



ADS 2008  
January 2008  
Importing and Exporting Designs

# Advanced Design System 2008

© Agilent Technologies, Inc. 2000-2008

395 Page Mill Road, Palo Alto, CA 94304 U.S.A.

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

## Acknowledgments

Mentor Graphics is a trademark of Mentor Graphics Corporation in the U.S. and other countries. Microsoft®, Windows®, MS Windows®, Windows NT®, and MS-DOS® are U.S. registered trademarks of Microsoft Corporation. Pentium® is a U.S. registered trademark of Intel Corporation. PostScript® and Acrobat® are trademarks of Adobe Systems Incorporated. UNIX® is a registered trademark of the Open Group. Java™ is a U.S. trademark of Sun Microsystems, Inc. SystemC® is a registered trademark of Open SystemC Initiative, Inc. in the United States and other countries and is used with permission. 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.

**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 manual 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** If software is for use in the performance of a U.S. Government prime contract or subcontract, Software is delivered and licensed as "Commercial computer software" as defined in DFAR 252.227-7014 (June 1995), or as a "commercial item" as defined in FAR 2.101(a) or as "Restricted computer software" as defined in FAR 52.227-19 (June 1987) or any equivalent agency regulation or contract clause. Use, duplication or disclosure of Software is subject to Agilent Technologies' standard commercial license terms, and non-DOD Departments and Agencies of the U.S. Government will receive no greater than Restricted Rights as defined in FAR 52.227-19(c)(1-2) (June 1987). U.S. Government users will receive no greater than Limited Rights as defined in FAR 52.227-14 (June 1987) or DFAR 252.227-7015 (b)(2) (November 1995), as applicable in any technical data.

## Contents

- About Importing and Exporting Designs
  - Layout Export Considerations
    - Available File Formats
- Importing and Exporting Schematic and Layout Designs
  - Importing a Schematic
  - Exporting a Schematic
  - Importing a Layout
  - Exporting a Layout
  - Defining Layers
- Using the Pre-Production Editor
  - File Menu
  - Edit Menu
  - View Menu
  - Window Menu
  - Viewing Files in the Pre-Production Editor During Export
  - Viewing Files in the Pre-Production Editor During Import
  - Editing Files in the Pre-Production Editor
- DXF/DWG Translator
  - Importing DXF/DWG Files to ADS
    - Import DXF/DWG Options
    - Mapping DXF to ADS
  - Exporting an ADS Design to DXF/DWG Files
    - Export DXF/DWG Options
    - Mapping ADS to DXF (hierarchical)
- EGS Archive Files
  - Importing EGS Archive Files
    - Import EGS Archive Options
    - Mapping EGS Archive to ADS
  - Exporting EGS Archive Files
    - Export EGS Archive Options
    - Mapping ADS to EGS Archive
- EGS Generate Files
  - Importing EGS Generate Files
    - Import EGS Generate Options
    - Mapping EGS Generate to ADS
  - Exporting EGS Generate Files
    - Export EGS Generate Options
    - Mapping ADS to EGS Generate
- GDSII Stream File Translator
  - Importing GDSII Files
    - Guidelines/Considerations
    - Filenames Versus Instance Names
    - Import GDSII Options
  - Exporting GDSII Files
    - Guidelines/Considerations
    - Export GDSII Options

## Advanced Design System 2008

- Gerber Translator
  - ◦ Gerber Command Format
  - ◦ Importing Gerber Files
  - ◦ Exporting Gerber Files
    - - Gerber File Options
    - - Gerber Layer Options
    - - Limitations and Considerations
  - ◦ Exporting Drill Files
    - - Considerations:
  - ◦ Exporting Gerber Files Using MTools
    - - Gerber File Options
    - - Translation Settings
    - - Edit Apertures
    - - View Mask
    - - View Gerber
    - - Export Gerber Options
    - - Limitations and Considerations
  - ◦ Using the Gerber Viewer
    - - Criteria for Viewing Gerber Files
    - - Launching the Viewer from a Layout Window
    - - Launching the Viewer During File Export
    - - Loading a File to View
  - ◦ Gerber Viewer Menu Options
    - - Gerber Settings
    - - Snap Settings
    - - Gerber Merge
    - - Layer Options
    - - Aperture
    - - Gerber Viewer Keyboard Commands
  - ◦ Configuring the Gerber Translator for Photoplotters
    - - Vector Plotter Configuration
    - - FIRE 9000 Photoplotter Configuration
    - - RS274X Output Configuration
  - Creating an Excellon Drill File from an ADS Layout Using MTools
- HPGL/2 Files
  - ◦ Exporting HPGL/2 Graphics Files
    - - Export HPGL/2 Options
- IFF Files
  - ◦ Importing IFF Files
    - - Import IFF Options
  - ◦ Exporting IFF Files
    - - Export IFF Options
- IGES Translator
  - ◦ Translator Description
  - ◦ Importing IGES Files
    - - Import IGES Options
  - ◦ Exporting IGES Files
    - - Export IGES Options
- Mask Files
  - ◦ Importing Mask Graphics Files

- - Import MASK Options
- Exporting Mask Graphics Files
- - Export MASK Options
- MGC/PCB Files
- ◦ Exporting MGC/PCB Files
- - Export MGC/PCB Options

## 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 Window
DXF/DWG (hierarchical) (.dxf, .dwg)	-	-	Import/Export
DXF/DWG (flattened) (.dxf, .dwg)	-	-	Export
EGS Archive Format (_a)	-	-	Import/Export
EGS Generate Format (_g)	-	Import	Import/Export
GDSII Stream Format (.gds)	-	-	Import/Export
Gerber (.gbr)	-	-	Export
Gerber Viewer (.msk, .gbr)	-	-	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/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.

## Advanced Design System 2008

Steps for importing and exporting these files are outlined in [Importing and Exporting Schematic and Layout Designs](#).

### DXF/DWG

The Drawing Exchange Format (DXF) was developed by Autodesk for its AutoCAD 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](#).

### 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](#) and [EGS Generate Files](#).

### GDSII Stream

GDSII Stream Format (Calma) is an industry standard for translating final mask data to foundries. The Advanced

## Advanced Design System 2008

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](#).

### Gerber

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](#).

### Gerber Viewer

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](#)).

You can use the Gerber Viewer to view Gerber or mask files to help verify the correctness of your data. Also, you can use the Gerber Viewer to configure mask file data for photoplotting.

### 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](#).

### 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](#).

### 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](#).

### 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](#).

### 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.

### 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](#).

## Importing and Exporting Schematic and Layout Designs

This chapter 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](#).

 **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](#).


## Importing a Schematic

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

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 chapter.


To import a schematic design:

1. Open a project and choose File > Import. The Import dialog box appears.
2. Select the appropriate file format from the File Type drop-down list.
3. To define options for the imported file, click the More Options button. The Import 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 project directory). When you make changes in the options dialog box and click OK, a local copy of the options file is written to the current project directory.

4. To specify the path and filename of the file you want to import, click Browse.
5. Double-click as needed to locate the directory containing the design. By default, all files are listed that have the file suffix appropriate for the chosen file format.
6. Select the 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).
7. 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 filename to determine the new design name.

8. 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](#).

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](#).

To export a schematic design:

1. Open your design and choose File > Export. The Export dialog box appears.
2. Select IFF file format from the File Type drop-down list.
3. To set export options, click the More Options button. The Export 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

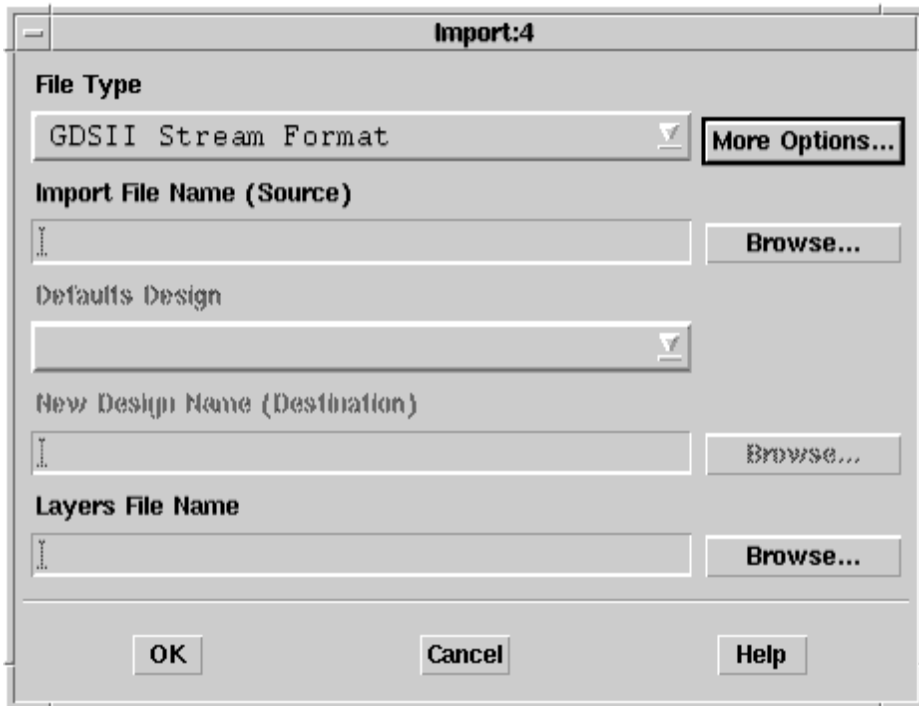
For available file formats that you can import from an ADS Layout window, refer to [Available Formats and Import/Export Options](#).

The procedure for importing each format is the same, but the available options differ. For options relating to a

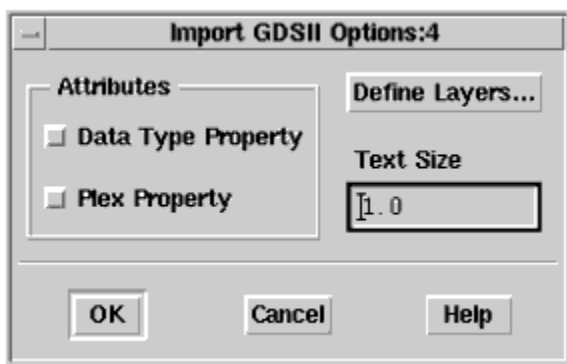
particular file format, refer to the associated chapter.

To import a layout file:

1. In the Advanced Design System layout window, choose File > Import to open the Import dialog box.



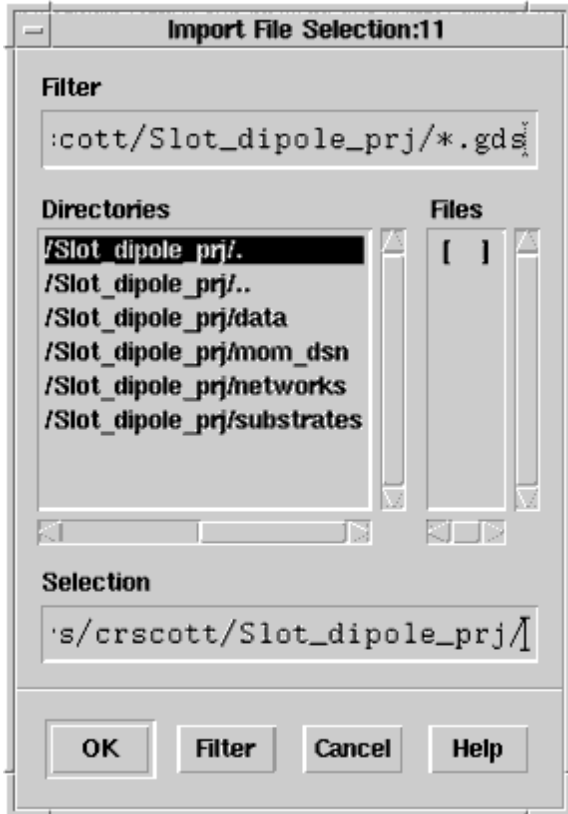
2. In the File Type field, click the arrow in the right-hand corner to display a drop-down list of available formats. Select the format you want to import.
3. Click the More Options button to define the options for your selected format.



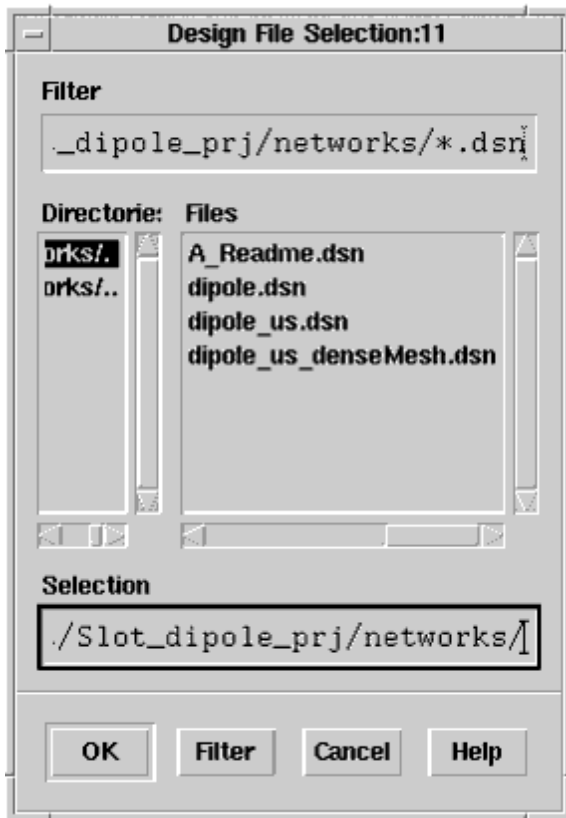
**Note**  
Each format has its own set of import options. Refer to the individual translator topic.

4. Click Define Layers to set the layer options. For information on the Layer Editor, see [Defining Layers](#). Close the Layer Editor dialog box.
5. In the Import Options dialog box, click OK to save your settings and return to the Import dialog box.

- In the Import File Name (Source) field enter the full path of the source file. Alternatively, click Browse to open the Import File Selection dialog box and locate the file you want. After locating the file, click OK to accept the selection and return to the Import dialog box. The appropriate suffix (.dsn) is appended to the filename automatically.



- 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 Design File Selection dialog box and locate the destination path. After locating the file, click OK to accept the selection and return to the Import dialog box. The appropriate suffix is appended to the filter automatically.

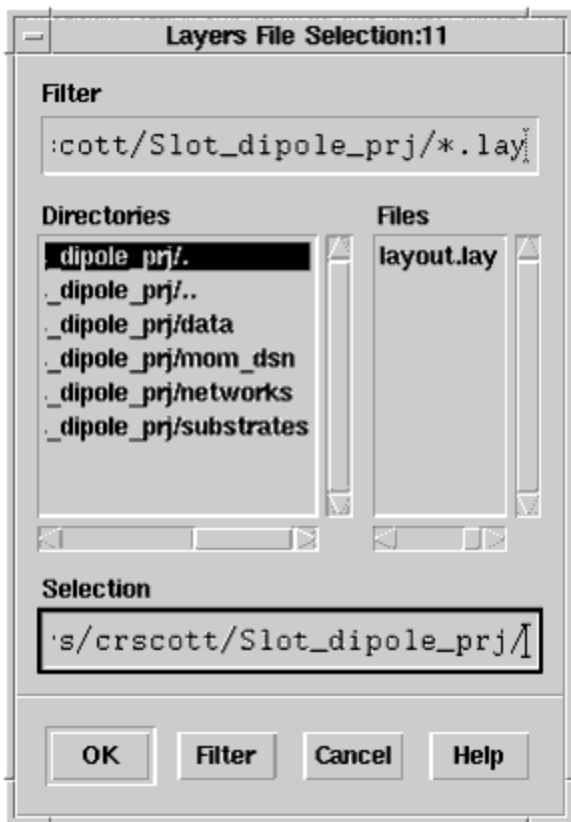


**Note**  
New Design Name (Destination) cannot be specified for EGS Archive Format, GDSII Stream Format, IFF, or IGES files.

- In the Layers File Name field, enter the full path and name of the layers file to be referenced during import. If the name you enter does not exist, a .lay file will be created using the information defined in the Layer Editor.

**Note**  
If the import results in changes to the layers that you have defined, then those updates are written to the custom layers file. If the custom layers file is defined as non-writable, then a warning is generated.

Alternatively, you can click Browse to open the Layers File Selection box and locate the path and file name. After locating the file, click OK to accept the selection and return to the Import dialog box. The appropriate suffix (.lay) is appended to the filename automatically.



The specified layers file contains the layer definition from the file you are importing.

**Note**  
Layers File Name cannot be specified for HPGL/2, IFF, or Mask files.

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

## Exporting a Layout

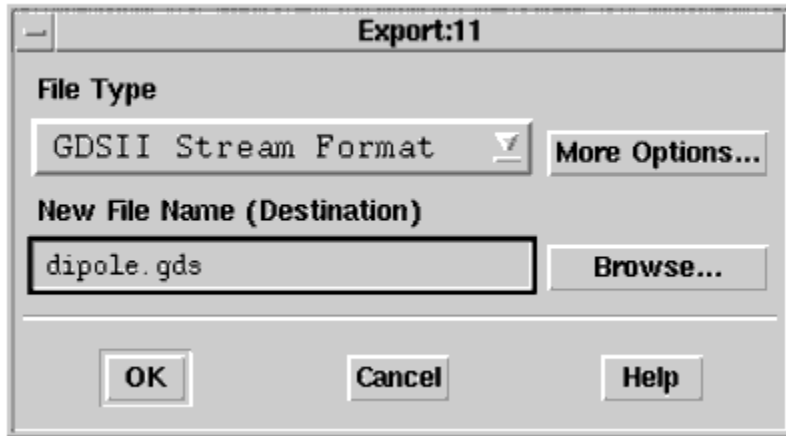
For available file formats that you can export from an ADS Layout window, refer to [Available Formats](#).

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 chapter.

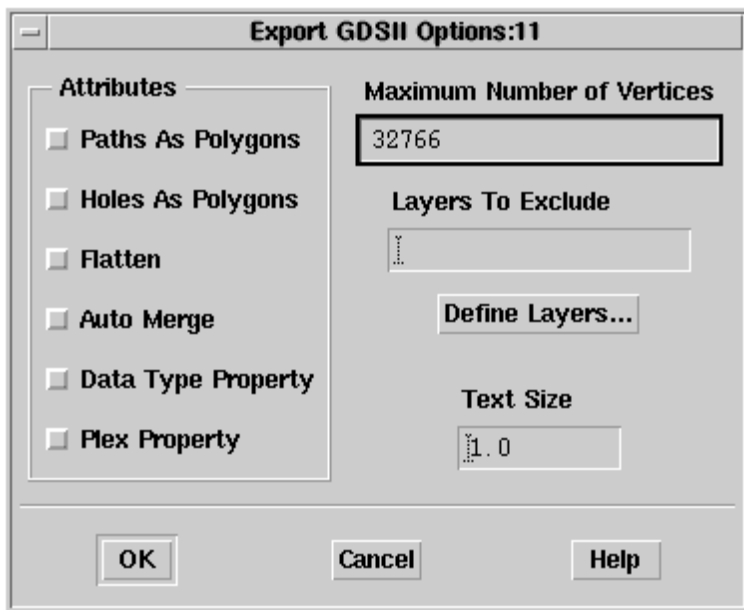
**Note**  
Gerber Viewer appears as an export file option. It is not a file format, but is placed on the export file menu so you can open the Gerber Viewer at any time. In addition, you can access the Viewer during a DXF or Gerber export (see [Using the Gerber Viewer](#)).

To export a layout file:

1. In the Advanced Design System layout window containing your design, choose File > Export. The Export dialog box appears.



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 export.
3. Click the More Options button to define the options for your selected format.

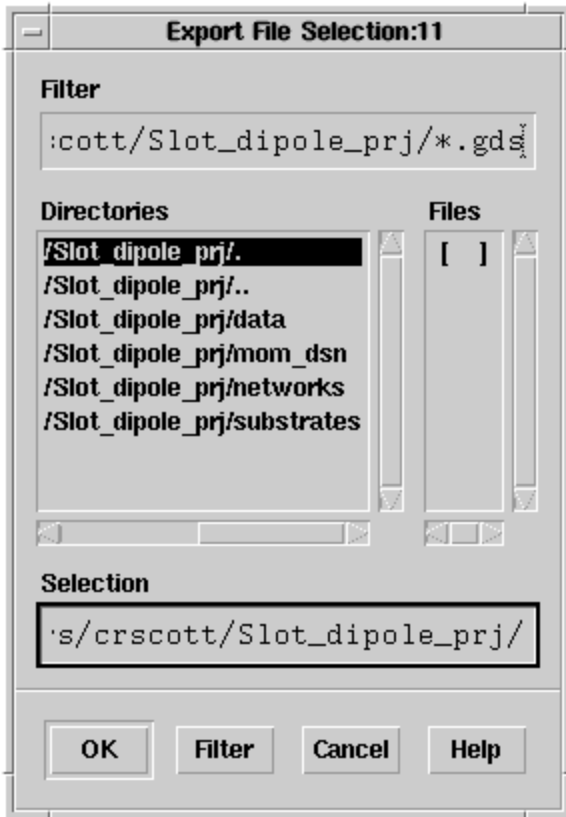


**Note**  
Each format has its own set of import options. Refer to the appropriate chapter in this book.

4. Click Define Layers to set the layer options. For information on the Layer Editor, see [Defining Layers](#).

**Note**  
ADS layer numbers are mapped to DXF layer names. You can modify layer mappings and DXF layer names by editing the DXF Layer Name column in the Advanced tab section of the Layer Editor.

5. In the Export Options dialog box, click OK to save your settings and return to the Export dialog box.
6. In the New File Name (Destination) field enter the full path of the destination file. Alternatively, click Browse to open the Export File Selection dialog box. After locating the destination directory, click OK to accept the selection and return to the Export dialog box. Enter a filename. The appropriate suffix is appended to the filename automatically.

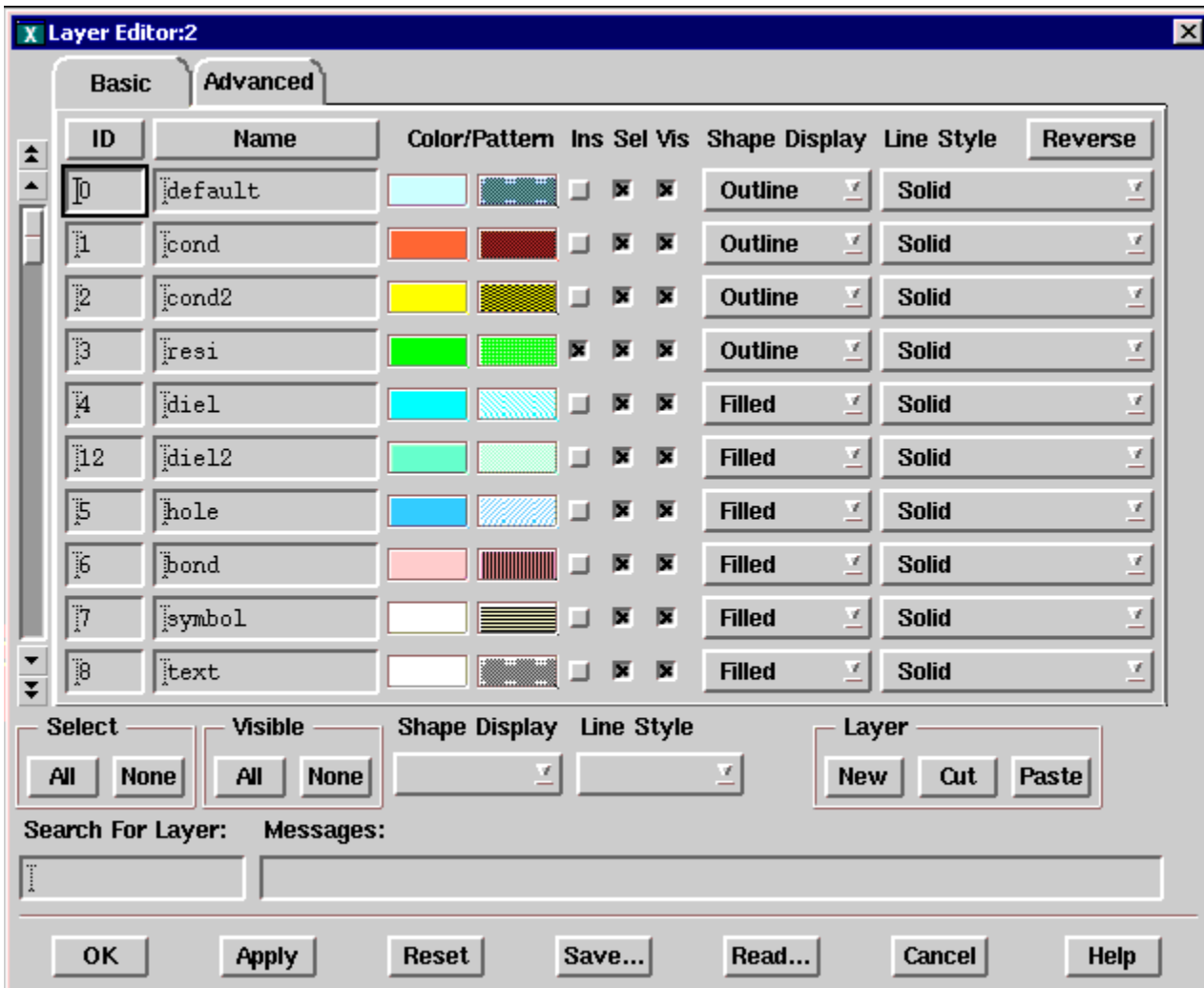


**Note**  
 New File Name (Destination) cannot be specified for MGC/PCB files. MGC/PCB files are written to the same predetermined location each and every time. For more information on this file type, see [MGC/PCB Files](#).

7. 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 chapter in this book.

## Defining Layers

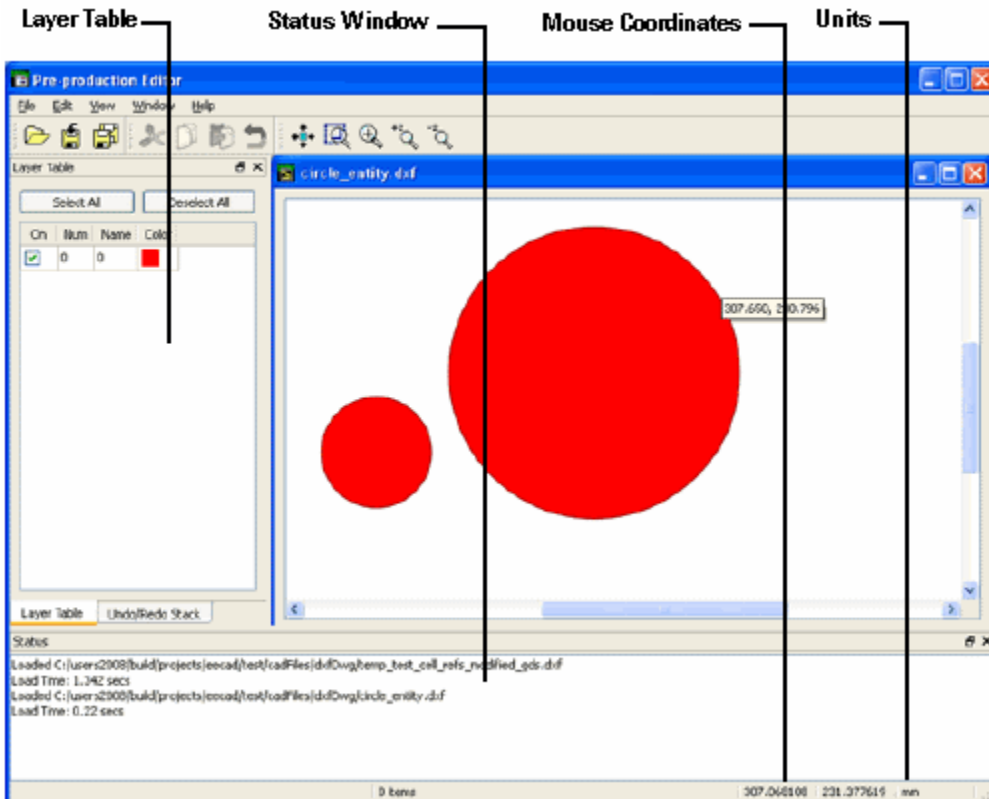
Using the Advanced Design System Layer Editor (More Options > Define Layers) in the Import or Export dialog box, you can define design layers and add, delete, or rearrange layer priority. The contents of this dialog box change to reflect the layer assignments associated with the open design. For information on the defining layers using the ADS Layer Editor, refer to [Setting Layout Options](#).



## Using the Pre-Production Editor

ADS includes an editor that can display DXF/DWG designs. 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.



## 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.

## Edit Menu

The Edit menu contains common and specialized editing commands.

### Enable Editing

Enables editing commands to be performed on the current design. Deselect 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

## Advanced Design System 2008

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.

 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.

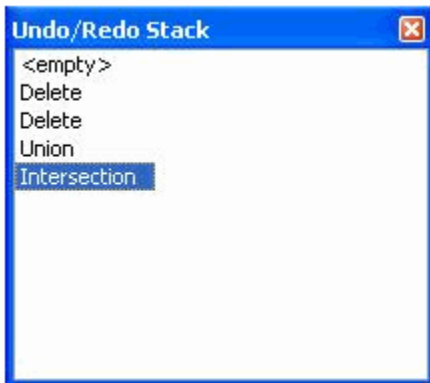
#### Measure



The Measure dialog allows you to measure lengths.

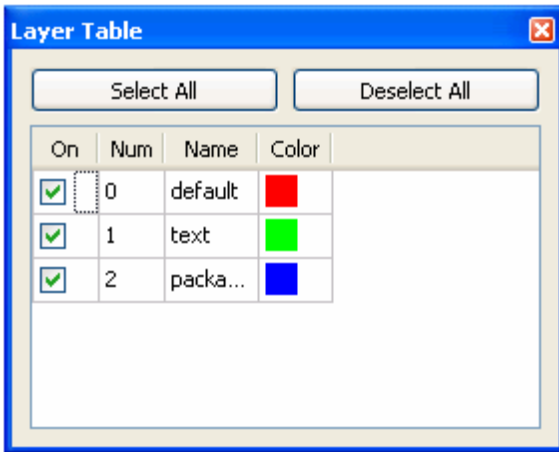
- Absolute X, Y displays the absolute coordinates of the point of mouse click.
- Delta X, Y gives the relative X and Y coordinates from previous mouse click.
- Angle gives the angle with respect to a line horizontal to the X axis.
- Cumulative distance gives the total length from starting point to current 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 file during export from ADS:

1. Select DXF/DWG 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 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 file during import to ADS:

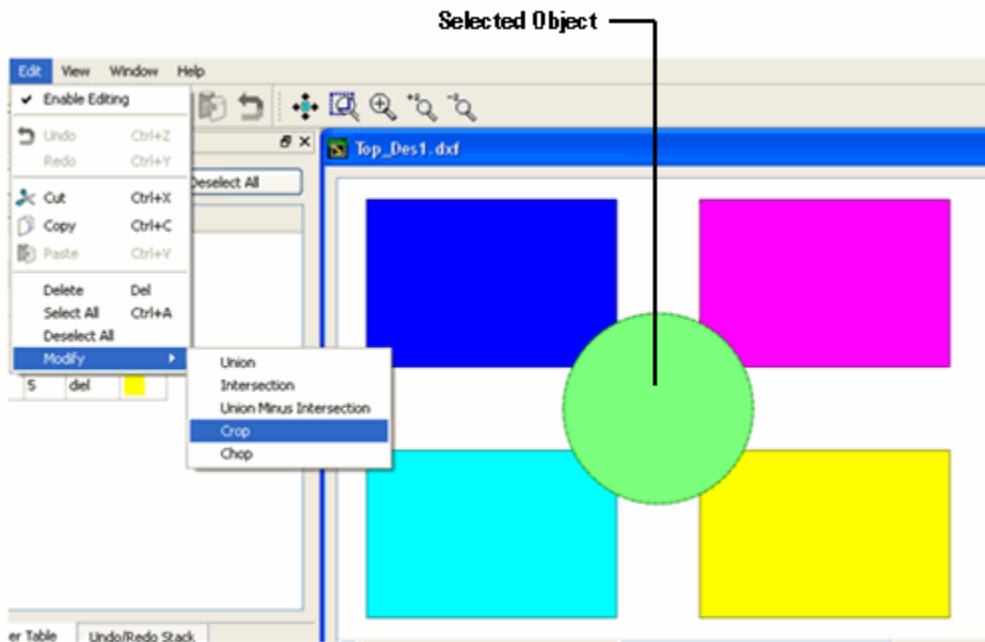
1. Select DXF/DWG from File type drop-down list in Import dialog.
2. Click Preview to view the file in the Pre-Production Editor.

## Editing Files in the Pre-Production Editor

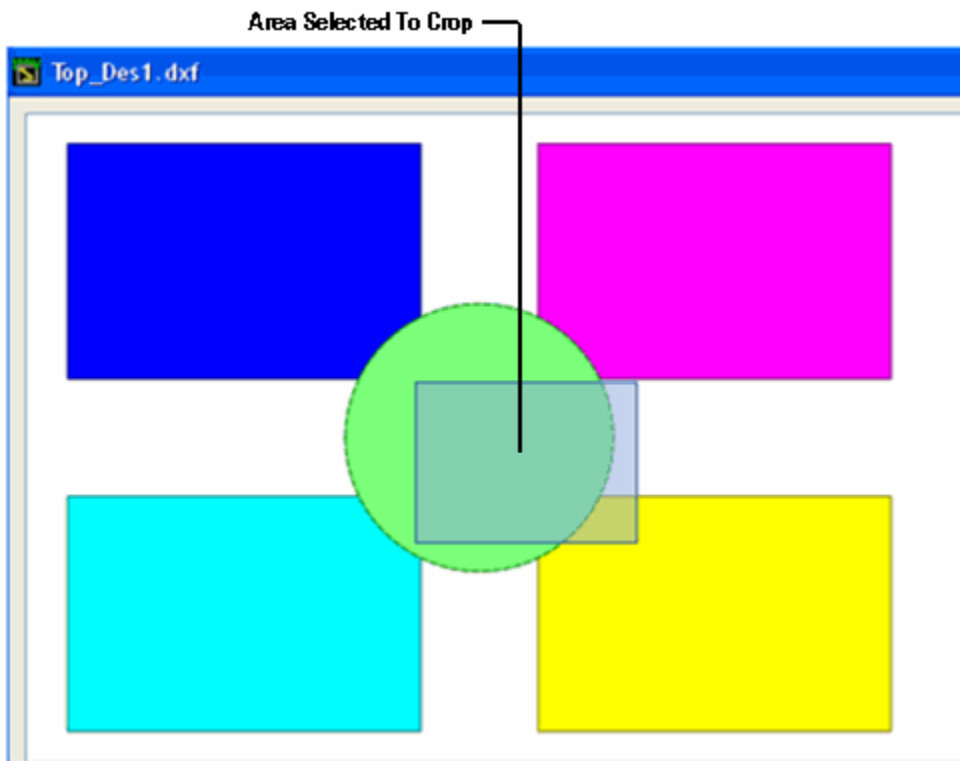
Using the Pre-Production Editor you can cut, copy, paste and modify designs. From the Modify menu, you can perform editing functions like union, intersection, union minus intersection, crop, and chop.

To edit using the Pre-Production Editor:

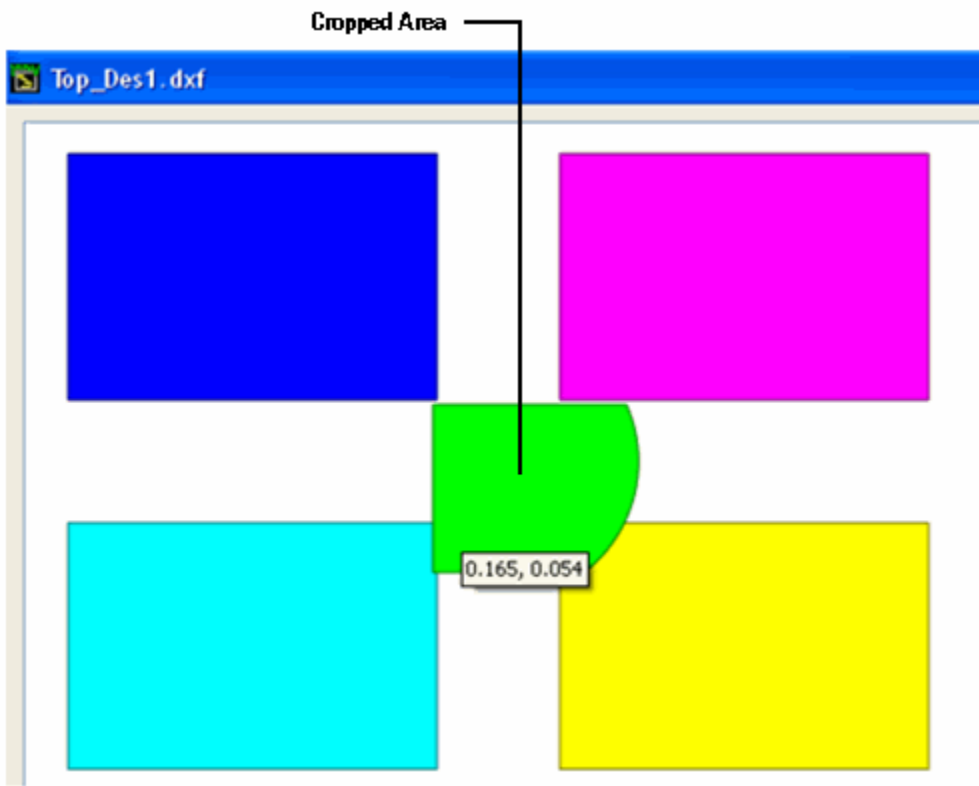
1. Open a design in the Pre-Production Editor as described above.
2. Click Edit > Edit Enable.
  - To cut a figure, select the figure and then click Edit > Cut.
  - To copy a figure, select the figure and then click Edit > Copy.
3. To crop or chop a figure:
  - a. Select the figure.



- b. Click Crop or Chop.
- c. Mark the area to crop/chop using mouse.



d. Release the mouse to crop/chop the selected area.



4. Click Delete to delete selected item.

5. Click Undo to revoke last Edit command.

## 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. 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.

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](#).

## Importing DXF/DWG Files to ADS

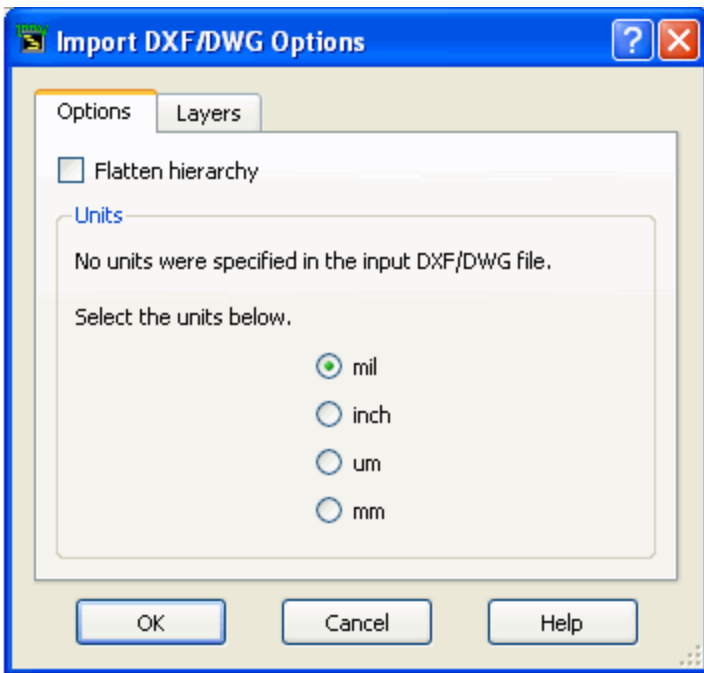
The procedure for importing each format is generally the same, however the available options differ. For a step-by-step tutorial on importing a layout file, refer to [Importing a Layout](#). For specific import options related to importing DXF/DWG files, refer to [Import DXF/DWG Options](#). For information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

### Import DXF/DWG Options

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

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

### Options



### Flatten hierarchy

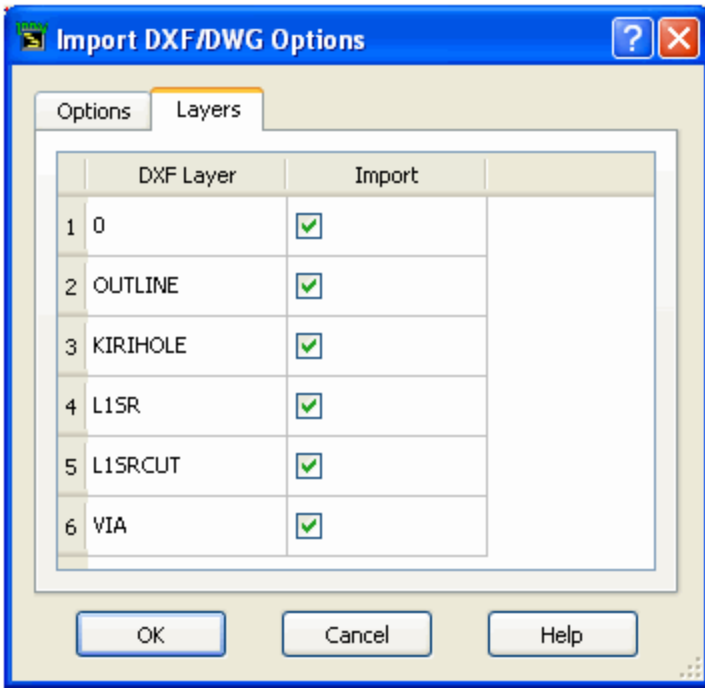
Select Flatten hierarchy to remove any hierarchy that may be present in the DXF/DWG selected. Default setting retains the hierarchy.

Note: If you select a DXF/DWG that contains more than 50 component designs this option gets automatically enabled. This is 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.

Layers



The Layers 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.

Mapping DXF to ADS

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

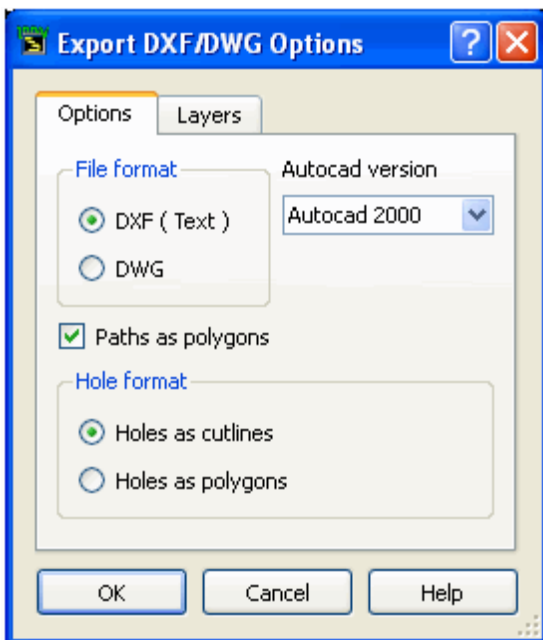
To export DXF/DWG files:

1. Follow the steps as outlined in [Exporting a Layout](#). 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 Export DXF/DWG Options dialog box enables you to set file format, Autocad version, path geometry, and hole format.



### Options

## Advanced Design System 2008

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 setting.

#### 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.

### 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
polyline	open LWPOLYLINE
path	open LWPOLYLINE with width
path with PathAsPolygons	closed LWPOLYLINE
wire	open LWPOLYLINE
circle	circle
text	text
rectangle	closed LWPOLYLINE
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 Name 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.

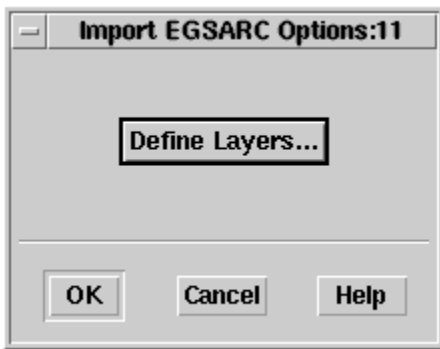
## Importing EGS Archive Files

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](#). 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

File > Import > EGS Archive Format > More Options

Other than layer characteristics, there are no definable options available for importing files in the EGS Archive format.



The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

**Note**  
 ADS does not support the character '\$' in layer names. Translation will fail if the import file contains this character in the layer names.

### 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

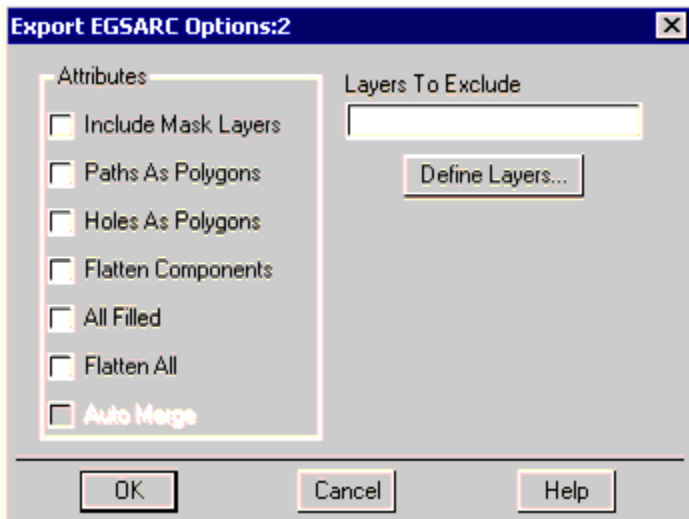
### Exporting EGS Archive Files

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](#). 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

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.
- The 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.

## Define Layers

The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

## 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 Name 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.

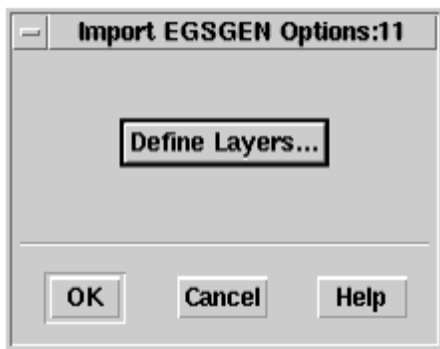
## Importing EGS Generate Files

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](#). 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

File > Import > EGS Generate Format > More Options

Other than layer characteristics, there are no definable options available for importing files in the EGS Generate format.



The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

**Note**  
 ADS does not support the character ` \$ ' in layer names. Translation will fail if the import file contains this character in the layer names.

### 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

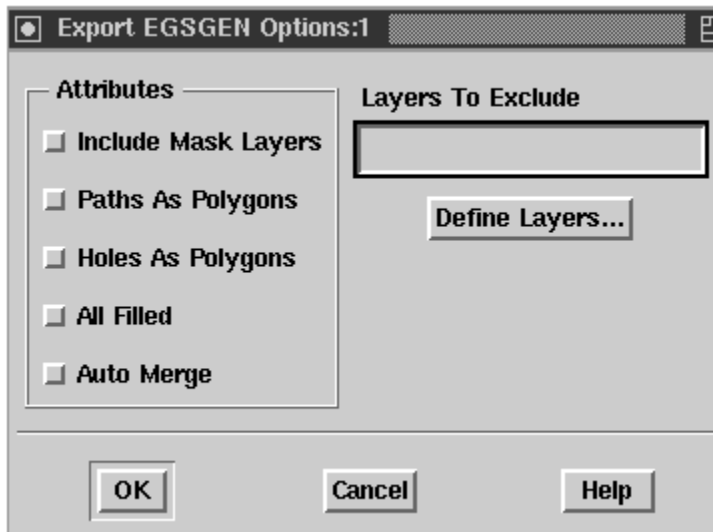
Rectangle with width = 0	Rectangle
Rectangle with width > 0	Polygon with cutline on longest segment
Circle with width = 0	Circle
Circle with width > 0	Polygon with cutline to the right
Arc with width = 0	Polyline
Arc with width > 0	Polygon

## 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](#). 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

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.

### Define Layers

The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

### 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

## GDSII Stream File Translator

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.

GDSII has some limitations that may affect your layout, including:

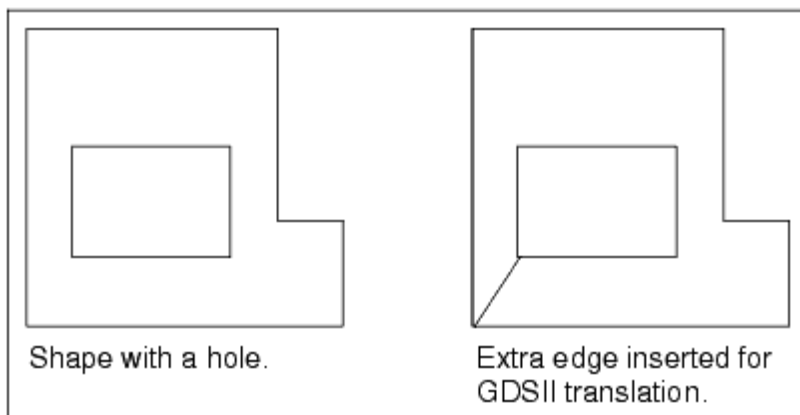
- No support for arcs or circles
- No support for holes or empty regions
- 32-character name limitation
- Limit of 255 mask layers

## Curved Element Limitation

Although the Advanced Design System has options for re-mapping layer numbers and structure names, it does not have the full ability to overcome the GDSII limitations regarding curved elements. The program stores arcs as true arcs. When translating arcs to GDSII format, the program converts each arc into line segments. This number can be controlled via the Arc Resolution dialog box (select Edit > Modify > Arc/Circle Resolution from the Layout window). However, a shape with several arcs, such as a transmission line with curved elements, may not be approximated with sufficient resolution given the 200 vertex limitation. Therefore, some consideration should be given to the use of curved elements and the extensive use of arcs and circles in your layout design.

## Lack of Support for Holes

Lack of support for holes is usually not a problem, but should be considered. The Advanced Design System can convert shapes with holes into contiguous single polygons by inserting an extra edge (shown below). Inserting an edge is valid for GDSII output, but may be a problem for certain mask generation software.



## Filename Limitation

## Advanced Design System 2008

Very long file names (greater than 32 characters) are not supported by GDSII.

### Limit on the Number of Layers

If your layout has more than 255 layers, some of these can be re-mapped to appear on the same GDSII layer.

### UNITS Limitation

The UNITS record within the GDSII file contains two numbers. The first indicates the ratio of (user units)/(db units), the second indicates the number of db units in a meter. GDSII specifications state that the first number should typically be set to less than 1. ADS requires that this GDSII value to be less than 1 since values greater than 1 are not valid for ADS resolution.

### Conclusion

GDSII limitations can affect how you design your layout. The limitation that bears particular consideration is the fact that GDSII Stream format is limited to 200 vertices per polygon/polyline and does not include arcs.

In the Advanced Design System, arcs are stored as true arcs, but are evaluated as polygon segments when translated to GDSII. Although the number of vertices used to approximate an arc can be controlled via the Arc Resolution dialog box (see [Curved Element Limitation](#)), the 200 vertices limit may prevent the program from correctly evaluating polygons consisting of several arcs. Therefore, it is best to consider the number of arcs your shapes contain before designing your layout.

GDSII does not have the ability to empty a polygon or create a hole. The translator can output polygons with holes as single-segment polygons with an extra edge added, or as separate polygons. If this is not acceptable, you may want to prohibit the use of holes in your design.

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

Calma/GDSII Element	Layout Equivalent(s)
Box element	Rectangle
Boundary element	Polygon, Rectangle
Path element	Path
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

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](#). For import options, see [Import GDSII Options](#).

## Guidelines/Considerations

- If a layers file name is defined in the Import dialog, the file is loaded into memory before the GDSII importer starts. See [Importing a Layout](#) for more information.
- If the importer encounters elements on layers not present in the layer definition file, it will automatically add new layers. If new layers are added, the importer saves the revised layer definition file to disk.
- When transferring a GDSII file via FTP, you must specify the binary option.
- When transferring a file via tape, the 2048 block size must be preserved. To write fixed-block sized tapes for transfer to other systems, refer to your computer system documentation.
- When reading a GDSII file, the Advanced Design System ignores reference libraries, font files, generations, format types, and nodes. Element flags (ELFLAGS), complexes (PLEX, see [Plex Property](#)), data types, path types, extensions, and element properties are also skipped.
- Reading a GDSII file may also fail if the file is the wrong version or has been corrupted.

 **Note**  
Because the GDSII stream is a block-structured binary file, it can easily be corrupted when transferring the file from one system to another.

- When reading GDSII path elements, note the following for each path type:

type 0	GDSII path is imported as an ADS path with square corner type.
type 1	GDSII path is imported as an ADS path with square

	corner type resulting in the loss of data in the path's round-ended endpoints and corners.
type 2	GDSII path is imported as an ADS path with square corner type and with endpoints that have been extended 1/2 of the path's width.
GDSII type 3 is non-existent	
type 4	GDSII path is imported as an ADS path with square corner type resulting in the loss of data in the path's variable defined endpoints.

### Filenames Versus Instance Names

When importing a GDSII file into the Advanced Design System, the design name on the Layout title bar may differ from the GDSII filename. The reason for this is as follows.

The GDSII file format can contain multiple top-level instances that may or may not be related to one another. Because of this, the GDSII filename may not be identical to the top-level instance name.

When importing a GDSII file, the top-level instance is automatically displayed in the Layout window. The name of that instance is reflected in the Layout title bar. In some cases, this name does not match that of the imported file.

GDSII file instance names simplify the tracking of hierarchies within the file. Exporting a GDSII file under a new name will have no effect on the named data contained in the file.

### Import GDSII Options

The Import GDSII Options dialog box enables you to specify certain local attributes to control the import of GDSII Stream files. An x appears in the box to the left of selected attributes. To select or deselect an attribute, click it.



**Note**  
To control attributes globally, click the Define Layers button to invoke the Layer Editor. For more information on the Layer Editor, see [Defining Layers](#).

#### Data Type Property

Selecting Data Type Property enables import of GDSII files that use the Data Type record in GDSII and retain the information in a property attached to the corresponding Advanced Design System data structure. When enabled, any Data Type records encountered on Boundary or Path structures while importing a GDSII file cause a property named Data Type to be created and attached to the resulting data group in the Advanced Design System. The value of the property is set to the integer value found in the GDSII record. In the reverse manner, Data Type records are created in the GDSII file when enabled for data groups that have a Data Type property with integer value.

#### Plex Property

Selecting Plex Property enables import of GDSII files that use the PLEX record in GDSII and retain the information in a property attached to the corresponding Advanced Design System data structure. When enabled, any PLEX records encountered on Boundary, Path, SREF, AREF, and TEXT structures while importing a GDSII file cause a property named PLEX to be created and attached to the resulting structure in the Advanced Design System. The value of the property is set to the integer value found in the GDSII record. In the reverse manner, PLEX records are created in the GDSII file when enabled for structures that have a PLEX property with integer value.

#### Define Layers

This button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

### Text Size

The size, in layout units, for text in the imported design. The default value is 1.0. The same size is assumed for all fonts in the GDSII file.

## Exporting GDSII Files

When writing a Calma/GDSII file, the GDSII 6.0 release format is used.

For a step-by-step tutorial on exporting files, see [Exporting a Layout](#). For export options, see [Export GDSII Options](#).

### Guidelines/Considerations

- You can create a bidirectional GDSII file from the current design file. Translated GDSII files appear in layout representation only. Although no electrical connectivity information is included, hierarchy and mask layer information is preserved.
- The Advanced Design System writes the layout representation of a hierarchical design into GDSII stream format using the current layout units and precision.
- The GDSII file created is of fixed-block size (2048 bytes), and is a proper subset of the stream format.
- Valid GDSII layer numbers are from 0 to 255. The data type for all objects is set to 0.
- Note that only some object attributes are translated:
  - A polyline is translated as a path with no specified width, path type, or extension.
  - When paths are converted, the endpoint type is ignored and the type 0 is applied (square-ended at digitized point).
  - Holes in the program are translated as filled polygons, or as a single polygon with the hole joined to other polygons by a new edge.
  - Circles are converted to polygons and arcs are converted to chords.
  - Text is translated with a bottom left justification. A scale factor showing magnification of the text height over one user unit and an optional rotation angle are also translated.
  - Relative mirroring, translation, scaling, and rotation are transferred. Absolute rotation and magnification are not supported.
- Data groups with more vertices than the number specified by the Maximum Number of Vertices option will be skipped during the translation and a warning will be issued. This value can be as high as 8191 (the UI will not allow higher values). Error and warning messages generated during translation are also printed to the log file. After the translation, this file can be viewed.
- Precision and unit information is stored differently in the GDSII Stream format than in Advanced Design System, but no information is lost in translation. A database-unit to user-unit size appears in the GDSII format; this is equivalent to the Advanced Design System's data base precision. This measurement specifies the smallest dimension obtainable in a design. In the Advanced Design System, this is established as a power of ten. For example, a precision of 3 indicates that .001 is the smallest dimension achievable in the design. Since GDSII format contains database to user units, a 3 precision in the Advanced Design System becomes 1000 database to user units in GDSII.

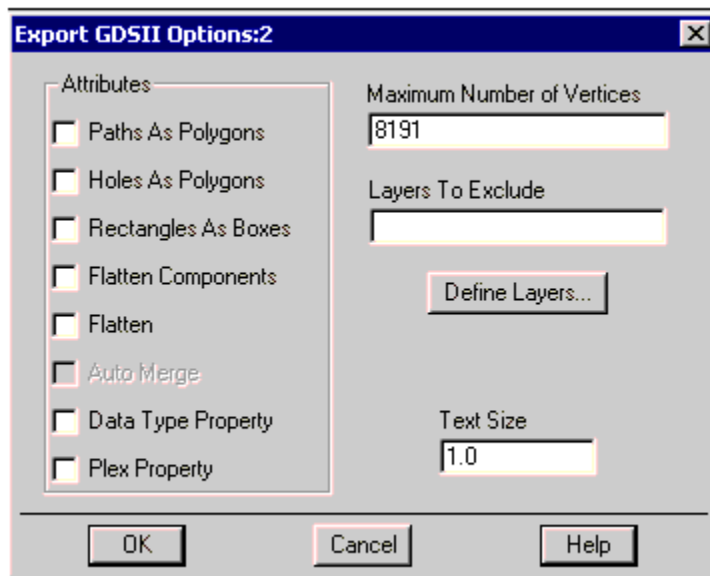
GDSII dB/user unit = 10 | dB precision |

GDSII further employs user units in terms of number of user units to a meter. Since Advanced Design System units can be inches, microns, mils, etc., the GDSII number of user units to meters is equal to the program dB unit converted to meters.

When reading a new GDSII file, the program precision and units should be set to match those that are in the GDSII file, otherwise information may be lost (especially if the program's database precision is less than that of the GDSII Stream file).

## Export GDSII Options

The Export GDSII Options dialog box enables you to specify certain local attributes to control the export of GDSII Stream files. An x appears in the box to the left of selected attributes. To select or deselect an attribute, click it.



**Note**  
To control attributes globally, click the Define Layers button to invoke the Layer Editor. For more information on the Layer Editor, see [Defining Layers](#).

## Paths As Polygons

When Paths As Polygons is selected, the program paths or traces are exported as polygons and GDSII paths are imported as polygons. This should be selected for the following conditions:

## Advanced Design System 2008

- Paths or traces have mitered or curved corners that need to be preserved in the GDSII translation.

This option is deselected as the default.

### Holes As Polygons

When this is selected, holes are converted to polygons. GDSII has no concept of holes. 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. 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.

### Rectangles as Boxes

ADS rectangles get mapped into GDSII box elements.

### 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.

### Flatten

When Flatten is selected, all levels of hierarchy are automatically removed and a single flat design is translated. A component is flattened into its geometric primitives. 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 GDSII Stream 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.

### Auto Merge

When this attribute is selected, the program outputs shapes and forms with data from simulator elements (mlins, slins) and performs an or operation on each layer before outputting. The original dataset is not modified. This option is deselected as the default.

### Data Type Property

Selecting Data Type Property enables export of GDSII files that use the Data Type record in GDSII and retain the information in a property attached to the corresponding Advanced Design System data structure. When enabled, any Data Type records encountered on Boundary or Path structures while importing a GDSII file cause a property named Data Type to be created and attached to the resulting data group in the Advanced Design System. The value of the property is set to the integer value found in the GDSII record. In the reverse manner, Data Type records are created in the GDSII file when enabled for data groups that have a Data Type property with integer value.

This option is deselected as the default.

### Plex Property

Selecting Plex Property enables export of GDSII files that use the PLEX record in GDSII and retain the information in a property attached to the corresponding Advanced Design System data structure. When enabled, any PLEX records encountered on Boundary, Path, SREF, AREF, and TEXT structures while exporting a GDSII file cause a property named PLEX to be created and attached to the resulting structure in the Advanced Design System. The value of the property is set to the integer value found in the GDSII record. In the reverse manner, PLEX records are created in the GDSII file when enabled for structures that have a PLEX property with integer value.

This option is deselected as the default.

### Maximum Number of Vertices

This enables you to define the maximum number of vertices allowed for one polygon. Valid values are 1 - 8191 inclusive. The default value for this option is 8191.

### Layers To Exclude

This enables you to exclude one or more layers from the output file by specifying the layer numbers to exclude, separated by commas. For example:

1,4,9

### Define Layers

This button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

### Text Size

The size, in layout units, for the text in the exported design. The default value is 1.0. The same size is assumed for all fonts in the GDSII file.

## Gerber Translator

The Gerber Artwork Translator can translate artwork directly from circuit layouts created with Advanced Design System into Gerber format. It exports ADS layouts into ASCII files that control Gerber photoplotting equipment.

The Gerber Viewer is a companion program included with the ACS Gerber translator. The Viewer is used for viewing artwork on a computer monitor and for printing it to a graphics printer before creating artwork on the Gerber photoplotter. It is also used for generating drill data and tooling reports.

The Gerber Union tool is also available within the Gerber Viewer. This tool requires a separate license from the Gerber export license. The Gerber Union tool provides the ability to import Gerber files into ADS. It can create EGS Archive files from a set of Gerber files which can then be imported into ADS.

## Gerber Command Format

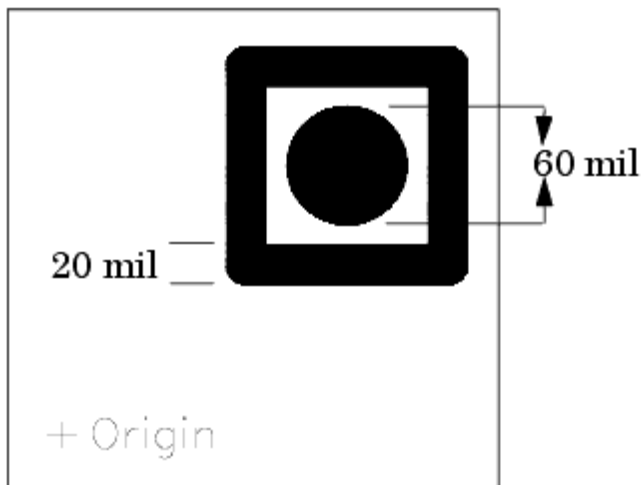
The Gerber format is a numerical control language developed to generate photo artwork. The output of this translator is an ASCII file that contains the following Gerber commands:

- G01E = 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 figure.



The output files are:

- The Gerber Drawing File (.gbr) is the actual Gerber command file.
- The Aperture Usage Report (.apt) reports primitive counts, aperture utilization, and the total distance traveled by a mechanical photoplotting system.

The photoplotter vendor receives both output files.

## Importing Gerber Files

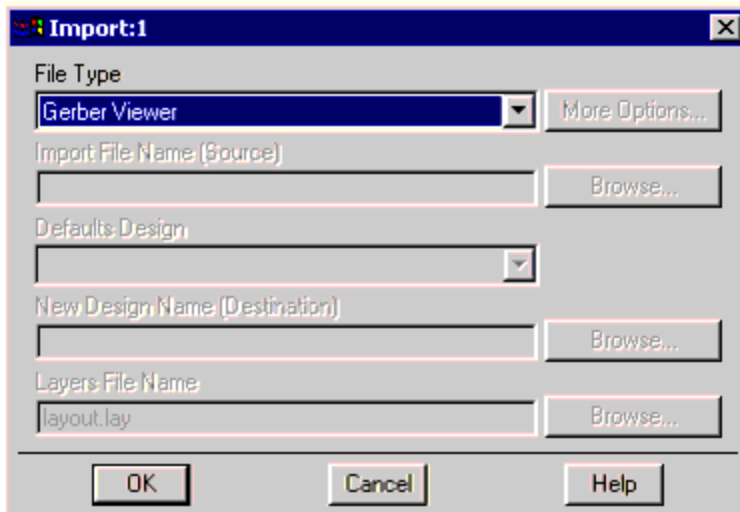
Gerber files can be imported into ADS layout by following a two step process. First, one or more Gerber files are converted into a single EGS Archive file using the Gerber Union tool. This EGS file is then imported into ADS layout.

**Note**  
Only Gerber RS274X files can be processed by the Gerber Union tool.

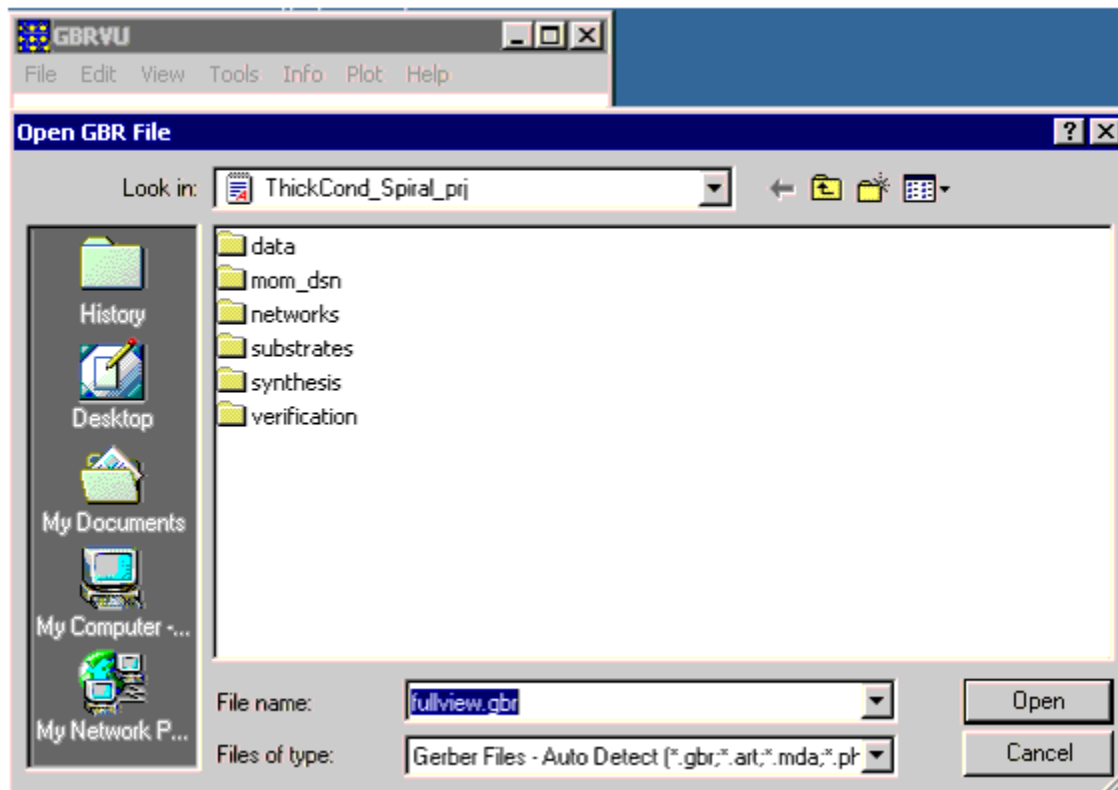
A layer will be created for each Gerber file that is imported. The names of the layers created will be "\_1", "\_2", "\_3", ..., "\_n". The layer table (\_File > Layer) in the Gerber Viewer will show how the Gerber files are mapped to each layer number. This is the default behavior when using the option Multi-Layer Output. Changing this option to Single-Layer Output places all elements on the layer "\_1".

To import Gerber Files into ADS layout:

1. In the Layout window, select File > Import.
2. Set the File Type to Gerber Viewer and click OK, this invokes the Gerber Viewer tool.



3. When the Gerber Viewer (GBRVU) opens, a browser window entitled Open GRB File appears. This window enables you to browse to the folder containing the Gerber files. Select the Gerber file(s) you wish to use and click Open. This dismisses the Open GRB File window and moves the Gerber Viewer tool to the foreground, enabling you to view the contents of the file(s).



4. Select Tools > Gerber Union from the GBRUV window. A dialog appears with the following options.
5. Setting the Gerber Union options:  
 Settings Use Multi-Layer Output to generate a layer for each Gerber file imported. Use Single-Layer Output to place all elements on the layer "\_1".  
 Window Specifies an area of the Gerber files to import by defining its region here.  
 Output Defines the units and scale of the output EGS Archive file.  
 Output File Name Destination path and filename of the EGS Archive file.

**Note**  
 Details on all options can be seen by clicking the Help button in the Gerber Union options dialog.

6. Set the options as desired, then select OK to create the EGS Archive file.
7. Close the Gerber Viewer tool.
8. In the layout window, select File > Import.
9. Set the File Type to EGS Archive.
10. Select the file to import and click OK.

## Exporting Gerber Files

ADS Release 2008 contains a simplified Gerber export tool. This tool can generate Gerber files in RS274X and MDA format for raster photo plotters. For instructions on using MTools to export Gerber files, see [Exporting Gerber Files Using MTools](#).

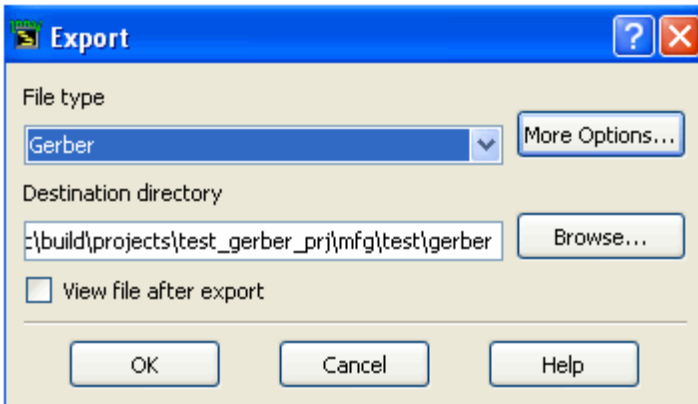
The Gerber export tool creates one output file for each different layer used in the layout design. For example, if the

design contains two layers, cond and cond2, two Gerber files will be generated:

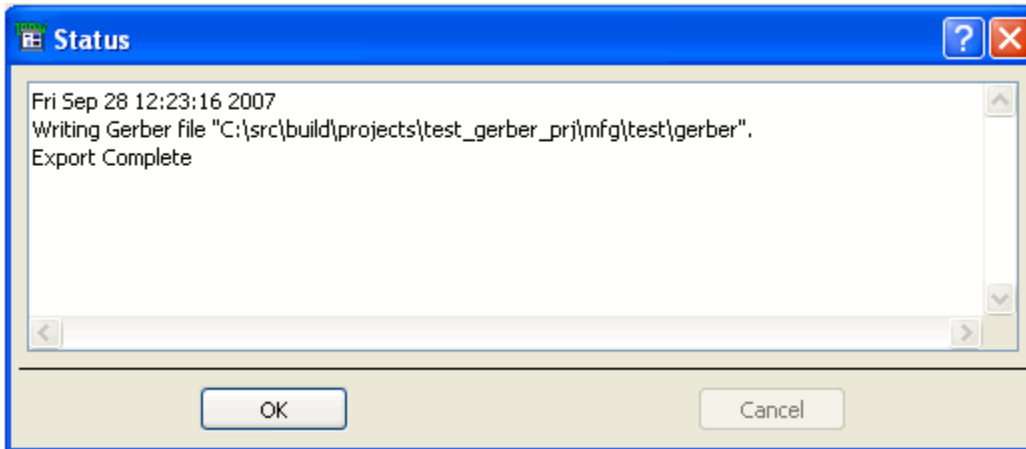
- cond.gbr
- cond2.gbr

To export Gerber files:

1. In the layout window containing your design, choose File > Export.  
The Export dialog box appears.
2. In the File type field, select Gerber.



3. To define export options, click More Options.
4. Set the options that apply.  
For available options, see [Gerber File Options](#).
5. Click OK to save your settings or Cancel to retain the default settings.
6. In the Destination Directory field, accept the default directory, or click Browse to locate a new destination directory.  
The destination directory will be the <Project Dir>/mfg/<Design Name>/gerber/  
where:  
<Project Dir> is the project directory  
<Design Name> is directory named with the design name  
The checkbox option View file after export is not applicable for the Gerber file type.
7. Click OK to export the ADS layout design to a Gerber file.  
A Status window appears detailing the export information.

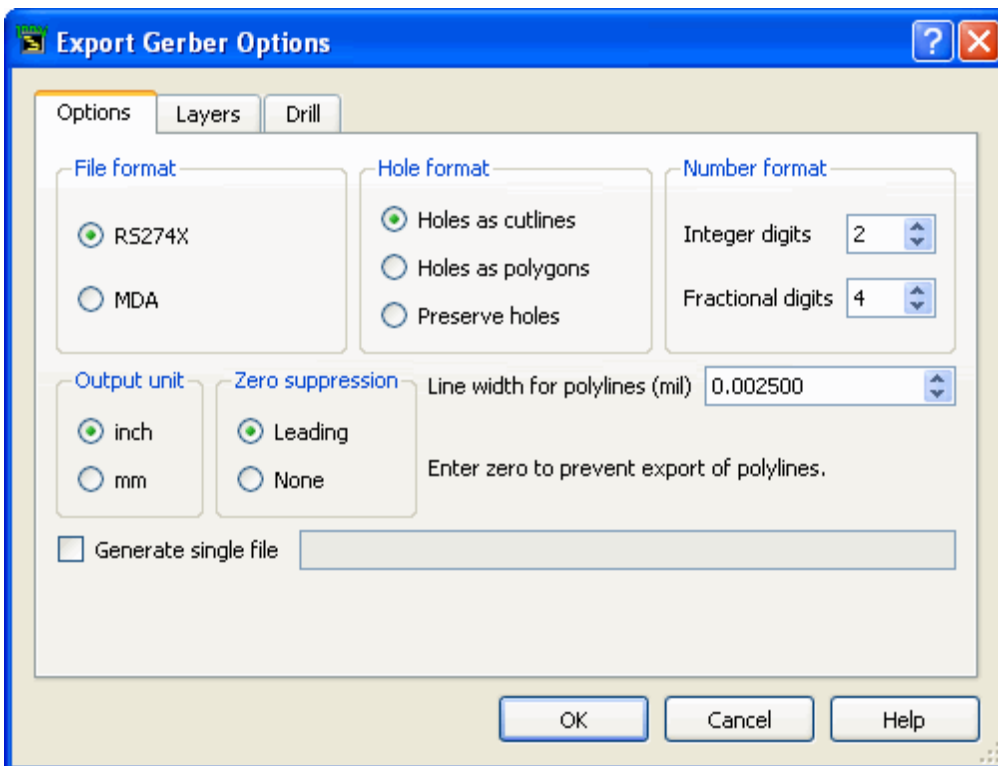


- Examine the log, searching for any warnings or errors that may have occurred during translation. To save the log message in a file: select the text, copy, and then paste into any text editor.

## Gerber File Options

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

The Export Gerber Options dialog appears, providing access to these options.



Gerber file choices can be edited in the Options tab.

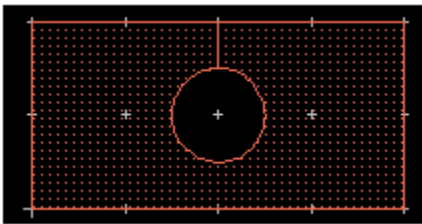
## File format

- 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

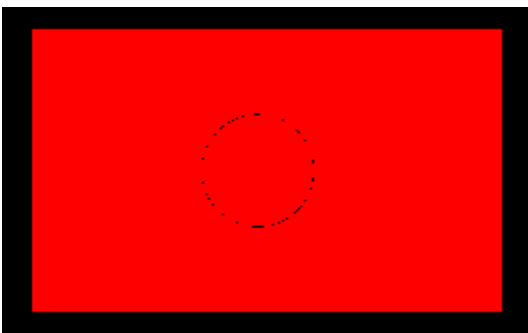
In ADS, when a hole is created in a geometrical figure (like polygon), a cutline is introduced. This is a false edge connecting the outer boundary of the polygon with the inner boundary. This polygon is actually a single re-entrant polygon. When you select Holes as cutlines this re-entrant polygon is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

### 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.

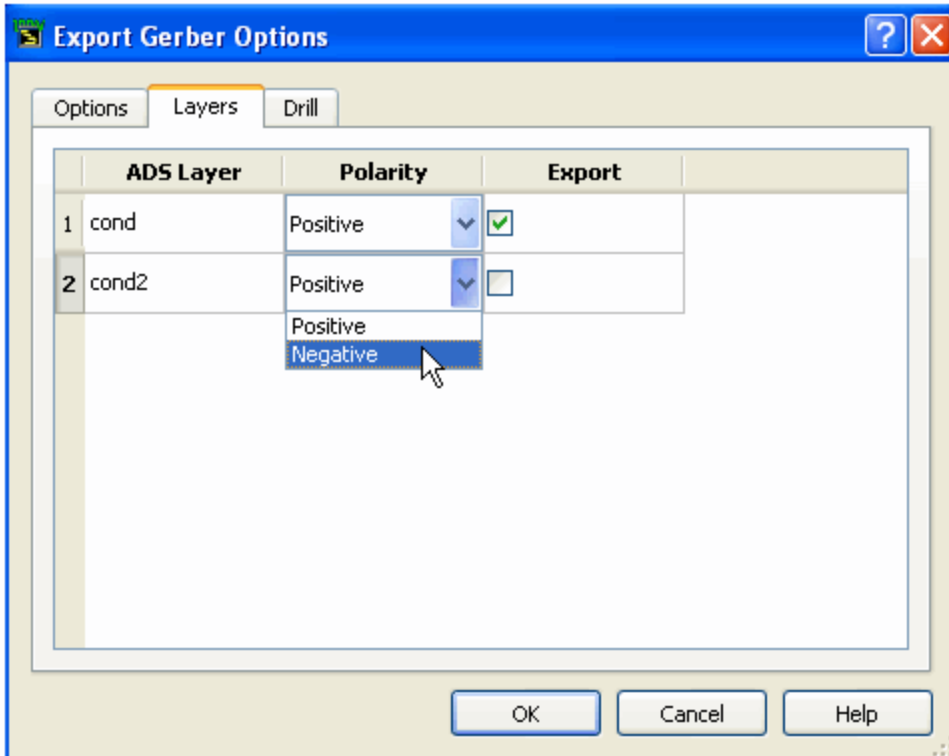
If the polylines and arcs (both zero width entities) are not required to be exported, then set the width to zero.

### Generate single file

When this checkbox is selected, a single file is generated for all the selected layers in the Layer tab. Specify the file name with extension .gbr. This file will be created in the destination directory selected in the Export dialog box.

## Gerber Layer Options

All the layers used in the design are displayed in a table in the Layers tab of the Export Gerber 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

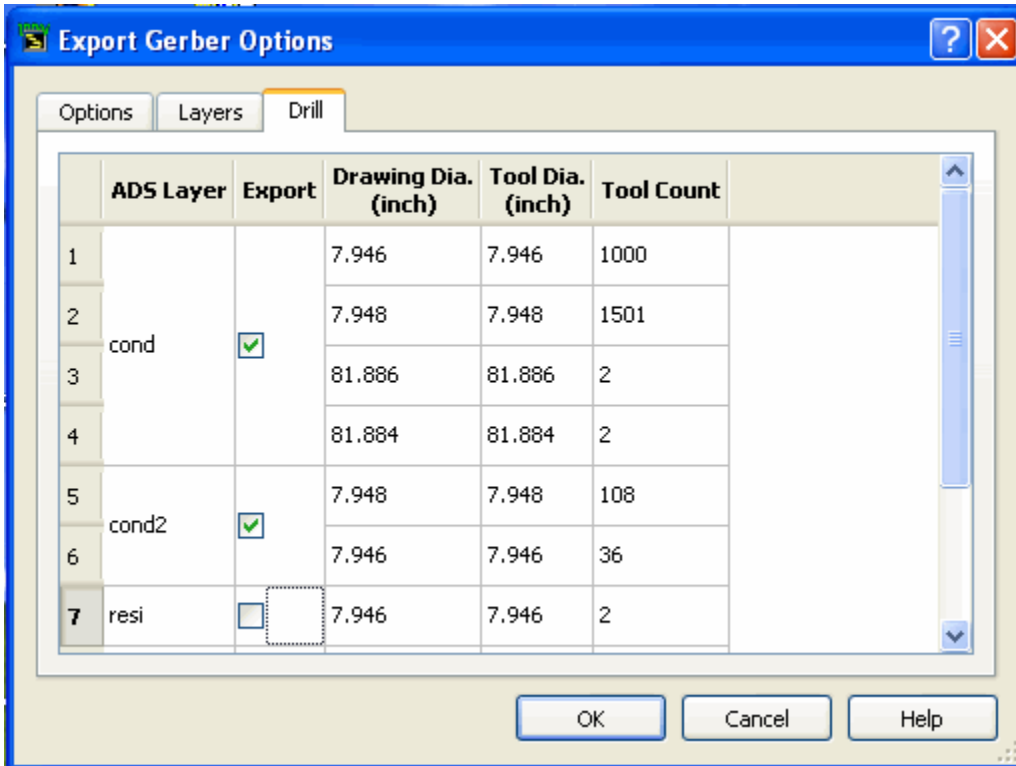
- No intermediate mask file is generated.
- The new Gerber export tool is targeted for the raster photoplotters to generate RS274X and MDA file format.
- No aperture filling is supported for polygons.
- No separate aperture file is required for RS274X and MDA files.
- It is recommended to create the ADS design using inch or mm as the unit and set the appropriate resolution.
- You must set the appropriate number format in the Export Gerber Options dialog so that the precision in Gerber data is more than the precision in layout design.
- The Gerber translator can generate a single Gerber file for all layers. However, the Gerber Viewer cannot process a Gerber file in which multiple layers are defined in a single file. It imports only the first layer defined in the file.

## Exporting Drill Files

In ADS 2008, 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 Export Gerber 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 Export Gerber 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.

## Exporting Gerber Files Using MTools

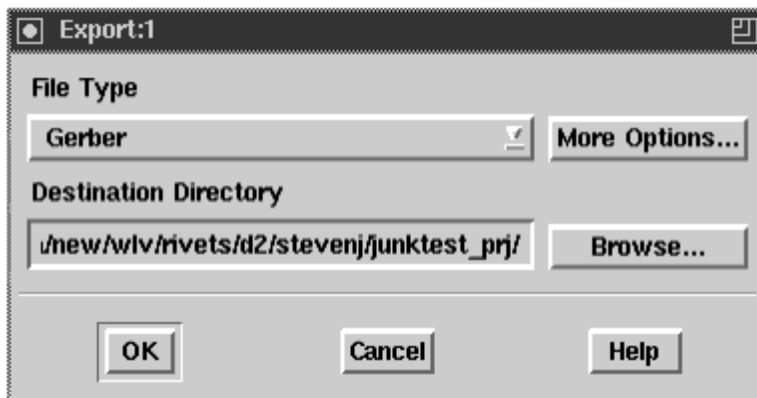
The ADS-to-Gerber translator creates one output file for each different layer used in the layout design. For example, if the design contains two layers, cond and cond2, two Gerber files will be generated:

```
cond.gbr
cond2.gbr
```

In ADS 2002C and earlier versions of ADS it was not possible to perform multiple Gerber translations in a project, as these filenames would be overwritten. However, starting with the ADS 2003A release it is possible to specify a destination folder for each Gerber file, allowing for multiple translations in a single project.

To export Gerber files:

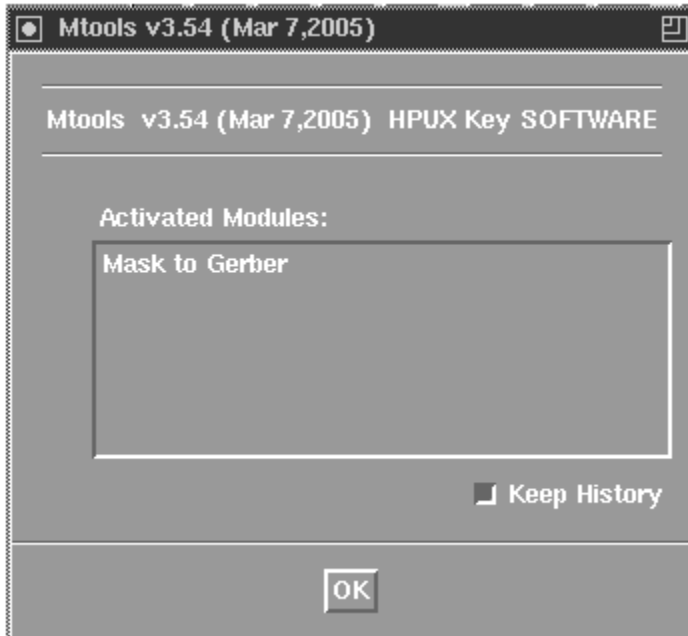
1. In the layout window containing your design, choose File > Export.
2. The Export dialog box appears. In the File Type field, click the arrow in the right-hand corner to display the available formats. Select Gerber.



3. To define export options, select More Options. Set the options that apply. For available options, see [Export Gerber Options](#). Click OK to save your settings or Cancel to retain the default settings.
4. In the Destination Directory field, accept the default directory, or use the Browse button to locate a new destination directory.

**Note**  
If you generate Gerber files for more than one design in the same project, you need to specify different destination folders to avoid overwriting common files (i.e., layers common to most projects like cond and cond2).

5. Click OK to accept the settings. The status window appears:



The entry Mask to Gerber displays under the heading Activated Modules. The Gerber translator converts your design into a mask file before outputting it in Gerber format.

The option Keep History is selected as the default. When this option is selected, the preferences you specify are saved and used as the default for future translations. If you do not want your settings saved, deselect this option.

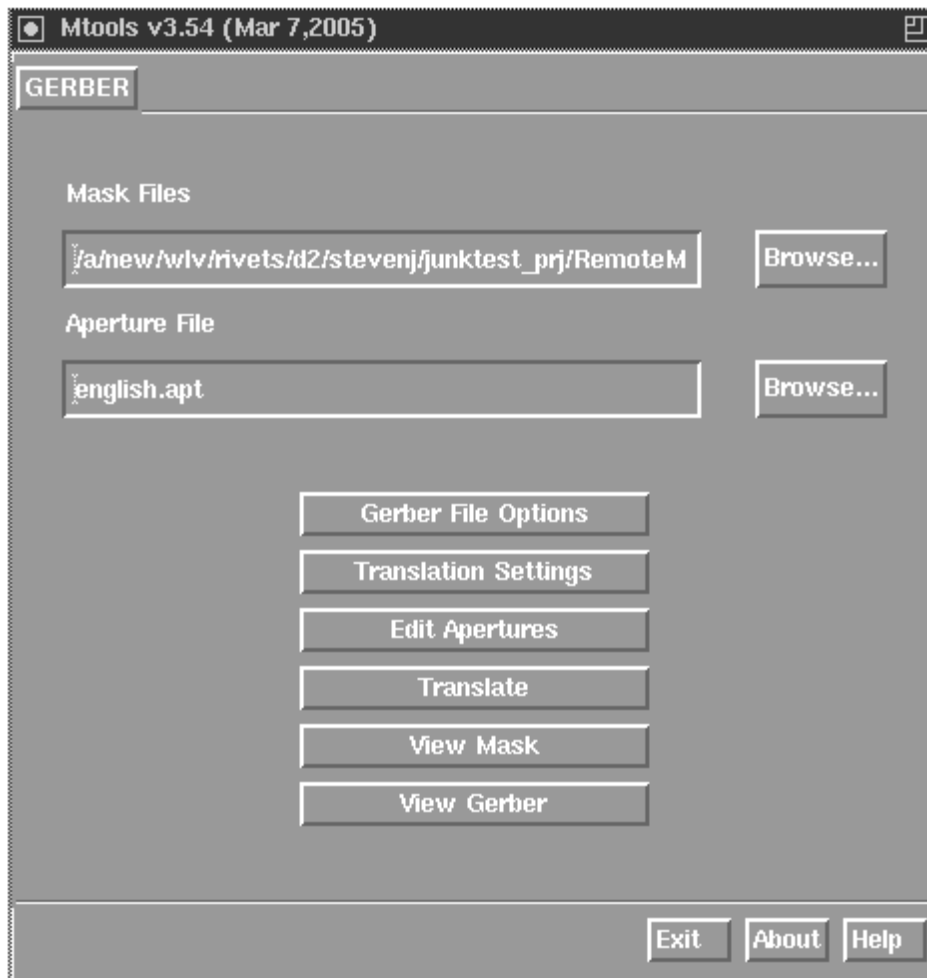
Click OK to proceed. The Mtools Log and Gerber Translator Interface window appear.

6. Verify that the Mtools Log contains the correct information. If necessary, edit the information. To dismiss the Mtools Log window, click OK.

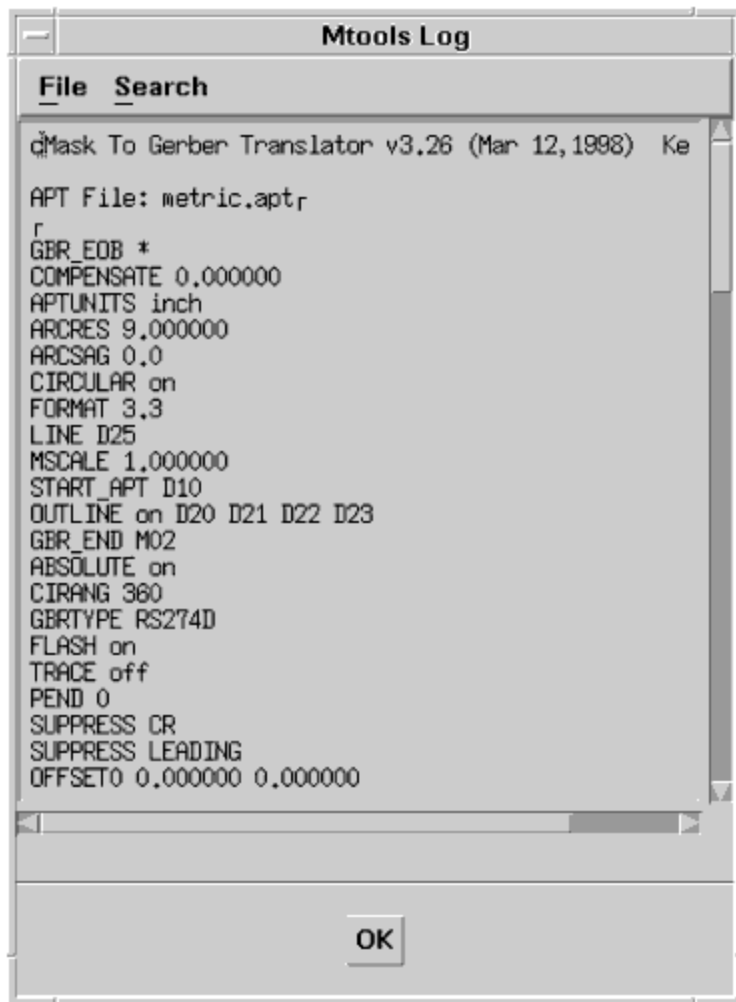


**Note**  
You may view the Mtools Log at any time by clicking the GERBER tab in the translator interface window.

- In the Gerber Translator Interface window, set the Mask Files and Aperture File paths. The Mask Files field displays the mask file created in the Export dialog box. This is the file that the Gerber translator converts into Gerber format.  
The Aperture File field displays the configuration file used to hold all of the translation parameters and Gerber apertures. When performing a translation, you can create a new configuration file or reuse the same file. To modify this file, see [Gerber File Options](#).  
To view the Mask Files and Aperture File paths, click the adjacent Browse button.



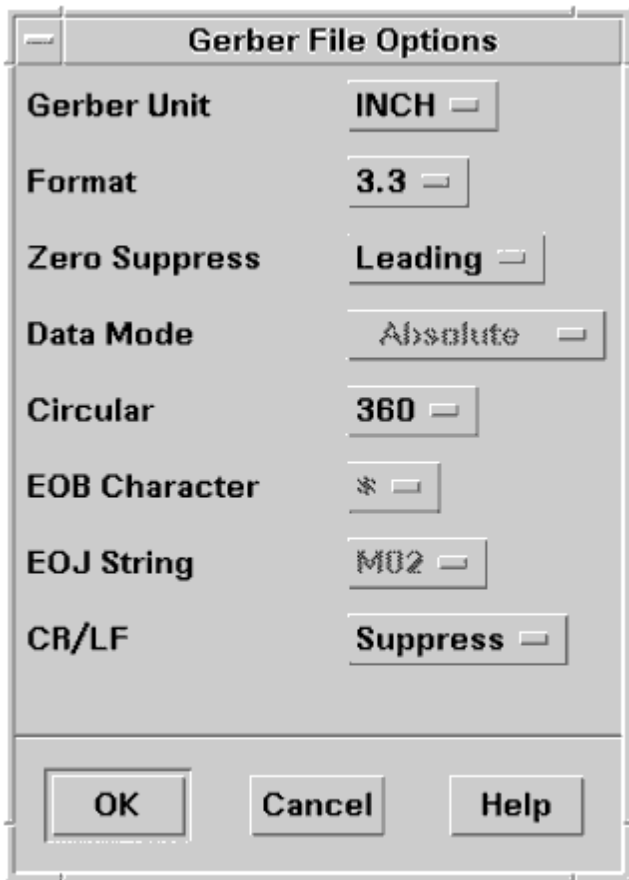
8. Select Gerber File Options , Translation Settings , and Edit Apertures to specify settings. Select View Mask to invoke the Gerber Viewer for DXF and Gerber translations and View Gerber for Gerber translations only. For details on Gerber translator interface options, see the appropriate section: [Gerber File Options](#), [Translation Settings](#), [Edit Apertures](#), and [Using the Gerber Viewer](#).
9. Click Translate to select the layers you want to include in the translated file (see [Translate](#)). After selecting the layers, click OK to complete the translation. A window appears briefly, indicating that the file is being converted to Gerber format.
10. When the translation is complete, the Mtools Log appears, detailing the export information. Examine the log file, searching for any warnings or errors that may have occurred during translation.



When you are finished viewing the log file, click OK to dismiss the MTools Log and exit the Gerber translator.

## Gerber File Options

You can view or edit the translator settings, by selecting Gerber File Options in the Gerber Translator Interface window.



A dialog appears, providing access to these options:

**Gerber Unit.** Available units are INCH or MM. MM is the default.

**Format.** The number of integers placed before and after the decimal point. If chosen incorrectly, Gerber data resolution can be poor. The default is 3.3.

**Zero Suppress.** Available settings are Leading and None. Leading (the default) removes all leading zeros in the coordinate data, making the Gerber file smaller.

If the setting is inappropriate, the Gerber data display is nonsensical.


**Data Mode.** The program always writes out absolute coordinates.

**Circular.** Available settings are 360 or Off. When Off is selected, arcs are fractured. When 360 is selected, arcs are written by means of GO2/GO3 with 360 interpolation. The default is 360.

**EOB Character.** Sets the character that denotes the end of a Gerber data block. Currently set as an asterisk (\*).

EOJ String. This string, inserted at the end of the Gerber file, indicates that the plot is complete. Currently set as MO2.

CR/LF. Available settings are Include or Suppress. Some Gerber files include a carriage return/line feed (CR/LF) at the end of each command. When the CR/LF is suppressed, the file size is reduced by twenty percent and the translation is completed more quickly. Suppressing the CR/LF does not effect your ability to view the Gerber data in the Gerber Viewer. Suppress is selected as the default.

 Note  
After you have set these options, you do not need to do so again unless you deselect the Keep History option in the initial Mtools dialog (see [Exporting Gerber Files](#)).

### Translation Settings

You can control how the program converts the mask data into Gerber format by selecting Translation Settings in the Gerber Translator Interface window. These settings, crucial to correct output, are described in this section.

**Translation Settings**

---

**Global Parameters**

Line DCode       ArcRes   
 Scale Factor       APT Out

---

**Outline/Fill**

Outline      Filled        Empty    
 Fill      Start Apt

---

**Compensation**

inch       Swell  
 Shrink  
 None

---

**Output Offset**

X       Y

---

**Gerber Output Format**

RS274D (Gerber)       MDA Autoplot  
 RS274X       Barco DPF

**Use RS274D format if you plan to merge the output files in the Gerber viewer.**

Global Parameters

Line DCode. Open figures output to this D-code.

Scale Factor. Output data is scaled up or down according to this factor.


## Advanced Design System 2008


ArcRes. Value, in degrees, by which arcs are broken up.

APT Out. Drop-down list for selecting aperture output from popular CAM software.

### Outline/Fill

Outline or Fill. Each closed area is either outlined or filled, depending upon selection.

 Note  
Maximum vertex count for polygons is 50k.

 Note  
When Fill mode is selected, large shapes will be chopped into blocks of 1000 pt Gerber polygons. When Outline mode is selected, large shapes will be chopped into blocks of 5000 pt Gerber polygons.

- Filled. D-codes used for outlining the closed filled figures.
- Empty. D-codes used for outlining the closed empty figures.
- StartApt. The smallest aperture used to start the filling.

### Compensation

Compensate for etch factor by the given inch amount, shrink or swell , as needed.

### Output Offset

The Gerber data coordinates are moved by the amount defined in the X and Y fields.

### Gerber Output Format

The flavor of Gerber output: standard RS274D, extended RS274X or MDA with the Autoplot header. Barco DPF is planned for the future.

### Considerations

When setting the translation options, consider the following:

- Outline vs. Fill - Outline is much more efficient if your vendor accepts it. (Use POEX aperture for polygon fill as needed.)
- Compensation - We recommend that you leave the default setting of None. The compensation available in the Gerber translator is not intelligent and does not take butting or overlapping polygons into account.
- Gerber Output Format - RS274X is the best choice if your vendor accepts it. This format already contains an embedded aperture list. RS274X and RS274D do not currently support empty polygons.

### Edit Apertures

When you click Edit Apertures , a table listing the aperture settings appears.

Click DCode to select the desired aperture type.



Gerber files use apertures much like plotters use pens. Defining available photoplotter apertures is very similar to installing pens into a pen plotter carousel.

To insert a standard set of apertures into the list, click Add Default Set in the Aperture dialog box.

To add a setting, change a coordinate, or change an aperture type, click in the section you wish to change and enter the desired value. You can also change the aperture type via the Dcode dialog:

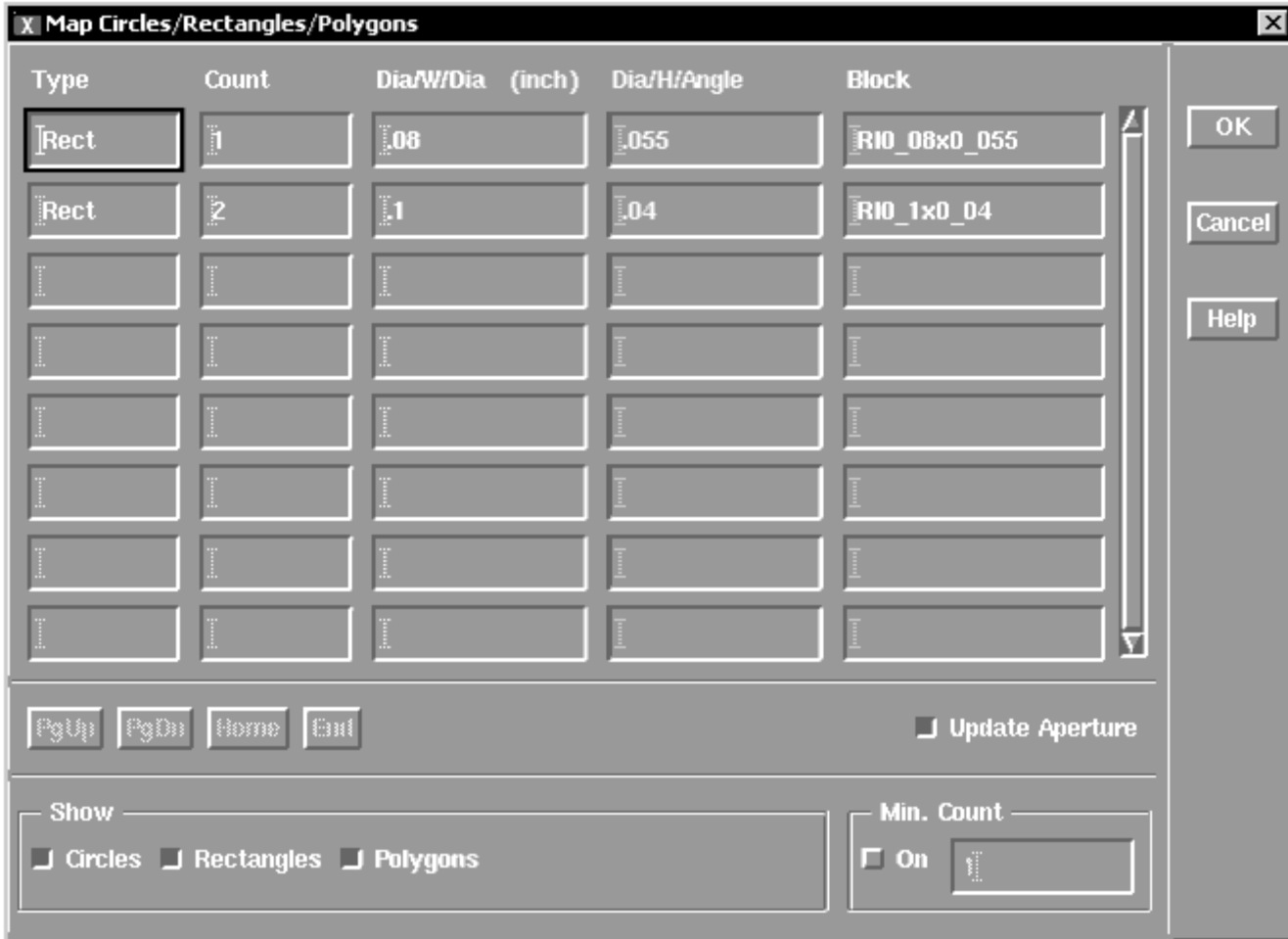
1. Click the D-code number under the heading DCode. A dialog box appears, listing the available aperture types.



2. Select the desired type and click OK. The dialog box disappears.

Among the available aperture types are POEX and POIN. These are special apertures for the FIRE 9000 laser plotter. This plotter can take outline data and fill the inside of each polygon. If you are using such a plotter, the D-codes should be defined for POEX and POIN and the translator should be run in outline mode (see [Translation Settings](#)). For more information about this plotter, see [FIRE 9000 Photoplotter Configuration](#).

Cir/Rect/Poly allows you to flash specific shapes with fixed dimensions, and will bring up the Map Circles/Rectangles/Polygons dialog. When a shape is flashed it will only be defined once in the Gerber file. Every instance of the shape can be referenced in the Gerber file with only the x,y coordinates of its location. Flashing shapes in Gerber files reduces the file size and simplifies the output file.



Show Circles, Rectangles, Polygons check boxes can be used to filter the list of shapes. Unchecking a box will remove those elements from the list of shapes that can be flashed. The default position is checked.

When the Min. Count On checkbox is selected, only shapes that have a minimum count specified in the field will be shown in the list of shapes that can be flashed. The default position is unchecked.

Polygons can be prevented from being processed by the boolean engine if it has the property DB\_NO\_BOOL and its value is 1.

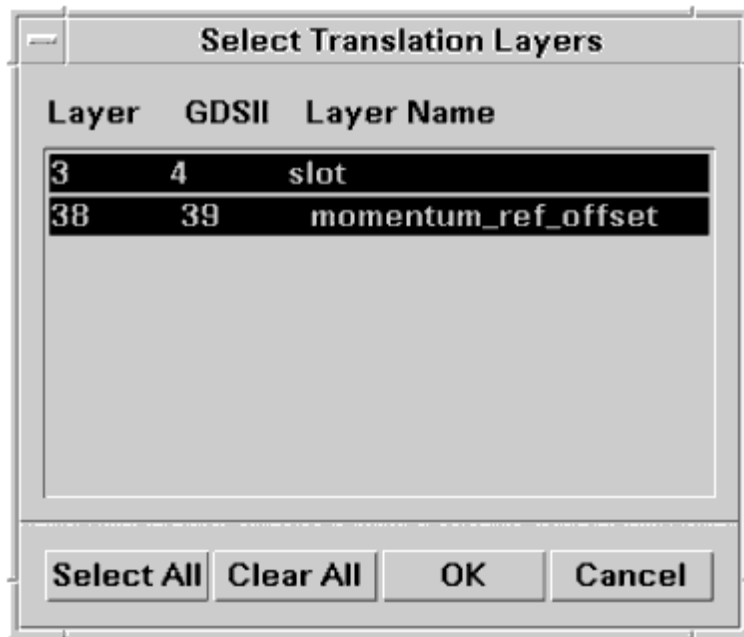
To add this property to a shape:

1. Select the shape.
2. Select Edit > Properties.
3. Type DB\_NO\_BOOL in the name field, with a value of 1 in the value field.
4. click Add > OK.

**Note**  
Selecting the Auto Flash button in the Aperture dialog will automatically add all elements that can be flashed to the aperture table, bypassing the Map Circles/Rectangles/Polygons dialog.

When you are satisfied with the aperture settings, click Save or Save As , or click Cancel to return to the default settings. The Aperture settings window disappears.

## Translate



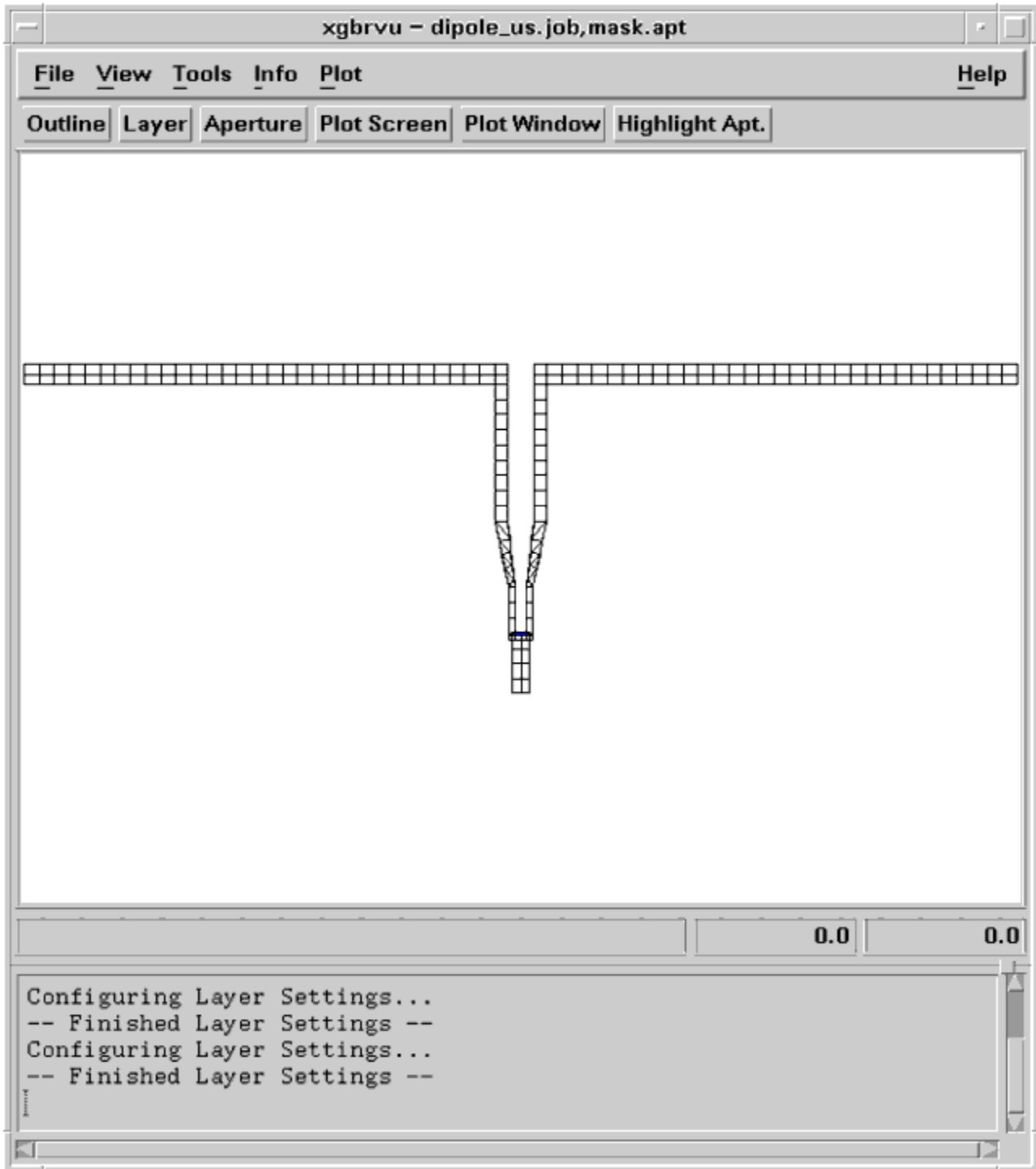
The Translate button invokes the Select Translation Layers dialog. This dialog enables you to select the layers to be included in the translated file. (The Gerber translator creates a Gerber file for each layer in the mask file.) All layers are selected as the default.

**Note**  
Layers are selected when they are highlighted.

To select or deselect a layer, click it. You may also choose Select All or Clear All.

To complete the translation, click OK. A dialog briefly appears, informing you that the mask file is being converted into Gerber format. When the translation is complete, the Mtools Log appears. Inspect the log to verify that the file was translated as expected. (To dismiss the log window, click OK.)

## View Mask



The View Mask button invokes the Gerber Viewer to display the mask file. For information on the Gerber Viewer, see [Using the Gerber Viewer](#).

View Gerber

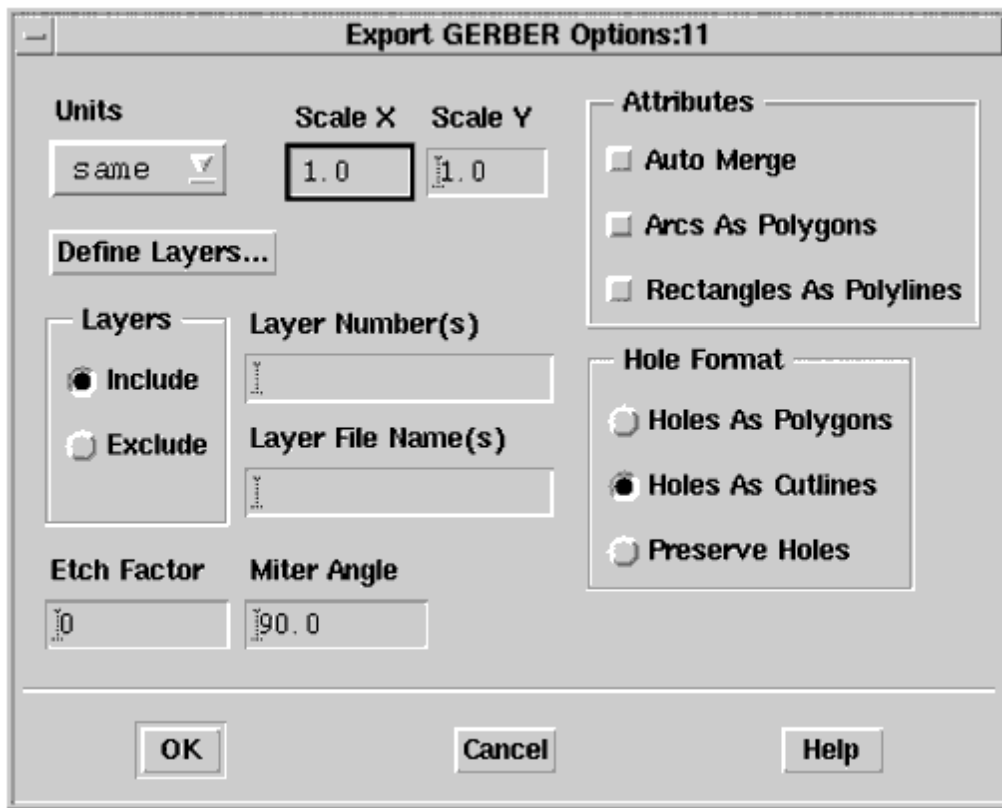
Once the translation is complete, click View Gerber to launch the Gerber Viewer and load the Gerber file(s). The

Gerber Viewer enables you to view your file(s) and generate drill data.

Check the bottom of the Viewer window for messages or instructions.

For more information on the Gerber Viewer, see [Using the Gerber Viewer](#).

Export Gerber Options



Units

These are the units that the Gerber 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 information on choosing layout units, refer to [Setting Units/Scale Factors](#) in the Customization and Configuration manual.

### 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.

### Define Layers

Clicking the Define Layers button invokes the Layer Editor. For Layer Editor Options, see [Defining Layers](#).

### Layers Include, Layers Exclude

The Include and Exclude buttons enable you to specify layers to either include or exclude.

- Include. Exports information only from layers specified in the Layer Number(s) and Layer File Name(s) fields. Include is selected as the default. Unless you specify layer numbers to include in the Layer Number(s) field, all layers are automatically included.
- Exclude. Exports all layers except those specified in the Layer Number(s) field (it is not necessary to fill in the Layer File Name(s) field).

### Layer Number(s)

The numbers of the layers to be included or excluded in the export process. Entries are separated by commas. For example:

1,6,20

### Layer File Name(s)

The numbers and names of the layers to be included in the export process. The information must be presented in pairs, as follows:

<layer\_number> <layer\_name> <layer\_number> <layer\_name>...

where all are separated by spaces. For example:

1 msk1 3 msk3

### 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.

### 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.

### Rectangles As Polylines

When Rectangles As Polylines is selected, all rectangles are translated as open plane figures bounded by straight lines. When this option is deselected, all rectangles are translated as closed plane figures bounded by straight lines. This option is deselected as the default.

### 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 gerber 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. 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 gerber files.

### Etch Factor

The etch factor applies a global over/undersize amount to each shape translated. This is meant to compensate for etch effect during processing. However, using this option can be problematic. Thus, we recommend that you retain the default setting of 0.

If you use Etch Factor, carefully verify the correctness of the compensation to minimize problems. Limitations include the following: When a figure has a side smaller than the etch factor, this function may fail. If two boundaries butt up against one another before compensation, because each boundary is handled independently, such boundaries will either overlap or show a gap when compensation is specified. When Etch Factor is applied, re-entrant polygons may be transformed into illegal polygons.

### Miter Angle

This is the angle cutoff used with the etch factor. The miter angle controls acute angle edge over-extension. Any angle below the miter angle amount is mitered. The default value is 90.0.

### Limitations and Considerations

How you want to use the Gerber output-including layer numbering, use of holes, and polygon shapes-should be extensively considered before beginning your layout design. Setting up the proper layout rules can save a lot of time in generating acceptable Gerber output. For specific considerations or limitations, particularly in relation to apertures and film wheels, consult with your photoplotter vendor.

## Using the Gerber Viewer

The Gerber Viewer enables you to view mask (.msk) and Gerber (.gbr) files. The Gerber viewer can be invoked from the layout window File menu, or during export from the Mtools DXF or Gerber translator window.

### Criteria for Viewing Gerber Files

In order to be viewed, the files must meet the following criteria:

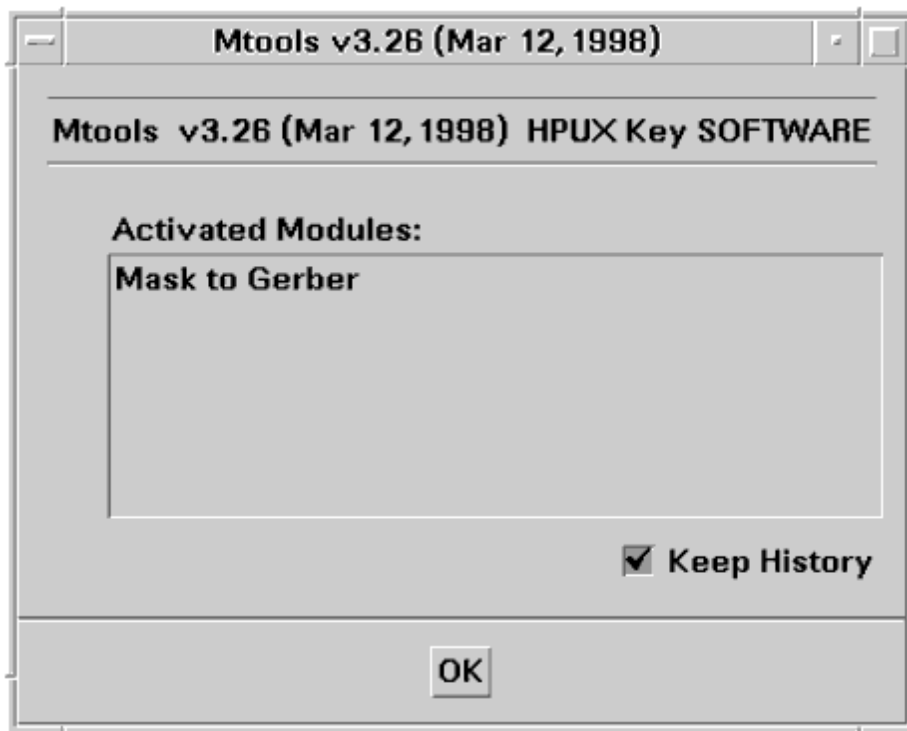
- Use either absolute or incremental data coordinates
- Support apertures from D10-D999
- Have data formats from 0.1 to 4.5

Launching the Viewer from a Layout Window

1. From an Advanced Design System layout window, choose File > Export. The Export dialog box appears.
2. Select Gerber Viewer as the file type.

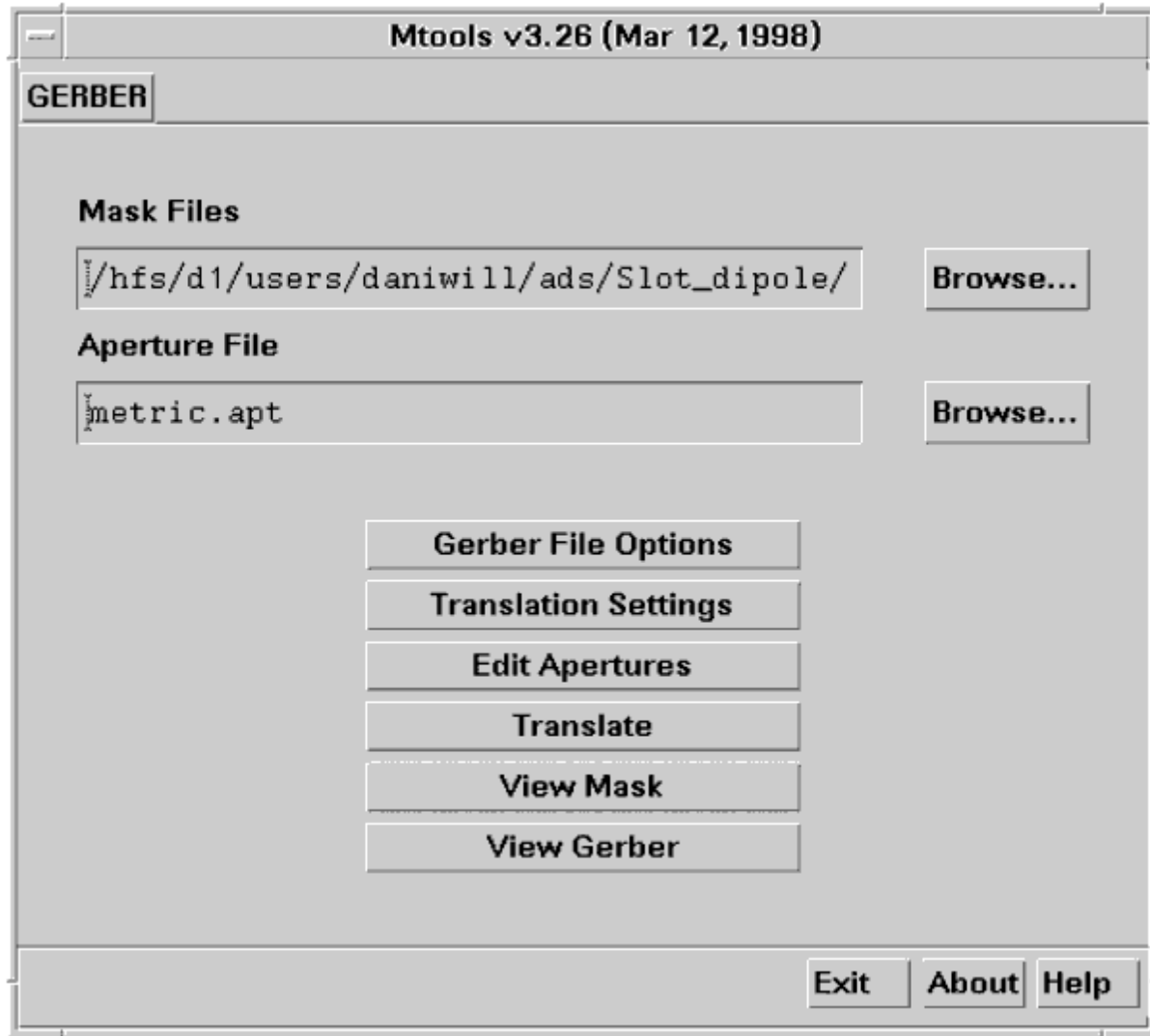


3. In the New File Name (Destination) field, enter the name of the file you wish to view. Alternatively, you can click Browse and the Export File Selection dialog box appears, enabling you to browse your directories for the desired file.
4. Click OK and the following window appears:



5. In this window, click OK. The Mtools translator and log windows appear.
6. To dismiss the Mtools Log window, click OK in the window. You may review it at any time by clicking the GERBER tab in the translator window.
7. In the Mtools translator window, enter the name of the file to view in the Mask Files field. This file must carry the

extension .msk.

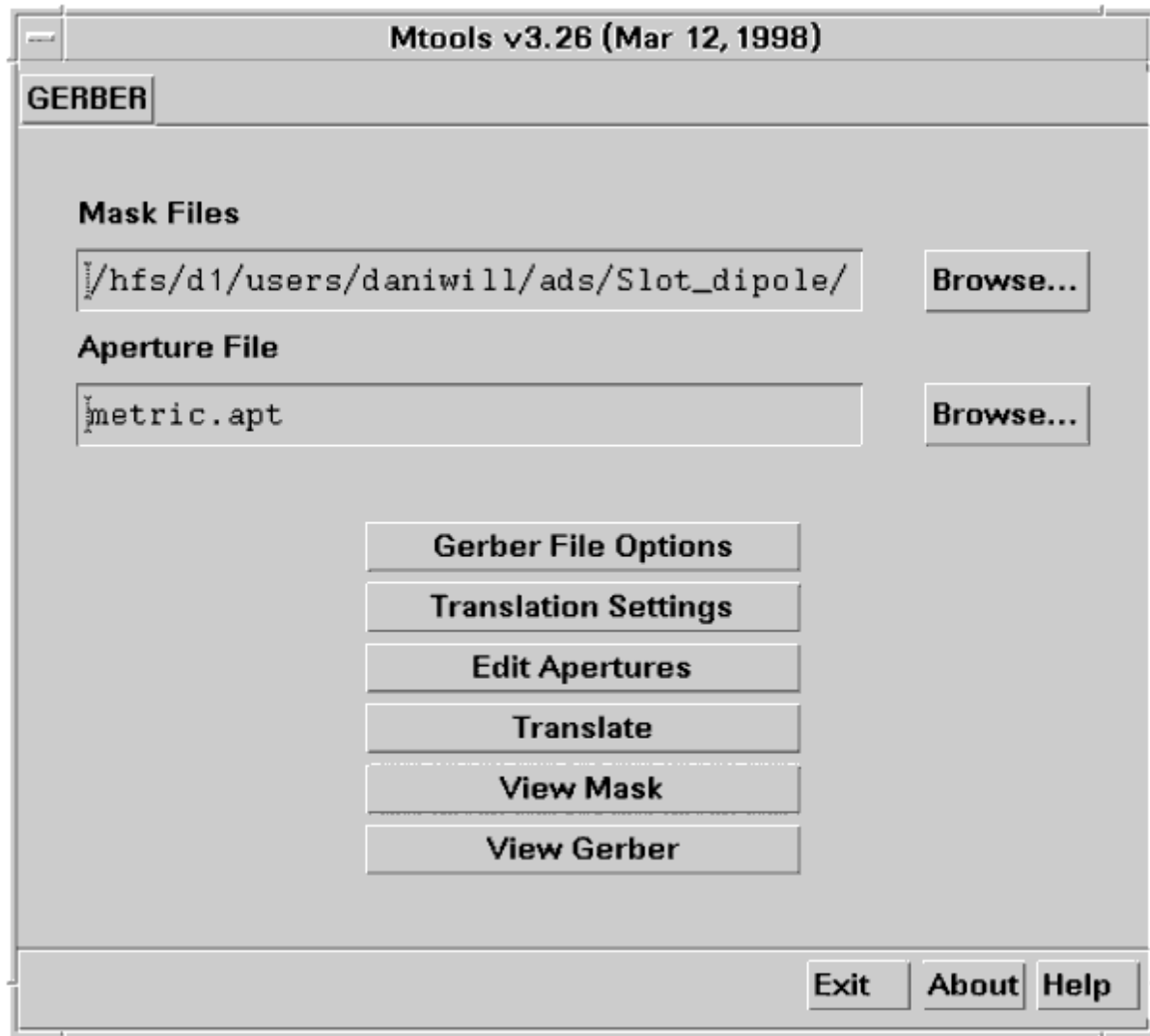


8. To invoke the Gerber Viewer, click either View Mask (for DXF and Gerber translations) or View Gerber (for Gerber translations only). The Gerber Viewer appears.  
For information about Gerber Viewer options, see [Gerber Viewer Menu Options](#). For information on the buttons available from the Gerber translator interface, see [Exporting Gerber Files](#).
9. To exit the Viewer, select File > Exit Xgbrvu.

### Launching the Viewer During File Export

You can launch the Gerber Viewer from the Mtools translator window when exporting DXF or Gerber files.

1. From the Mtools translator window, click View Mask (for DXF and Gerber translations) or View Gerber (for Gerber translations only). The Gerber Viewer appears.

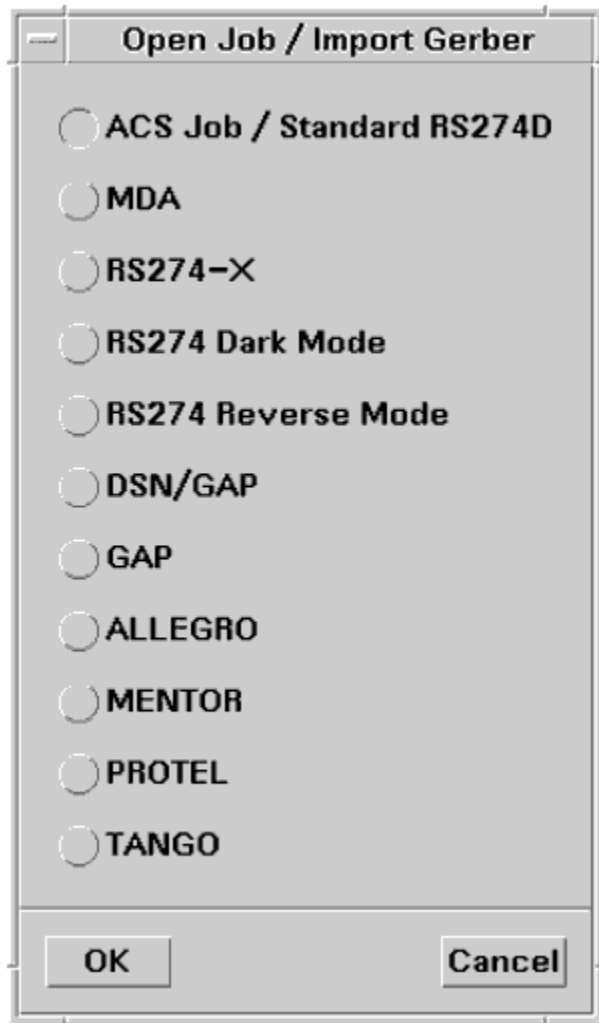


For information about Gerber Viewer options, see [Gerber Viewer Menu Options](#). For information on Gerber translation options, see [Exporting Gerber Files](#). For information on the options available from the Mtools DXF translator interface, see [DXF Translator](#).

2. To exit the Viewer, select File > Exit Xgbrvu.

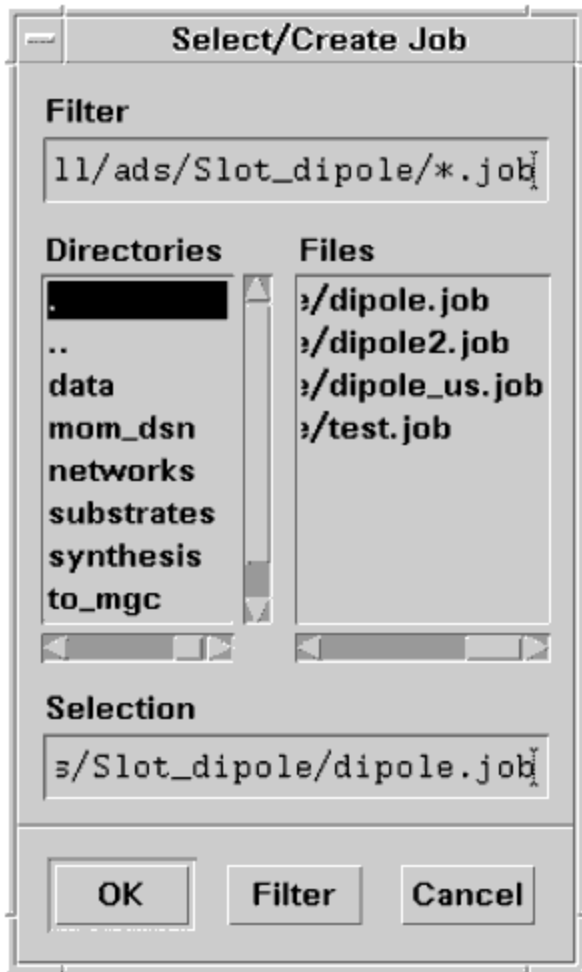
## Loading a File to View

1. Choose File > Open/Import  
If there is a file in current memory (that is, you have initiated the Viewer during a file export) and it is not yet loaded, that file is automatically loaded at this time. There is no need to proceed with the following steps. Otherwise, the Open Job/Import Gerber dialog appears.

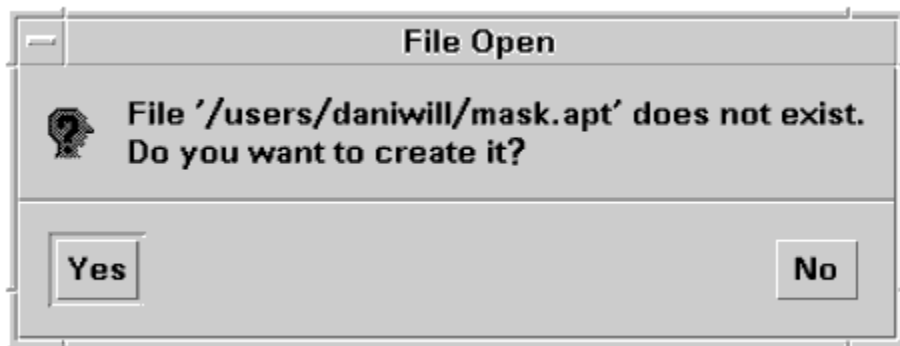


This dialog enables you to load an existing file, or to create a new job file. (The job file is used for purposes internal to the Viewer and is not required by the vendor.)

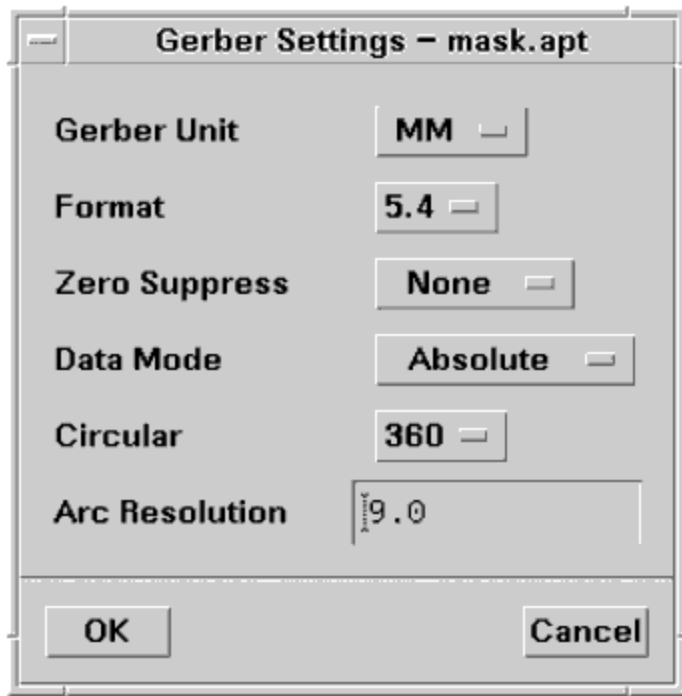
2. Select the desired file type and click OK. A browser dialog that is specific to the file type you selected appears. For example, if you chose ACS Job/Standard RS274D , the Select/Create Job dialog appears.



3. Select the file you want to load or create and click OK.  
 If you cannot remember the name of the file you want to load, you can use a wildcard (\*). The browser will display all files that match the wildcard specification.  
 If the file you choose does not exist, a dialog appears asking if you want to create it. Click Yes to create the file, or No to cancel.
4. The Select/Create Aperture dialog appears. Select the file you want to load or create and click OK.  
 If the file you choose does not exist, a dialog appears asking if you want to create it. Click Yes to create the file, or No to cancel.



5. The Gerber Settings dialog appears. Set options as desired and click OK. (For more information, see [Gerber Settings.](#))

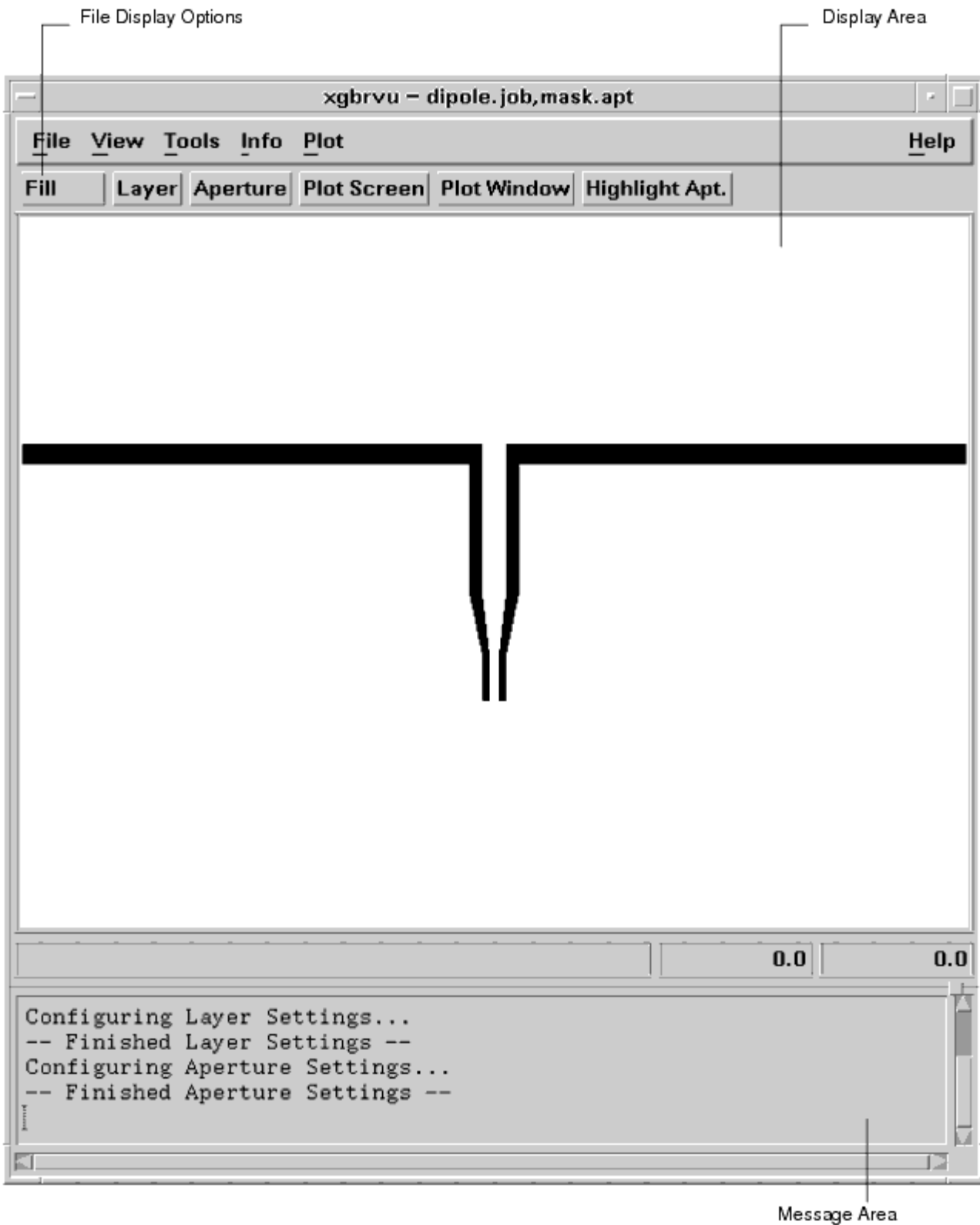


6. The Layer dialog appears. Select layers as desired and click OK. (For more information, see [Layer.](#))
7. The Aperture dialog appears. Set apertures as desired (see [Aperture](#)) and click Save.
8. Set options, view, and plot as desired. See [Gerber Viewer Menu Options.](#)

## Gerber Viewer Menu Options

This section describes the Gerber Viewer menu options. A message area at the bottom of the Viewer window provides information specific to your situation.

The two file display options are: Outline or Fill. Click the appropriate button to change the window display. The button title changes to reflect the display.



### File

Click File and a drop-down menu containing the following options appears:

- Open/Import loads the job file. The job file contains the name of the aperture file, data units, and Gerber files to load. For more information, see [Loading a File to View](#).
- Layer invokes the Gerber Viewer Layer menu. For more information, see the section [Layer](#).
- Aperture invokes the Gerber Viewer Aperture menu. For a description of this menu, see the section entitled [Aperture](#).
- Preferences leads to the following menu choices:
  - Gerber Settings invokes the Gerber Settings dialog. See [Gerber Settings](#).
  - Grid/Snap Settings opens the Snap Settings dialog. See [Snap Settings](#).
  - Poex Check
- Exit Xgbrvu dismisses the Gerber Viewer.

### View

The following display options are available:

- All displays the extents of the data. This is the default option.
- Window enables you to use the mouse to draw a window around the area you wish to view.
- In zooms in, or magnifies, the view by two times.
- Out zooms out by two times
- Pan enables you to pan the display window.
- Last restores the previous view.
- Redraw refreshes the screen.

### Tools

The Tools menu contains the following options:

- Drill enables you to select your drill output. Available choices are listed below. When you click your selection, the Drill Output dialog appears, stating layer, drill output and tool count. To dismiss this dialog, click Close.
  - Excellon (Suppress Leading Zero) is the default option.
  - Excellon (No Zero Suppress)
  - XY Table outputs a drill summary table.
- Report displays the Gerber Report dialog. This dialog provides a text summary of the current file, including layer and aperture information. To dismiss this dialog, click Close.
- Film Merge invokes the Gerber Merge dialog. For more information, see [Gerber Merge](#).

## Info

The Info menu provides the following options:

- Vertex Query searches for the vertex point nearest to where you click the mouse.
- Window Query summarizes the trace information of a specified area.+
- Flash Query identifies the flash point nearest to where you click the mouse.
- Measure measures the distance between two points.
- Highlight Apertures invokes the Highlight Aperture dialog. For more information about this dialog, see [Highlight Apt.](#)

When you select (click) an option, information and instructions appear in the message area at the bottom of the Gerber Viewer window. To quit a task, press the ESC key.

**Note**  
Actions prompted for in the Viewer message area (such as pressing ESC to abort) are effective only when the mouse is pointed in the Viewer display area.

## Plot

The following options are available from the plot menu:

- Plot/Page Setup invokes the Page/Plotter Setup dialog, enabling you to specify page setup, plot format, output, and setup strings.
- Plot Screen plots only the area displayed on the screen.
- Plot Window enables you to plot a selected area. You use the mouse to draw a box around the specific area you wish to plot. For instructions on how to do this, see the Gerber Viewer message area at the bottom of the Viewer window.
- Plot Window Coords invokes the Plot Window dialog, enabling you to specify plot coordinates.



## Outline/Fill

The Outline/Fill button reads as either Outline or Fill , depending upon the display type activated. That is, if Outline appears, the file is displayed in outline form, and if Fill appears, it is displayed as filled.

To change the display from outlined to filled, or vice-versa, click this button.

## Layer

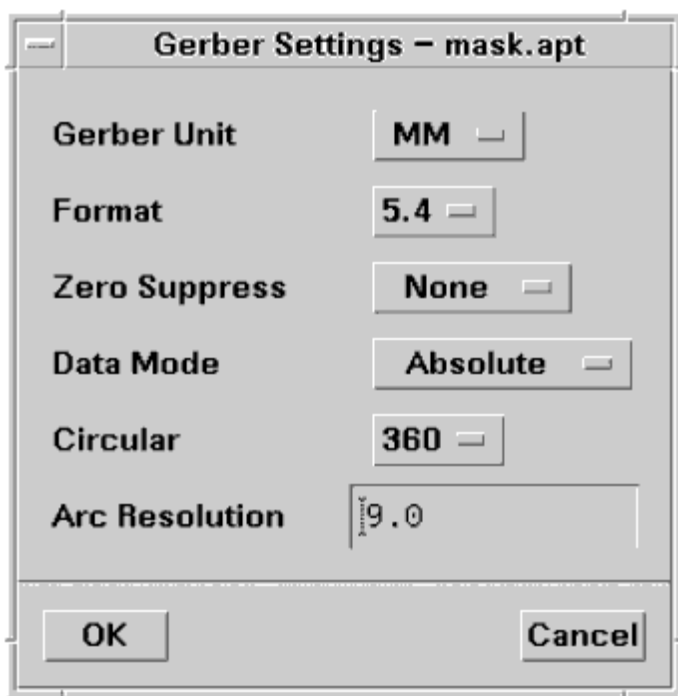
The Layer button invokes the Gerber Viewer Layer menu. (You can also choose the menu command File > Layer to open the Layer menu.)

The Layer menu enables you to select which layer files are loaded for display; whether these are displayed as paint, scratch, or negative; and the color they are displayed in. Up to 48 layers may be simultaneously displayed. For information, see Layer Menu.

**Note**  
Changes to the Layer dialog options do not affect the raw Gerber data. They change the display only.

## Gerber Settings

File > Preferences > Gerber Settings



## Advanced Design System 2008

**Gerber Unit.** The units the Gerber data adopts when it is loaded. You may select either INCH or MM. MM is the default.

**Format.** Gerber data does not contain the decimal point. For coordinate data, you must tell the program the number of digits to insert before and after the decimal point. The default Format value is 3.3. Other common formats are 2.3 and 4.4.

Data formats from 0.1 to 6.3 are acceptable. However, arithmetic problems may arise when very large data values are rendered precisely. For example, if you have a circuit board that extends out 20 inches from the origin and you require 4-place accuracy across the board, you may find that some data points out at the 20,20 coordinate have only 3-place accuracy. This may cause the program to incorrectly fill polygons in this region.

**Zero Suppress.** Gerber files are normally compressed by suppressing either the leading or trailing zeros of the Gerber data. You may specify Leading , Trailing , or None. Leading is the default. If you choose the wrong setting, the Viewer displays nonsensical data.

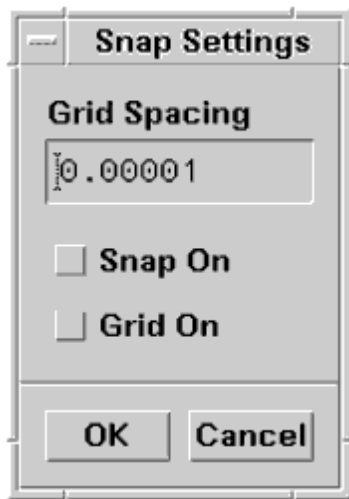
**Data Mode.** The data coordinates can represent either absolute or incremental values relative to 0,0. Absolute is the default. When Incremental is selected, each coordinate represents the distance from the previous coordinate. Selecting an incorrect data mode results in an incoherent display of the data.

**Circular.** You may select either 90 or 360. Older photoplotters used 90 degree arc interpolation. Newer machines support 360 degree interpolation. The default setting is 360. If your arcs display incorrectly, try changing this setting.

**Arc Resolution.** To display Gerber circular commands (G01/G02), the Gerber Viewer uses small, straight segments to approximate the data. How fine the approximation is depends upon the Arc Resolution value. The default value of 9 degrees is appropriate in most cases. This value offers a good compromise between speed and resolution. Valid values are between 0.5 and 30 degrees.

### Snap Settings

File > Preferences > Grid/Snap Settings



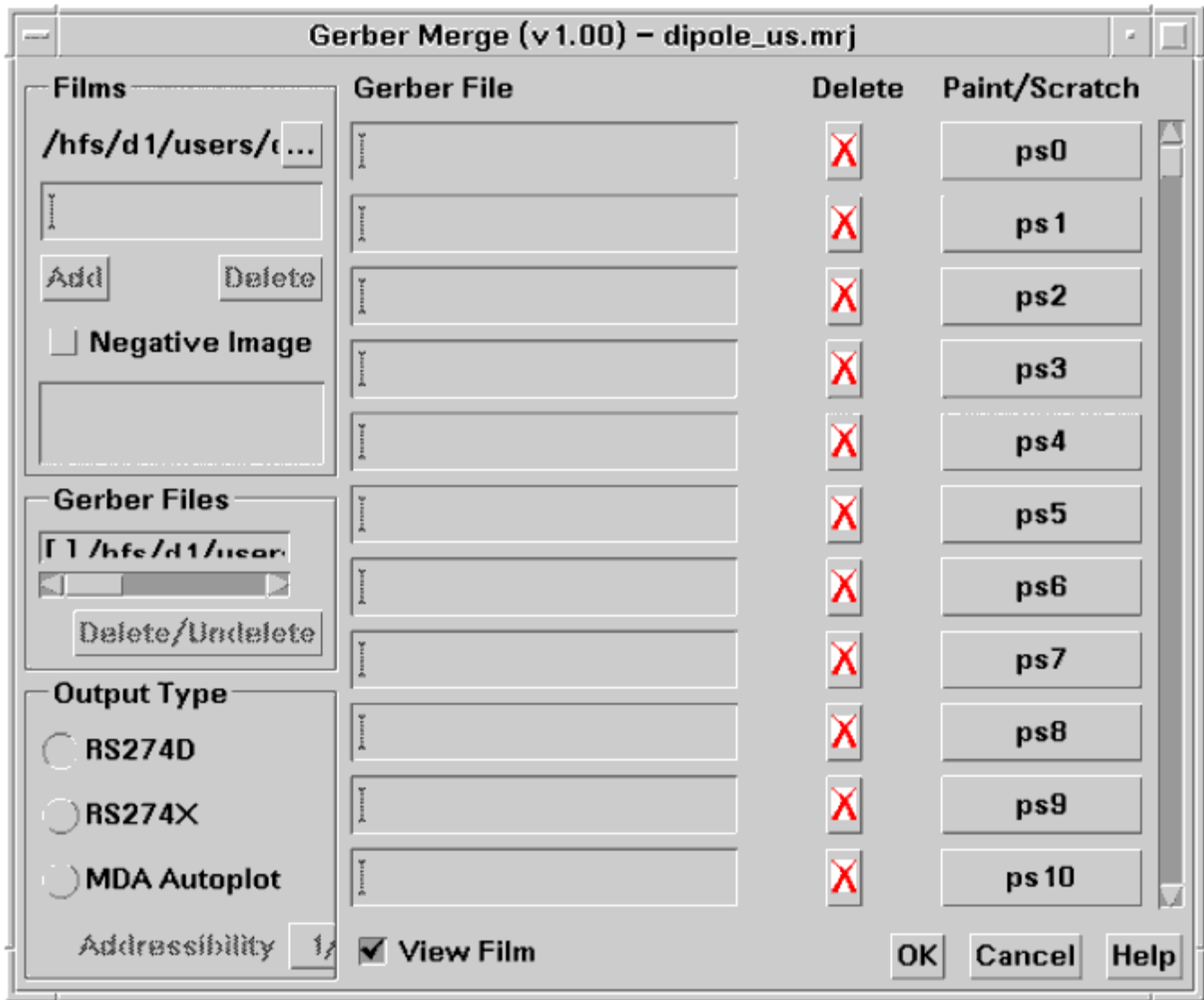
Grid Spacing. Use Grid Spacing field to set the grid spacing as desired.

Snap On. Click the selection box to turn this option on or off. This option is selected as the default.

Grid On. Click the Grid On selection box to turn this option on or off. This option is deselected as the default.

## Gerber Merge

To invoke the Gerber Merge dialog, select Tools > Film Merge



Films. The current filename is displayed in this field. To enter a filename, click in the entry box and type the desired file path. You can also click the ellipses box (...) in the right-hand side of the Film field to select a file from the Working Directory browse dialog.

Negative Image. Click the selection box to turn this option on or off. This option is deselected as the default.

Gerber Files. Your Gerber files are listed in this field. You may delete or undelete these by clicking the Delete/Undelete button below this field.

Output Type. You may select RS274D, RS274X, or MDA Autoplot as the output format. RS274D is selected as the default.

Gerber File. List the Gerber files that you wish to merge in this column.

Delete. This field is disabled.

Paint/Scratch. This field is disabled.

View Film. Click the selection box to select or deselect this option. View Film is selected as the default.

### Layer Options

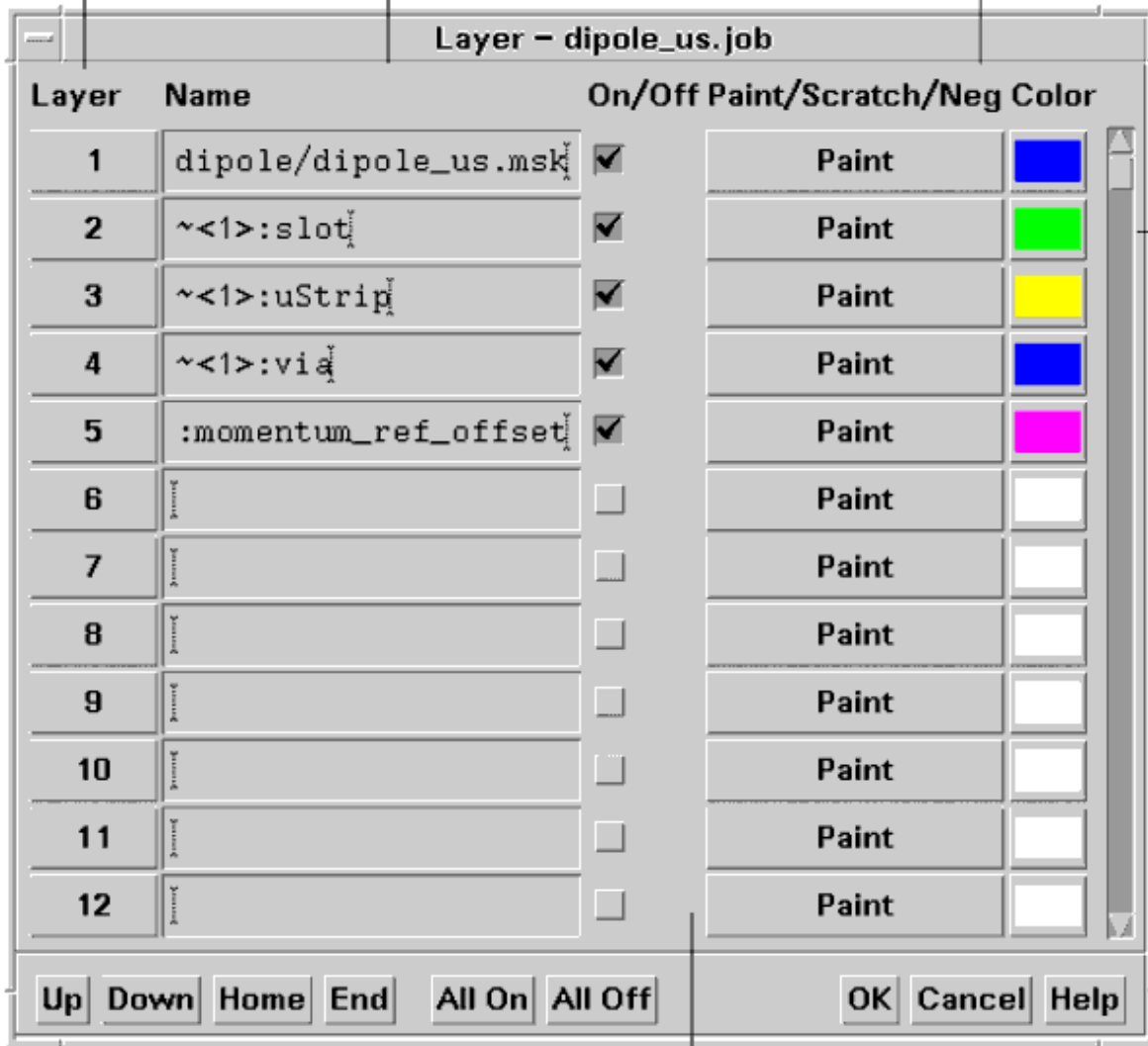
To load your Gerber files, click on a Layer number button. A file selection dialog box appears, enabling you to select the file(s) you want loaded. You may select display options as desired.

The Layer dialog and its options are shown in the figure.

Click the Layer number to open the Select Gerber File dialog. Browse your directories, select the file(s) you desire, and click OK.

Type the full name of the Gerber or mask file in this field. Alternatively, click the Layer number to open the Select Gerber File dialog and select the file from your directories (see left).

Click this button to specify Paint, Scratch, or Negative. For more information on these display types, refer to the text following this figure.



Click the Color field to open the Layer Color palette. Click the desired layer color.

This field toggles the layer display. When highlighted, the layer is displayed. To turn all layers on or off, use the All On or All Off buttons at the bottom of this dialog.

Paint displays the layer data normally.

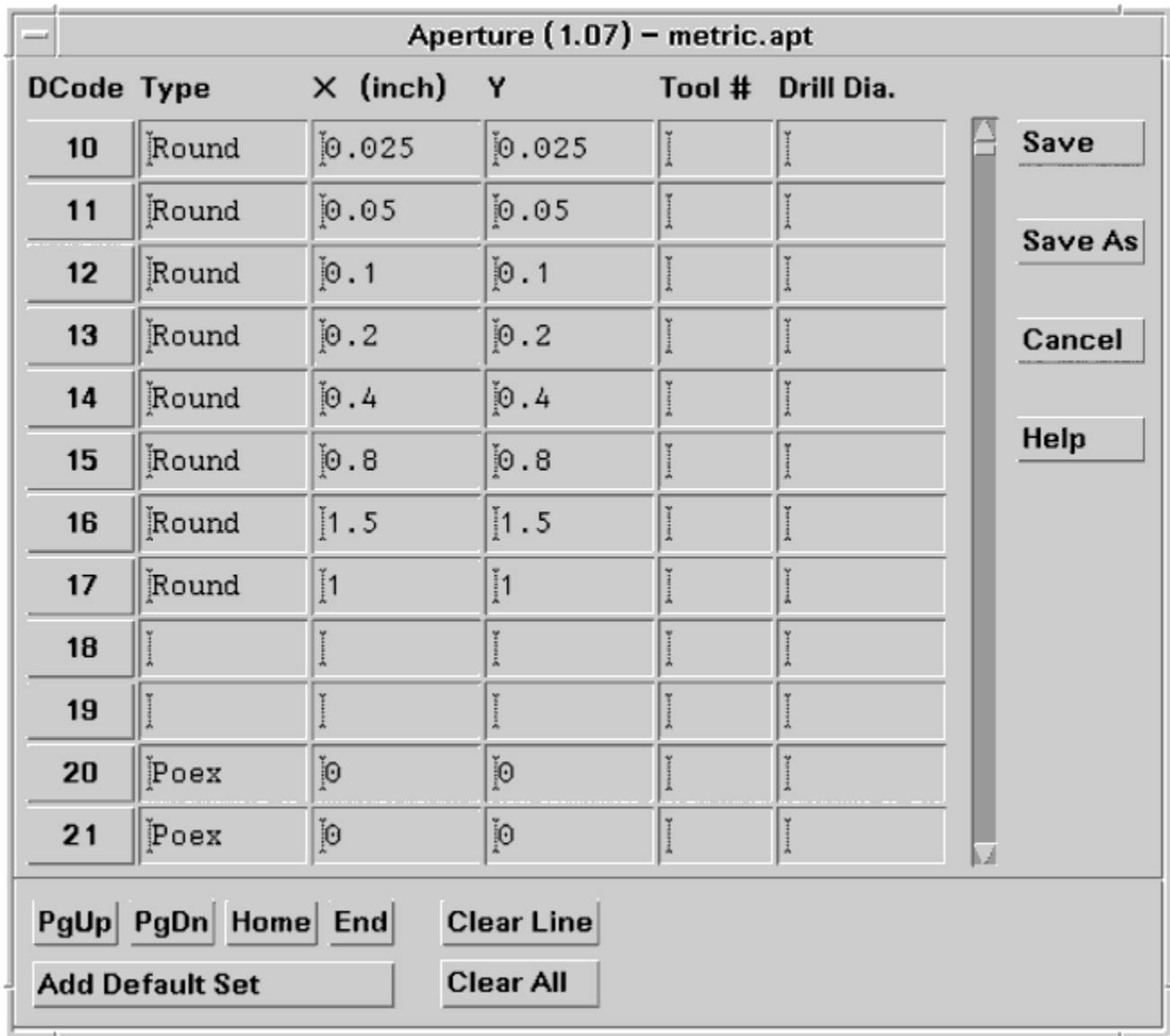
Scratch performs an XOR operation on all data previously loaded. The resulting display mimics the effect of scratch behavior available on many photoplotters.

Negative reverses the polarity of the layer and is useful for displaying ground planes.

### Aperture

File > Aperture

The aperture button in the Gerber Viewer Options window invokes the Aperture menu, enabling you to define apertures from D10 through D999 and assign drill tools to create Excellon drill codes.



To accurately display your data, you must set the correct apertures.

DCode. You may define D-codes from D10 through D999. Click in this field and the Dcode dialog appears, enabling you to select one of 12 standard types (see below).

Type. Twelve types are available: round, donut, square, rectangle, thermal, target, octagon, oblong, poex, poin, thmsq, and custom.

To specify a type, click in the Type field and enter the desired type. Alternatively, click the DCode number and the Dcode dialog appears, presenting the available types for selection.

X , Y. These are the aperture dimensions in units of inches or millimeters (mm), depending on the Gerber unit selected (see [Gerber Settings](#)). Some apertures, such as round pads, have only an X value; others require X and Y

values.

**Tool #.** This field is for specifying an Excellon drill data tool. This field is optional and need only be specified when converting Gerber flashes to Excellon drill commands. Valid ranges are T01 through T99.

**Drill Dia.** Enter the drill tool diameter in this field. This field is optional. Valid ranges are 0.001 inch to 0.99 inch.

**Add Default Set.** When you click this button, a standard set of apertures is inserted into the list.

### Plot Screen

You may select this option by clicking the Plot Screen button on the Gerber Viewer menu bar or by selecting the menu command Plot > Plot Screen. The screen is plotted, and the plot information appears in the message area at the bottom of the Viewer.


### Plot Window

You may select this option by clicking the Plot Window button on the Gerber Viewer menu bar or by selecting the menu command Plot > Plot Window. This option enables you to plot a portion of the screen by using the mouse to draw a box around the area you wish to plot. This is done by clicking the mouse on the first and second corners, diagonally, of the plot window. Instructions and plot information appear in the message area at the bottom of the Viewer.

### Highlight Apt.

The Highlight Apt. button opens the Highlight Aperture dialog. This dialog enables you to select apertures to be highlighted on the display.

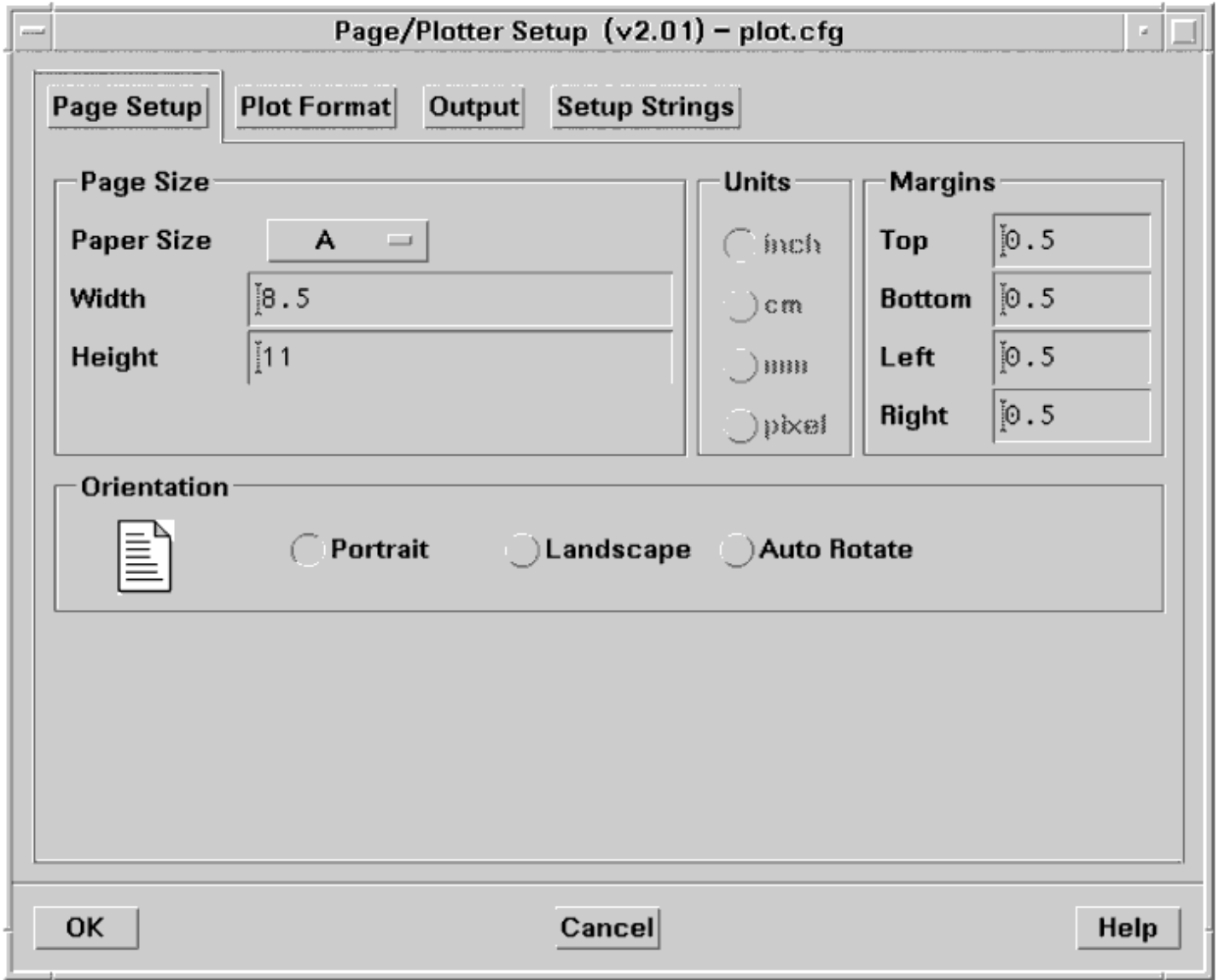
!trans-08-1-33.gif!

 **Note**  
Double click an aperture to select and display it. All traces associated with a highlighted aperture appear in light gray rather than in the assigned layer color. Click OK to dismiss the dialog box.

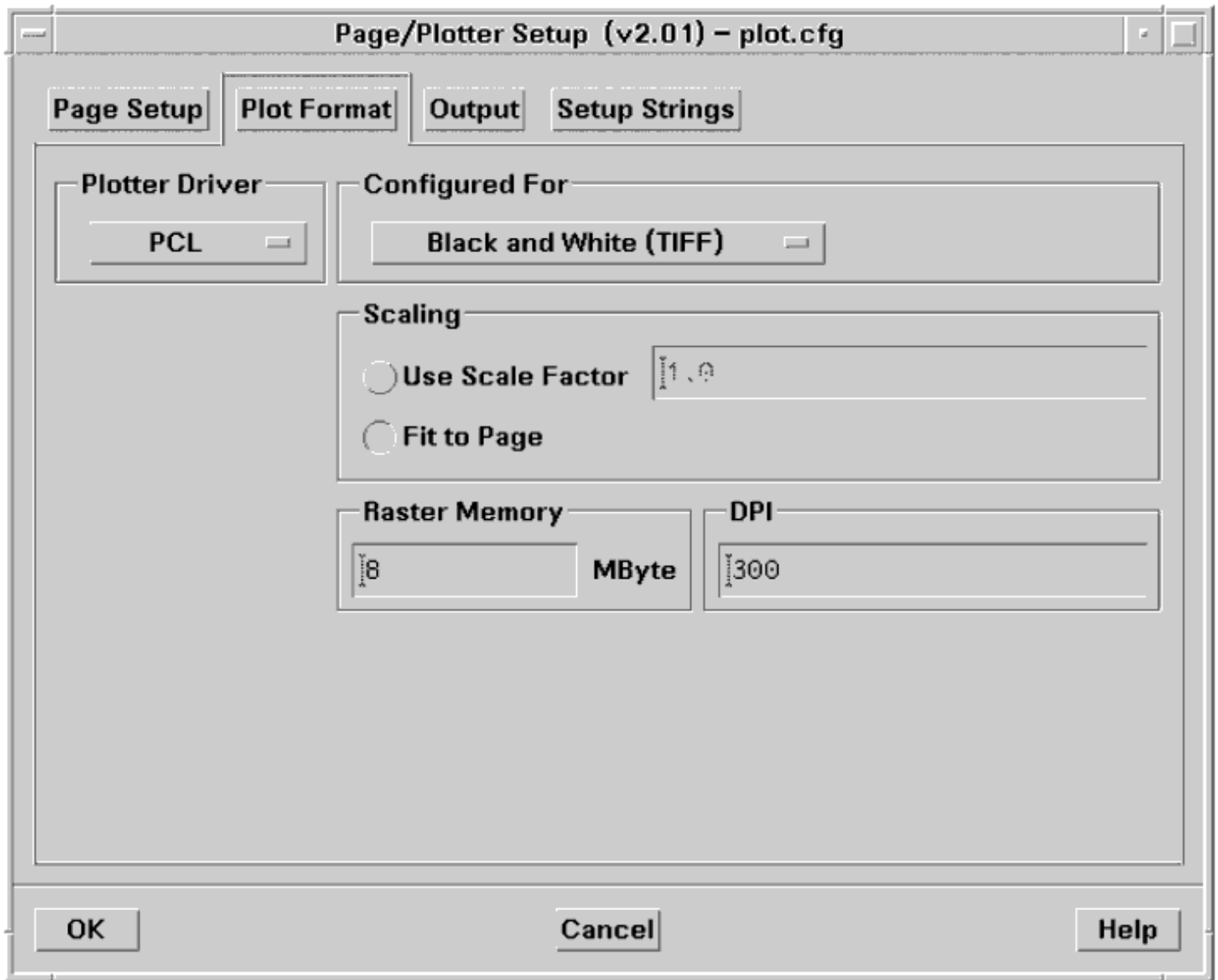
### Page/Plotter Setup

This dialog box contains four tabs: Page Setup, Plot Format, Output, and Setup Strings. The options available from each tab are as follows. To access this dialog box, select Plot > Plot/Page Setup

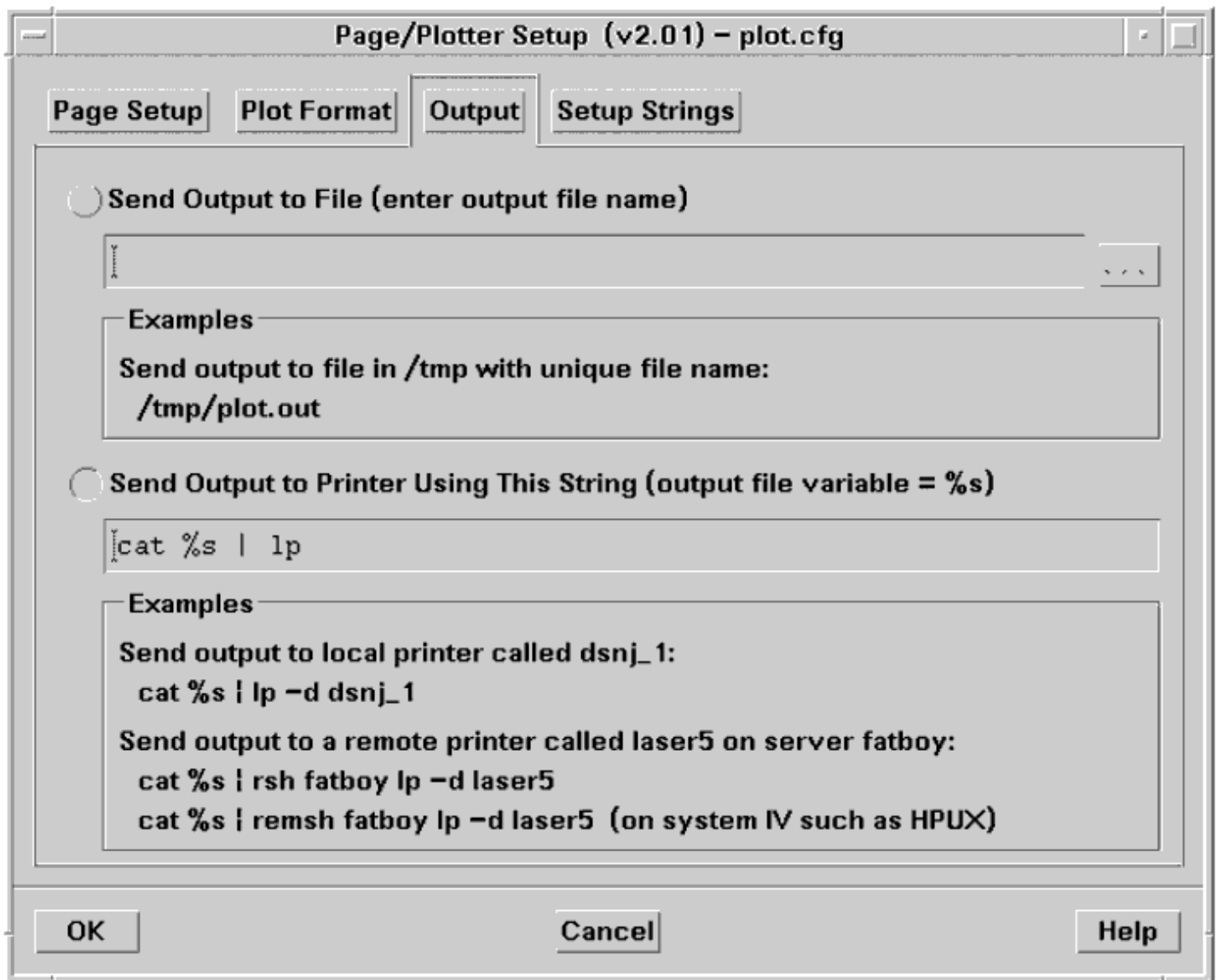
Page Setup. This tab provides access to page size, margin values, and orientation options.



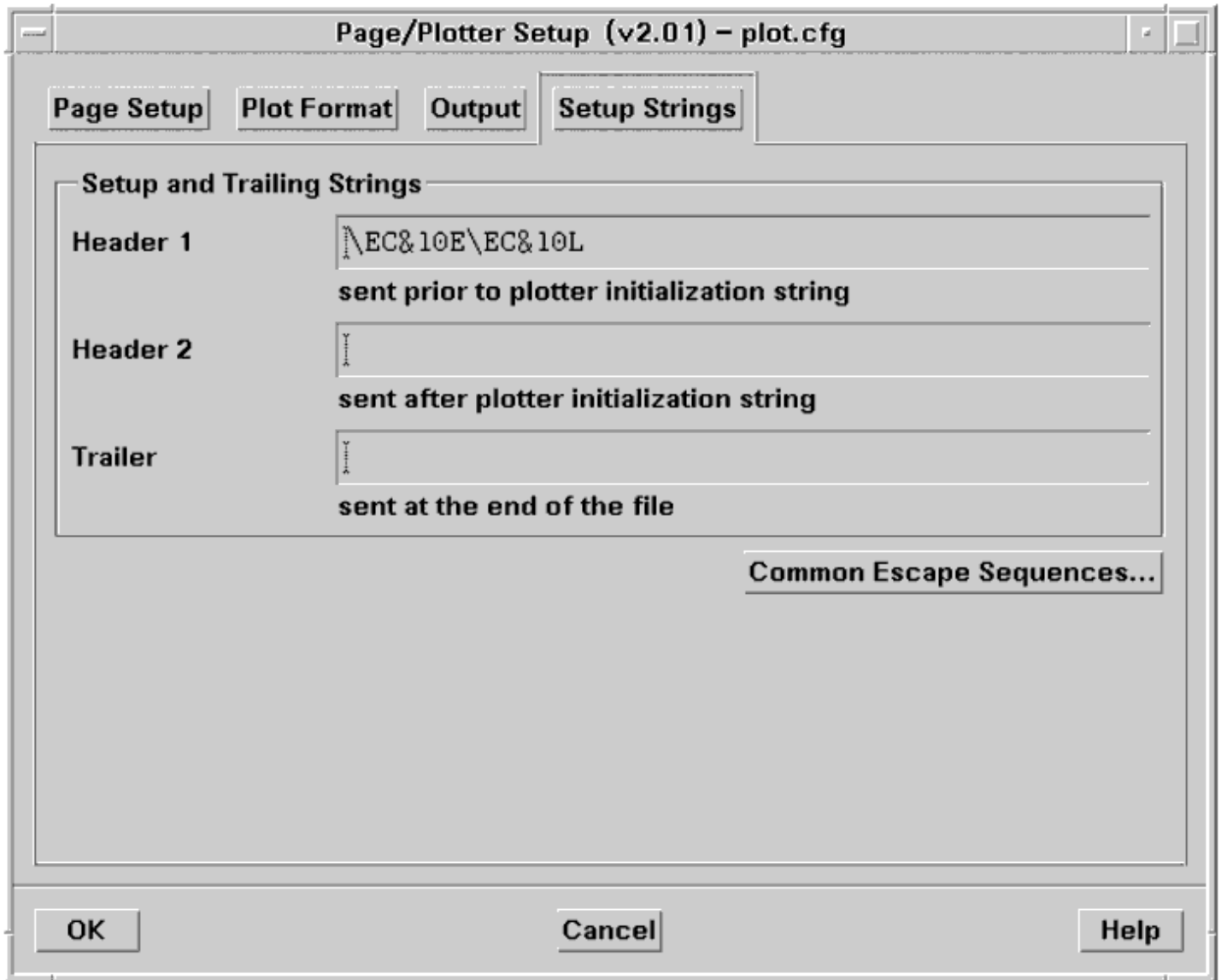
Plot Format. This tab provides access to the plotter driver type, print type, scaling factor, raster memory size (in MBytes), and dpi (dots per inch).



Output. This tab enables you to select the output file name and designate a printer path.



Setup Strings. This tab enables you to set up headers and trailers.



## Gerber Viewer Keyboard Commands

You can access many Gerber Viewer commands either through the mouse or the keyboard. In fact, a few commands are available only through the keyboard.

Key	Action
Esc	Cancels command or moves up one level in the menu structure.
.	Toggles the grid on and off.

+	Zooms in 2x.
-	Zooms out 2x.
Ins	Same as Pan command.
Home	Zooms to extents. Same as All command.

## Configuring the Gerber Translator for Photoplotters

### Types of Photoplotters

There are two types of photoplotters: vector and raster.

#### Vector Photoplotters

Vector photoplotters process each draw and flash command directly from the Gerber database. These are normally mechanical plotters with an X-Y table, a light head, and an aperture wheel. Examples of vector plotters include the Gerber 3200 and 4000 series flatbed plotters.

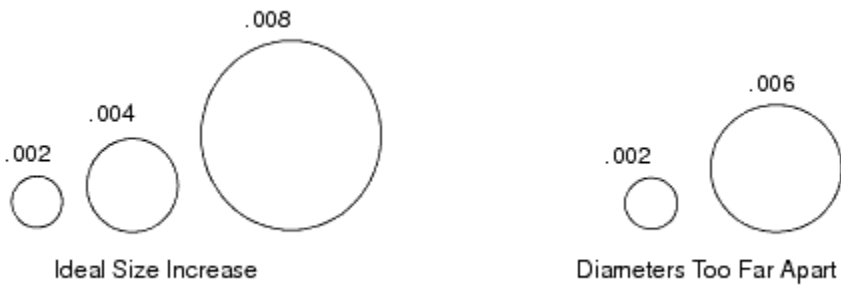
#### Raster Photoplotters

Raster photoplotters rasterize the input Gerber data using a computer, and then use the resulting bitmap to modulate a laser that is scanned across the film. What is interesting about raster plotters is that many of them can accept polygons in addition to draws and flashes. The ability of a photoplotter to fill a polygon is extremely useful to the microwave and RF designer. Examples of powerful raster plotters that support polygons include the Gerber Crescent family of plotters and the Cymbolic Sciences family of FIRE 9000 plotters.

### Vector Plotter Configuration

The mask file input to the Gerber translator is essentially a collection of polygons that need to be filled. Therefore, when you run the translator you should:

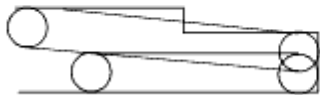
- Set the Translation Settings option Outline/Fill to FILL.
- Have a reasonable selection of round apertures available to fill. The smallest aperture diameter should be at least 2-3 times less than your smallest line width, otherwise you may not get a good representation of your design.



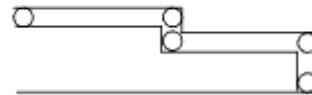
Typical D-code diameters range from 0.001 inch up to 0.200 inch. Most mechanical photoplotters support up to 24 D-codes.

The Gerber translator uses an advanced multi-aperture fill instead of what we call a pen plotter fill. The multi-aperture fill generates Gerber files with the same resolution as a pen plotter fill, but creates data files 5-10 times smaller.

Aperture Too Large to Resolve Small Feature



Aperture Smaller than Critical Feature

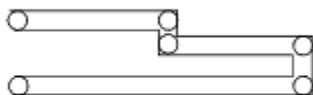


### Polygon Filling Rules

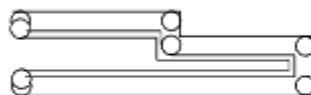
Each polygon is filled independently of any other polygon in the mask file.

Any arcs that are part of a polygon boundary are broken into segments using the ArcRes parameter. This parameter is the number of degrees per segment. The default of ArcRes is 9 degrees. If you need smoother arcs in your film, reduce the number to 6 or even 4 degrees.

The routines start filling at the inner edge of the polygon with the smallest aperture (the one you specify as the Start Aperture in the translation configuration menu). This aperture is normally used twice and is offset from the edge of the polygon by 1/2 diameter (see A below). The first two strokes overlap by 1/2 diameter (see B below).



A



B

The routines then select a larger aperture (but normally no larger than 2 times the starting aperture) and repeat this process.

The interior of most large polygons is scan-filled with a fat aperture. There is no overlap between strokes once the routines jump into scan-fill mode.

### Empty Polygons

Because the Gerber translator fills each polygon as it encounters it in the data stream, when it encounters an empty polygon it cannot clear away areas already filled. Therefore, for vector photoplotters, avoid using empty polygons in your layout. (This limitation is not in effect for some raster plotters.)

The translator issues a warning in the log file when it does encounter an empty polygon so that you do not accidentally plot over it.

If you have used empty polygons in your design, you may select the Advanced Design System Gerber export option Auto Merge , enabling you to merge filled and empty polygons to form a single filled polygon. This creates what we call a re-entrant polygon and is supported by the Gerber translator. However, while we recommend this function for relatively simple structures, we do not recommend you use such a function where hundreds of drill holes must appear in a power plane.

### Compensation

Compensation works by swelling or shrinking each polygon prior to filling it. Again, because the translator views polygons independently, it cannot take into account spatial relationships between polygons. If you attempt to use a shrink compensation with butting polygons, a narrow gap will form between them.

### FIRE 9000 Photoplotter Configuration

The Symbolic Sciences FIRE 9000 photoplotter is a raster laser plotter that is ideal for creating microwave and RF artwork. Not only does this photoplotter have a very high resolution (typically 1/8 mil), but its RIP front end supports two very important extensions to standard Gerber (RS274D) data:

- POEX - external polygons (filled areas)
- POIN - internal polygons (empty areas)

Because of these high level commands, the Gerber translator can translate a mask file with empty polygons directly into a stream of POEX and POIN commands. Configuring the translator for MDA output is the only mode that supports empty figures in the mask file.

Not only does configuring the translator for MDA output eliminate the need to stroke out the interior of each polygon,

## Advanced Design System 2008

but the resulting artwork is limited only by the precision of the photoplotter. The FIRE 9000 autoplot format also embeds all Gerber format, unit, and data mode information into its header so that a separate aperture and information list is not needed.

We highly recommend (if possible) that you send your data to a photoplot or board shop equipped with a raster photoplotter such as the FIRE 9000.

The Gerber Viewer can properly view both POEX and POIN data so that you can verify the correctness of the output.

### Recommended Settings for FIRE 9000 Output

The proper settings for FIRE 9000 output are summarized in the table below. When Outline/Fill is set to OUTLINE , each mask polygon is outlined. If a polygon is filled, then it is sent to either D20 or D21; these are both assigned as POEX. If it is an empty polygon or a hole, it is assigned to D22 or D23 which correspond to a POIN. Any open mask entities are sent to D10, which is a standard round aperture.

#### Gerber Translator Settings for FIRE 9000 Output

Gerber File Options		Translation Settings		Apertures (inches)		
Option	Setting	Option	Setting	D-Code	Type	Inch (X,Y)
Unit	INCH or MM	Line DCode	d10	10	Round	0.005
Format	4.4 or 4.3	Scale Factor	1	20	Poex	0.000
Zero Suppression	Leading	Outline/Fill	Outline	21	Poex	0.000
Circular	360	Filled D-codes (POEX)	d20, d21	22	Poin	0.000
CR/LF	Suppress	Empty D-codes (POIN)	d22, d23	23	Poin	0.000
		Compensation	None			
		Output Offset	0,0			
		Gerber Output Format	MDA Autoplot			

### RS274X Output Configuration

Gerber Scientific's laser photoplotters read the extended RS274X specification. These photoplotters also support a

## Advanced Design System 2008

polygon definition. Unfortunately, however, "empty" polygons are not supported. If you use empty figures in the mask file they will be covered up.

Other photoplotters may also support the RS274X specification, but before using them you should verify that they properly support the G36/G37 command used to switch into polygon mode.

Recommended Gerber Translator Settings for RS274X Output

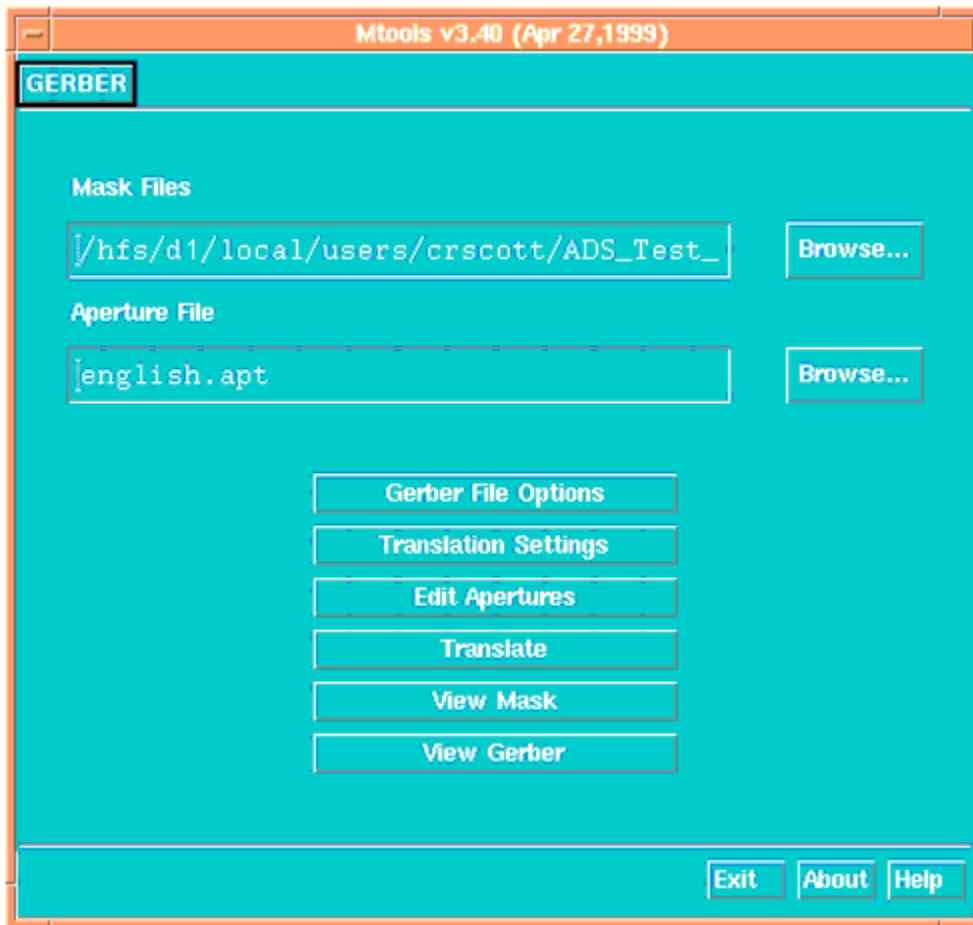
Gerber File Options		Translation Settings		Apertures (inches)		
Option	Setting	Option	Setting	D-Code	Type	Inch (X,Y)
Unit	INCH or MM	Line DCode	d10	10	Round	0.005
Format	4.4 or 4.3	Scale Factor	1	20	Poex	0.000
Zero Suppression	Leading	Outline/Fill	Outline	21	Poex	0.000
Circular	360	Filled D-codes (POEX)	d20, d21			
CR/LF	Suppress	Compensation	None			
		Output Offset	0,0			
		Gerber Output Format	RS274X			

## Creating an Excellon Drill File from an ADS Layout Using MTools

This section describes the procedure for creating an Excellon drill file from an ADS Layout. Excellon drill files define x and y coordinates for hole location and drill size. These files are used to automate the drilling process in manufacturing environments.

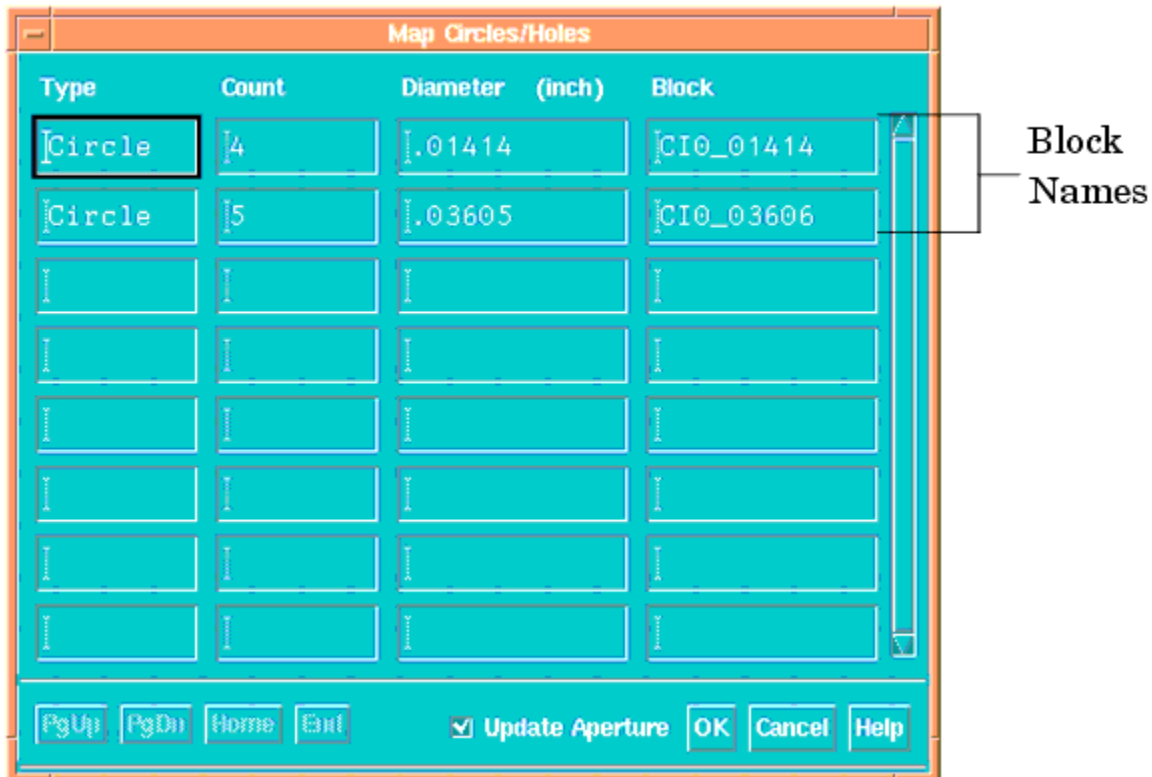
To create a drill file:

1. In the layout window containing your design, choose File > Export. The Export dialog box appears.
2. Select Gerber from the File Type drop down list then specify the file name in the New File Name (Destination) field. Click OK. The Mtools status window appears.
3. Click OK in the Mtools status window to accept the settings. The Mtools Log and Gerber Translator Interface windows appear.



Gerber Translator Interface Window

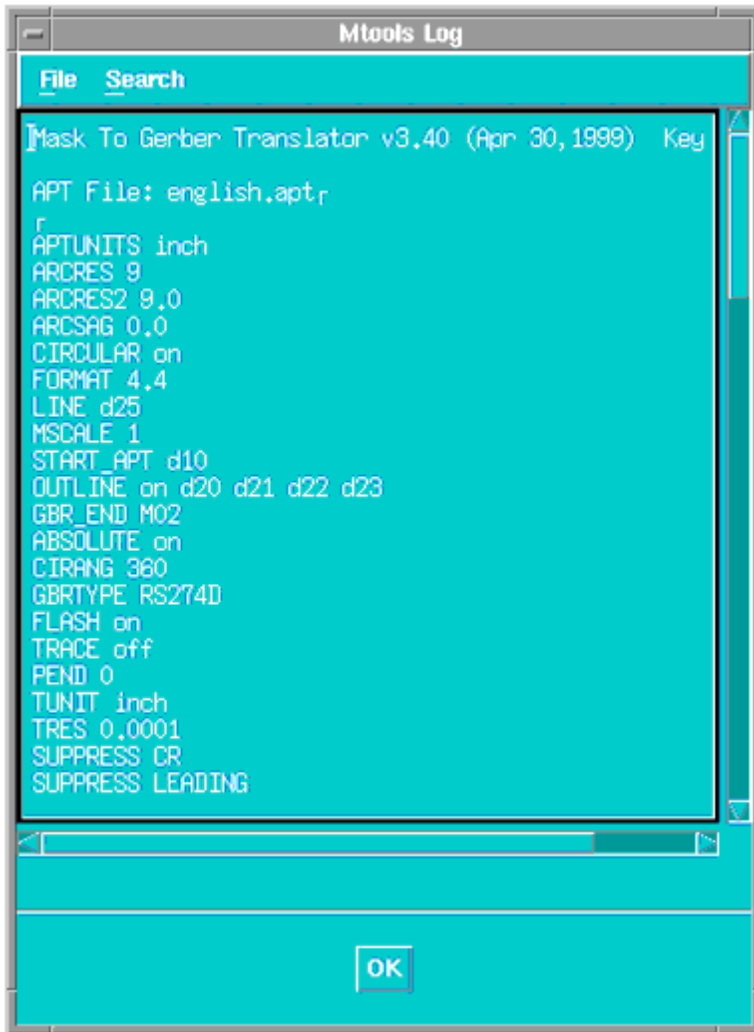
4. From the Gerber Translator Interface window, click Edit Apertures. A table listing the aperture settings appears.
5. In the Aperture dialog box, click Flash Circles. A Map Circles/Holes dialog box appears.



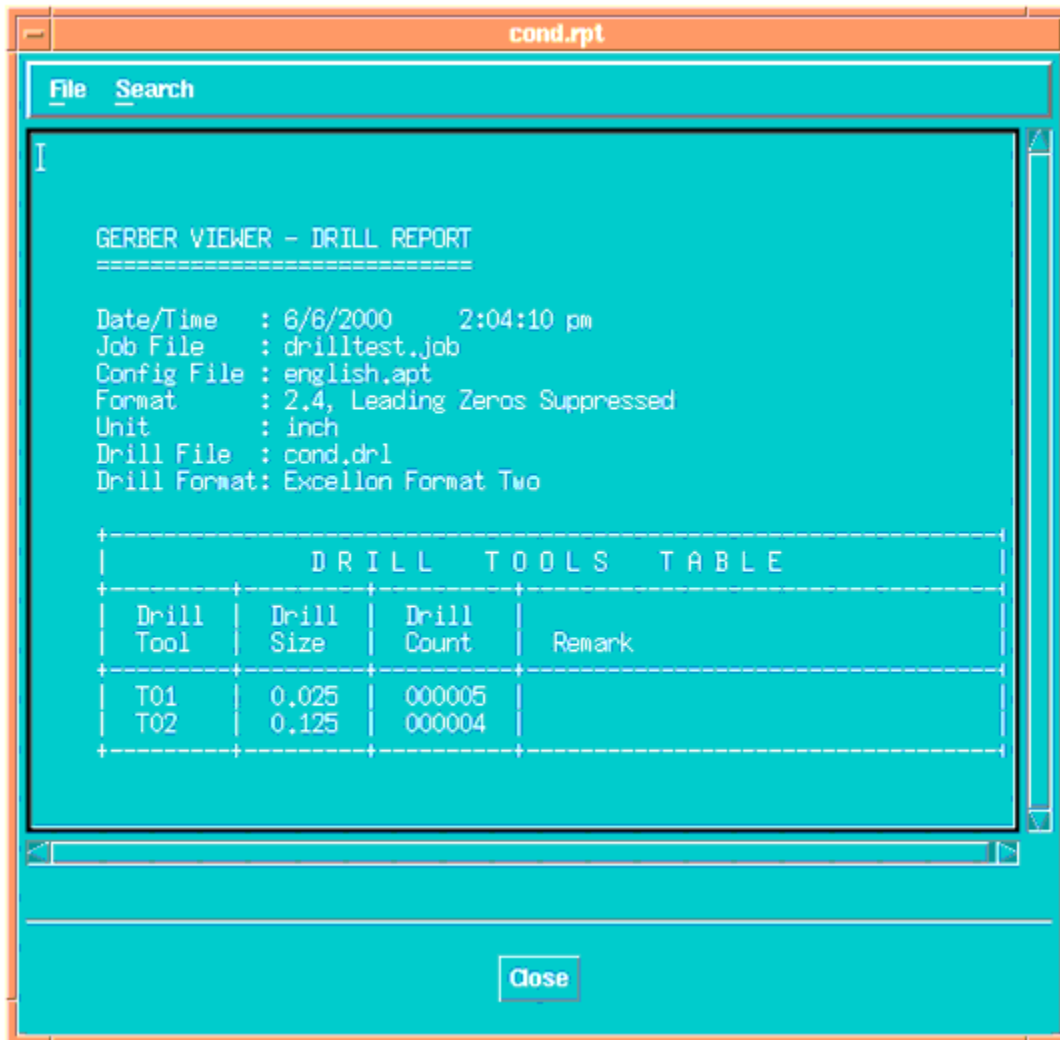
6. In the Map Circles/Holes dialog box, verify that the circle Count and Diameter for each Block are correct. Ensure that the Update Aperture box is checked. Note the Block name of each of the circles in the Map Circles/Holes dialog box. Click OK in the Map Circles/Holes dialog box.
- In the Aperture dialog box, using the information from step 6, note which D-codes have the Block names that were mapped to circles and then click Save. A small dialog box asking if you want to save the changes appears. Click OK.



- From the Gerber Translator Interface window, click Translate to select the layers you want to include in the translated file (see [Translate](#)). After selecting the layers, click OK to complete the translation. A window appears briefly, indicating that the file is being converted to Gerber format.
- When the translation is complete, the Mtools Log appears, detailing the export information. Examine the log file, searching for any warnings or errors that may have occurred during translation.



9. When you are finished viewing the log file, click OK to dismiss the MTools Log and exit the Gerber translator. You can also save the MTools Log before exiting using the File > Save As menu pick.
10. From the Gerber Translator Interface window, click View Gerber. The GBRVU dialog appears with a Gerber view of your layout.
11. From the GBRVU dialog, click Aperture. The Aperture dialog box appears.
12. In the Aperture dialog box, enter the Tool # (integer only) and Drill Dia. for each of the D-codes that were noted in step 7. Click Save. A small dialog box asking if you want to save the changes appears. Click OK.
13. From GBRVU, choose Tools > Drill > Excellon. Choose whether or not to suppress the leading zero's. The Drill Output dialog box appears.
14. Click the appropriate Drill Output and then click Report. The GERBER VIEWER - DRILL REPORT file is displayed. This file contains a DRILL TOOLS TABLE that lists tool numbers, tool size, quantity and remarks.  
Example Drill Report:




15. Use any ASCII text editor to open and view the drill file (file extension .drl) stored in the current ADS project directory.

Example Drill File:

```

M48
INCH,TZ
VER,1
FMAT,2
DETECT,ON
%
M72
G05
T01C.025
X-03000Y001000
X-03050Y-00500
X-01500
X002000
    
```

X001500Y000000  
T02C.125  
X-01500Y001000  
Y000500  
X-01000  
Y001000  
M30

 **Note**  
Both the drill report and drill file are created in the ADS project directory with the layer name as a prefix and .rpt and .drl as suffixes.  
Example: layer\_name.rpt is the drill report and layer\_name.drl is the drill file.

## 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](#). 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, Precision, Scale, Attributes, Etch Factor and Miter Angle as well as access the Layer Editor and specify layer options.



### 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](#).

### 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  $-3$ .

### Define Layers

The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

## 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

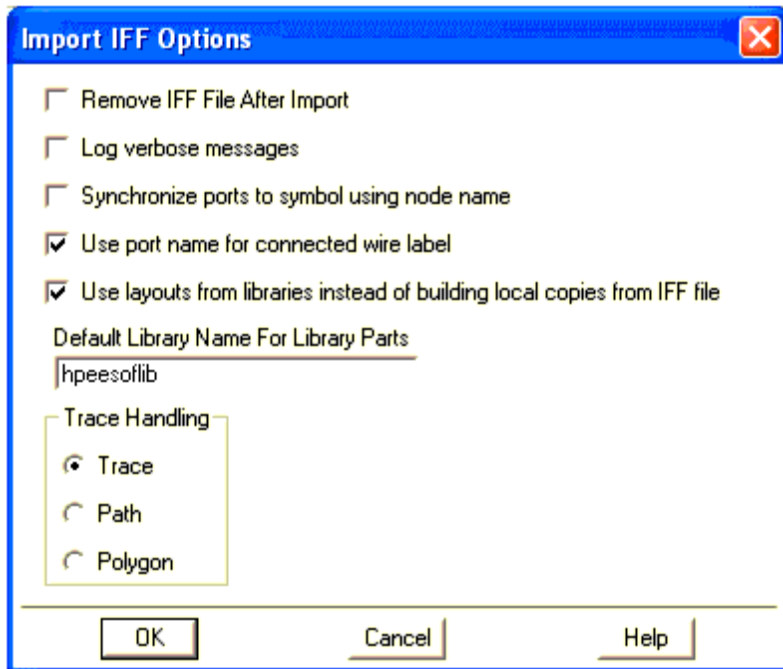
IFF files are used as the exchange mechanism when transferring designs between Advanced Design System and third-party EDA tools such as Mentor Graphics Design Architect and Cadence Analog Artist.

### 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](#). 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. If the import is invoked from the ADS Main window, the following Import IFF More 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 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 project.

### 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 manual. Creating an ADS component library using IFF requires specialized tools and training. If you're interested in learning more about this topic, contact Agilent EEsof-EDA's Solution Services.

### 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.

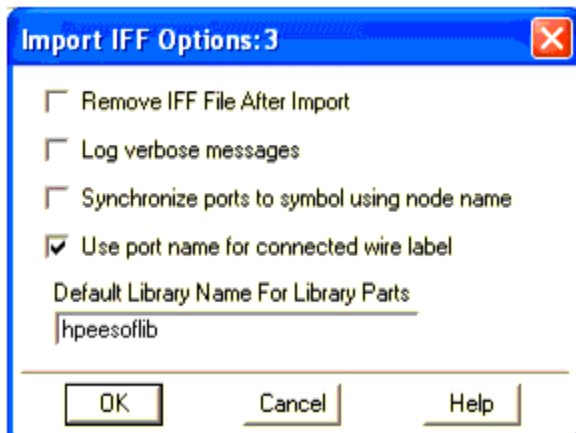
**Path** When this option is selected, meander elements are translated as primitive data type with center line and width. The default for this option is deselected.

**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 More Options dialog box appears.



Note that the same options are used with the exception of 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](#).

### Exporting IFF Files

To export an IFF file:

1. Follow the steps as outlined in [Exporting a Layout](#). 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:

```

Hpeesofedit: [iffexport.log]
File Edit Search
[iffexport.log
HP IFF translation ($Revision: 100.63 $)

Output format: HP Intermediate File Format
Output filename: test.iff

No export options defined - using default values.

Export Options:
  DefaultLibraryPath: hpeesoflib
  The IFF file will be appended to.
  Export IFF Version 2

Reading design /hfs/hpsrwip/d1/users/tracyjs/ads/ModemTimed/networks
Finished writing layout /hfs/hpsrwip/d1/users/tracyjs/ads/ModemTimed

Hpeesofedit (TM) 1.0 100.400 1, 1

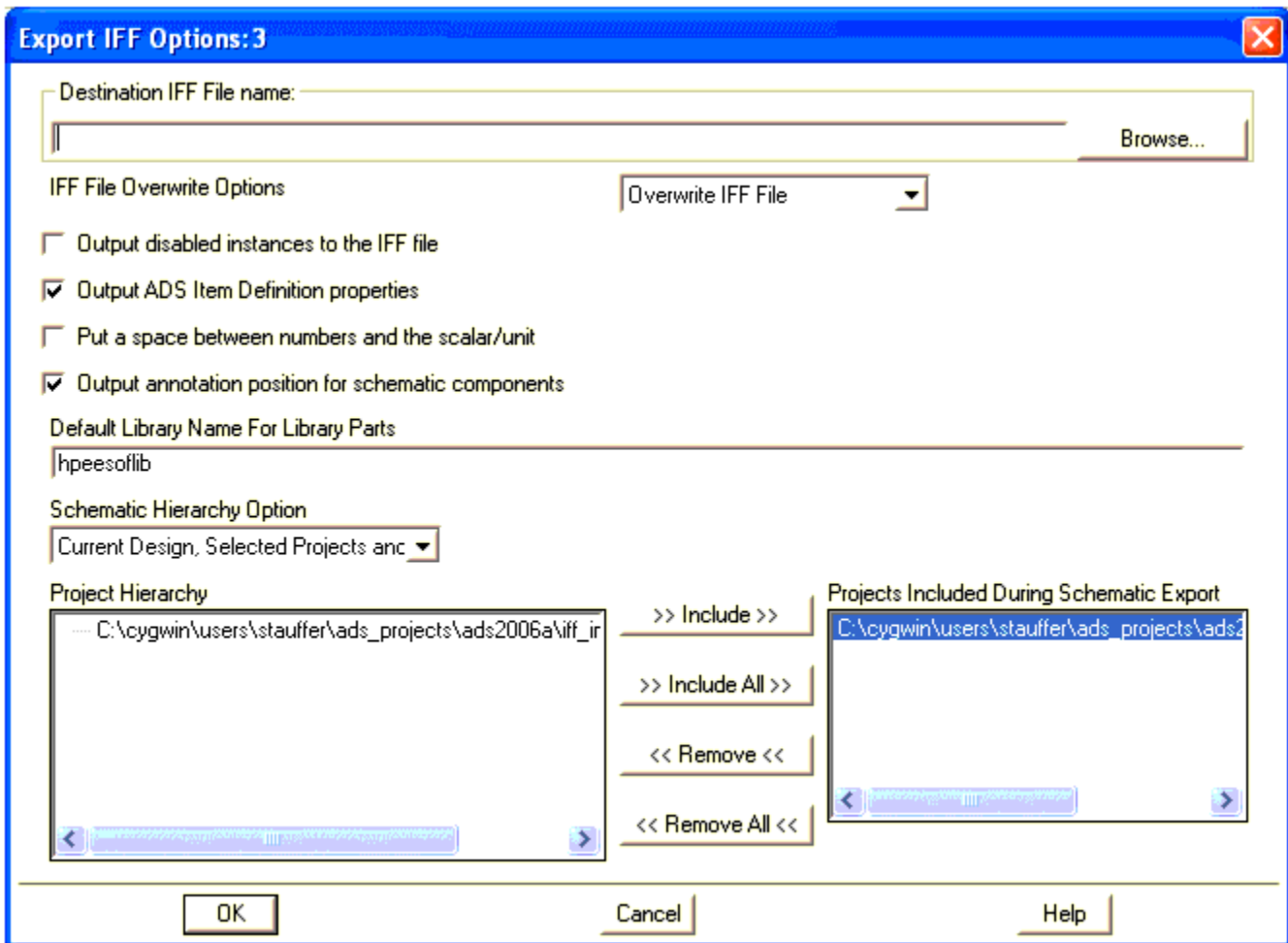
```

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

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

- To dismiss the log window, choose File > Quit.

## 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 manual. Creating an ADS component library using IFF requires specialized tools and training. If you're interested in learning more about this topic, contact Agilent EEsof-EDA's Solution Services.

### Schematic Hierarchy Option

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

**Current Design Only** Write current level only. Complete design information for the current design is exported.

**Instance-specific information** (parameter values and coordinates identifying position) is also exported. Detailed definitions of a referenced design are not exported.

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

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

### Project Hierarchy

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

### Projects Included During Schematic Export

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

To add a project to this list:

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

To include all projects, click Include All.

To remove a project from the Projects Included list:

1. In the Projects Included list, click the entry you want to remove.
2. Click the Remove button. The project is removed from the list.

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

## 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 chapter.

## 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](#). For specific import options related to importing IGES Files, refer to [Import IGES Options](#).

## Import IGES 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.

## Define Layers

The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

Note that the layer-to-IGES level number is controlled in the Layer Editor IGES Layer entry field for each program layer.

## 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](#). For specific export options related to exporting IGES Files, refer to [Export IGES Options](#).

## Export IGES Options

**Export IGES Options:3**

**IGES Extension**  
igs Define Layers...

**Units**  
same Addnl. Scale Factor  
1

**Exponent Character**  
E Precision  
-2

**Author**  
.  
**Organization**  
.

**Attributes**

- Create Top
- Include Fill
- Include Text
- Holes As Polygons
- Flatten
- Reverse Arcs
- Conv Arc to Lines
- Polygons As Open Polylines
- Polygons As Segments
- Explicitly Define Defaults
- ZeroFillSequenceNumbers
- ZeroFillTerminateSection
- Name Properties

OK Cancel Help

### 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](#) in the Customization and Configuration manual.

### 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.

### Define Layers

The Define Layers button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

### 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

## Advanced Design System 2008

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 file 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

## Advanced Design System 2008

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. This option is deselected as the default.

### 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 sequential 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.

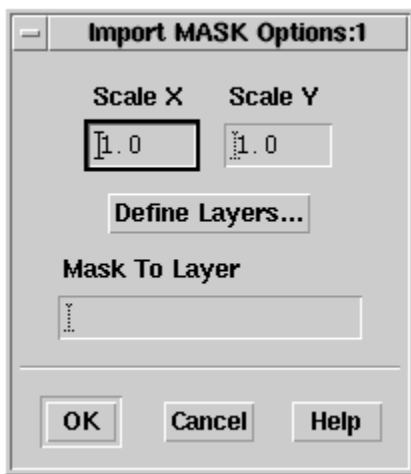
The Gerber and DXF translators use the mask file format as an intermediate file when converting data to Gerber and DXF. The mask format is also used as an intermediate file when translating other formats such as HPGL/2. In addition, the simplicity of the mask file format makes it an appealing option 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](#).

#### Import MASK Options



#### 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.

#### Define Layers

This button invokes the Layer Editor. For Layer Editor options, see [Defining Layers](#).

### Mask To Layer

This field enables you to specify a list of mask files to be read and the layer numbers to be created within them. The information must be presented in pairs, as follows:

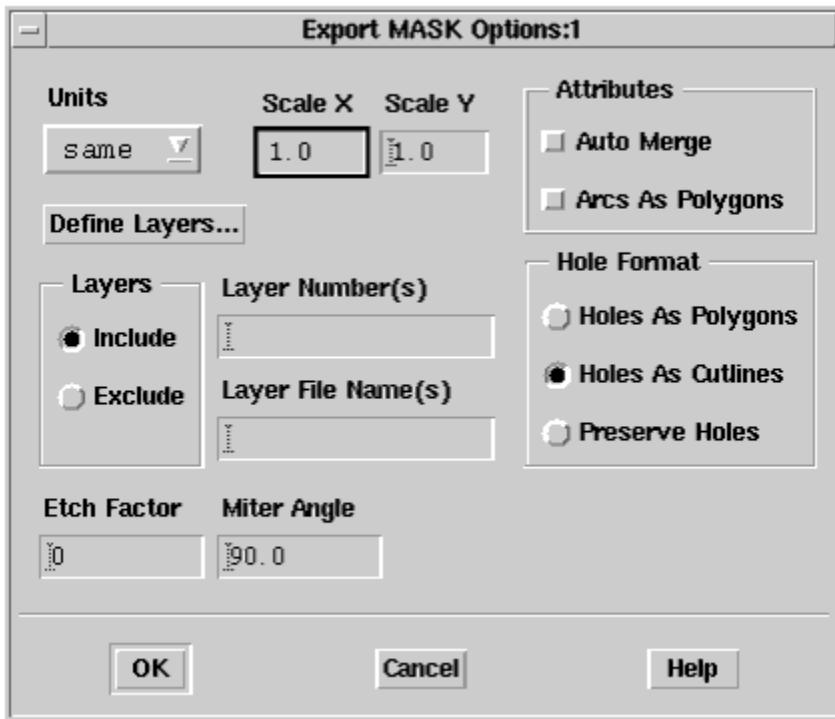
<layer\_number> <file\_name> <layer\_number> <file\_name>...

where all entries are separated by spaces.

### Exporting Mask Graphics Files

For a step-by-step tutorial, see [Exporting a Layout](#).

### Export MASK Options



### Units

These are the units that the mask 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 Layout Options > Preferences for Layout > Setting Units/Scale Factors in [Customization and Configuration](#).

### 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.

### Define Layers

Clicking the Define Layers button invokes the Layer Editor. For Layer Editor Options, see [Defining Layers](#).

### Layers Include, Layers Exclude

These buttons enable you to specify layers to either include or exclude.

- Include – exports information only from layers specified in the Layer Number(s) and Layer File Name(s) fields (see descriptions below).
- Exclude – exports all layers except those specified in the Layer Number(s) field (it is not necessary to fill in the Layer File Name(s) field).

Include is selected as the default. Unless you specify layer numbers to include in the Layer Number(s) field, all layers are automatically included.

### Layer Number(s)

The layer numbers to include or exclude in the mask file output. Entries are separated by commas. For example: 1, 6, 20.

### Layer File Name(s)

This option is used only when Layers Include is selected. The information must be presented in pairs, as follows:

## Advanced Design System 2008

<layer\_number> <layer\_name> <layer\_number> <layer\_name>...

where all are separated by spaces. For example:

1 msk1 3 msk3

### 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.

### 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.

### 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. 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.

### Etch Factor

The etch factor applies a global over/undersize amount to each shape translated. This is meant to compensate for etch effect during processing. However, using this option can be problematic. Thus, we recommend that you retain the default setting of 0.

If you use Etch Factor, carefully verify the correctness of the compensation to minimize problems. Limitations include the following: When a figure has a side smaller than the etch factor, this function may fail. If two boundaries butt up against one another before compensation, because each boundary is handled independently, such boundaries will either overlap or show a gap when compensation is specified. When Etch Factor is applied, re-entrant polygons may be transformed into illegal polygons.

### Miter Angle

The angle cutoff used with the etch factor. The miter angle controls acute angle edge over-extension. Any angle below the miter angle amount is mitered. The default is 90.0.

## 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 project 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 be 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](#). 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 following message window appears:



Click OK to dismiss this window.

4. The IFF Export log appears:

```

Hpeesofeedit: [iffexport.log]
File Edit Search
[iffexport.log
HP IFF translation ($Revision: 100.63 $)

Output format: HP Intermediate File Format
Output filename: test.iff

No export options defined - using default values.

Export Options:
  DefaultLibraryPath: hpeesoflib
  The IFF file will be appended to.
  Export IFF Version 2

Reading design /hfs/hpsrwip/d1/users/tracyjs/ads/ModemTimed/networks
Finished writing layout /hfs/hpsrwip/d1/users/tracyjs/ads/ModemTimed

Hpeesofeedit (TM) 1.0 100.400 1, 1

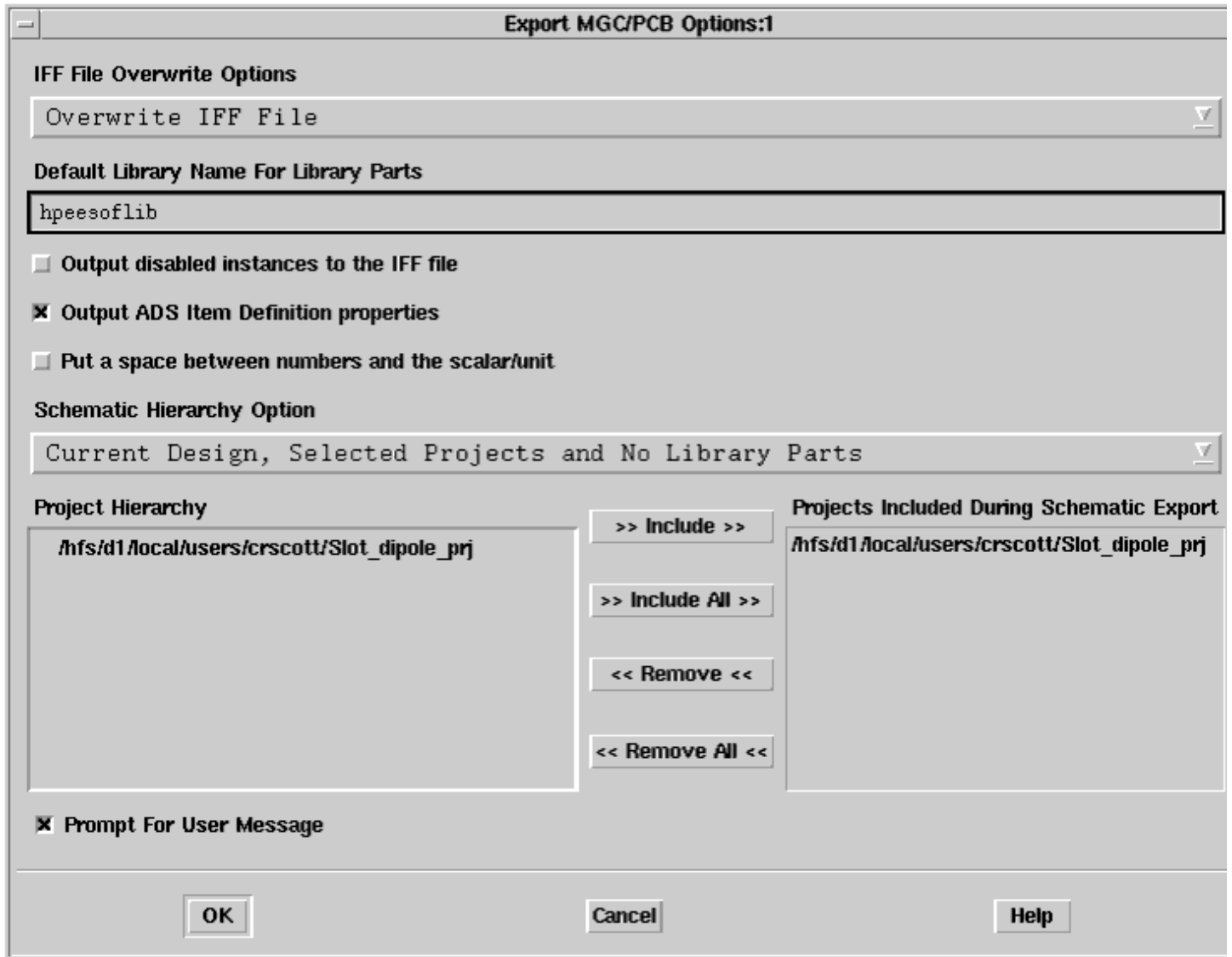
```

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.

- To dismiss the log window, choose File > Quit.

## 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

## Advanced Design System 2008

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 project 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

## Advanced Design System 2008

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

### Project Hierarchy

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

### Projects Included During Schematic Export

The projects for which design information will be exported. You may customize this list if the current project is hierarchical. (Note that complete layout hierarchy is always exported.)

To add a project to this list:

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

To include all projects, click Include All.

To Remove a project from the Projects Included list:

1. In the Projects Included list, click the entry you want to remove.
2. Click the Remove button. The project is removed from the list.

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

### Prompt For User Message

## Advanced Design System 2008

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.