Agilent Technologies

Advanced Design System 2011.01

Feburary 2011
Importing and Exporting Designs

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. FLEXlm 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 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 Importing and Exporting Designs

Translating CAD data between various systems can be difficult. Different CAD systems have different representations of data. For instance, some systems contain the concept of a hole and some do not. Differences in translation formats often cause data to be lost or transformed.

In Advanced Design System, translating physical designs is initiated from the layout window using the **File > Import** and **File > Export** menu commands. The program's import/export translators are highly configurable. Each translator has associated options that you can set from the Import and Export dialog boxes. These options control how the translator works. For example, with the IGES and GDSII format *flatten* option, a layout can be translated as a flat, non-hierarchical design. Similarly, the *merge* option for these formats creates two new polygons from two intersecting closed shapes.

Other options include arc conversions, treatment of holes and text, and in the layers file, specification of layer translation tables. The IGES translator has the greatest number of options because IGES is such a broad standard. System defaults are included with the product and are automatically used if no others are specified.

Translating schematic designs is initiated from the ADS *Main* or *Schematic* window using the **File > Import** and **File > Export** menu commands. Similar to the Layout translators, each schematic translator has associated options that control how the translator works.

Layout Export Considerations

Before starting a layout, you should consider how the final output could affect the layout process. Different output formats impose different restrictions. The restrictions imposed by GDSII are very different from those imposed by DXF, HPGL, Mask, or Gerber.

With all export formats, you must consider layout units and data base precision. The units and precision used in layout should match those you want in the final output.

Usually no problems are associated with translating units that are in the same measurement system (such as, mils to inches or centimeters to millimeters). However, round-off errors can occur when translating from metric-to-English units or vice versa.

Similarly, no problem is associated with translating a less precise data base resolution in the program to a more precise output resolution. However, the reverse process (such as, 0.001 Layout to 0.01 GDSII) can result in loss of data.

Available File Formats

The Import and Export commands enable you to import and export files in a variety of different formats. You can import files through Advanced Design System's *Main*, *Schematic*, or *Layout* window. All file export is currently done from the ADS *Layout* window with the exception of Intermediate File Format (IFF) files. The following table

shows the available file formats and the individual import/export options.

Available Formats and Import/Export Options

Available File Formats and File Extensions	ADS Main Window	ADS Schematic Window	ADS Layout
DXF/DWG (hierarchical) (.dxf, .dwg)	-	-	Import/Export
DXF/DWG (flattened) (.dxf, .dwg)	-	-	Import
EGS Archive Format (_a)	-	-	Import/Export
EGS Generate Format (_g)	-	Import	Import/Export
GDSII Stream Format (.gds)	-	-	Import/Export
Gerber/Drill (.gbr)	-	-	Import/Export
Gerber Viewer (.msk, .gbr)	-	-	Import/Export
HPGL/2 (.hpg)	-	-	Export
IFF (.iff)	Import	Import/Export	Import/Export
IGES (.igs)	-	-	Import/Export
Mask File (.msk)	-	Import	Import/Export
MGC/PCB (.iff)	Import	-	Import/Export
Cadence/PCB	Import	-	Import/Export
ODB++	-	-	Import/Export
Netlist File [†]	Import	Import	-

[†] For detailed information on SPICE (.cir, .cki) or Spectre (.scs) netlist file import, refer to the Netlist Translator for SPICE and Spectre documentation.

Steps for importing and exporting these files are outlined in *Importing and Exporting Schematic and Layout Designs* (trans).

DXF/DWG

The Drawing Exchange Format (DXF) was developed by Autodesk for its A utoCAD product and is widely used to transfer geometric data between systems. Like the mask file format, it provides a simple geometric representation of data. DXF files can be transferred between PC-based or UNIX-based systems. DXF is a very simple file format that can be read by most CAD programs that support DXF.

DWG is a binary format, therefore the files take less time to load and save. Advanced Design System supports importing and exporting to/from AutoCAD. AutoCAD versions 12 to 2007 are supported for import. AutoCad versions 2000, 2004 and 2007 are supported for export. Carefully consider how you want to use DXF/DWG output - including layer numbering, use of holes, and polygon shapes - before beginning your layout design. Setting up the proper layout rules in ADS can save a lot of time in generating acceptable DXF/DWG output. For specific considerations or limitations, consult your AutoCAD documentation. For details, see *DXF/DWG Translator* (trans).

EGS (Archive/Generate)

Engineering Graphics System (EGS) format is a general graphics format used for capturing manually entered designs. EGS has been applied to ICs, Micro-circuits, Hybrids, and PC Board design applications. Using this format, you can easily exchange data with other programs using EGS formats.

The layout portion of Advanced Design System has adopted many of the primitive types and styles that are part of the EGS standard, but has a more enriched set of capabilities that structure information not representable in EGS. For example, EGS files cannot represent embedded arcs.

Two variations of the EGS format are:

- The *Generate* format is a flattened list of EGS primitives specified in the user-defined unit space. No additional information is supplied such as supplied with the Archive format.
- The *Archive* format is a hierarchically organized list of EGS primitives specified in the user-defined unit space. Information such as drawing shapes, layout units, database precision, and grid spacing is included.

For details, see *EGS Archive Files* (trans) and *EGS Generate Files* (trans).

GDSII Stream

 GDSII Stream Format (Calma) is an industry standard for translating final mask data to foundries. The Advanced Design System reads GDSII versions 4.0 through 6.0 and writes GDSII version 6.0.

Unlike other data formats, GDSII stream format is binary. You cannot easily view or edit a stream format file using a text editor. This format is easily translated between different CAD systems because it represents a highly restrictive data type. However, the format has a number of significant limitations; these limitations are discussed in *GDSII Stream File Translator* (trans).

Gerber/Drill

 Gerber refers to various data input formats that Gerber Scientific uses to drive its photoplotters. The Gerber format is used by photoplotters produced by other manufacturers also. Advanced Design System supports various types of Gerber output via mask files to either the Gerber or DXF translator. For details, see *Gerber Translator* (trans).

HPGL/2

HPGL/2 output is a subset of the HPGL/2 printer/plotter language. In Advanced Design

System, you can export HPGL/2 files from the Layout window. For details, see *HPGL/2 Files* (trans).

IFF

The Intermediate File Format (IFF) is an ASCII file with a simple, line-oriented command structure and a fairly rich set of constructs. This format is machine- and application-independent, thus simplifying design data transfer.

IFF files are used as the exchange mechanism when transferring designs between Advanced Design System and third-party EDA tools such as Mentor Graphics Design Architect and Cadence Analog Artist. For more information about these framework links, contact your Agilent Technologies sales representative.

For details, see *IFF Files* (trans).

IGES

 The Initial Graphics Exchange Specification (IGES) is an approved ANSI standard of the U.S. Department of Commerce (ANSI Y14.25M) that is used extensively throughout the computer-aided design and manufacturing world. The IGES format can represent both mechanical and electrical design data in two and three dimensions.

Because IGES is such a broad standard, the government has attempted to further define a stricter standard of IGES for the transfer of electrical design data. This standard is known as CALS specification. The CALS specification for IGES is officially contained in the military specification MIL-D-28000, *Digital Representation for Communication of Product Data: IGES Application Subsets*.

The Advanced Design System supports version 4.0 and 5.0 IGES formats. The program reads and writes IGES CALS Level 1 (technical illustration) and Level 3 (electrical/electronic applications) files. Level 2 (engineering drawings) is not fully supported since Level 2 is used primarily for drafting applications.

Even with the CALS standards, CAD systems accept very different IGES formats. Although it is impossible to accommodate every format, you can use a number of options to configure various IGES translators, including translators supplied by Autodesk, Mentor Graphics, and Cadence. For details, see *IGES Translator* (trans).

Mask File

 Mask file format is a simple flat (non-hierarchical) geometric description. The format facilitates the transfer of simple geometric data for final mask processing. Only geometric forms are described in a mask file; simulation data, element parameters, substrate definitions, and hierarchy are not included.

The Gerber and DXF translators use the mask file format as an intermediate file when

converting data to Gerber and DXF. For details, see *Mask Files* (trans).

MGC/PCB

 MGC/PCB files are IFF files that are used exclusively for Mentor Graphics design transfers. Although this format is available from the Advanced Design System layout menu only, you can transfer both schematic and layout information.

MGC/PCB files write to a specific location each and every time. When you select this format, the filename and location of the IFF transport is determined automatically. For details, *MGC/PCB* (trans).

ODB++

ODB++ is widely accepted as a practical de-facto standard within the electronics industry as an efficient way to move printed circuit bareboard, assembly and test data on the manufacturing-engineering level within design/manufacturing supply chains. It is designed as a simple yet comprehensive description of all entities needed in the manufacturing of a printed circuit board. Advanced Design System supports importing and exporting to/from ODB++ file. For details, see *ODB++ Translator* (trans)

Netlist Files

Simulation Program with Integrated Circuit Emphasis (SPICE) is a simulation tool used by engineers throughout the world for simulating circuits of all types. Since its development at the University of California Berkeley, SPICE has been commercialized and modified by a large number of vendors and also adopted and modified by electronics companies for their own in-house use. Many designers and companies have large investments in existing subcircuits or device models described by SPICE netlists that they want to use with the Advanced Design System from Agilent Technologies.

 Spectre is an EDA (Electronic Design Automation) tool produced by Cadence Design Systems, Inc. Spectre is used by engineers throughout the world for simulating circuits of all types. Many designers and companies have large investments in existing subcircuits or device models described by Spectre netlists that they want to use with the Advanced Design System (ADS) from Agilent Technologies.

For detailed information on SPICE or Spectre file import, refer to *Netlist Translator for SPICE and Spectre* (netlist).

Importing and Exporting Schematic and Layout Designs

This section provides information on importing and exporting schematic and layout files as well as how to use the Advanced Design System Layer Editor. For information on importing Netlists, refer to *Importing a Netlist File* (netlist).

> **ⓘ Note**
> To prevent performance and memory consumption problems with large designs such as reticles or imported designs for use with Momentum, refer to *Disabling Layout Connectivity Features* (usrguide).

Importing a Schematic

For available file formats that you can import from an ADS Schematic window, refer to *Available Formats* (trans). For information on importing SPICE and Spectre netlist files, refer to the *Netlist Translator for SPICE and Spectre* (netlist).

The procedure for importing each format is generally the same, however the available options differ. For options relating to a particular file format, see the appropriate section.

To import a schematic design:

1. Open a schematic design in a workspace.
2. Click **File** > **Import...** from the schematic window. The *Import* dialog box appears.
3. Select the appropriate file format from the *File Type* drop-down list.
4. Click **More Options** to define options for the imported file. The *More Options* dialog box appears.

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

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

   > **ⓘ Note**
   > For certain file types, the translator uses the existing file name to determine the new design name.

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

Exporting a Schematic

For available file formats that you can export from an ADS Schematic window, refer to *Available Formats* (trans).

The procedure for exporting IFF files is the same as the import, but the available options differ. For options relating to the IFF file format, see *IFF Files* (trans).

To export a schematic design:

1. Open the schematic design and click **File** > **Export**. The *Export* dialog box appears.
2. Select *IFF* file format from the File Type drop-down list.
3. Click **More Options** to set export options. The *More Options* dialog box appears.
4. Change options as needed and click **OK**.
5. To specify a path for the exported file, click **Browse**.
6. Double-click as needed to locate the directory for the exported design. By default, all files are listed that have the file suffix appropriate for the chosen file format.
7. Click **OK**.
8. Type a new filename in the *Export* dialog box, following the path, and click **OK**. The file is written to the specified directory.

Importing a Layout

For available file formats that you can import from an ADS Layout window, refer to *Available Formats and Import/Export Options* (trans).

The procedure for importing each format is the same, but the available options differ. For options relating to a particular file format, refer to the associated section.

To import a layout file:

1. In the ADS Layout window, click **File** > **Import** to open the *Import* dialog box.
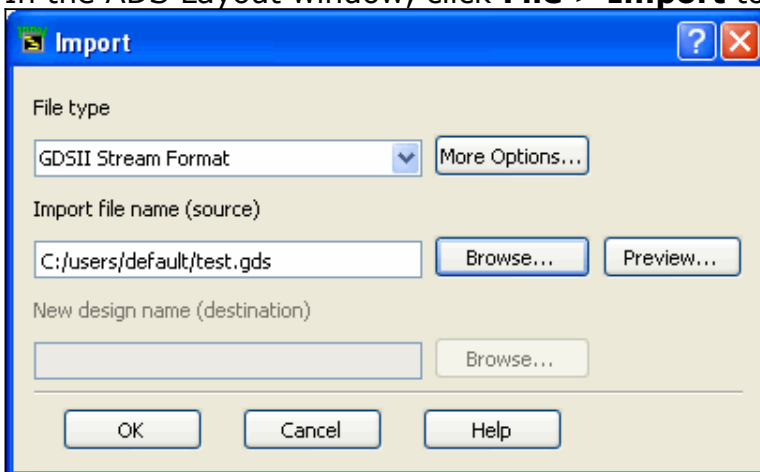

2. In the *File Type* field, click the arrow in the right-hand corner to display a drop-down list of available formats. Select the format you want to import.
3. Click **More Options** to define the options for your selected format.
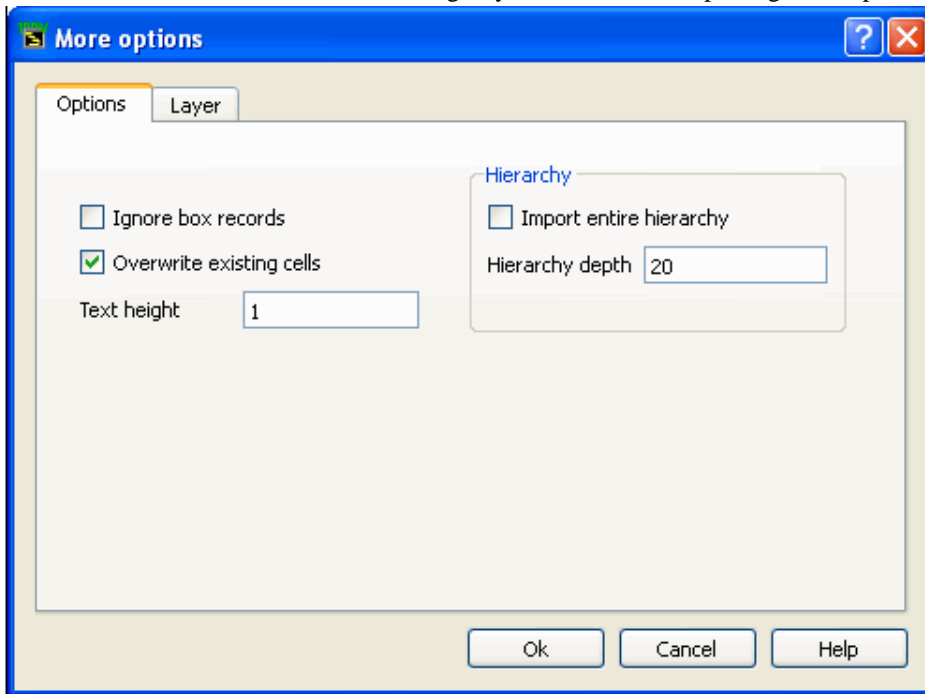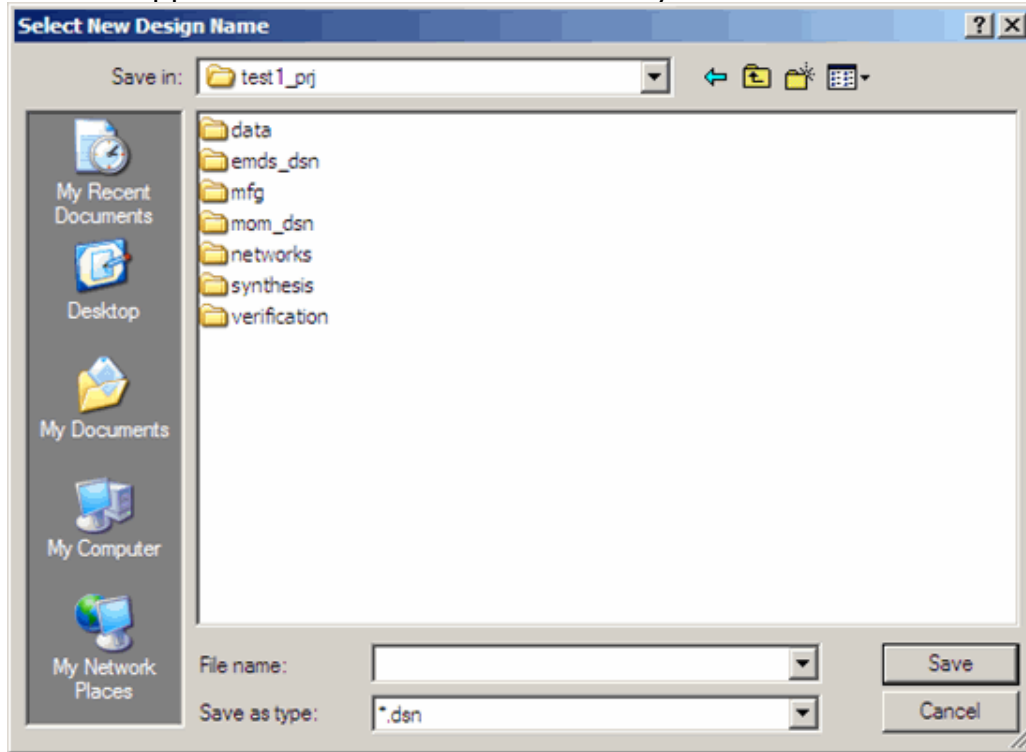
Note
Each format has its own set of import options. Refer to the individual translator topic.

4. In the *More Options* dialog box, click **OK** to save your settings and return to the *Import* dialog box.
5. In the *Import File Name (Source)* field enter the full path of the source file. Alternatively, click **Browse** to open the *Select Source File* dialog box and locate the file you want. After locating the file, click **Open** to accept the selection and return to the *Import* dialog box. The appropriate suffix () is appended to the filename automatically.
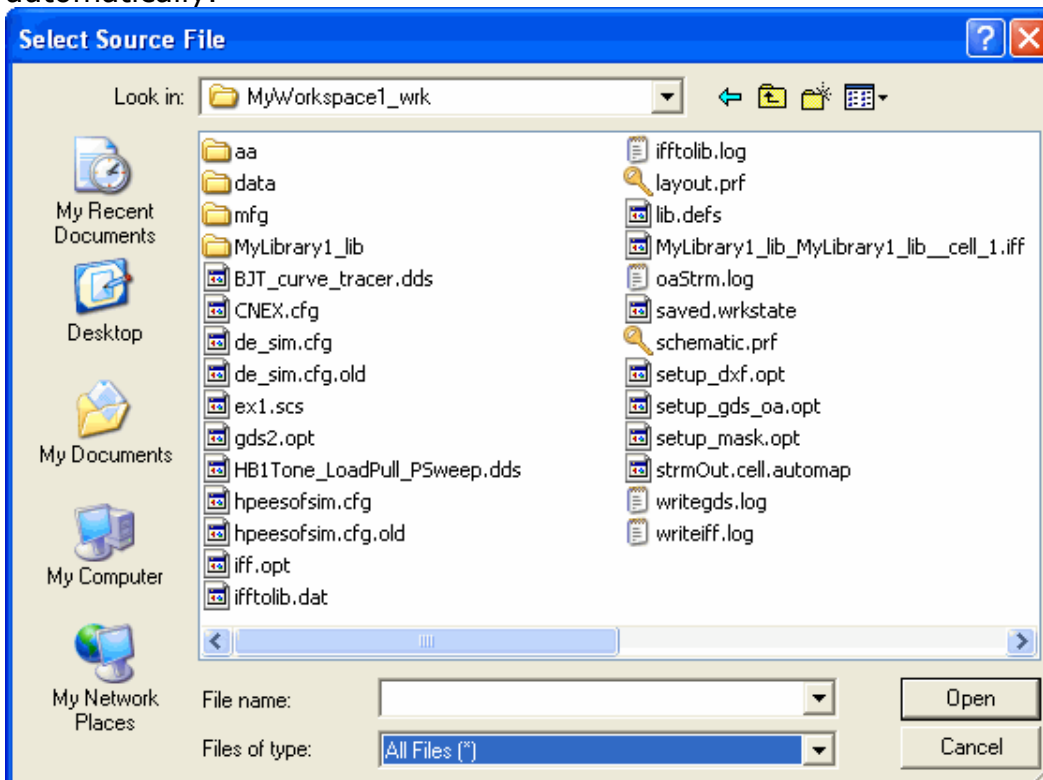
6. In the **New Design Name (Destination)** field, enter the full destination path and a name for the new file. Alternatively, you can click **Browse** to open the *Select New*

Design Name dialog box and locate the destination path. After locating the file, click **Open** to accept the selection and return to the Import dialog box. The appropriate suffix is appended to the filter automatically.

> **ⓘ Note**
> *New Design Name (Destination)* cannot be specified for EGS Archive Format, GDSII Stream Format, IFF, Gerber, DXF, Mask, ODB__ or IGES files.

7. Click **OK** to accept the selections and start the import process. One or more files may be created.

Exporting a Layout

For available file formats that you can export from an ADS Layout window, refer to *Available Formats* (trans).

The procedure for exporting each format is the same, but the available options differ. For options relating to a particular file format, see the appropriate section.
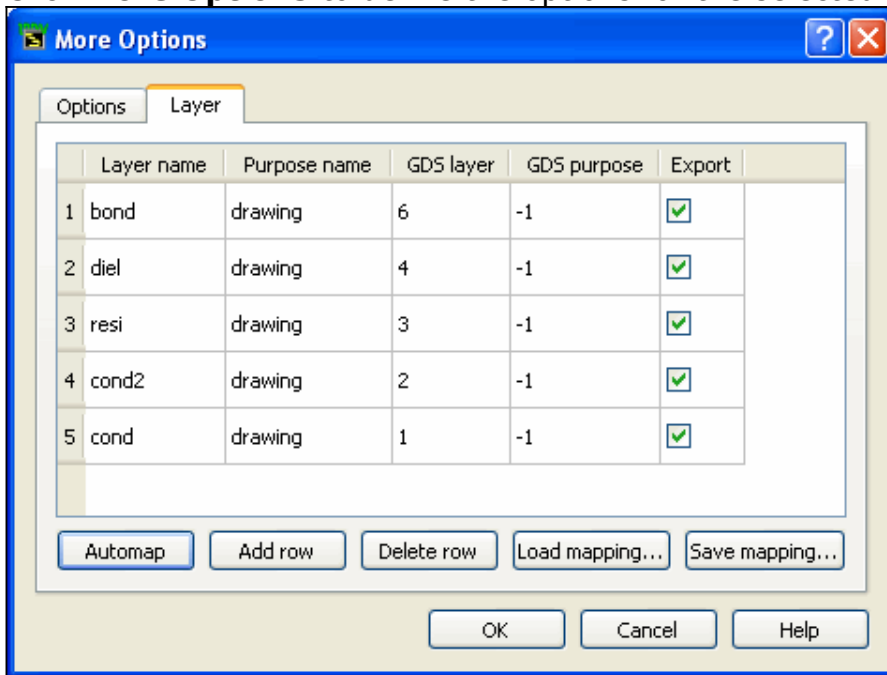
> **ⓘ Note**
> Gerber Viewer appears as an export file option. It is not a file format, but is placed on the export file menu so you can open the Gerber Viewer at any time. In addition, you can access the Viewer during a DXF or Gerber export (see *Using the Gerber Viewer* (trans)).

To export a layout file:

1. In the ADS Layout window, click **File** > **Export**. The *Export* dialog box appears.
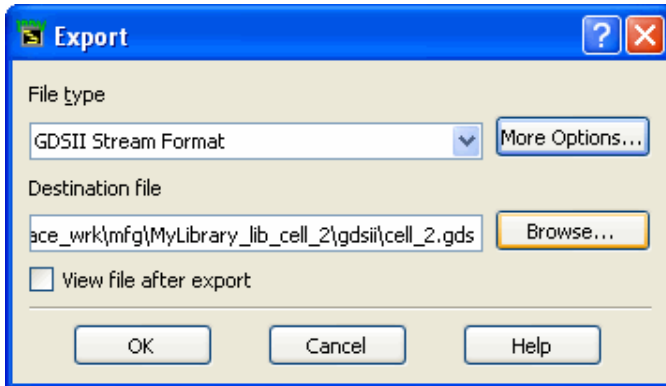
2. In the *File type* drop-down list to open the available formats. Select the format you want to export.
3. Click **More Options** to define the options for the selected format.

> ℹ **Note**
> Each format has its own set of import options. Refer to the appropriate section.

4. In the *More Options* dialog box, click **OK** to save your settings and return to the *Export* dialog box.
5. In the *Destination file*, enter the full path of the destination file. Alternatively, click **Browse** to open the *Select Destination File* dialog box. After locating the destination directory, click **OK** to accept the selection and return to the Export dialog box. Enter a file name. The appropriate suffix is appended to the file name automatically.
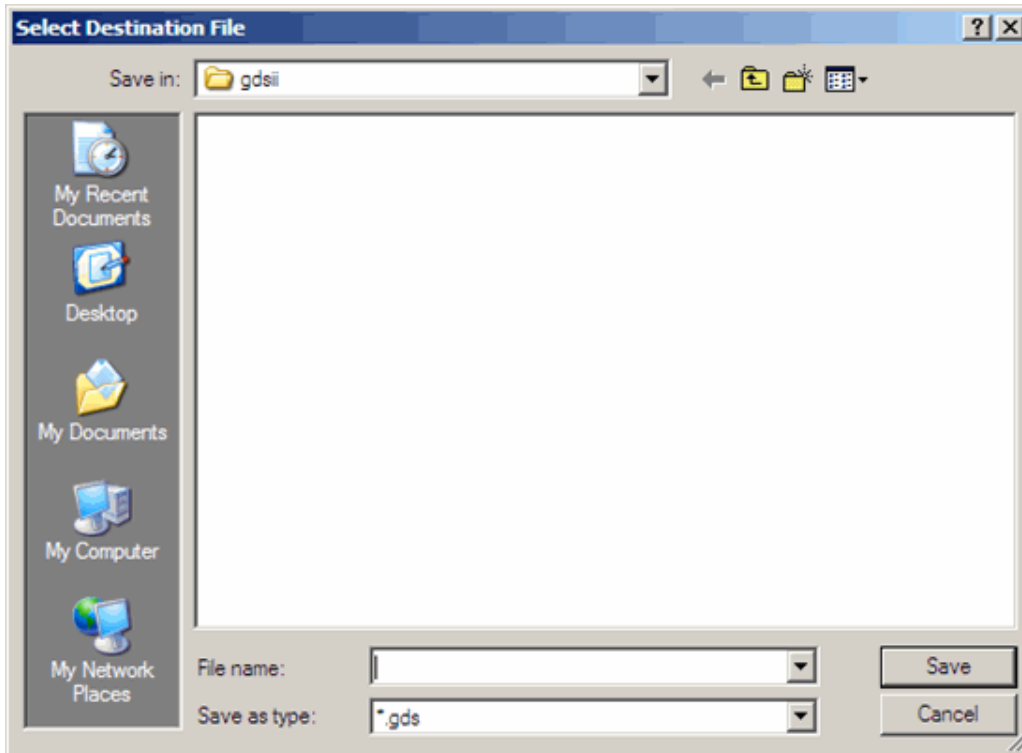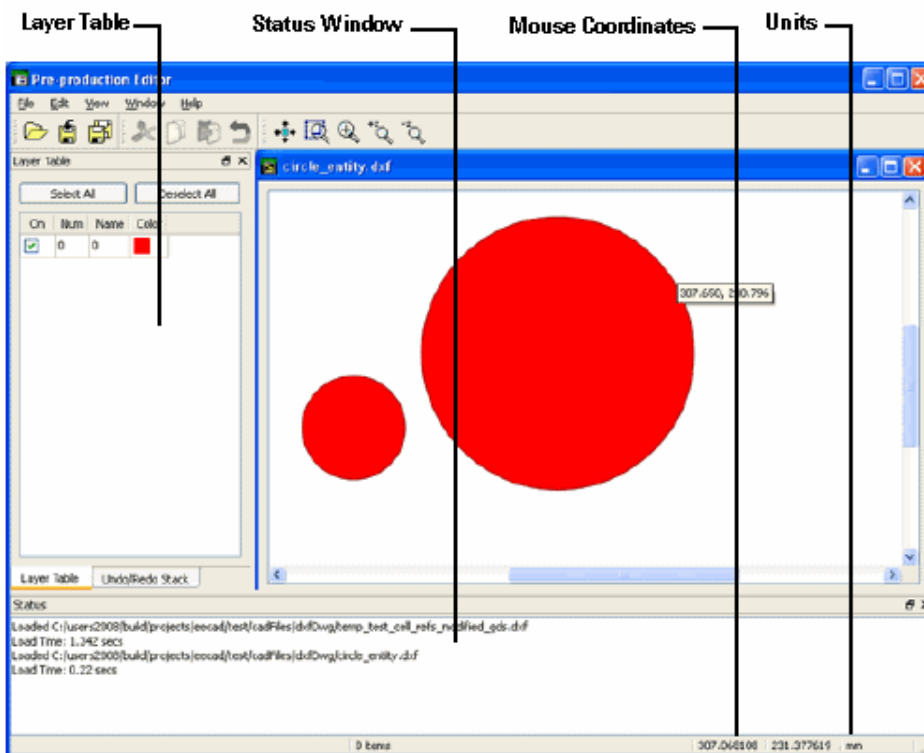
> **ⓘ Note**
> *Destination file* cannot be specified for MGC/PCB files. MGC/PCB files are written to the same predetermined location each and every time. For more information on this file type, see *MGC/PCB Files* (trans).

6. Click **OK** to accept the selections and start the export process. Additional dialog boxes display for the DXF, Gerber, IFF, and MGC/PCB formats. To complete translation of these formats, refer to the appropriate section.

Using the Pre-Production Editor

ADS includes an editor that can display DXF/DWG and Gerber/Drill designs. Using the editor, you can compare the DXF/DWG and Gerber/Drill files to the original ADS layout design to verify that the design was exported correctly. Similarly, DXF/DWG files can be previewed prior to import into ADS:

- to edit a large design down to a smaller design prior to importing it into ADS. This edited design can be saved and then imported in ADS.
- or after import, to verify that the DXF/DWG design data and the ADS-imported design appear similar.

File Menu

The File menu contains common Open, Save, Save As, and Close functions and lists the four most recent files viewed in the editor.

> ℹ️ **Note**
> When files are saved as Gerber/Drill files using the Pre-Production editor, the layer name is used as the file name.

Edit Menu

The Edit menu contains common and specialized editing commands.

Enable Editing

Enables editing commands to be performed on the current design. De-select this item to view the design in read-only mode. In this mode, editing commands are unavailable.

Undo

Undoes the last editing command. A stack of edit commands is created enabling you to choose Undo repeatedly to return to an earlier state of your design. A stack is maintained for each window, thus the Undo command works independently from window to window.

Redo

Returns the file to the pre-undo state.

Cut

Enables you to delete one or more items from one window, and paste in another window.

Copy

Enables you to copy items in a given design window and then paste those items within the same design window or another design window.

Paste

Enables you to paste items that you previously cut or copied. You are prompted to enter the X and Y coordinates of the position where you want to paste the copied items. Select **Apply** to paste or **Cancel** to dismiss pasting.

Delete

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

Select All

Enables you to select all figure(s).

Deselect All

Deselects all the selected figure.

Modify

This choice lets you modify the following items:

- Select figures and click **Union** to perform union of the figures.
- Select figures and click **Intersection** to perform union of the figures.
- Select figures and click **Union Minus Intersection** to perform union of the figures
- Select **Crop** to crop an area from a figure(s).
- Select **Chop** to chop an area from a figure.

> ℹ️ **Note**
> Union, Intersection, Union Minus Intersection, Crop, and Chop do not work for open ended figures with zero width (for example arc and polylines).

View Menu

The view menu contains commands that allow you to alter the view of the current design.

View All

Your design is scaled and repositioned to fit within the viewing area.

The Zoom Commands

The Zoom commands enable you to enlarge or shrink the area being viewed. You can zoom in and zoom out using mouse wheel. The enlarged or condensed figure is moved towards the center of the window.

Click **Zoom To Area** > click a point and then with the mouse button pressed move the mouse to the bottom right of the area and release the mouse button to define a new view.

Click **Zoom to Point** to zoom in on a specified point in the window. Click to specify a point and the current view is magnified by a factor of two, moving the point you specify to the center of the window.

Click **Zoom In By 2** to zoom in by a factor of 2.

Click **Zoom Out By 2** to zoom out by a factor of 2.

Measure
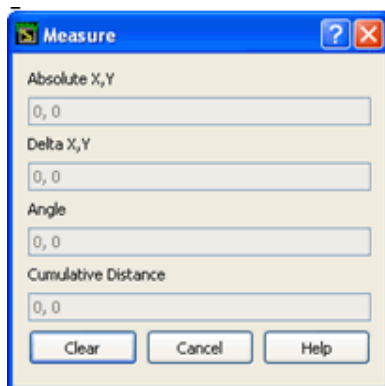

The Measure dialog allows you to measure lengths.

- *Absolute X, Y* displays the absolute coordinates of the point of mouse click.
- *Delta X, Y* gives the relative X and Y coordinates from previous mouse click.
- *Angle* gives the angle with respect to a line horizontal to the X axis.
- *Cumulative distance* gives the total length from starting point to current point.
- Click **Clear** to reset the values for Absolute X,Y, Delta X, Y, Angle and Cumulative distance.
- Click **Cancel** to dismiss the measure dialog box.

Undo/Redo Stack
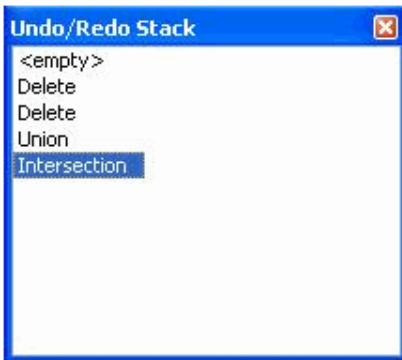
Shows last performed actions. You have an option to undo or redo the last operations performed.

Layer Table
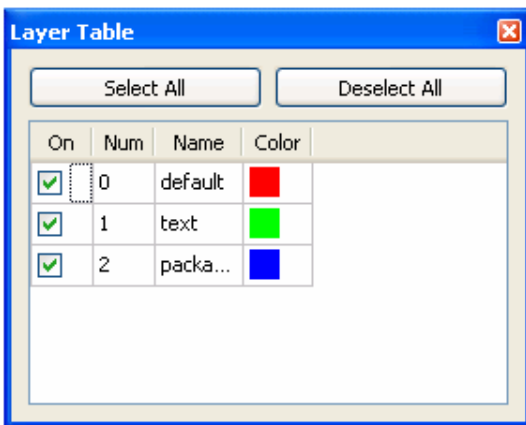
The layer Table displays the layer information. It enables you to switch on/off layers from display. It also enables you to view layer number, layer name and layer color. Click **Select All** to view all the layers and **Deselect All** to switch off all the layers. All the layers are displayed by default.

Status

The status window displays messages about the status of the current design, as well as warning and error messages.

Window Menu

This menu contains the following common Window commands.

Click **Close** to close the active window.
Click **Close All** to close all open designs.
Click **Tile** to arrange all child windows in a tile pattern.
Click **Cascade** to arrange all the child windows in a cascade pattern.
Click **Arrange Icons** to arrange all iconified windows at the bottom of the workspace.
Click **Next** to navigate to next open window.
Click **Previous** to navigate to previous open window.

Viewing Files in the Pre-Production Editor During Export

To view a DXF/DWG or Gerber/Drill file during export from ADS:

1. Select **DXF/DWG** or **Gerber/Drill** from File type drop-down list in *Export* dialog.
2. Select the **View file after export** checkbox and click **OK**.
   This will display the exported DXF/DWG or Gerber/Drill file. A status window shows if the export is complete or failed and the design is displayed in the Pre-Production Editor.

Viewing Files in the Pre-Production Editor During Import

To view a DXF/DWG or Gerber/Drill file during import to ADS:

1. Select **DXF/DWG** or **Gerber/Drill** from File type drop-down list in *Import* dialog.
2. Click **Preview** to view the file in the Pre-Production Editor.

Editing Files in the Pre-Production Editor

Using the Pre-Production Editor you can cut, copy, paste and modify designs. From the *Modify* menu, you can perform editing functions like union, intersection, union minus intersection, crop, and chop.

To edit using the Pre-Production Editor:

1. Open a design in the Pre-Production Editor as described above.
2. Click **Edit** > **Edit Enable**.
   - To cut a figure, select the figure and then click **Edit** > **Cut**.
   - To copy a figure, select the figure and then click **Edit** > **Copy**.
3. To crop or chop a figure:
   1. Select the figure.
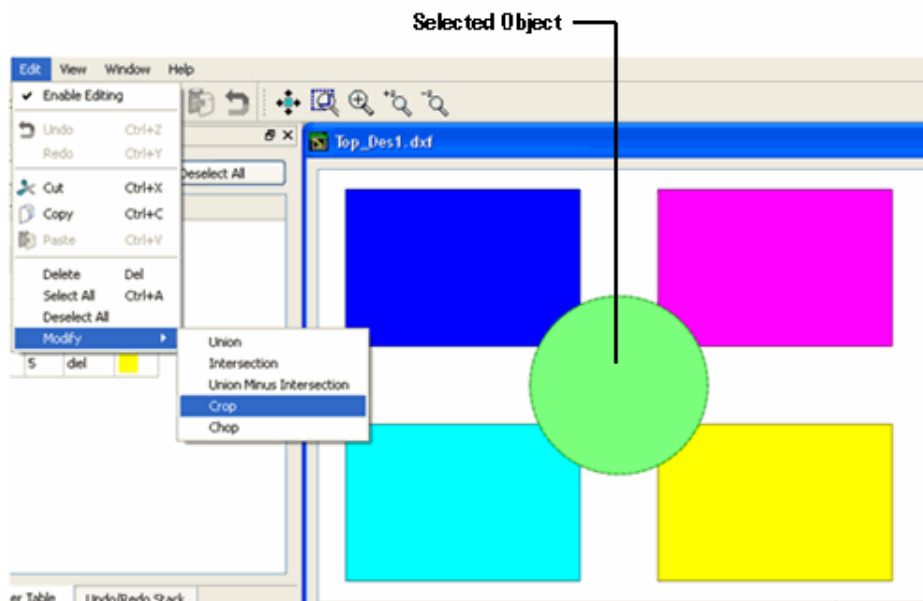
2. Click **Crop** or **Chop**.
3. Mark the area to crop/chop using mouse.
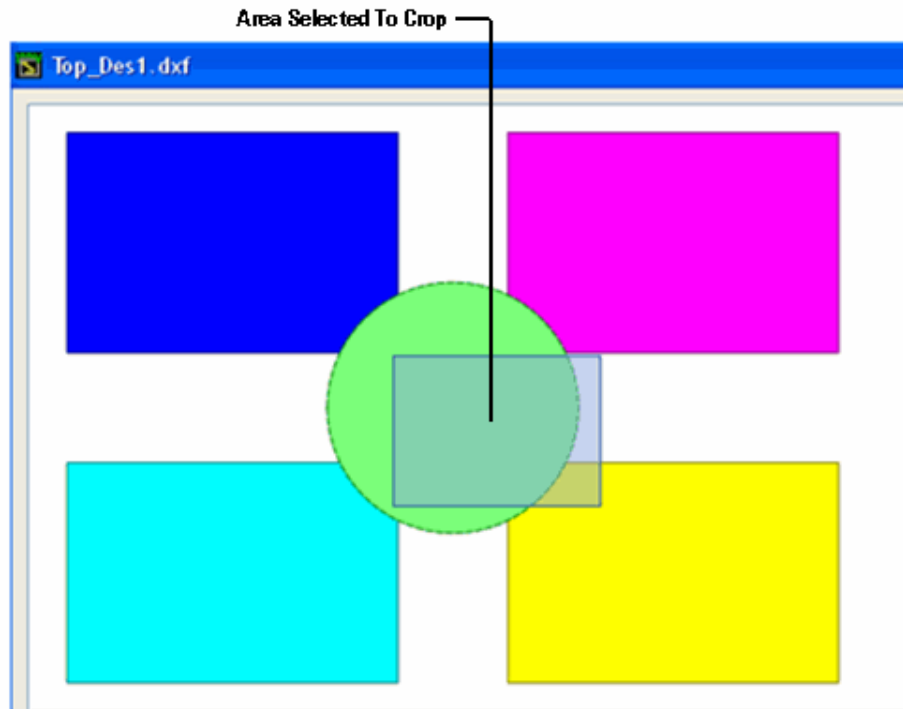
4. Release the mouse to crop/chop the selected area.
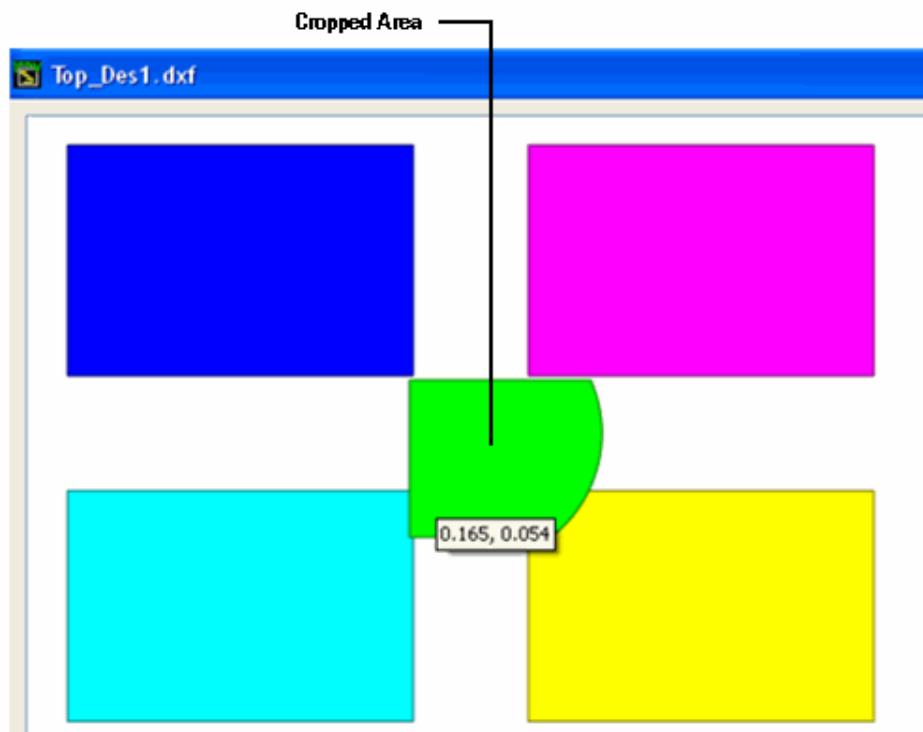
4. Click **Delete** to delete selected item.
5. Click **Undo** to revoke last Edit command.

Cadence/PCB Translator

Cadence/PCB files are IFF files that are used to transfer schematic and layout data simultaneously to Cadence Concept and Allegro PCB editors. This format is available from the Advanced Design System layout menu only, yet it enables the transfer of both schematic and layout information.

When you select the Cadence/PCB export format from an Advanced Design System Layout window, both a layout and schematic IFF file are exported in a single step. The design data is exported into a standard directory tree contained in the program's workspace directory structure. The standard directory is called *to_cadence*.

The exported files are placed within this directory in a subdirectory that is named the same as the design being exported. This subdirectory contains an information file and the translated schematic and layout IFF files. Thus, if you were translating a design called *test* , the exported files (*design_info*, *schematic.iff* and *layout .iff*) would found in a directory called *to_cadence/test.hpxfer*.

For information on how to import the files in the Cadence tools, consult the Cadence Allegro PCB Editor user guide.
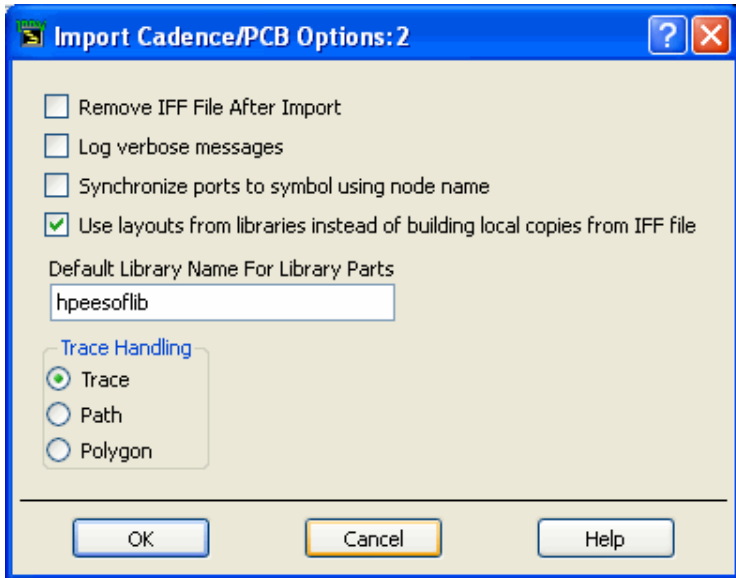
Importing Cadence/PCB Files

This section outlines the procedure for translating designs from Cadence/PCB format. For more information on transferring designs between the Advanced Design System and Cadence Analog Design Environment, contact your Agilent Technologies sales representative.

To import an Cadence/PCB file follow the steps as outlined in *Importing a Layout* (trans). For available options (accessed via *More Options* in the Import dialog box), see Import Cadence/PCB Options.

Import Cadence/PCB Options

This section describes the definable options available for importing files in the Cadence/PCB format. To view the options, click **File > Import > Cadence/PCB > More Options**.

The *Import Cadence/PCB Options* dialog enables you to specify configuration an trace handling to control the import of Cadence/PCB files.

Remove IFF File After Import

When this option is selected, the *.iff* file is removed once the file is successfully imported. This option is not selected as the default.

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 from the Cadence environment. To maintain proper connectivity in the ADS environment, this option must be enabled if connectors share a common name in the Cadence environment.

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

When this option is selected, library elements that already exist in the system libraries are *not* recreated for the imported file. Instead, these elements are read from the local libraries; if an element does not exist in a local library, then it is newly created. This option is deselected as the default, and all elements are created/recreated in the local workspace.

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.

> **ⓘ 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.

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-EDA's Solution Services.

Trace Handling

Trace Handling enables you to select how you want your *meander elements* interpreted during a translation.

Trace When this option is selected, *meander elements* are translated as simulatable traces with pins. This is the default setting.
Path When this option is selected, *meander elements* are translated as primitive data type with center line and width. The default for this option is deselected.
Polygon When this option is selected, *meander elements* are translated as polygons. The default for this option is deselected.

> **ⓘ Note**
> If you choose *Use layouts from libraries instead of building local copies from IFF file* in the Import IFF Options dialog box, choose *Path* in the *Trace Handling* options. Fixed artwork is not simulatable, therefore it is not necessary for the interconnects to be simulatable.

Exporting Cadence/PCB Files

This section outlines the procedure for translating designs into Cadence/PCB format. For more information on transferring designs between the Advanced Design System and the Mentor Graphics Falcon Framework, contact your Agilent Technologies sales representative.

To export an Cadence/PCB file:

1. Follow the steps as outlined in *Exporting a Layout* (trans). For available options (accessed via *More Options* in the Export dialog box), see Export Cadence/PCB Options.
2. If the option *Prompt For User Message* was selected, an Export Cadence/PCB Options message dialog appears:
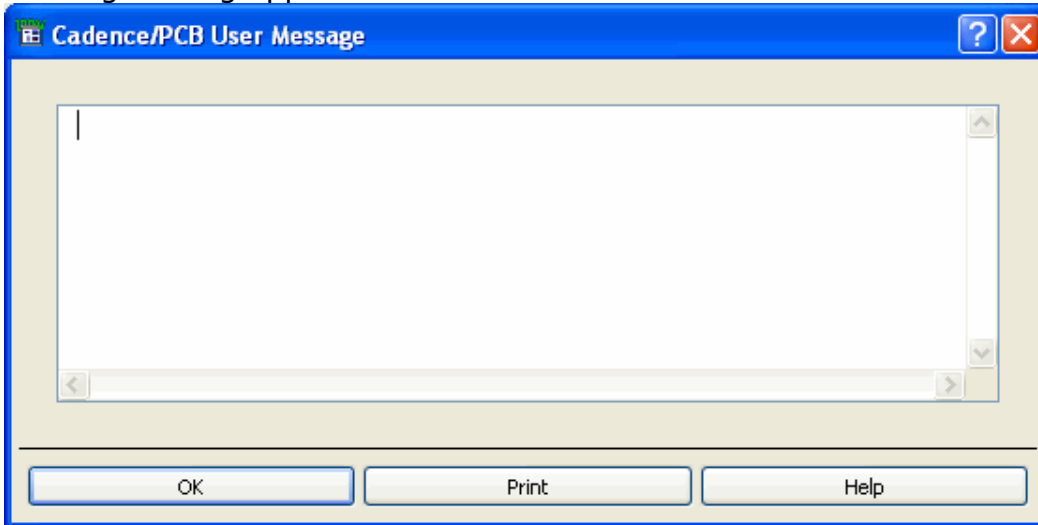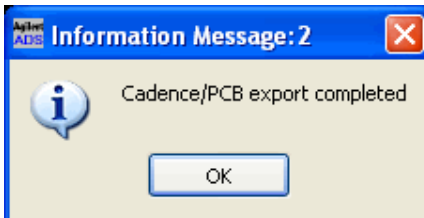

    Enter any messages to be included with the design transfer. This information is used only by the *import_hpeesof* utility (see *Cadence/PCB Files*) and is not kept with the design.
    If you do not wish to include a message in the file, you may leave this window empty. If you want to print the message, click **Print**.
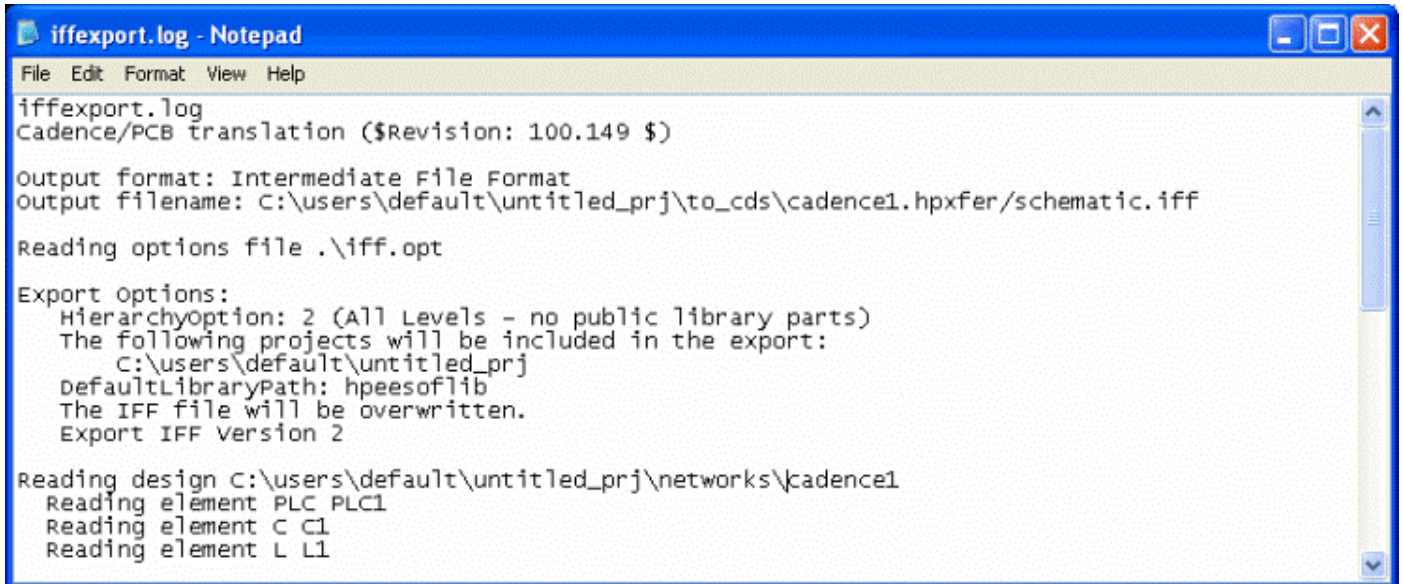   To proceed with the transfer, click **OK**.
3. When the translation is completed, the following message window appears:

    Click **OK** to dismiss this window.
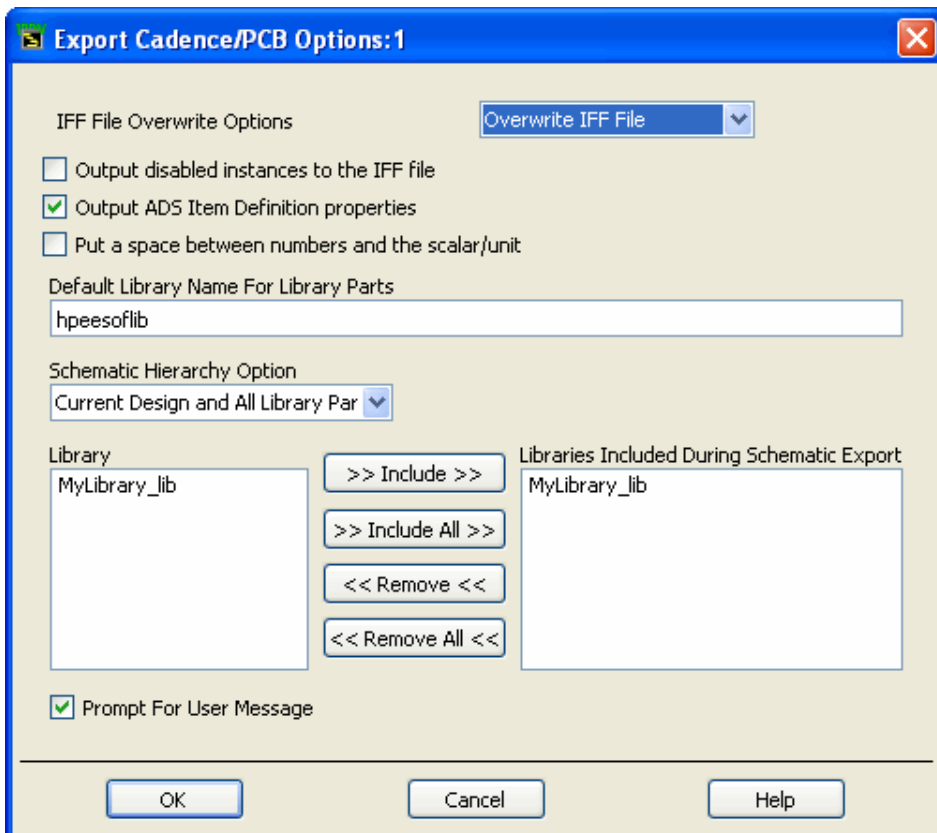4. The IFF Export log appears:

Review the log, searching for any warnings or error messages generated during export.

The log file appears in the *hpeesofeedit* window by default. This window is provided as a means of viewing the file and is not intended for editing.

5. To dismiss the log window, choose **File** > **Quit**.

Export Cadence/PCB Options

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.
- *Append to IFF File*
  When writing to an existing file, the new data is appended to the existing file.

Default Library Name For Library Parts

The name of the library to which the library parts are written. Design objectives are stored in a group that uses the same name as the workspace directory, but library parts are stored in either the default library *hpeesoflib* or a library that you specify.

> ⓘ **Note** The default library name can contain only alphabetic and numeric characters.

Output disabled instances to the IFF file

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

Output ADS Item Definition properties

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

Put a space between numbers and the scalar/unit

When this option is selected, parameter values are exported as they normally appear in ADS (i.e. with a space between the number and the scalar, e.g. "1 pF"). If the checkbox is deactivated, the exporter converts the values into the IFF value specification, which is to have no space between a number and a scalar (e.g. "1pF"). Ideally, an IFF exporter should interpret either form of number, and set the value internally to whatever is normal for that environment. Some environments do not interpret the IFF property values in any way and can exhibit problems with the values when they are imported.

Schematic Hierarchy Option

This establishes 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 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 library. If hierarchical, all included libraries are listed in the appropriate order.

Libraries Included During Schematic Export

The libraries for which schematic design information is exported. You may customize this list if the current library is hierarchical. (Note that complete layout hierarchy is always exported.)
To add a library to this list:

In the Library list, click the desired library.

1. Click **Include**. The library is added to the Libraries Included list.
2. 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

When selected, a user message window appears before the transfer is initiated. This window enables you to enter any messages that you want included in the translated file. For more information about the user message, see Export Cadence/PCB Options.

This option is selected as the default.

DXF/DWG Translator

The DXF/DWG translator enables you to convert ADS designs into AutoCAD's DXF/DWG file format, as well as convert DXF/DWG files into ADS designs. DXF is a very simple file format that can be read by most CAD programs that support DXF. The DXF/DWG translator is bidirectional and provides support for hierarchical and all layer separation.

DWG is a binary format, therefore the files take less time to load and save. ADS supports importing and exporting to/from AutoCAD. AutoCAD versions 12 to 2007 are supported for import. AutoCad versions 2000, 2004 and 2007 are supported for export. Carefully consider how you want to use DXF/DWG output - including layer numbering, use of holes, and polygon shapes - before beginning your layout design. Setting up the proper layout rules in ADS can save a lot of time in generating acceptable DXF/DWG output. For specific considerations or limitations, consult your AutoCAD documentation.

> **ⓘ Note**
> Password protected DWG files are not supported.

ADS includes an editor that can display the DXF/DWG design. Using the editor, you can compare the DXF/DWG files to the original ADS layout design to verify that the design was exported correctly. Similarly, DXF/DWG files can be previewed prior to import into ADS:

- to edit a large design down to a smaller design prior to importing it into ADS. This edited design can be saved and then imported in ADS.
- or after import, to verify that the DXF/DWG design data and the ADS-imported design appear similar.

For details, see *Using the Pre-Production Editor* (trans).

Importing DXF/DWG Files to ADS

The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to *Importing a Layout* (trans). For specific import options related to importing DXF/DWG files, refer to *More Options*. For information on mapping DXF/DWG entities to ADS layout components, refer to Mapping DXF to ADS.

This section describes the definable options available for importing files in the DXF/DWG format. To view the options:

1. Click **File > Import** from the *Layout* window.
2. Select **DXF/DWG** from the *File type* list box.
3. Enter the *Import file name* and *Layers file name* in the appropriate fields.
4. Click **More Options...** to open the *More Options* dialog box.

The *More Options* dialog box enables you to specify flattening and layers to control the import of DXF/DWG format files. A DXF/DWG file and layers file must be selected in the main export dialog for the *More Options* dialog box to open.

DXF Options
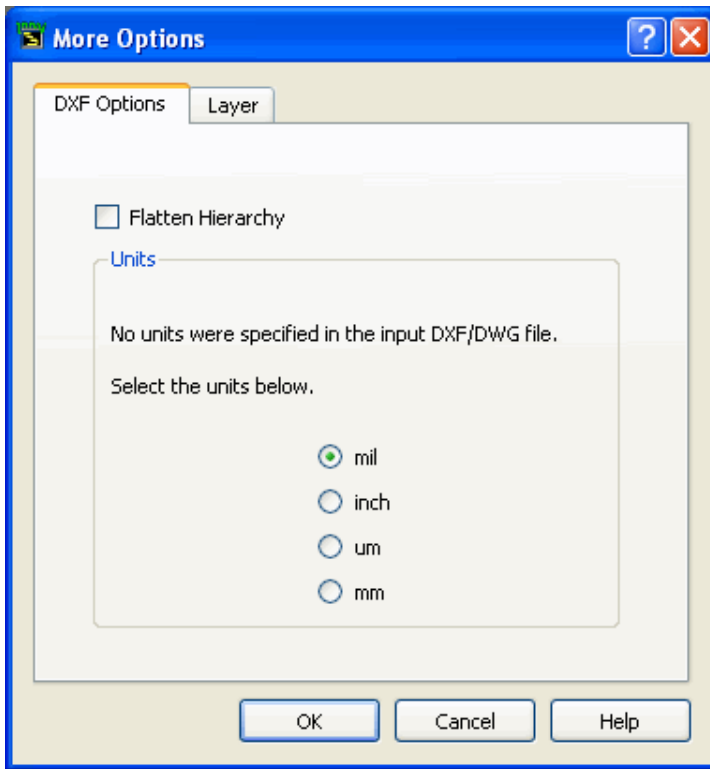
Flatten Hierarchy

Select **Flatten Hierarchy** to remove any hierarchy that may be present in the DXF/DWG selected. Default setting retains the hierarchy.

> **ⓘ Note**
> If you select a DXF/DWG file that contains more than 50 component designs this option gets automatically enabled in order to avoid long import times.

Units

Older versions of the DXF file format may not specify units. If no Units were specified in the input DXF file, this option is enabled. Default setting is *mil*.

If the unit mentioned in the import file is different form layout unit then a *warning message* (trans) is displayed.

Layer Options
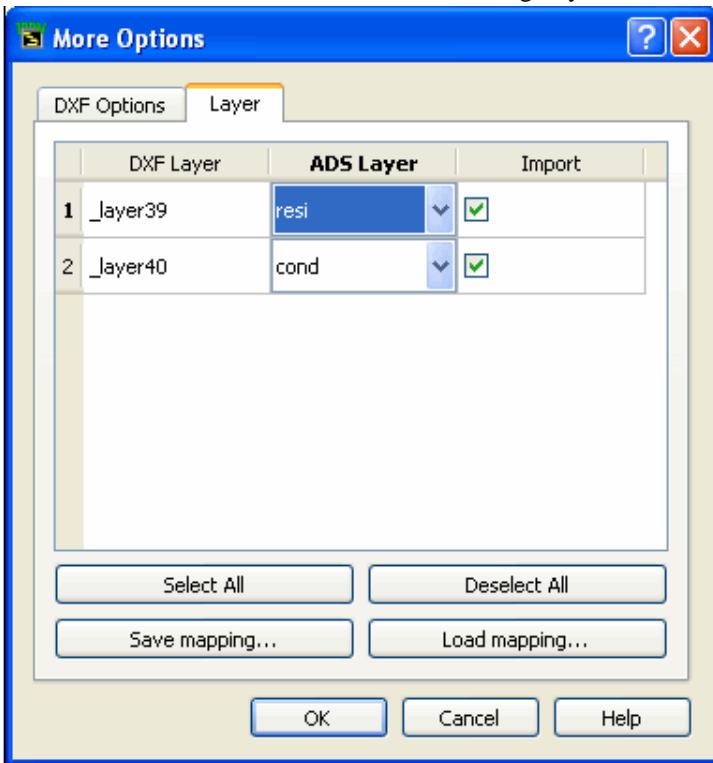
The *Layer* tab displays a list all the layers in DXF/DWG file. Layers can be selectively imported to ADS using the import column. By default all layers are imported. Using this tab user can map DXF/DWG layers to ADS layers during import. By default DXF/DWG layers are mapped to ADS layers with same name. If no match is found the mapped layer is *Mismatch*. During import if a *Mismatch* mapping is encountered a new layer with same name as DXF/DWG layer name is created in current workspace technology in ADS.

User can save layer mapping in .map file using **Save Mapping** button.
During import user can load a previously saved layer mapping file (.map) using **Load Mapping** button.

> ⓘ **Note**
> Set the *Arc/Circle resolution (degrees)* value before importing the DXF/DWG file. From the *Layout* window, click **Options > Preferences**. From the **Preferences** dialog box, click **Entry/Edit** tab and change the *Arc/Circle resolution (degrees)* field.

Mapping DXF to ADS

The following table represents the mapping of DXF/DWG file shapes to ADS file shapes.

Mapping DXF File Shapes to ADS File Shapes

DXF/DWG	ADS Layout Equivalent
Line with width = 0	Polyline
Line with width > 0	Path
Polygon with width = 0	Polygon
Polygon with width > 0	Polygon with cutline
Rectangle with width = 0	Rectangle
Rectangle with width > 0	Polygon with cutline
Circle with width = 0	Circle
Circle with width > 0	Polygon with cutline
Arc with width = 0	Arc
Arc with width > 0	Polygon

Exporting an ADS Design to DXF/DWG Files

To export DXF/DWG files:

1. Follow the steps as outlined in *Exporting a Layout* (trans). For available options (accessed via More Options in the *Export* dialog box), see Export DXF/DWG Options.
2. After clicking **OK** in the *Export* dialog box, the translator converts your ADS Layout to DXF format.

Once in DXF/DWG file format, the file is available for use in AutoCAD or other Mechanical CAD systems.

Export DXF/DWG Options

The *More Options* dialog box enables you to set file format, Autocad version, path geometry, and hole format.
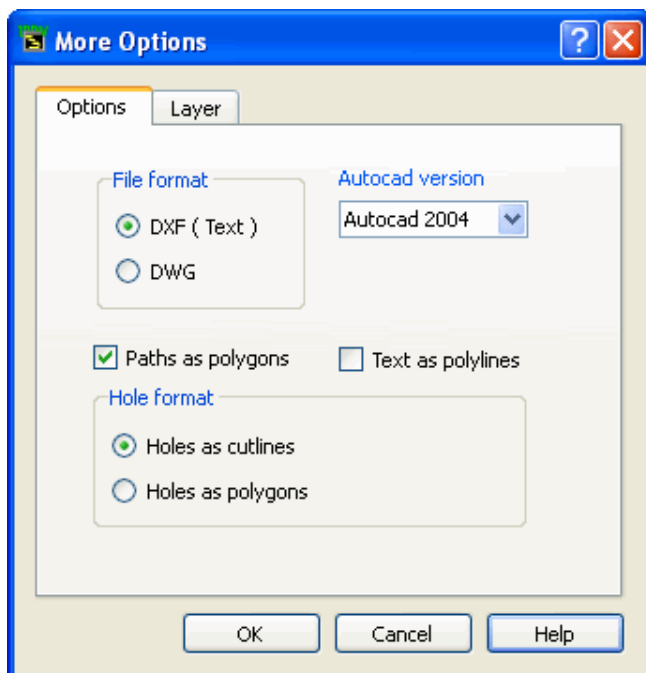
Options

The *Options* tab enables you to select various design attributes and AutoCad File Formats in the translation.

File format

DXF (Text)

Select this option to export the ADS design to DXF format as an ASCII text file. This selection is the default settling.

DWG

Select this option to export your ADS design to a binary DWG file format. Exporting to DWG file format is faster than the DXF (Text) format. Also file size of the DWG file will be smaller as compared to corresponding DXF (Text) file.

Autocad version

Use this drop-down list to select the AutoCad version. ADS layout can be exported in three AutoCad versions: 2000, 2004, and 2007.

Paths as polygons

Select **Paths As Polygons** to export the design paths or traces as polygons. Paths or traces have mitered or curved corners that need to be preserved in the translation. The ADS layout has paths with endpoint types other than flush that need to be preserved in the program database. This option is selected as the default.

Text as polylines

Select **Text as polylines** option to convert all layout text to set of polylines. The fonts are maintained as in layout.

Hole format

The Hole format section enables you to define how the translator deals with holes in a design.

Holes as polygons

Select **Holes As polygons** to convert holes into polygons. One polygon will be created for each hole in the same layer.

> **ⓘ Note**
> Some systems may not be able to tolerate complex polygons with cutlines. For these systems, select **Holes as polygons**. This option is deselected as the default.

Holes as cutlines

Select **Holes as cutlines** to convert holes into cutlines. This option is selected as the

default.

Layers

The Export column enables you to deselect one or more layers to exclude it from the output file. Default setting is all layers selected and exported.
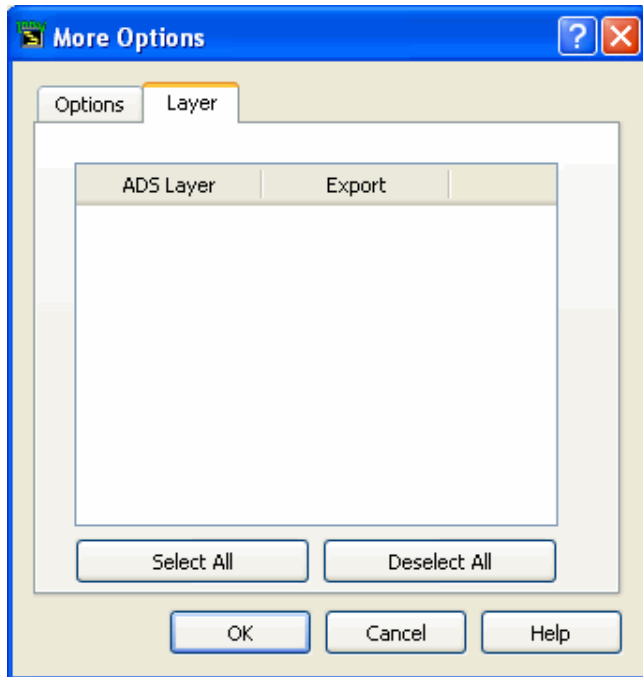
Mapping ADS to DXF (hierarchical)

The DXF file created by the DXF/DWG translator is as simple as possible so that many DXF parsers, even primitive ones, can read the file. The DXF translator performs the following mapping between ADS Entity and DXF files.

Hierarchical DXF Output

ADS Entity	DXF Entity
Polygon	closed LWPOLYLINE, solid HATCH
polyline	open LWPOLYLINE
path	closed LWPOLYLINE, solid HATCH
wire	open LWPOLYLINE
circle	circle
text	text
rectangle	closed LWPOLYLINE, solid HATCH
arc	arc

EGS Archive Files

The EGS Archive format is an ASCII text format that represents multiple hierarchical designs. EGS Archive files include information such as drawing shapes, layout units, database precision, grid spacing and, optionally, a layers definition including layer colors, numbers, and names. They do not contain information about the connectivity of a layout design.

Originating from a former *Hewlett-Packard* product called the Engineering Graphics System (EGS), the EGS Archive format is useful as an intermediate file for transferring input/output to various third-party translators. It is also the best format for translating graphic shapes into the Series IV and MDS products. In addition, the EGS Archive format is easy to parse and read, so if you have created your own software tools, this is a very useful format for transferring graphics shapes into these tools.

When importing an EGS Archive file into Advanced Design System, you may specify a Layers File Nam e for the creation of a layers definition file (the *Layers File Name* field is located in the Import dialog box). This file will be automatically created and will contain the layer information from the EGS Archive file, as well as layers automatically added as necessary during import. All designs created during import reference this layers file.

Oval or hatch shapes of the EGS Archive format are not supported in Advanced Design System.

Importing EGS Archive Files

The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to *Importing a Layout* (trans). For specific import options related to importing EGS Archive Files, refer to Import EGS Archive Options. For information on mapping EGS Archive files to ADS, refer to Mapping EGS Archive to ADS.

Import EGS Archive Options

There are no definable options available for importing files in the EGS Archive format.

Mapping EGS Archive to ADS

The table below shows the mapping of EGS Archive file shapes to Advanced Design System layout shapes.

EGS Archive Object	Advanced Design System Layout Equivalent
Line with width = 0	Polyline
Line with width > 0	Path
Polygon with width = 0	Polygon
Polygon with width > 0	Polygon with cutline on longest segment
Rectangle with width = 0	Rectangle
Rectangle with width > 0	Polygon with cutline on longest segment
Circle with width = 0	Circle
Circle with width > 0	Polygon with cutline to the right
Arc with width = 0	Polyline
Arc with width > 0	Polygon

Exporting EGS Archive Files

The procedure for exporting each format is generally the same, however the available options differ. For a step-by-step tutorial on exporting a layout file, refer to *Exporting a Layout* (trans). For specific export options related to exporting EGS Archive Files, refer to Export EGS Archive Options. For information on mapping ADS to EGS Archive files, refer to Mapping ADS to EGS Archive.

Export EGS Archive Options

To explore Export EGS Archive Options, in *Layout* window, click **File > Export > EGS Archive Format > More Options**.

Include Mask Layers

When this option is selected, a list of mask layers is included with the translated design. This option is selected as the default.

Paths As Polygons

When *Paths As Polygons* is selected, the design paths or traces are exported as polygons. This should be selected for the following conditions:

- Paths or traces have mitered or curved corners that need to be preserved in the translation.
- T he EGS Archive file has paths with endpoint types other than *flush* that need to be preserved in the program database.

This option is deselected as the default.

Holes As Polygons

When this is selected, holes are converted to polygons. When *Holes As Polygons* is *not* selected (this is the default), polygons with holes are translated as single-segment polygons, the *false edge* segment becoming part of the polygon. Some systems may not be able to tolerate this type of complex polygon. For these systems, select *Holes As Polygons* .
This option is deselected as the default.

Flatten Components

When *Flatten Components* is selected, all instances whose artwork type is defined with an AEL macro will be flattened when exported. For example, if a design contains an MLIN it will be flattened. This option is deselected as the default. It only affects the current design and will not change the hierarchy.

All Filled

When this is selected, all data is transferred as filled. This option is deselected as the default.

Flatten All

When *Flatten All* is selected, all levels of hierarchy are automatically removed and a single flat design is exported. There are no references from the top-level structure to any other structure in the design. This option is useful when your post-processor does not support or correctly translate hierarchy in EGS Archive files. Please note: if a substructure was instanced more than once, selecting this option can substantially increase the size of the file.
This option is deselected as the default.

Auto Merge

When *Auto Merge* is selected, all shapes for every mask layer that intersect or overlap are merged. This option is deselected as the default.

Layers To Exclude

Layers can be excluded from the EGS Archive export by entering the layer IDs. Multiple IDs can be entered using space or comma delimiter.

Mapping ADS to EGS Archive

The table below represents the mapping Advanced Design System layout shapes to EGS Archive file shapes.

Advanced Design System Layout Object	EGS Archive Equivalent
Polyline	Line
Path with PathsAsPolygons	Polygon
Path without PathsAsPolygons	Line
Circle	Circle
Rectangle	Rectangle
Polygon	Polygon
Wire	Line

EGS Generate Files

The EGS Generate format is an ASCII text format representing a single, flat design. It is a subset of the EGS Archive format that contains only the shape information. If an hierarchical design is exported, it is automatically flattened during the export process.

Originating from a former *Hewlett-Packard* product called the Engineering Graphics System (EGS), the EGS Generate format is useful as an intermediate file for transferring input/output to various third-party translators. Also, the EGS Archive format is easy to parse and read, so if you have created your own software tools, this is a very useful format for transferring graphics shapes into these tools.

When importing an EGS Generate file into the Advanced Design System, you may specify a Layers File Nam e for the creation of a layers definition file (the *Layers File Name* field is located in the Import dialog box). This file will be automatically created and will contain the layer information from the EGS Generate file, as well as layers automatically added as necessary during import. All designs created during import reference this layers file.

Oval or hatch shapes of the EGS Generate format are not supported in the Advanced Design System.

Importing EGS Generate Files

The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to *Importing a Layout* (trans). For specific import options related to importing EGS Generate Files, refer to Import EGS Generate Options. For information on mapping from EGS Generate files to an ADS layout, refer to Mapping EGS Generate to ADS.

Import EGS Generate Options

There are no definable options available for importing files in the EGS Generate format.

Mapping EGS Generate to ADS

The table below represents the layout mapping from EGS Generate format to Advanced Design System layout.

EGS Generate Object	Advanced Design System Layout Equivalent
Line with width = 0	Polyline
Line with width > 0	Path
Polygon with width = 0	Polygon
Polygon with width > 0	Polygon with cutline on longest segment
Rectangle with width = 0	Rectangle
Rectangle with width > 0	Polygon with cutline on longest segment
Circle with width = 0	Circle
Circle with width > 0	Polygon with cutline to the right
Arc with width = 0	Polyline
Arc with width > 0	Polygon

Exporting EGS Generate Files

The procedure for exporting each format is generally the same, however the available options differ. For a step-by-step tutorial on exporting a layout file, refer to *Exporting a Layout* (trans). For specific export options related to exporting EGS Generate Files, refer to Export EGS Generate Options. For information on mapping ADS to EGS Generate files, refer to Mapping ADS to EGS Generate.

Export EGS Generate Options

To explore Export EGS Generate Options, click **File** > **Export** > **EGS Generate Format** > **More Options**.

Include Mask Layers

When this option is selected, a list of mask layers is included with the translated design. This option is selected as the default.

Paths As Polygons

When *Paths As Polygons* is selected, the design paths or traces are exported as polygons. This should be selected for the following conditions:

- Paths or traces   have mitered or curved corners that need to be preserved in the translation.
- The EGS Generate file has paths with endpoint types other than *flush* that need to be preserved in the program database.

This option is deselected as the default.

Holes As Polygons

When this is selected, holes are converted to polygons. When *Holes As Polygons* is *not* selected (this is the default), polygons with holes are translated as single-segment polygons, the *false edge* segment becoming part of the polygon. Some systems may not be able to tolerate this type of complex polygon. For these systems, select *Holes As Polygons*.
This option is deselected as the default.

All Filled

When this is selected, all data is transferred as filled. This option is deselected as the default.

Auto Merge

When *Auto Merge* is selected, all shapes for every mask layer that intersect or overlap are merged. This option is deselected as the default.

Layers To Exclude

Layers can be excluded from the EGS Archive export by entering the layer IDs. Multiple IDs can be entered using space or comma delimiter.

Mapping ADS to EGS Generate

The table below represents the layout mapping from Advanced Design System layout to EGS Generate format.

Advanced Design System Layout Object	EGS Generate Equivalent
Polyline	Line
Path with PathsAsPolygons	Polygon
Path without PathsAsPolygons	Line
Circle	Circle
Rectangle	Rectangle
Polygon	Polygon
Wire	Line

Advanced Design System Layout Object	EGS Generate Equivalent

Gerber Translator

The *Gerber Artwork Translator* translates artwork directly from circuit layouts created with Advanced Design System into Gerber format. It exports ADS layouts into ASCII files that control Gerber photo-plotting equipment.

ADS includes an editor that can display the Gerber/Drill design. Using the editor, you can compare the Gerber/Drill files to the original ADS layout design to verify that the design was exported correctly.

For details, see *Using the Pre-Production Editor* (trans).
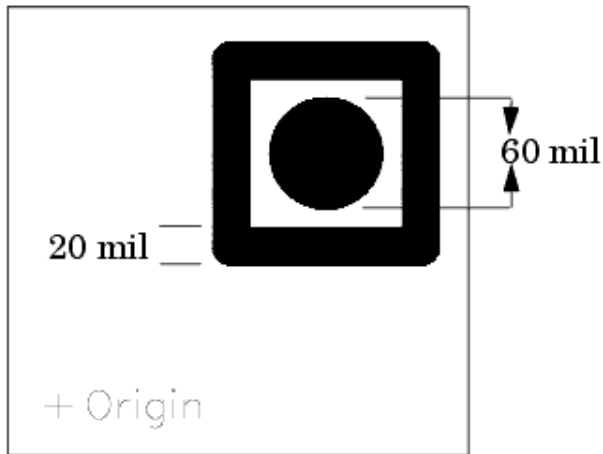
Gerber Command Format

The Gerber format is a numerical control language developed to generate photo artwork. The output of this translator is an ASCII file that contains the following Gerber commands:

    G01 = linear interpolation
    G54 = aperture select
    D01 = shutter open
    D02 = shutter close
    D03 = flash
    M02 = end of program
    X and Y = coordinates
    * = end of block

Coordinates are absolute, with implied decimal point and optional leading zero suppression. However, the literal string values may be modified in the message file. For example, *G01* may be changed to *AB*. A sample listing of the Gerber file commands, with interpretation to the right, might look like this:

```
G54D16*             <- select aperture 16 (20 mil circle trace)
G01X90Y85D02*       <- move the shutter to 90,85
Y185D01*            <- open the shutter and draw to 90,185
X190*               <- while shutter remains open, draw to 190,185
Y85*                <- draw to 190,85
X90*                <- draw to 90,85
G54D45*             <- select aperture 45 (60 mil circle flash)
X140Y135D02*        <- close shutter and move to 140,135
D03*                <- flash expose
X0Y0D02*M02*         <- return to the origin and stop
```

This example would produce the following:

The output file (Gerber command file) is the Gerber Drawing File (*.gbr*).

Importing Gerber/Drill Files

The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to *Importing a Layout* (trans). For specific import options related to importing Gerber/Drill files, refer to *More Options*. While importing Gerber/Drill files multiple files can be selected at a time. The drill files are imported in seperate drill layers. For example a drill file named *cond.drl* is imported in layer *cond_drl* in ADS.

> **Note**
> Only RS274X and MDA formats are supported in gerber import.

This section describes the definable options available for importing files in the Gerber/Drill format. To view the options:

1. Click **File > Import** from the *Layout* window.
2. Select *Gerber/Drill* from the **File type** list box.
3. Enter the Import file name and Layers file name in the appropriate fields.
4. Click **More Options...** to open the *More Options* dialog box.

The *More Options* dialog box enables you to specify units, number format, mode, zero suppresion and layers to control the import of Gerber/Drill format files. A Gerber/Drill file and layers file must be selected in the main *Import* dialog box for the *More Options* dialog box to open. User options preferences are saved in *setup_gerber.opt* file.
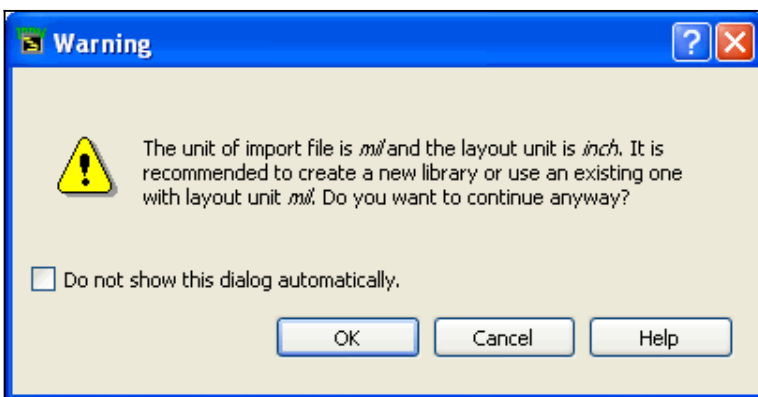
Gerber/Drill Options
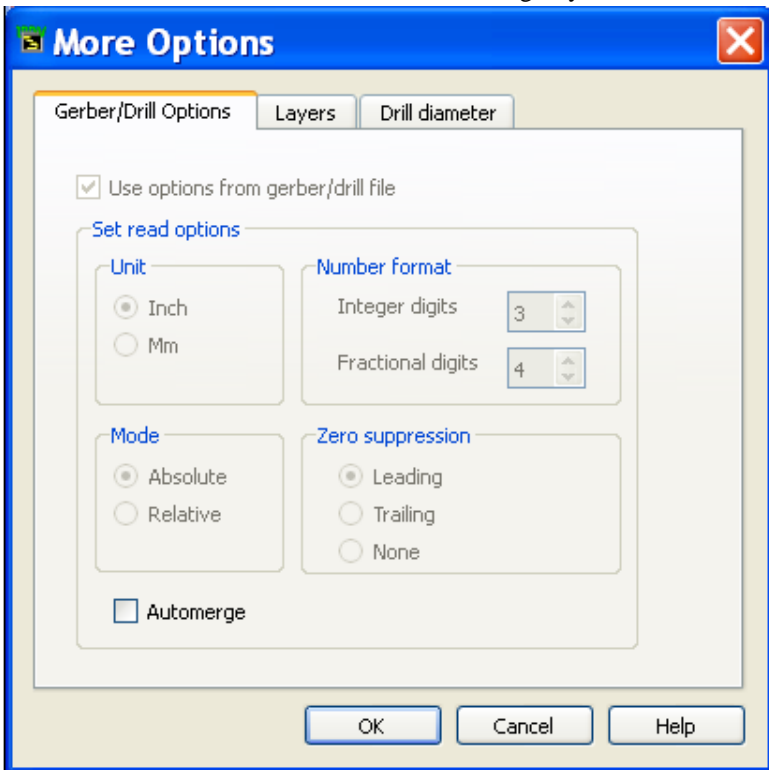
Units

Older versions of the Gerber file format may not specify units. If no Units were specified in the input Gerber file, this option is enabled. Default setting is *inch*.

If the unit mentioned in the import file is different form layout unit the following warning message is displayed.

For step-by-step tutorial on creating a new library, refer to *Creating Libraries* (adstour). To change the units of the current library refer to *Technology Setup* (usrguide). Click **OK** to continue importing on current layout.

If you check the *Do not show this dialog automatically.* check box then no unit mismatch warning message is displayed. To display the warning message box again edit *%HOME%/config/de_sim.cfg* and set the value of *UNIT_WARNING* environment variable to **TRUE**.

> **ℹ Note**
> This Unit Warning also appears if there is a unit mismatch while importing DXF, ODB__, and Mask files.

Number format

The number of integers placed before and after the decimal point. If no number format is specified in the gerber file, then this option is enabled. While importing multiple files together the number format specified in the first gerber file is used for all other files. For drill files if number format is not set in the file itself user can specify the number format. The default is 2.4.

Mode

If no Mode is set in gerber file this option is enabled. The default Mode setting is *Absolute*.
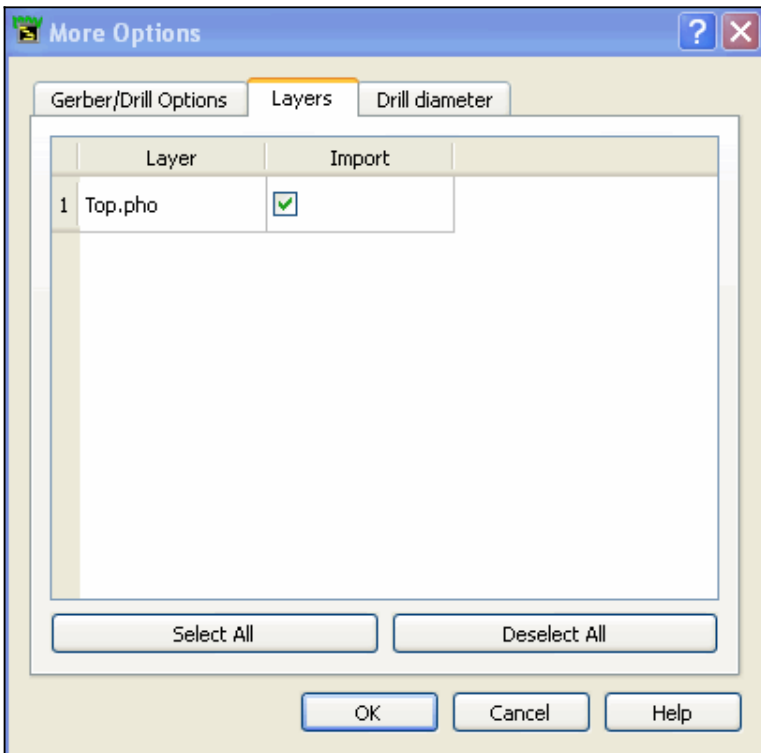
Zero Suppression

Available settings are Leading and None. If zero suppression is not specified in the Gerber file this option is enabled. The default setting is *leading* zero suppression.
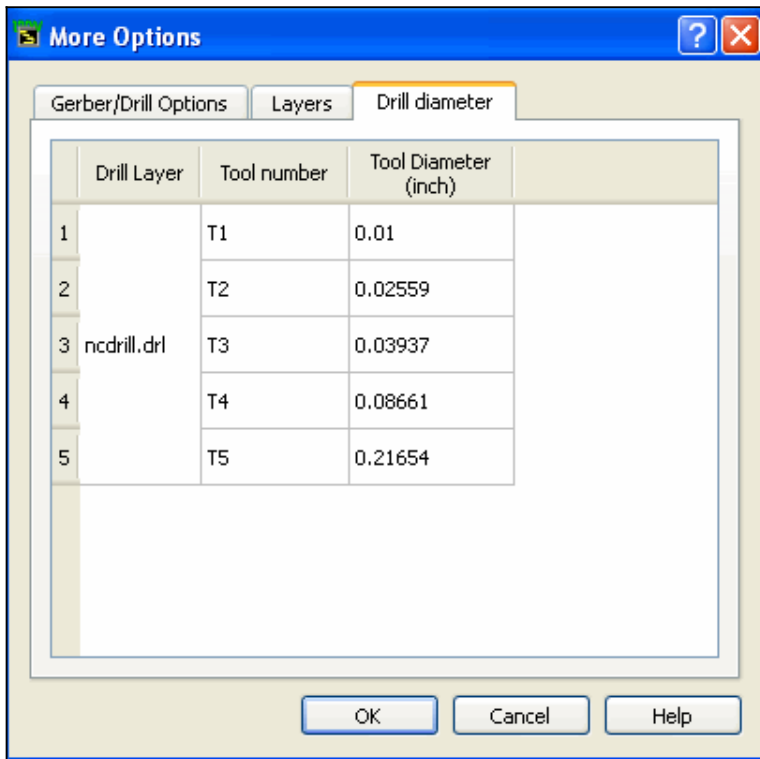
Automerge

By default, the **Automerge** option is unchecked. The figures are merged during the import if the **Automerge** option is checked.

Layers Options

The Layers tab displays a list all the layers in Gerber file. Layers can be selectively imported to ADS using the import column. By default all layers are imported.

Drill diameter Options

The Drill diameter tab allows user to set drill diameter during import. This is particularly useful when the drill diameter is not mentioned in the file or mentioned in seperate drill header file.

> **Note**
> Since the Gerber and drill files have seperate options it is recommended to import Gerber and drill file seperately in same layout.
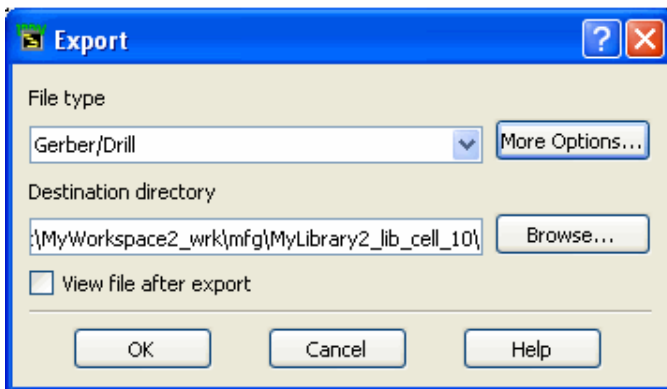
Exporting Gerber Files

This tool can generate Gerber files in RS274X and MDA format for raster photo plotters.

The Gerber export tool creates one output file for each different layer used in the layout design. For example, if the design contains two layers, cond and cond2, two Gerber files will be generated:

- cond.gbr
- cond2.gbr

To export the Gerber files:

1. In the layout window containing your design, choose **File > Export**.
   The Export dialog box opens.
2. In the *File type* field, select **Gerber/Drill**.
3. Optionally, you can select the *View file after export* check box to open the Pre-Production Editor where you can compare the Gerber files to the original ADS layout design to verify that the design was exported correctly. For details, see *Using the Pre-Production Editor* (trans).

4. To define export options, click **More Options**.
5. Set the options that apply.
   For available options, see Gerber File Options.
6. Click **OK** to save your settings or **Cancel** to retain the default settings.
7. In the *Destination Directory* field, accept the default directory, or click **Browse** to locate a new destination directory.
   The destination directory will be the *<Workspace Dir>/mfg/<Design Name>/gerber/* where:
   <Workspace Dir> is the workspace directory
   <Design Name> is directory named with the design name
8. Click **OK** to export the ADS layout design to a Gerber file.
   A Status window appears detailing the export information.
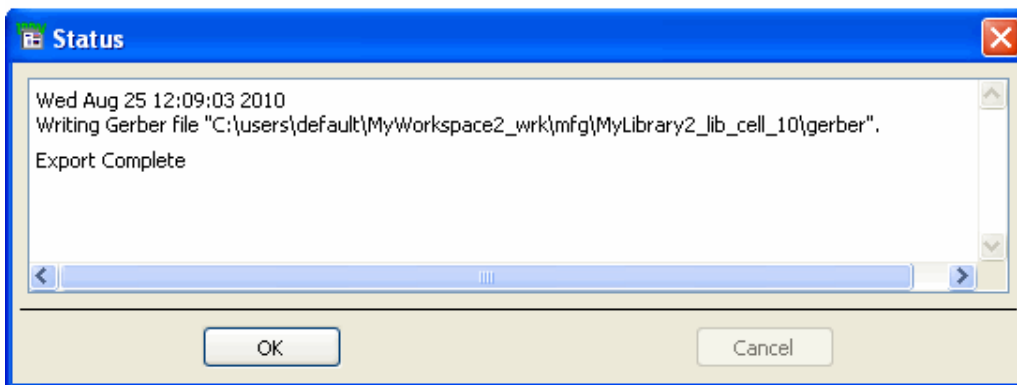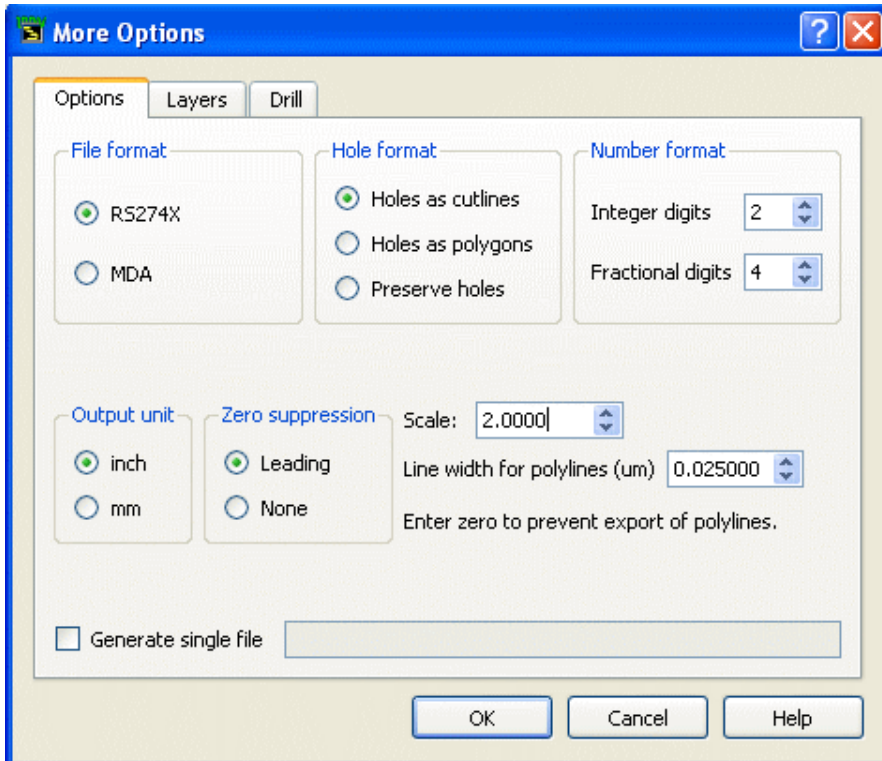

9. Examine the log, searching for any warnings or errors that may have occurred during translation.
   To save the log message in a file: select the text, copy, and then paste into any text editor.

Gerber File Options

You can view or edit the translator settings, by clicking **More Options** in the *Export* dialog box.

More Options dialog opens, providing access to these options.

Gerber file choices can be edited in the *Options* tab.

File format

- Select **RS274X** for an enhanced version of Gerber RS274D format that supports embedded apertures and G36/G37 polygons.
- Select **MDA** if your photoplot shop uses a FIRE 9000 plotter. It embeds apertures and supports POEX/POIN polygons for fill on the fly.

Hole format

In ADS, when a hole is created in a geometrical figure (like polygon), a cutline is introduced. This is a false edge connecting the outer boundary of the polygon with the inner boundary. This polygon is actually a single re-entrant polygon. When you select **Holes as cutlines** this re-entrant polygon is translated to Gerber as-is. The default is **Holes as cutlines**.

Holes as Cutlines
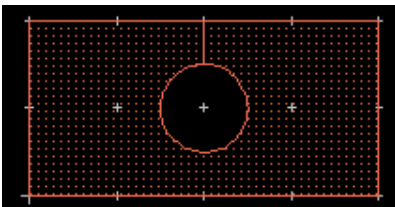
Select **Holes as polygons** or **Preserve holes** to remove the false edge from the polygon.
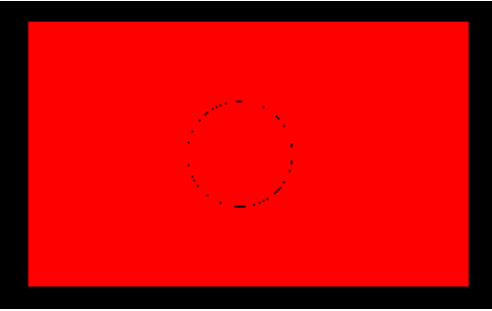
Holes as Polygons

When **Holes as polygons** is selected, holes are exported as filled elements. Therefore the polygon will appear to have no holes.

Preserve holes

When **Preserve holes** is selected, the resultant polygon in Gerber contains a dark area and empty area. This option is not available for exporting Gerber files in RS274X format.

Unit

Available units are inch or mm. Inch is the default.

Number format

The number of integers placed before and after the decimal point. If chosen incorrectly, Gerber data resolution can be poor. The default is 2.4. If the unit is set to mm then you can set the number format to 3.3.

A warning message is displayed in the Status window to notify the user that to preserve the precision in the Gerber data, number format has to be selected carefully. If the Gerber data is not generated correctly for the selected number format then a suitable error message is displayed in the Status window.

Zero suppression

Available settings are Leading and None. Select **Leading** (the default) to remove all leading zeros in the coordinate data, making the Gerber file smaller.

Scale

Set *Scale* to a value other than 1.0 to scale the design during Gerber export.

> **ℹ Note**
> The drill data is scaled by the same scale factor. The polyline width set in the dialog is *not* scaled.

Line width for polylines

Polylines in ADS have zero width, but you can use this option to provide a width to be used for drawing this polyline in the Gerber file.

If the polylines and arcs (both zero width entities) are not required to be exported, then set the width to zero.

Gerber Layer Options

All the layers used in the design are displayed in a table in the *Layers* tab of the *More Options* dialog. By default, all the layers will be displayed with positive polarity and all layers will be exported. You can avoid exporting a specific layer by deselecting the checkbox in the Export column for that layer.
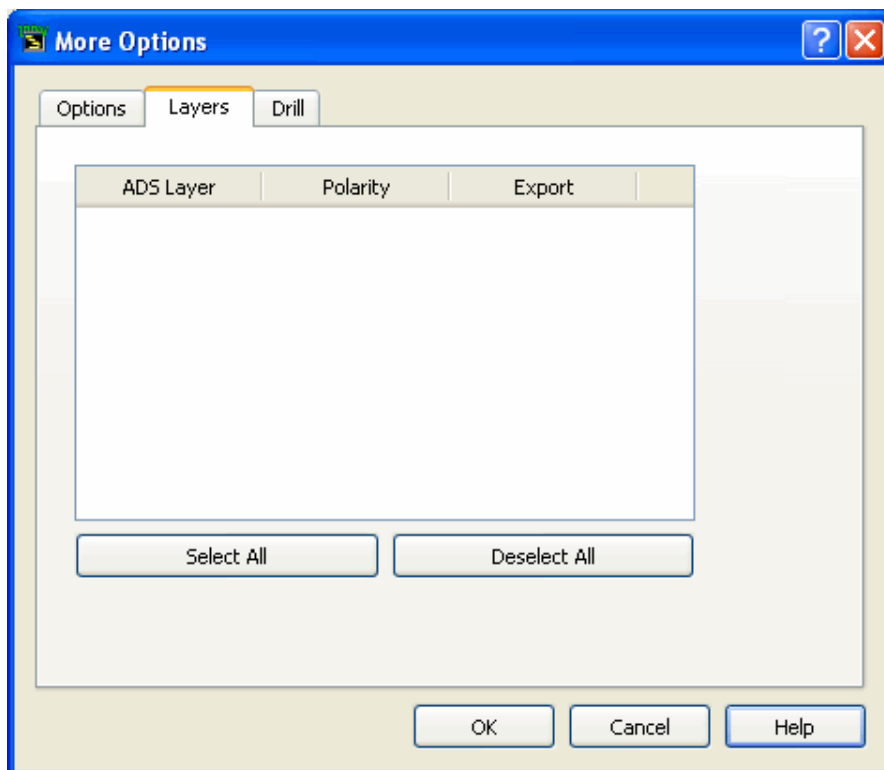

Limitations and Considerations

- No intermediate mask file is generated.
- The new Gerber export tool is targeted for the raster photoplotters to generate RS274X and MDA file format.
- No aperture filling is supported for polygons.
- No separate aperture file is required for RS274X and MDA files.
- It is recommended to create the ADS design using inch or mm as the unit and set the appropriate resolution.
- You must set the appropriate number format in the *More Options* dialog so that the precision in Gerber data is more than the precision in layout design.

Exporting Drill Files

In ADS, drill file export is accomplished simultaneously as part of the Gerber export. The procedure for generating the drill file is similar to generating the Gerber file.

Drill files are configured from the *Drill* tab in the *More Options* dialog.
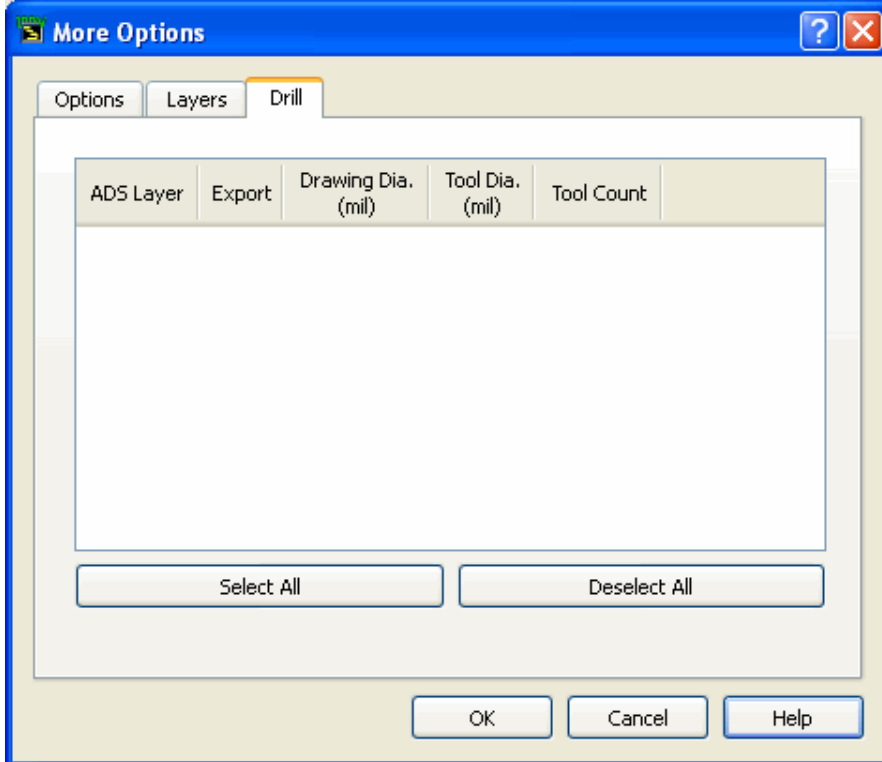
Select the checkbox to designate the layer for which you want to generate the drill file. When a layer is selected, all the tools on that layer are automatically selected. By default, no drill file will be generated during Gerber export.

The drill file options *Number format*, *Output unit*, and *Zero suppression* are same as options set for Gerber export in *Options* tab of *More Options* dialog. However, note that the zero suppression in Drill is actually a zero inclusion.

Tool Dia. (Diameter) is an editable field. By default the Tool Diameter value equals the Drawing Diameter value.

Considerations:

- If a layer does not have any via (circles) then it will not be displayed in the *Drill* table and no drill file will be generated for that layer.
- One drill file will be generated for each layer.
- Drill files will be created in the Destination directory selected in the *Export* dialog.
- Drill file name is the same as layer name with extension .drl. For example, drill file for the layer *cond* will be *cond.drl*.
- If Tool Diameter is set to 0.0, the Tool Diameter will be equal to the Drawing Diameter in the output drill file.
- The drill data is scaled by the factor set by *Scale* on the *Options* tab.

GDSII Stream File Translator

In this section:

- [Introduction](#)
- [Importing GDSII Files to ADS](#)
- [Exporting an ADS Design to GDSII](#)

Introduction

The end result of many MMIC designs is a Calma or GDSII stream format file. Advanced Design System's Layout provides a flexible GDSII translator for both reading and writing this format. The GDSII Stream File Translator is a bi-directional graphics file translator. It can create files in GDSII Stream file format from Advanced Design System layouts, and it can translate graphics files from GDSII Stream file format into Advanced Design System layouts.

The Advanced Design System to GDSII Stream file format translation links the Advanced Design System directly to mask-making equipment that uses the popular GDSII Stream file format. This format, in turn, gives access to a wide range of photoplotters, coordinatographs, E-beam machines, and pattern generators. The GDSII Stream file format to Advanced Design System format translation makes it easy to use the cell libraries offered by many GaAs semiconductor foundries.

Translation from a layout in the Advanced Design System into GDSII Stream file format is done with the menu command **File > Export > GDSII Stream Format**.

The table below describes GDSII elements and their Advanced Design System layout equivalents.

Table: GDSII Elements and their ADS Layout Equivalents

Calma/GDSII Element	Layout Equivalent(s)
Box element	Rectangle
Boundary element	Polygon, Rectangle
Path element	Polygon
Boundary element	Circle, Polygon
Element	Text
Boundary element	Hole
Structure element	Design
Sref element	Instance
None	Pin
Path element	Wire
Path element	Trace
Plex	None
Array	Instances
Node	None

Importing GDSII Files to ADS

The Advanced Design System reads GDSII files into a layout design without circuit or schematic information, internally converting the drawing data to the current layout units. For a step-by-step tutorial, see *Importing a Layout* (trans). For import options, see *Import GDSII Options.*

To import GDSII files to ADS:

1. Click **File > Import > GDSII Stream Format**
2. Enter the **Import file name** and **Layers file name** in the appropriate fields
3. Click **More Options...**

Import GDSII More Options

The *More Options* dialog enables you to specify flattening and layers to control the import of GDSII format files. A GDSII file and layers file must be selected in the main export dialog for the *More Options* dialog to open.
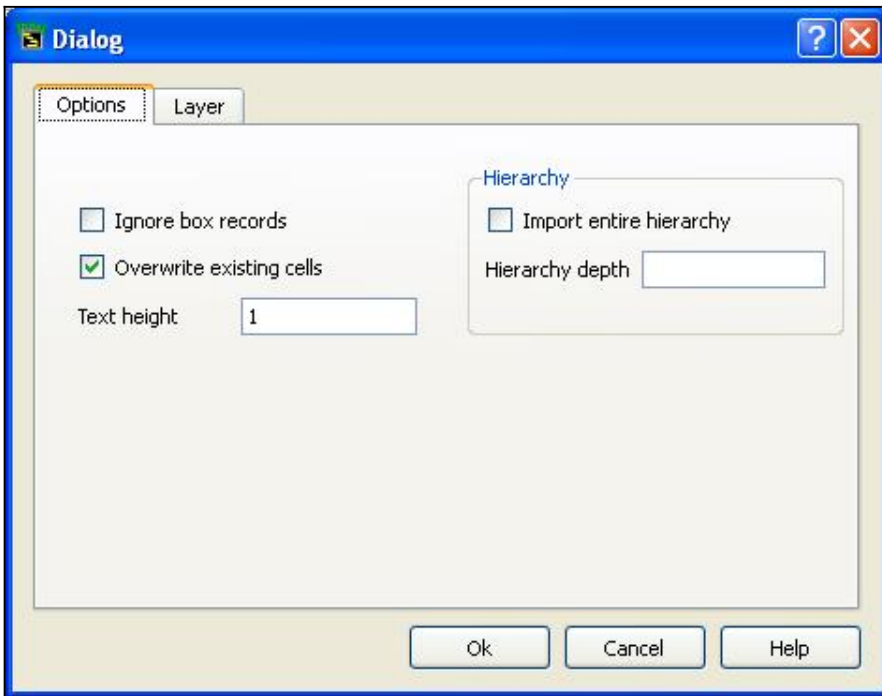

Figure: Options for importing GDSII files to ADS

You can Check/Uncheck/specify information in the following check/edit boxes:

Ignore box records

This option specifies that GDSII Box Records are not translated. By default, Box Records are translated as rectangles.

Overwrite existing cells

If this option is checked, GDSII import will overwrite an existing design with the imported

data. If this option is omitted, any cell in the GDSII file that corresponds to a pre-existing design is silently skipped. By default existing cells are overwritten.

Text height

This option specifies the text height for labels. The default value is 1 user unit.
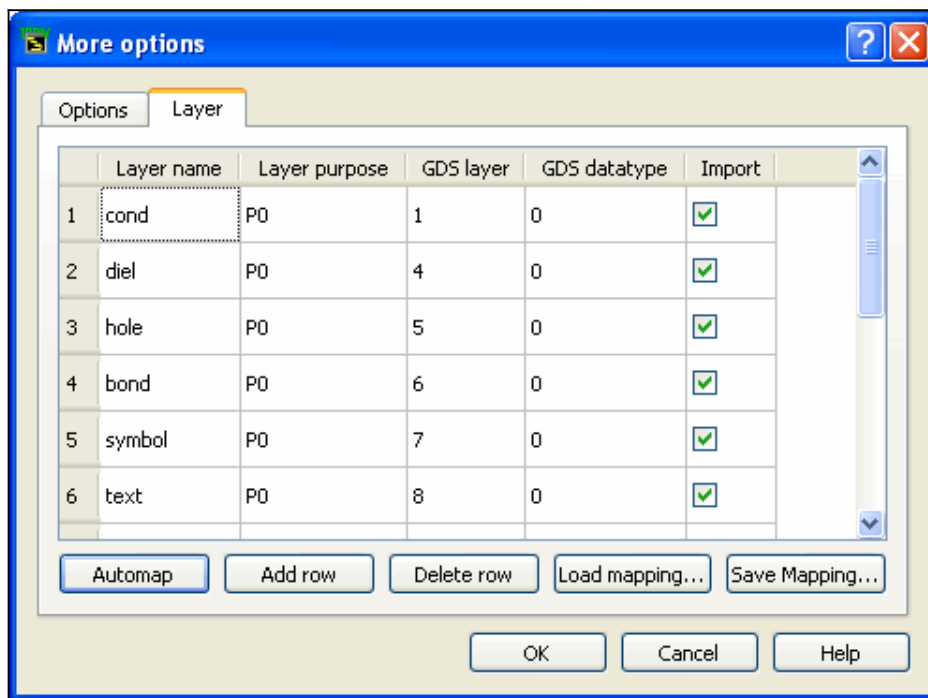
Hierarchy

Import entire hierarchy

If selected GDSII import imports entire hierarchy.

Hierarchy depth

Specifies the depth of hierarchy to import. The default hierachy depth is 20.

Layers

Figure: Layer options for importing GDSII files to ADS

Layer tab allows you to perform a layer mapping between ADS Layer purpose, GDS Layer, and GDS Datatype during import. The following details are listed for layers under Layers tab:

- Layer name
- Layer purpose
- GDS layer
- GDS datatype
- Import

Importer creates GDSII objects on the ADS layer and purpose found in the technology

library using the GDS Layer Number and GDS Datatype specified in the mapping. GDSII objects on GDS Layer and GDS Datatype combinations not found in the mapping are not translated. Default layer mapping or Automap, creates GDSII objects on same layer and purpose as GDS Layer and GDS datatype. If no layer is found for GDS layer and purpose a new layer L<GDS Layer Number> and P<GDS Datatype> is created in the technology.

You can create a custom layermap file for layer mapping in the following format:

`<ADS Layer Name> <ADS Layer Purpose> <GDS Layer Number> <GDS Purpose>`

Lines in the layerMap file that are blank and lines beginning with the comment character # are ignored by the importer.

Click on the following buttons to perform the following actions:

- **Add row** - Adds a new row in the layers list.
- **Delete row** - Deletes selected row.
- **Load mapping** - Allows user to browse and load an already defined layermap file. The loaded custom map is remembered by the importer for a single ADS session. Restart ADS to reset layer mapping to default mapping or automap.
- **Save mapping** - Allows user to save the layer map file.
- **AutoMap** - Enables automatic mapping. With Automapping during import, GDSII layers and purpose are translated to L<*GDSLayer*> and P<*GDS Datatype*> in ADS. During Export with automap, objects are created in same GDSII layer and purpose as ADS. Click **Automap** to clear any prevously loaded custom layer map.

Click **OK** to import GDSII files to ADS.

Exporting an ADS Design to GDSII

To export GDSII files:

1. Follow the steps as outlined in *Exporting a Layout* (trans).
2. After clicking **OK** in the *Export* dialog box, the translator converts your ADS Layout to GDSII format.
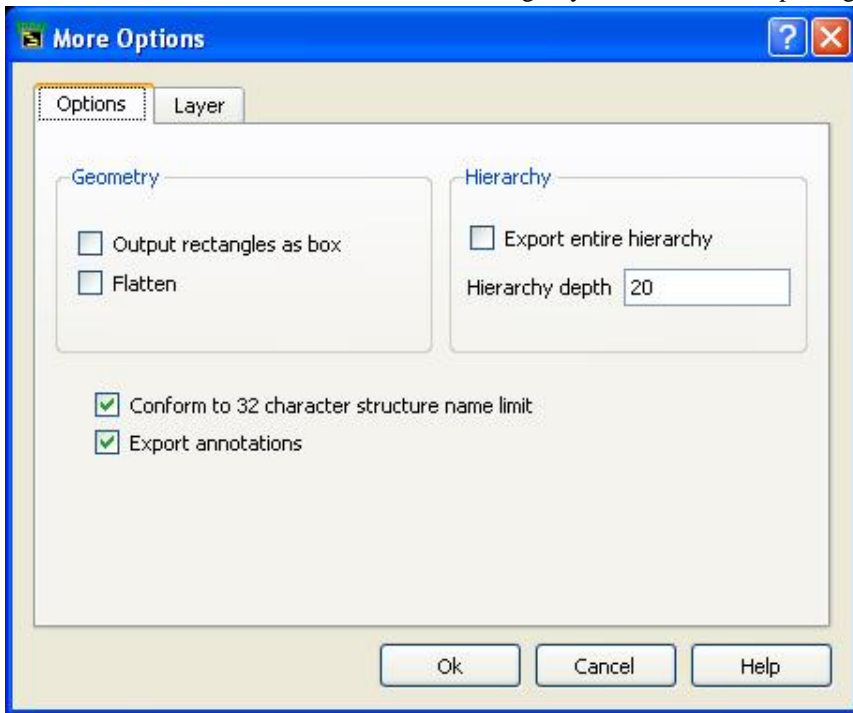
Once in GDSII file format, the file is available for use in AutoCAD or other Mechanical CAD systems.

Export GDSII More Options

The Export GDSII *More Options* dialog box enables you to select the geometry and hierarchy options.

Options

Figure: Options tab for exporting GDSII files

You can Check/Uncheck/specify information in the following check/edit boxes:

Geometry

Output rectangles as box

Selecting this option generates output rectangles as box.

Flatten

Selecting this option flattens a hierarchical design.

Conform to 32 character structure name limit

Selecting this option will not allow you to enter more than 32 characters for a structure name.

Export annotations

Selecting this option will enable you to export annotations.

Hierarchy
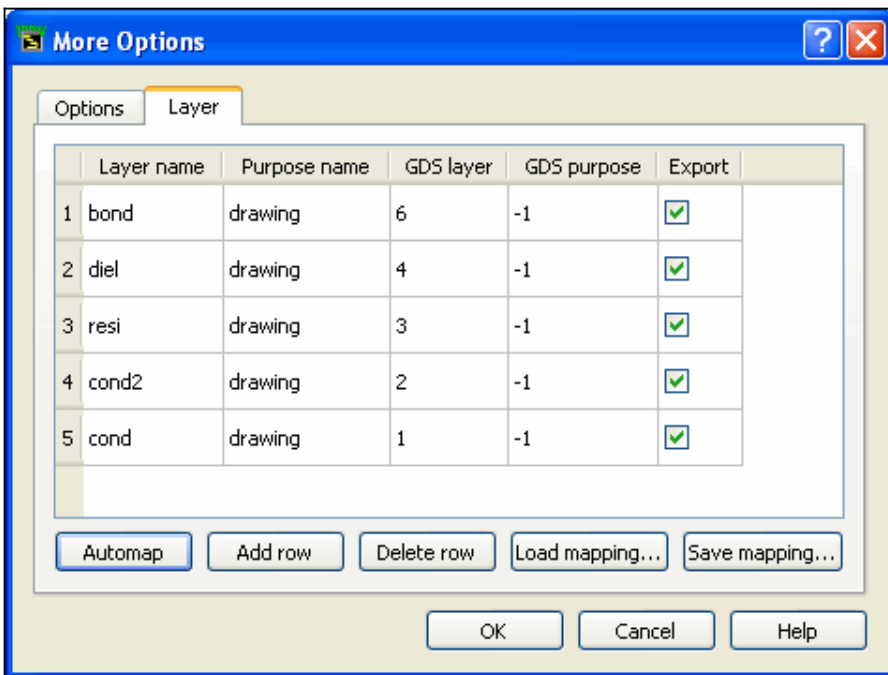
Export entire hierarchy

If selected GDSII exports entire hierarchy.

Hierarchy depth

Specifies the depth of hierarchy to import.

Layers

Figure: Layer options for exporting GDSII files

Layer tab allows you to perform a layer mapping between ADS Layer purpose and GDS Layer and GDS Datatype. The following details are listed for layers under Layer tab:

- Layer name
- Purpose name
- GDS layer
- GDS purpose
- Export

Objects on ADS layer and purpose combinations found in the mapping are created on the GDS layer and datatype specified in the mapping. Objects on ADS layer and purpose combinations not included in the mapping are not created in GDS file. Default layer mapping or Automap creates GDS object on same layer and datatype as in ADS layer and purpose.

User can create custom layer mapping file for the purpose of export. The format of layer mapping file is

<ADS layer name> <ADS layer purpose> <GDS layernumber> <GDS datatype>

Click on the following buttons to perform the following actions:

- **Add row** - Adds a new row in the layers list
- **Delete row** - Deletes selected Row
- **Load mapping** - Allows user to browse and load an already defined layermap file. The loaded custom map is remembered by the exporter for a single ADS session. Restarting ADS resets layer mapping to default mapping or automap.
- **Save mapping** - Allows user to save the layer map file.
- **AutoMap** - Enables automatic mapping. During Export with automap, objects are created in same GDSII layer and purpose as ADS. Click **Automap** to clear any prevously loaded custom layer map.
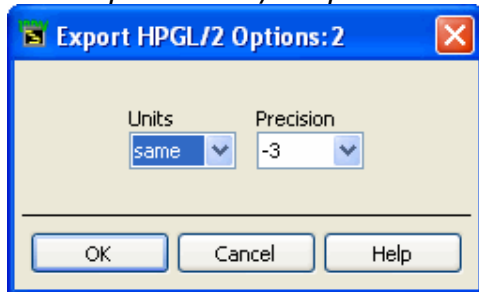
HPGL/2 Files

The HPGL/2 format is a subset of the HPGL22 printer/plotter language and can be exported from a Layout.

Exporting HPGL/2 Graphics Files

The procedure for exporting each format is generally the same, however the available options differ. For a step-by-step tutorial on exporting a layout file, refer to *Exporting a Layout* (trans). For specific export options related to exporting HPGL/2 Files, refer to Export HPGL/2 Options.

Export HPGL/2 Options

The *Export HPGL/2 Options* dialog box enables you to set *Units* and *Precision*.

Units

These are the units with which the HPGL/2 file will be exported. You may select from the following options: *same*, *mil*, *inch*, *um*, *mm*, *cm*. The default is *same*. When *same* is selected, the design is written in the same units that are stored in the design file. For more information on choosing layout units, refer to Setting Layout Options > Preferences for Layout > Setting Units/Scale Factors in *Customization and Configuration* (custom).

Precision

This value should be the same as the precision with which the drawing file was created. A warning is generated if the precision is less than the drawing file precision. Possible values are −2 and −3. The default is −3.

IFF Files

The Intermediate File Format (IFF) is an ASCII intermediate file. The file has a simple, line-oriented command structure with a fairly rich set of constructs, thus simplifying design transfer between Agilent Technologies products and third-party EDA tools. This format is machine- and application-independent.

The types of information that can be represented in the IFF format include:

- Design Objects
    - Symbols
    - Layouts
    - Schematics
- Connectivity
- Design Object Hierarchy
- Hierarchical and Design Object Properties

IFF files are used as the exchange mechanism when transferring designs between Advanced Design System and third-party EDA tools such as Mentor Graphics Design Architect and Cadence Analog Artist.
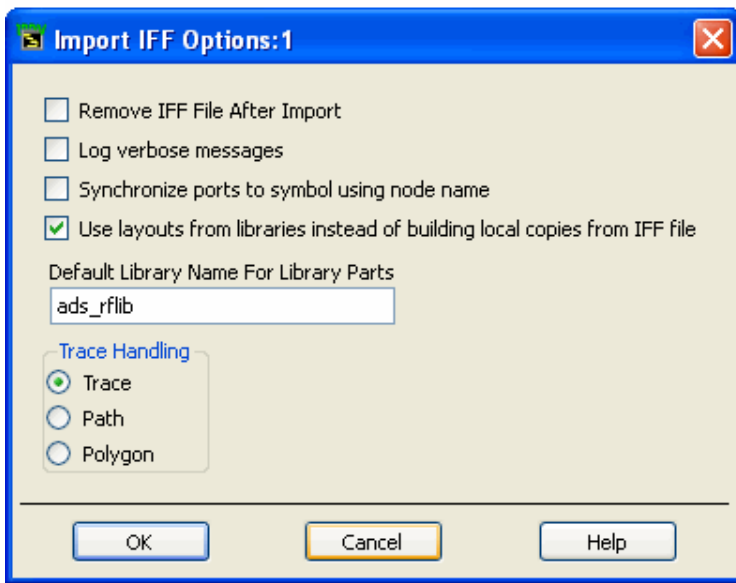
Importing IFF Files

The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to *Importing a Layout* (trans). For specific import options related to importing IFF Files, refer to Import IFF Options.

Import IFF Options

The *Import IFF Options* dialog box that appears is dependent upon where you execute the IFF import from.

Importing from a Layout Window

If the import is invoked from the Layout window, the following *Import IFF Options* dialog box appears.

Remove IFF File After Import

When this option is selected, the *.iff* file is removed once the file is successfully imported. This option is not selected as the default.

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

When this option is selected, library elements that already exist in the system libraries are *not* recreated for the imported file. Instead, these elements are read from the local libraries; if an element does not exist in a local library, then it is newly created. This option is deselected as the default, and all elements are created/recreated in the local

workspace.

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.

> **ⓘ 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.

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-EDA's Solution Services.

Trace Handling

Trace Handling enables you to select how you want your *meander elements* interpreted during a translation.

Trace When this option is selected, *meander elements* are translated as simulatable traces with pins. This is the default setting.
**Path** When this option is selected, *meander elements* are translated as primitive data type with center line and width. The default for this option is deselected.
Polygon When this option is selected, *meander elements* are translated as polygons. The default for this option is deselected.

> **ⓘ Note**
> If you choose *Use layouts from libraries instead of building local copies from IFF file* in the Import IFF Options dialog box, choose *Path* in the *Trace Handling* options. Fixed artwork is not simulatable, therefore it is not necessary for the interconnects to be simulatable.

Importing from a Schematic Window

If the import is invoked from the ADS *Schematic* window, the following *Import IFF Options* dialog box appears.
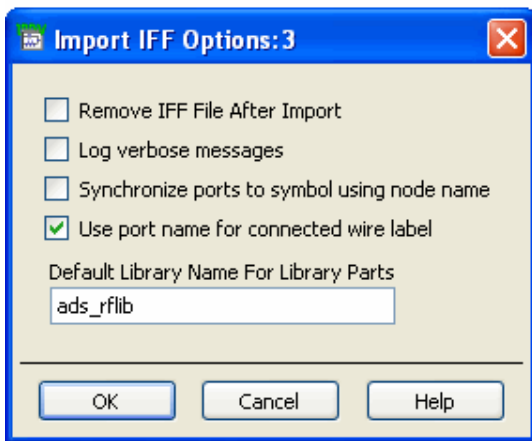
Note that the options for Importing from a Schematic window and from a layout window are same except the *Use layouts from libraries instead of building local copies from IFF file* check box and the *Trace Handling* options. For information on these options, refer to Import IFF Options. Apart from this a new option *Use port name for connected wire label* is added.

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. To maintain proper connectivity in the ADS environment, it is necessary to enable this option if connectors share a common name.

Exporting IFF Files

To export an IFF file:

1. Follow the steps as outlined in *Exporting a Layout* (trans). For available options (accessed via *More Options* in the Export dialog box), see Export IFF Options.
2. When the translation is complete, an Information Message window appears. Click **OK** to dismiss this window.
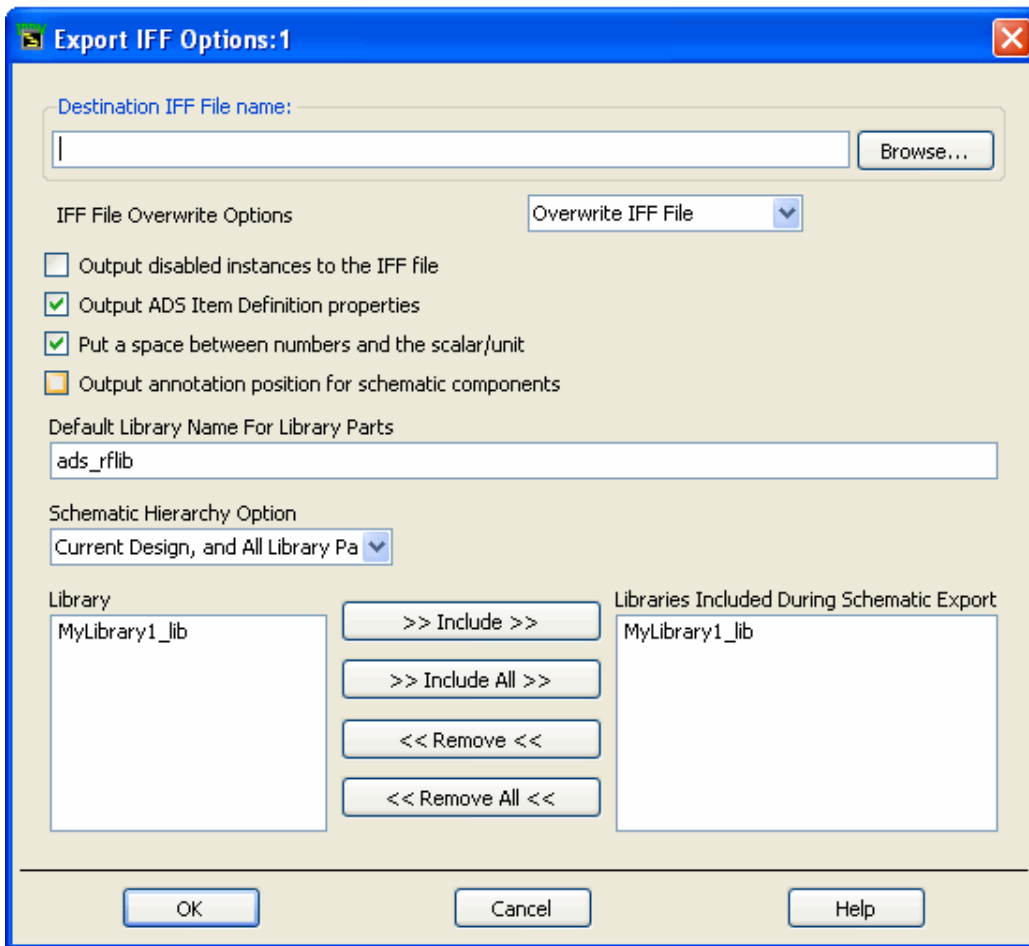3. The IFF Export log appears:
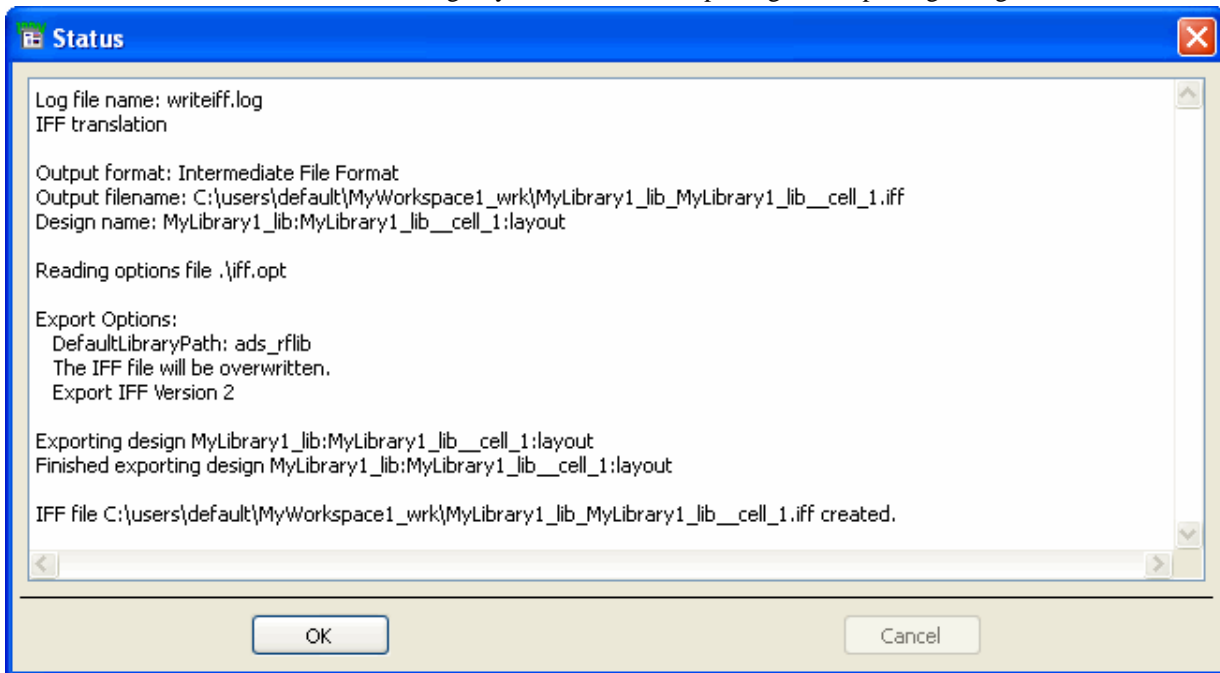
Review the log, searching for any warnings or error messages generated during export.

The log file appears in the *Status* window by default. This window is provided as a means of viewing the file and is not intended for editing.

4. Click **OK** to dismiss this window.

Export IFF Options

Destination IFF File name

Use the *Destination IFF File name* field to enter the full path of the destination file. Alternatively, you can click **Browse** to open the Export File Selection dialog box and locate the destination path. After selecting the destination path and entering the file name, click **OK** to accept the selection and return to the Export IFF Options dialog box. The appropriate suffix is appended to the filename automatically.

IFF File Overwrite Options

Overwrite IFF File When writing to an existing file, the contents of that file are overwritten.
Append to IFF File When writing to an existing file, the new data is appended to the existing file. This is the default setting.

Output disabled instances to the IFF file

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

Output ADS Item Definition properties

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

Put a space between numbers and the scalar/unit

When this option is selected, parameter values are exported as they normally appear in ADS (i.e. with a space between the number and the scalar, e.g. "1 pF"). If the checkbox is deactivated, the exporter converts the values into the IFF value specification, which is to have no space between a number and a scalar (e.g. "1pF"). Ideally, an IFF exporter should interpret either form of number, and set the value internally to whatever is normal for that environment. Some environments (e.g. Mentor Graphics) do not interpret the IFF property values in any way. For Mentor IC, this means the numbers need to have no space in them, because, when they are used within SPICE simulations, the space will cause syntax errors in the simulator. However, for Mentor Board, they require the ADS components to have a space in them, because the RF Architect ADS library is set up to expect values to have a space between a number and a scalar/unit.

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

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, though 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.

> **ⓘ 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-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 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 library. If hierarchical, all included libraries are listed in the appropriate order.

Libraries Included During Schematic Export

The libraries for which schematic design information is exported. You may customize this list if the current library is hierarchical. (Note that complete layout hierarchy is always exported.)
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**.

IGES Translator

IGES (Initial Graphics Exchange Specification) is a neutral graphics database format designed primarily for data exchange between mechanical CAD systems. The IGES file format links mechanical CAD systems to Advanced Design System. Two-dimensional geometry can be used to interchange layout or package outline information.

The Advanced Design System IGES translator can either create or read an IGES ASCII form file. The basic input requirements for the translator are the file to be translated and the configuration message file. The output is the translated file.

The IGES translator can be run via the layout window menu commands **File > Import** and **File > Export**.

The IGES format can represent both mechanical and electrical design data in two and three dimensions.

For IGES output, it is important to consider the limitations and capabilities of the intended receiving system. IGES is a very general language. Many IGES translators understand only a sub-set of IGES entities. If the receiving system is CALS Level 1 compliant, there should be no problem. If it is not, before you begin layout carefully review the types of entities the receiving system is able to accept and what options are available in the layout output translator. The translator is extremely configurable, but may still be unable to output every entity in a form acceptable to another system.

Translator Description

 The Advanced Design System's IGES output is compatible with IGES versions 4.0 and 5.0. The output is CALS Level 1 compliant. However, because many IGES pre-processors accept different *types* of IGES files, the translator is designed to be highly configurable through the use of the Import and Export Options dialog boxes. For more information on the options provided in these dialog boxes, see the sections Import IGES Options and Export IGES Options later in this section.

Importing IGES Files

The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to *Importing a Layout* (trans). For specific import options related to importing IGES Files, refer to Import IGES Options.

Import IGES Options

To explore Import IGES Options, in *Layout* window, click **File > Import > IEGS > More Options**.

Precision

This value should be the same as the precision with which the drawing file was created. A warning is generated if the precision is less than the drawing file precision. Possible values are 0, −1, −2, −3, −4, or −5. The default is −3.

Exporting IGES Files

The procedure for exporting each format is generally the same, however the available options differ. For a step-by-step tutorial on exporting a layout file, refer to *Exporting a Layout* (trans). For specific export options related to exporting IGES Files, refer to Export IGES Options.

Export IGES Options

To explore Export IGES Options, in *Layout* window, click **File > Export > EGS > More Options**.
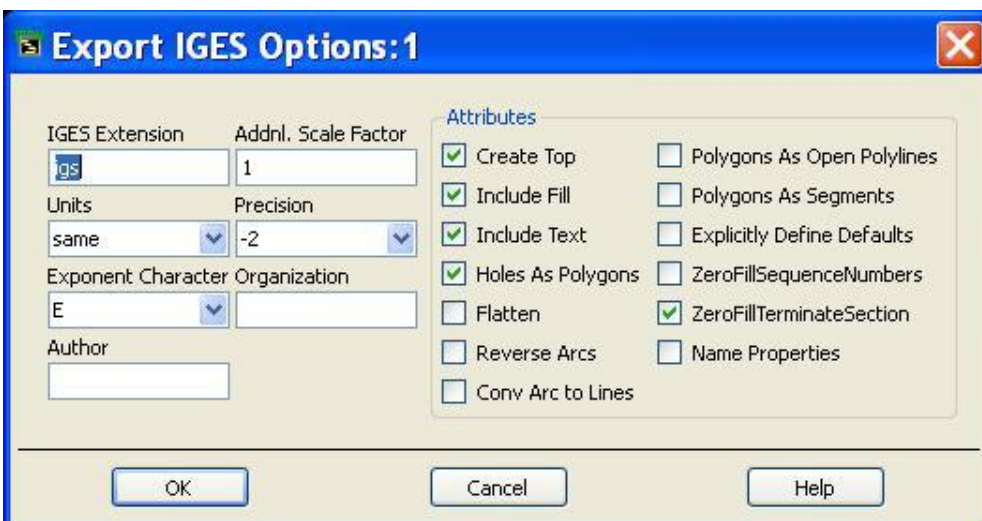
IGES Extension

If you do not specify an output file name when exporting the file, the program takes the input file name and appends this extension to create the output file name. The default extension is *igs* , but any three character string can be used.

Units

These are the units that the IGES file will be written in. You may select from the following options: *same*, *mil*, *inch*, *um*, *mm*, *cm*. The default is *same*. When *same* is selected, the design is written in the same units that are stored in the design file. For more information on choosing layout units, refer to *Setting Units/Scale Factors* (custom) in *Customization and Configuration* (custom).

Exponent Character

For case sensitive post processors, this option enables you to specify either upper-case (*E*) or lower-case (*e*) exponents. The default is *E*, or upper-case.

Author

Your name. This is written into the Global Section (the first three lines of the IGES file) and is not required.

Addnl. Scale Factor

If this value is other than 1.0, the output coordinates are multiplied by that value. The default is 1.0. You may enter any integer or double precision float number.

Precision

This value should be the same as the precision with which the drawing file was created. A warning is generated if the precision is less than the drawing file precision. Possible values are 0, −1, −2, −3, −4, or −5. The default is −2.

Organization

Name of your company or organization. This is written into the Global Section (the first three lines of the IGES file) and is not required.

Create Top

When selected, this adds a Substructure Instance Entity to the IGES file at the top level of hierarchy. *Create Top* is selected as the default.

Include Fill

This is selected as the default. However, not all post-processors support fill patterns. To eliminate fill patterns in filled polygons, deselect this item.

Include Text

This is selected as the default. However, not all post-processors support text. When this option is deselected, text is eliminated from the IGES file.

Holes As Polygons

This is selected as the default. However, not all post-processors support holes in polygons. When *Holes As Polygons* is deselected, polygons are written with false edges.

Flatten

 When *Flatten* is selected, all levels of hierarchy are automatically removed and a single flat design is translated. There will be no references from the top level structure to any other structure. This option is useful when your post-processor does not support or correctly translate hierarchy in IGES files. Please note: if a substructure was instanced more than once, selecting this option increases the size of the file.
This option is deselected as the default.

Reverse Arcs

When selected, the program writes all arc− as arc+ by interchanging the start and end points. This forces polygons to be written as line segments. This option is deselected as the default.

Conv Arc to Lines

When selected, the program translates an arc into of line segments. You should select this option if your post-processor places random arcs on a drawing. This option is deselected as the default.

Polygons As Open Polylines

When selected, closed polygons are written as open contours. This retains arcs in outline form. This option is deselected as the default.

Polygons As Segments

When selected, all copious data entities are changed to composite curve entities. Arcs are retained in outline form. This is useful for post-processors that cannot read IGES solids. However, when this option is selected, the size of the output f ile greatly increases because each line segment is written on a separate line, and each coordinate that is a vertex gets written twice. Also, knowledge of segments belonging to a polygon is *not* retained.
This option is deselected as the default.

Explicitly Define Defaults

If your post-processor complains because there are blanks in the Directory Entry Section fields, or that the parameter and record delimiters are not specified, select this option. Otherwise, *Explicitly Define Defaults* is deselected as the default.

ZeroFillSequenceNumbers

If your post-processor complains because the sequence numbers do not have zero fill, select this option. Otherwise, *ZeroFillSequenceNumbers* is deselected as the default.

ZeroFillTerminateSection

If your post-processor complains because the terminate section has zero fill, deselect this option. Otherwise, *ZeroFillTerminateSection* is selected as the default.

Name Properties

When selected, all *Name Properties* are preserved and exported with the IGES file. By

default, this option is deselected.

Overcoming Limitations of Other IGES Readers

IGES Reader Problem	Advanced Design System Solution
Can't read clockwise arcs	Convert arcs to lines (*Conv Arc to Lines*)
Can't read hierarchical IGES	Flatten the design
Can't read IGES solids	Reverse arcs (polygons are converted to line sets)
Can't read lower-case exponent	Specify Exponent Character as *E* (or *e*)
Text won't transfer	Deselect *Include Text*
Defaults not explicitly defined	Select *Explicitly Define Defaults*
Zero fill in termination section	Deselect *ZeroFillTerminateSection*
Zero fill not in se quential numbers	Select ZeroFillSequenceNumbers

Mask Files

Created by Agilent EEsof EDA for use with its EDA tools, the mask format is a simple ASCII format that provides a flat geometric representation of a layout. Because of its simplicity, this format offers great flexibility when transferring designs between the Advanced Design System and other design environments.

The Gerber and DXF translators use the mask file format as an intermediate file when converting data to Gerber and DXF. The mask format is also used as an intermediate file when translating other formats such as HPGL/2. In addition, the simplicity of the mask file format makes it an appealing option for post-processing designs and extracting drill hole file information (you will need to create your own programs for these tasks).

Mask files can include multiple layers of data, but this data describes geometric forms exclusively. Simulation data, element parameters, substrate definitions, and hierarchy are not included.
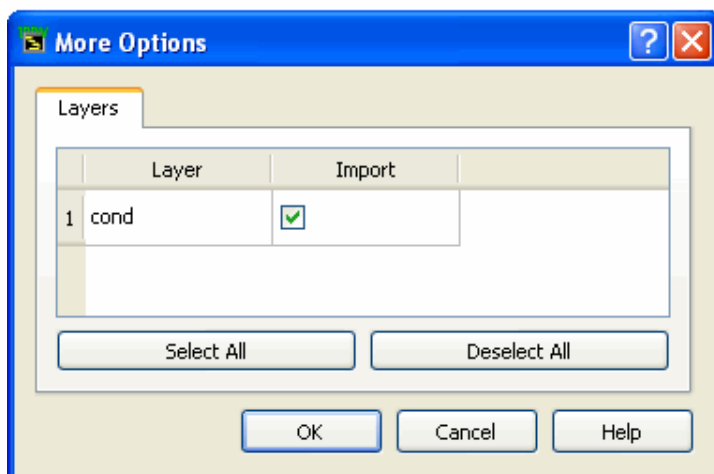
Importing Mask Graphics Files

For a step-by-step tutorial, see *Importing a Layout* (trans).

Import MASK Options

The *More Options* dialog box allows you to specify the layers for the import of MASK format files.

Layers

The *Layers* tab displays a list all the layers in a MASK file. Layers can be selectively imported to ADS using the import column. By default all layers are imported.

If the unit mentioned in the import file is different form layout unit then a *warning*
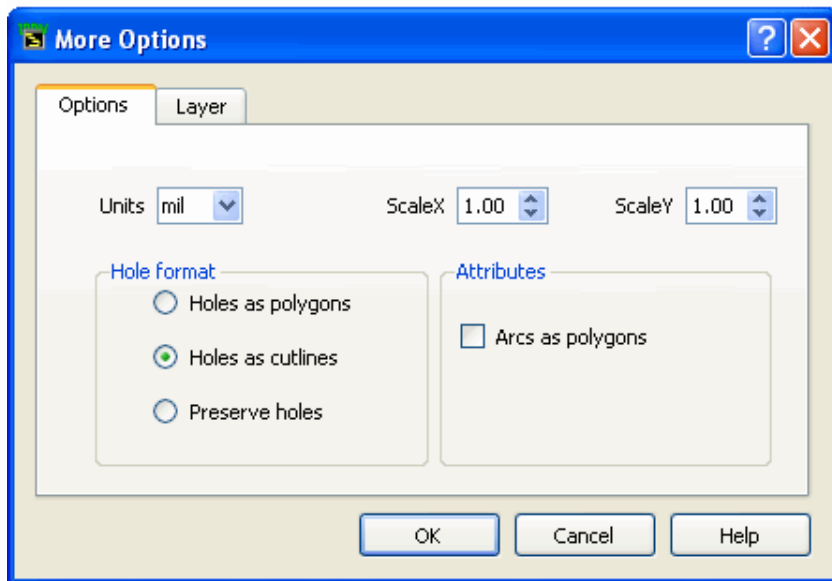
message (trans) is displayed.

Exporting Mask Graphics Files

For a step-by-step tutorial, see *Exporting a Layout* (trans).

Export MASK Options

The *More Options* dialog box allows you to set the options for units, formats, and layers for export.

More Options

Options

The *Options* tab allows you to select Units, Hole Format and Attributes in the translation.

Units

These are the units that the mask file will be written in. You may select from the following options: *mil*, *inch*, *um*, *mm*, *cm*. The default is *mil*. For more information on choosing layout units, refer to **Setting Layout Options > Preferences for Layout > Setting Units/Scale Factors** in *Customization and Configuration* (custom).

Scale X, Scale Y

These are the fields for inputting the scale factors for shapes in the direction of X and Y.

The default settings are 1.0, 1.0.

Hole Format

The Hole format section enables you to define how the translator deals with holes in a design.

Holes As Polygons

When **Holes As Polygons** is selected, holes are exported as a filled element, which is also how polygons are exported in the mask format. When **Holes As Polygons** is *not* selected, polygons with holes are translated as single-segment polygons, the *false edge* segment becoming part of the polygon.

> **ⓘ Note**
> Some systems may not be able to tolerate this type of complex polygon. For these systems, make certain that **Holes As Polygons** is selected. This option is deselected as the default.

Holes As Cutlines

When **Holes As Cutlines** is selected, holes are converted into cutlines. This option is selected as the default.

Preserve Holes

When **Preserve Holes** is selected, holes are exported as an empty element. This option is not available for exporting mask graphics files.
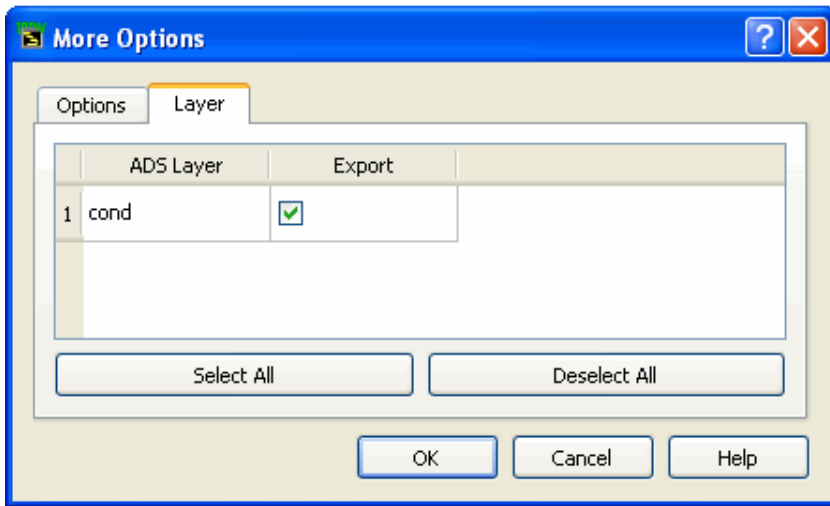
Attributes

Arcs As Polygons

When **Arcs As Polygons** is selected, the design arcs are exported as line segments (or polygons). This option is deselected as the default.

Layers

The *Layers* tab displays a list all the layers in a MASK file which can be selectively exported to ADS using the export column.

Layers

The **Export** column enables you to deselect one or more layers to exclude it from the output file. Default setting is all layers are selected and exported.

MGC/PCB Files

 MGC/PCB files are IFF files that are used exclusively for Mentor Graphics design transfers. This format is available from the Advanced Design System layout menu only, yet it enables the transfer of both schematic and layout information.

When you select the MGC/PCB export format from an Advanced Design System Layout window, both a layout and schematic IFF file are exported in a single step. The design data is exported into a standard directory tree contained in the program's workspace directory structure. The standard directory is called *to_mgc*.

The exported files are placed within this directory in a subdirectory that is named the same as the design being exported. This subdirectory contains an information file and the translated schematic and layout IFF files. Thus, if you were translating a design called *test* , the exported files (*design_info*, *schematic.iff* and *layout .iff*) would found in a directory called *to_mgc/test.hpxfer*.

From the Mentor Graphics Design Manager, a single command called   *import_hpeesof* imports the schematic and layout data into Boardstation and Design Architect. *This command automates to a single procedure the steps required to transfer both the schematic and the layout*.

Mentor PCB products do not accept layout hierarchy, so the entire layout is flattened prior to building the Mentor layout.

Exporting MGC/PCB Files

This section outlines the procedure for translating designs into MGC/PCB format. For more information on transferring designs between the Advanced Design System and the Mentor Graphics Falcon Framework, contact your Agilent Technologies sales representative.

To export an MGC/PCB file:

1. Follow the steps as outlined in *Exporting a Layout* (trans). For available options (accessed via *More Options* in the Export dialog box), see Export MGC/PCB Options.
2. If the option *Prompt For User Message* was selected, an Export MGC/PCB Options message dialog appears:
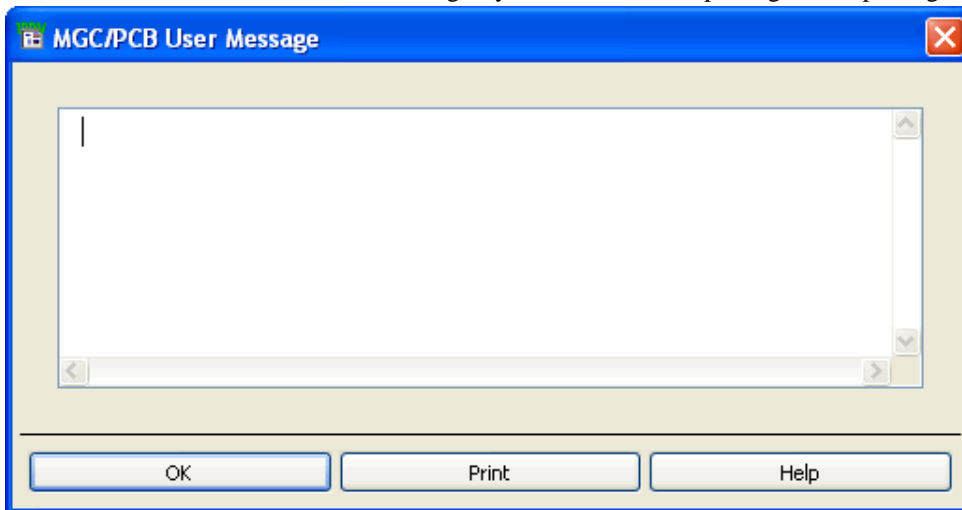
Enter any messages to be included with the design transfer. This information is used only by the *import_hpeesof* utility (see MGC/PCB Files) and is not kept with the design.

If you do not wish to include a message in the file, you may leave this window empty. If you want to print the message, click **Print**.

To proceed with the transfer, click **OK**.

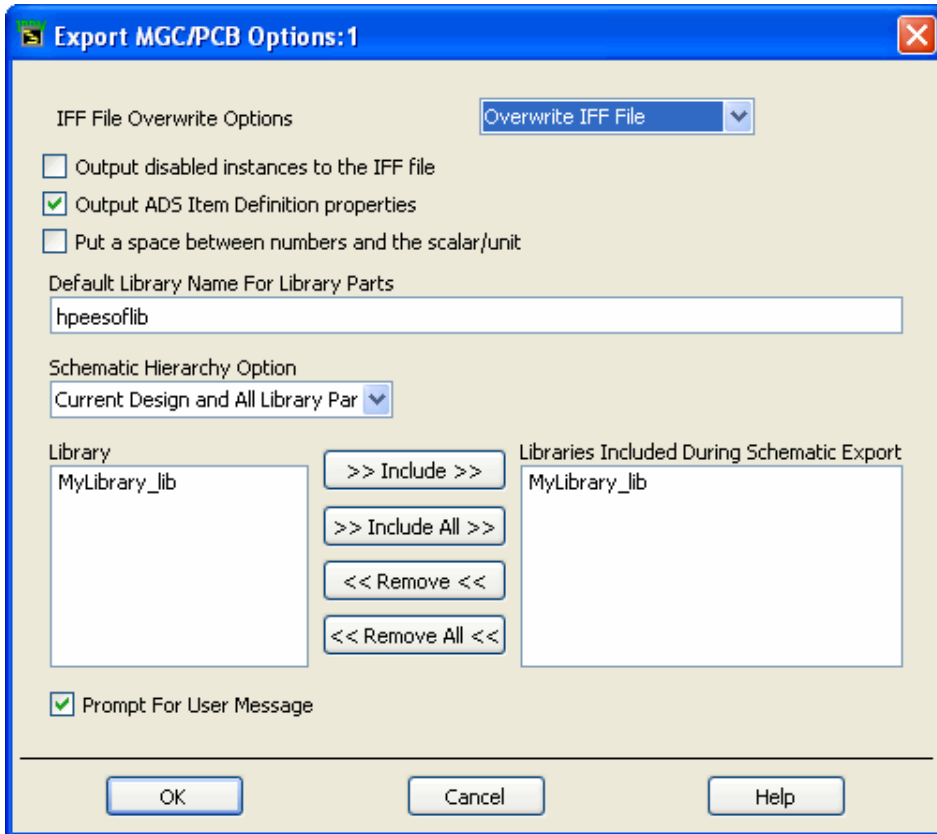3. When the translation is completed, the IFF Export status appears:
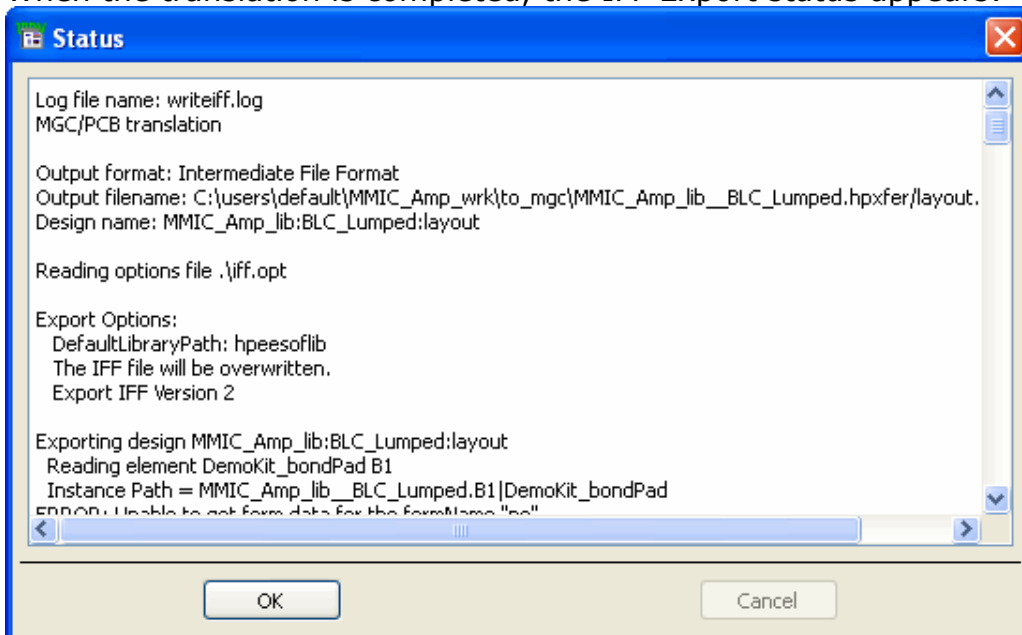
Review the log, searching for any warnings or error messages generated during export.

This window is provided as a means of viewing the file and is not intended for editing.

4. Click **OK** to dismiss the *Status* window.

Export MGC/PCB Options

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.
- *Append to IFF File*
  When writing to an existing file, the new data is appended to the existing file.

Default Library Name For Library Parts

The name of the library to which the library parts are written. Design objectives are stored in a group that uses the same name as the workspace directory, but library parts are stored in either the default library *hpeesoflib* or a library that you specify.

> **Note** The default library name can contain only alphabetic and numeric characters.

Output disabled instances to the IFF file

When this option is selected, if an instance is disabled in the schematic, it will still be output into the IFF file. If the checkbox is deselected (default), disabled instances will not be exported. This option can be utilized to omit certain components from being transferred to remote environments that might not support the components (e.g. disable the

simulation components prior to creating an IFF file to send to Cadence, which does not have any definitions for the simulator components). Activate this option if you want to get everything. Deactivate this option if you want to filter out the unused/unwanted components.

Output ADS Item Definition properties

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

Put a space between numbers and the scalar/unit

When this option is selected, parameter values are exported as they normally appear in ADS (i.e. with a space between the number and the scalar, e.g. "1 pF"). If the checkbox is deactivated, the exporter converts the values into the IFF value specification, which is to have no space between a number and a scalar (e.g. "1pF"). Ideally, an IFF exporter should interpret either form of number, and set the value internally to whatever is normal for that environment. Some environments (e.g. Mentor Graphics) do not interpret the IFF property values in any way. For Mentor IC, this means the numbers need to have no space in them, because, when they are used within SPICE simulations, the space will cause syntax errors in the simulator. However, for Mentor Board, they require the ADS components to have a space in them, because the RF Architect ADS library is set up to expect values to have a space between a number and a scalar/unit.

Schematic Hierarchy Option

This establishes 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 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 library. If hierarchical, all included libraries are listed in the appropriate order.

Libraries Included During Schematic Export

The libraries for which schematic design information is exported. You may customize this list if the current library is hierarchical. (Note that complete layout hierarchy is always exported.)
To add a library to this list:

In the Library list, click the desired library.

1. Click **Include**. The library is added to the Libraries Included list.
2. 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

When selected, a user message window appears before the transfer is initiated. This window enables you to enter any messages that you want included in the translated file. For more information about the user message, see Export MGC/PCB Options.

This option is selected as the default.

ODB++ Translator

The ODB++ Artwork Translator translates artwork directly from circuit layouts created with Advanced Design System into ODB++ format. Alternatively, ODB++ files can be converted to ADS designs also.

Importing ODB++ Files to ADS

To import ODB++ files:

1. From the layout window, choose **File > Import**. The *Import* dialog box opens.
2. Enter the **Import file name** and **Layers file name** in the appropriate fields.
3. Select either **ODB++ Directory Format** or **ODB++ File Format** from the **File type** drop-down list.
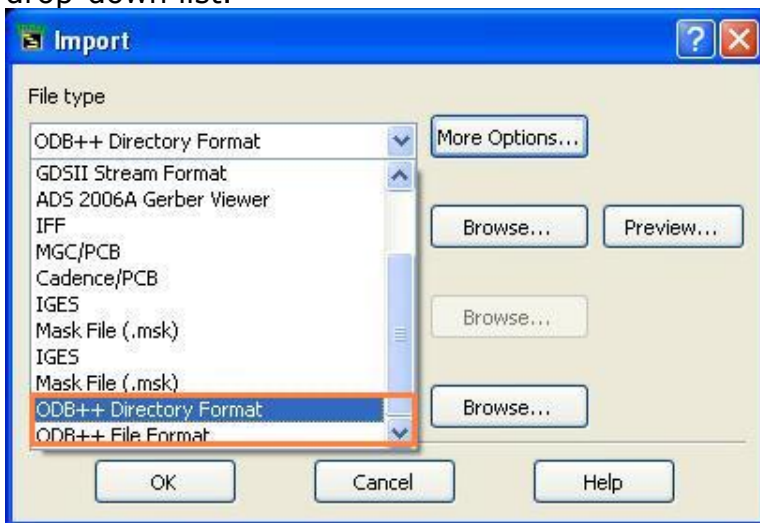
4. To view and/or define import options, click **More Options**.
5. Set the desired options. For available options, see Import ODB++ Options.
6. Click **OK** to import ODB++ file into ADS.
7. If the unit mentioned in the import file is different form layout unit then a *warning message* (trans) is displayed.
8. If the selected ODB++ file is too large, a Confirmation dialog box opens.
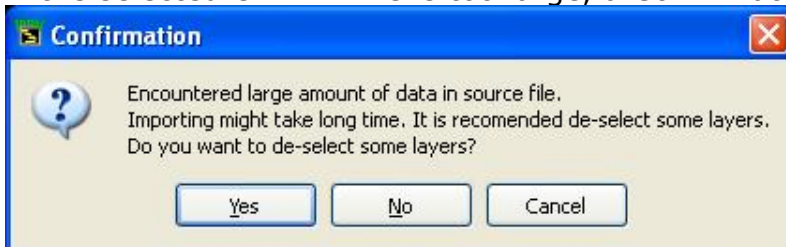
9. Select **Yes**. *Layers* dialog box opens. You can check or un-check layers. If you select **No**, all layers are imported.
10. Click **Cancel** to stop import.
11. A *Status* window opens displaying the import information.
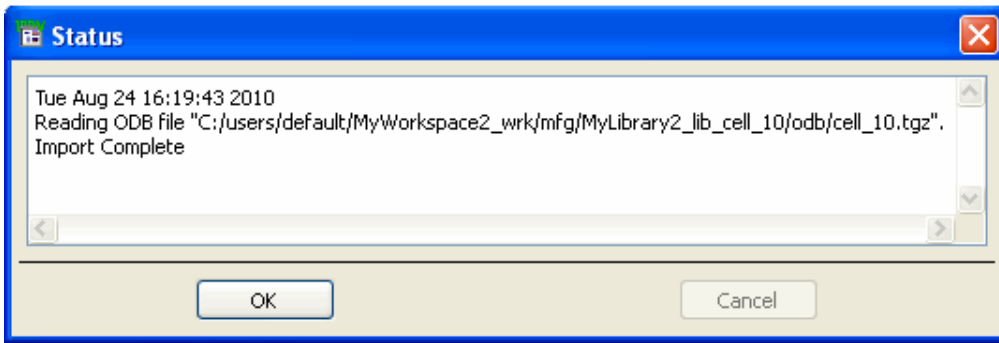
> **ⓘ Note**
> Status Window displays warning occurred during ODB__ import. The layers on which drill is applicable are also mentioned in status dialog. Additionally,the warning in status window is saved in *<Workspace Dir>/readodb.log*

Import ODB++ Options

The *More Options* dialog enables you to select layers and ODB Options to control the import of ODB++ files. An ODB++ file and layers file must be selected in the main import dialog for *More Options* dialog to open.

ODB++ Options
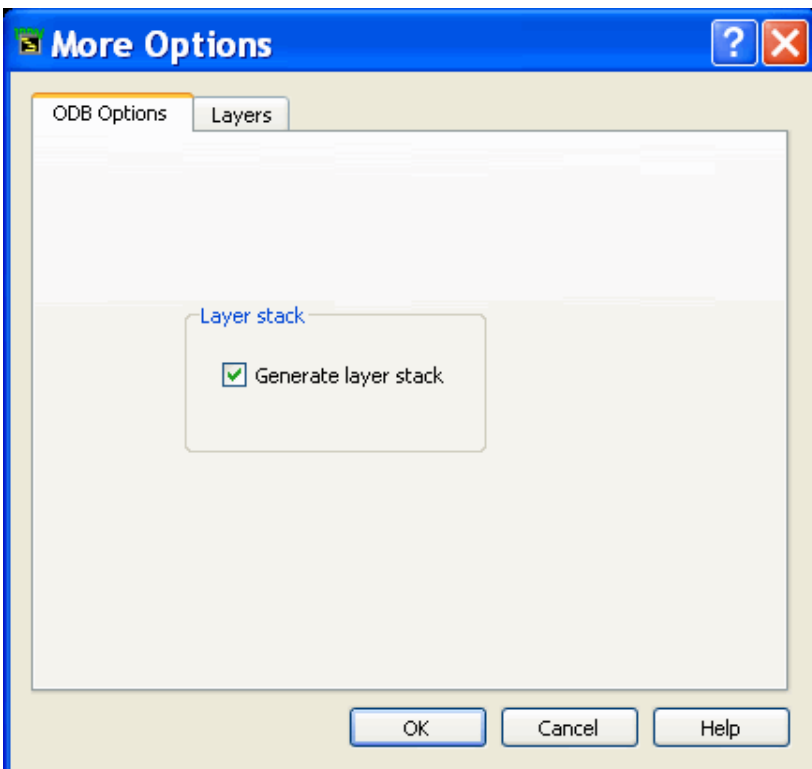
By default Layer stack information is imported. To import only layers un-check the **Generate layer stack** from *ODB Options* tab
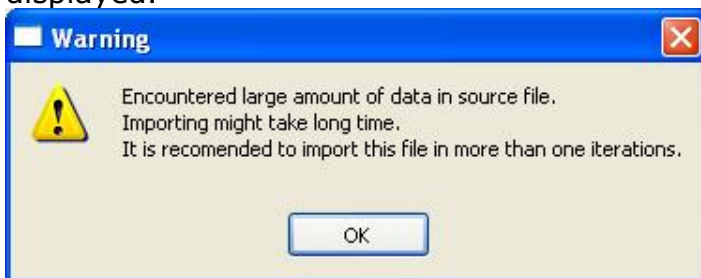
Layers
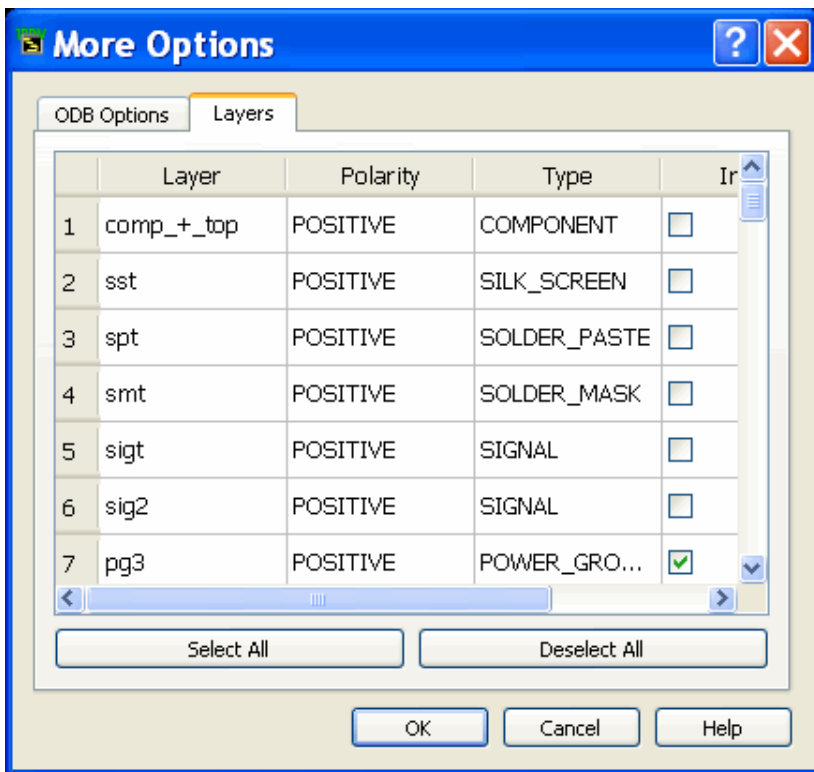
The *Layers* tab displays a list all the layers in ODB++ file. By default all layers are imported. To import selective layers, un-check the layers from **Import** column under Layers tab. **Polarity** and **Type** are non-editable fields and are displayed only for information purpose.

Limitations and Considerations

1. To import ODB++ data reliably in ADS, ensure that the ODB++ data is exported with finer precision than the original design precision by increasing the layout precision before doing export.
2. When you click **More Options** button and if the ODB++ file is too large, a warning is displayed.

3. Text and other properties for components are not imported while importing the component layer.
4. Only geometrical information will be imported. Net name and other net properties will be ignored.

Exporting ADS Design to ODB++ Files

ADS exports ADS layouts into a single compressed archive with the *.tgz* file extension. You may specify the output path and file name, but the default is *<Workspace Dir>/mfg/<Design Name>/odb/<Design Name>.tgz*. These translated files can be viewed

with any third party viewer that supports ODB++ (e.g. Valor Universal Viewer from Valor, AutoVue from Cimmetry, ViewMate from Pentalogix) to verify the exported design.

To export ODB++ files:

1. From the layout window containing your design, choose **File > Export**.
   The *Export* dialog box opens.
2. In the *File type* drop-down list, select **ODB++**.
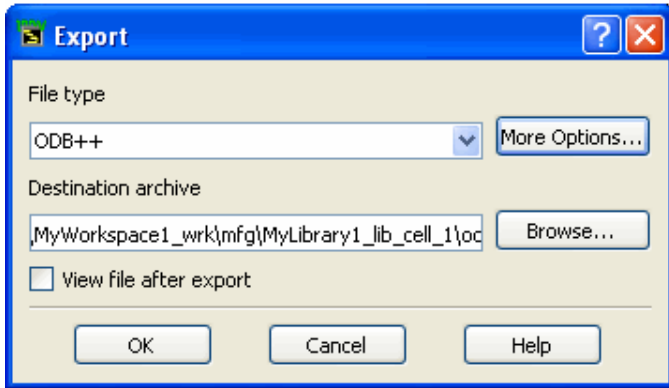
3. To view and/or define export options, click **More Options**.
4. Set the desired options.
   For available options, see *Export ODB++ Options*.
5. Click **OK** to save your settings or **Cancel** to retain the default settings.
6. In the *Destination Archive* field, accept the default directory, or click **Browse** to locate a new destination directory.
   The archive directory is defined as *<Workspace Dir>/mfg/<Design Name>/odb/* where:
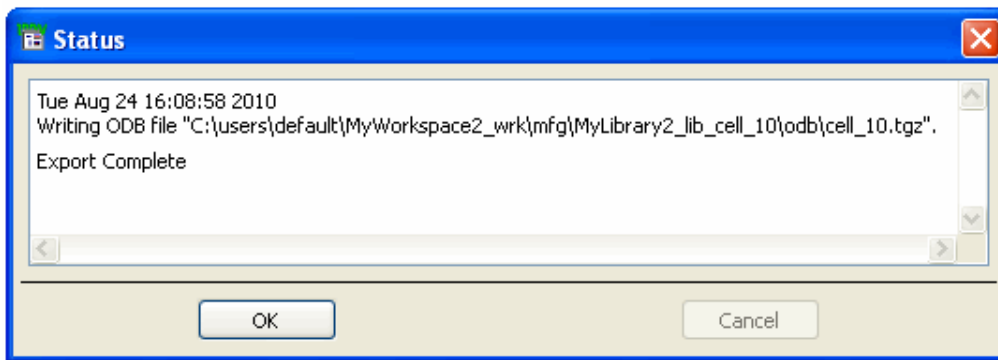   *<Workspace Dir>* is the workspace directory
   <Design Name> is directory named with the design name

   > ℹ **Note**
   > The check box option *View file after export* is not applicable for the ODB_+ file type.
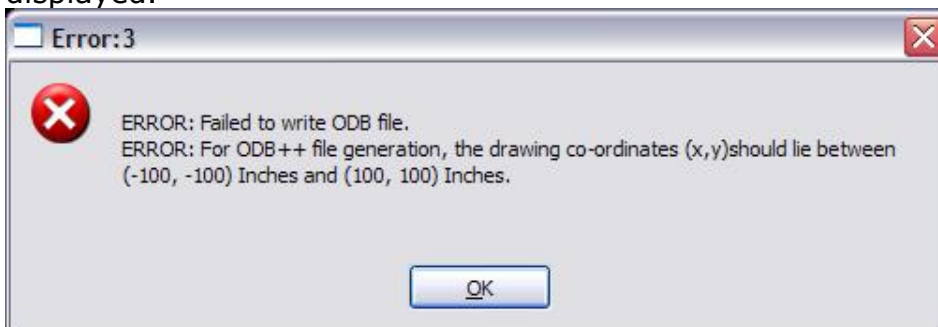
7. Click **OK** to export the ADS layout design to an ODB++ archive.
8. If you attempt to write to a folder which is not empty and/or previously contains ODB++ data, a confirmation dialog is displayed.
9. Choose **Yes** to overwrite with the current export data. Otherwise, choose **No** to return to the *Export* dialog.

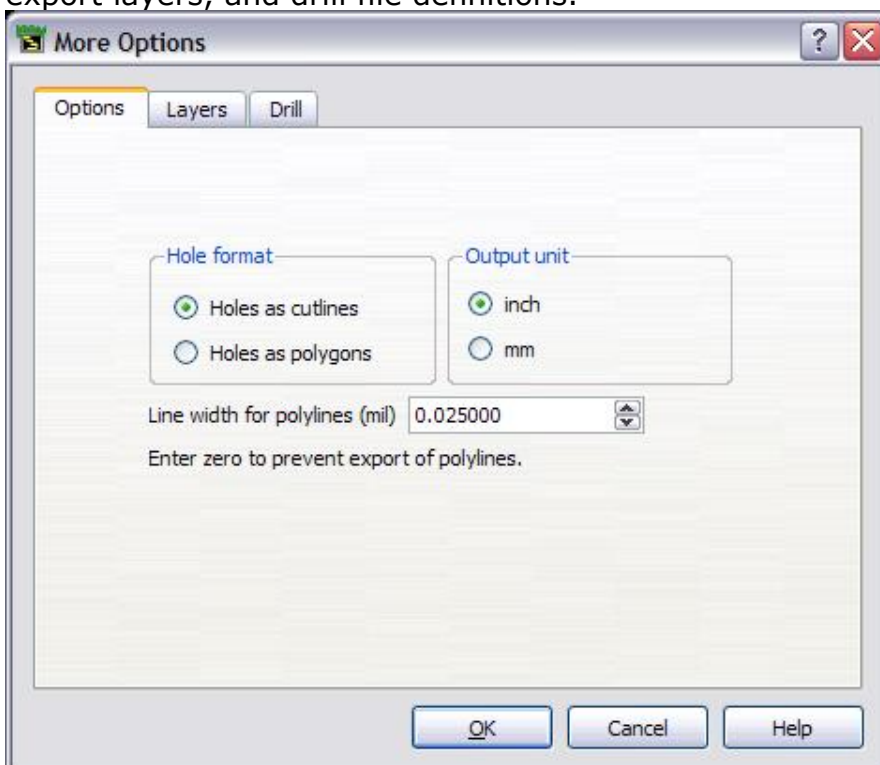A *Status* window opens displaying the export information, if successful.

Status

```
Tue Aug 24 16:08:58 2010
Writing ODB file "C:\users\default\MyWorkspace2_wrk\mfg\MyLibrary2_lib_cell_10\odb\cell_10.tgz".

Export Complete
```

OK   Cancel

10. If the export is unsuccessful due to out of range co-ordinates, an *Error* dialog is displayed.

Error:3

ERROR: Failed to write ODB file.
ERROR: For ODB++ file generation, the drawing co-ordinates (x,y)should lie between (-100, -100) Inches and (100, 100) Inches.

OK

11. To save the log message, select text, copy, and then paste into any text editor.

Export ODB++ Options

The *More Options* dialog enables you to set hole format, output units, path geometry, export layers, and drill file definitions.

More Options

Options   Layers   Drill

Hole format
- ⦿ Holes as cutlines
- ○ Holes as polygons

Output unit
- ⦿ inch
- ○ mm

Line width for polylines (mil) 0.025000

Enter zero to prevent export of polylines.

OK   Cancel   Help

Options

Hole format

In ADS, when a hole is created in a geometrical figure (like polygon), a cutline is introduced. This is a false edge connecting the outer boundary of the polygon with the inner boundary. This polygon is actually a single re-entrant polygon. When you select **Holes as cutlines**, this re-entrant polygon is translated to ODB++ as-is. The default is **Holes as cutlines**.

Holes as cutlines

Select **Holes as polygons** to remove the false edge from the polygon.
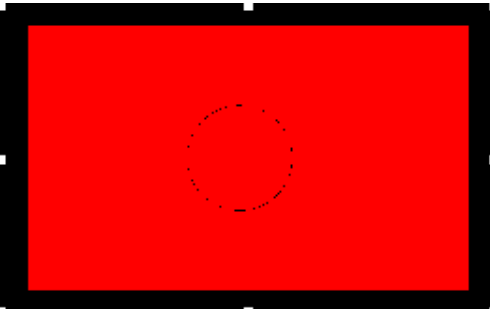
Holes as polygons

Select **Holes as polygons** to export holes as filled elements. The polygon will appear to have no holes when this option is selected.

Unit

Available units are inch or mm. Inch is the default.

Line width for polylines

Polylines in ADS have zero width. Use this option to provide a width for drawing this polyline in the ODB++ format.

If the export of polylines and arcs (both zero width entities) is not required, then set the width to zero.
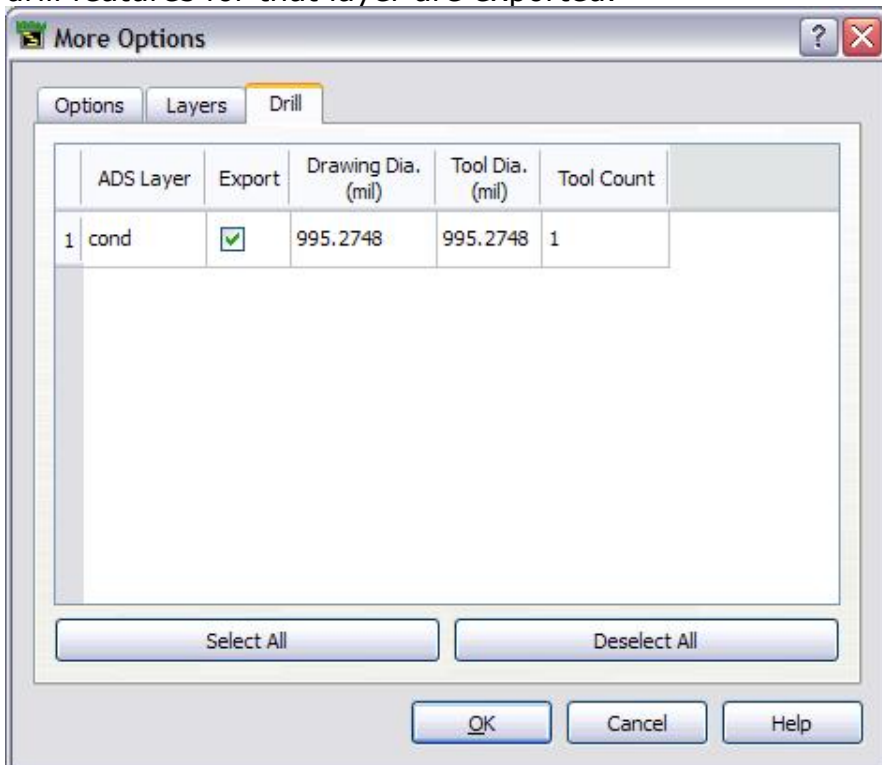
Layers

All the layers used in the design are displayed on the *Layers* tab. By default, all layers are displayed and selected for export. You can avoid exporting a specific layer by deselecting the checkbox in the *Export* column for that layer.
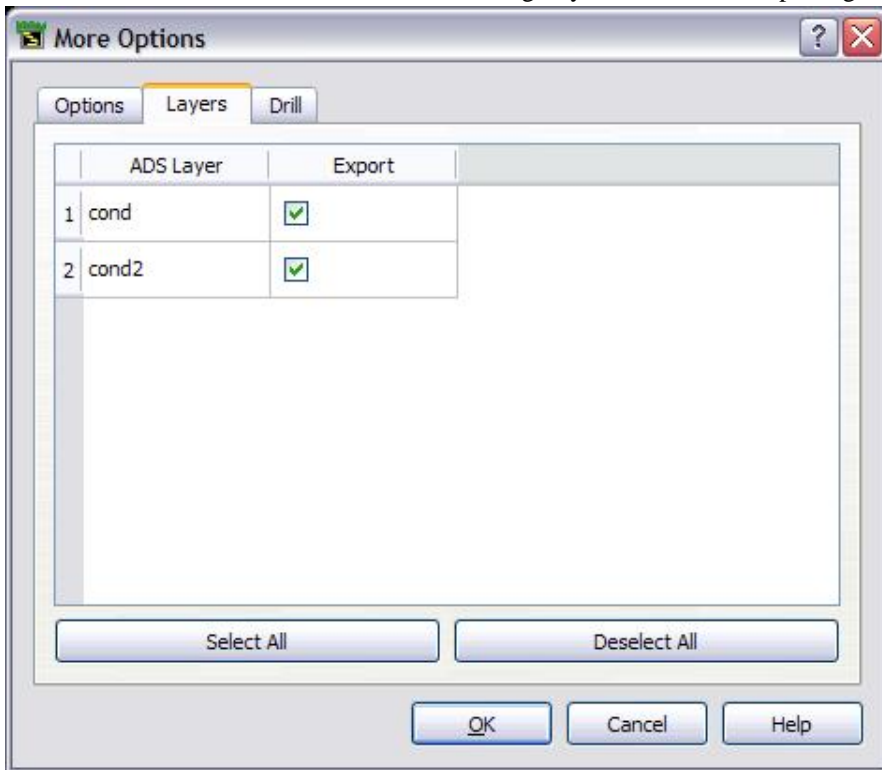
Drill

Drill files are configured from the *Drill* tab. While exporting a layout design in ODB++ format, layers names containing drill features are appended with *_drl*. Only the selected drill features for that layer are exported.

Select the check box to designate the drill layer to export. When a layer is selected, all the tools on that layer are automatically selected. By default, no drill layer is selected.
Tool Dia. (Diameter) is an editable field. By default the *Tool Diameter* value equals the *Drawing Diameter* value. If Tool Diameter is set to 0.0, the Tool Diameter will be equal to
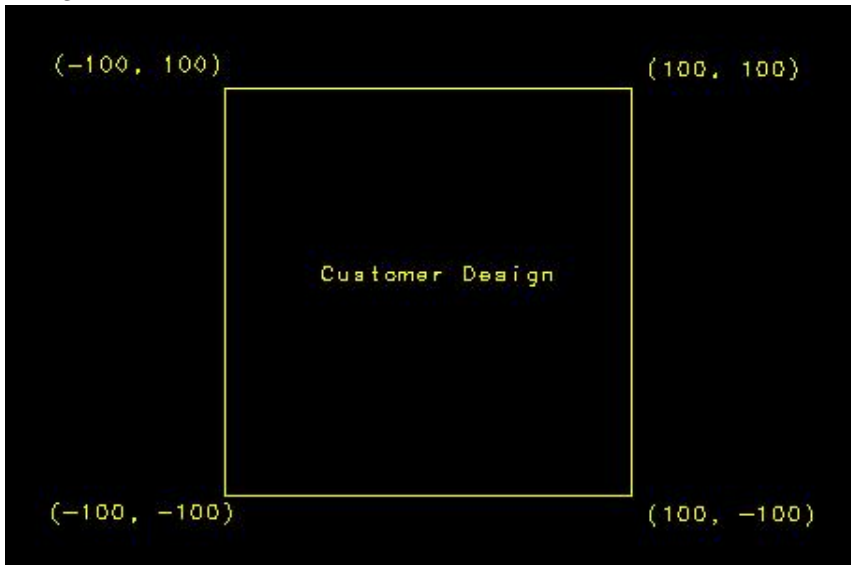
the Drawing Diameter in the output drill file.

Limitations and Considerations

> **ⓘ Note**
> The tool only exports the geometrical data. No netlist data, layer stack, or component data is exported.

- Designs beyond the (x,y) co-ordinates of (-100, -100) and (100, 100) will not be exported and an error message will be issued. Ensure that your design follows this limit.

- It is recommended to create the ADS design using inch or mm as the unit and set the appropriate resolution.
- If a layer does not have any via (circles), then it will not be displayed in the *Drill* tab and no drill file will be generated for that layer.
- One drill file will be generated for each layer having at least one drill feature.
- Drill files will be created and packaged in the *Destination archive* selected in the *Export* dialog.
- Drill layer name is the same as the ADS layer name with extension *_drl* appended to it. For example, drill file for the layer *cond* will be *cond_drl*.