



**Agilent Technologies**

**Advanced Design System 2002  
IFF Schematic Translation for Cadence**

**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

Cadence®, Analog Artist®, and Virtuoso® Schematic Composer are registered trademarks of Cadence Design Systems Incorporated.

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

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

# Contents

## 1 Introduction

Advanced Design System .....	1-1
Virtuoso Schematic Composer .....	1-1
The IFF Translator .....	1-2
IFF Interface Major Benefits .....	1-2
IFF Interface Major Features .....	1-2
Intended Audience.....	1-2
Main Requirements .....	1-3
The IFF Transfer Process .....	1-3
What's in this Manual.....	1-5
About Design Translation.....	1-6

## 2 Before Using the IFF Translators

Configuring your Software for IFF Translation .....	2-1
Software Requirements .....	2-1
License Requirements.....	2-2
Modifying the Cadence Initialization File .....	2-2
Setting up the System '.cdsenv' File.....	2-3
Verifying your Cadence IFF Configuration.....	2-4
Checking for the "basic" and "analogLib" Libraries.....	2-6
Understanding Component Library Requirements .....	2-7
Simulating a Design Transferred via IFF.....	2-8
Getting Help .....	2-8
Constructing Schematics for IFF Translation .....	2-9
Implementing Power Supply Connections.....	2-9
Separating Simulation Control Elements.....	2-9
Including Design Variables and Equations .....	2-9
Known Issues and Limitations .....	2-10

## 3 Importing IFF Schematic Files into ADS

Opening an ADS Project and Schematic Window.....	3-2
Importing an IFF Schematic File .....	3-3
Accessing the Schematic Import dialog .....	3-3
Specifying the Import File Selection .....	3-4
Selecting Import IFF Options.....	3-5
Completing the IFF Import.....	3-7

## 4 Exporting IFF Schematic Files from ADS

Exporting an IFF Schematic File .....	4-2
Accessing the Schematic Export dialog .....	4-2
Specifying the Export File Selection.....	4-3

Selecting Export IFF Options .....	4-4
Completing the IFF Export .....	4-9
<b>5 Importing IFF Schematic Files into DFII</b>	
Importing an IFF Schematic File .....	5-2
Accessing the Framework Input Transfer Form .....	5-2
Specifying the File Name and Setting Import Options.....	5-3
Completing the Import .....	5-5
<b>6 Exporting IFF Schematic Files from DFII</b>	
Exporting an IFF Schematic File .....	6-2
Accessing the Framework Output Transfer Form .....	6-2
Specifying the File Name and Setting Export Options .....	6-4
Completing the Export.....	6-7
Additional Export Options .....	6-7
Export library data for formsets .....	6-7
Configure IFF Variables.....	6-9
Framework Software Version.....	6-11
<b>Index</b>	

# Chapter 1: Introduction

Agilent Technologies and Cadence Design Systems both offer powerful EDA design tools. Many of today's design engineers prefer to use a combination of these tools to take advantage of the strengths of both design environments. Because of this desire to use multiple tools, Agilent Technologies has developed the *Intermediate File Format* translators as a method for transferring designs between the Agilent Technologies Advanced Design System (ADS) and Cadence environments.

Intermediate File Format (IFF) is an ASCII file format that is both platform and application independent. The file has a simple, line-oriented command structure with a fairly rich set of constructs, thus simplifying design transfer. The IFF translators offered by Agilent Technologies provide a means for transferring IFF files between Advanced Design System and third-party electronic design automation (EDA) tools such as Design Framework II (DFII) from Cadence Design Systems.

## Advanced Design System

Advanced Design System has been developed specifically to simulate the entire communications signal path. This unique solution integrates the widest variety of proven RF, DSP, and electromagnetic design tools into a single, flexible environment. Building on years of expertise developing new technologies for our EDA tools, such as Series IV and MDS, Advanced Design System provides a broad range of high-performance capability. This makes it easy to explore design ideas, then model the electrical and physical design of the best candidates.

## Virtuoso Schematic Composer

The Virtuoso Schematic Composer from Cadence Design Systems is a hierarchical design entry tool used by RFIC circuit designers. Useful for both analog and digital designs, the database created is accessible by the Cadence simulation and physical layout tools. The tool supports multi-sheet schematics, including cross-referencing, symbol creation, automatic HDL cell template generation, global nets and hierarchical property definition for most database objects. The tool also provides hierarchical checking of connectivity, consistency of different cell representations and label attachments.

## The IFF Translator

The IFF translator provided by Agilent Technologies is an EDA framework integration software product that stores circuit component and connectivity information. This product enables you to exchange design information between ADS and other EDA frameworks that provide an IFF interface. Agilent's IFF Interface enables you to generate IFF files from ADS Schematic information as well as receive IFF files from other design environments that support IFF translation. Agilent Technologies has also developed an IFF translator for use with Cadence DFII. The combination of these two translators enables you to share schematics between the two EDA design tools.

### IFF Interface Major Benefits

The IFF Interface enables you to translate schematic information between Advanced Design System and the Cadence design environment resulting in the following benefits:

- Avoiding re-entry of designs
- Helps eliminate the possibility of errors in design re-entry
- Time savings

### IFF Interface Major Features

Key features of the IFF Interface enable you to:

- Import IFF files into ADS from DFII and vice versa
- Export IFF files from ADS to DFII and vice versa
- Preserve circuit component and connectivity information during transfer

### Intended Audience

The audience intended for this manual consists of CAD System Administrators and RFIC Designers. It is assumed that the designer using the IFF translators has some working knowledge of both Advanced Design System and Cadence DFII.

# Main Requirements

To enable the successful IFF translation of an RFIC schematic design between ADS and Cadence DFII, you must ensure that:

- The link between IFF and the Cadence design environment is available and installed.
- The component libraries used for creating the schematic in ADS and DFII have been implemented to support translation via IFF. For more detailed information on library requirements, refer to [“Understanding Component Library Requirements” on page 2-7](#).

---

**Note** The link between IFF and DFII for IC design is developed and supported directly by Agilent Technologies. Agilent Technologies is generally not responsible for the development or support of the link between IFF and other 3rd party frameworks. Contact your 3rd party framework vendor for information on IFF translation availability.

---

## The IFF Transfer Process

Once the main requirements have been satisfied, your schematic can be transferred between Advanced Design System and Cadence DFII. Importing and exporting can be initiated from either EDA design environment.

The diagram shown in [Figure 1-1](#) describes the general use model for translating a design using the IFF Interface as it applies to Advanced Design System and Cadence DFII. The link between the two EDA environments is established via the ADS and DFII IFF Interface. The two component libraries in both ADS and DFII must be compatible to support an IFF translation.

For more information on the library requirements, refer to [Chapter 2, Before Using the IFF Translators](#).

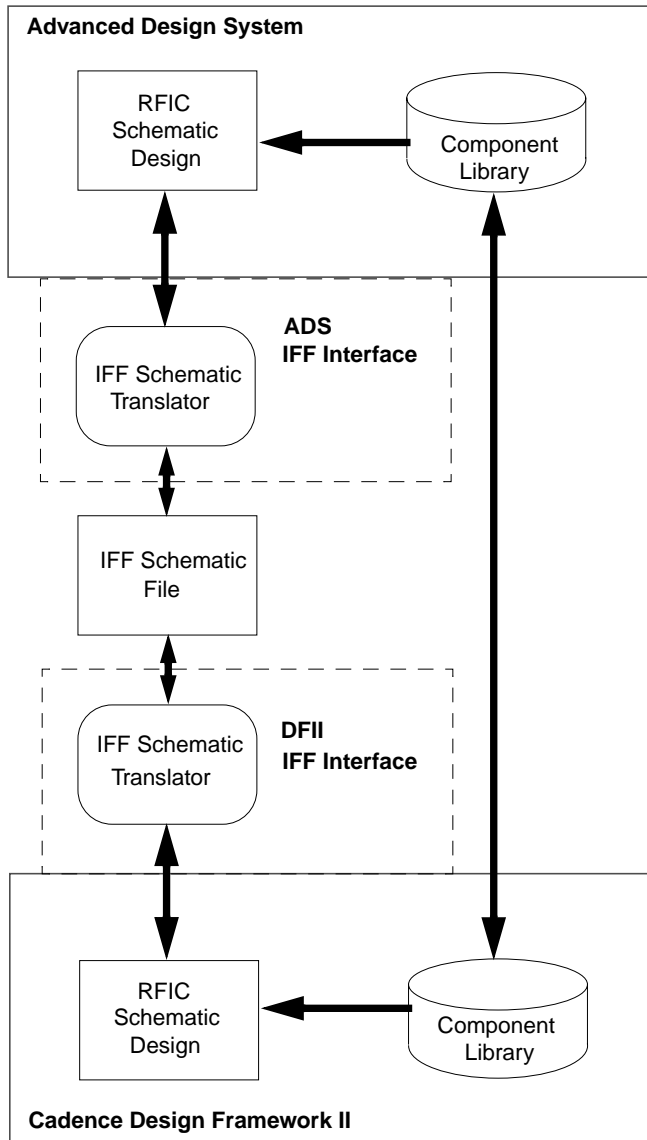


Figure 1-1. IFF Schematic Transfer Use Model



# What's in this Manual

The goal of this manual is to help you get started, providing relevant examples that teach you how to setup and use the software, and showing you where you can get more information as you need it. This manual contains:

- [Chapter 2, Before Using the IFF Translators](#) describes how to configure your Cadence design environment to support IFF translation between ADS and DFII. This chapter also discusses some of the issues related to IFF schematic translation in the ADS and DFII environments.
- [Chapter 3, Importing IFF Schematic Files into ADS](#) describes the procedure for importing an IFF file into an Advanced Design System Schematic.
- [Chapter 4, Exporting IFF Schematic Files from ADS](#) describes the procedure for exporting an ADS Schematic to an IFF file from Advanced Design System.
- [Chapter 5, Importing IFF Schematic Files into DFII](#) describes the procedure for importing an IFF file into DFII for use in Virtuoso Schematic Composer.
- [Chapter 6, Exporting IFF Schematic Files from DFII](#) describes the procedure for exporting a Composer Schematic to an IFF file from DFII.

## About Design Translation

It is highly recommended that the IFF Translator be used *only* for schematic translation in the presence of compatible component libraries in the two environments that have been developed to support this translation process. The IFF Translator can also be used to perform partial translation of a Cadence library into an ADS library and vice versa. To complete the translation task however, a significant amount of manual intervention is required.

This manual exclusively covers the schematic translation aspect of IFF translations. If you're interested in translating a component library from Cadence to ADS or vice versa, *Agilent Technologies Solution Services* can provide you with specialized tools and help. For more information, contact your Agilent Technologies sales representative.

For information about Spectre netlist to ADS translation, see the *Netlist Translator for SPICE and Spectre* manual.

# Chapter 2: Before Using the IFF Translators

Before using the IFF translators, there are several topics that must be addressed. This chapter is broken down into three main sections that cover these issues:

- [“Configuring your Software for IFF Translation” on page 2-1](#)
- [“Understanding Component Library Requirements” on page 2-7](#)
- [“Constructing Schematics for IFF Translation” on page 2-9](#)

After covering the information in this chapter, you’ll be ready to begin sharing your schematic files between ADS and DFII using the IFF translators.

## Configuring your Software for IFF Translation

This section describes the details related to software requirements and configuring the ADS and Cadence design environments to support IFF translation. Before attempting an IFF translation, the appropriate licenses must also be installed and the Cadence environment must be configured to operate with the IFF translator.

### Software Requirements

The IFF Translator described in this manual requires ADS 1.5 or later. IFF is supported by Cadence DFII versions 4.4.2\*, 4.4.3QSR1, 4.4.5 and 4.4.6. IFF is also supported on all UNIX operating system versions from Hewlett-Packard, Sun and IBM which run this Cadence software. Refer to [Table 2-1](#) for a summary of supported platforms. For additional information, please contact Cadence Design Systems, Inc.

Table 2-1. Supported Platforms

Cadence DFII Version	AIX4	HPUX10	HPUX11	SUN55	SUN56	SUN57
4.4.2*	yes	yes	no	yes	yes	no
4.4.3QSR1	yes	yes	yes	yes	yes	yes
4.4.5	yes	no	yes	yes	yes	yes
4.4.6	yes	no	yes	yes	yes	yes

\* Sub-version 100.1 or later.

## License Requirements

In addition to the mainstream Cadence (DFII, Composer Schematic, etc.) and Advanced Design System licenses, the following additional product licenses are required.

### Agilent Technologies Licenses

- Hp\_eda\_schematic\_trans

---

**Note** Before continuing, ensure that you have a valid license for the ADS schematic environment. For more information on ADS Licenses, refer to “*Setting up Licenses on UNIX Systems*” in the ADS Installation on UNIX Systems manual.

---

### Cadence Design Systems Licenses

- Virtuoso Schematic Composer (34500)

---

**Note** This and all other Cadence licenses must be purchased from Cadence Design Systems, Inc.

---

## Modifying the Cadence Initialization File

This section describes the information that must be added to the Cadence initialization file (*.cdsinit*) in order to configure DFII to run the IFF translator. Normally, only one *.cdsinit* file is read by DFII. DFII searches for the first *.cdsinit* file in the following order:

<code>\$CDS_INST_DIR/tools/dfii/local/.cdsinit</code>	(global Cadence startup directory)
<code>./cdsinit</code>	(local directory)
<code>\$HOME/.cdsinit</code>	(home directory)

Add the following lines to your Cadence initialization file (*.cdsinit*):

```
load(strcat(getShellEnvVar("HPEESOF_DIR")
"/links/tools/iff/cadence/composer/mdsinit.il"))
```

Adding the above information to your *.cdsinit* file loads the SKILL code that sets up the *IFF* pull-down menu on the Cadence CIW as well as other code necessary for performing IFF transfers between DFII and Advanced Design System.

---

**Note** SKILL is Cadence's C/lisp-like interpretive programming language for framework and database integration.

---

## Restarting Your Cadence Session

If you have already started your Cadence session and do not wish to interrupt it, source your *.cdsinit* file before attempting to use the IFF translator. For more information on Cadence initialization files (*.cdsinit*), refer to your Cadence documentation under “*Setting Cadence Environment Variables*”. If you're using version 4.4.5 or 4.4.6, you will need to restart your Cadence session to initiate the change.

## Setting up the System ‘.cdsenv’ File

A specific *.cdsenv* file must be created in the Cadence installation to support the operation of the IFF translator. To perform this operation, a script is provided that copies the default *.cdsenv* file from the ADS installation to the Cadence installation.

To install the *.cdsenv* file:

1. Change to the `$HPEESOF_DIR/links/tools/iff/cadence/composer` directory.
2. Locate the script called *setupCdsEnv*. This script copies the default *.cdsenv* file into the Cadence root currently set in your path. The Cadence root is determined by using the *cds\_root* command.

---

**Note** The *cds\_root* command is located in the `tools/bin` directory of the Cadence directory hierarchy. You must have the `tools/bin` directory in your path for the *setupCdsEnv* script to function properly.

---

3. Run the *setupCdsEnv* script. The *.cdsenv* file is copied to:

`<cds_root>/tools/dfII/etc/tools/iff/.cdsenv`

If the *iff* directory does not exist, it will be automatically created.

**Note** If you do not have this environment file installed, you will get messages similar to the following:

```
*WARNING* Cannot find /apps/cadence/4.4.3/tools.hppa/dfII/etc/tools/iff
directory to load environment variables
*WARNING* envGetVal: could not find tool[.partition] 'iff'
```

---

## About the Local '.cdsenv' File

There is also a local *.cdsenv* file that is created and updated every time you click **Apply** or **OK** in the IFF Import and Export dialog boxes. This file contains the settings of the various fields and check boxes and it can be found in the directory where you started the Cadence application. For example, if you change the scaling factor (see [“Scaling Factor for Cadence to Target System” on page 6-9](#)) from 200 to 250 and then click **Apply**, the setting is saved and will appear the next time you open either the IFF Import or Export dialog box. These settings can become system defaults by manually editing the system *.cdsenv* file, located in the *<cds\_root>/tools/dfII/etc/tools/iff* directory, setting each of the variables in the *.cdsenv* file to the desired setting.

## Verifying your Cadence IFF Configuration

After meeting your license requirements and modifying the Cadence initialization file, you can run Cadence DFII to ensure that your configuration was successful. To launch DFII and verify your setup:

1. In a terminal window, change to the appropriate directory.
2. Run DFII by typing the appropriate command (typically *icms* or *icfb*). The Cadence *Command Interpreter Window* (CIW) appears.

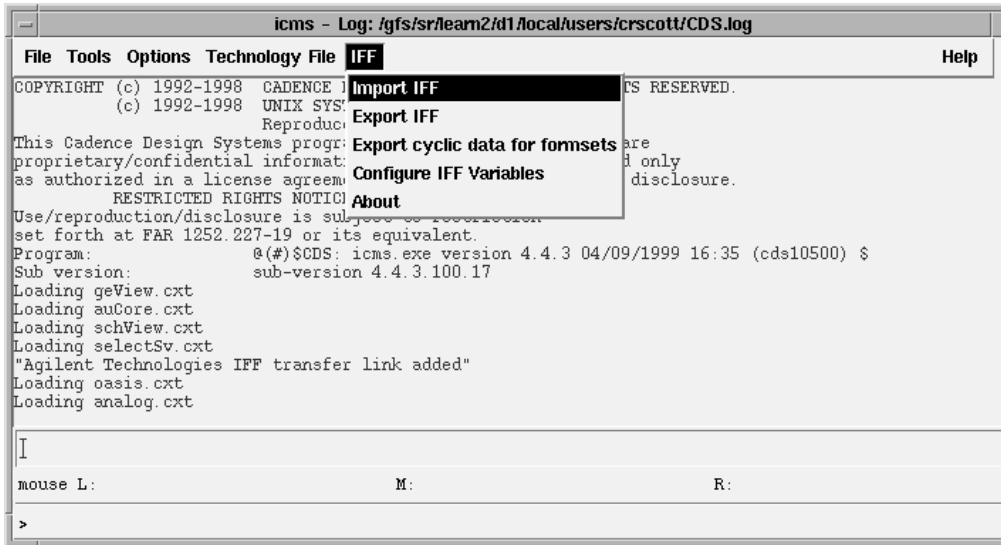


Figure 2-1. Cadence CIW Window

3. Check that the *IFF* menu item is displayed in the CIW menu bar as shown in [Figure 2-1](#). For detailed information on the available command options, refer to [Chapter 5, Importing IFF Schematic Files into DFII](#) and [Chapter 6, Exporting IFF Schematic Files from DFII](#).

---

**Note** Non-standard or customized start-up scripts for Cadence Design Framework II may not be supported. If you have difficulties, contact your system administrator.

---

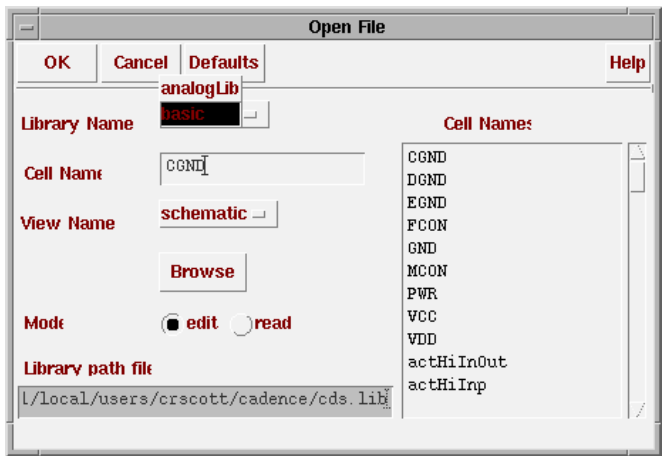
## Checking for the “basic” and “analogLib” Libraries

The *basic* library must be installed in Cadence DFII before attempting an IFF translation. If the *basic* library is not installed in the Cadence *cdslib* directory, there are potential transfer issues related to pin mapping. Fundamentally, DFII will not be able to generate pins when translating an IFF schematic from ADS without having the *basic* library available in Cadence.

If the *analogLib* library is not installed in the Cadence *cdslib* directory, there are potential transfer issues related to ground symbols and connector components. If you’ve developed your own ground symbol and placed it in an existing library, ensure that you activate the *Export standard connectors as components* option when exporting an IFF schematic file from DFII to ADS. The *Export standard connectors as components* option can be found in the *Framework Output Transfer Form*. For more information on this option, refer to [“Accessing the Framework Output Transfer Form” on page 6-2](#). Also ensure that you use your own connector components in ADS, as opposed to the standard ADS connectors (i.e. *Port* and *Ground*), to avoid any potential problems.

To ensure that the *basic* and *analogLib* libraries are available from the Cadence CIW:

1. Choose **File > Open**. The Open File dialog box appears.



2. In the *Open File* dialog box, click the *Library Name* pull-down menu item. If the *basic* and *analogLib* libraries are installed, they should both be available in this menu.



3. If the *basic* and/or *analogLib* libraries are not installed, work with your system administrator to obtain a copy of the missing library or libraries.

Copy the *basic* library to:

```
<Cadence_install_dir>/tools/dfII/etc/cdslib/basic
```

Copy the *analogLib* library to:

```
<Cadence_install_dir>/tools/dfII/etc/cdslib/analogLib
```

4. Repeat steps 1 and 2 to verify that the libraries are now available from the Open File dialog box.

## Understanding Component Library Requirements

The fundamental library requirement for a successful IFF transfer is that both the Advanced Design System and Cadence Design Framework II contain compatible component libraries. This section provides an overview of compatible libraries and discusses the issues related to simulating your design.

Compatible libraries between two design frameworks can be considered equivalent component libraries for all practical purposes. While two EDA frameworks may have some differences in the way component information is handled, the fundamental component parameters for each primitive component must map each other to have the same function in one framework as it does in the other. Components and parameters such as component names, symbols, size and shape, symbol pin locations, etc. in the source framework must all map to corresponding components and parameters in the target framework. The diagram below displays an example of two equivalent, or compatible component libraries within two separate EDA frameworks.

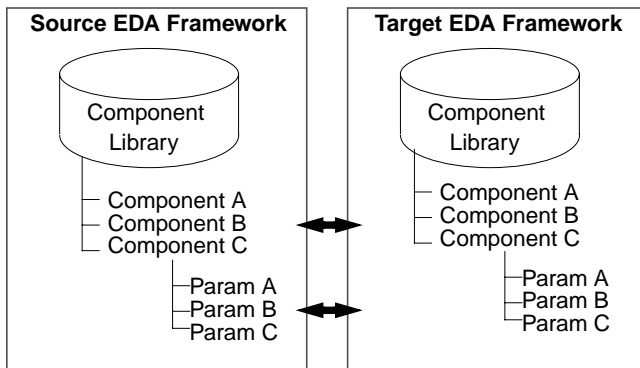


Figure 2-2. Compatible Component Libraries

## Simulating a Design Transferred via IFF

A circuit can be transferred from a source environment into a target environment via IFF translator and be simulated in the target environment only if the component libraries that have been used to create the circuit are compatible between the two environments.

## Getting Help

Due to the detailed nature of creating compatible component libraries, it is recommended that you consult Agilent EEsof-EDA Solution Services for more information. Solution Services can provide training on how to configure compatible component libraries as well as complete library development solutions. Contact your local Agilent Technologies sales representative for more information on working with Solution Services.

# Constructing Schematics for IFF Translation

The IFF schematic translator module enables bi-directional, fully hierarchical schematic transfer between the ADS and Cadence DFII Frameworks. You can edit your schematic in either environment, and when you transfer edited material to the other environment, all edits are preserved, including property changes.

Before attempting to transfer a schematic via IFF, ensure that all component libraries are compatible in all frameworks. This step is essential for a successful IFF transfer. For more information on compatible libraries, refer to [“Understanding Component Library Requirements” on page 2-7](#).

To have a successful IFF transfer, it is required that only components that are contained within the compatible libraries be used in your designs. A hierarchical design approach is recommended for the implementation of RFIC schematic designs in ADS.

## Implementing Power Supply Connections

Power supply connections (*power rails*) should be implemented through port construction in the subcircuits or through global named connections.

## Separating Simulation Control Elements

Simulation control elements are not transferred between ADS and DFII. Because of the different way the two environments handle simulation setup, it is recommended that your simulation control elements in Advanced Design System be separated from your circuit schematic information using a hierarchical approach (i.e. top level contains simulation control and instance(s) of subcircuits containing DUT). This enables you to transfer only the schematic information in the subcircuit and will require you to set up simulation controls independently in the Cadence environment. For more information, refer to *“Creating Hierarchical Designs”* in the ADS User’s Guide.

## Including Design Variables and Equations

If the design being transferred from ADS to DFII contains design variables or equations, the same variable should be manually set to the appropriate value or equation in the Cadence environment.

**Note** Advanced Design System equations are transferred in the IFF file; however, they are not translated to a Cadence equivalent.

---

## Known Issues and Limitations

There are several known issues or limitations that you should become familiar with before attempting to perform an IFF translation.

- Only one schematic is allowed per cell in Advanced Design System.
- Extracted views are not supported by the IFF translator.
- Parameterized subnetworks require special setup not covered in this manual. It is recommended that parameterized subnetworks not be used if IFF is used to transfer designs between ADS and Cadence.

For information, see the *Netlist Translator for SPICE and Spectre* manual.

- Inherited Connections are not supported in IFF transfers in ADS version 1.5 SP1A or earlier.
- Inherited Connections are supported in ADS via Power Pins. It is necessary to manually place Power Pins on symbols in ADS. IFF library transfers will not set up pins that utilize the ADS version of Inherited Connections.

If IFF is used to transfer designs between ADS and Cadence, it is recommended that symbols use explicitly defined pins rather than Inherited Connections.

- ADS ports to have unique names. If multiple ports with the same name are used within a Cadence schematic, the IFF exporter will keep the first port with the name. All subsequent ports with the common name will be converted to wire labels.
- Area pins are not supported in ADS. It is important to have wires in your Cadence design connect to the center point of a symbol pin.

- Design variable in Cadence are not transferred to ADS via IFF. It is necessary to manually create a VAR component in ADS that will contain the Cadence design variables. Additionally, VAR components in ADS are not transferred to Cadence design variables. If VAR components are set up in ADS, you must manually create design variables in Cadence.



# Chapter 3: Importing IFF Schematic Files into ADS

This chapter describes the procedure for importing an Intermediate File Format (IFF) schematic file into Advanced Design System.

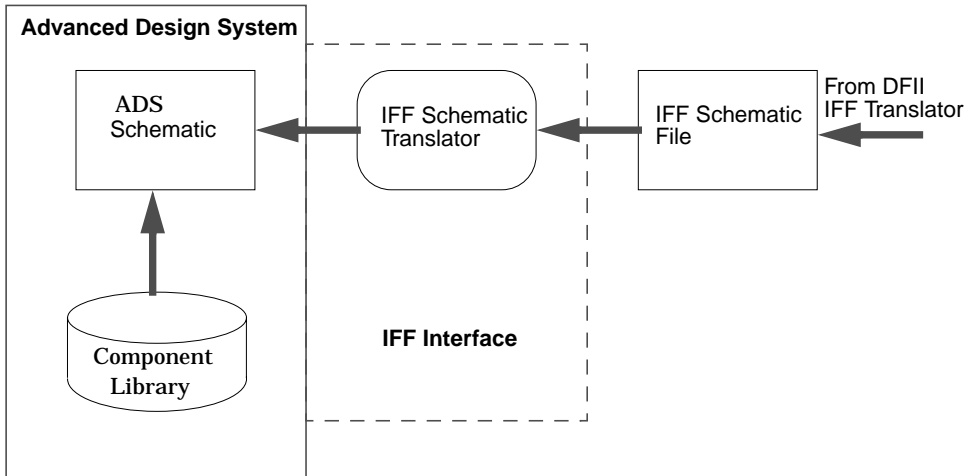


Figure 3-1. IFF Schematic Import in ADS

The basic procedure for importing an IFF schematic file into ADS can be broken down into several simple steps:

1. [“Opening an ADS Project and Schematic Window” on page 3-2](#)
2. [“Accessing the Schematic Import dialog” on page 3-3](#)
3. [“Specifying the Import File Selection” on page 3-4](#)
4. [“Selecting Import IFF Options” on page 3-5](#)
5. [“Completing the IFF Import” on page 3-7](#)

## Opening an ADS Project and Schematic Window

Open a project in ADS before attempting to import your design. Working in project directories enables the translator to organize design files in the standard ADS file structure. The import option will not be active in the File menu unless you open a project. From the ADS Main window:

1. Choose **File > New Project** to open a new project or **File > Open Project** to open an existing project.

For more information on working in project directories, refer to “*Managing Projects and Designs*” in the ADS User’s Guide.

2. Before invoking the import procedure, close any open designs to remove any active designs from memory. In the ADS Main window, choose **File > Close All**.
3. Open a new ADS Schematic window by clicking the *New Schematic Window* icon in the ADS Main window tool bar.



A new ADS *untitled* schematic window appears.



# Importing an IFF Schematic File

This section describes the procedure for importing an IFF schematic file into Advanced Design System using the IFF Translator's User Interface. Before importing an IFF schematic file, refer to [“Opening an ADS Project and Schematic Window” on page 3-2.](#)

---

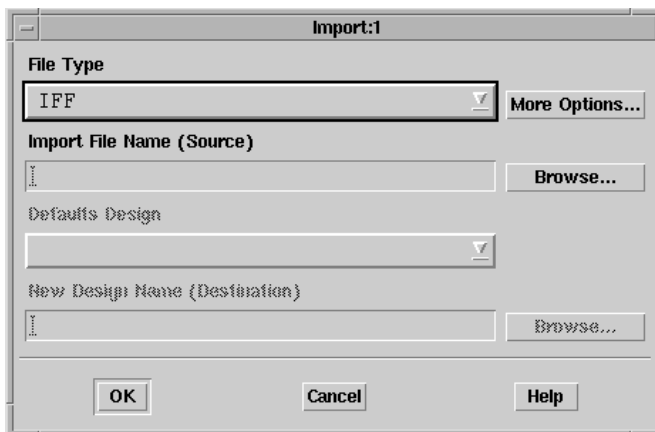
**Caution** When a design being imported has the same name as an existing design, the existing design will be *overwritten with no questions asked*. The designs are not merged together.

---

## Accessing the Schematic Import dialog

To access the schematic import dialog, launch your import from an ADS *Schematic* window.

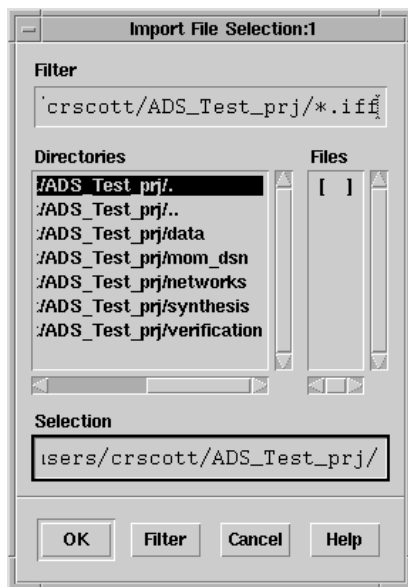
Choose **File > Import**. The schematic *Import* dialog box appears.



## Specifying the Import File Selection

In the schematic *Import* dialog box, choose the type of file to import, specify the filename, and supply other basic information needed by the translator.

1. In the *Import* dialog box, select *IFF* from the *File Type* drop-down list if it isn't already displayed.
2. To specify the path and filename of the file you want to import, click **Browse** in the *Import* dialog box. The *Import File Selection* dialog box appears.

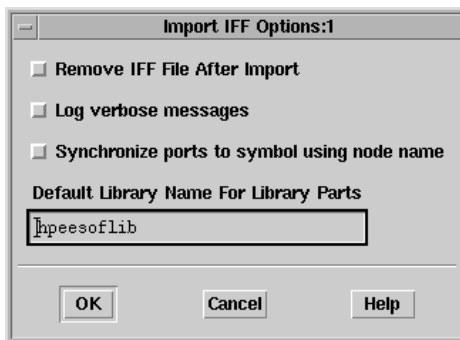


3. Double-click as needed to locate the directory containing your IFF file in the *Directories* field, then click the file in the *Files* field. Alternatively, you can enter the full path and file name in the *Selection* field.
4. After selecting the design you want to import, click **OK**. You are returned to the *Import* dialog box and the selected filename appears in the field labeled *Import File Name (Source)*.
5. Click **More Options** to define the import options. The *Import IFF Options* dialog box appears.

## Selecting Import IFF Options

This section describes the choices available in the *Import IFF Options* dialog box. Import options for other file formats are detailed in the ADS “*Importing and Exporting Designs*” manual.

To access the schematic *Import IFF Options* dialog box from the Import dialog, click **More Options**. The schematic *Import IFF Options* dialog box appears.



In the *Import IFF Options* dialog box, select the appropriate options for your translation using the information below.

### Remove IFF File After Import

Set the *Remove IFF File After Import* as desired. When selected, the *.iff* file is removed once the IFF file is successfully imported. This option is deselected as the default, and the IFF file remains after import.

### Log verbose messages

When the *Log verbose messages* option is selected, all translation information is recorded in the *iffolib.log* file resulting in step-by-step description of what happened internally during your translation. This option is primarily intended to be used as a diagnostics tool so the default mode for this option is deselected. Note that error and warning messages will always appear in your status window regardless of this selection.

## Synchronize ports to symbol using node name

The *Synchronize ports to symbol using node name* option is not supported in the ADS 1.5 or 2001 IFF importer. Ensure that this checkbox is deactivated.

## Default Library Name For Library Parts

When the IFF file does not specify a library name for a component that needs to be created, the library name specified in this field is used. This is only necessary for environments that do not support the concept of a library. Cadence will always provide a library name.

---

**Note** The *Default Library Name For Library Parts* field is identical to the field of the same name in the *Export IFF Options* dialog box. Changes made to this field will modify the contents of the field in the *Export IFF Options* dialog box. For more information on Exporting IFF Schematic files, refer to [Chapter 4, Exporting IFF Schematic Files from ADS](#).

---

## About Component Libraries

A component library in Advanced Design System consists of a collection of component definitions. Each primitive component has an associated component name, symbol and predefined component parameters that include relevant physical and electrical characteristics.

The IFF translator can be used as the initial step in creating an ADS component library however; this topic is outside of the scope of this manual. Creating an ADS component library using IFF requires specialized tools and training. If you're interested in learning more about this topic, contact Agilent EEsof-EDA's Solution Services.

## Completing the IFF Import

After specifying the IFF import options, click **OK** in the *Import IFF Options* dialog box to save your settings or **Cancel** to retain the default settings. After clicking **OK**, you are returned to the *Import* dialog box. Click **OK** to begin the translation.

When translation is complete, an *Information Message* dialog box appears stating, *IFF Import Completed*. The *IFF Import* log window also appears. Review the log message searching for any error messages or warnings generated during export.



# Chapter 4: Exporting IFF Schematic Files from ADS

This chapter describes the procedure for exporting an Intermediate File Format (IFF) schematic from Advanced Design System.

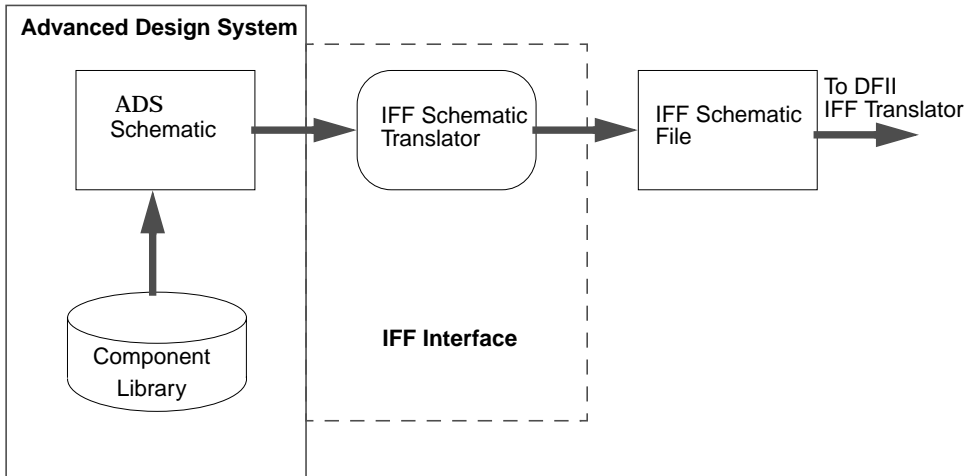


Figure 4-1. IFF Schematic Export from ADS

The basic procedure for exporting an IFF file from ADS can be broken down into several simple steps:

1. [“Opening an ADS Project and Schematic Window” on page 3-2](#)
2. [“Accessing the Schematic Export dialog” on page 4-2](#)
3. [“Specifying the Export File Selection” on page 4-3](#)
4. [“Selecting Export IFF Options” on page 4-4](#)
5. [“Completing the IFF Export” on page 4-9](#)

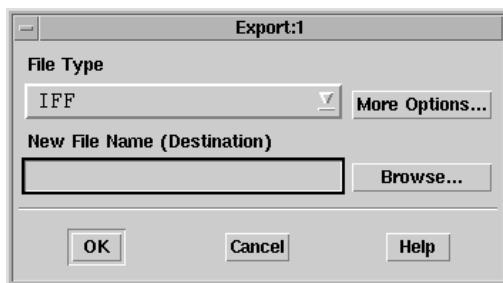
## Exporting an IFF Schematic File

This section describes the procedure for exporting an IFF schematic file from Advanced Design System using the IFF Translator's User Interface. Before exporting an IFF schematic file, refer to [“Opening an ADS Project and Schematic Window” on page 3-2.](#)

### Accessing the Schematic Export dialog

To access the schematic export dialog, launch your export from an ADS *Schematic* window.

1. Open your ADS Schematic Design in the ADS Schematic window.
2. Choose **File > Export**. The schematic *Export* dialog box appears.

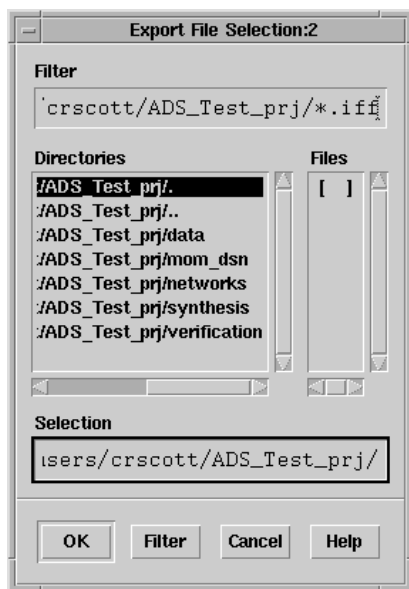




## Specifying the Export File Selection

In the schematic *Export* dialog box, choose the type of file to export, specify the new file name, and supply other basic information needed by the translator.

1. In the *Export* dialog box, select *IFF* from the *File Type* drop-down list if it isn't already displayed.
2. To specify the path and filename of the file you want to import, click **Browse** in the *Export* dialog box. The *Export File Selection* dialog box appears.

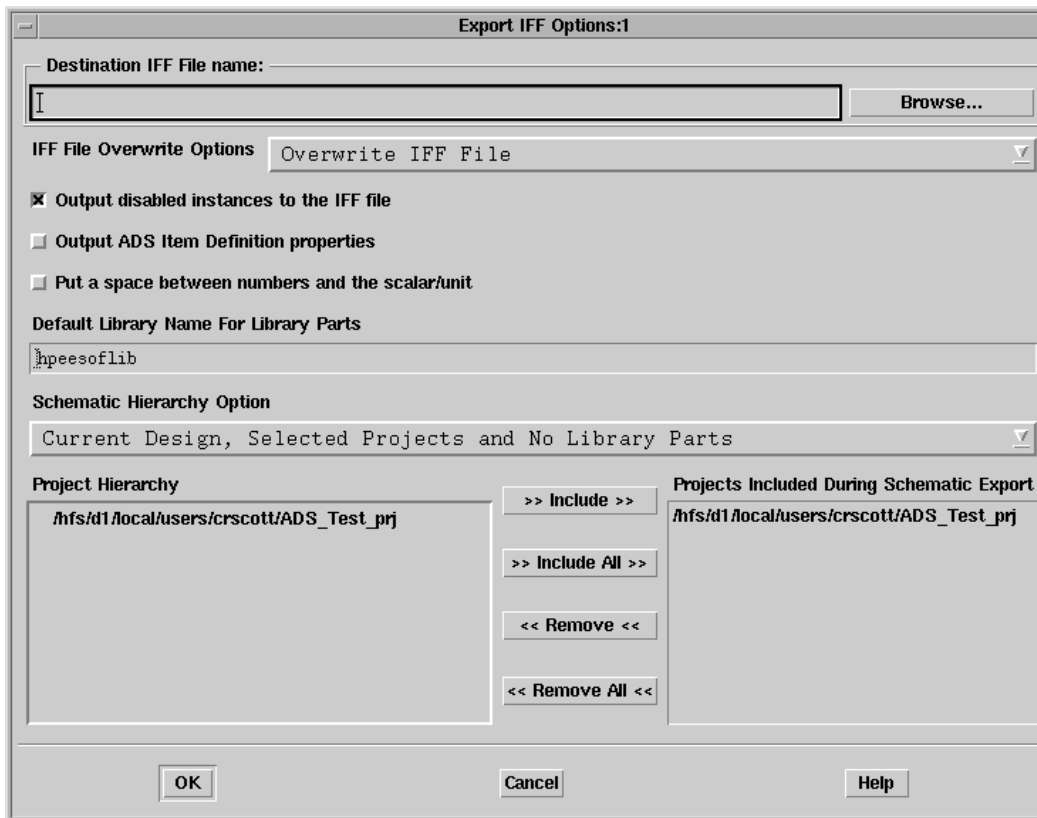


3. Double-click as needed to locate the directory for your exported IFF schematic file in the *Directories* field then enter the new file name in the *Selection* field. Alternatively, you can enter the full path and file name in the *Selection* field.
4. After selecting the new location and name of your design, click **OK**. You are returned to the *Export* dialog box and the selected path and file name appear in the field labeled *New File Name (Destination)*. When translated, the file name is automatically appended with suffix *.iff*. Note that the *.iff* extension is only added if the file name doesn't already contain it.
5. Click **More Options** to define the export options. The *Export IFF Options* dialog box appears.

## Selecting Export IFF Options

This section describes the choices available in the schematic *Export IFF Options* dialog box.

To access the schematic *Export IFF Options* dialog box from the Export dialog, click **More Options**. The schematic *Export IFF Options* dialog box appears.

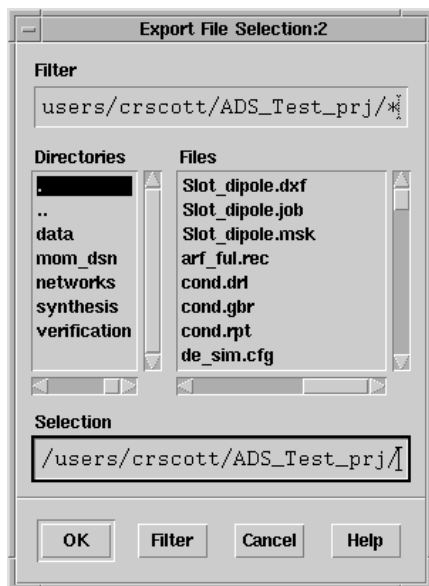


In the *Export IFF Options* dialog box, select the appropriate options for your translation using the information below.

## Destination IFF File name

Use this field to specify the full path and filename for the IFF file destination. To set the directory path and filename:

1. Click **Browse** in the *Export IFF Options* dialog box. The *Export File Selection* dialog box appears.



2. Double-click as needed to locate the directory to place your exported IFF file in the *Directories* field. If a file that you want to *overwrite* already exists in this directory, click the filename in the *Files* field. Alternatively, you can append the file name to the directory path in the *Selection* field.
3. After selecting the destination for the design you want to export, click **OK**. You are returned to the *Export IFF Options* dialog box and the selected path and filename appears in the field labeled *Destination IFF File name* field.

## IFF File Overwrite Options

*Overwrite IFF File* - When writing to an existing file, the contents of that file are overwritten. While this is not the default setting, it is generally the preferred setting.

*Append to IFF File* - When writing to an existing file, the new file is appended to the existing file. This is the default setting. The IFF file is not overwritten during the ADS export by default. To transfer multiple designs, simply use the same file multiple times. Each design is created in turn. This method enables you to transfer a limited amount of hierarchy when you don't want to overwrite reference elements. To implement this method, set the hierarchy level to *None*, then export one sub-network followed by the main design. No IFF data is generated for any of the referenced components in either design, so the only two circuits overwritten during an import are the two designs transferred.

## **Output disabled instances to the IFF file**

When this option is selected, if an instance is disabled in the schematic, it will still be output into the IFF file. If the checkbox is deselected (default), disabled instances will not be exported. This option can be utilized to omit certain components from being transferred to remote environments that might not support the components (e.g. disable the simulation components prior to creating an IFF file to send to Cadence, which does not have any definitions for the simulator components). Activate this option if you want to get everything. Deactivate this option if you want to filter out the unused/unwanted components.

## **Output ADS Item Definition properties**

When this option is selected, ADS Item Definition properties are utilized to recreate the information necessary to simulate a component for ADS. For example, if you have parameters on a resistor, some Item Definition properties are created in the IFF file (e.g. R\_ADS\_UNIT=1), which allow the IFF importer to exactly recreate the component as it exists in ADS. However, other tools will not recognize the Item Definition parameters, and may misinterpret the properties as being separate. If library symbols are being exported to other environments that do not recognize the ADS Item Definition parameters, the option should be turned off. This option is deselected by default.

## **Put a space between numbers and the scalar/unit**

When this option is selected, parameter values are exported as they normally appear in ADS (i.e. with a space between the number and the scalar, e.g. "1 pF"). If the checkbox is deactivated, the exporter converts the values into the IFF value specification, which is to have no space between a number and a scalar (e.g. "1pF"). Ideally, an IFF exporter should interpret either form of number, and set the value

internally to whatever is normal for that environment. Some environments (e.g. Mentor Graphics) do not interpret the IFF property values in any way. For Mentor IC, this means the numbers need to have no space in them, because, when they are used within SPICE simulations, the space will cause syntax errors in the simulator. However, for Mentor Board, they require the ADS components to have a space in them, because the RF Architect ADS library is set up to expect values to have a space between a number and a scalar/unit.

If you are exporting designs to Mentor Boardstation, you must select this option for IFF imports to work into their environment. An additional issue can come up if you create variables, and then assign scalar values to the variable (e.g. "R1 kOhms"). When this is exported, if the option is not set, it would convert to "R1koh", which could no longer be interpreted correctly. Note that this second option is considered bad practice (the scalar should be included in the variable value for R1, and no units should be specified); however, ADS does allow you to format variables in this way. If you are using variables in this way, you must set this option to true. This option is deselected by default.

## Default Library Name for Library Parts

When the IFF file does not specify a library name for a component that needs to be created, the library name specified in this field is used. This is necessary for environments that do not support the concept of a library.

Design objects are stored in a group that uses the same name as the project directory, but library parts are stored in either the default library *hpeesoflib* or a library that you specify. The default library name can contain only alpha numeric characters.

---

**Note** The *Default Library Name For Library Parts* field is identical to the field of the same name in the *Import IFF Options* dialog box. Changes made to this field will modify the contents of the field in the *Import IFF Options* dialog box.

---

## About Component Libraries

A component library in ADS consists of a collection of component definitions. Each primitive component has an associated component name, symbol and predefined component parameters that include relevant physical and electrical characteristics.

The IFF Translator can be used as the initial step in creating an ADS component library however, this topic is outside of the scope of this manual. Creating an ADS component library using IFF requires specialized tools and training. If you're

interested in learning more about this topic, contact Agilent EEsof-EDA's Solution Services.

## Schematic Hierarchy Option

The *Schematic Hierarchy Option* drop-down list enables you to establish how much of the schematic hierarchy is exported:

**Current Design Only** Write current level only. Complete design information for the current design is exported. Instance-specific information (parameter values and coordinates identifying position) is also exported. Detailed definitions of a referenced design are not exported.

**Current Design, Selected Projects and No Library Parts** Complete design information for the current design is exported. Referenced designs that reside in a project selected for inclusion during export and are part of the current design's hierarchy are also exported. Library parts are not exported. This is the default setting.

**Current Design, Selected Projects and All Library Parts** Complete design information for the current design is exported. Referenced designs that reside in a project selected for inclusion during export and are part of the current design's hierarchy are also exported. In addition, library parts are exported.

## Project Hierarchy

Displays the current project. If hierarchical, all included projects are listed in the appropriate order.

## Projects Included During Schematic Export

This field contains the projects for which schematic design information is exported. You can customize this list if the current project is hierarchical.

To add a project to this list:

1. In the *Project Hierarchy* list, click the desired project.
2. Click the **Include** button. The project is added to the *Projects Included* list.

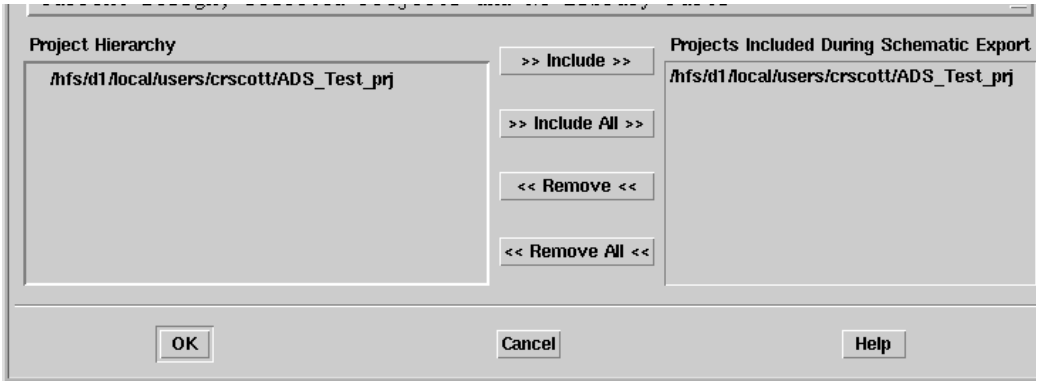
To include all projects, click **Include All**.

To remove a project from the *Projects Included* list:

1. In the *Projects Included* list, click the entry you want to remove.

2. Click the **Remove** button. The project is removed from the list.

To remove all entries from the *Projects Included* list, click **Remove All**.



This is an example of using the *Project Hierarchy* and *Projects Included During Schematic Export* fields. First you make a project called *Proj\_A* that includes several designs. Then you make another project called *Proj\_B* and you want to reuse some of the designs from project *Proj\_A* in project *Proj\_B*. You can then include *Proj\_A* in *Proj\_B* by using the **Include** button to have access to all the designs in *Proj\_A* after your export is complete.

## Completing the IFF Export

After specifying the IFF export options, click **OK** in the *Export IFF Options* dialog box to save your settings or **Cancel** to retain the default settings. After clicking **OK**, you are returned to the *Export* dialog box. Click **OK** to begin the translation.

When translation is complete, an *Information Message* dialog box appears stating, *IFF Export Completed*. The IFF Export log window also appears. Review the log message searching for any error messages or warnings generated during export.





# Chapter 5: Importing IFF Schematic Files into DFII

This chapter describes the procedure for importing an Intermediate File Format (IFF) schematic file from the Cadence CIW.

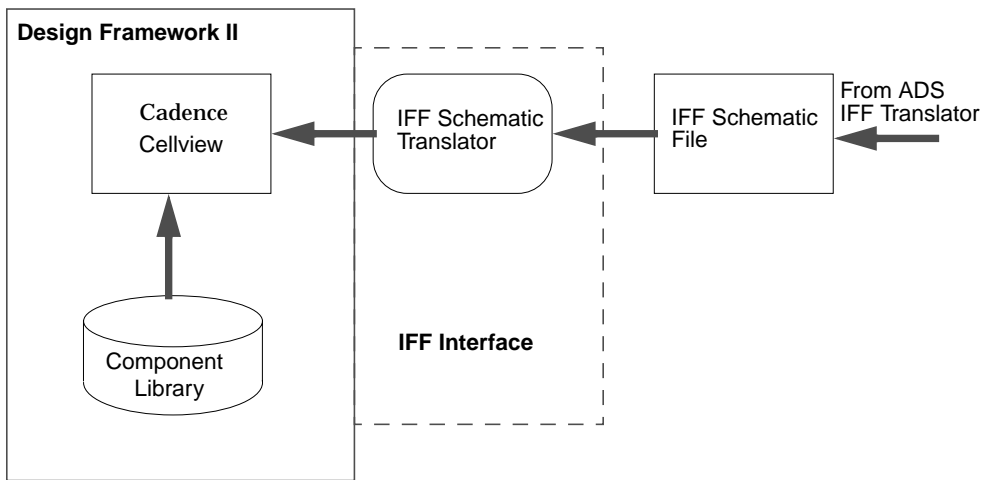


Figure 5-1. IFF Schematic Import into DFII

Before attempting to import an IFF file into DFII, refer to [Chapter 2, Before Using the IFF Translators](#). The basic procedure for importing an IFF schematic file into DFII can be broken down into several simple steps:

1. [“Accessing the Framework Input Transfer Form” on page 5-2](#)
2. [“Specifying the File Name and Setting Import Options” on page 5-3](#)
3. [“Completing the Import” on page 5-5](#)

## Importing an IFF Schematic File

This section describes the procedure for importing an IFF schematic file into the Cadence DFII using the IFF Translator's User Interface. Before importing an IFF schematic file, refer to the beginning of [Chapter 5, Importing IFF Schematic Files into DFII](#).

The menu items described in this section are only available after the `.cdsinit` file has been configured for an Intermediate File Format file transfer between DFII and ADS. For more information, refer to [Chapter 2, Before Using the IFF Translators](#).

### Accessing the Framework Input Transfer Form

To access the schematic import dialog, launch your import from the Cadence *Command Interpreter Window* (CIW).

1. In a terminal window, change to the appropriate directory.
2. Run DFII by typing the appropriate command (typically `icms` or `msfb`). The Cadence *Command Interpreter Window* (CIW) appears.

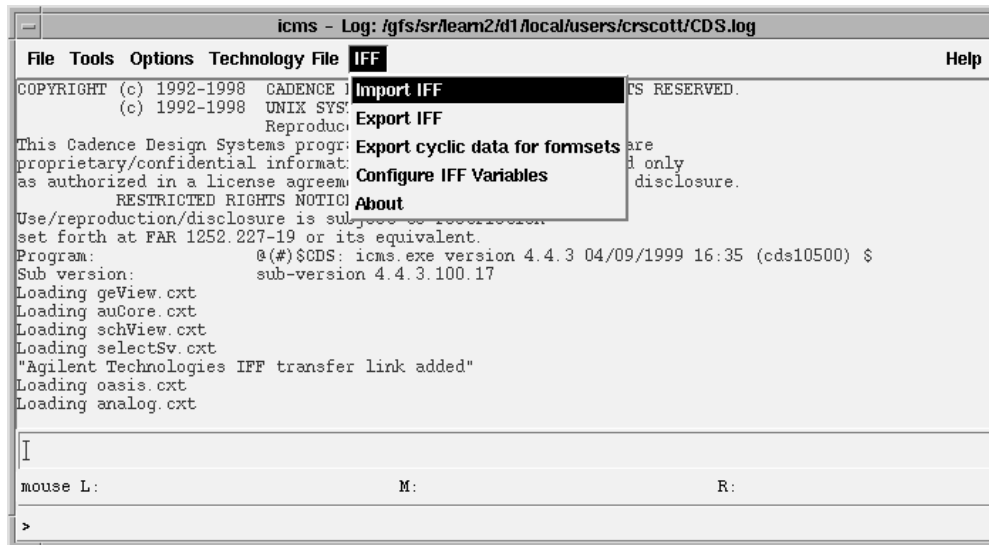
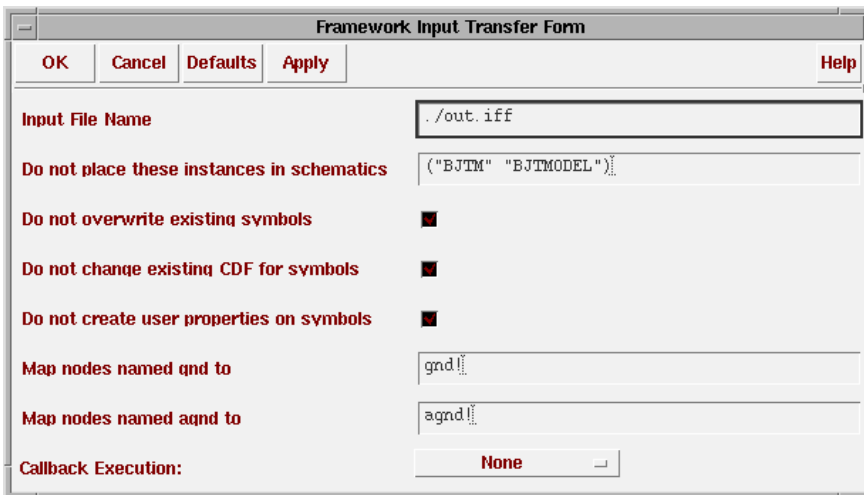


Figure 5-2. Cadence CIW Window

3. From the Cadence CIW, choose **IFF > Import IFF**. The *Framework Input Transfer Form* appears.



The screenshot shows the 'Framework Input Transfer Form' dialog box. It has a title bar with the text 'Framework Input Transfer Form'. Below the title bar are four buttons: 'OK', 'Cancel', 'Defaults', and 'Apply', followed by a 'Help' button on the right. The main area of the dialog contains several fields and checkboxes:

- Input File Name:** A text field containing the path `./out.iff`.
- Do not place these instances in schematics:** A text field containing `("BJTM" "BJTMODEL")`.
- Do not overwrite existing symbols:** A checked checkbox.
- Do not change existing CDF for symbols:** A checked checkbox.
- Do not create user properties on symbols:** A checked checkbox.
- Map nodes named gnd to:** A text field containing `gnd`.
- Map nodes named agnd to:** A text field containing `agnd`.
- Callback Execution:** A dropdown menu currently set to `None`.

## Specifying the File Name and Setting Import Options

In the *Framework Input Transfer Form*, specify the file name, enter the names of any components that you want ignored and supply other basic information needed by the translator.

### Input File Name

In the *Input File Name* field, enter the full path and file name of the IFF file generated in Advanced Design System that you want to import into Cadence.

### Do not place these instances in schematics

In the *Do not place these instances in schematics* field, enter the names of any components that you want Cadence to ignore during the import. If the IFF file contains symbol data for these components, the symbols are created, however; they are not placed in the Cadence schematic. This option enables you to prevent the import of ADS simulation and model components which do not have equivalent components in DFII.

## Do not overwrite existing symbols

Set the *Do not overwrite existing symbols* checkbox as desired. When this option is selected, existing symbols are preserved rather than overwritten. If a symbol does not exist prior to the transfer, it is created during the import. The default value is selected.

## Do not change existing CDF for symbols

Set the *Do not change existing CDF for symbols* checkbox as desired. When this option is selected, Cadence cell Component Description Formats (CDFs) are not updated during the import. The default value is selected.

## Do not create user properties on symbols

When this option is selected, user instance properties are not created during the import. The default value is selected and it is recommended to keep the default setting when using ADS 1.3 and above to export the IFF file.

## Map nodes named gnd

In the *Map nodes named gnd to* field, enter the ground node name mapping information. The default value is the *gnd!* used by Advanced Design System.

## Map nodes named agnd

In the *Map nodes named agnd to* field, enter the analog ground node name mapping information. The default value is the *agnd!* used by Advanced Design System.

## Callback Execution

There are three options available from the *Callback Execution* pull-down menu.

- **All** This option executes the evaluation of all functions or expressions used.
- **Changed Parameters** This option executes the evaluation of only those functions or expressions that have changed.
- **None** This option disables any callback execution and is the default setting.

## Completing the Import

After you have completed the *Framework Input Transfer Form*, click **Apply** in the *Framework Input Transfer Form* to save your settings or **Defaults** to retain the default settings. You can also click **Cancel** to abort the import operation. Click **OK** to begin the translation.

After the translation begins, all import log information is displayed in the Cadence CIW. One of the log lines will display what was created. Any warnings or error messages encountered during the translation are also displayed in the log messages.

While cells are created in the appropriate Cadence libraries, the cell view is not automatically displayed in the CIW. If full hierarchy was specified for the Advanced Design System export, existing schematics common to both program environments are overwritten on import. Symbols are overwritten only when you deselect the *Do not overwrite existing symbols* checkbox in the *Framework Input Transfer Form*. If you have deselected this option, symbols only are overwritten; existing CDF data is not overwritten. Only data needing to be overwritten is overwritten; and new versions are generated for all Cadence cell views.



# Chapter 6: Exporting IFF Schematic Files from DFII

This chapter describes the procedure for exporting an Intermediate File Format (IFF) schematic file from the Cadence CIW.

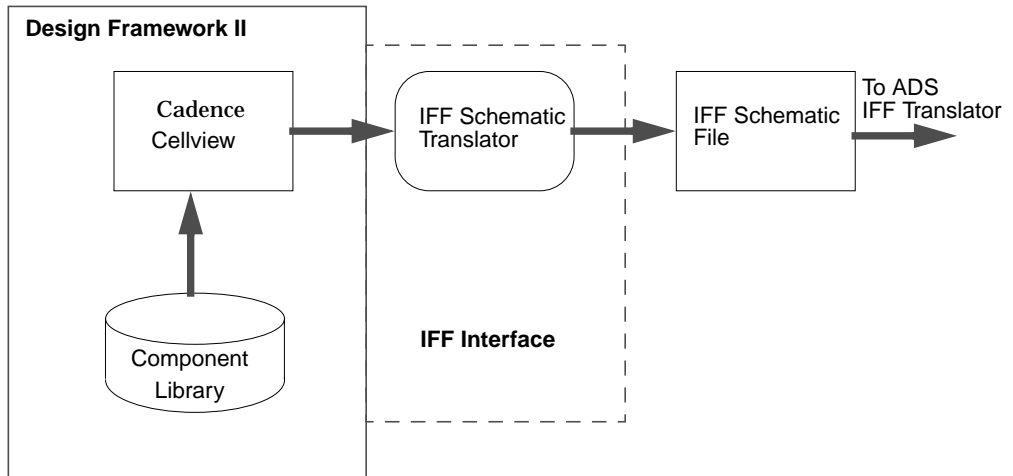


Figure 6-1. IFF Schematic Export from DFII

Before attempting to export an IFF file from DFII, refer to [Chapter 2, Before Using the IFF Translators](#).

The basic procedure for exporting an IFF schematic file from DFII can be broken down into several simple steps:

1. "Accessing the Framework Output Transfer Form" on page 6-2
2. "Specifying the File Name and Setting Export Options" on page 6-4
3. "Completing the Export" on page 6-7

## Exporting an IFF Schematic File

This section describes the procedure for exporting an IFF schematic file from Design Framework II using the IFF Translator from the Cadence CIW. It is a recommended practice to export only design schematic information. This implies excluding everything else from the export process such as simulation control blocks as well as any other elements that do not map directly to Advanced Design System. Such elements should exist at the top cell (symbol) view. To avoid complications, descend one level into this symbol view before beginning the export process.

The menu items described in this section are only available after the `.cdsinit` file has been configured for an Intermediate File Format file transfer between DFII and ADS. For more information, refer to [Chapter 2, Before Using the IFF Translators](#).

### Accessing the Framework Output Transfer Form

To access the schematic export tools, launch your export from the Cadence *Command Interpreter Window* (CIW).

1. In a terminal window, change to the appropriate directory.
2. Run Cadence by typing the appropriate command (typically `icms` or `msfb`). The Cadence *Command Interpreter Window* (CIW) appears.



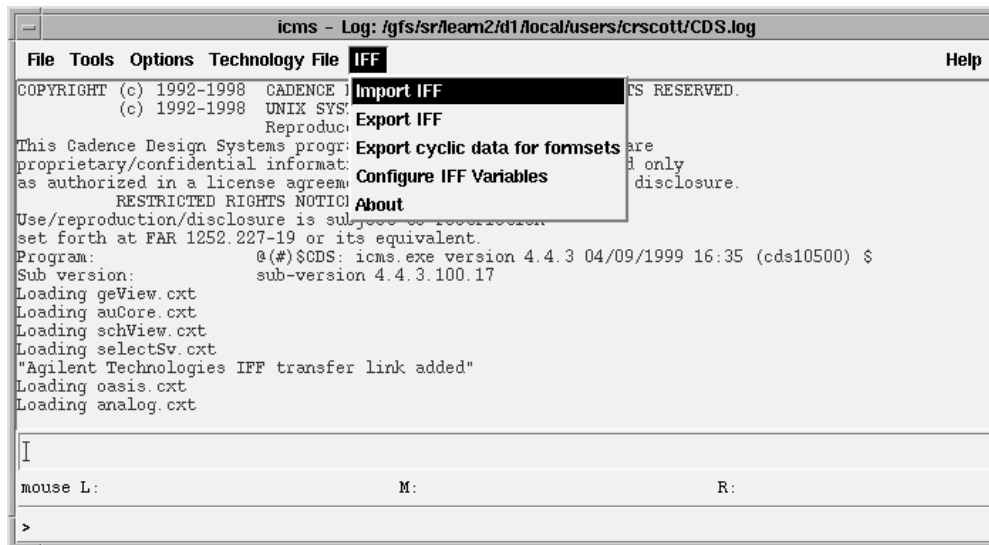


Figure 6-2. Cadence CIW Window

3. From the Cadence CIW, choose IFF > Export IFF. The *Framework Output Transfer Form* appears.

**Framework Output Transfer Form**

OK Cancel Defaults Apply Help

**Output File Name**

**Output Library Name**

**Output Cell Name**

**Type of Hierarchical Transfer:**

- The design only.
- The design and its parts found in the same
- The design and its parts except those found
- The entire library.

**Simulator to use for pin ordering**

**Export standard connectors as components**

**Exclude these libraries in Hierarchical Export** ("MDSprims")"/>

**Do not export these properties when encountered**

**Export Component Parameter Modifiers from CDF**

## Specifying the File Name and Setting Export Options

In the *Framework Output Transfer Form*, specify the file name, enter the library and output cell names and supply other basic information needed by the translator.

### Output File Name

In the *Output File Name* field, enter the full path and file name of the IFF file generated in Virtuoso Schematic Composer that you want to export into Advanced Design System.

### Output Library Name

In the *Output Library Name* field, enter the name of the library containing the cell you want to export. If you are exporting the entire library, enter the name of the library you want to export. The library name will default to the library name of the cell that was in the last active schematic window.

## Output Cell Name

In the *Output Cell Name* field, enter the name of the Cadence cell that you want Cadence to export. The cell name will default to the cell name of the cell that was in the last active schematic window.

## Type of Hierarchical Transfer

There are four choices available from the *Type of Hierarchical Transfer* section of the *Framework Output Transfer Form*. For design exports, it is recommended that the second button is selected so that the output IFF file contains only hierarchical blocks of the design and none of the library parts. For detailed information on each of the selections, refer to the descriptions below.

**The design Only** When this option is selected, only the schematic and symbolic views of the top-level design are exported. No hierarchy is translated. For this option to function properly, all of the instances used in the design must have library elements that already exist in one of the libraries available in ADS.

**The design and its parts found in the same library** When this option is selected, the schematic and symbolic views of the open design are exported. Additionally, all schematic and symbolic views of cells (included in the cell) in the same library as the open design are output into the IFF file. Any instance of reference library cells (that is, cells from a library other than that of the open design) are not exported. For this option to function properly, all of the cells placed from other libraries must already exist in one of the libraries available in ADS.

**The design and its parts except those found in HP\_excludeLibs** When this option is selected, the schematic and symbolic views of the open design are exported. Additionally, the schematic and symbolic views of any instance encountered in the design are exported unless the instance belongs to a library appearing in the *HP\_excludeLibs* list. For this option to function properly, any cell placed from a library listed in *HP\_excludeLibs* must already exist in one of the libraries available in ADS.

---

**Caution** *The design and its parts except those found in HP\_excludeLibs* is the default setting for this form. Exporting with this option set may cause simulation problems in Advanced Design System and should be avoided. This setting is typically used for establishing a reference library in ADS.

---

**The entire library** When this option is selected, the schematic and symbolic views of the open design are exported. The schematic and symbolic views of all other designs in the library are also exported, whether the cells have been placed in the open design or not. Nothing from any other library is exported.

This option is provided so that reference libraries from Cadence can be moved into Advanced Design System. After the file has been exported, it will still be necessary to perform simulation setups on all components. The steps required to build a functional ADS version of any Cadence reference library is complex and currently undocumented as discussed in [“Understanding Component Library Requirements” on page 2-7](#). For information on creating compatible component libraries for use with IFF, refer to [“Getting Help” on page 2-8](#).

## Simulator to use for pin ordering

The *Simulator to use for pin ordering* field is used to designate the simulator to use for component pin ordering. Enter (*spectreS*) since ADS models are typically translated from *Spectre* subcircuit models. The default value is (*spectreS*).

## Export standard connectors as components.

The IFF translator normally converts ports and grounds to an Advanced Design System equivalent. When this option is selected, ports and grounds are *not* translated to ADS equivalents. This requires that the Cadence port or ground be transferred as a library component. When the *Export standard connectors as components* option is selected, all standard connectors are translated as components in ADS. The default setting for this option is deselected.

## Exclude these libraries in Hierarchical Export

Components listed in the *Exclude these libraries in Hierarchical Export* field are not hierarchically exported. Thus, no symbol or schematic information is created for these components. Instance data is still generated.

## Do not export these properties when encountered

The properties listed in the *Do not export these properties when encountered* field are not included in symbols or instance records. This option is useful for suppressing Cadence system properties.

## Export Component Parameter Modifiers from CDF

When the *Export Component Parameter Modifiers from CDF* option is selected, component parameter modifiers contained in the Cadence Component Description Format (CDF) are exported to Advanced Design System. The default setting for this option is deselected.

## Completing the Export

After you have completed the *Framework Output Transfer Form*, click **Apply** in the *Framework Output Transfer Form* to save your settings or **Defaults** to retain the default settings. You can also click **Cancel** to abort the export operation. Click **OK** to begin the translation.

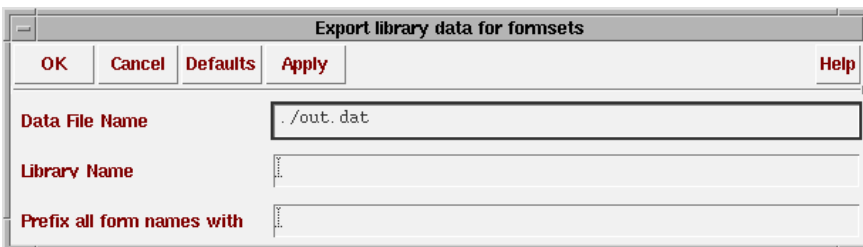
After the translation begins, all export log information is displayed in the Cadence Command Interpreter Window (CIW). One of the log lines will display what was created. Any warnings or error messages encountered during the translation are also displayed in the log messages.

## Additional Export Options

This section describes some of the additional options available from the Cadence CIW *IFF* command menu.

### Export library data for formsets

This section describes the selections available from the *Export library data for formsets* dialog box. When the menu command *IFF > Export cyclic data for formsets* is selected from the Cadence CIW, the *Export library data for formsets* dialog box appears.



**Note** This dialog box was created exclusively for library customizing and should not be used without consulting Agilent Technologies Solution Services. For more information, contact your Agilent Technologies sales representative.

---

## Data File Name

The data file is a Cadence file that can be utilized by the Library Translator, a custom tool available from Agilent Technologies Solution Services. The file name entered into this field contains information about all of the cyclic fields for a given library. Do not enter a file extension when entering the file name as two separate files are created during the export. One will have a *.dat* extension, which is a semi-binary representation of the cyclic fields, which will be used in the Library Translator so that you can choose what formset a Cadence parameter will utilize. The second file will have a *.ael* extension that will contain the ADS Application Extension Language (AEL) code necessary to create the needed forms and formsets for the ADS environment.

## Library Name

This field is used to enter the name of the library that you want to export cyclic field data from.

## Prefix all form names with

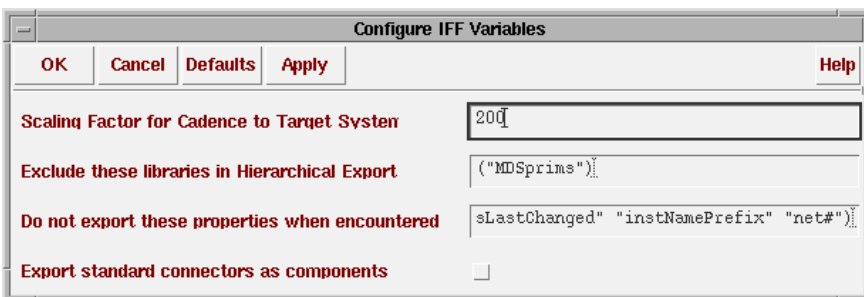
In some cases, it may be necessary to prefix the form names with the library name or the library and cell name. This field enables you to designate a prefix in order to generate unique form and formset names.

As an example, if you were exporting data from the *analogLib* and *generic\_foundry* libraries, both of these libraries contain a cell named *npn* that has a parameter called *region*. In *analogLib*, the values are *on* and *off*. In *generic\_foundry*, the values are *on*, *off* and *reverse*. When the formsets are generated, if the formset name is *npn\_region*, the *analogLib* and *genericLib* components could not be used simultaneously in ADS as formsets would overlap. It is necessary to prefix the formset names so that they will be unique. This is done by specifying *analogLib* as the prefix when exporting the *analogLib* library, and specifying *generic\_foundry* when exporting the *generic\_foundry* library. Now, the formset name for *analogLib* would be *analogLib\_npn\_region*, and the formset name for *generic\_foundry* would be *generic\_foundry\_npn\_region*.

---

## Configure IFF Variables

This section describes the selections available from the *Configure IFF Variables* dialog box. When the menu command **IFF > Configure IFF Variables** is selected from the Cadence CIW, the *Configure IFF Variables* dialog box appears.



### Scaling Factor for Cadence to Target System

When transferring cells from Cadence to Advanced Design System, the Cadence symbol sizes tend to be about one half the size of ADS symbol sizes. This results in poor quality schematic resolution. This scaling factor can be used during an export to improve the resolution by using the default value (200) to make all Cadence symbols and schematics twice the size in ADS. Additionally, during an import, all values are reduced to 50% of their original size.

### Exclude these libraries in Hierarchical Export

This field enables you to specify a set of libraries that should not have symbols and schematics included in hierarchical exports. The primary purpose of excluding libraries is that when a reference library is used in Cadence, a duplicate of that library must be created in ADS. The duplicate library contains all of the simulation setup data that is not contained within the Cadence library. If the components from the reference library are output into the IFF file, new components without the necessary simulation setups are created in ADS and the design will not simulate.

As an example, let's say that you have a design in a library *mylib* called *myschematic*. Within this design, you place an instance of the cell *res* from the *analogLib* library. If you specify *analogLib* in the *Exclude these libraries in Hierarchical Export* field, and then proceed to perform a full hierarchical export, the symbol for *mylib myschematic* is exported and the schematic *mylib myschematic* is created in the IFF file. However,

the symbol for `analogLib` is not placed in the IFF file because it has been designated as being excluded. If `analogLib` is not specified in the exclude library list, the symbol for `analogLib res` is created in the IFF file. Note that if `analogLib res` is created as a symbol, it is up to you to create the proper simulation setup for the component when it is imported into ADS.

---

**Note** This field matches the field found on the IFF export dialog.

---

## Do not export these properties when encountered

This field is provided primarily so that Cadence system properties can be omitted from the IFF export if desired. It's main purpose is to avoid clutter in the property fields of instances. You can decide if you want to omit properties that are not necessarily Cadence system properties, or whether you want to have the properties translated.

## Export standard connectors as components

This checkbox enables you to specify how to translate connectors. A connector in this context is defined as a pin or a ground component. In Cadence, this means `basic ipin`, `basic opin`, `basic iopin`, `analogLib gnd`, `analogLib gnda`, and `analogLib gndd`. When any of these components are encountered, and this checkbox is deactivated, the components are converted into the ADS equivalent components (either `Port` for the basic pins or `Ground` for the analogLib ground component). Because the ADS components are shaped differently from the Cadence components, it may be more desirable to transfer the components without converting them into ADS equivalents ensuring that the schematic will look identical to its Cadence representation. Activate this checkbox if you want to translate your connectors as Cadence components.

---

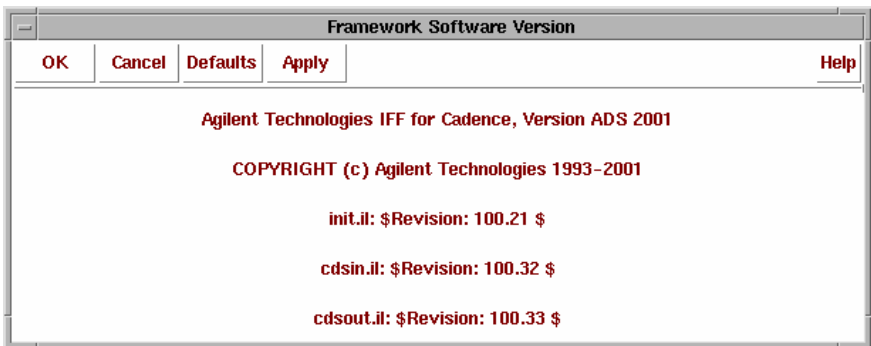
**Note** The `analogLib` Design Kit for ADS contains all of these components and they are already setup to function properly as the default ADS connectors. If you want to activate this checkbox, it is recommended that the factory generated `analogLib` Design Kit be used in the ADS environment.

---



# Framework Software Version

This section describes the *Framework Software Version* dialog box. When the menu command **IFF > About** is selected from the Cadence CIW, the *Framework Software Version* dialog box appears.



This dialog box displays:

- The IFF for Cadence version number.
- The Agilent Technologies (previously “HEWLETT-PACKARD”) Copyright information.
- Revision information for the *init.il*, *cdsin.il* and *cdsout.il* files.
- Information on where to obtain the latest version of the *mdsinit.il* file.

Use this information when contacting Agilent Technologies for technical support.



# Index

## A

- ADS, 1-1
  - Application Extension Language, 6-8
  - Item Definition properties, 4-6
  - projects, 3-2
  - schematic, 3-2, 4-2
- analogLib, 6-8, 6-9
  - Design Kit, 6-10
  - gnd, gnda & gndd, 6-10
  - library, 2-6

## B

- basic
  - ipin, opin & iopin, 6-10
  - library, 2-6

## C

- Cadence
  - Cadence Design Systems, 2-1, 2-2
  - CDF, 6-7
  - cell view, 6-5
  - CIW, 2-3, 2-4, 5-2, 6-2
  - components, 6-10
  - customized start-up scripts, 2-5
  - data file, 6-8
  - environment variables, 2-3
  - initialization file, 2-2
  - restarting your session, 2-3
  - running DFII, 2-4
  - symbols, 6-9
  - system properties, 6-6, 6-10
  - Virtuoso Schematic Composer, 2-2
- cell name, 6-5, 6-8
- command interpreter window, 2-3, 2-4, 5-2, 6-2
- component libraries, 1-3, 1-6
  - translation, 1-6
- components
  - Cadence, 6-10
  - component description format, 6-7
  - connector components, 2-6
  - mapping, 2-7
  - parameter modifiers, 6-7
  - parameters, 2-7
  - pin ordering, 6-6

- translating, 4-6
- connectors, 6-6, 6-10
- coordinates, 4-8
- copyright information, 6-11
- cyclic field data, 6-8

## D

- data file, 6-8
- Design Kits
  - analogLib, 6-10
- designs
  - design objects, 4-7
  - design variables, 2-9
  - identical names, 3-3
  - referenced designs, 4-8
  - transferring multiple designs, 4-6
- DFII, 1-1, 1-2
  - non-standard start-up scripts, 2-5
  - supported versions, 2-1
- diagnostics, 3-5
- directories
  - cdslib, 2-6

## E

- EDA, 1-1
- environment variables
  - Cadence, 2-3
- equations, 2-9
- error messages, 3-5, 6-7
- export log, 6-7
- exporting schematics, 4-1, 6-1
- extracted views, 2-10

## F

- files
  - .ael extension, 6-8
  - .cdsinit, 5-2, 6-2
  - .dat extension, 6-8
  - cdsinit, 2-2
  - cdsout.il, 6-11
  - data file, 6-8
  - file desitination, 4-5
  - file extension, 6-8
  - file selection, 3-4, 4-3
  - init.il, 6-11
  - mdsinit.il, 6-11

- form names, 6-8
- formsets, 6-8
- Framework
  - Input Transfer Form, 5-3
  - Output Transfer Form, 2-6, 6-4
  - Software Version, 6-11

## G

- generic\_foundry, 6-8
- genericLib, 6-8
- global named connections, 2-9
- ground symbols, 2-6
- grounds, 6-6, 6-10

## H

- Hewlett Packard, 2-1
- hierarchical
  - designs, 2-9, 6-5
  - exports, 6-9
  - projects, 4-8
- HP\_excludeLibs, 6-5
- hpeesoflib, 4-7

## I

- IBM, 2-1
- IC design, 1-3
- icfb, 2-4
- icms, 2-4, 5-2, 6-2
- IFF, 1-1
  - export options, 4-4
  - import options, 3-5
  - pull-down menu, 2-3, 2-5
  - translation availability, 1-3
  - translators, 1-2, 2-1
  - use model, 1-4
- iffplib.log, 3-5
- importing
  - schematics, 3-1, 5-1
- instance data, 6-6
- instance records, 6-6

## L

- libraries, 6-9
  - analogLib, 2-6, 6-8, 6-9
  - basic, 2-6
  - compatible component libraries, 1-6, 2-7
  - component library requirements, 2-7, 3-6
  - default library name, 4-7
  - generic\_foundry, 6-8
  - library cells, 6-5
  - library customization, 6-8
  - library development, 2-8
  - library name, 3-6, 4-7, 6-4, 6-8
  - missing libraries, 2-7
- Library Translator, 6-8
- licenses, 2-1, 2-2
- log messages, 6-7

## M

- missing libraries, 2-7
- msfb, 5-2, 6-2

## N

- npn, 6-8
- npn\_region, 6-8

## O

- operating system versions, 2-1

## P

- parameter
  - spacing, 4-6
  - values, 4-8
- partial translation, 1-6
- pins, 6-10
  - pin mapping, 2-6
  - pin ordering, 6-6
- ports, 6-6
  - port construction, 2-9
- power rails, 2-9
- projects
  - ADS, 3-2
  - hierarchical, 4-8
- property fields, 6-10

## R

- region, 6-8

## S

- scaling factor, 6-9
- schematic, 6-6
  - ADS, 3-2, 4-2
  - export, 6-1
  - hierarchy, 4-8
  - import, 3-1, 5-1
  - resolution, 6-9

- schematics per cell, 2-10
- translation, 1-6, 2-9
  - view, 6-6
- simulation, 2-8, 6-9
  - control elements, 2-9
- SKILL, 2-3
- specialized tools, 1-6
- spectre, 6-6
- subcircuits, 2-9
- Sun, 2-1
- symbol, 6-6
  - sizes, 6-9
  - view, 6-5, 6-6
- system properties, 6-10

## **T**

- third party frameworks, 1-3
- translation
  - schematic, 2-9

## **U**

- unique form and formset names, 6-8

## **V**

- versions
  - Cadence version number, 6-11
  - cdsin.ilfiles
    - cdsin.il, 6-11
  - cdsout.il, 6-11
  - init.il, 6-11
  - mdsinit.il, 6-11
- Virtuoso Schematic Composer, 1-1, 2-2, 6-4

## **W**

- warning messages, 3-5, 6-7

