty space.)
14. When the Print dialog box appears, select any other options as desired, such as *Color Output* if your printer is a color printer.

Note The *Fit to page* option overrides any scaling percentage set. Disable this option if you have intentionally set a scaling percentage.

Chapter 12: Using the Text Editor

Hpeesofeedit is a general-purpose ASCII text editor capable of editing one file at a time. It enables you to cut and paste, search for text, replace text, and go to a specific line of text. You can access Hpeesofeedit from the Options menu in the Main window or you can start it from any command shell. The topics described here are:

Starting the Text Editor Program

To start the text editor program:

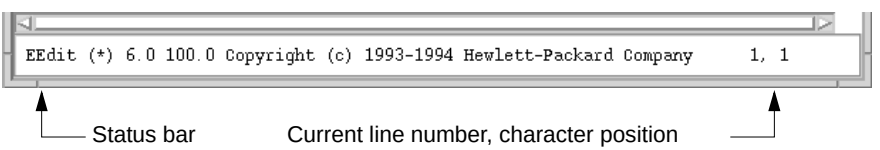
- Choose **Options** > **Text Editor** in the Main window.

  or

- Type **hpeesofeedit** in any command shell, and press **Return**.

The Hpeesofeedit window appears.

The status bar displays warnings, the status of the search/replace operation, and the current line number and character position. The copyright message is displayed in the status bar until you begin typing. The text entry portion of the edit window displays all text you type or paste into the window.



Command Line Options

The Hpeesofeedit program accepts most standard X/Xt command line options. These options are listed in Table 12-1. (For further details on these options, refer to your UNIX documentation.)

Table 12-1. Command line options

Option	Argument	Example
-background, -bg	color_name	-bg white
-foreground, -fg	color_name	-fg blue
-display	display_name	-display unix:0
-geometry	geometry string	-geometry 300x400+10+30
-iconic		
-xrm	X-resource	-xrm "*background:blue"

To start the Hpeesofeedit program with a command line option:

In any command shell, type **hpeesofeedit** <*desired option*> and press **Return**.

Alternatively, any of these resources can be specified in your resource file using *Hpeesofeedit* as the class name. *Hpeesofeedit* is also the resource filename for the program.

Note Any option not preceded by a dash (-) is treated as a filename and the program will attempt to open a file by this name. If a file by this name does not exist, a new file is created.

Text File Management

Basic file operations include creating new files, opening existing files, inserting existing files into other files, and printing and saving files.

Creating a Text File

To crete a new text file:

1. Select **File** > **New**. If there is currently any text in the window, you are prompted *Text exists, clear?*
   - To avoid clearing the text in the window, click **No** and then take the desired action.
   - To clear it and start a new file, click **Yes.**
2. When the window is empty and labeled *untitled*, type the desired text. (You assign a name to the file when you save it.)

Opening an Existing File

To open an existing file:

1. Select **File** > **Open**. If there is currently any text in the window, you are prompted *Text exists, clear?*
   - To avoid clearing the text in the window, click **No** and then take the desired action**.**
   - To clear it and open a file, click **Yes,** and a dialog box appears.

   The default file filter is an asterisk (*) so all files in the current directory are listed.
2. Select the desired file and click **Open**. The file appears in the *Hpeesofeedit* window for editing.

Inserting One Text File into Another

The *Insert* command enables you to insert a text file, in its entirety, into another text file.

To insert a text file into the currently open file:

1. Select **File** > **Insert**. A dialog box appears.
2. Select the desired file and click **Insert**. The selected file is inserted at the current cursor position in the currently open file.

Saving Text Files

The File menu contains two commands related to saving files: *Save* and *Save As*.

- The *Save* command enables you to save changes to an existing file.
- The *Save As* command enables you save a new file, and name it in the process, or save an existing file with a new name. For example, if you would like a copy of an existing file so that you can make changes to it while preserving the original, you can use the *Save As* command to create a copy of the file with another name.

To save an existing file:

Select **File** > **Save**. The file is saved and the number of bytes and path/filename are displayed in the status bar.

To save a new file or to save an existing file with a new name:

1. Select **File** > **Save As...** and a dialog box appears.
2. Adjust the path, if desired.
3. Enter a name for the file in the Selection field.
4. Choose **Save As**. The file is saved.

Printing Text Files

To print a text file:

Select **File** > **Print**. The file is sent to the printer and a status window appears briefly displaying the status of the print request and noting which printer was used.

Note The setting of the environment variable *PRINTER* determines which printer is used. If this variable is not set, the default printer is used.

Exiting the Text Editor

To exit the program:

Select **File** > **Quit**. If the file has been modified since the last save, you are prompted, *File Modified, exit anyway?*

- To exit without saving the file, click **Yes**.
- To cancel so that you can save the file, click **No**.

Editing Text Files

The Edit menu enables you to copy, cut, paste, and delete text.

To copy text from one location to another:

1. Select the text you wish to copy.
2. Select **Edit** > **Copy**. The selected text is copied to the buffer.
3. Position the cursor in the desired location (in any window) and select **Paste**.

To move text from one location to another:

1. Select the text you wish to move.
2. Select **Edit** > **Cut**. The selected text is removed from that location and placed in the buffer.
3. Position the cursor in the desired location (in any window) and select **Paste**.

To paste text currently being held in the buffer:

Position the cursor in the desired location (in any window) and select **Edit** > **Paste**. The contents of the buffer appear at the current cursor location.

To delete text:

Select the text you want to delete and select **Edit** > **Delete**. The selected text disappears.

Performing Search and Replace Operations

The Search menu contains commands to enable you to jump to a specified line number in the text file and to search for and replace specified text strings.

To jump to a specific line number in the file:

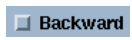
1. Select **Search** > **Go To...** and a dialog box appears.
2. Type the desired line number and click **OK**. The cursor jumps to that line number in the file.

To perform the search and replace operation:

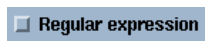
1. Select **Search** > **Search/Replace...** and a dialog box appears. In the *Search String* field, type the string of characters you want to search for, including wildcard characters, if desired. (If you use wildcard characters, you must check the box labeled *Regular Expression*.)

2. In the *Replace String* field, type the string of characters you want to use to replace the Search String characters.

3. Optionally, check this box if you want to search backward from the current cursor position.



4. Optionally, check this box if you want to include wildcard characters in your search.



Note For details on including *Regular Expressions* in your search, refer to your UNIX documentation.

---

5. Click **Search**. If the requested text is found, it is highlighted; if it is not found, a pop-up appears and displays the message *String not found*.
6. If you want to replace the highlighted occurrence with the contents of the Replace String field, click **Replace**. The highlighted text is replaced.

Hint To clear the highlight from text at any time, simply click somewhere inside the text editor window.

---

7. If you want to replace every occurrence of the specified search string at once, click **Replace All**. Every occurrence is replaced.
8. When you are through using the Search and Replace function, click **Cancel** to clear any remaining highlights from text and dismiss the dialog box.

Keyboard Mappings

The text entry area in the editor supports the complete Motif set of keyboard mappings. These mappings are primarily controlled by the *XKeysymDB* file (or its equivalent), which is typically found in */usr/lib/X11*. Table 12-2 describes the mapping of the Sun keyboard. (The mappings for the HP and IBM platforms follow the Sun's mappings with few exceptions.) Note that the *Meta* keys are labeled differently on different keyboards, but are always the keys on either side of the space bar.

Table 12-2. Keyboard mappings

Keyboard or Mouse Operation	Action
Copy	Copies selected text to the clipboard
Paste	Pastes text from the clipboard to current insertion point
Cut	Cuts selected text to the clipboard
Arrow Keys	Moves the insertion point cursor/scrolls window
Shift Arrow Keys	Moves insertion point, selects text
Page Up/Down	Moves the window up/down by the number of visible lines
Home/End	Moves cursor to beginning/ending of current line
Ctrl Home/End	Moves cursor to first/last visible line in window
Back Space	Deletes text to left of the insertion cursor
Meta Ctrl left arrow	Moves cursor backward one word
Meta Ctrl right arrow	Moves cursor forward one word
Meta Ctrl down arrow	Moves cursor down one paragraph
Meta Ctrl up arrow	Moves cursor up one paragraph
Left mouse button down/move	Selects text range
Shift left mouse button down/move	Extends selected text range

Appendix A: Using Online Documentation

The Advanced Design System includes online help and a complete set of online manuals, and is installed based on the software you choose to install.

The online documentation is in the form of HTML and can be viewed using Netscape or Internet Explorer. (Other browsers may work but are not supported at this time.) Most online manuals are also available as PDF files and can be viewed and printed using Adobe Acrobat Reader.

Individual browsers provide information on using various browser features. To access this information in Netscape, choose *Help* > *Help* followed by *Index*. In Internet Explorer, choose *Help* > *Contents and Index*.

In the Adobe Acrobat Reader, you can access information on how to use the Reader by choosing *Help* > *Reader Guide*.

Accessing Documentation

To access the online documentation, choose *Help* > *Topics and Index* from any ADS window. The browser opens and displays the online documentation front page. You can select a topic by clicking the appropriate link at the top of the page.

Searching

The online documentation provides the capability to search for a word or phrase in the currently open document, a category of manuals, or the entire (installed) documentation library.

To search for a word or phrase in the currently open document:

- From Netscape, select **Edit** > **Find In Page**. When the Find dialog box opens, enter the word or phrase in the Find field and click **Find**.
- From Internet Explorer, select **Edit** > **Find (on this page)**. In the Find dialog box, enter the word or phrase in the Find What field and click **Find Next**.

To search for a word or phrase in the entire documentation library:

1. From any documentation page, click **Search** from the set of links at the top of the content pane (on the right).
2. Enter the word(s) in the Query field, and click **Find**.

Hint You can restrict the global search to a specific category of documentation by selecting a category from the drop-down *Search in* list. Tips on searching are provided in the search window itself.

Printing

You can print individual help topics or entire books.

To print a help topic, click once in the content frame (to make it the active frame) and:

- In Netscape, choose **File** > **Print Frame**. When the Print dialog box appears, make the desired selections, and click **Print**.
- In Internet Explorer, choose **File** > **Print**. When the Print dialog box appears, make the desired selections, and click **OK**.

To print an entire book, or contiguous sections (formatted for printing):

Click the **Print version of this book (PDF file)** link at the top of any page of a currently displayed manual. When the document is displayed, choose **File** > **Print**. In the Print dialog box, specify the desired print range.

Hint The page numbering of the PDF file is displayed in the status bar at the bottom of the window.

Appendix B: Shortcut Keys

Table B-1 lists the default keyboard shortcuts found in the Main, Schematic, Layout and Data Display windows. Some of the ADS individual tools also offer the ability to customize shortcut keys. For details on customizing these shortcuts, refer to the section, "Customizing Keyboard Shortcuts" on page 9-30.

Table B-1. Shortcut Keys

Edit > Copy/Paste Commands	
Cut	Ctrl + x
Copy	Ctrl + c
Paste	Ctrl + v
Copy Using Reference	Shift + C
View Commands	
View All	Ctrl + f
Restore Last View	Ctrl + l
Zoom In x2	+ (addition key)
Zoom Out x2	- (subtraction key)
Grid/Snap Commands	
Grid Display, Minor Grid Display (toggle on/off)	Ctrl + g
Minor Grid Display	Ctrl + g
Major Grid Display	Shift + g
Snap Enabled	Ctrl + e
Move Commands	
Move Using Reference	Ctrl + m
Move Wire/Trace Endpoint	Ctrl + Shift + m
Move Component Text (Edit > Move)	F5
Rotate Commands	
Rotate	Shift + r
Mirror About X	Shift + x
Mirror About Y	Shift + y
Rotate -90	Ctrl + r

Table B-1. Shortcut Keys (continued)

File Management Commands	
New Design	Ctrl + n
Open Design	Ctrl + o
Save Design	Ctrl + s
Window Commands	
Close Window (Schematic and Layout windows)	Ctrl + F4
Schematic (Schematic and Layout windows)	Ctrl + Shift + s
Layout (Schematic and Layout windows)	Ctrl + Shift + l
New Schematic (Main window)	Ctrl + Shift + n
New Layout (Main window)	Ctrl + Shift + a
Designs Open	Ctrl + d
Shape Commands	
Polygon (Insert)	Ctrl + Shift + p
Rectangle (Insert)	Ctrl + Shift + r
Undo Commands	
Undo	Ctrl + z
Undo Vertex	Shift + z
Help Commands	
What's This?	Shift F1
Topics and Index	F1
Simulate Commands	
Simulate	F7
Select Commands	
Select All (Select)	Ctrl + a
Miscellaneous Commands	
Print (File)	Ctrl + p
Change Entry Layer To (Insert)	Ctrl + Shift + c
Delete (Edit)	Del
End Command (Edit)	Esc
Text (Insert)	Ctrl + Shift + t

Table B-1. Shortcut Keys (continued)

Clear Highlighting (View)	F8
Wire (Insert)	Ctrl + w
Trace (Insert)	Ctrl + t
Exit Advanced Design System (File)	Alt + F4

Table B-2 lists the default shortcut keys, in alphabetical order, to enable you to see at a glance which ones are in use. If you attempt to assign any of these to other commands, you are warned and given the choice to proceed or to select another key sequence (see footnote for *Alt*).

Table B-2. Sorted Default Shortcut Keys

Ctrl	**Ctrl + Shift**	**Shift**	**Alt***	**Others**
Ctrl + a	Ctrl + Shift + a	Shift F1	a	+ (add key)
Ctrl + c	Ctrl + Shift + c	Shift + C	d	- (subtract key)
Ctrl + d	Ctrl + Shift + l	Shift + g	e	Alt + F4
Ctrl + e	Ctrl + Shift + m	Shift + r	f	Del
Ctrl + f	Ctrl + Shift + n	Shift + x	h	Esc
Ctrl + f4	Ctrl + Shift + p	Shift + y	i	F1
Ctrl + g	Ctrl + Shift + r	Shift + z	l	F5
Ctrl + l	Ctrl + Shift + s		m	F7
Ctrl + m	Ctrl + Shift + t		o	F8
Ctrl + n			r	
Ctrl + o			s	
Ctrl + p			t	
Ctrl + r			v	
Ctrl + s			w	
Ctrl + t				
Ctrl + v				
Ctrl + w				
Ctrl + x				

Table B-2. Sorted Default Shortcut Keys (continued)

Ctrl	Ctrl + Shift	Shift	Alt*	Others
Ctrl + z				
* If you use *Alt* as the modifier key (when creating custom shortcuts), and a letter that is already assigned as an accelerator for a menu (listed in this table), the menu accelerator is replaced by your custom shortcut with no warning.				

Appendix C: Using Advanced Design System Across Platforms

A device library or design project stored on a shared volume can be accessed and modified from either a PC or a UNIX workstation. In addition, files created on one platform can be transferred and used on another.

When creating and modifying a file on a shared volume, keep in mind the following:

- You need read and write access to modify a file on a shared volume.
- Only one user can access a file at a time.

Files can be transferred electronically via FTP or they can be moved physically using a medium such as floppy disks. When transferring libraries and projects from one platform to another, keep in mind the following:

- Use a Binary (image) file transfer method.
- Maintain the original file and directory structure at all times.
- Copy all hidden files.
- Use the *Archive Project* command before copying a project to a diskette or transferring it via FTP. For details, refer to "Archiving a Project" on page 2-11.

Note If a project created on UNIX contains two or more designs whose names are only distinguishable from one another by differences in case, do not archive and transfer this project to a PC without renaming the designs such that they all have unique names. This requirement is due to the fact that the PC is case insensitive.

Opening Projects

To open a project, you need a licensed copy of Advanced Design System for your current platform and access to the current location of the project files.

To open a project:

1. In the Main window, choose **File** > **Open Project** or click the **Open Project** button.
2. In the dialog box that appears, select the project and click **OK**.

Guidelines for Cross-platform Use

Use the following guidelines when creating libraries or projects for cross-platform use:

- Use filename characters and conventions that are legal on both platforms. For example, Windows does not recognize case-sensitive differences in filenames.
- Use only the standard ASCII character set for text.
- Use fonts that are standard on both platforms.
- Use colors that are standard on both platforms.

Appendix D: Glossary

accelerator keys

See *shortcut keys*.

active design

The design currently displayed in any window.

active window

The window to which the next keystroke or command will apply. If a window is active, its title bar changes color to differentiate it from other open windows.

AEL

Application Extension Language. A C-like general purpose interpreted language used in Advanced Design System products to configure and define extensions to the design environment. It is available to any application as a linkable library.

annotation

Old Series IV term used to denote explanatory text that is automatically placed with a component. Now referred to as component text.

back annotation

Synchronizing values between optimization and the Schematic window, or between a schematic and its associated layout.

background

The drawing area of a screen upon which a design or data display is placed.

bias-dependent analysis

Whenever a single-point DC bias analysis is required to carry out a linear analysis, it is referred to as a bias-dependent analysis.

boot

To start or restart your computer by loading the operating system.

budget analysis

A single-tone bandpass analysis that determines the system internodal signal and noise performance for components in the top level of a system network.

bundle

A collection of wires (or buses) that do not share the same base name, for example, *Data<1>,ABC<1>*.

bus

A set of wires or a single wire carrying a set of signals. For example, *Data<1>*, *Data<2>*, *Data<3>* represents a bus because all three wires share the base name *Data*.

bus pin

A vectorized pin is a component pin that represents multiple connections into the component.

check box

A small box or circle inside a dialog box. A check box represents an option and it can be turned on and off. When the option is selected, it appears darkened.

click

To press and release a mouse button quickly (most often, the left mouse button).

command

A word or phrase found on a menu or represented by a button on a toolbar, that you choose to carry out an action.

component text

Explanatory text that is automatically placed with a component.

connection width

The total number of pins at a given port of an *iterated instance*. Computed by multiplying the pin width at the port by the *instance width*.

convolution

A SPICE-type analysis involving the solution of a set of integro-differential equations that express the frequency dependence of the currents and voltages of the circuit under analysis. Convolution represents all the distributed elements in the frequency domain to enable analysis of their frequency-dependent behavior. It is based upon the premise that characterization of many RF and microwave distributed elements is best accomplished in the frequency domain.

cursor

A graphical representation of the location of the mouse. A cursor appears when the location requires text input.

data display window

Window in which to view simulation results. A data display window can be opened from the Main window, a Schematic window, or a Layout window.

data set

A file that contains data from a given simulation.

design environment

The entire group of windows used for creating and simulating designs and viewing results for the Agilent EEsof simulator suite. It includes the Main window, all Design windows, the Message/Status window, and the Data Display window.

design files

Project files, which include schematic and layout information. These files have a *.dsn* extension.

design units

See *layout units*, *schem units*.

design windows

Schematic and Layout windows.

dialog box

A rectangular box that requests and/or provides information.

directory

A collection of files and subdirectories that are stored at the same location on a disk. See also *subdirectory*.

double-click

To press and release a mouse button twice in rapid succession without moving the mouse. Double-clicking carries out an action, such as displaying the dialog box for editing component parameters.

drag

To move an item on the screen by pressing and holding down the mouse button and then moving the mouse. For example, you can move a window to another location on the screen by dragging its title bar.

drawing formats

Templates for use as a design sheet that include a title block and space for design information. Standard sizes include: A, B, C, D, E, and metric.

Electrical component models

Models based on circuit theory signal S-parameters and noise wave parameters. Each port of any n-port electrical element has a bidirectional signal flow; that is, each port has an input and output signal and noise characteristic. At any port of an *n*-port electrical element, incident signal and noise produce reflected signal and noise at the incident port; the incident signal and noise produce signal and noise output at all other *n*-1 element ports.

component

An item placed in the Schematic or Layout window, typically an electrical component or specification.

Functional, optical, and DSP component models

Models based on transformation of input signals and noise into output signals and noise. Each component port is unidirectional, and is either an input port or an output port.

graph

A diagram showing the values of a function.

grid

A set of evenly spaced horizontal and vertical dots or lines to aid in locating specific points.

harmonic balance

An iterative method of analysis based on the assumption that for a given sinusoidal excitation, there exists a steady-state solution that can be approximated to satisfactory accuracy using a finite Fourier series.

hierarchical design

A network design containing one or more subnetworks.

histograms

There are two types of histograms available with yield analysis: *sensitivity* and *measurement* histograms.

- The *sensitivity* histogram plots percent pass versus parameter value bin. Each sample has parameter values randomly assigned to those parameters with yield value types.
- The *measurement* histogram plots percent of samples versus measurement value bin. The random parameter values causes the measurements listed in a MEAS_HIST item to be statistically distributed.

instance width

The number of instances in an *iterated instance.*

iterated instance

An instance whose instance name has been modified to include a vectorized label. For example, Rx<0:2> refers to Rx<0>,Rx<1>,Rx<2>.

iterated pin

See *bus pin*.

keyboard shortcuts

See *shortcut keys.*

large-signal S-parameter analysis

The calculation of large-signal, power-dependent S-parameters of a 2-port network, which can be stored in a P2D disk file for use in the system simulator.

layout units

The unit used for grid display and cursor snapping, usually the same as length unit. (See also *length unit* and *schem units*.)

length unit

In the Schematic and Layout windows, the unit of measure for parameters with physical length. Also the unit used for grid display and cursor snapping in the Layout window. (See also *layout units* and *schem units*.)

library

A collection of components sharing similar characteristics.

list box

A region within a dialog box that lists all available choices.

macro

A series of actions represented by AEL functions that can be played back using the *Playback Macro* command on the Options menu of the Main window. May also refer to AEL artwork macros that you can assign to networks or new components.

menu bar

The horizontal bar containing the names of all the menus. It appears below the title bar.

message window

A window that is displayed during simulation and optimization. Messages displayed within this window report on the status of the current operation.

meta key

The key on either side of the space bar. The label on this key varies depending on the keyboard manufacturer.

mnemonics

A series of keys pressed to initiate a command. The key combination typically includes a significant letter in the command name, such as Alt + F to access the File menu. See also *shortcut keys*.

net width

The number of individual wires in a bus/bundle.

netlist

A textual listing of all the parts in a design showing how they are interconnected.

network

A design consisting of interconnected components with input/output ports, and with associated data, units, and optional variables and equation items. In general, a network can have a schematic and a layout representation.

node

The connection of pins of two or more components or items.

nonlinear steady-state circuit analysis

An analysis performed by the Harmonic Balance, Oscillator, or Large-signal S-parameter test benches. The types of analyses are:

- Harmonic balance analysis (single- and multi-tone)
- Mixer analysis
- Dynamic load-line analysis
- Oscillator analysis
- Large-signal S-parameter analysis

nonlinear transient time-domain circuit analysis

An analysis involving the solution of a set of integro-differential equations that express the time dependence of the currents and voltages of the circuit under analysis. The result is a nonlinear analysis with respect to time and, possibly, a swept variable.

optimization

There are two kinds of optimization: *performance optimization* and *yield optimization*.

- *Performance optimization* is the process of modifying parameter values in order to satisfy predetermined performance criteria. Optimizers compare the computed and desired responses and modify design parameter values to bring the computed response closer to the desired response.
- *Yield optimization* involves a series of yield analyses where the yield variable nominal values are adjusted to maximize the yield estimate. In yield optimization, each yield improvement is referred to as a design iteration.

oscillator analysis

A special kind of harmonic balance analysis in which the frequency of operation varies as the algorithm progresses, allowing the simulator to zoom in on the actual frequency of oscillation for a given oscillator circuit.

palette

A set of buttons with icons representing various commands that allow quick execution of those commands (found on the left side of all design windows).

parameters

The characteristics defining a component.

parametric subnetwork

A network design with associated parameters that are passed to the main network when it is placed as a subnetwork within the main network.

PDE

See project design environment.

pin

A terminal on a schematic or layout component.

pin width

The number of pins at the given port of a component. See also *bus pin*.

pointer

A graphical representation of the current location of the mouse

port

Refers to either a terminal on a network or symbol to indicate input or output.

post production tuning

A process that combines with yield analysis to allow certain parameter values to receive additional tuning. Typically, post production tuning is used to ensure that yield specifications are met.

project

A set of designs stored in a predetermined directory structure referred to as a project directory. It contains subdirectories for different types of files.

Project Design Environment

The entire group of windows used for creating and simulating designs and viewing results for a project using the Agilent EEsof simulator suite.

schem units

In the Schematic window, the unit used for grid display and cursor snapping, inches. (See also *length unit* and *layout units*.)

scroll bars

The bars at the bottom and right edge of a window or list box that appear when the contents are not entirely visible. Each scroll bar contains a small box, called a scroll box, and two scroll arrows to allow different directions in scrolling.

shortcut keys

A combination of one or more modifier keys (Ctrl, Shift, Alt, etc.) and a letter that equates to a command. Pressing these keys initiates the command, for example, Ctrl + r initiates the rotate command.

small-signal analysis

An alternate mode of analysis that often increases simulation speed. When used for mixers, small-signal analysis assumes that amplitude of the RF signal is small enough to generate negligible harmonics, and that the power level of the RF signal is smaller than that of the LO signal.

spurious signals

Spurious signals are generated within RF systems by mixers, nonlinear amplifiers, and spectrally impure oscillators.

statistical design

The process of varying a set of parameter values over specified probability distributions to determine how many possible combinations satisfy predetermined performance criteria. The statistical design methods used are yield analysis, yield optimization.

subnetwork

A network that is used as a component inside of another network.

symbol

A graphical representation of a component or subnetwork.

tap

A tap is a wire connected to a bus that splits a wire (or in the case of multi-wire tapping, a new bus/bundle) off of a bus/bundle.

toolbar

A group of buttons with icons representing various commands that allow quick execution of those commands (found at the top of all design windows).

units

See individual entries for *design units*, *length units*, *layout units*, and *schem units*.

user units

Old Series IV term for *schem units* or *layout units*.

vertex

A beginning or ending point, turn, or junction in a segment.

wildcard character

A character that represents another character. In filenames, you can use the asterisk (*) as a wildcard character to indicate any character or group of characters that might match that position in other filenames. For example, *.*dsn* represents all files with the *.dsn* filename extension.

yield analysis

A process involving a yield estimation over a given number of trials in which the yield variables have values that vary statistically about their nominal values according to specified probability distribution functions.

yield optimization

See *optimization*.

Index

D

E

F

N

O

P

R

S

T

U

V

W

Z