



Agilent Technologies

Advanced Design System 2002
Netlist Exporter

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.

Copyright © 2002, Agilent Technologies. All Rights Reserved.

Acknowledgments

Design Framework II™ and Composer™ are trademarks of Cadence Design Systems Incorporated.

Copyright © 2001 Cadence Design Systems Incorporated. All rights reserved.

Mentor Graphics® is a registered trademark of Mentor Graphics Corporation.

Copyright © 1997-2001 Mentor Graphics Incorporated. All rights reserved.

Contents

1 Introduction

Front End Flow	1-2
Design Flow Using Front End Flow	1-2
General Process.....	1-4
Intended Audience.....	1-6
Main Requirements	1-6
What's in this Manual.....	1-6

2 Creating Netlists

Creating a Netlist	2-1
Including Files in Netlists	2-5
Defining Files to Include	2-5
Setting up Options for a Netlist	2-11
Setting Netlist Options	2-11
Setting Netlist Options Example	2-11

Index

Chapter 1: Introduction

Front End Flow (FEF) is a design flow tool in which the Advanced Design System (ADS) schematic editor is used as a single point of entry to drive an entire design flow. ADS supports three other design flow tools: Netlist Translator, Intermediate File Format (IFF), and RFIC Dynamic Link.

Design Flow Tool Overview

Netlist Translator	Third party schematic entry tools are used to create SPICE or Spectre netlists. The netlists are translated into ADS netlists or schematics. This requires that schematic data be duplicated and synchronized in multiple environments.
IFF	ADS is used as a schematic entry tool. The schematic is transferred from ADS to other tools using IFF to complete the design. This requires that schematic data be duplicated and synchronized in multiple environments.
RFIC Dynamic Link	A single schematic entry tool, Cadence Virtuoso Composer, is used as the point of entry for schematic designs. ADS is used to add test bench data around the top level IC schematic symbol. All of the schematic data is represented in a single environment (Cadence); and there is a <i>limited</i> link to the ADS simulation tools and Instrument Server.
Front End Flow	Front End Flow allows all schematic design entry to be done in ADS. There is a single data representation for the schematic data with a direct link to the ADS simulation tools and Instrument Server. In addition, there are new customizable netlisters and tool interfaces available to allow the ADS schematic tool to interface with third party layout and Layout Verses Schematic (LVS) tools.

Front End Flow

The Front End Flow tool enables you to perform the following:

- Create schematic and layout designs from a single design entry point (ADS).
- Use the powerful ADS simulation and Instrument Server tools.
- Use third-party layout and LVS tools while maintaining a single schematic data source (no redundant data to maintain).
- Use third-party simulation tools.

Front End Flow can be used to generate netlists on one platform that are transferred to other systems. This is necessary in cases where the third party tool does not run on the same system as ADS. For example, you may be running ADS on a PC and an LVS tool from Cadence that will only run on UNIX.

Design Flow Using Front End Flow

The key feature of the ADS Front End Flow tool is a configurable netlister that is able to target the netlist format required for any CAD tool. The most common target format is HSpice, which is supported by many CAD tools. ADS 2001 includes configurations for several HSpice derived formats including those for Cadence Assura, Cadence Dracula, and Mentor Graphics* Calibre. In addition, the netlister can be configured by modifying certain functions to support any other target format. The *Netlist Exporter Installation and Configuration* Guide covers the steps required to custom configure the Front End Flow netlister.

Figure 1-1 depicts the design flow using the Front End Flow tool. Note that the Front End Flow netlister can generate netlists for several different tools.

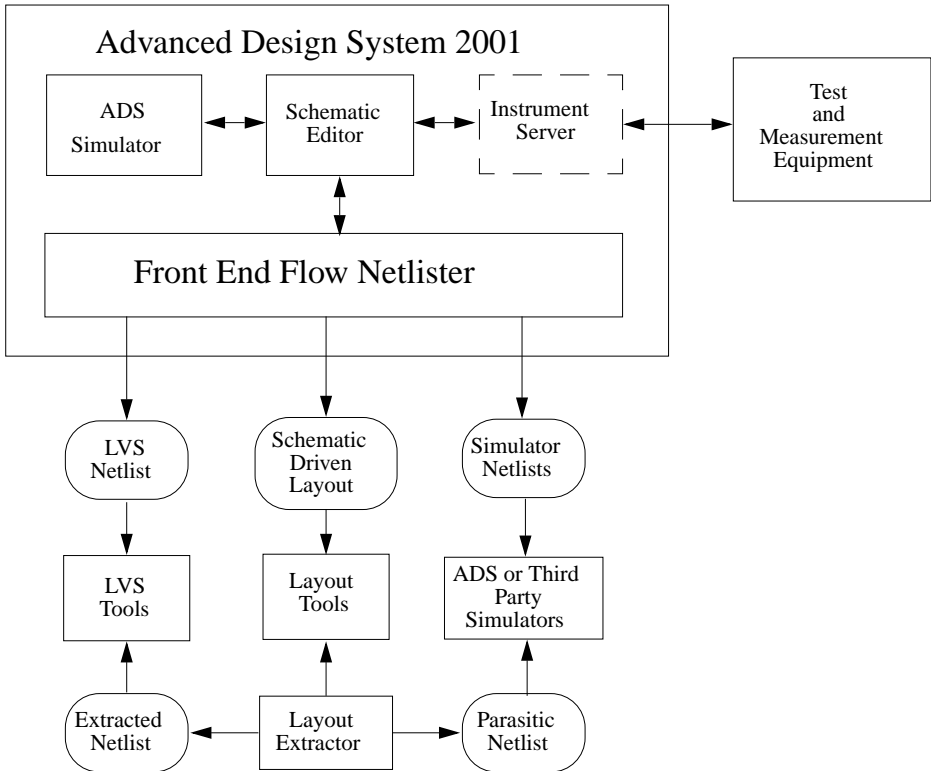


Figure 1-1. Front End Flow Flowchart

General Process

The following describes the typical task flow using Front End Flow. Refer to [Figure 1-2](#).

1. A schematic design is created within ADS.
2. This design is simulated using ADS.
3. When the simulation results match the circuit specification, a layout for the schematic is generated.
4. The Front End Flow netlister can create netlists that can be utilized to generate layouts, such as VirtuosoXL.
5. Once the layout has been generated, an LVS tool is used to verify that the layout representation is the same as the schematic representation.
6. To do this, a netlist is generated in an appropriate format for an LVS tool. Additionally, a layout extraction utility generates a netlist that the LVS tool will compare to the Front End Flow netlist.
7. When the layout and schematic match, the layout extractor can be used to generate netlists with parasitics.
8. If ADS is not the simulator used for the parasitic resimulation, the Front End Flow netlister can generate a netlist for a third party simulator.
9. Resimulate the netlist.

[Figure 1-2](#) is a simplified task flow for using Front End Flow.

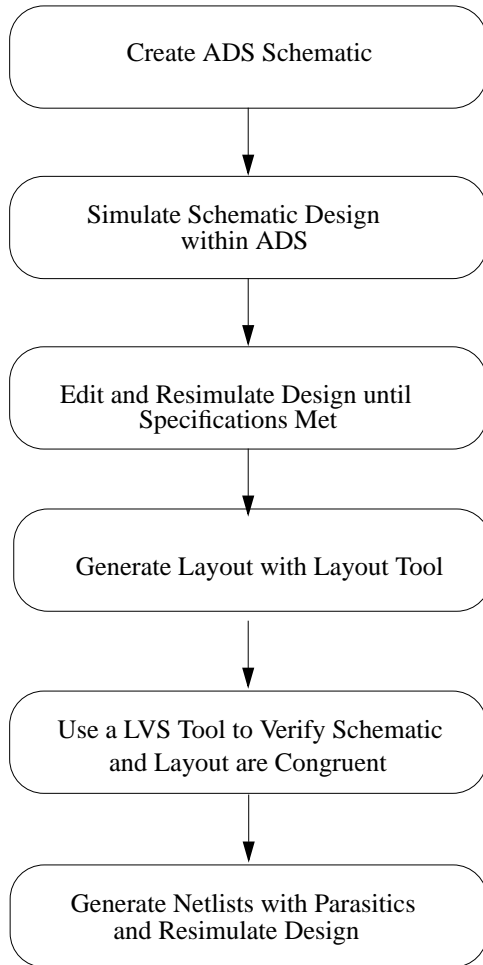


Figure 1-2. Simplified Task Flow

Intended Audience

The audience intended for this manual consists of CAD System Administrators and RFIC Designers who are using Advanced Design System 2001 or newer to create RFIC designs. It is assumed that the designer using Front End Flow has some working knowledge of Advanced Design System. This manual does not cover how to use third party tools that can be accessed from ADS. For information on other vendor's tools, consult that vendor's manuals.

Main Requirements

To use Advanced Design System Front End Flow, you must have Advanced Design System 2001 or newer installed. Front End Flow can be used on all platforms that are supported by Advanced Design System.

What's in this Manual

The goal of this manual is to help you get started, provide relevant examples that teach you how to set up and use the software and show you where you can get more information as you need it.

[“Creating Netlists” on page 2-1](#) provides step by step instructions for generating netlists using Front End Flow. This chapter covers procedures to use the pre-configured Front End Flow tools. Refer to the *Netlist Exporter Setup* documentation for instructions on generating custom netlists.

Chapter 2: Creating Netlists

Front End Flow is able to produce netlists in non-ADS formats. These netlists can then be used within various non-ADS design tools. For more information, refer to the *Netlist Exporter Setup* documentation.

Creating a Netlist

To create a Front End Flow netlist, use the following procedure:

1. Open the design to be netlisted (refer to the ADS "*User's Guide*" for information on opening designs).

Note The design must be saved with a title at least once prior to netlisting. Front End Flow netlisting options will not be activated for untitled designs.

2. From the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**. The *Create ADS Front End Netlists* dialog appears (see [Figure 2-1](#)).
3. Select the desired design tool from the **Tool** drop-down list. Front End Flow will produce a netlist compatible with that tool.

Note The tool will be remembered between ADS sessions, so it will not be necessary to select the tool every time a netlist is generated. The Tool drop-down list is constructed each time the dialog window is brought up, and will give you choices that are available based on the component definition directories that are within the component path. In this example, the tools Assura, Calibre, and Dracula are available.



Figure 2-1. Create Front End Flow Netlists Dialog

4. The *Design to Netlist* field contains the design name that will be netlisted (source file). The field defaults to the design name that is in the active window. If you prefer, enter a different design to netlist.
5. Click the *Netlist file* **Browse** button, or directly enter the full path and file name, to define the file that will be created by the netlister (target file).

Note If no path is specified, the file will be created in the currently opened project's directory.

Caution If the specified *Netlist file* already exists, it will be overwritten without warning.

6. Click the *Log file* **Browse** button, or directly enter the full path and file name, to define the log file name. The *Log file* will contain any netlisting errors or warnings.

Note If no path is specified, the file will be created in the currently opened project's directory.

Caution If the specified *Log file* already exists, it will be overwritten without warning.

7. Click the **Modify Include File List** checkbox to specify the files to include in the final netlist. For more information, refer to [“Including Files in Netlists” on page 2-5](#).
8. Click the **Modify Option List** checkbox to specify the options that will be output into the header of the netlist file. For more information, refer to [“Setting up Options for a Netlist” on page 2-11](#).
9. Specify comments within the *Comments* section that you would like to have output into the header of the netlist.
 - Select the **Include date and time as a comment** to place a comment line in the file that specifies the date and the time that the netlist was generated.
 - Select the **Include design name as a comment** to place a comment line in the file that specifies the design used to generate the netlist.
 - Enter comment text, to be included in the header of the netlist file, within the edit text box (located below the *Include design name as a comment* checkbox).

Note Comments from the edit box will be automatically output with the line comment character, so it is not necessary to type the comment character with the comments. The comments will be remembered through the current ADS session only. They are not stored between ADS sessions.

10. Click the **View netlist file when finished** checkbox to load the netlist file into the standard text editor after netlisting is finished. This allows you to visually inspect it for netlisting errors. An example netlist file is shown in [Figure 2-2](#).

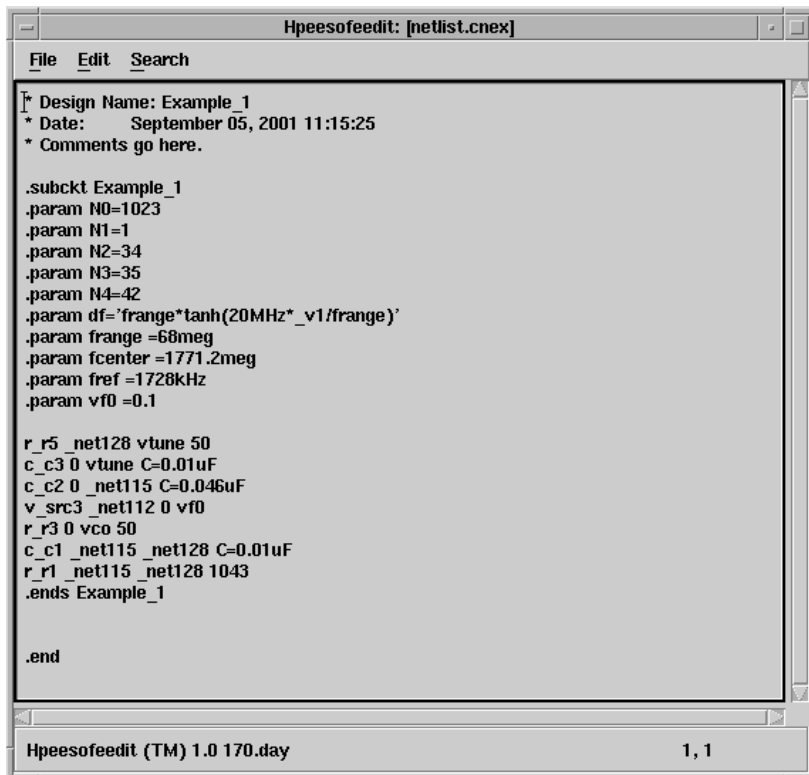


Figure 2-2. Example Netlister Netlist File

11. Click the **View log file when finished** checkbox to load the netlist file into the standard text editor after netlisting is finished. An example log file is shown in [Figure 2-3](#).

If neither the *View netlist file when finished* nor the *View log file when finished* checkbox is set, an information dialog will appear that informs you that netlisting has completed.

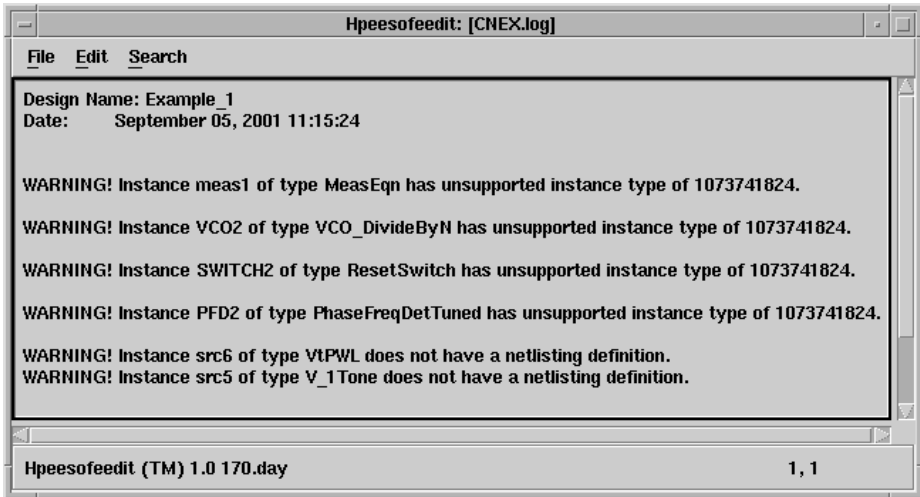


Figure 2-3. Example Netlister Log File

12. Click **OK** to create the netlist.

Including Files in Netlists

Most IC Foundries place symbols within schematics that refer to subcircuits. These subcircuits will typically contain parasitic and non-parasitic devices. Subcircuit component representation allows for a more accurate representation than primitive device representation. Since a single foundry process may use multiple design tools with different subcircuit definitions, most foundries elect to include model files in the final netlist. The Front End Flow netlister allows the inclusion of model files within netlists, although this is not a preferred practice for the ADS simulator.

Caution Options defined in include files will be set regardless of the settings in the *Options* dialog (see [“Setting up Options for a Netlist” on page 2-11](#)).

Defining Files to Include

To define the files to include in a netlist, use the following procedure:

1. If not already open, open the *Create ADS Front End Netlists* dialog (from the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**).
2. Select the desired design tool from the **Tool** drop-down list.
3. Click the **Modify Include File List** button. The *Include Files* dialog will appear (see [Figure 2-4](#)).

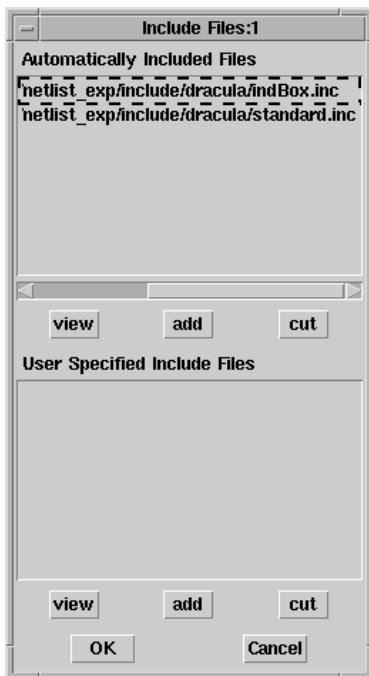


Figure 2-4. The Include Files Dialog

4. Add or cut automatically included files as required. See [“Automatically Included Files” on page 2-7](#) for information about these files.
5. Add or cut user specified include files as required. See [“User Specified Include Files” on page 2-9](#) for information about these files.
6. Click **OK** to close the *Include Files* dialog.
7. Refer to [“Creating a Netlist” on page 2-1](#) to create the netlist.

Automatically Included Files

To determine the files listed in *Automatically Included Files*, Front End Flow will check all paths, looking for an *include* subdirectory under any *netlist_exp* directory that contains components (the component definitions should always be in an *netlist_exp* directory). If the *include* directory contains a directory for the tool selected in the *Create ADS Front End Netlists* dialog, all of the files within the tool directory will be included in the *Automatically Included Files* list. For example, all of the files located within the directory *\$HOME/hpeesof/netlist_exp/include/dracula* will be included in the list if *\$HOME/hpeesof/netlist_exp* contains components and *dracula* is the selected tool. These files can be manually removed from the list, see [“Cutting Automatically Included Files” on page 2-7](#).

Note If new files are added into the directories that have automatically included files, they will show up in the *Automatically Included Files* list the next time the *Create ADS Front End Netlists* dialog is opened.

Normally, the automatically included files will be a part of a design kit, and will contain specific subcircuits necessary for simulation or LVS. These files can also be placed in other directories if there are particular options that must be set for all newly created designs. In the example, the two files, *standard.inc* and *indBox.inc*, were placed in *\$HOME/hpeesof/netlist_exp/include/dracula*, see [“User Specified Include Files” on page 2-9](#). The two files will be included automatically in every project.

Viewing Automatically Included Files

Files within the *Automatically Included Files* list can be viewed by selecting the file to view and clicking the **view** button. This will open the file in the ADS text editor.

Cutting Automatically Included Files

All files that are under the *include* directories will be in the *Automatically Included Files* list until manually removed, see [“Automatically Included Files” on page 2-7](#).

To remove files from the list, use the following procedure:

1. If not already open, open the *Create ADS Front End Netlists* dialog (from the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**).
2. Select the desired design tool from the **Tool** drop-down list.

3. Click the **Modify Include File List** button. The *Include Files* dialog will appear.
4. Select the file within the *Automatically Included Files* field to cut.
5. Click the **cut** button under the *Automatically Included Files* field.
6. Repeat steps 3 and 4 as required. Click **OK**.

This will remove the selected files from the *Automatically Included Files* list. The removed files will remain removed from session to session. Files can be added back into the *Automatically Included Files* list by using the add procedure.

Adding Automatically Included Files

To add files back into the list (that were previously cut), use the following procedure:

1. If not already open, open the *Create ADS Front End Netlists* dialog (from the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**).
2. Select the desired design tool from the **Tool** drop-down list.
3. Click the **Modify Include File List** button. The *Include Files* dialog will appear.
4. Click the **add** button under the *Automatically Included Files* field. The *Add Automatically Included Files* dialog appears (see [Figure 2-5](#)).



Figure 2-5. Add Automatically Included Files Dialog

5. Select the file to include.
6. Click **OK** or **Apply** to add the file.

7. Repeat steps 5 and 6 as required.

If there are no files left to include, and you press the **Apply** dialog, the dialog will automatically be dismissed.

User Specified Include Files

Files that are not in an *include* directory can be selected for inclusion into the netlist by adding these files in the *User Specified Include Files* list. These files can reside in any path. User specified include files are useful for selecting files that need to be included for a one time usage or files that are under test.

Adding User Specified Include Files

To add files into the list, use the following procedure:

1. If not already open, open the *Create ADS Front End Netlists* dialog (from the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**).
2. Select the desired design tool from the **Tool** drop-down list.
3. Click the **Modify Include File List** button. The *Include Files* dialog will appear.
4. Click the **add** button under the *User Specified Included Files* field. The ADS file browser appears.
5. Select the file to include.
6. Click **OK** in the browser.

Note Make sure to select the correct file; Front End Flow does not validate the format of the included file. The list of files to include will be stored in a configuration file, so the user specified include file list is maintained between ADS sessions.

Cutting User Specified Include Files

To remove files from the list, use the following procedure:

1. If not already open, open the *Create ADS Front End Netlists* dialog (from the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**).
2. Select the desired design tool from the **Tool** drop-down list.

3. Click the **Modify Include File List** button. The *Include Files* dialog will appear.
4. Select the file within the *User Specified Included Files* field to cut.
5. Click the **cut** button under the *User Specified Included Files* field.
6. Repeat steps 3 and 4 as required. Click **OK**.

Example Include File

For this example the *Include Files* dialog shown in [Figure 2-4](#) will be used. Clicking the *view* button with the *standard.inc* file selected in the *Include Files* dialog, will open the file in the text editor, as shown in [Figure 2-6](#).

The *standard.inc* file in this example automatically sets the appropriate Dracula options for LVS. The file contains three lines, and will set the `.BIPOLAR`, `.CAPVAL`, and `.RESVAL` options.

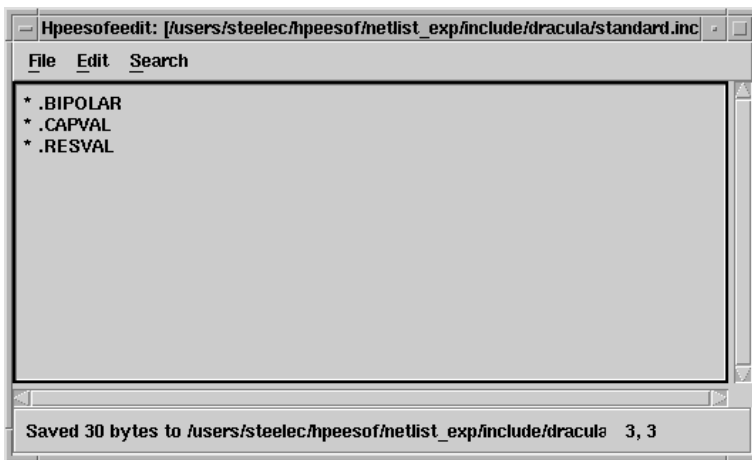


Figure 2-6. The *standard.inc* file

Since *standard.inc* files has options set within it, you may not want it to be included in the final netlist. That is, the appropriate options are already set by using the options dialog and you do not want the *standard.inc* options to override them. In this case, cut the *standard.inc* file from the *Automatically Included File* list, see [“Cutting Automatically Included Files” on page 2-7](#).

Setting up Options for a Netlist

This section refers to options for a particular tool that are output into a netlist (for example, `.Temp=25` in a SPICE netlist). Front End Flow has an options dialog that can be configured for each tool. For information on how to configure the options dialog, refer to the *Netlist Exporter Setup* documentation.

Note Options defined in include files will be set regardless of the settings in the *Options* dialog (see [“Including Files in Netlists” on page 2-5](#)).

Setting Netlist Options

To set or modify netlist options, use the following procedure:

1. If not already open, open the *Create ADS Front End Netlists* dialog (from the Schematic window, select the menu option **Tools > Netlist Export > Create ADS Front End netlist**).
2. Select the desired design tool from the **Tool** drop-down list.
3. Click the **Modify Option List** button.

A dialog that is specific to the selected tool will be displayed. For information on each option for a specific tool, you must consult the documentation for that tool.

4. Select the options you want contained in your netlist.
5. Click **OK** to close the options dialog.

The selected options will be saved in the file, `<tool>.cfg`, in the current project's directory. The options are available for later use.

Note The option format in the netlist will be dependent on the tool being used.

6. Refer to [“Creating a Netlist” on page 2-1](#) to create the netlist.

Setting Netlist Options Example

In this example, the selected tool is Dracula. When the *Modify Option List* button is clicked, the *Dracula Options* dialog appears (see [Figure 2-7](#)).

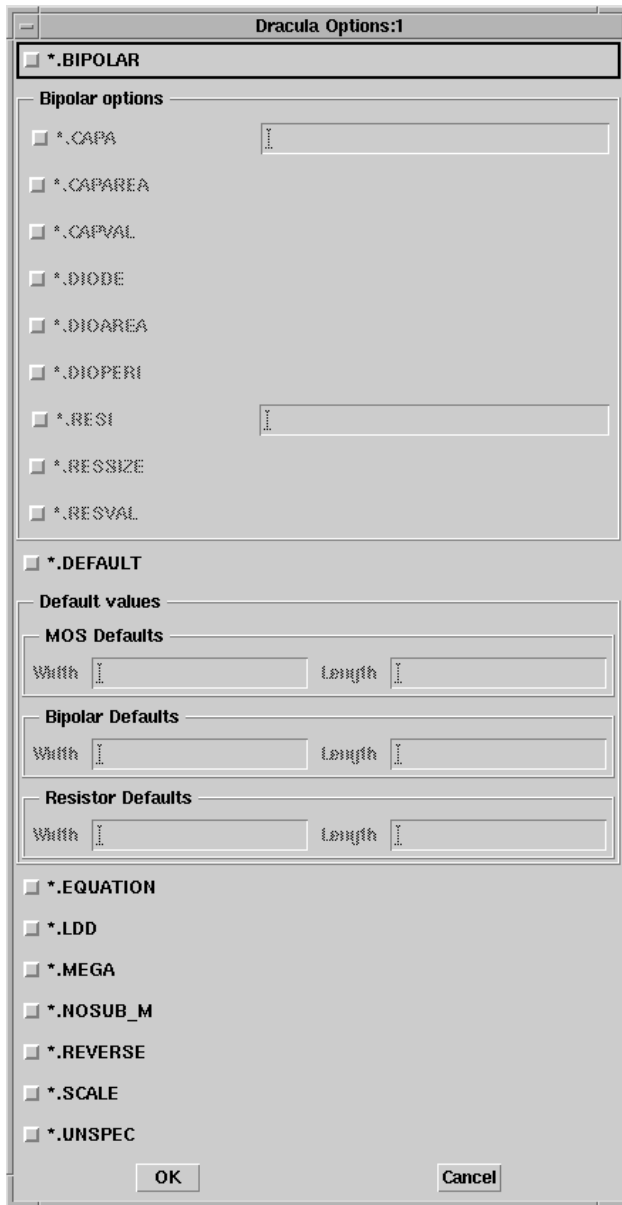


Figure 2-7. Dracula Option Dialog

The *Dracula Options* dialog has been configured to contain only the Dracula netlist options relevant to ADS. In addition, the *Dracula Options* dialog is set up to have dependent options become enabled or disabled based on whether certain options are set. For example, if .BIPOLAR is set as an option, the options within the Bipolar Options group box will all become enabled.

Each tool's options dialog may have different behavior depending on the approach taken when customizing for your use. For more information, refer to the *Netlist Exporter Setup* documentation.

When the **OK** button is clicked, the selected options will be saved in the file, *<tool>.cfg*, in the current project's directory. The options are available for later use.

The option format in the netlist will be dependent on the tool being used. For Dracula, each option outputs with the line comment character (therefore, for a simulator, the Dracula options would be seen as comments).

Index

- A
 - Advanced Design System, 1-1
 - automatically included files, 2-7
 - C
 - CAD, 1-2
 - Cadence
 - Assura, 1-2
 - Dracula, 1-2
 - D
 - defining files to include, 2-5
 - designs
 - layout, 1-2
 - E
 - example include file, 2-10
 - extraction utility, 1-4
 - F
 - FEF, 1-1
 - files
 - automatically included, 2-7
 - user specified include, 2-9
 - Front End Flow, 1-1
 - G
 - general process, 1-4
 - H
 - HSpice, 1-2
 - I
 - IFF, 1-1
 - including files in netlists, 2-5
 - Intermediate File Format, 1-1
 - L
 - layout designs, 1-2
 - layout extraction utility, 1-4
 - layouts, 1-4
 - LVS tools, 1-2
 - M
 - main requirements, 1-6
 - Mentor Graphics
 - Calibre, 1-2
 - N
 - netlist
 - automatically included files, 2-7
 - creating, 2-1
 - defining files to include, 2-5
 - example include file, 2-10
 - including files in, 2-5
 - user specified include files, 2-9
 - Netlist Translator, 1-1
 - netlists, 1-2
 - O
 - options for a netlist, setting up, 2-11
 - P
 - parasitic resimulation, 1-4
 - R
 - requirements, main, 1-6
 - RFIC Dynamic Link, 1-1
 - S
 - schematic, 1-4
 - setting up options for a netlist, 2-11
 - U
 - user specified include files, 2-9
-