



**February 2011**  
**IFF Transfer for PCB Design Environments**

© **Agilent Technologies, Inc. 2000-2011**

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

No part of this documentation may be reproduced in any form or by any means (including electronic storage and retrieval or translation into a foreign language) without prior agreement and written consent from Agilent Technologies, Inc. as governed by United States and international copyright laws.

**Acknowledgments**

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Mentor products and processes are registered trademarks of Mentor Graphics Corporation. \* Calibre is a trademark of Mentor Graphics Corporation in the US and other countries. "Microsoft®, Windows®, MS Windows®, Windows NT®, Windows 2000® and Windows Internet Explorer® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Oracle and Java and registered trademarks of Oracle and/or its affiliates. Other names may be trademarks of their respective owners. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. MATLAB® is a U.S. registered trademark of The Math Works, Inc.. HiSIM2 source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code in its entirety, is owned by Hiroshima University and STARC. FLEXIm is a trademark of Globetrotter Software, Incorporated. Layout Boolean Engine by Klaas Holwerda, v1.7 <http://www.xs4all.nl/~kholwerd/bool.html> . FreeType Project, Copyright (c) 1996-1999 by David Turner, Robert Wilhelm, and Werner Lemberg. QuestAgent search engine (c) 2000-2002, JObjects. Motif is a trademark of the Open Software Foundation. Netscape is a trademark of Netscape Communications Corporation. Netscape Portable Runtime (NSPR), Copyright (c) 1998-2003 The Mozilla Organization. A copy of the Mozilla Public License is at <http://www.mozilla.org/MPL/> . FFTW, The Fastest Fourier Transform in the West, Copyright (c) 1997-1999 Massachusetts Institute of Technology. All rights reserved.

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

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

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

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

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

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

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

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

as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. UMFPACK Availability: <http://www.cise.ufl.edu/research/sparse/umfpack> UMFPACK (including versions 2.2.1 and earlier, in FORTRAN) is available at <http://www.cise.ufl.edu/research/sparse> . MA38 is available in the Harwell Subroutine Library. This version of UMFPACK includes a modified form of COLAMD Version 2.0, originally released on Jan. 31, 2000, also available at <http://www.cise.ufl.edu/research/sparse> . COLAMD V2.0 is also incorporated as a built-in function in MATLAB version 6.1, by The MathWorks, Inc. <http://www.mathworks.com> . COLAMD V1.0 appears as a column-preordering in SuperLU (SuperLU is available at <http://www.netlib.org> ). UMFPACK v4.0 is a built-in routine in MATLAB 6.5. UMFPACK v4.3 is a built-in routine in MATLAB 7.1.

Qt Version 4.6.3 - Qt Notice: The Qt code was modified. Used by permission. Qt copyright: Qt Version 4.6.3, Copyright (c) 2010 by Nokia Corporation. All Rights Reserved. Qt License: Your use or distribution of Qt or any modified version of Qt implies that you agree to this License. This library is free software; you can redistribute it and/or modify it under the terms of the GNU Lesser General Public License as published by the Free Software Foundation; either version 2.1 of the License, or (at your option) any later version. This library is distributed in the hope that it will be useful, but WITHOUT ANY WARRANTY; without even the implied warranty of MERCHANTABILITY or FITNESS FOR A PARTICULAR PURPOSE. See the GNU Lesser General Public License for more details. You should have received a copy of the GNU Lesser General Public License along with this library; if not, write to the Free Software Foundation, Inc., 51 Franklin St, Fifth Floor, Boston, MA 02110-1301 USA Permission is hereby granted to use or copy this program under the terms of the GNU LGPL, provided that the Copyright, this License, and the Availability of the original version is retained on all copies. User documentation of any code that uses this code or any modified version of this code must cite the Copyright, this License, the Availability note, and "Used by permission." Permission to modify the code and to distribute modified code is granted, provided the Copyright, this License, and the Availability note are retained, and a notice that the code was modified is included. Qt Availability: <http://www.qtsoftware.com/downloads> Patches Applied to Qt can be found in the installation at: `$HPEESOF_DIR/prod/licenses/thirdparty/qt/patches`. You may also contact Brian Buchanan at Agilent Inc. at [brian\\_buchanan@agilent.com](mailto:brian_buchanan@agilent.com) for more information.

The HiSIM\_HV source code, and all copyrights, trade secrets or other intellectual property rights in and to the source code, is owned by Hiroshima University and/or STARC.

**Errata** The ADS product may contain references to "HP" or "HPEESOF" such as in file names and directory names. The business entity formerly known as "HP EEsof" is now part of Agilent Technologies and is known as "Agilent EEsof". To avoid broken functionality and to maintain backward compatibility for our customers, we did not change all the names and labels that contain "HP" or "HPEESOF" references.

**Warranty** The material contained in this document is provided "as is", and is subject to being changed, without notice, in future editions. Further, to the maximum extent permitted by applicable law, Agilent disclaims all warranties, either express or implied, with regard to this documentation and any information contained herein, including but not limited to the implied warranties of merchantability and fitness for a particular purpose. Agilent shall not be liable for errors or for incidental or consequential damages in connection with the furnishing, use, or performance of this document or of any information contained herein. Should Agilent and the user have a separate written agreement with warranty terms covering the material in this document that conflict with these terms, the warranty terms in the separate agreement shall control.

**Technology Licenses** The hardware and/or software described in this document are furnished under a license and may be used or copied only in accordance with the terms of such license. Portions of this product include the SystemC software licensed under Open Source terms, which are available for download at <http://systemc.org/>. This software is redistributed by Agilent. The Contributors of the SystemC software provide this software "as is" and offer no warranty of any kind, express or implied, including without limitation warranties or conditions or title and non-infringement, and implied warranties or conditions merchantability and fitness for a particular purpose. Contributors shall not be liable for any damages of any kind including without limitation direct, indirect, special, incidental and consequential damages, such as lost profits. Any provisions that differ from this disclaimer are offered by Agilent only.

**Restricted Rights Legend** U.S. Government Restricted Rights. Software and technical data rights granted to the federal government include only those rights customarily provided to end user customers. Agilent provides this customary commercial license in Software and technical data pursuant to FAR 12.211 (Technical Data) and 12.212 (Computer Software) and, for the Department of Defense, DFARS 252.227-7015 (Technical Data - Commercial Items) and DFARS 227.7202-3 (Rights in Commercial Computer Software or Computer Software Documentation).

About IFF Translation for PCB Design Environments . . . . .	7
Advanced Design System . . . . .	7
Boardstation . . . . .	7
Allegro . . . . .	7
The IFF Translator . . . . .	8
Intended Audience . . . . .	8
Main Requirements . . . . .	8
The IFF Transfer Process . . . . .	9
What's in this Documentation . . . . .	10
About Design Translation . . . . .	11
Before Using the IFF Translators with PCB Design Environments . . . . .	12
Configuring your Software for IFF Translation . . . . .	12
Understanding Component Library Requirements . . . . .	13
Known Issues and Limitations . . . . .	15
Importing IFF Schematic Files into ADS from PCB Design Environments . . . . .	16
Opening an ADS Workspace and Schematic Window . . . . .	16
Importing an IFF Schematic File . . . . .	17
Exporting IFF Schematic Files from ADS to PCB Design Environments . . . . .	20
Exporting an IFF Schematic File . . . . .	20
Importing IFF Files into ADS from PCB Design Environments . . . . .	26
Opening an ADS Workspace and Layout Window . . . . .	26
Accessing the */PCB Import dialog . . . . .	27
Selecting */PCB Import Options . . . . .	28
Completing the IFF Import . . . . .	30
Exporting IFF Files from ADS to PCB Design Environments . . . . .	31

# About IFF Translation for PCB Design Environments

Many of today's design engineers prefer to use a combination of Agilent ADS and enterprise PCB EDA 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 PCB design 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 and Allegro 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, 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 generating 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) style libraries, and the special library that is provided with Mentor Graphics' 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 allows the user to create parameterized layout instances, which is a requirement when generating transmission line components in a PC Board environment.

## Allegro

Allegro from Cadence Design Systems is an enterprise level PCB design environment supporting schematic design capture and board physical layout. The Allegro IFF interface to ADS uses the same LMS-style intermediate files as Mentor Boardstation. Optional products support the import and creation of ADS-style schematics into Allegro through the



IFF link.

## 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.

### IFF Interface Major Benefits

The IFF interface enables you to translate schematic information between Advanced Design System and PCB design environments, 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 perform the following:

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

## Intended Audience

The audience intended for this documentation 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 the PCB design environment.

## Main Requirements

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



- The link between ADS and the PCB design environment is available and installed. The IFF translator for the PCB environment is supplied and supported by the PCB tool vendor.
- The component libraries used for creating the schematic in ADS and the PCB tool have been implemented to support translation via IFF. For more detailed information on library requirements, refer to *Understanding Component Library Requirements* (iffm). If equivalent libraries have not been set up for the PCB library components in ADS, the PCB library components will not be usable in simulations.

**Note** Agilent Technologies supports the IFF Import and Export tools for ADS. The PCB tool vendor is responsible for their product, which includes ADS equivalent part libraries as well as IFF Import and IFF export. ADS and PCB environment products are licensed separately. Contact your PCB tool vendor for information on IFF translation availability for their products.

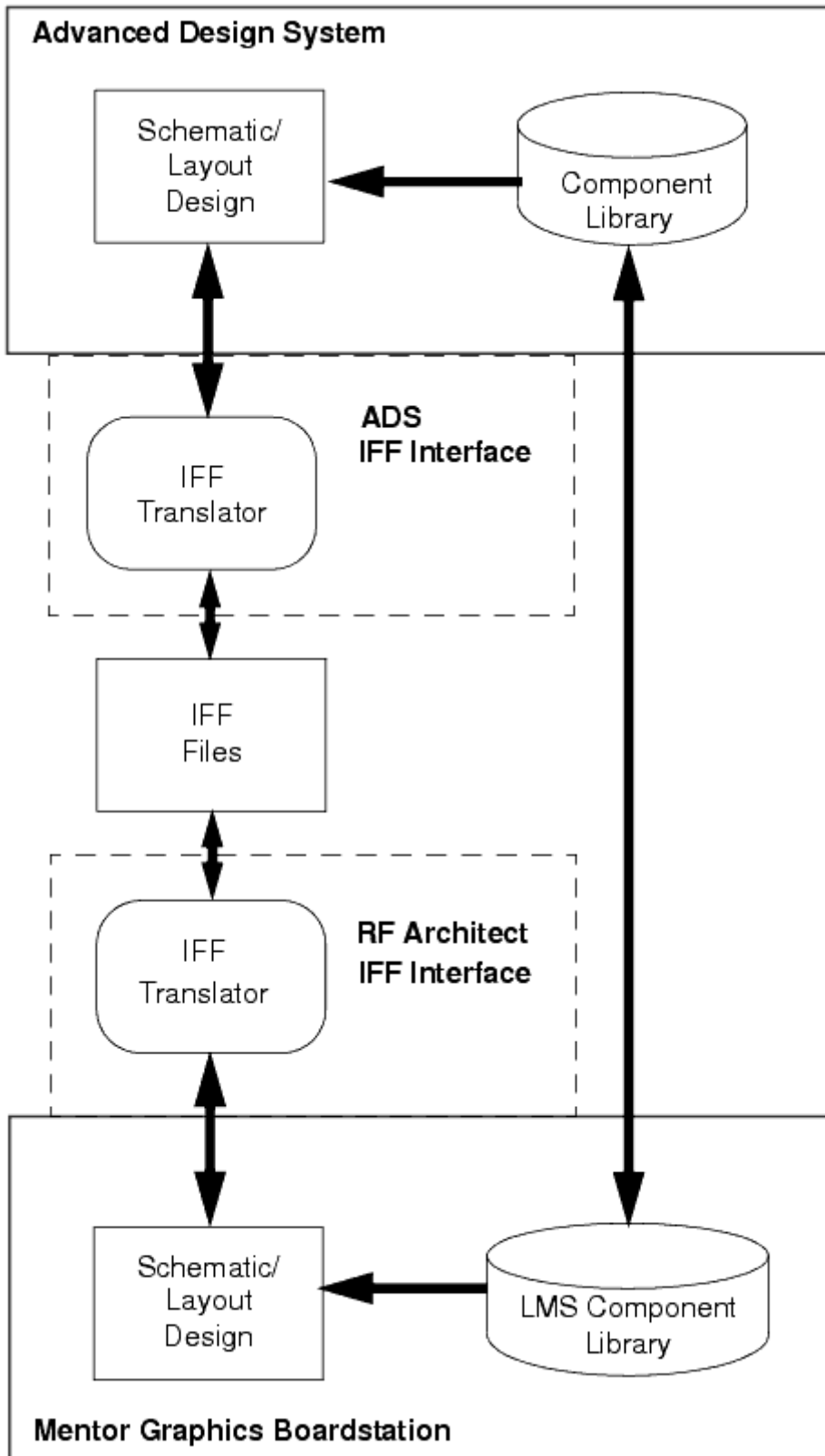
## The IFF Transfer Process

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

The diagram shown in *IFF Schematic and Layout Transfer Use Model* (adshbapp) describes the general use model for translating a design using the IFF Interface as it applies to Advanced Design System and PCB packages, using Mentor Boardstation as a typical example. The link between the two EDA environments is established via the IFF Interface. The component libraries in both ADS and the PCB environment must be compatible to support an IFF translation.

For more information on the library requirements, refer to *Before Using the IFF Translators* (iffm).

**Figure: IFF Schematic and Layout Transfer Use Model**



## What's in this Documentation

The goal of this documentation 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 documentation contains the following:

- *Before Using the IFF Translators with PCB Design Environments (iffm)*
- *Importing IFF Schematic Files into ADS from PCB Design Environments (iffm)*
- *Exporting IFF Schematic Files from ADS to PCB Design Environments (iffm)*
- *Importing IFF Files into ADS from PCB Design Environments (iffm)*
- *Exporting IFF Files from ADS to PCB Design Environments (iffm)*

## 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 PCB environment library into an ADS library and vice versa. To complete the translation task however, a significant amount of manual intervention is required.

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

# Before Using the IFF Translators with PCB Design Environments

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

- [Configuring your Software for IFF Translation](#)
- [Understanding Component Library Requirements](#)
- [Constructing Designs for IFF Translation](#)

After covering the information in this section, you'll be ready to begin sharing your designs between ADS and your PCB environment 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 documentation requires ADS 2011 or newer. For version requirements for your PCB environment, please contact your vendor representative.

### License Requirements

For ADS, no licenses are required beyond the basic set. For license requirements for your PCB environment, please contact your vendor representative.

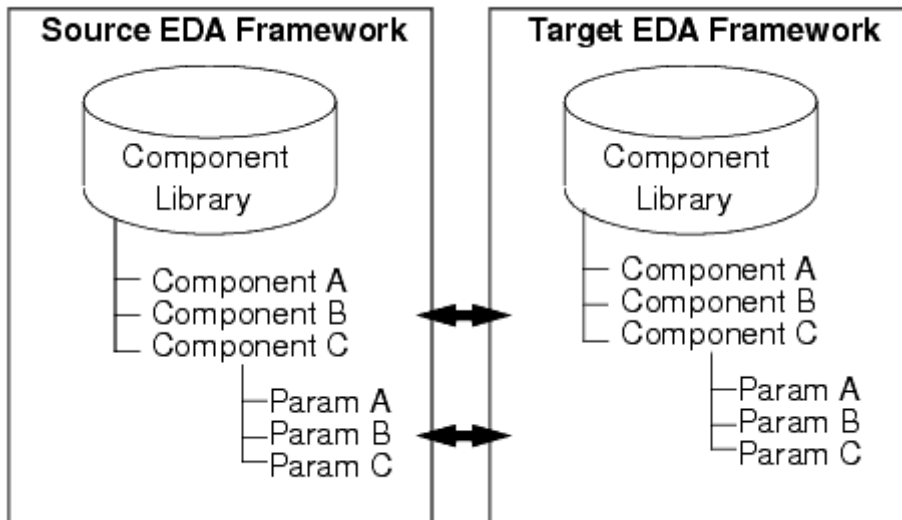
### Setting Environment for PCB Design Tools

Some PCB design tools may require special setup to enable IFF import/export and other RF features. Please consult your PCB environment manuals or representative for the details.

## Understanding Component Library Requirements

The fundamental library requirement for a successful IFF transfer is that both ADS and the PCB design environment 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.



### 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 the figure [Compatible Component Libraries](#), 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 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 schematic and layout transfer between the ADS and your PCB design environment. 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](#).

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 should not be transferred between ADS and the PCB tool. 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 allow you to set up any simulation controls independently in the PCB design environment. For more information, refer to *Creating Hierarchical Designs* (usrguide).

## Known Issues and Limitations

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

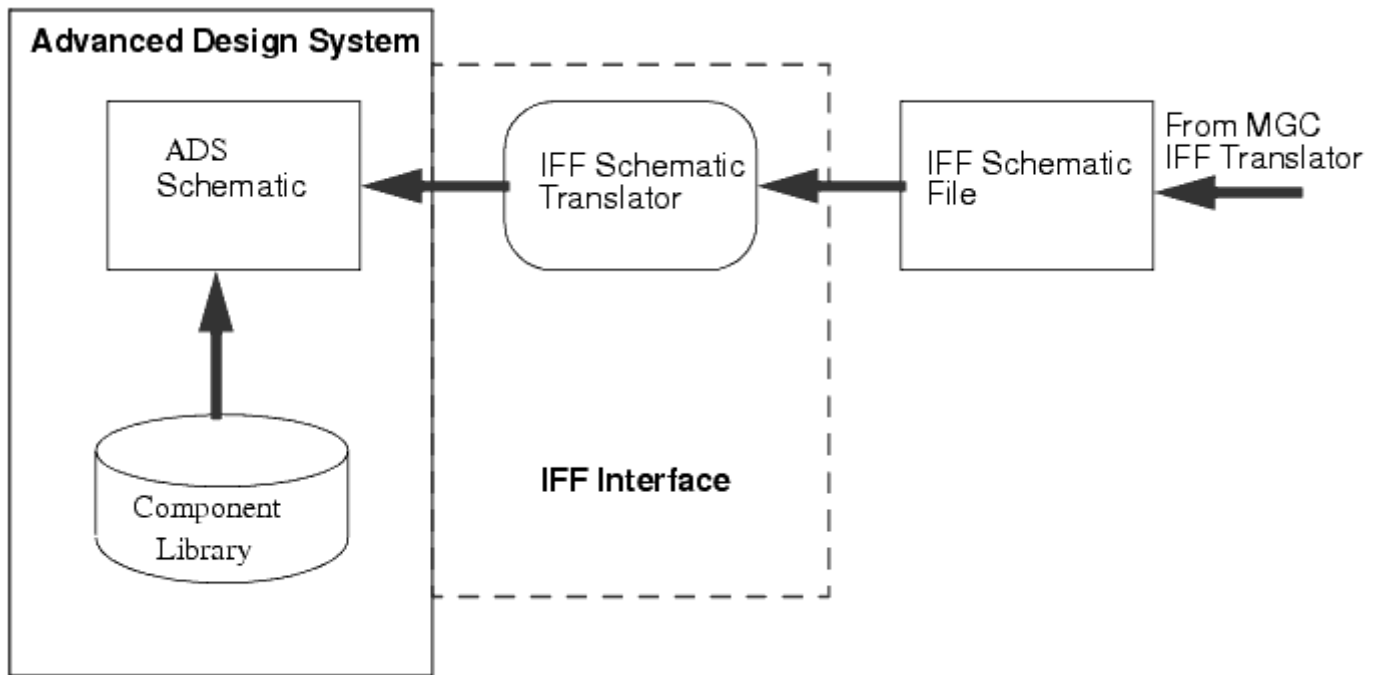
- Only one schematic is allowed per cell in Advanced Design System.
- Only one symbol is allowed per cell in Advanced Design System.
- Only one layout is allowed per cell in Advanced Design System.
- Schematic view points can not be exported from Mentor Graphics Boardstation. Only the base schematic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



# Importing IFF Schematic Files into ADS from PCB Design Environments

This section describes the procedure for importing an Intermediate File Format (IFF) schematic file produced by a PCB design tool into ADS.

Figure: Typical IFF Schematic Import Flow - Mentor Boardstation



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


1. [Opening an ADS Workspace and Schematic Window](#)
2. [Accessing the Schematic Import dialog](#)
3. [Specifying the Import File Selection](#)
4. [Selecting Import IFF Options](#)
5. [Completing the IFF Import](#)

## Opening an ADS Workspace and Schematic Window

Open a workspace in ADS before attempting to import your design. Working in workspace 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 workspace. From the ADS Main window:

1. Choose **File > New > Workspace** to open a new workspace or **File > Open >**

**Workspace** to open an existing workspace. For more information on working in workspace directories, refer to *Using Workspace* (adstour).

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

## Importing an IFF Schematic File

This section describes the procedure for importing an IFF schematic file into ADS using the IFF Translator's User Interface.

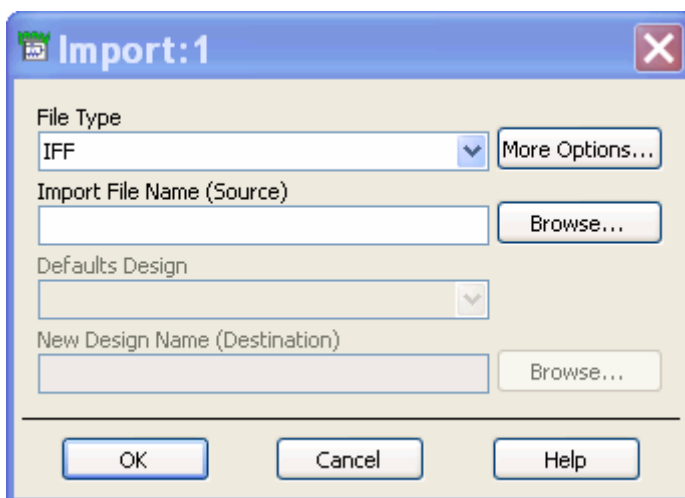
### Caution

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 box:

1. Open ADS Schematic view.
2. Choose **File > Import** to open the schematic **Import** dialog box.



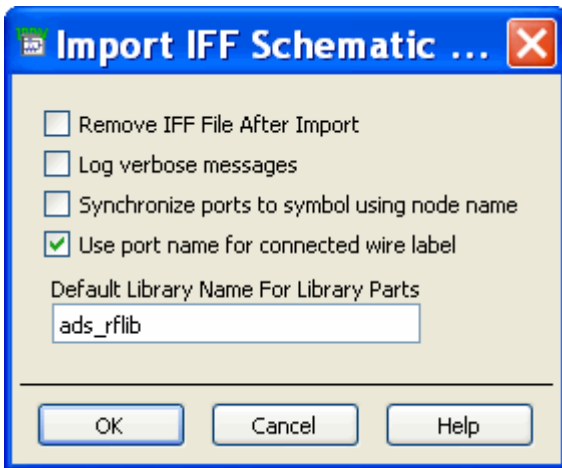
## Specifying the Import File Selection

In the schematic **Import** dialog box, choose the type of file to import, specify the file name, 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. Click **Browse** to specify the path and file name of the file you want to import. The **Import File Selection** dialog box appears.
3. Double-click as needed to locate the directory containing the IFF file. Alternatively, you can enter the full path and file name.
4. Select the file you want to import and click **Open**. You are returned to the **Import** dialog box and the selected file name 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. For more details on Import IFF Options, see [Selecting Import IFF Options](#).

## Selecting Import IFF Options

This section describes the choices available in the **Import IFF Options** dialog box. Select the appropriate options for your translation using the information below.



**Note**  
Import options for other file formats are explained in *Importing and Exporting Designs* (trans).

### Remove IFF File After Import

When this option is 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 this option is selected, all translation information is recorded in the `ifftolib.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

When this option is selected, the IFF import resets the symbol pin numbers to match port numbers based on the node name of the schematic port. By default, symbol pin numbers are matched to schematic port numbers based on the port's instance name.

## Use port name for connected wire label

For each wire attached to a port, the wire label will be set to name of the port in the IFF file. ADS does not allow ports to share a common name. This option is useful in tools where ports may have the same name to share a connection, since connectivity through the ports will be maintained in ADS even though the port names differ.

## 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.

### 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 (iffm)*.

## 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 documentation. 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 Solution Services.

## Completing the IFF Import

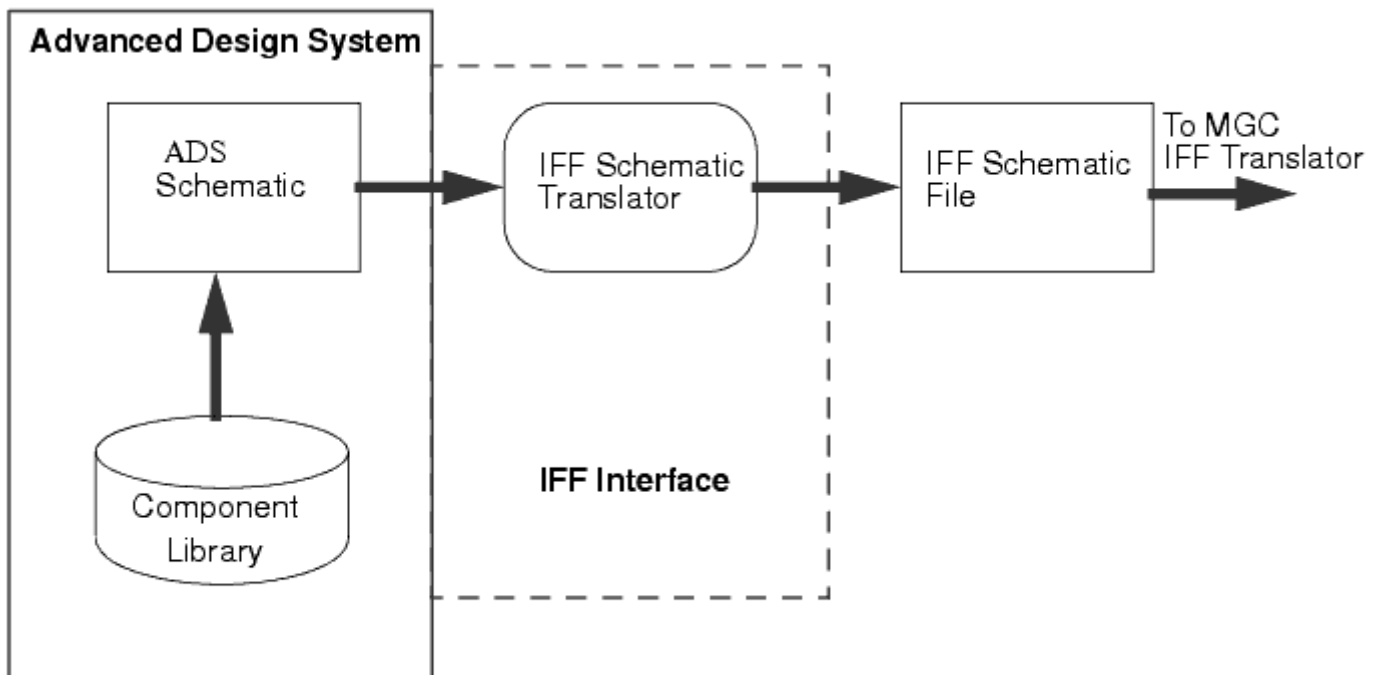
After specifying the IFF import options, click **OK** in the **Import IFF Options** dialog box to save your settings or **Cancel** to retain the initial settings. After clicking **OK**, you are returned to the **Import** dialog box. Click **OK** to begin the translation.

When translation is complete, a *Status* window appears which logs the progress and completion of the import process, including any error messages or warnings generated during import.

# Exporting IFF Schematic Files from ADS to PCB Design Environments

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

Figure: Typical IFF Schematic Export Flow - Mentor Boardstation



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

1. *Opening an ADS Workspace and Schematic Window (iffm)*
2. [Accessing the Schematic Export dialog](#)
3. [Specifying the Export File Selection](#)
4. [Selecting Export IFF Options](#)
5. [Completing the IFF Export](#)

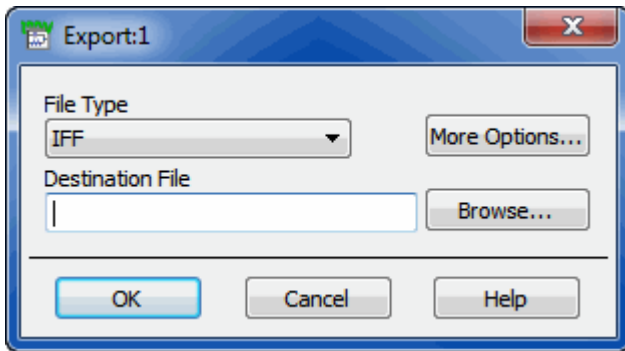
## Exporting an IFF Schematic File

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 Workspace and Schematic Window (iffm)*.

### Accessing the Schematic Export dialog

To access the schematic export dialog box:

1. Open ADS Schematic view.
2. Choose **File > Export**. The schematic **Export** dialog box appears.



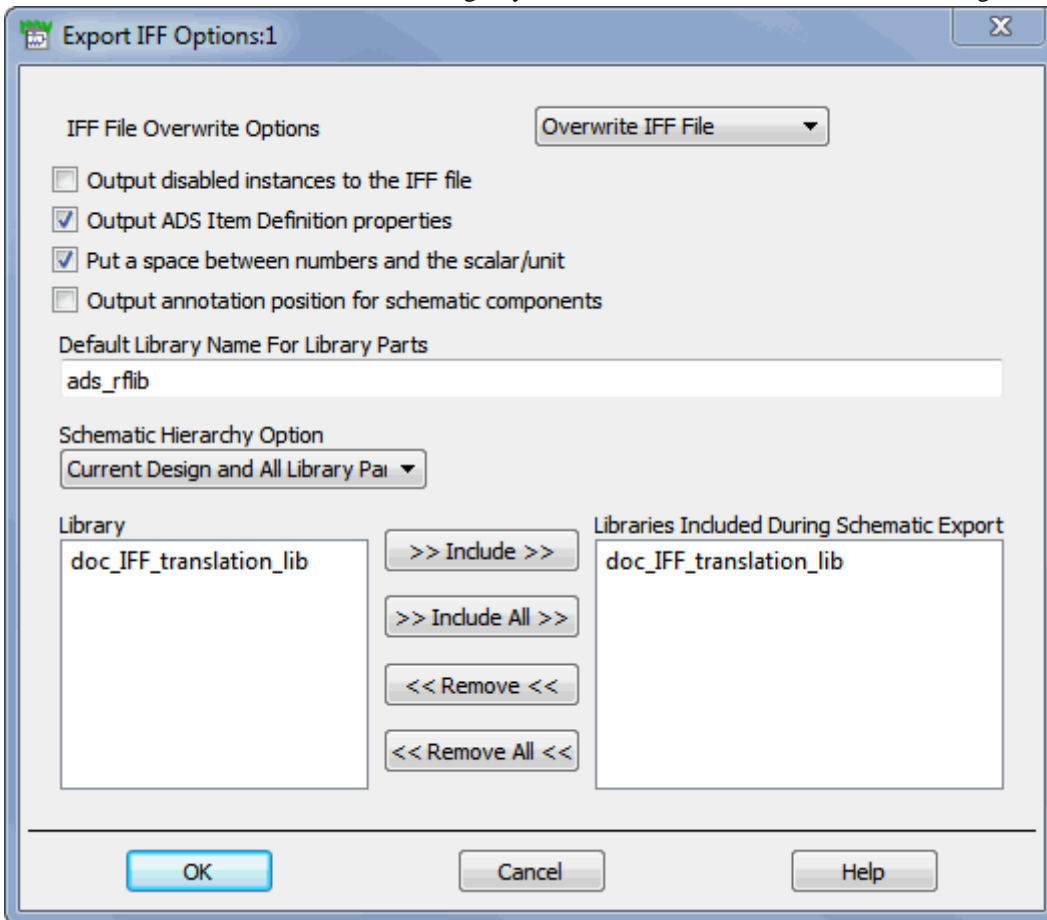
## Specifying the Export File Selection

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. In the **Destination File Name** specify the full path of the destination file you want to export. Alternatively, click **Browse** to open **Export File Selection** dialog box .
3. After selecting the destination location, click **Save** to return to the **Export** dialog box. The selected path and file name appear in **Destination File Name**. 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.
4. Click **More Options** to define the export options. The **Export IFF Options** dialog box appears. For more details on Export IFF Options, see [Selecting Export IFF Options](#).

## Selecting Export IFF Options

This section describes the choices available in the schematic **Export IFF 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 generally the preferred 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. 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.

## 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.

## Output annotation position for schematic components

This option stores information about the position of the component text annotation in the schematic window. Annotation text includes the component type, name and parameters, as displayed next to the component. Enabling this option may be useful if the annotation

has been moved in ADS. However, even with this option enabled, some environments may not preserve the position of the annotation.

## 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 the local workspace library, but library parts are stored in either the default library *ads\_lib* 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 documentation. 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 Solution Services.

## Schematic Hierarchy Option

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

## Library

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

### Libraries Included During Schematic Export

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

To add a library to this list:

1. In the Library list, click the desired library.
2. Click **Include**. The library is added to the Libraries Included list.

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

To remove a library from the Libraries Included list:

1. In the Libraries Included list, click the entry you want to remove.
2. Click **Remove**. The library is removed from the list.

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

### Completing the IFF Export

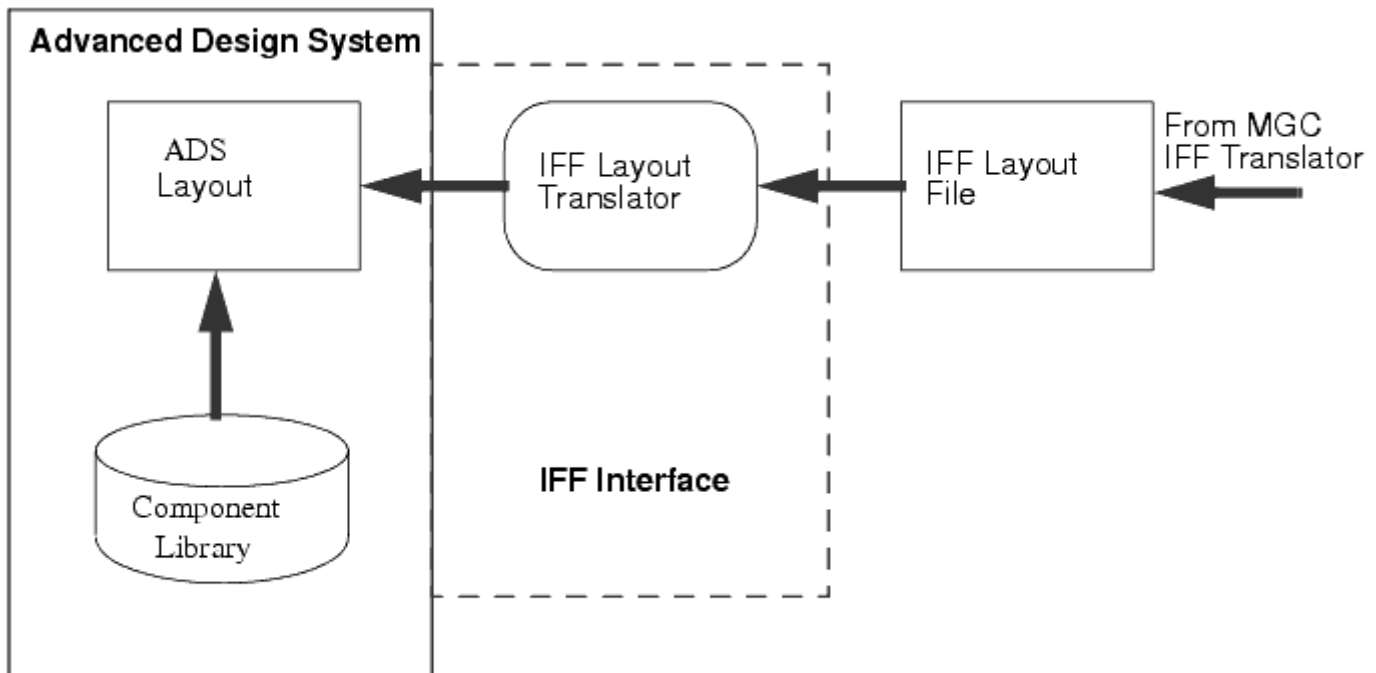
After specifying the IFF export options, click **OK** in the **Export IFF Options** dialog box to save your settings or **Cancel** to retain the default settings. After clicking **OK** , you are returned to the **Export** dialog box. Click **OK** to begin the translation.

When translation is complete, a Status window appears which logs the progress and completion of the import process, including any error messages or warnings generated during import.

# Importing IFF Files into ADS from PCB Design Environments

This section describes the procedure for importing Intermediate File Format (IFF) files from a PCB design tool into Advanced Design System.

Figure: Typical IFF Import into ADS: Mentor Boardstation example




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

1. [Opening an ADS Workspace and Layout Window](#)
2. [Accessing the \\*/PCB Import dialog](#)
3. [Selecting \\*/PCB Import Options](#)
4. [Completing the \\*/PCB Import](#)

## Opening an ADS Workspace and Layout Window

Open a workspace in ADS before attempting to import your design. From the ADS Main window:

1. Choose **File > New > Workspace** to open a new workspace or **File > Open > Workspace** to open an existing workspace.  
For more information on working in workspace directories, refer to *Using Workspace* (adstour).

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  from the ADS Main window tool bar.

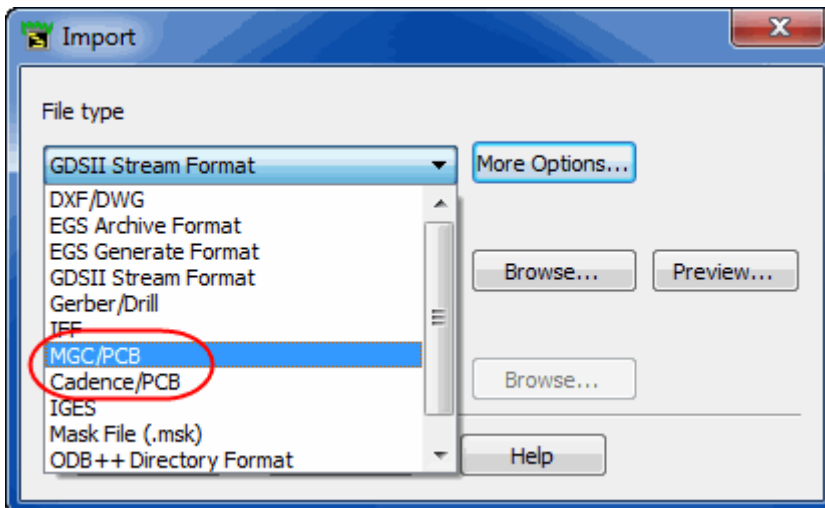
## Accessing the \*/PCB Import dialog

### Caution

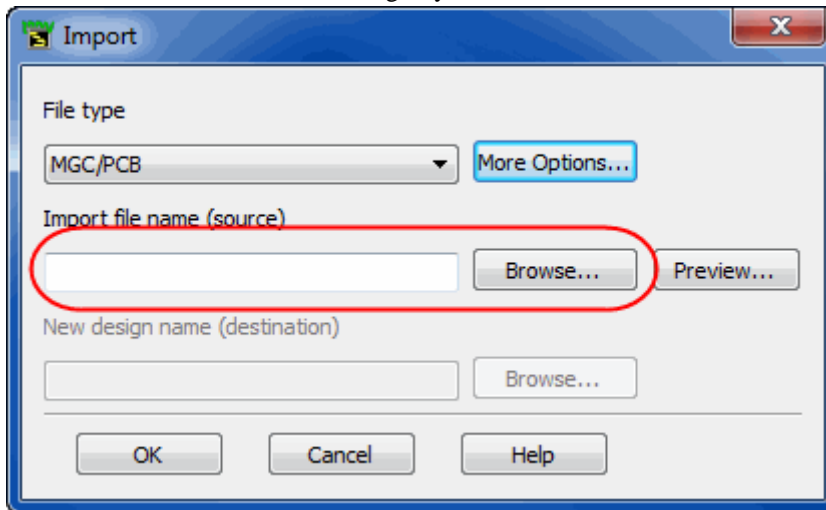
When a design being imported has the same name as an existing design, the existing design will be *overwritten with no questions asked*. The designs are not merged together. ADS will force all designs to be closed prior to import. Designs are saved to disk, not to memory. If you wish to keep your existing ADS designs intact, you must make a backup copy prior to doing your import.

To access the \*/PCB import dialog box:

1. Open ADS Layout view.
2. Choose **File > Import**. The layout **Import** dialog box appears.
3. Select one of the vendor specific \*/PCB types (currently MGC/PCB for Mentor Boardstation and Cadence/PCB for Cadence Allegro) from the **File Type** drop-down list.

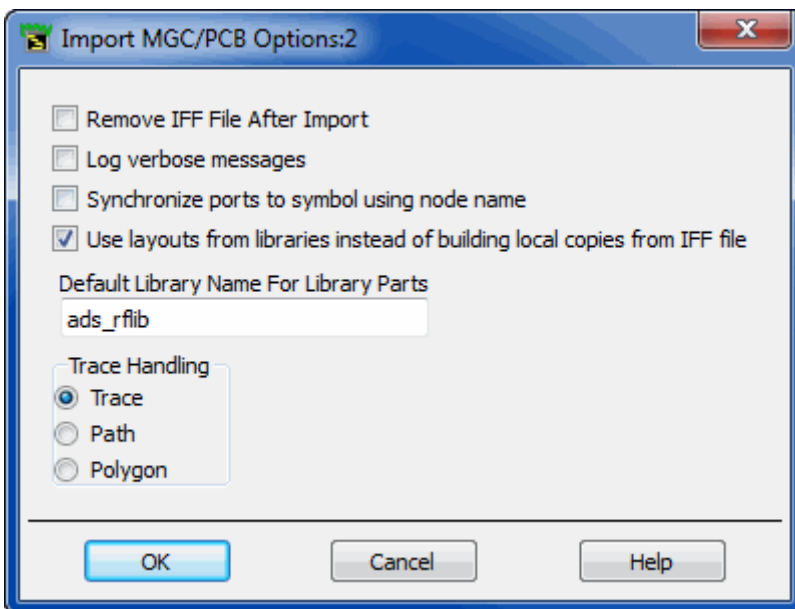


4. Browse to the folder that holds the IFF data exported from the PCB environment. This is not a single IFF file, but a folder with schematic and layout IFF files along with other data. The folder will be named after the PCB environment top level block, and there may be more than one block in the export folder (named *to\_hp* by the Mentor Boardstation export, for example).



## Selecting \*/PCB Import Options

This section describes the choices available in the **Import \*/PCB Options** dialog box. Select the appropriate options for your translation using the information below.



### Remove IFF File After Import

When this option is 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 this option is selected, all translation information is recorded in the *ifftolib.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

When this option is selected, the IFF import resets the symbol pin numbers to match port numbers based on the node name of the schematic port. By default, symbol pin numbers are matched to schematic port numbers based on the port's instance 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 the PCB tools of the day. 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.

## 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. PCB tools will always provide a library name.

## Trace Handling

The trace handling option allows you to specify how you wish to handle metal lines. ADS supports three different ways of interpreting metal lines:

- **Trace:** 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.
- **Path:** 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.

- **Polygon:** 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.

## Completing the IFF Import

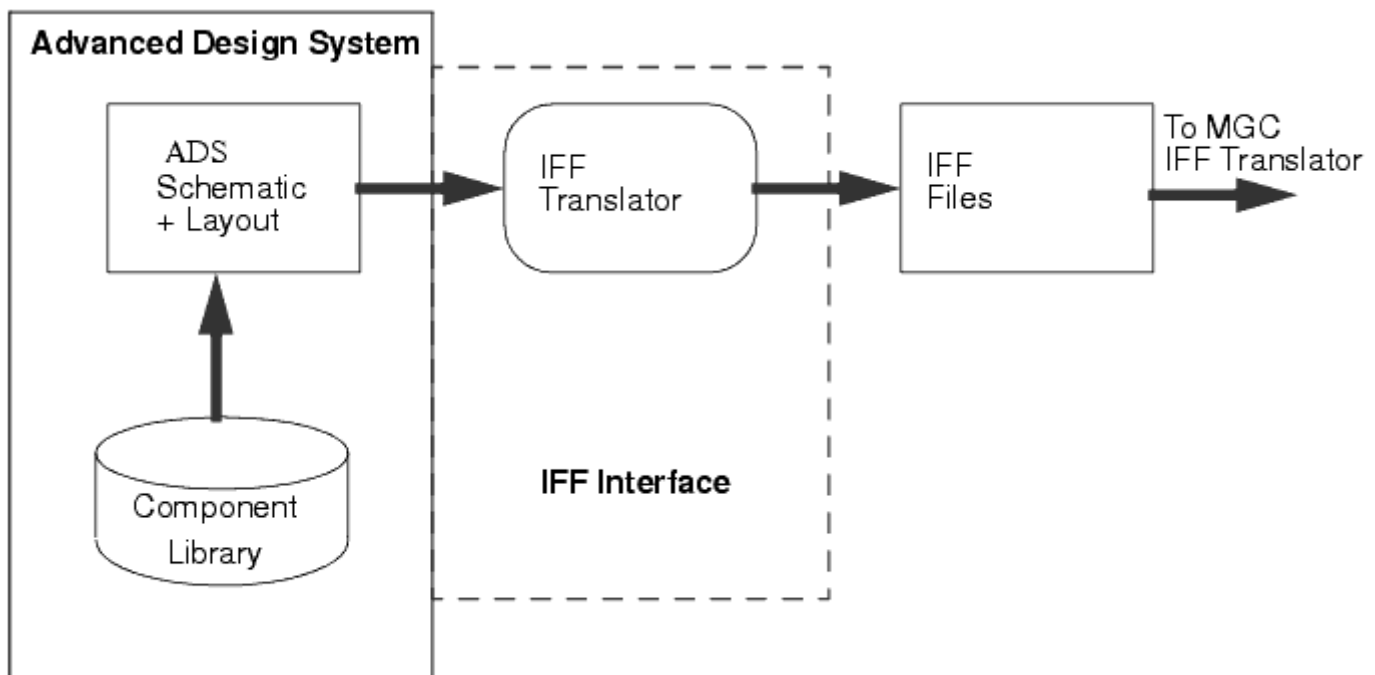
After specifying the IFF import options, click **OK** in the **Import \*/PCB Options** dialog box to save your settings or **Cancel** to retain the default settings. After clicking **OK**, you are returned to the *Import* dialog box. Click **OK** to begin the translation.

When translation is complete, a Status window appears which logs the progress and completion of the import process, including any error messages or warnings generated during import.

# Exporting IFF Files from ADS to PCB Design Environments

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

Figure: Typical IFF Layout and Schematic Export - Mentor Boardstation example



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

1. [Opening an ADS Workspace and Layout Window](#)
2. [Accessing the \\*/PCB Export dialog](#)
3. [Selecting \\*/PCB Export Options](#)
4. [Completing the \\*/PCB Export](#)

## Opening an ADS Workspace and Layout Window

Open a workspace in ADS before attempting to export your design. From the ADS Main window:

1. Choose **File > Open > Workspace** to open an existing workspace. For more information on working in workspace directories, refer to *Using Workspace* (adstour).
2. Open an existing ADS Layout window.

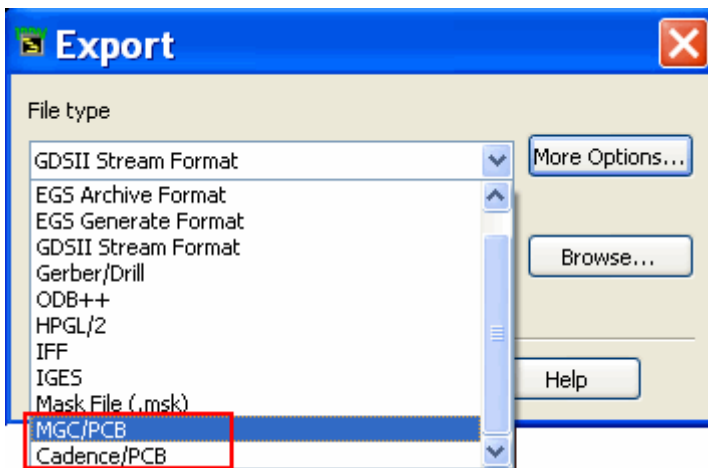
## Accessing the \*/PCB Export dialog

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

1. Open ADS Layout view.
2. Choose **File > Export** to open the layout **Export** dialog box.
3. Select one of the vendor specific \*/PCB types (currently, MGC/PCB for Mentor Boardstation and Cadence/PCB for Cadence Allegro) from the **File Type** drop-down list.

### Note

It is recommended that the \*/PCB export option be used in favor of the IFF option when transferring layouts to a PCB design environment.



## Where do the files go?

When exporting schematic data to IFF files, you are able to choose the destination file name. The \*/PCB Export does not allow you to specify where to save the data. The \*/PCB export is actually exporting both the schematic data and the layout data simultaneously. Some PCB design tools, 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. Other tool requirements may vary, but this technique ensures that all necessary information will be available for the PCB environment import.

Because both files are needed to do a successful import, the \*/PCB export hard codes the file names, so that they will be easily identified when you later choose to import them into the PCB tool. In the workspace containing your design, a new directory (currently either to\_mgc or to\_cds) will be created. Within that directory, another 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.

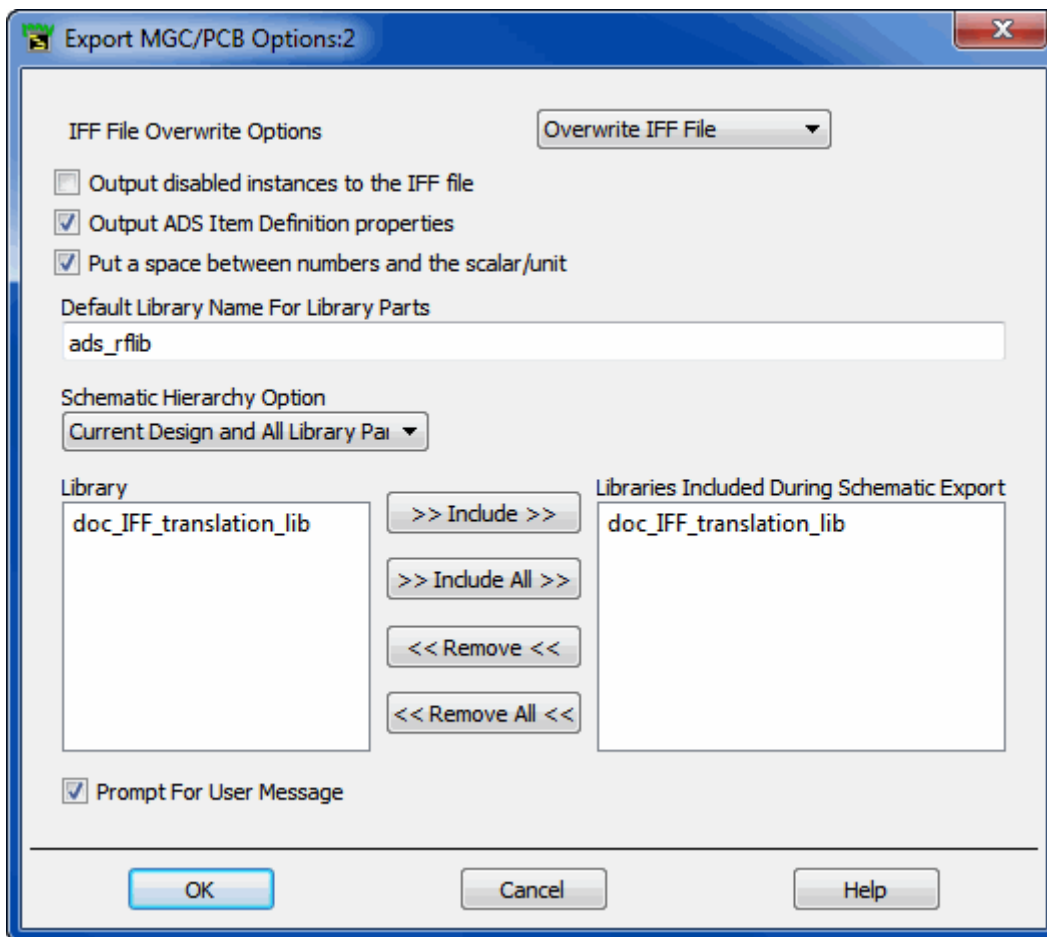
For example, if a design was created called demo, in the workspace directory, a new output directory would be created. Within that output directory, there would be a directory called *demo.hpxfer* . Within *demo.hpxfer*, there would be following three files:

- design\_info
- layout.iff
- schematic.iff

## Selecting \*/PCB Export Options

This section describes the choices available in the layout Export **/PCB Options dialog box**. **To access the \*Export \*/PCB Options dialog box**:

1. Click **More Options** in the **Export dialog** box. The **Export \*/PCB Options dialog** box appears.



2. 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 \*/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 the PCB tool, 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.

### 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 workspace directory, but library parts are stored in either the default library `_ads_lib` 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 documentation. 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 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 and All Library Parts** Complete design information for the current design is exported. Referenced designs that reside in a library selected for inclusion during export and are part of the current design's hierarchy are also exported. In

addition, library parts are exported.

## Library

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

## Libraries Included During Schematic Export

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

To add a library to this list:

1. In the Library list, click the desired library.
2. Click **Include**. The library is added to the Libraries Included list.

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

To remove a library from the Libraries Included list:

1. In the Libraries Included list, click the entry you want to remove.
2. Click **Remove**. The library is removed from the list.

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

## 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\_\*/<design>.hpxfer directory.

## Completing the \*/PCB Export

After specifying the export options:

1. Click **OK** to save your settings or **Cancel** to retain the default settings in the **Export \*/PCB Options** dialog box.
2. After clicking **OK** , you are returned to the **Export** dialog box. Click **OK** to begin the translation.
3. When translation is complete, a Status window appears which logs the progress and



completion of the import process, including any error messages or warnings generated during import.