



Agilent Technologies

IFF Translation for Mentor Graphics

August 2005

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, Inc. 1983-2005
395 Page Mill Road, Palo Alto, CA 94304 U.S.A.

Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries.

Microsoft[®], Windows[®], MS Windows[®], Windows NT[®], and MS-DOS[®] are U.S. registered trademarks of Microsoft Corporation.

Pentium[®] is a U.S. registered trademark of Intel Corporation.

PostScript[®] and Acrobat[®] are trademarks of Adobe Systems Incorporated.

UNIX[®] is a registered trademark of the Open Group.

Java[™] is a U.S. trademark of Sun Microsystems, Inc.

SystemC[®] is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission.

Contents

1 Introduction

Advanced Design System	1-1
Boardstation	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.....	2-1
Setting Environment for Mentor Graphics.....	2-2
Understanding Component Library Requirements	2-3
Simulating a Design Transferred via IFF.....	2-4
Getting Help	2-5
Constructing Designs for IFF Translation	2-5
Separating Simulation Control Elements.....	2-5
Known Issues and Limitations	2-6

3 Importing IFF Schematic Files into ADS

Opening an ADS Project and Schematic Window	3-1
Importing an IFF Schematic File	3-2
Accessing the Schematic Import dialog	3-2
Specifying the Import File Selection	3-3
Selecting Import IFF Options.....	3-4
Completing the IFF Import.....	3-6

4 Exporting IFF Schematic Files from ADS

Exporting an IFF Schematic File	4-1
Accessing the Schematic Export dialog	4-2
Specifying the Export File Selection.....	4-2
Selecting Export IFF Options	4-3
Completing the IFF Export	4-9

5 Importing IFF Files into Mentor Graphics

Importing an IFF Schematic File	5-2
---------------------------------------	-----

Accessing the Framework Input Transfer Form	5-2
Specifying the File Name and Setting Import Options.....	5-4
Completing the Import.....	5-4
Setting Up a Library Search Path	5-5
Importing an IFF Layout File.....	5-6
Completing the Layout Import	5-6
6 Exporting Designs to IFF from Boardstation	
Exporting an IFF Schematic File	6-1
Accessing the IFF Write Dialog in Design Architect	6-2
Completing the Export.....	6-4
Exporting an IFF Layout File	6-4
Accessing the IFF Write Dialog in RF Layout.....	6-5
Completing the IFF Export	6-6
7 Importing IFF Layout Files into ADS	
Opening an ADS Project and Layout Window	7-1
Importing an IFF Layout File.....	7-2
Accessing the Layout Import dialog.....	7-2
Specifying the Import File Selection	7-3
Selecting Import IFF Options.....	7-4
Completing the IFF Import.....	7-7
8 Exporting IFF Layout Files from ADS	
Exporting an IFF Layout File	8-1
Accessing the MGC/PCB Export dialog	8-2
Where do the files go?	8-2
Selecting MGC/PCB Export Options	8-3
Completing the MGC/PCB Export.....	8-8
Index	

Chapter 1: Introduction

Agilent Technologies and Mentor Graphics Corporation 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 Advanced Design System (ADS) and Mentor Graphics 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 Boardstation from Mentor Graphics Corporation.

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, 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.

Boardstation

Boardstation from Mentor Graphics Corporation is an integrated environment for generation of PC Boards. Boardstation contains both a schematic tool (Design Architect), and a physical layout tool. The IFF interface to Boardstation will specifically only work with Library Management System (LMS) libraries, and the special library that is provided with Mentor's RF Architect product. RF Architect is a special add-on to the Boardstation environment that allows the standard ADS palette of components to be used within Mentor Graphics. Additionally, it enables the user to create parameterized layout instances, which is a requirement when generating transmission line components in a PC Board environment.

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. Mentor Graphics has also created an IFF translator for use with Boardstation. 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 Mentor Graphics design environment resulting in the following benefits:

- Avoids 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 Boardstation and vice versa
- Export IFF files from ADS to Boardstation 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 RF-Board Designers. It is assumed that the designer using the IFF translators has some working knowledge of both Advanced Design System and Mentor Graphics Boardstation.

Main Requirements

To enable the successful IFF translation of an RF Board design between ADS and Mentor Graphics Boardstation, you must ensure that the following requirements are met:

- The link between IFF and the Mentor Graphics design environment is available and installed. The IFF translator for Mentor Graphics is a part of RF Architect.
- The component libraries used for creating the schematic in ADS and Design Architect have been implemented to support translation via IFF. For more detailed information on library requirements, refer to [“Understanding Component Library Requirements” on page 2-3](#). If equivalent libraries have not been set up for Mentor Graphics components in ADS, the Mentor Graphics components will not be usable in simulations.

Note Agilent Technologies supports the IFF Import and Export tools for ADS. Mentor Graphics is responsible for the RF Architect product, which includes ADS equivalent part libraries, as well as IFF Import and IFF export. ADS and Mentor IFF products are licensed separately. Contact Mentor Graphics for information on IFF translation availability for Boardstation.

The IFF Transfer Process

Once the main requirements have been satisfied, your schematic can be transferred between Advanced Design System and Mentor Graphics Boardstation. 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 Mentor Graphics Boardstation. The link between the two EDA environments is established via the ADS and Boardstation IFF Interface. The two component libraries in both ADS and Boardstation must be compatible to support an IFF translation.

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

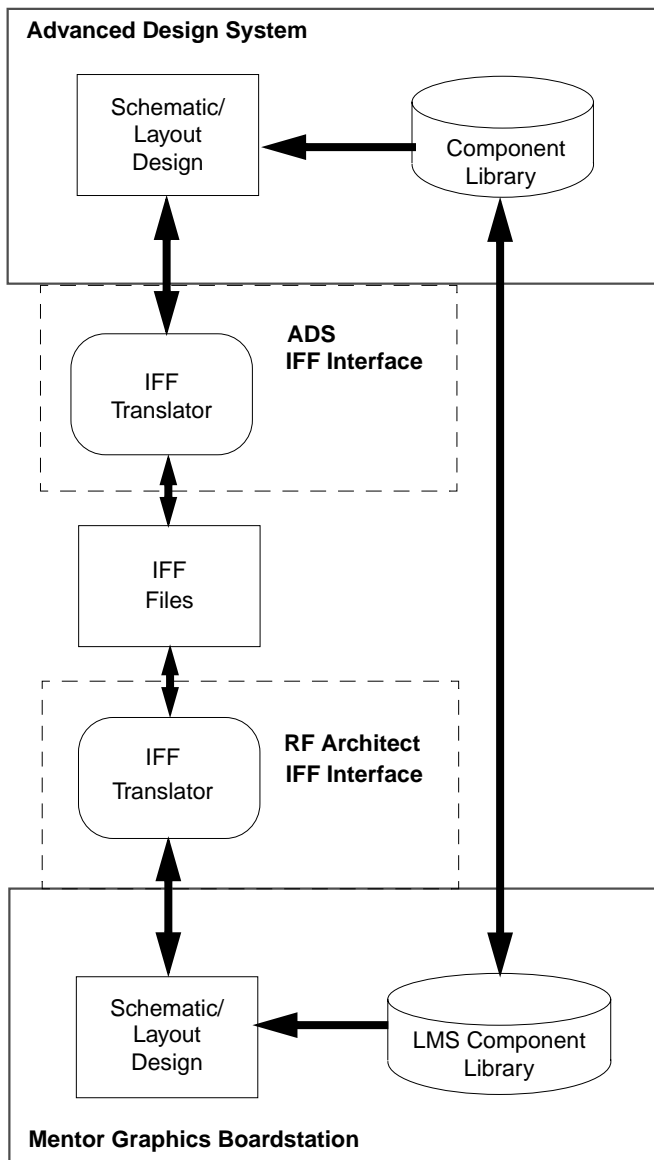


Figure 1-1. IFF Schematic and Layout Transfer Use Model

What's in this Manual

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

- [Chapter 2, Before Using the IFF Translators](#) describes how to configure your Mentor Graphics environment to support IFF translation between ADS and Boardstation. This chapter also discusses some of the issues related to IFF schematic translation in the ADS and Boardstation 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 Files into Mentor Graphics](#) describes the procedure for importing an IFF file into Boardstation for use in Design Architect.
- [Chapter 6, Exporting Designs to IFF from Boardstation](#) describes the procedure for exporting a Design Architect Schematic to an IFF file from Boardstation.
- [Chapter 7, Importing IFF Layout Files into ADS](#) describes the procedure for importing an IFF file into an Advanced Design System Layout.
- [Chapter 8, Exporting IFF Layout Files from ADS](#) describes the procedure for exporting an ADS Layout to an IFF file from Advanced Design System.

About Design Translation

It is highly recommended that the IFF Translator be used for schematic and layout translation only when compatible component libraries exist 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 Mentor Graphics 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 Mentor Graphics to ADS, *Agilent Technologies Solution Services* can provide you with specialized tools and help. For more information, contact your Agilent Technologies sales representative.

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 2-3](#)
- [“Constructing Designs for IFF Translation” on page 2-5](#)

After covering the information in this chapter, you'll be ready to begin sharing your designs between ADS and Boardstation using the IFF translators.

Configuring your Software for IFF Translation

This section describes the details related to software requirements and configuring the ADS and Mentor Graphics design environments to support IFF translation. Before attempting an IFF translation, the appropriate licenses must also be installed and the shell environment must be configured for Mentor Graphics.

Software Requirements

The IFF Translator described in this manual requires ADS 1.7 or newer. IFF is supported by Mentor Graphics versions C.4 or newer. IFF is also supported on all operating system versions which run Mentor Graphics software. Currently, this includes Solaris 2.8 or newer, HP-UX 11, Windows 2000, or Red Hat Linux 7.2, 7.3, or 8.0. For additional information, please contact Mentor Graphics Corporation.

License Requirements

In addition to the mainstream Mentor Graphics (Design Architect, Falcon Framework, etc.) and Advanced Design System licenses, the following additional product licenses are required.

Agilent Technologies Licenses

- `trans_iff`
- `trans_mentor_schem`
- `trans_mentor_layout`

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

Mentor Graphics Licenses

- RF Architect license (pcbrarch)
- RF Layout license (pcbrflayout)

Note This and all other Mentor Graphics licenses must be purchased from Mentor Graphics Corporation.

Setting Environment for Mentor Graphics

Mentor Graphics requires some special environment variables to be set so that their RF Architect product will be available in Design Architect. This section describes the information that must be added your environment, as well as the libraries that should be added to the Mentor Graphics location map file. This information is provided for convenience only. To obtain up-to-date information, refer to Mentor Graphic’s *RF Design Tools User’s and Reference Manual* (rf_useref.pdf), which is distributed with Mentor Graphic’s RF Architect product.

The following environment variables must be set, either in your UNIX profile (*.cshrc* or *.profile* depending on your shell), or in the system environment on Windows:

- MGC_HOME - Set this to the Mentor Graphics installation directory
- MGC_LOCATION_MAP - Set this to the full path of the location map file you wish to use.
- AMPLE_PATH - Add the path \$MGC_HOME/pkg/pcb_rf/userware/En_na to the value of AMPLE_PATH (delimit multiple paths with colons). On Windows, add %MGC_HOME%\pkg\pcb_rf\userware\En_na (delimit multiple paths with semicolons). Including this directory in the Ample path will allow Mentor Graphics to load the add on software required for doing IFF imports and exports from ADS.
- MGC_RF_SYSTEM - Set the value to ADS

Note *Ample* is Mentor Graphic's C/lisp-like interpretive programming language for framework and database integration.

Add the following libraries to your location map file:

- `$MGC_ADSLIB` - This is the Mentor Graphics equivalent library for ADS primitive components. This library is primarily used for drawing transmission line elements. A tar file containing this library is located at `$MGC_HOME/pkgs/pcb_rf/data/lib`. Consult your librarian to get the location where the library has been installed.
- `$HPEESOF_LIBS` - This is the Mentor Graphics equivalent library for ADS connectors. It contains ADS ports that can be used for generating hierarchical designs. A tar file containing this library is located at `$MGC_HOME/pkgs/hpeesof_int/data/lib`. Consult your librarian to get the location where the library has been installed.

Example of a location map file:

```
#
MGC_LOCATION_MAP_1
#
$MGC_ADSLIB
/sj/pcb/rf/data/mgc_adslib
$HPEESOF_LIBS
/sj/pcb/rf/data/hpeesof_libs
$MGC_GENLIB
/sj/auspcbfsl/mentor_8.2/mgc_libraries/gen_lib
```

Understanding Component Library Requirements

The fundamental library requirement for a successful IFF transfer is that both the Advanced Design System and Mentor Graphics Falcon Framework 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.

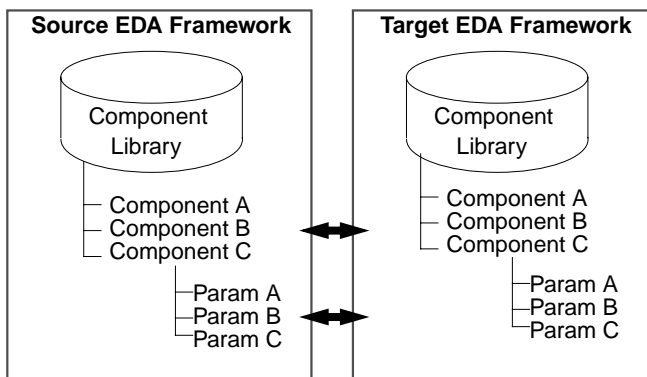


Figure 2-1. Compatible Component Libraries

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. This requires more than just a transfer of symbol graphics and properties. In ADS, it is necessary to set up a simulation definition for the component. In [Figure 2-1](#), assume Component A is a capacitor. When the component is sent to ADS via IFF, a symbol of the Component A capacitor will be created. ADS will *not* know it is a capacitor based on the IFF import. It is necessary to use utilities to tell ADS that Component A is a capacitor. This can be done by using the Design/Parameters dialog, using the Library Translator, or editing the AEL component definition file for Component A. Additionally, because ADS is designed to handle high frequency simulation effects, it is usually necessary to create a subcircuit model that represents the capacitor with parasitic effects that are not seen in low frequency simulations. These high frequency models will typically not exist in the source framework, and need to be created specifically in ADS.

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 Designs for IFF Translation

The IFF translator module enables bi-directional, fully hierarchical schematic and layout transfer between the ADS and Mentor Graphics Boardstation. You can edit your schematic and layout 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 design 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 2-3](#).

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 RF Board designs in ADS.

Separating Simulation Control Elements

Simulation control elements are not transferred between ADS and Mentor Graphics. 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 Mentor Graphics environment. For more information, refer to *“Creating Hierarchical Designs”* in the [Schematic Capture and Layout](#) manual.

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.
- Only one symbol is allowed per cell in Advanced Design System.
- Only one layout is allowed per cell in Advanced Design System.
- Schematic view points can not be exported from Mentor Graphics. Only the base schematic can be translated. This means that package annotations that are annotated to a view point can not be sent to Advanced Design System.

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.

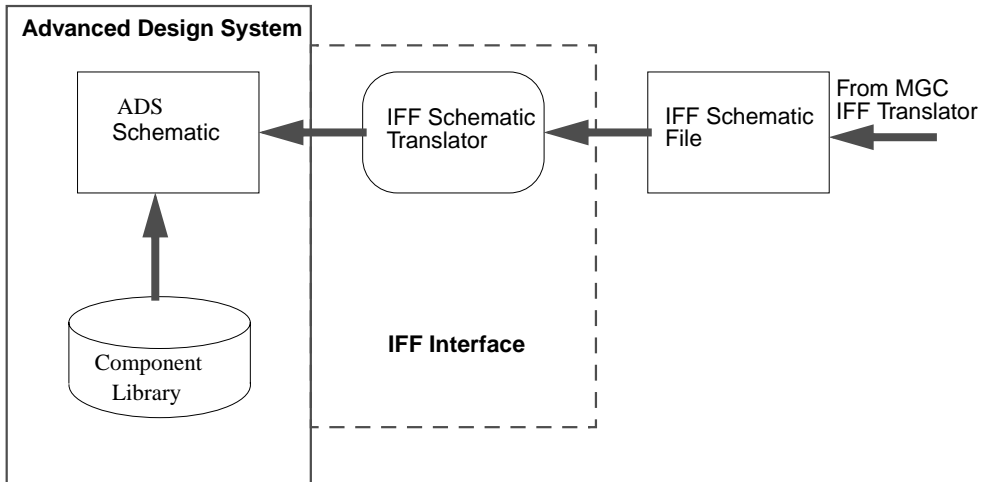


Figure 3-1. IFF Schematic Import in ADS

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 3-1](#)
2. [“Accessing the Schematic Import dialog” on page 3-2](#)
3. [“Specifying the Import File Selection” on page 3-3](#)
4. [“Selecting Import IFF Options” on page 3-4](#)
5. [“Completing the IFF Import” on page 3-6](#)

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 “*Schematic Capture and Layout*” manual.

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

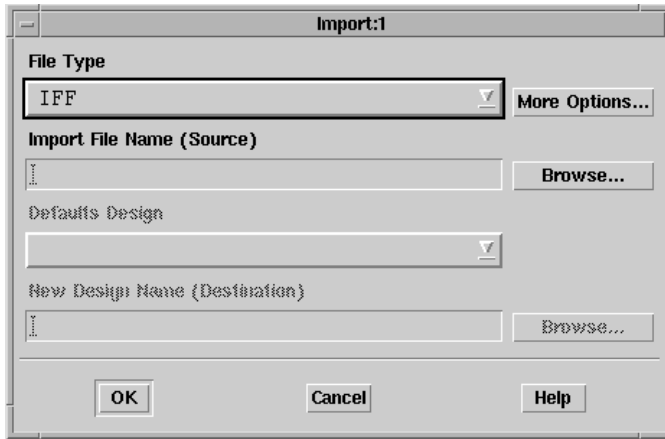
This section describes the procedure for importing an IFF schematic file into Advanced Design System using the IFF Translator’s User Interface. Before importing an IFF schematic file, refer to “[Opening an ADS Project and Schematic Window](#)” on [page 3-1](#).

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. ADS will force all designs to be closed prior to import. Designs are saved to disk, not to memory. If you wish to keep your existing ADS designs intact, you must make a backup copy prior to doing your import.

Accessing the Schematic Import dialog

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

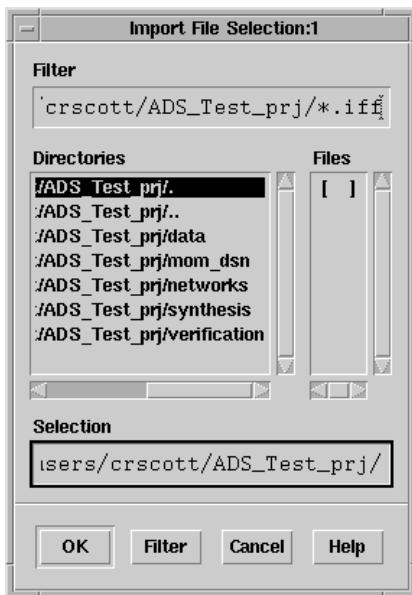
Choose **File > Import**. The schematic *Import* dialog box appears.



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.

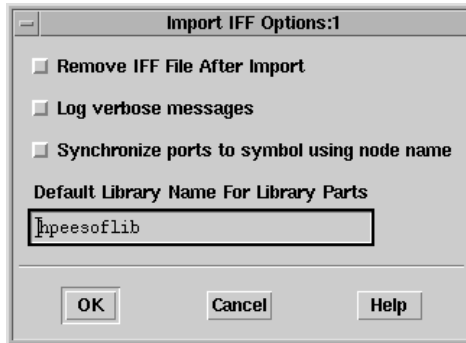


3. Double-click as needed to locate the directory containing your IFF file in the *Directories* field, then click the file in the *Files* field. Alternatively, you can enter the full path and file name in the *Selection* field.
4. After selecting the design you want to import, click **OK**. You are returned 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 *iff.lib.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 IFF importer. Ensure that this checkbox is un-checked.

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. Mentor Graphics 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 [“Exporting IFF Schematic Files from ADS” on page 4-1](#).

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.

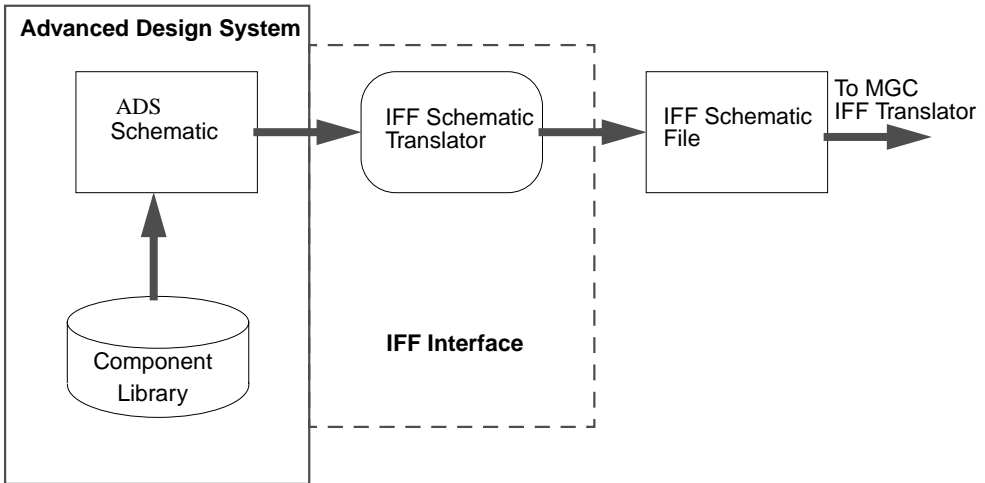


Figure 4-1. IFF Schematic Export from ADS

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 3-1](#)
2. [“Accessing the Schematic Export dialog” on page 4-2](#)
3. [“Specifying the Export File Selection” on page 4-2](#)
4. [“Selecting Export IFF Options” on page 4-3](#)
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 3-1](#).

Accessing the Schematic Export dialog

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

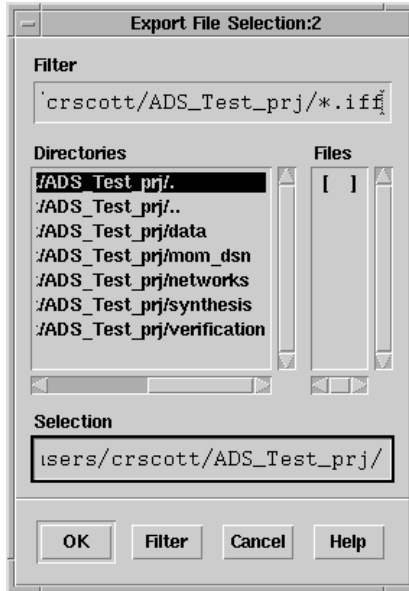
1. Open your ADS Schematic Design in the ADS Schematic window.
2. Choose **File > Export**. The schematic *Export* dialog box appears.



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 export, click **Browse** in the *Export* dialog box. The *Export File Selection* dialog box appears.

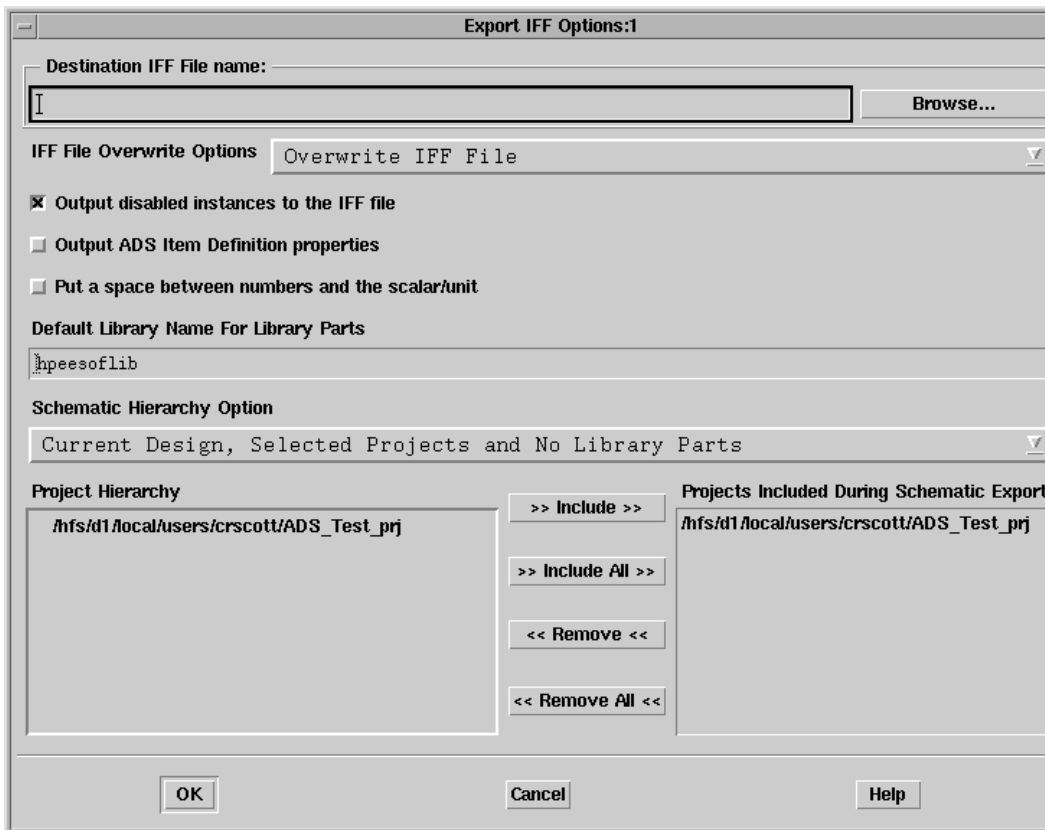


3. Double-click as needed to locate the directory for your exported IFF schematic file in the *Directories* field then enter the new file name in the *Selection* field. Alternatively, you can enter the full path and file name in the *Selection* field.
4. After selecting the new location and name of your 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.

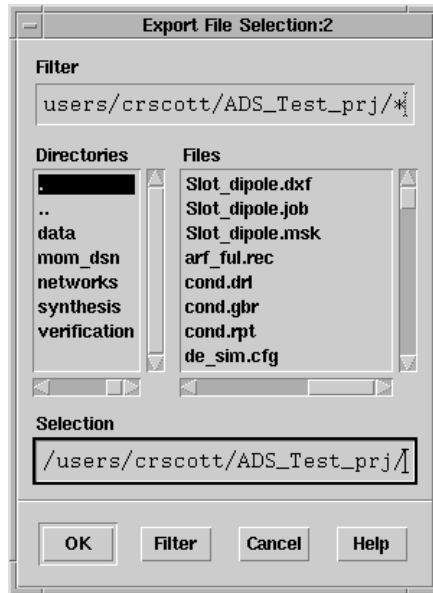


In the Export IFF Options dialog box, select the appropriate options for your translation using the information below.

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 deactivated in the schematic, it will still be output into the IFF file. If the checkbox is deselected (default), deactivated 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. deactivate the simulation components prior to creating an IFF file to send to Mentor Graphics, which does not have any definitions for the simulator components). Check this option if you want to get everything. Uncheck 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 *hpeesoffib* 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 Files into Mentor Graphics

This chapter describes the procedure for importing Intermediate File Format (IFF) files into Mentor Graphics Boardstation. This information is provided as a convenience. For detailed information on Mentor Graphics IFF imports, refer to Mentor Graphic's *RF Design Tools User's and Reference Manual* (rf_userref.pdf), which is provided with their RF Architect product. The Import procedure is detailed in "Chapter 7, Importing a Design".

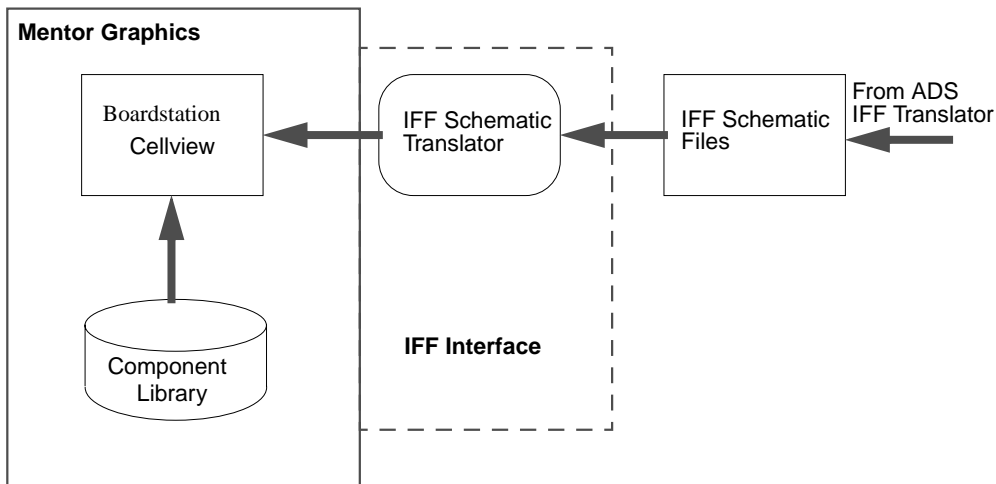


Figure 5-1. IFF Schematic Import into Mentor Graphics

Before attempting to import an IFF file into Mentor Graphics, refer to [Chapter 2, Before Using the IFF Translators](#). The basic procedure for importing an IFF schematic file into Boardstation 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 5-4
3. "Completing the Import" on page 5-4

Importing an IFF Schematic File

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

The menu items described in this section are only if the appropriate *Ample* path has been set up for the RF Architect option of Design Architect. 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 Boardstation *Design Architect* window.

1. In a terminal window, change to the appropriate directory.
2. Run Design Architect by typing the appropriate command (typically *da* or *da_lms*). The Mentor Graphics *Design Architect* window appears.

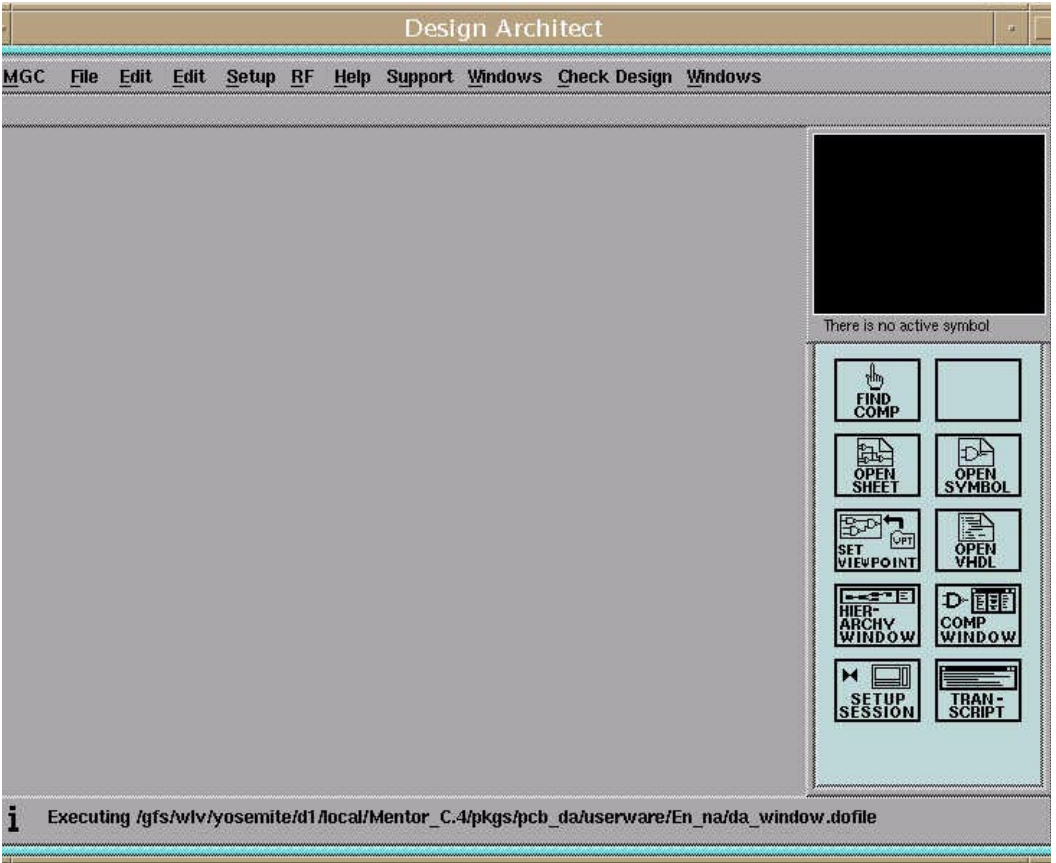


Figure 5-2. Design Architect Window

3. From the Design Architect window, choose **RF > IFF Read**. The *IFF Read Dialog* appears.

Specifying the File Name and Setting Import Options

In the *IFF Read Dialog*, specify the file name, the transfer directory name, and whether to use the ADS reference designators.

Name of input file

In the *Name of input file* field, enter the full path and file name of the IFF file generated in Advanced Design System that you want to import into Mentor Graphics. Pressing the Navigator button will bring up a browsing dialog, that will allow you to graphically search for the IFF file name.

Transfer directory name

The *Transfer directory name* field specifies the directory where the Mentor Graphics design files will be created. The Navigator button can be pressed to bring up a graphical directory browser.

Import EEsof Reference Designators

Setting the *Import EEsof Reference Designators* checkbox causes the EEsof instance names to be used as the REF property on the MGC instance. Note that ADS Instance names are unique for only a single level of hierarchy. For hierarchical designs, it is possible to have multiple instances that will get the same REF property. To prevent this problem, this option should be used with care, or, preferably, only with a non-hierarchical design. It is also possible to have duplicate instance references when a PCB board contains two or more independent RF regions.

Completing the Import

After you have filled in the fields of the *IFF Read Dialog*, click **OK** to begin the translation. You can also click **Cancel** to abort the import operation. The **Reset** to button will reset the fields to the default values (“./schematic.iff” and “./mgc_hpeesof_xfer_dir”).

After the translation begins, all import log information is displayed in the terminal window that Design Architect was started from. 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. After the import is completed, the log information will be displayed in Notepad window.

While cells are created in the appropriate directory, the cell view is not automatically displayed in the Design Architect. If full hierarchy was specified for the Advanced Design System export, existing schematics common to both program environments are overwritten on import.

Setting Up a Library Search Path

This information is repeated from Mentor Graphics *RF Design Tools User's and Reference Manual*. The RF LAYOUT, DA and LIBRARIAN options support searching for the library symbols used in a RF design in \$HPEESOF_LIBS directory, based on an entry defined in the location map in Mentor Graphics environment. This is also applicable to user-defined symbol libraries. Additionally, you can set the environmental variable \$MGC_RF_LIB_SEARCH_PATH which makes all the symbols for the packaged parts available. For example,

```
setenv MGC_RF_LIB_SEARCH_PATH /company_lib/resistors:  
/company_lib/caps:/company_lib/relays
```

Note that the colon (:) is required between the libraries.

This setup is essential if you use translated LMS parts in the EEsof environment. This search path provides the symbol mapping path information for the design IFF translation process.

Importing an IFF Layout File

To access the layout import dialog, launch your import from the Boardstation *RF layout* window.

1. Start the RF Layout tool by typing the command **layout design -rf**. You may need to run the *to_layout* command on your schematic prior to running this command. For more information on running *to_layout* for an RF design, refer to Chapter 3 of Mentor Graphic's *RF Design Tools User's and Reference Manual* (rf_useref.pdf). Your PC Board layout must have an RF Region in it for you to import a layout IFF file. You cannot create a new RF region through the layout tool, only the *to_layout* tool can create an RF region.
2. In the layout window, right click to bring up the popup menu, and select the menu option **RF > Import IFF into Layout**. Use the Navigator button to select an IFF file through a graphical browser.
3. The board you will be importing to must have an RF Region set up. This should have been done by the *to_layout* command. Select the region you wish to import the IFF file into. This region should correspond to the schematic that the layout represented in ADS.

Completing the Layout Import

After you have filled in the fields of the *Import IFF into LAYOUT Dialog*, click **OK** to begin the translation. You can also click **Cancel** to abort the import operation. The **Reset** to button will reset the fields to the default values.

After the translation begins, all import log information is displayed in the terminal window that RF Layout was started from. 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. After the import finishes, the final layout will be displayed in the layout window.

Chapter 6: Exporting Designs to IFF from Boardstation

This chapter describes the procedure for exporting an Intermediate File Format (IFF) file from the Mentor Graphics Boardstation.

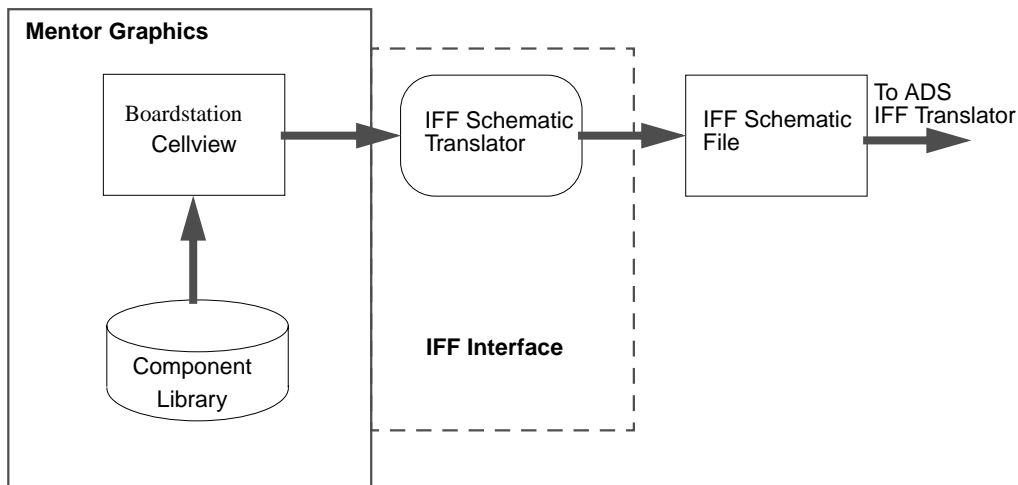


Figure 6-1. IFF Schematic Export from Boardstation

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

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

1. [“Accessing the IFF Write Dialog in Design Architect” on page 6-2](#)
2. [“Completing the Export” on page 6-4](#)

Exporting an IFF Schematic File

This section describes the procedure for exporting an IFF schematic file from Boardstation using the IFF Translator from a Design Architect schematic window. 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 if the correct *Ample* path has been set up for the RF Architect option of Design Architect. For more information, refer to [Chapter 2, Before Using the IFF Translators](#).

Accessing the IFF Write Dialog in Design Architect

To access the schematic export tools, launch your export from the Mentor Graphics *Design Architect* (da) window.

1. In a terminal window, change to the appropriate directory.
2. Run Design Architect by typing the appropriate command (typically *da* or *da_lms*). The Mentor Graphics *Design Architect* (da) window appears.

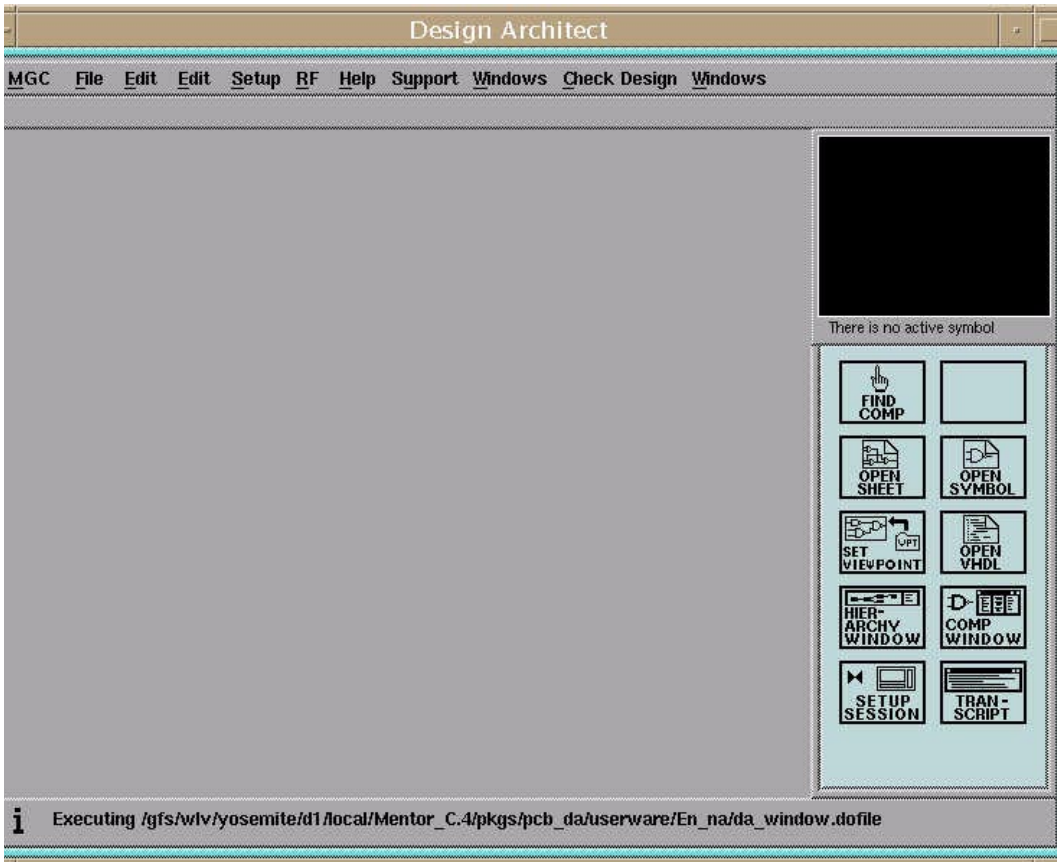


Figure 6-2. Mentor Graphics Design Architect Window

3. From the Mentor Graphics da window, choose **RF > Export IFF**. The *IFF Write* dialog appears.
4. Specify the name of the file where you wish the schematic data to be stored in IFF format. Use the Navigator button if you wish to graphically browse for a directory. The revert button will reset the file name to `./schematic`. The cancel button will abort the export.



Figure 6-3. Mentor Graphics IFF Write Dialog

Completing the Export

After you have specified the output file name on the IFF Write dialog, click **OK** to begin the translation.

After the translation begins, all export log information is displayed in the terminal window in which Design Architect was started. Any warnings or error messages encountered during the translation are also displayed in the log messages. After the export finishes, the log file will be displayed in Design Architect, in a Notepad window.

Note Mentor Graphics does not provide any export options. There is no way to limit the amount of hierarchy exported, it will always export the full schematic hierarchy. However, parts belonging to an LMS library will not be exported.

Exporting an IFF Layout File

This section describes the procedure for exporting an RF Region to an IFF layout file using the IFF Translator from a Boardstation RF Layout window. Only one RF region may be exported at a time.

The menu items described in this section are only available if the correct *Ample* path has been set up for the RF Architect option of Design Architect. For more information, refer to [Chapter 2, Before Using the IFF Translators](#).

Accessing the IFF Write Dialog in RF Layout

To access the layout export tools, launch your export from the Mentor Graphics RF Layout window.

1. In a terminal window, change to the appropriate directory.
2. Run Design Architect by typing the command `layout design -rf`. The Mentor Graphics RF Layout window appears.

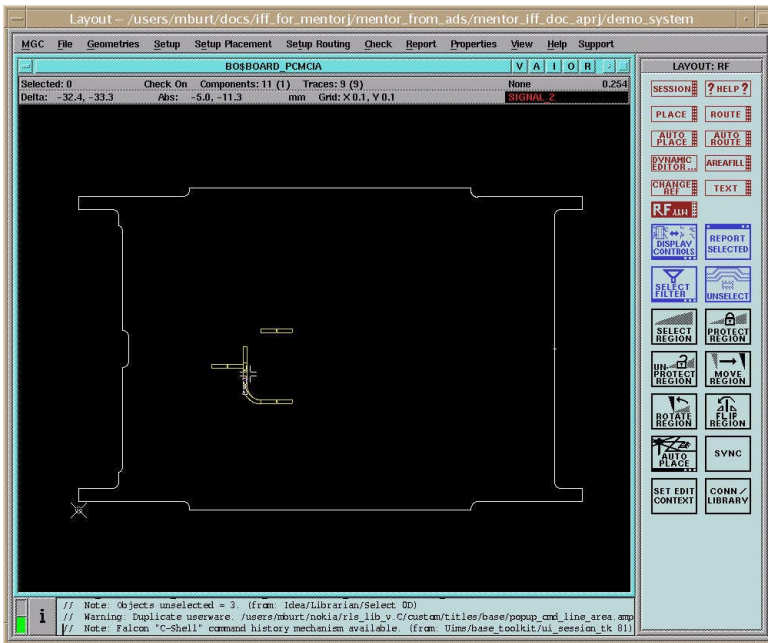


Figure 6-4. RF Layout Window

3. Click the right mouse button. On the popup menu, select `RF > Export to IFF`. The Export Layout to IFF dialog comes up.



Figure 6-5. Export Layout to IFF Dialog

Output File for IFF

There are two possible choices for this option, *Default* or *Specify*. If you leave *Default* checked, the IFF file will be named automatically, using the naming formula `<design>/eesof_pcbinfo/to_hp/<region>.xfer/layout.iff`. If you would prefer to output the file to a different location, select the *Specify* radio button. This will change the dialog, so you can enter in your own file pathname. A Navigator button enables you to graphically browse for a file name.

Region

There will additionally be a list box, that contains the RF Regions that exist for the current board design. You must select one of these regions. The data for that particular RF Region will be output into the IFF file that was specified.

Completing the IFF Export

After specifying the options, click **OK** in the Export Layout to IFF dialog box to begin the translation. Press **Cancel** to abort the export, or **Reset** to reset the dialog options to their original state. Log messages will be output into the terminal window where the RF Layout process was started.

Chapter 7: Importing IFF Layout Files into ADS

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

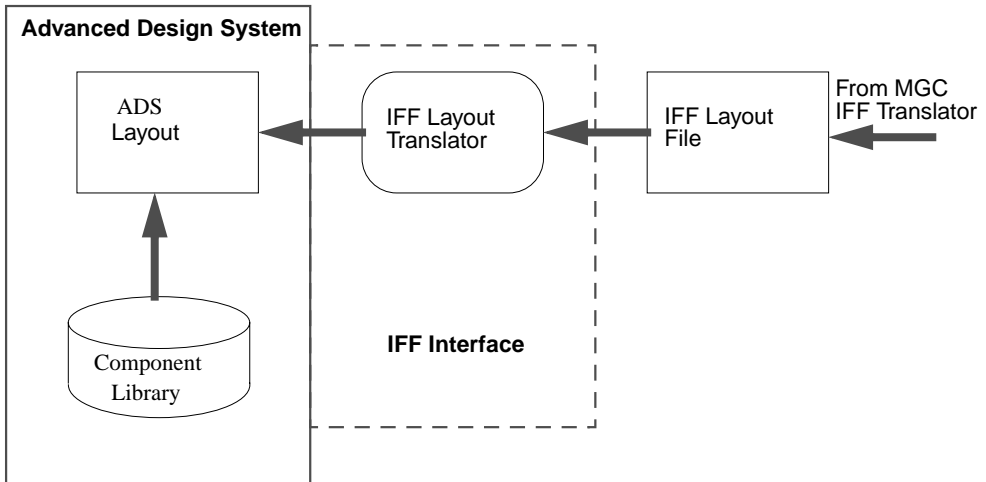


Figure 7-1. IFF Layout Import in ADS

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

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

Opening an ADS Project and Layout 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 *Schematic Capture and Layout* manual.

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 Layout window by clicking the *New Layout Window* icon in the ADS Main window tool bar.



A new ADS *untitled* layout window appears.

Importing an IFF Layout File

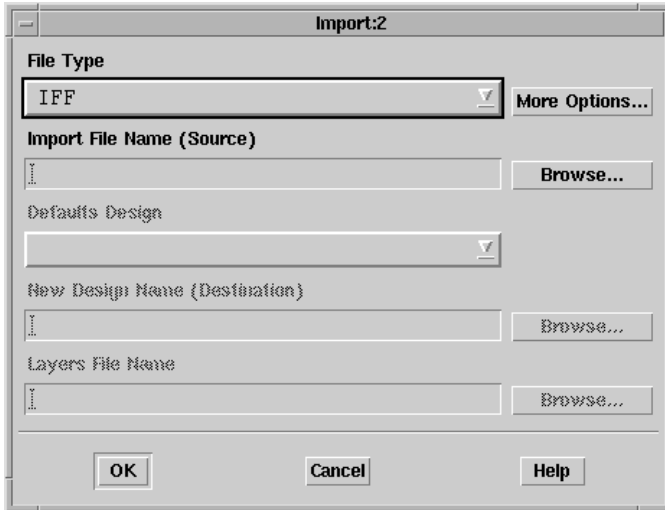
This section describes the procedure for importing an IFF layout file into Advanced Design System using the IFF Translator’s User Interface. Before importing an IFF layout file, refer to “[Opening an ADS Project and Layout Window](#)” on page 7-1.

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. ADS will force all designs to be closed prior to import. Designs are saved to disk, not to memory. If you wish to keep your existing ADS designs intact, you must make a backup copy prior to doing your import.

Accessing the Layout Import dialog

To access the layout import dialog, launch your import from an ADS *Layout* window.

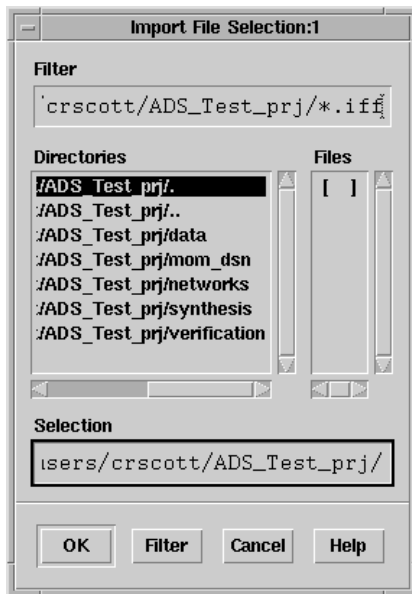
Choose **File > Import**. The layout *Import* dialog box appears.



Specifying the Import File Selection

In the layout *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.

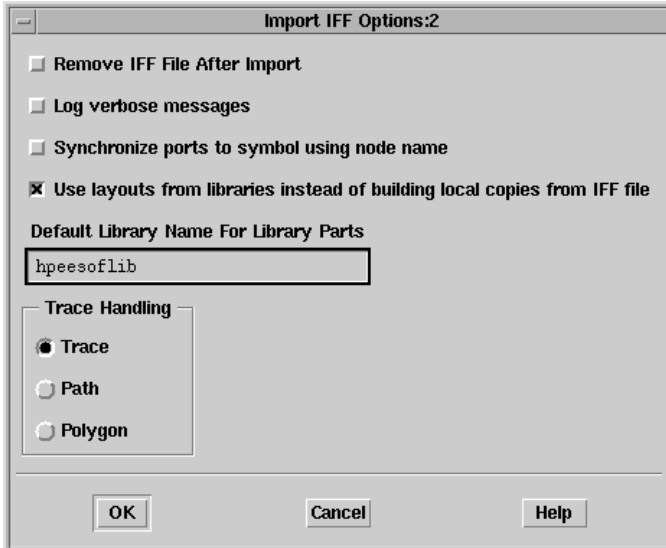


3. Double-click as needed to locate the directory containing your IFF file in the *Directories* field, then click the file in the *Files* field. Alternatively, you can enter the full path and file name in the *Selection* field.
4. After selecting the design you want to import, click **OK**. You are returned 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 layout *Import IFF Options* dialog box from the Import dialog, click **More Options**. The layout *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 *iff_{to}lib.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 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. Mentor Graphics will always provide a library name.

Use layouts from libraries instead of building local copies from IFF file

Layout IFF files contain all of the graphics primitives required to represent a footprint of a component. This is a historical requirement, dating back to when it was not possible to represent components parametrically in Mentor Graphics. When this option is not checked, all footprints and transmission line components will have new components created for them in ADS. This will have the effect of flattening any parametrically defined footprints/layouts (e.g. an MLIN component). Normally, this option should be checked. When checked, ADS will consult with the Library Browser, and get centrally defined layout definitions, including parameterized layout definitions, for all footprints. If a definition cannot be found, the geometries in the IFF file will be used.

Trace Handling

ADS supports three different ways of interpreting metal lines, as a Trace, as a Path, or as a Polygon. This option allows you to specify how you wish to handle metal lines. Specifying trace means that there will be a connection point for the metal line on each end, making the metal line similar to a microstrip element. Specifying a path means that the metal line will be a path. Paths in ADS have a width, and are defined by points, but they do not have connectivity pins like a trace does. Specifying a polygon means the metal line will instead be created as a polygon. A polygon, like a path, also does not have any pins to define connectivity.

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 layout files, refer to [“Exporting IFF Layout Files from ADS” on page 8-1.](#)

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 8: Exporting IFF Layout Files from ADS

This chapter describes the procedure for exporting an Advanced Design System design, including layout and schematic, to Intermediate File Format (IFF).

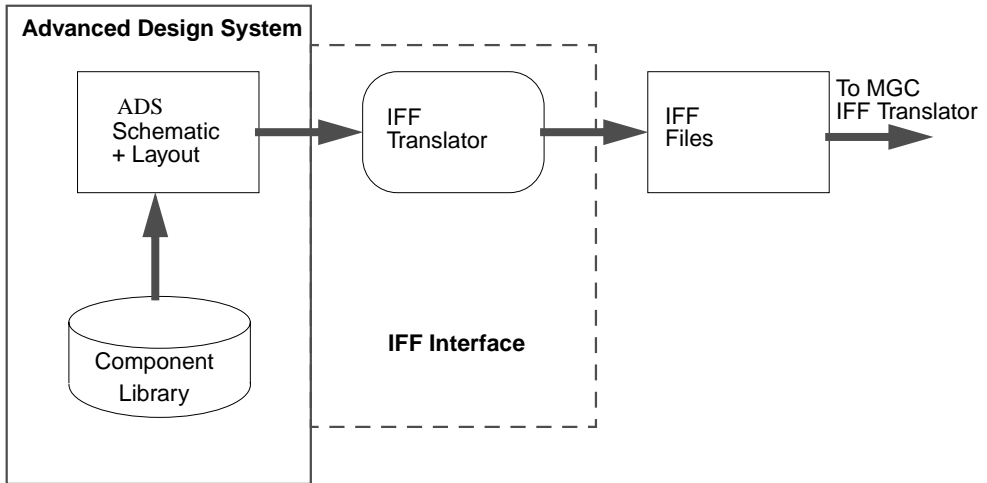


Figure 8-1. IFF Layout and Schematic Export from ADS

The basic procedure for exporting a design from ADS can be broken down into several simple steps:

1. [“Opening an ADS Project and Layout Window” on page 7-1](#)
2. [“Accessing the MGC/PCB Export dialog” on page 8-2](#)
3. [“Selecting MGC/PCB Export Options” on page 8-3](#)
4. [“Completing the MGC/PCB Export” on page 8-8](#)

Exporting an IFF Layout File

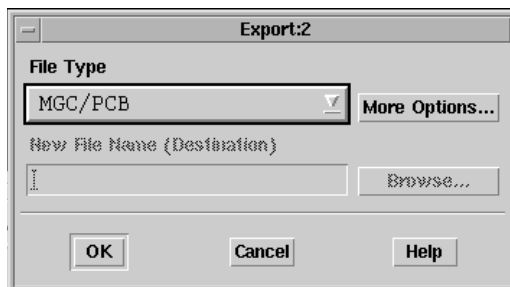
This section describes the procedure for exporting an IFF layout file from Advanced Design System using the MGC/PCB Translator’s User Interface. Before exporting an IFF layout file, refer to [“Opening an ADS Project and Layout Window” on page 7-1](#).

Accessing the MGC/PCB Export dialog

To access the PCB/MGC export dialog, launch your export from an ADS *Layout* window.

1. Open your ADS Layout Design in the ADS Layout window.
2. Choose **File > Export**. The schematic *Export* dialog box appears. Select MGC/PCB for the File Type.

Note It is recommended that the MGC/PCB export option be used in favor of the IFF option when transferring layouts to Mentor Graphics.



Where do the files go?

When exporting schematic data to IFF files, you are able to choose the destination file name. The MGC/PCB Export does not allow you to specify where to save the data. The MGC/PCB export is actually exporting both the schematic data and the layout data simultaneously. Mentor Graphics Boardstation, in order to do packaging and layout, requires that there be a schematic associated with a layout. It is actually impossible to import the layout data alone into Mentor Graphics - you must import both schematic and layout data simultaneously.

Because both files are needed to do a successful import, the MGC/PCB export hard codes the file names, so that they will be easily identified when you later choose to import them into Mentor. In the project containing your design, a new directory, `to_mgc`, will be created. Within that directory, a directory will be created, called `<design name>.hpxfer`. Within that directory, there will be three files:

- `design_info` - contains optional user comments.

- *layout.iff* - contains the layout representation of the design.
- *schematic.iff* - contains the schematic representation of the design.

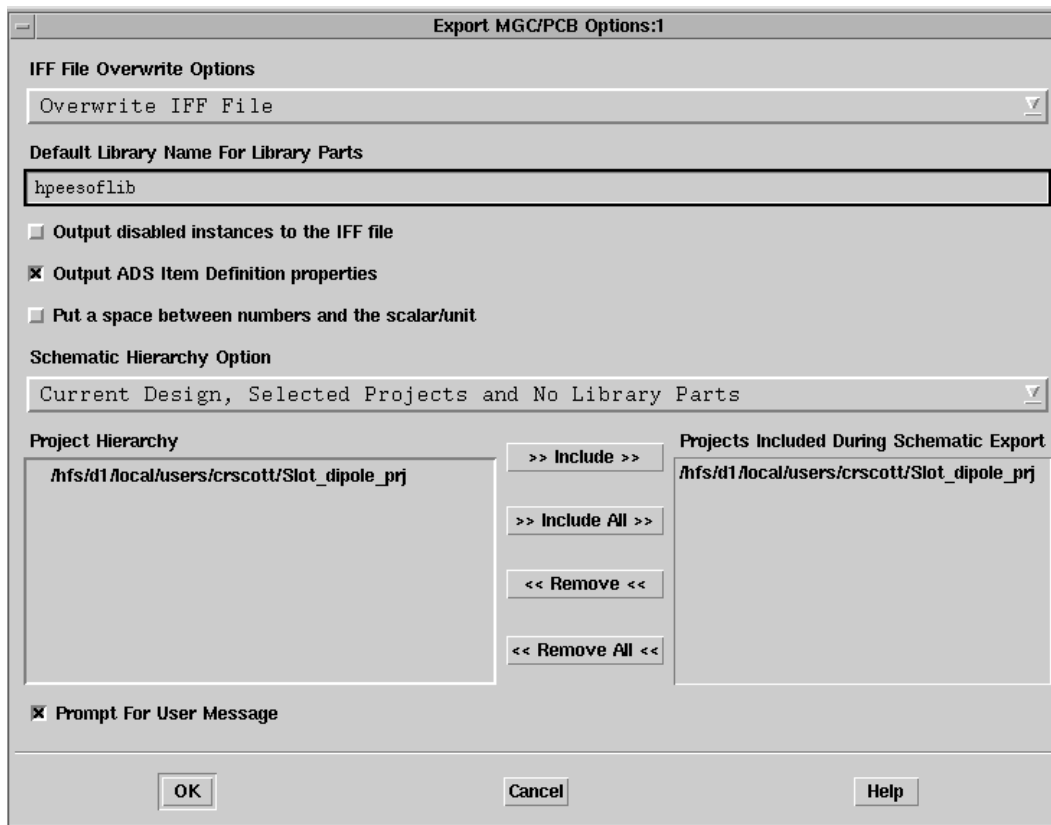
For example, if a design was created called *demo*, in the project directory, a new directory would be created called *to_mgc*. Within *to_mgc*, there would be a directory called *demo.hpxfer*. Within *demo.hpxfer*, there would be a file called *design_info*, a file called *layout.iff*, and a file called *schematic.iff*.

Selecting MGC/PCB Export Options

This section describes the choices available in the schematic MGC/PCB *Export Options* dialog box.

To access the MGC/PCB *Export IFF Options* dialog box from the Export dialog

1. Click **More Options**. The MGC/PCB Export IFF Options dialog box appears.



2. In the MGC/PCB Export Options dialog box, select the appropriate options for your translation using the information below.

IFF File Overwrite Options

Overwrite IFF File - When writing to an existing file, the contents of that file are overwritten. This is the default setting, and, for MGC/PCB export, you should always use this setting.

Append to IFF File - When writing to an existing file, the new file is appended to the existing file. 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 Mentor Graphics, 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. Mentor Graphics does not interpret the IFF property values in any way, and expects that the values from ADS will have a space in them. The RF Architect ADS library is set up to expect values to have a space between a number and a scalar/unit. If the space is not there, the ADS

components (e.g. MLIN) will not be placed properly. For an MGC/PCB transfer, this option should always be selected. 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 *hpeesoffib* 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 design 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.

Prompt for User Message

This option is selected by default. If the option is selected, a dialog will be brought up prior to the export of the layout and schematic files. You may enter as much text into this dialog as you want. The text will be stored in the design_info file that is created the to_mgc/<design>.hpxfer directory.

Completing the MGC/PCB Export

After specifying the export options, click **OK** in the *MGC/PCB Export 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.

Index

A

- ADS, 1-1
 - Item Definition properties, 4-6, 8-5
 - layout, 8-2
 - projects, 3-1, 7-1
 - schematic, 3-2, 4-2, 7-2
- Ample, 2-3

B

- Boardstation, 1-1, 1-2, 5-2, 6-2
 - IFF Read Dialog, 5-4

C

- component libraries, 1-3, 1-6
 - translation, 1-6
- components
 - mapping, 2-4
 - parameters, 2-4
 - translating, 4-6, 8-5
- constructing designs for IFF translation, 2-5
- coordinates, 4-8, 8-6

D

- da, 5-2, 6-2, 6-5
- da_lms, 5-2, 6-2, 6-5
- Design Architect, 1-1, 5-2
 - da, 5-2
- designs
 - design objects, 4-7, 8-6
 - identical names, 3-2, 7-2
 - referenced designs, 4-8, 8-6
 - transferring multiple designs, 4-5, 8-4
- diagnostics, 3-5, 7-5

E

- EDA, 1-1
- error messages, 3-5, 6-4, 7-5
- export log, 6-4
- exporting layouts, 8-1
- exporting schematics, 4-1, 6-1

F

- files
 - file destination, 4-4
 - file selection, 3-3, 4-2, 7-3

For, 5-2

H

- help, 2-5
- hierarchical
 - designs, 2-5
 - projects, 4-8, 8-7
- hpeesoflib, 4-7, 8-6

I

- IFF, 1-1
 - export options, 4-3, 8-3
 - import options, 3-4, 7-4
 - layout file, 7-2, 8-1
 - schematic file, 3-2, 4-1, 6-1
 - translation availability, 1-3
 - translators, 1-2, 2-1
 - use model, 1-4
 - write dialog in design architect, 6-2
 - write dialog in RF layout, 6-5
- ifftolib.log, 3-5, 7-5
- importing
 - layouts, 7-1
 - schematics, 3-1, 5-1
- input transfer form, 5-2

L

- layout
 - ADS, 8-2
 - hierarchy, 8-6
 - import, 7-1
 - layout per cellview, 2-6
- libraries
 - compatible component libraries, 1-6, 2-3, 2-4
 - component library requirements, 2-3, 3-6, 7-7
 - default library name, 4-7, 8-6
 - library development, 2-5
 - library name, 3-5, 4-7, 7-6, 8-6
- licenses, 2-1
- log messages, 6-4

M

- Mentor, 2-1
 - Design Architect, 6-2, 6-5

Mentor Graphics, 5-2
Mentor Graphics Corporation, 2-2
supported versions, 2-1

W
warning messages, 3-5, 6-4, 7-5

O

opening
ADS project and schematic window, 3-1
operating system versions, 2-1

P

parameter
spacing, 4-6, 8-5
values, 4-8, 8-6
partial translation, 1-6
PCB/MGC export dialog, 8-2
pcbrarch, 2-2
pcbrflayout, 2-2
projects
ADS, 3-1, 7-1
hierarchical, 4-8, 8-7

R

RF Architect, 2-2
RF Layout, 2-2

S

schematic
ADS, 3-2, 4-2, 7-2
export, 6-1
hierarchy, 4-7
import, 3-1, 5-1
schematics per cell, 2-6
symbol per cellview, 2-6
translation, 1-6, 2-5
setting
environment for mentor graphics, 2-2
simulation, 2-4
control elements, 2-5
software requirements, 2-1
specialized tools, 1-6
subcircuits, 2-5

T

third party frameworks, 1-3
translation
schematic, 2-5