Agilent Technologies

Advanced Design System 2002

Importing and Exporting Designs

February 2002

Notice

The information contained in this document is subject to change without notice.

Agilent Technologies makes no warranty of any kind with regard to this material, including, but not limited to, the implied warranties of merchantability and fitness for a particular purpose. Agilent Technologies shall not be liable for errors contained herein or for incidental or consequential damages in connection with the furnishing, performance, or use of this material.

Warranty

A copy of the specific warranty terms that apply to this software product is available upon request from your Agilent Technologies representative.

Restricted Rights Legend

Use, duplication or disclosure by the U. S. Government is subject to restrictions as set forth in subparagraph (c) (1) (ii) of the Rights in Technical Data and Computer Software clause at DFARS 252.227-7013 for DoD agencies, and subparagraphs (c) (1) and (c) (2) of the Commercial Computer Software Restricted Rights clause at FAR 52.227-19 for other agencies.

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

Contents

Chapter 1: Introduction

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. Table 1-1shows the available file formats and the individual import/export options.

Table 1-1. Available Formats and Import/Export Options

Available File Formats and File Extensions	ADS Main Window	ADS Schematic Window	ADS Layout Window
DXF (.dxf)	-	-	Import/Export
EGS Archive (_a)	-	-	Import/Export
EGS Generate (_g)	-	Import	Import/Export
GDSII Stream (.gds)	-	-	Import/Export
Gerber (.gbr)	-	-	Export
Gerber Viewer (.msk, .gbr)	-	-	Export
HPGL/2 (.hpg)	-	Import	Import/Export
IFF (.iff)	Import	Import/Export	Import/Export
IGES (.igs)	-	-	Import/Export
Mask File (.msk)	-	Import	Import/Export
MGC/PCB (.iff)	-	-	Export
SPICE (.cir, .cki)†	Import	Import	-
Spectre (.scs)††	Import	Import	-

† For detailed information on SPICE file import, refer to "Importing a SPICE File" in the ADS Netlist Translator for SPICE documentation.

†† For detailed information on Spectre file import, refer to "Importing a Spectre File" in the ADS Netlist Translator for Spectre documentation.

Steps for importing and exporting these files are outlined in Chapter 2, Importing and Exporting Schematic and Layout Designs.

DXF

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. For details, see Chapter 3, DXF 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 Chapter 4, EGS Archive Files and Chapter 5, EGS Generate Files.

GDSII Stream

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

Unlike other data formats, GDSII stream format is binary. You cannot easily view or edit a stream format file using a text editor. This format is easily translated between different CAD systems because it represents a highly restrictive data type. However, the format has a number of significant limitations; these limitations are discussed in Chapter 6, 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 Chapter 7, Gerber Artwork Translator and Gerber Viewer.

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” on page 7-18).

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. When creating a graph or chart in another tool, you can write the graphics data to an HPGL/2 output file, then import the file into Advanced Design System.

In Advanced Design System, the HPGL data is transformed into forms and shapes that can be edited and manipulated like any other drawing. Additional text, annotation, scaling or editing may be added. For details, see Chapter 8, 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 Chapter 9, 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 Chapter 10, 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 Chapter 11, 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 export 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.

SPICE

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.

For detailed information on SPICE file import, refer to "*Importing a SPICE File*" in the ADS *Netlist Translator for SPICE* documentation.

Spectre

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 Spectre file import, refer to "*Importing a Spectre File*" in the ADS *Netlist Translator for Spectre* documentation.

Chapter 2: 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.

Importing a Schematic

In Advanced Design System, you can Import a Schematic in these formats:

- DXF
- EGS Archive Format
- EGS Generate Format
- Gerber
- Gerber Viewer
- HPGL/2
- IFF
- IGES
- Mask File (.msk)
- MGC/PCB
- SPICE (See *Importing a SPICE File* in the *ADS Netlist Translator for SPICE* documentation.)
- Spectre (See *Importing a Spectre File* in the *ADS Netlist Translator for Spectre* documentation.)

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

In Advanced Design System, you can Export a Schematic in this format:

- IFF

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" on page 9-1.

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

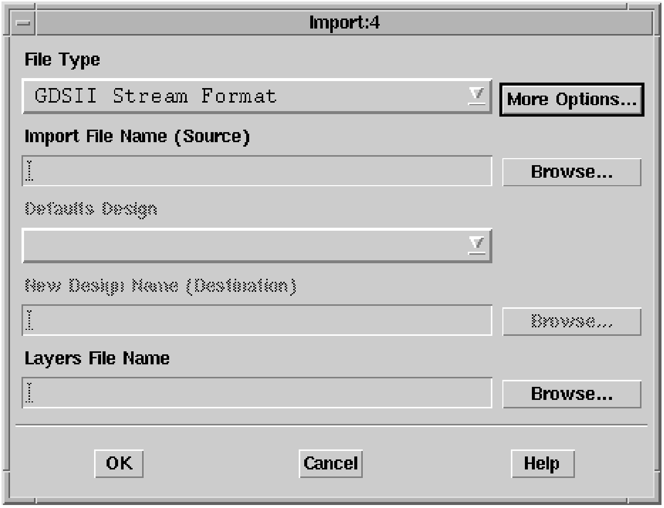
In Advanced Design System, you can import files in these formats:

- EGS Archive Format
- EGS Generate Format
- GDSII Stream Format
- HPGL/2
- IFF
- IGES
- Mask File (.msk)

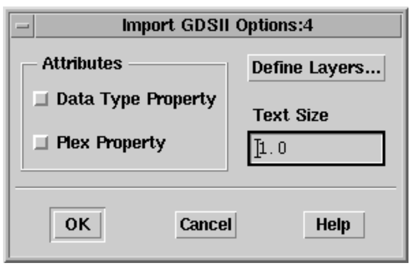
The procedure for importing each format is the same, but the available options differ. For options relating to a particular file format, see the appropriate chapter.

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.

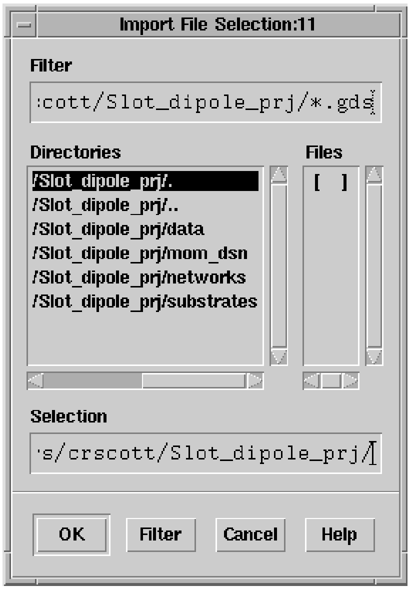


Note Each format has its own set of import options. Refer to the appropriate chapter in this book.

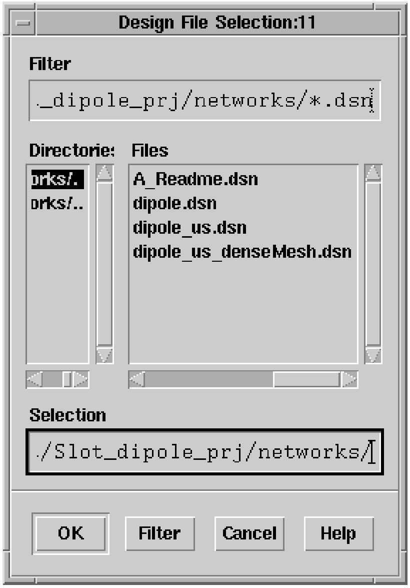
---

4. Click **Define Layers** to set the layer options. For information on the Layer Editor, see “Defining Layers” on page 2-11. 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.

6. 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.



7. 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.



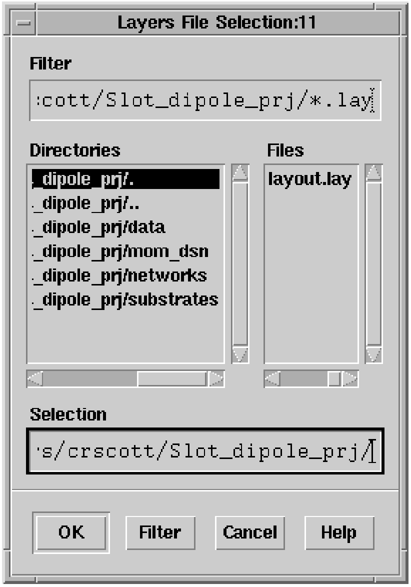
Note *New Design Name (Destination)* cannot be specified for EGS Archive Format, GDSII Stream Format, IFF, or IGES files.

---

8. 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

In Advanced Design System, you can export files in these formats:

- DXF
- EGS Archive Format
- EGS Generate Format
- GDSII Stream Format
- Gerber

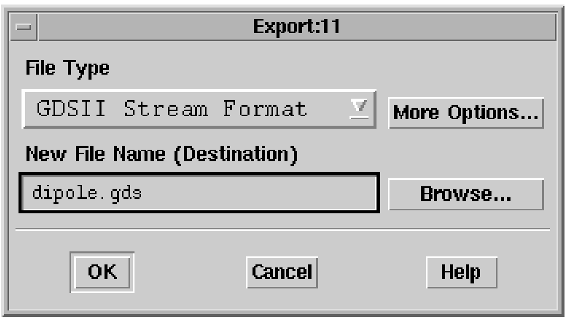
- HPGL/2
- IFF
- IGES
- Mask File (.msk)
- MGC/PCB

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.

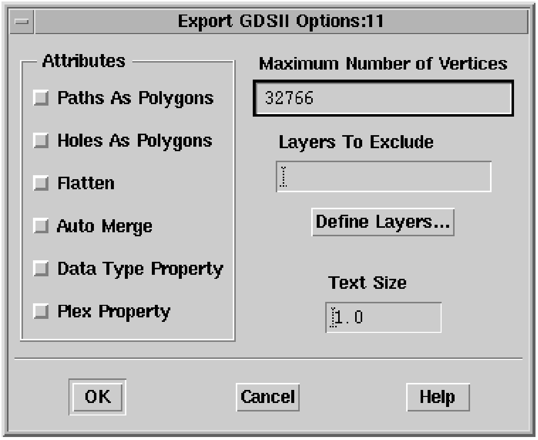
Note Gerber Viewer appears as an export file option. It is not a file format, but is placed on the export file menu so you can open the Gerber Viewer at any time. In addition, you can access the Viewer during a DXF or Gerber export (see “Using the Gerber Viewer” on page 7-18).

To export a layout file:

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



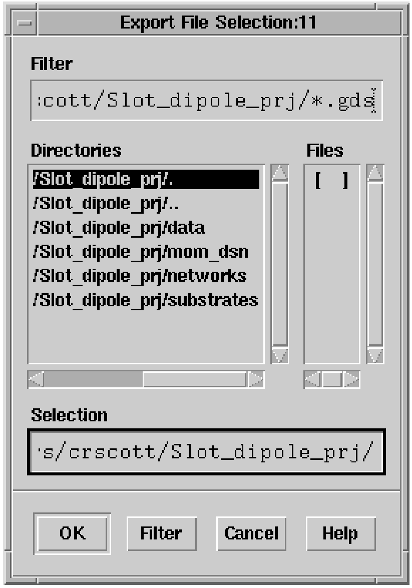
2. In the **File Type** field, click the arrow in the right-hand corner to display a drop down list of available formats. Select the format you want to export.
3. Click the **More Options** button to define the options for your selected format.



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" on page 2-11.
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.



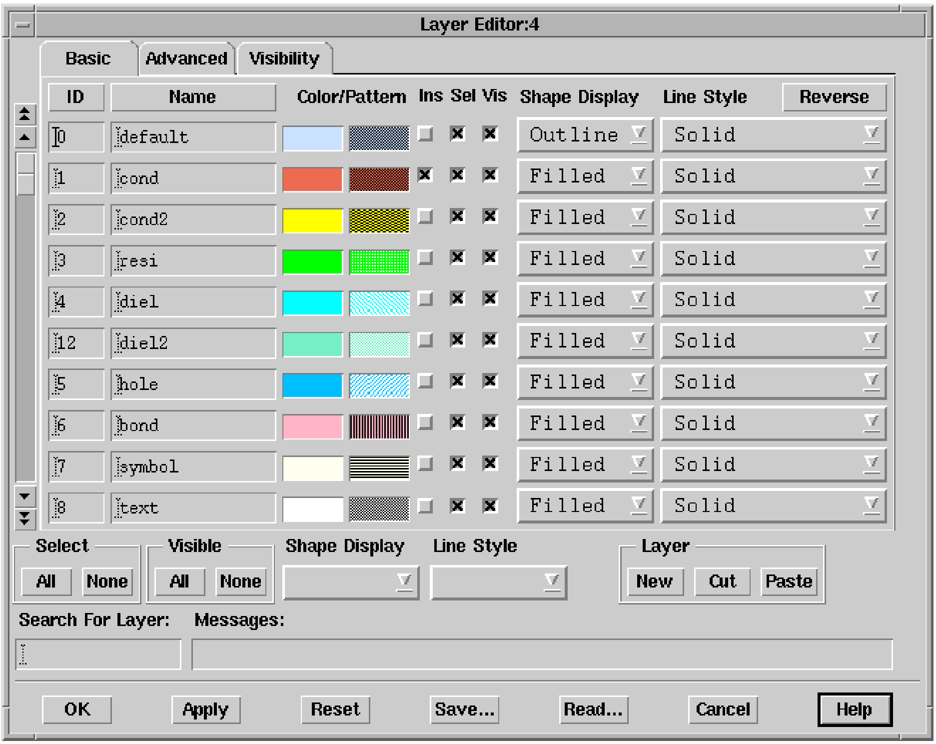
Note *New File Name (Destination)* cannot be specified for MGC/PCB files. MGC/PCB files are written to the same predetermined location each and every time. For more information on this file type, see Chapter 12, MGC/PCB Files.

---

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 “*Layer Editor*” in the ADS “*Layout*” manual.



Chapter 3: DXF Translator

The DXF translator enables you to convert ADS designs into AutoCAD's DXF file format as well as convert DXF files into ADS designs. DXF is a very simple file format and can be read by most any CAD program that supports DXF. Advanced Design System supports import from and export to AutoCAD versions 12, 14, and 2000.

How you want to use DXF output—including layer numbering, use of holes, and polygon shapes—should be considered before beginning your layout design. Setting up the proper layout rules in ADS can save a lot of time in generating acceptable DXF output. For specific considerations or limitations, consult your AutoCAD documentation.

Advanced Design System provides two different DXF translators, a hierarchical translator and a flattened translator. The *DXF (hierarchical)* translator is a bidirectional translator that supports both full hierarchy and all layer separation. The *DXF (flattened)* translator supports export only with limited capability.

Importing DXF (hierarchical) 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" on page 2-3. For specific import options related to importing DXF files, refer to "Import DXF (hierarchical) Options" on page 3-2. For information on mapping DXF files to EGS Archive files to ADS files, refer to "Mapping DXF to ADS" on page 3-7.

In order to import a DXF file, you must use the *DXF (hierarchical)* translator as opposed to the *DXF (flattened)* translator. When importing files using the *DXF (hierarchical)* translator, the program first accesses the DXF file. This file is then translated into EGS Archive file format as shown in Figure 3-1. The file can then be opened up in an Advanced Design System Layout window.

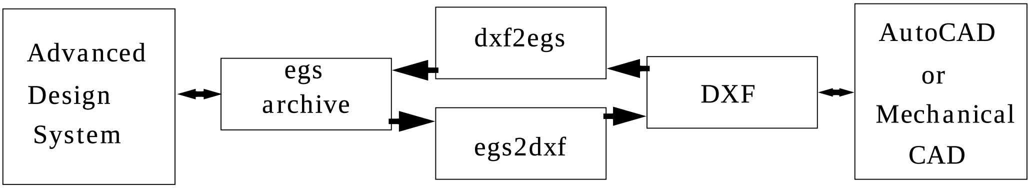


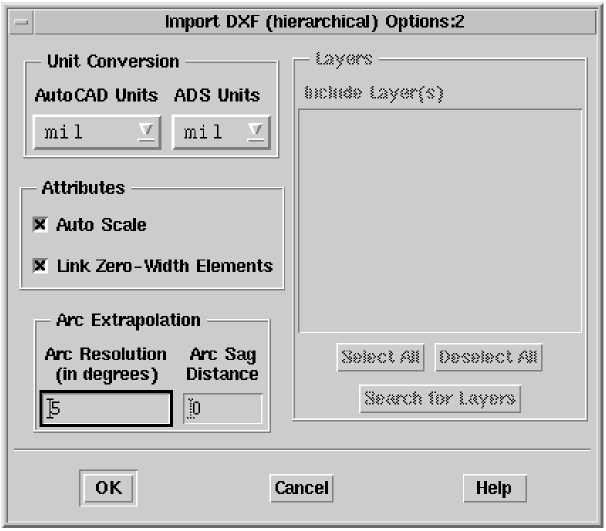
Figure 3-1. DXF (hierarchical) Block Diagram

Import DXF (hierarchical) Options

This section describes the definable options available for importing files in the DXF hierarchical format.

File > Import > DXF (hierarchical) > More Options

The *Import DXF (hierarchical) Options* dialog box enables you to specify *Unit Conversion*, *Attributes*, *Arc Extrapolation* and *Layers* to control the import of DXF format files.



Unit Conversion

Unit conversion must be specified for both the DXF file and EGS Archive file. The *AutoCAD Units* (DXF file) and *ADS Units* (EGS Archive file) drop-down menu selections display the available units. These units are described in Table 3-1.

Table 3-1. Available Units

Unit	Meaning
um	micrometer
mm	millimeter
cm	centimeter
mil	mils
in	inches

AutoCAD Units

Specifies the DXF file's units. This information is *not* included in the DXF file. The default is *mil*.

ADS Units

Specifies the EGS Archive file's units. The default is *mil*.

Attributes

The Attributes section enables you to set an auto scale and/or the ability to link zero-width elements.

Auto Scale

The Auto Scale attribute is used to automatically scale each length in the DXF file to the specified ADS units. The default is active.

Link Zero-Width Elements

The Link Zero-Width Elements attribute is used to merge line segments into polylines or polygons. (A polygon is formed if the polylines form a closed object.)

Note In order for the DXF Translator to merge line segments into polylines or polygons, each adjacent line segment must *share* a common endpoint (overlap endpoints).

This feature is especially useful when sending data to ADS Momentum, because Momentum can only mesh and simulate bounded elements. The default is active.

For more information on Momentum mesh and simulation, refer to "*Mesh*" and "*Simulation*" in the ADS Momentum manual.

Arc Extrapolation

The Arc Extrapolation section enables you to optimize arcs for a particular application.

Arc Resolution (in degrees)

The Arc Resolution defines the arc segment in degrees. The lower the value of the Arc Resolution, the smoother the arc. The default value for Arc Resolution is 5 degrees.

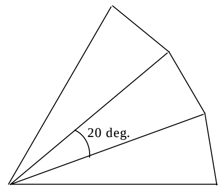


Figure 3-2. Arc Resolution

Arc Sag Distance

The Arc Sag helps to compensate for the triangle flat caused by each of the arc segments. By working with the dimension between the flat and the peak of the arc, the arc becomes gradually smoother. The default value for Arc Sag Distance is 0. The units for Arc Sag distance are defined in the *Unit Conversion* section.

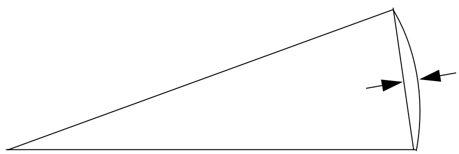


Figure 3-3. Arc Sag Distance

Note Use *Arc Resolution* and *Arc Sag* together for optimum results.

Layers

When importing large DXF files, you can save valuable computation time by including only specific layers. The following fields enable you to select particular layers that you want to include in your import. In order to obtain information about the DXF file's layers, ADS scans the DXF file and stores the layer information in a

.lyr file of the same name. If no DXF file was specified in the *Import* dialog box, the *Layers* options are left desensitized. If a DXF file was entered and ADS has already created a .lyr file for it, the list of layers is automatically displayed in the *Include Layer(s)* field. If a DXF file was entered but ADS cannot find a .lyr file for it, no layers are displayed and the *Search for Layers* button becomes active.

Include Layer(s)

This field displays all of the layers found within the DXF file. From this list, you can select specific layers to include when importing the file. If no specific layers are selected, ADS will import all layers by default.

Select All / Deselect All

Click the *Select All* button to select ALL layers for import. Click the *Deselect All* button to deselect ALL layers for import.

Search for Layers

If ADS cannot find a .lyr file with the same name as the import file, this button becomes active. Clicking this button causes the program to scan the DXF file and produce a .lyr file containing information about the file's layers. Once the .lyr file is created, the *Include Layer(s)* field is updated with all of the layers found within the DXF file, the *Search for Layers* button becomes desensitized, and you are now able to choose specific layers to include in your import.

Note If ADS has already created a .lyr file for a particular DXF file, and the DXF file is later modified, delete the old .lyr file before attempting to import only certain layers of the new DXF file.

Mapping DXF to ADS

Table 3-2 represents the mapping of DXF (hierarchical) file shapes to EGS Archive file shapes.

Table 3-2. Mapping of DXF File Shapes to ADS File Shapes

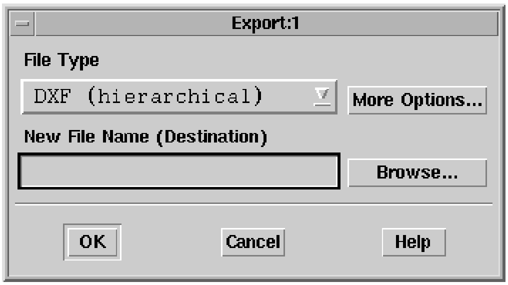
DXF (hierarchical)	Advanced Design System Layout Equivalent
Line with width = 0	Polyline
Line with width > 0	Path
Polygon with width = 0	Polygon
Polygon with width > 0	Polygon with cutline on longest segment
Rectangle with width = 0	Rectangle
Rectangle with width > 0	Polygon with cutline on longest segment
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 DXF (hierarchical) Files

When exporting files using the *DXF (hierarchical)* translator, the program first creates an intermediate EGS Archive file. This file is then translated into DXF format as shown in Figure 3-1.

To export DXF (hierarchical) files:

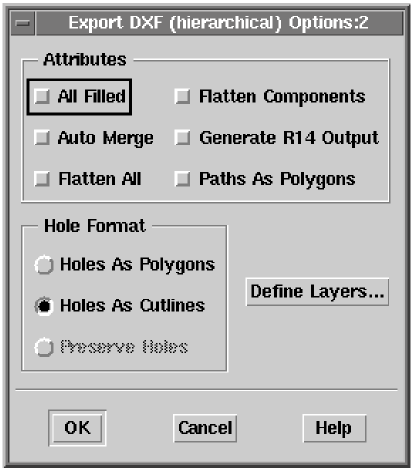
1. Follow the steps as outlined in “Exporting a Layout” on page 2-7. For available options (accessed via *More Options* in the Export dialog box), see “Export DXF (hierarchical) Options” on page 3-8.



2. After clicking **OK** in the Export dialog box, the translator converts your ADS Layout to EGS Archive format, then translates the EGS Archive file into DXF format as shown in Figure 3-1. Once in DXF file format, the file is available for use in AutoCAD or other Mechanical CAD systems.

Export DXF (hierarchical) Options

The *Export DXF (hierarchical) Options* dialog box enables you to set *Attributes* and *Hole Format*. This dialog also provides access to the *Layer Editor*.



Attributes

The *Attributes* section enables you to select various design attributes in the translation. All attributes are deselected as the default.

All Filled

When *All Filled* 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.

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. But beware: 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.

Flatten Components

When *Flatten Components* is selected, all parameterized components are flattened. This option is deselected as the default.

Generate R14 Output

When *Generate R14 Output* is selected, an AutoCAD version R14 database is generated. This option is deselected 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.

Hole Format

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

Holes As Polygons

When *Holes As Polygons* is selected, holes are converted into polygons. When *Holes As Polygons* is *not* selected, polygons with holes are translated as single-segment polygons, the *false edge* segment becoming part of the polygon.

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

Holes As Cutlines

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

Preserve Holes

This option is not available for DXF (hierarchical) Export. This option is desensitized as the default.

Define Layers

Clicking this button invokes the Layer Editor. For information on the Layer Editor, see “Defining Layers” on page 2-11.

Mapping ADS to DXF (hierarchical)

The DXF file created by the *DXF (hierarchical)* translator is as simple as possible so that as many DXF parsers, even primitive ones, can read the file. When exporting a layout in DXF format, a very simple header is created with an entry for each layer defined in the EGS Archive file. The DXF translator performs the following mapping between EGS Archive and DXF files.

Table 3-3. Hierarchical DXF Output

EGS Archive Entity	DXF Entity	Comment
Line	zero-width closed polyline	The difference between closed filled and closed empty is lost.
Polygon	zero-width closed polyline	
Line	open zero width polyline	
Circle	circle	The difference between circle and hole is lost.
Rectangle	circle	
Polygon	layer name	
Line	point	Any bad mask entity (one with zero area or length) is mapped to a point.

Exporting DXF (flattened) Files

When exporting files using the *DXF (flattened)* translator, the program first creates an intermediate mask file. This file is then translated into DXF format as shown in Figure 3-4.

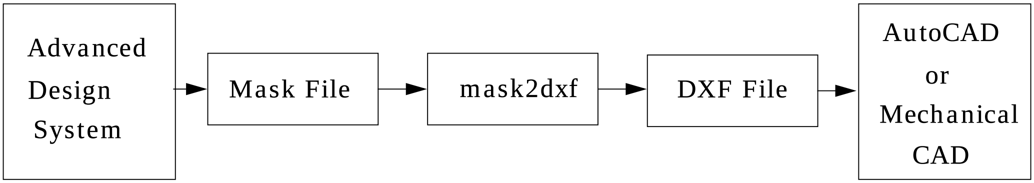


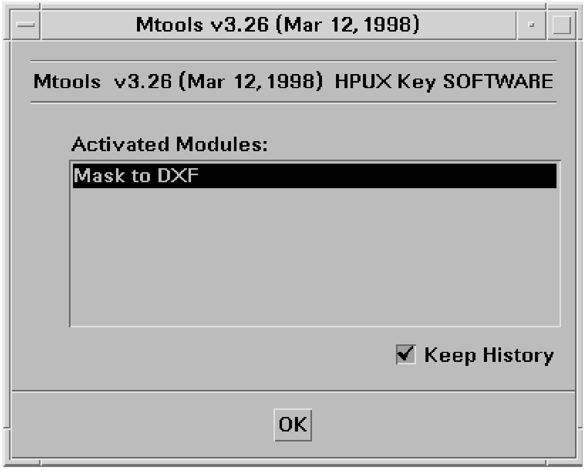
Figure 3-4. DXF (flattened) Block Diagram

For information on the Mask File format, refer to “Mask File” on page 1-5 and Chapter 11, Mask Files.

To export DXF (flattened) files:

1. Follow the steps as outlined in “Exporting a Layout” on page 2-7. For available options (accessed via *More Options* in the Export dialog box), see “Export DXF (flattened) Options” on page 3-17.

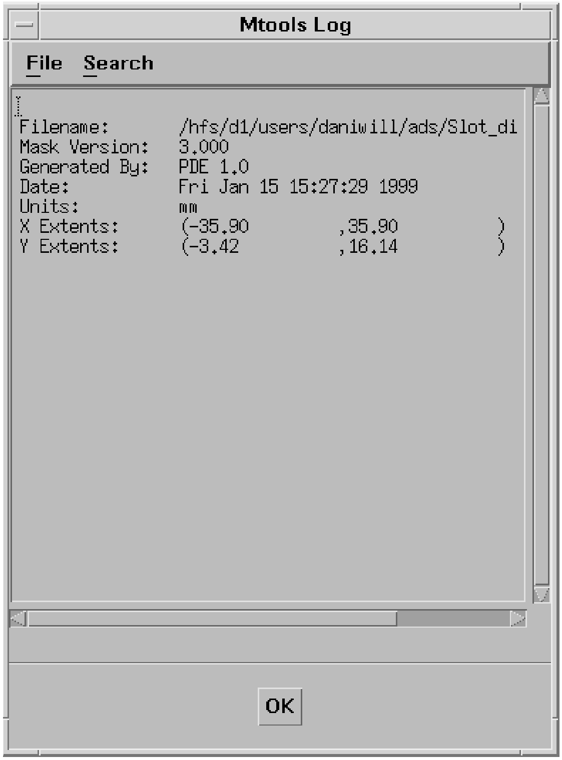
2. After clicking **OK** in the Export dialog box, the following window appears:



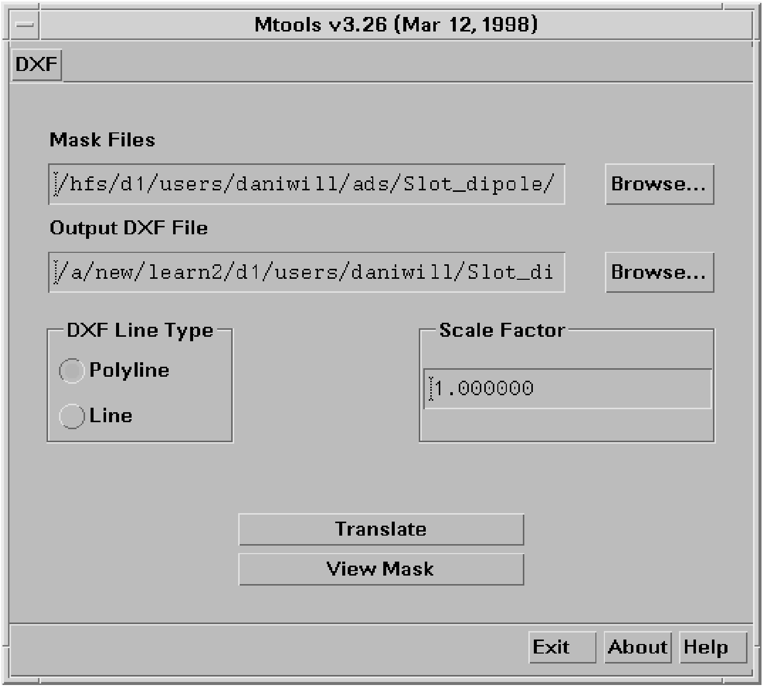
Under the heading *Activated Modules*, you will see the entry *Mask to DXF*. The DXF translator converts your design into a mask file before outputting it in DXF format.

Keep History is selected by 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.

3. Click **OK** to proceed. The Mtools Log window and DXF translator interface appear.
4. Verify that the Mtools Log contains the correct information, editing as necessary. To dismiss the log window, click **OK**.



5. In the DXF translator window, the *Mask Files* and *Output DXF File* paths should be set appropriately. If not, change the paths as desired. (Click **Browse** to view your directories.)



6. Set the *DXF Line Type* as desired. *Polyline* is the default.

You can configure the translator to use either lines or polylines when building each mask figure. The advantage of using polylines is that all segments forming the figure are connected into a single group. However, a few non-AutoCAD programs may not read polylines correctly.

Though lines are easier to edit, we recommend that you use polylines.

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

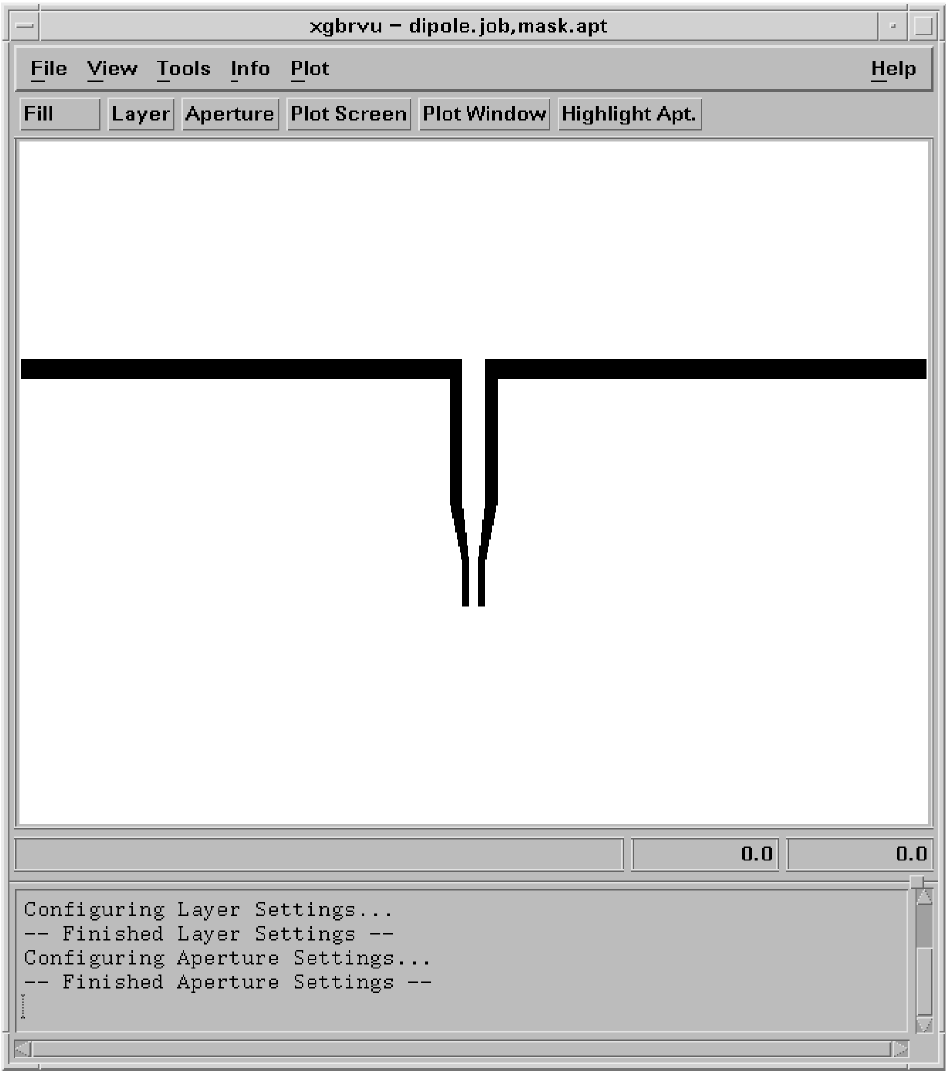
---

7. Set *Scale Factor* as desired. The default value is 1.000000.

The DXF output can be multiplied by a scale factor. This feature is useful when the mask file units are represented in mils, but the DXF file units need to be in inches.

By setting a scale factor of 0.001, each length in the mask file is multiplied by 0.001 and effectively scaled to inches.

8. To invoke the Gerber Viewer and view the mask file, click **View Mask**. The Gerber Viewer window appears.



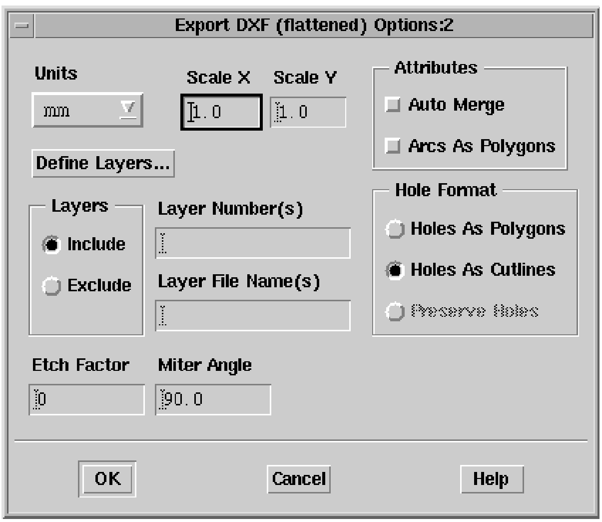
The Gerber Viewer presents a number of viewing options. For details, see Chapter 7, Gerber Artwork Translator and Gerber Viewer.

9. When you are finished viewing the file, choose **File > Exit Xgrbvu** to exit the Gerber Viewer. You may re-invoke the Viewer any time prior to exiting the translator interface by clicking **View Mask**.
10. To export the DXF file, click **Translate**. A dialog briefly appears stating that the mask file is being converted into DXF format.
11. When the translation is complete, the Mtools Log appears, detailing the DXF file information. To exit the log window, click **OK**.
12. To exit the DXF translator window, click **Exit**.

The export process is now complete.

Export DXF (flattened) Options

The *Export DXF (flattened) Options* dialog box enables you to define *Units* and *Scale*, perform layer definition, set an *Etch Factor* and *Miter Angle* and set *Attributes* and *Hole Format*.



Units

These are the units that the DXF 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 Advanced Design System *Layout* manual.

Scale X, Scale Y

The DXF output can be multiplied by a scale factor. This feature is useful when the mask file has units of mils (common in microwave design) and the DXF file is in units of inches. By setting a scale factor of 0.001, each length in the mask file is multiplied by 0.001 and effectively scaled to inches.

The default value for both *Scale X* and *Scale Y* is 1.0.

Define Layers

Clicking this button invokes the Layer Editor. For information on the Layer Editor, see "Defining Layers" on page 2-11.

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 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)

This option is used only when Layers Include is selected.

Each mask layer is exported to a separate DXF file as defined here. 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, arcs are exported as line segments (polygons). This option is deselected as the default.

Holes Format

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

Holes As Polygons

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

Preserve Holes

This option is not available for DXF (flattened) Export. This option is desensitized as the default.

Etch Factor

The etch factor applies a global over/under size 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.

Mapping ADS to DXF (flattened)

The DXF file created by the *DXF (flattened)* translator is as simple as possible so that as many DXF parsers, even primitive ones, can read the file. When exporting a layout in DXF format, a very simple header is created with an entry for each layer defined in the mask file. The DXF translator performs the mapping between mask and DXF files as shown in Table 3-4.

Table 3-4. Flattened DXF Output

Mask Entity	DXF Entity	Comment
closed filled	zero width closed polyline	The difference between closed filled and closed empty is lost.
closed empty	zero-width closed polyline	
open	open zero width polyline	
circle	circle	The difference between circle and hole is lost.
hole	circle	

Table 3-4. Flattened DXF Output (continued)

Mask Entity	DXF Entity	Comment
layer name	layer name	
zero length	point	Any bad mask entity (one with zero area or length) is mapped to a point.

Chapter 4: 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.

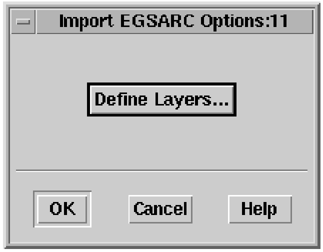
Importing EGS Archive Files

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” on page 2-3. For specific import options related to importing EGS Archive Files, refer to “Import EGS Archive Options” on page 4-1. For information on mapping EGS Archive files to ADS, refer to “Mapping EGS Archive to ADS” on page 4-2.

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" on page 2-11.

Mapping EGS Archive to ADS

Table 4-1 represents the mapping of EGS Archive file shapes to Advanced Design System layout shapes.

Table 4-1. Mapping of EGS Archive File Shapes to Advanced Design System 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" on page 2-7. For specific export options related to exporting EGS Archive Files, refer to "Export EGS Archive Options" on page 4-3. For information on mapping ADS to EGS Archive files, refer to “Mapping ADS to EGS Archive” on page 4-5.

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 parameterized components are flattened. 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.

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. But beware: 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.

Define Layers

The *Define Layers* button invokes the Layer Editor. For Layer Editor options, see “Defining Layers” on page 2-11.

Mapping ADS to EGS Archive

Table 4-2 represents the mapping Advanced Design System layout shapes to EGS Archive file shapes.

Table 4-2. Mapping Advanced Design System 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

Chapter 5: 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.

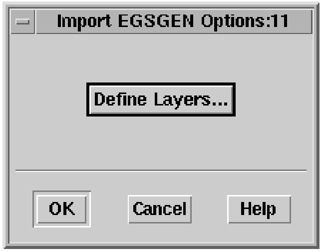
Importing EGS Generate Files

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" on page 2-3. For specific import options related to importing EGS Generate Files, refer to "Import EGS Generate Options" on page 5-1. For information on mapping from EGS Generate files to an ADS layout, refer to "Mapping EGS Generate to ADS" on page 5-2.

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" on page 2-11.

Mapping EGS Generate to ADS

Table 5-1 represents the layout mapping from EGS Generate format to Advanced Design System layout.

Table 5-1. Mapping from EGS Generate Files to Advanced Design System Layouts

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" on page 2-7. For specific export options related to exporting EGS Generate Files, refer to "Export EGS Generate Options" on page 5-3. For information on mapping ADS to EGS Generate files, refer to "Mapping ADS to EGS Generate" on page 5-4.

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.

Flatten Components

This option is irrelevant for EGS Generate files. The EGS Generate design is automatically flattened.

All Filled

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

Flatten All

This option is irrelevant for EGS Generate files. The EGS Generate design is automatically flattened.

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.

Define Layers

The *Define Layers* button invokes the Layer Editor. For Layer Editor options, see “Defining Layers” on page 2-11.

Mapping ADS to EGS Generate

Table 5-2 represents the layout mapping from Advanced Design System layout to EGS Generate format.

Table 5-2. 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

Chapter 6: 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:

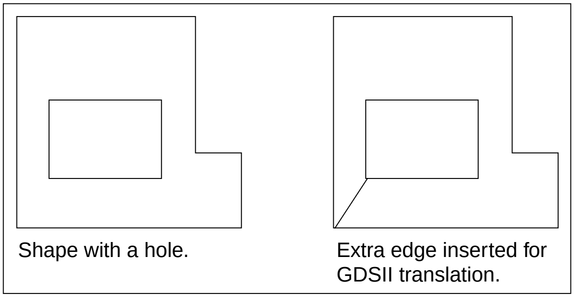
- No support for arcs or circles
- Limit of 200 vertices per shape
- No support for holes or empty regions
- 32-character name limitation
- Limit of 255 mask layers

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

If very long names are used for networks (greater than 32 characters), a translation table can be generated to map these names to shorter, unique names.

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.

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” on page 6-1), 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.

Table 6-1 is a mapping table for GDSII elements and their Advanced Design System layout equivalents.

Table 6-1. GDSII Elements and Advanced Design System Layout Equivalents

Calma/GDSII Element	Layout Equivalent(s)
Box element	Rectangle
Boundary element	Polygon
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” on page 2-3. For import options, see “Import GDSII Options” on page 6-5.

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" on page 2-3 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" on page 6-6), 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.

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.

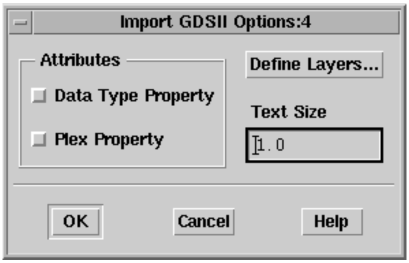
The GDSII file format can contain multiple top-level instances that may or may not be related to one another. Because of this, the GDSII filename may not be identical to the top-level instance name.

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" on page 2-11.

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” on page 2-11.

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” on page 2-7. For export options, see “Export GDSII Options” on page 6-8.

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 than 200 vertices are not translated (GDSII 200 vertex limit).
- When you attempt to translate shapes with more than 200 vertices, a warning message is printed to an error log. 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.

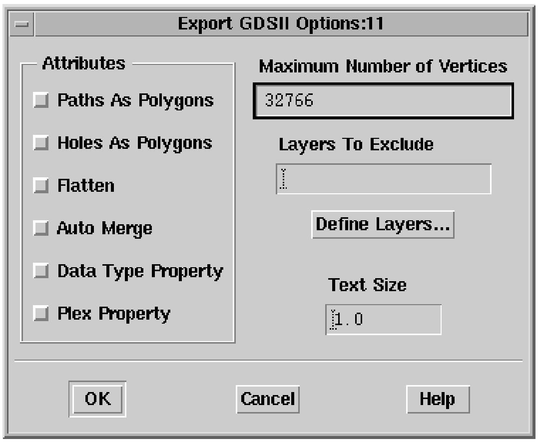
  GDSII dB/user unit = $10^{|\text{dB precision}|}$

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" on page 2-11.

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:

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

Flatten

When *Flatten* is selected, all levels of hierarchy are automatically removed and a single flat design is translated. 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. But beware: 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 - 32766 inclusive. The default value for this option is 32766.

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” on page 2-11.

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.

Chapter 7: Gerber Artwork Translator and Gerber Viewer

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

The Gerber Viewer is a companion program included with the 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.

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.

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

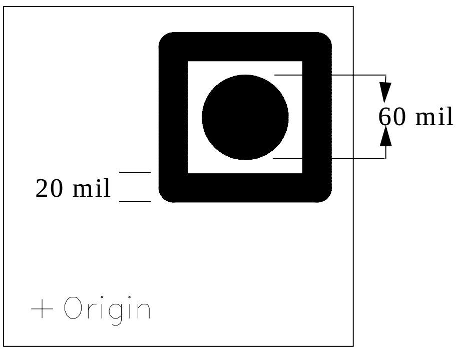
X and Y = coordinates

* = 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 *HP*. 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.

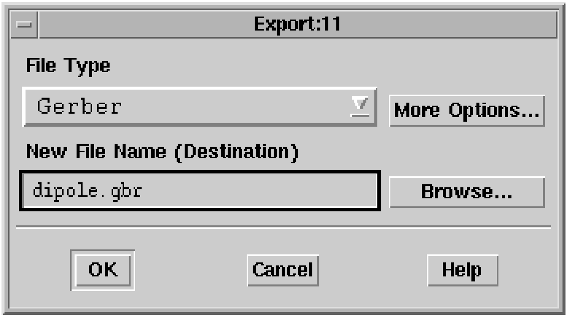
The photoplotter vendor receives both output files.

Exporting Gerber Files

To export Gerber files:

1. In the layout window containing your design, choose **File** > **Export**.

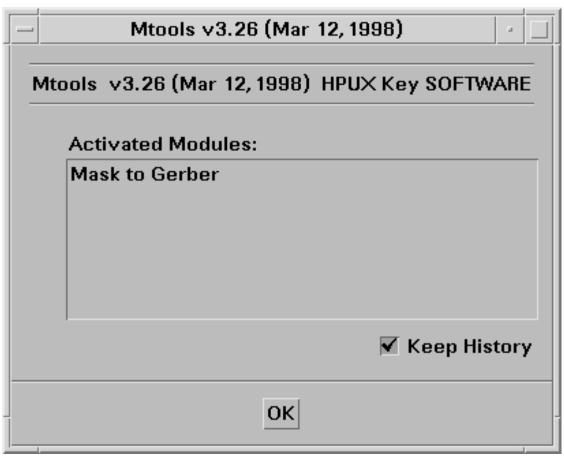
2. The Export dialog box appears. In the File Type field, click the arrow in the right-hand corner to display the available formats. Select **Gerber**.



3. To define export options, select **More Options**. Set the options that apply. For available options, see "Export Gerber Options" on page 7-15. Click **OK** to save your settings or **Cancel** to retain the default settings.

4. In the New File Name (Destination) field enter the new *file_name*. The file name is automatically appended with the appropriate suffix. Alternatively, you can select the **Browse** button to locate a destination directory and file name.

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

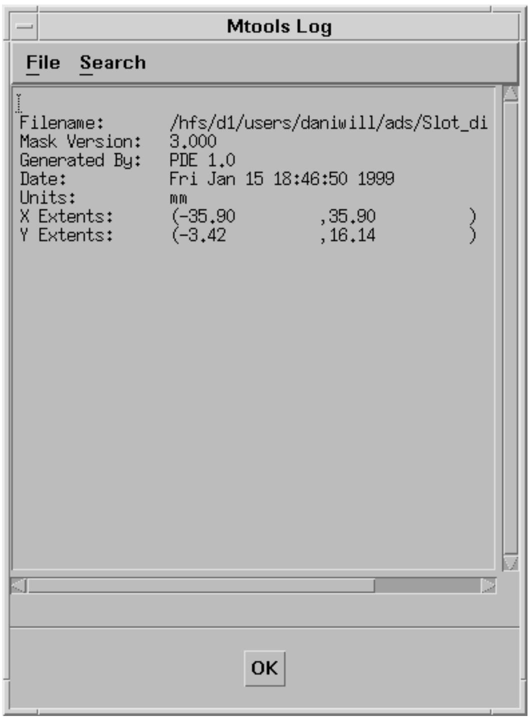


The entry *Mask to Gerber* displays under the heading *Activated Modules*. The Gerber translator converts your design into a mask file before outputting it in Gerber format.

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**.

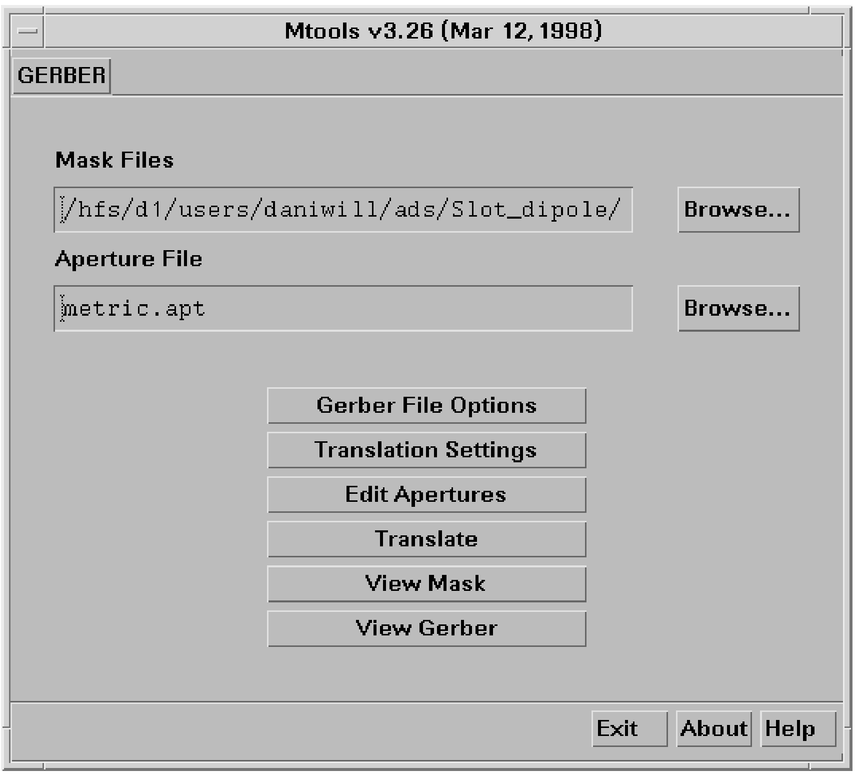


Note You may view the Mtools Log at any time by clicking the **GERBER** tab in the translator interface window.

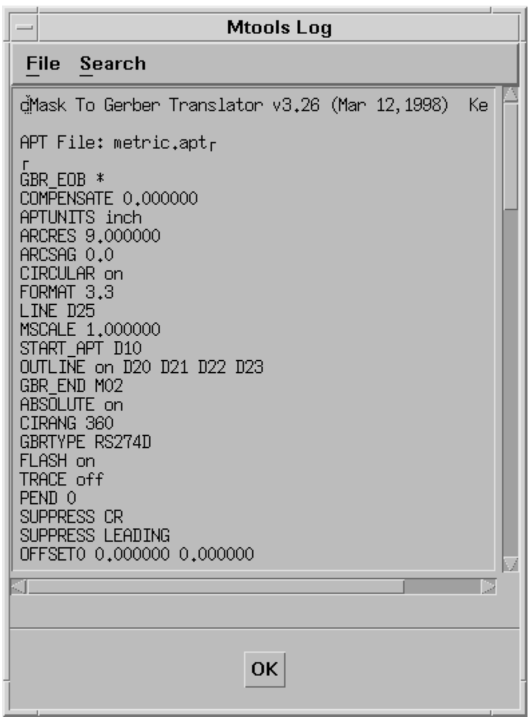
7. In the Gerber Translator Interface window, set the *Mask Files* and *Aperture File* paths. The *Mask Files* field displays the mask file created in the Export dialog box. This is the file that the Gerber translator converts into Gerber format.

   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" on page 7-7

   To view the *Mask Files* and *Aperture File* paths, click the adjacent **Browse** button.



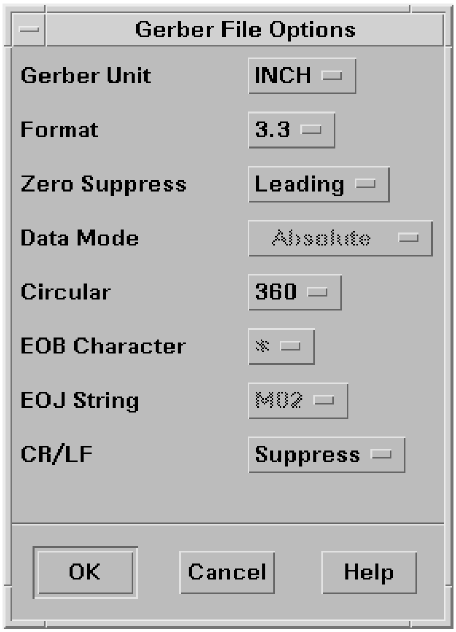
8. Select **Gerber File Options**, **Translation Settings**, and **Edit Apertures** to specify settings. Select **View Mask** to invoke the Gerber Viewer for DXF and Gerber translations and **View Gerber** for Gerber translations only. For details on Gerber translator interface options, see the appropriate section: “Gerber File Options” on page 7-7, “Translation Settings” on page 7-8, “Edit Apertures” on page 7-10, and “Using the Gerber Viewer” on page 7-18.
9. Click **Translate** to select the layers you want to include in the translated file (see “Translate” on page 7-13). 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 (*).

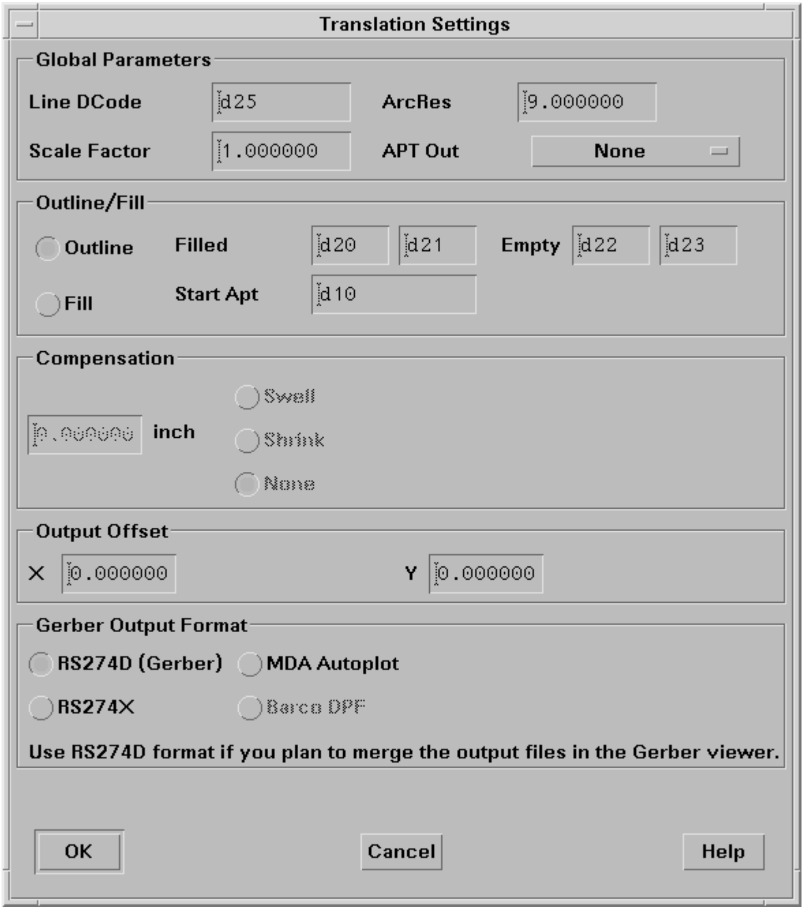
EOJ String. This string, inserted at the end of the Gerber file, indicates that the plot is complete. Currently set as *MO2*.

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” on page 7-2).

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.



Global Parameters

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

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

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.

- *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

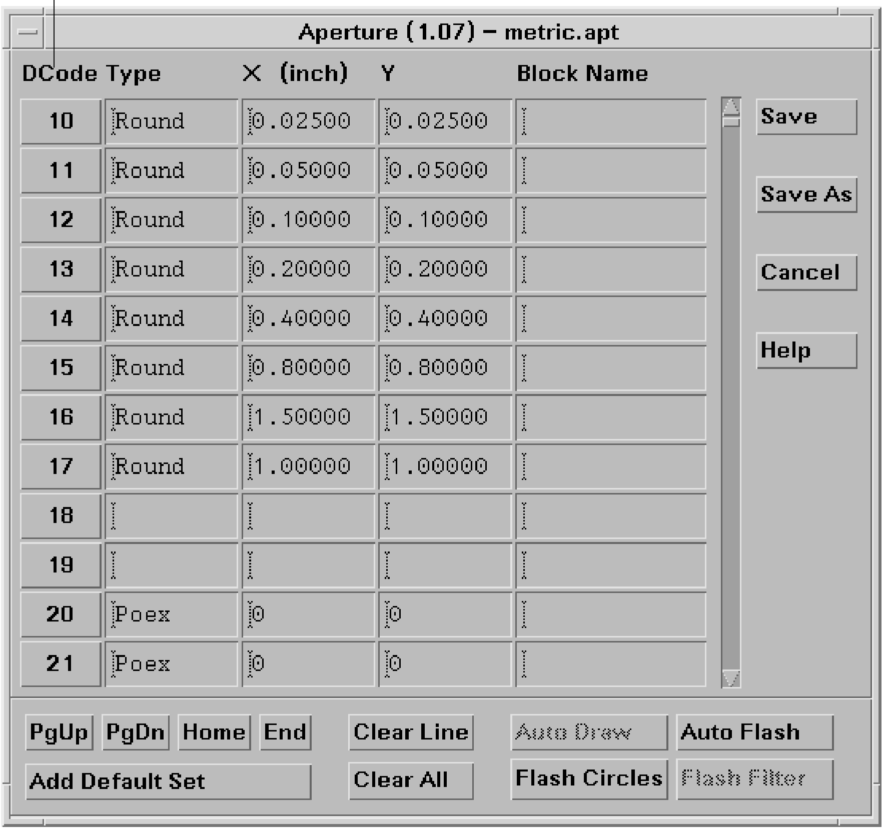
When setting the translation options, consider the following:

- *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.

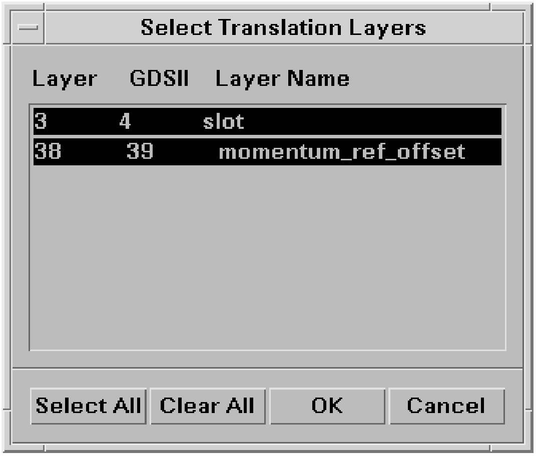
2. Select the desired type and click **OK**. The dialog box disappears.

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" on page 7-8.). For more information about this plotter, see "FIRE 9000 Photoplotter Configuration" on page 7-44.

To scan the mask file for circles and holes, and create a block name for each unique size, click **Flash Circles**. The created block names can then be assigned to unique apertures.

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



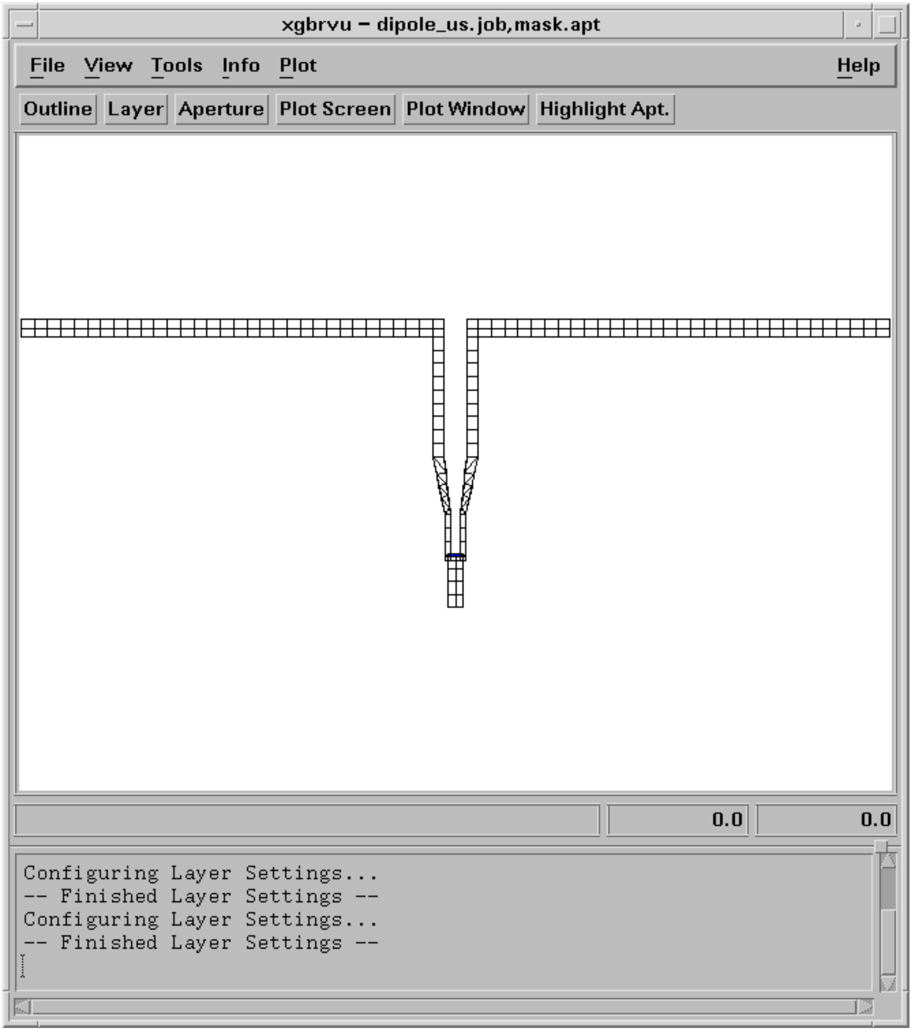
The *Translate* button invokes the Select Translation Layers dialog. This dialog enables you to select the layers to be included in the translated file. (The Gerber translator creates a Gerber file for each layer in the mask file.) All layers are selected as the default.

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" on page 7-18.

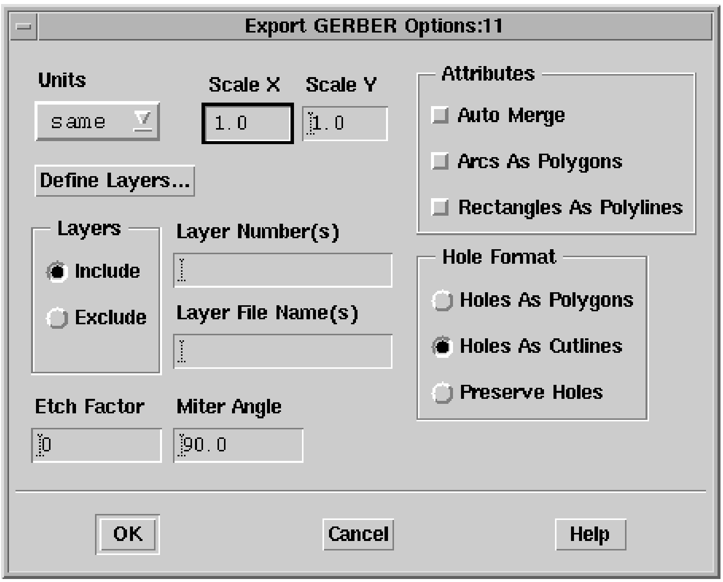
View Gerber

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" on page 7-18.

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 the *Advanced Design System Layout* 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" on page 2-11.

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 converted into polygons. 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

This option is not available for DXF (hierarchical) Export. This option is desensitized as the default.

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.

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

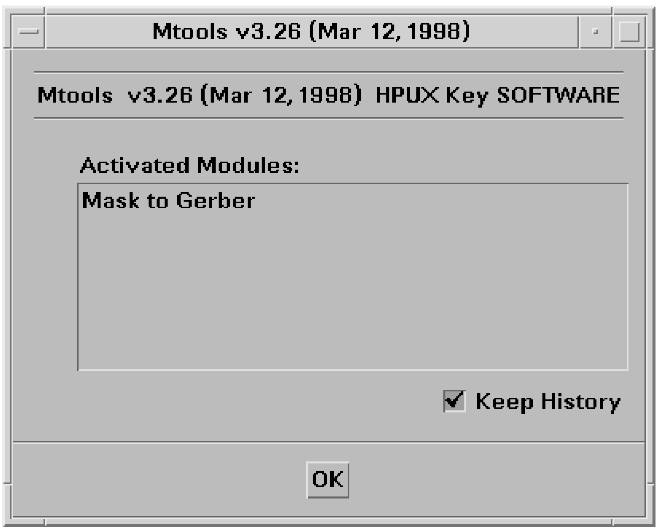
In order to be viewed, the files must meet the following criteria:

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

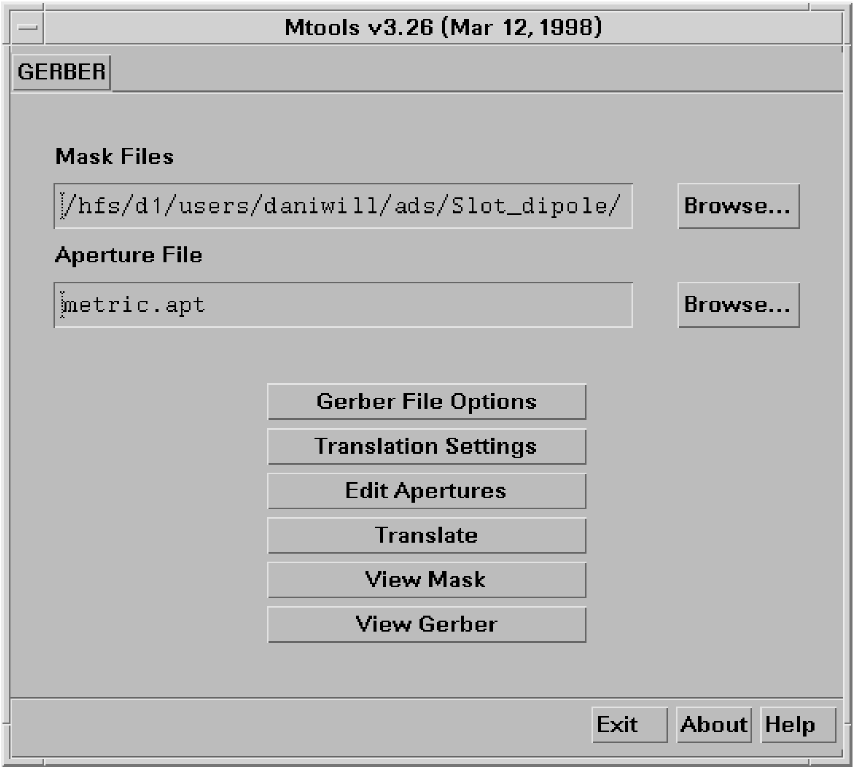
Launching the Viewer from a Layout Window

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.

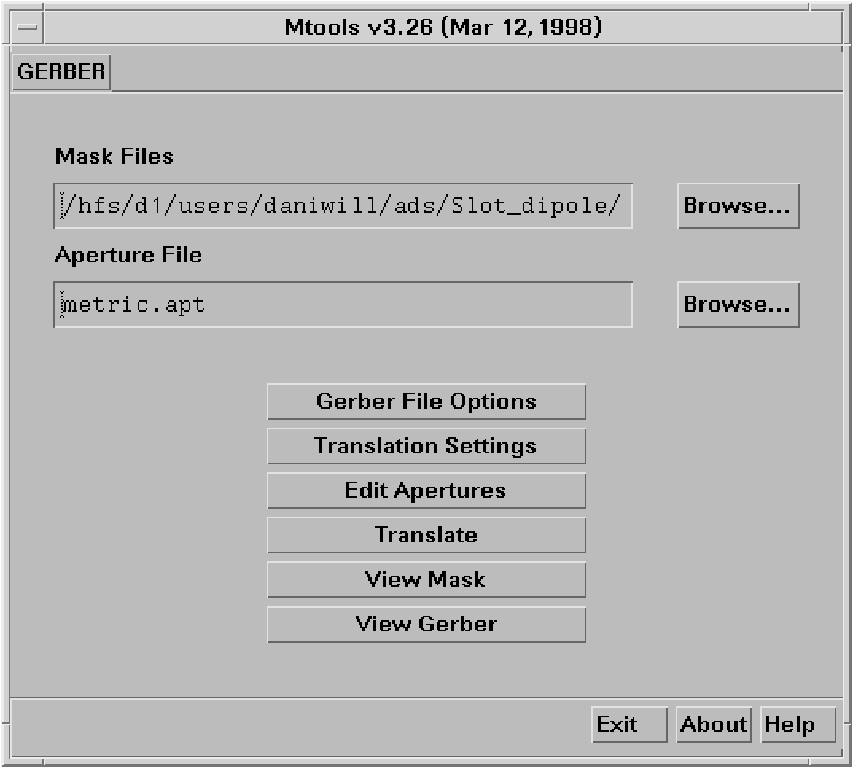
For information about Gerber Viewer options, see “Gerber Viewer Menu Options” on page 7-25. For information on the buttons available from the Gerber translator interface, see “Exporting Gerber Files” on page 7-2.

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” on page 7-25.. For information on Gerber translation options, see “Exporting Gerber Files” on page 7-2. For information on the options available from the Mtools DXF translator interface, see Chapter 3, DXF Translator.

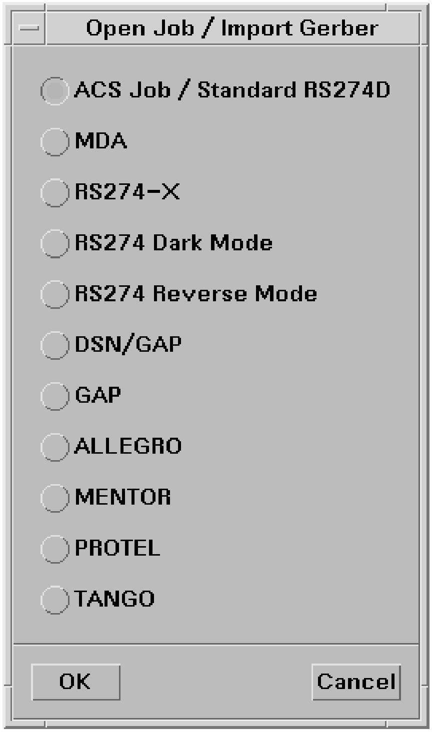
2. To exit the Viewer, select **File** > **Exit Xgbrvu**.

Loading a File to View

1. Choose **File** > **Open/Import**

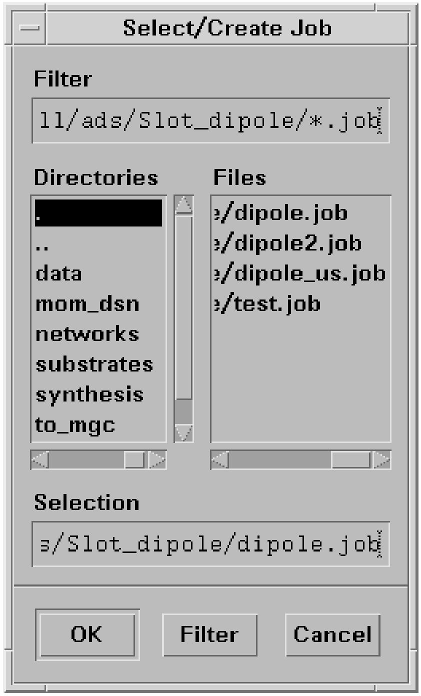
If there is a file in current memory (that is, you have initiated the Viewer during a file export) *and* it is not yet loaded, that file is automatically loaded at this time. There is no need to proceed with the following steps.

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.



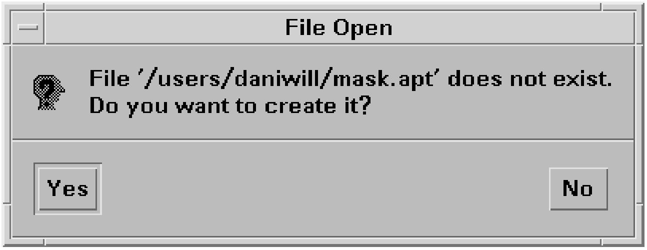
3. Select the file you want to load or create and click **OK**.

   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.

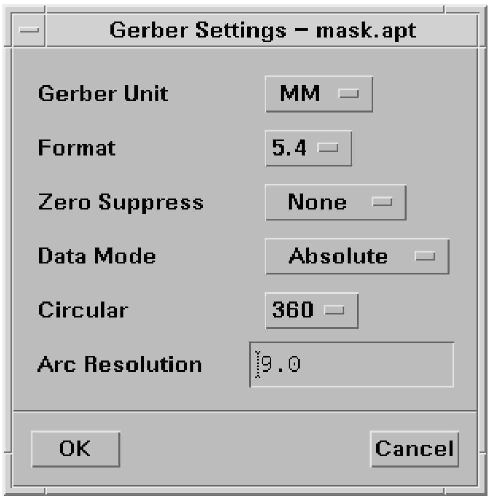
   If the file you choose does not exist, a dialog appears asking if you want to create it. Click **Yes** to create the file, or **No** to cancel.

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” on page 7-30.)

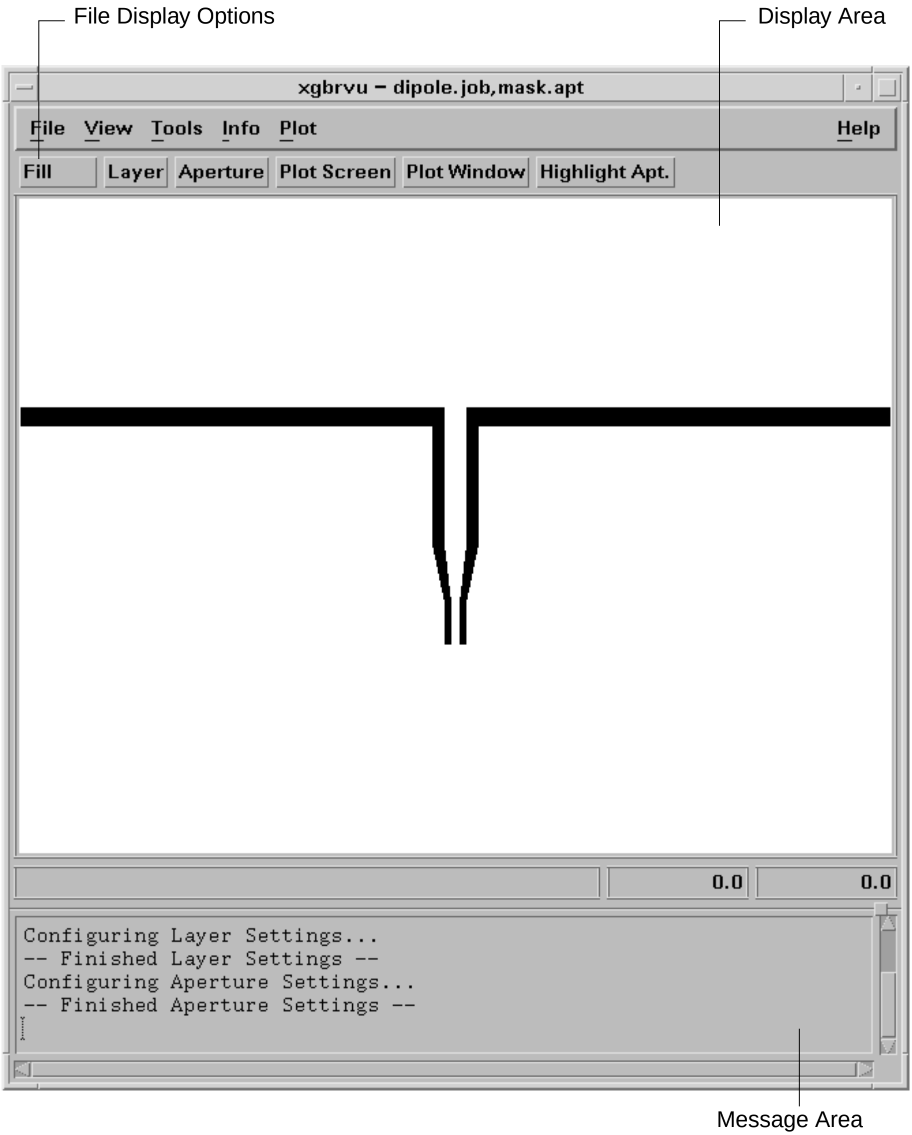


6. The Layer dialog appears. Select layers as desired and click **OK**. (For more information, see “Layer” on page 7-29.)
7. The Aperture dialog appears. Set apertures as desired (see “Aperture” on page 7-35) and click **Save**.
8. Set options, view, and plot as desired. See “Gerber Viewer Menu Options” on page 7-25.

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 appropiate button to change the window display. The button title changes to reflect the display.


File Display Options
Display Area
xgbrvu – dipole.job,mask.apt
File View Tools Info Plot
Help
Fill
Layer
Aperture
Plot Screen
Plot Window
Highlight Apt.
0.0
0.0
Configuring Layer Settings...
-- Finished Layer Settings --
Configuring Aperture Settings...
-- Finished Aperture Settings --
Message Area

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” on page 7-21.
- *Layer* invokes the Gerber Viewer Layer menu. For more information, see the section “Layer” on page 7-29.
- *Aperture* invokes the Gerber Viewer Aperture menu. For a description of this menu, see the section entitled “Aperture” on page 7-35.
- *Preferences* leads to the following menu choices:
    - *Gerber Settings* invokes the Gerber Settings dialog. See “Gerber Settings” on page 7-30.
    - *Grid/Snap Settings* opens the Snap Settings dialog. See “Snap Settings” on page 7-31.
    - *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" on page 7-32.

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." on page 7-37.

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.

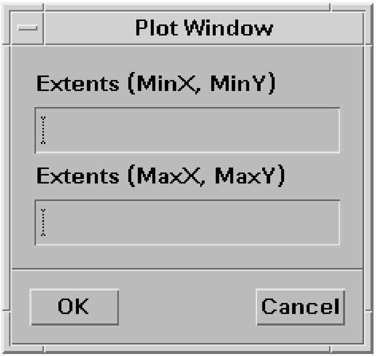
Note Actions prompted for in the Viewer message area (such as pressing **ESC** to abort) are effective only when the mouse is pointed in the Viewer display area.

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

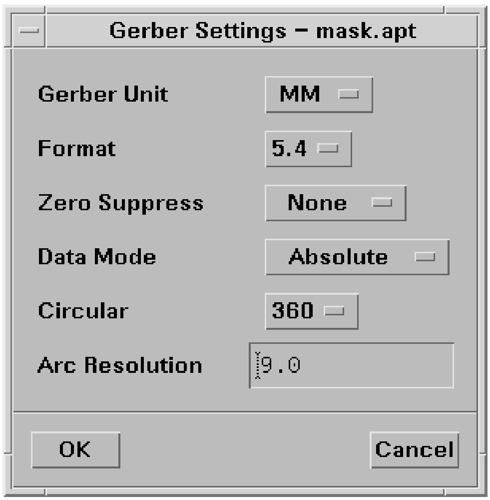
The *Layer* button invokes the Gerber Viewer Layer menu. (You can also choose the menu command *File* > *Layer* to open the Layer menu.)

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



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.

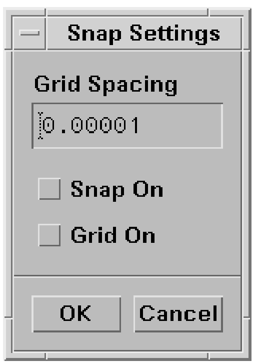
Circular. You may select either 90 or 360. Older photoplotters used 90 degree arc interpolation. Newer machines support 360 degree interpolation. The default setting is 360.

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



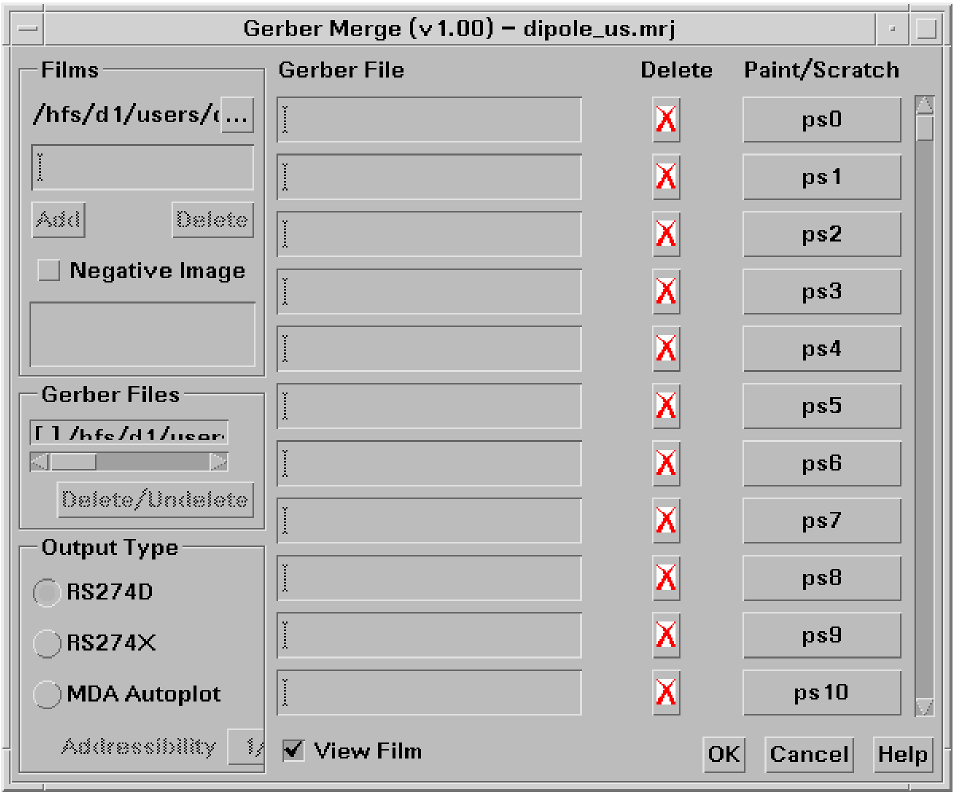
Grid Spacing. Use *Grid Spacing* field to set the grid spacing as desired.

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

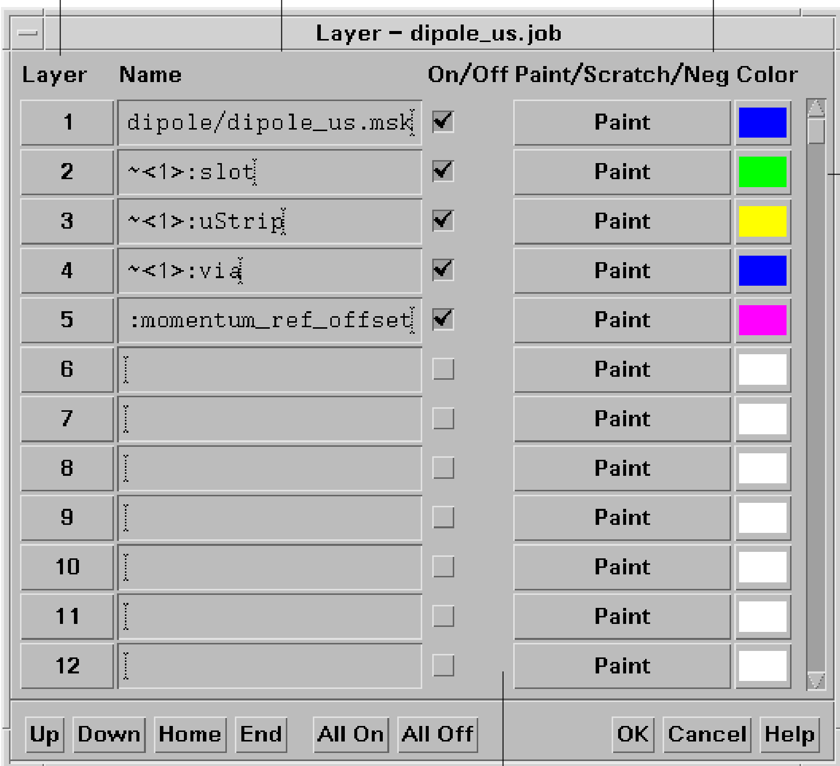
To load your Gerber files, click on a Layer number button. A file selection dialog box appears, enabling you to select the file(s) you want loaded. You may select display options as desired.

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.

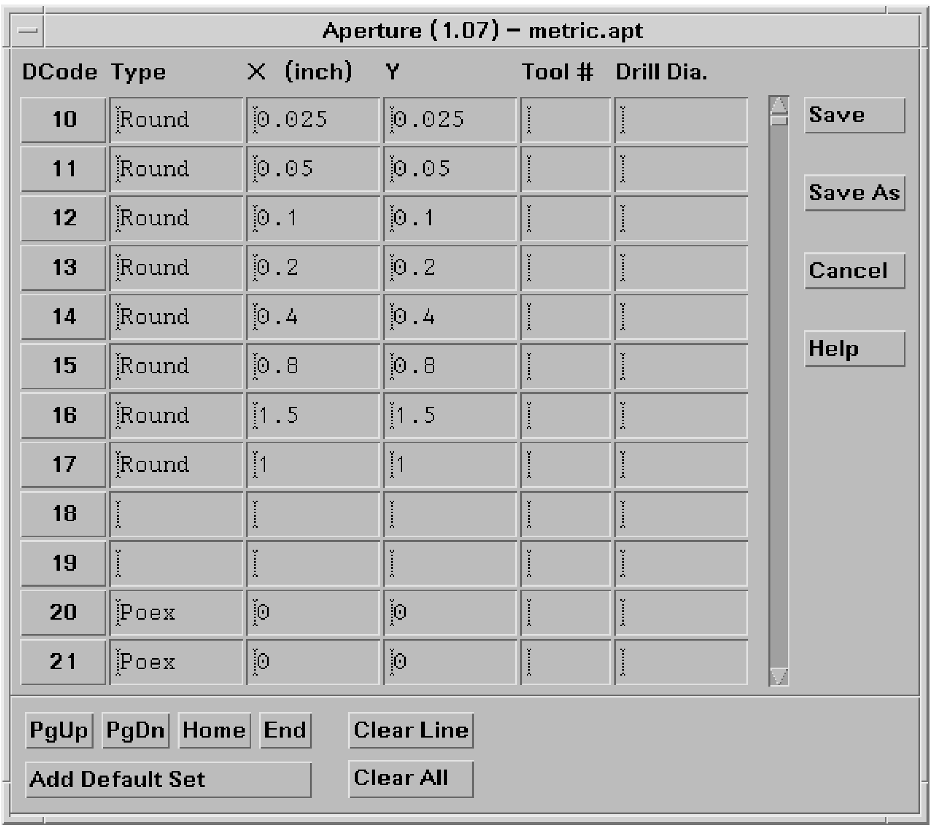
Scratch performs an XOR operation on all data previously loaded. The resulting display mimics the effect of scratch behavior available on many photoplotters.

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” on page 30). 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

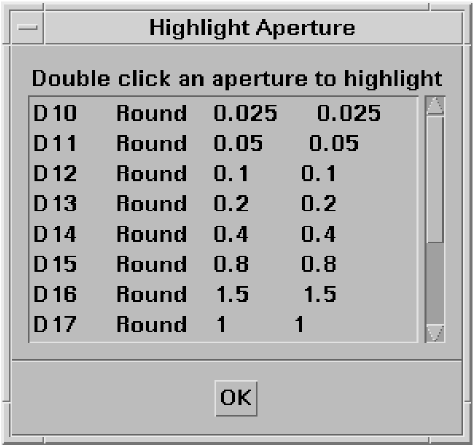
You may select this option by clicking the *Plot Screen* button on the Gerber Viewer menu bar or by selecting the menu command *Plot* > *Plot Screen*. The screen is plotted, and the plot information appears in the message area at the bottom of the Viewer.

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.

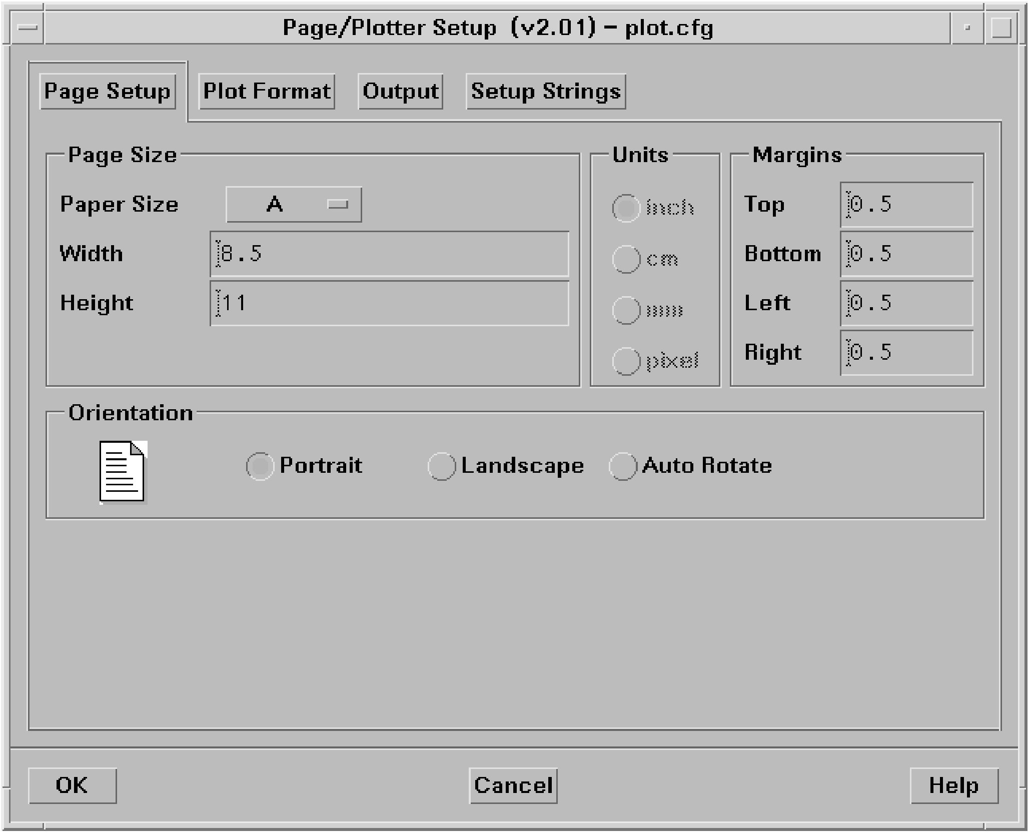


All traces associated with a highlighted aperture appear in light gray rather than in the assigned layer color.

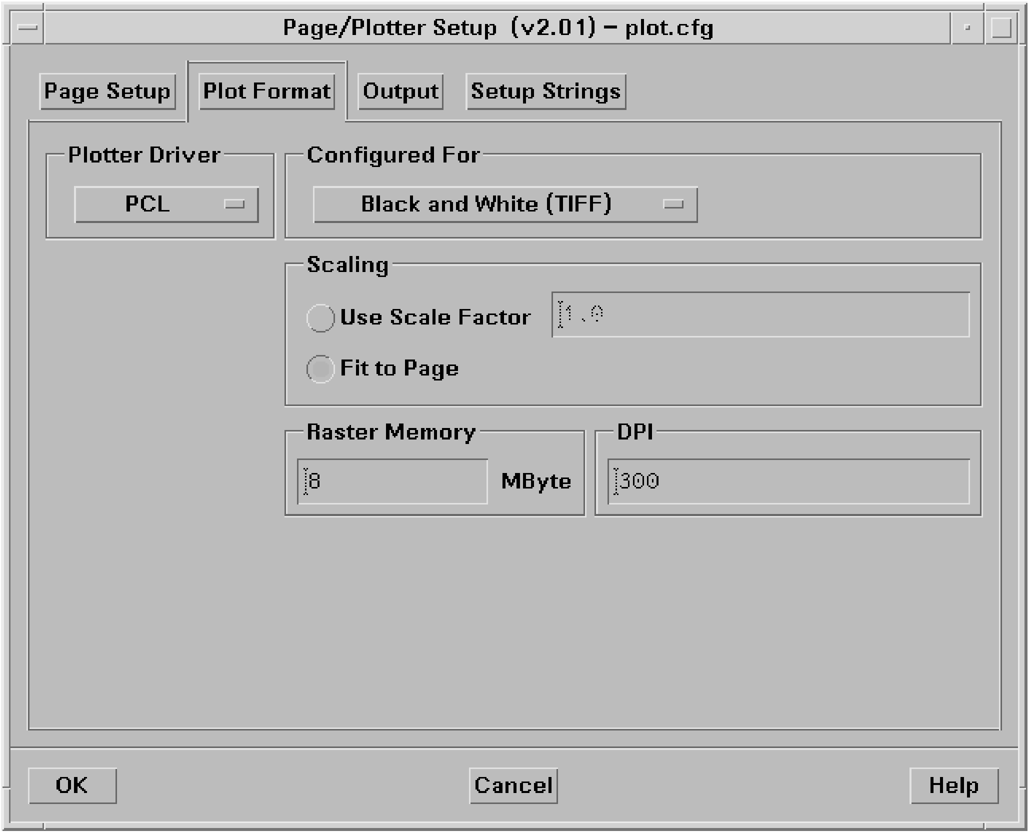
Page/Plotter Setup

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*

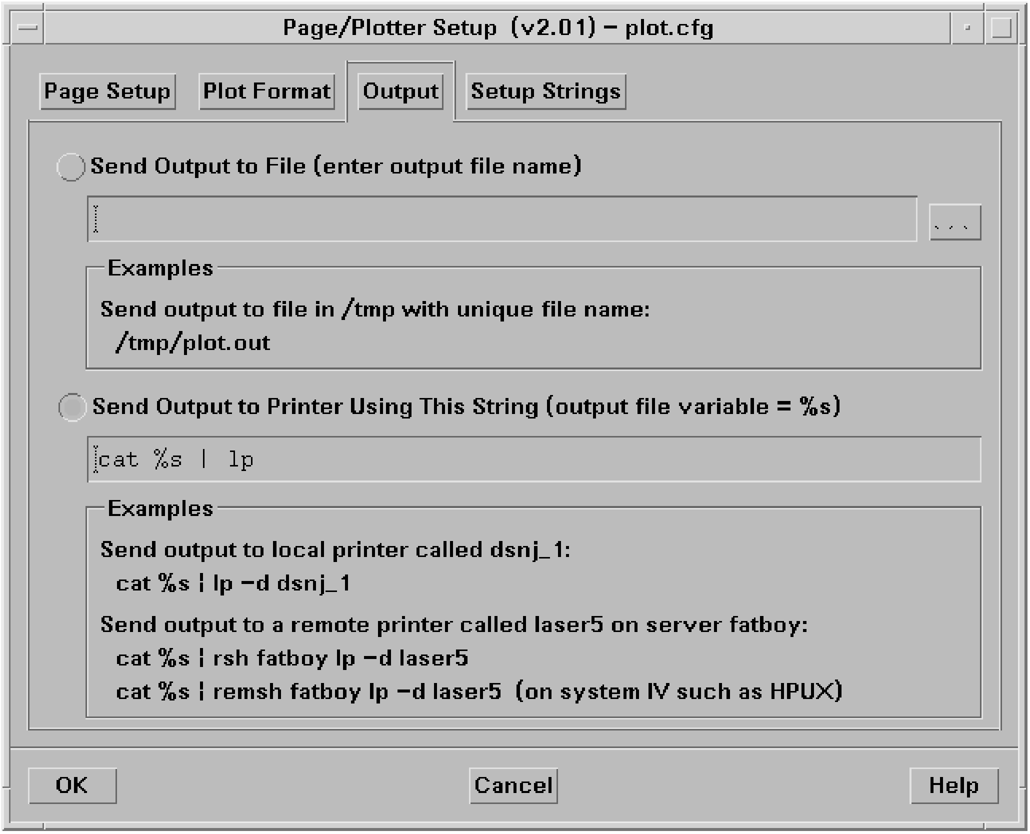
Page Setup. This tab provides access to page size, margin values, and orientation options.



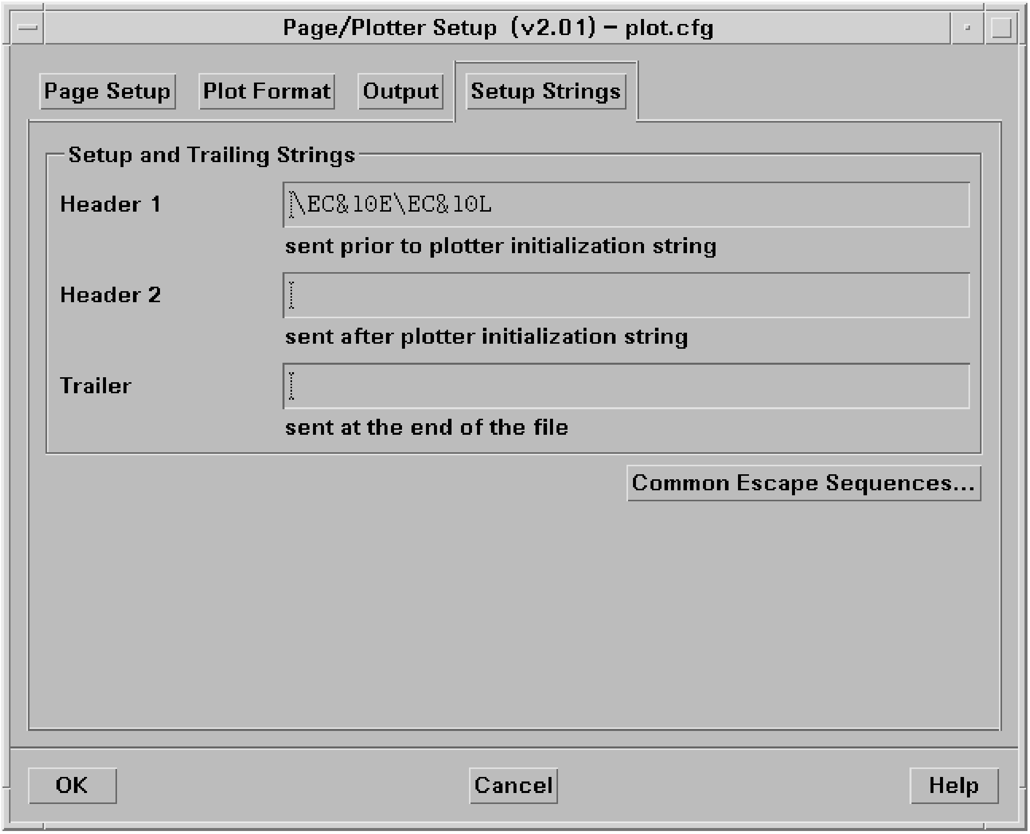
Plot Format. This tab provides access to the plotter driver type, print type, scaling factor, raster memory size (in MBytes), and dpi (dots per inch).



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.

Special shortcut keys for operating the Gerber Viewer are outlined in Table 7-1.

Table 7-1. Keyboard Shortcuts for the Gerber Viewer

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

Table 7-1. Keyboard Shortcuts for the Gerber Viewer

Key	Action
+	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.

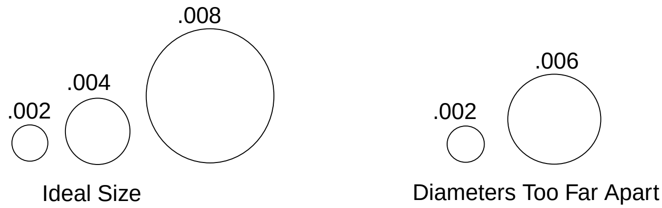
Raster Photoplotters

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

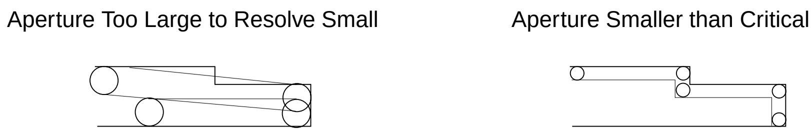
The mask file input to the Gerber translator is essentially a collection of polygons that need to be filled. Therefore, when you run the translator you should:

- 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.



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).

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 Cymbolic 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, 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.

Table 7-2. 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

Table 7-2. Gerber Translator Settings for FIRE 9000 Output

Gerber File Options		Translation Settings		Apertures (inches)		
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 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.

Table 7-3. 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

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.

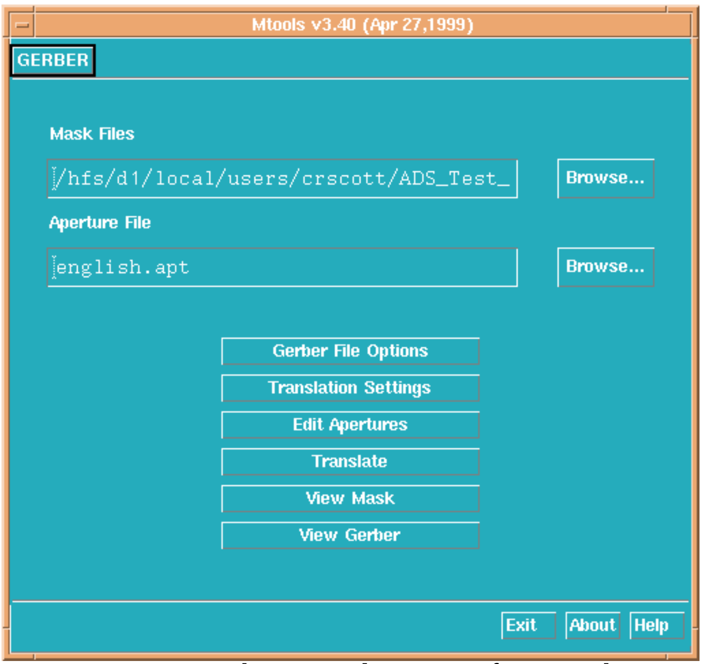
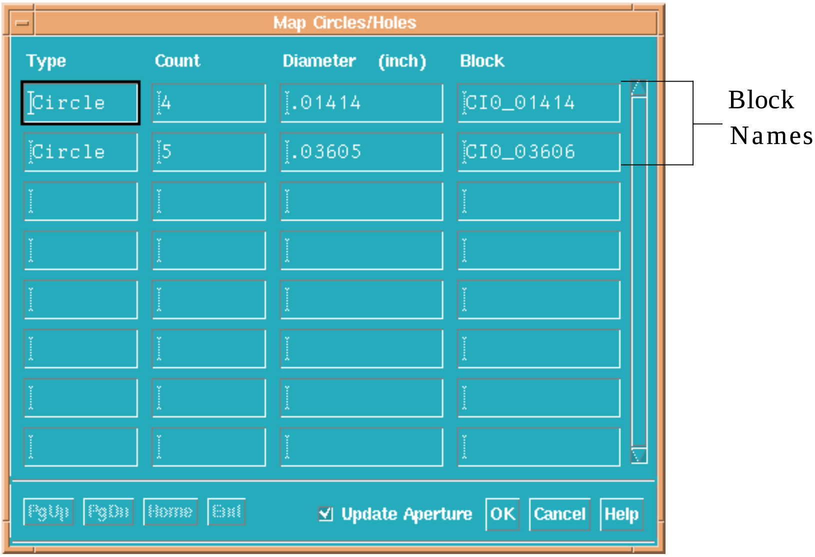


Figure 7-1. Gerber Translator Interface Window

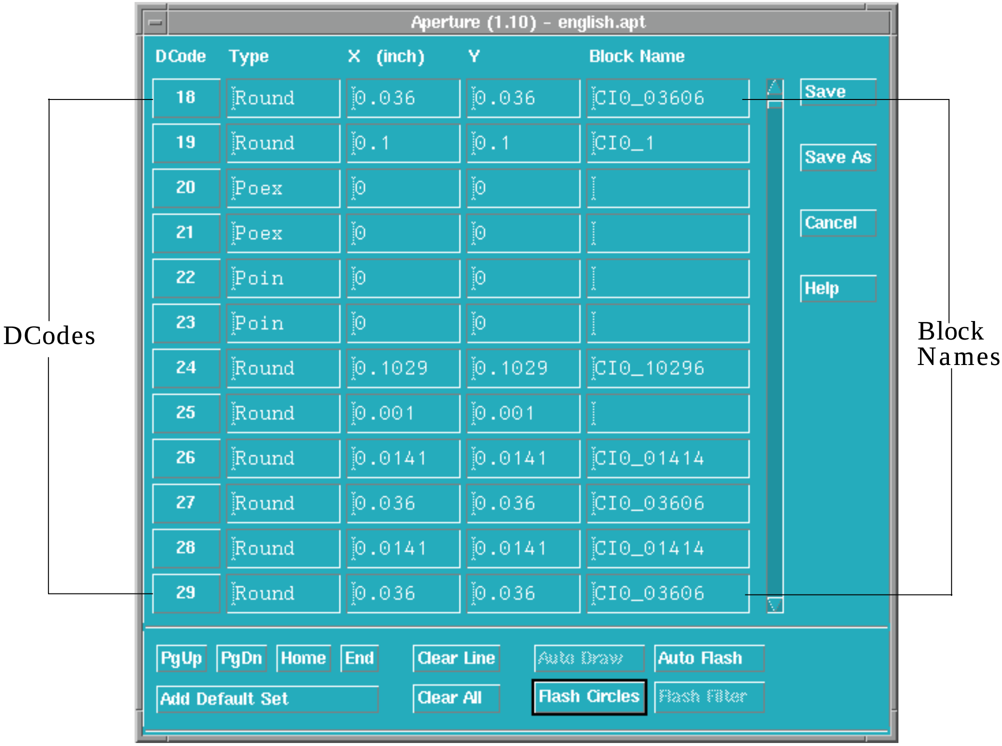
4. From the Gerber Translator Interface window, click **Edit Apertures**. A table listing the aperture settings appears.

5. In the Aperture dialog box, click **Flash Circles**. A Map Circles/Holes dialog box appears.



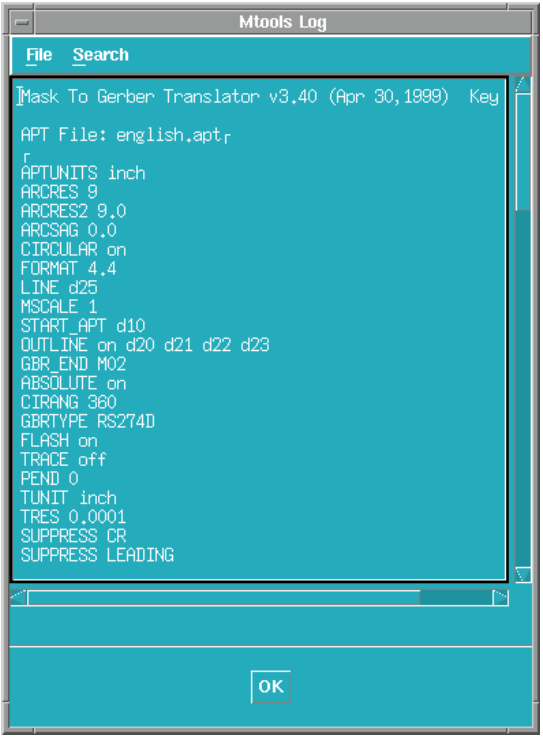
6. In the Map Circles/Holes dialog box, verify that the circle *Count* and *Diameter* for each *Block* are correct. Ensure that the *Update Aperture* box is checked. Note the Block name of each of the circles in the Map Circles/Holes dialog box. Click **OK** in the Map Circles/Holes dialog box.

   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**.



7. From the Gerber Translator Interface window, click **Translate** to select the layers you want to include in the translated file (see "Translate" on page 7-13). After selecting the layers, click **OK** to complete the translation. A window appears briefly, indicating that the file is being converted to Gerber format.

8. 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.

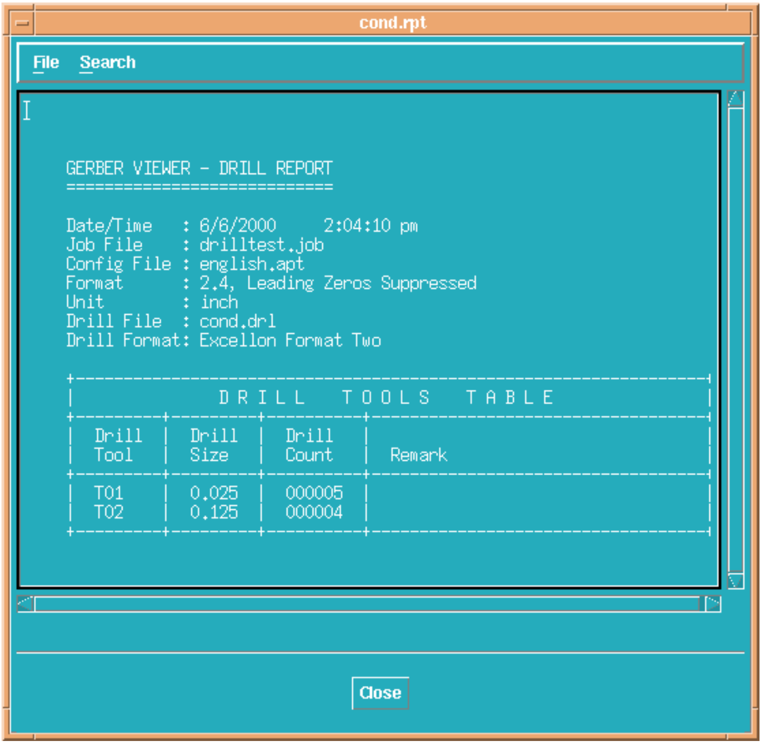
11. From the GBRVU dialog, click **Aperture**. The Aperture dialog box appears.

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.

Chapter 8: HPGL/2 Files

The HPGL/2 format is a subset of the HPGL/2 printer/plotter language. When creating a graph or chart, you can write the graphics data to an HPGL/2 simulator output file, then import the file into the Advanced Design System.

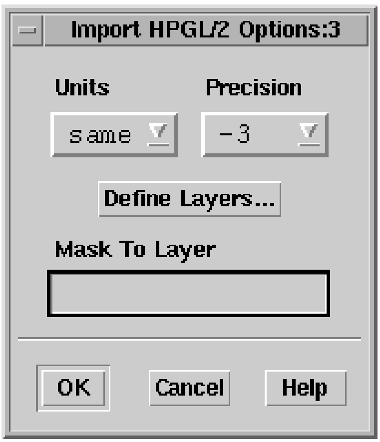
In the Advanced Design System, the HPGL/2 data is transformed into forms and shapes. These forms can be edited and manipulated like any other drawing. Additional text, annotation, scaling or editing may be added.

Importing HPGL/2 Graphics 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" on page 2-3. For specific import options related to importing HPGL/2 Files, refer to "Import HPGL/2 Options" on page 8-1.

Import HPGL/2 Options

The *Import HPGL/2 Options* dialog box enables you to set *Units* and *Precision*, access the *Layer Editor* and specify a list of mask files to be read by the translator.



Units

These are the units with which the HPGL/2 file will be imported. You may select from the following options: *same*, *mil*, *inch*, *um*, *mm*, *cm*. The default is *same*. When *same* is selected, the design is read in with the same units that are stored in the original design file. For more information on choosing layout units, refer to "*Setting Units/Scale Factors*" in the Advanced Design System *Layout* manual.

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 or –3. The default is –3.

Define Layers

This button invokes the Layer Editor. For Layer Editor Options, see "Defining Layers" on page 2-11.

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 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" on page 2-7. For specific export options related to exporting HPGL/2 Files, refer to "Export HPGL/2 Options" on page 8-2.

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 Units/Scale Factors*" in the Advanced Design System *Layout* manual.

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" on page 2-11.

Chapter 9: 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

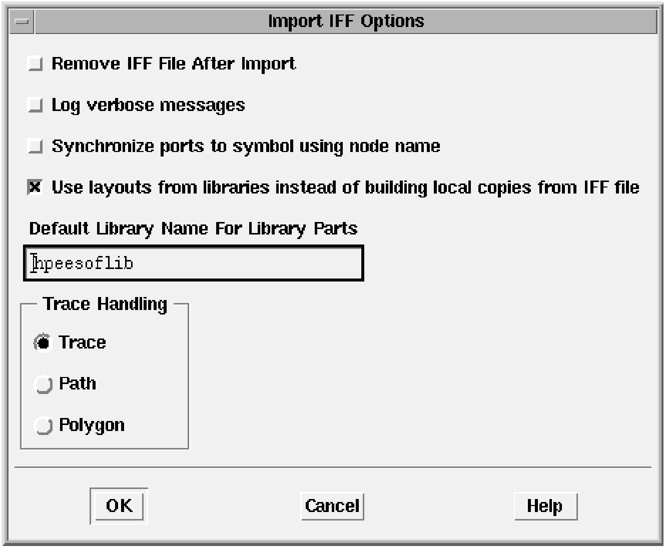
IFF files are used as the exchange mechanism when transferring designs between Advanced Design System and third-party EDA tools such as Mentor Graphics Design Architect and Cadence Analog Artist.

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" on page 2-3. For specific import options related to importing IFF Files, refer to "Import IFF Options" on page 9-1.

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* or *Layout* 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 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" on page 9-1.

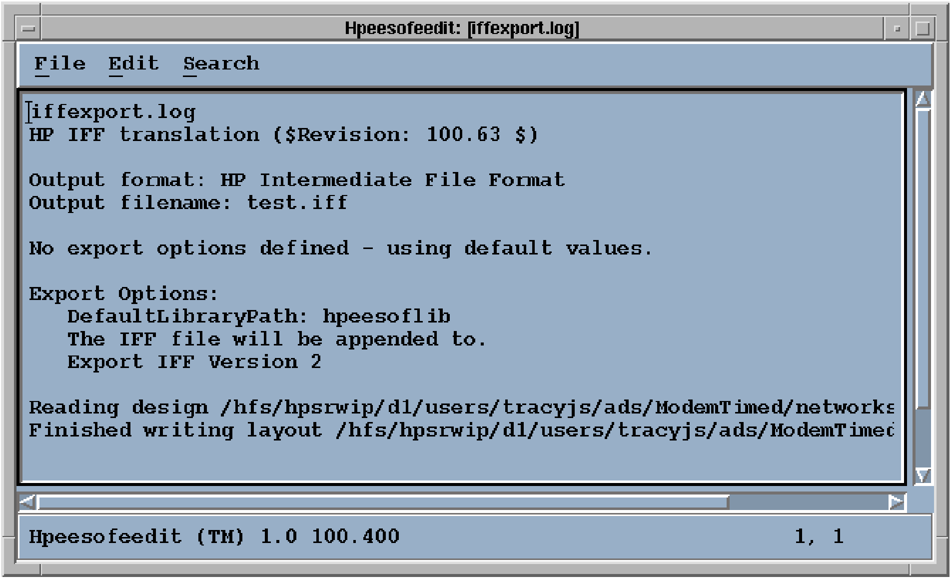
Exporting IFF Files

To export an IFF file:

1. Follow the steps as outlined in "Exporting a Layout" on page 2-7. For available options (accessed via *More Options* in the Export dialog box), see "Export IFF Options" on page 9-6.

2. When the translation is complete, an Information Message window appears. Click **OK** to dismiss this window.

3. The IFF Export log appears:

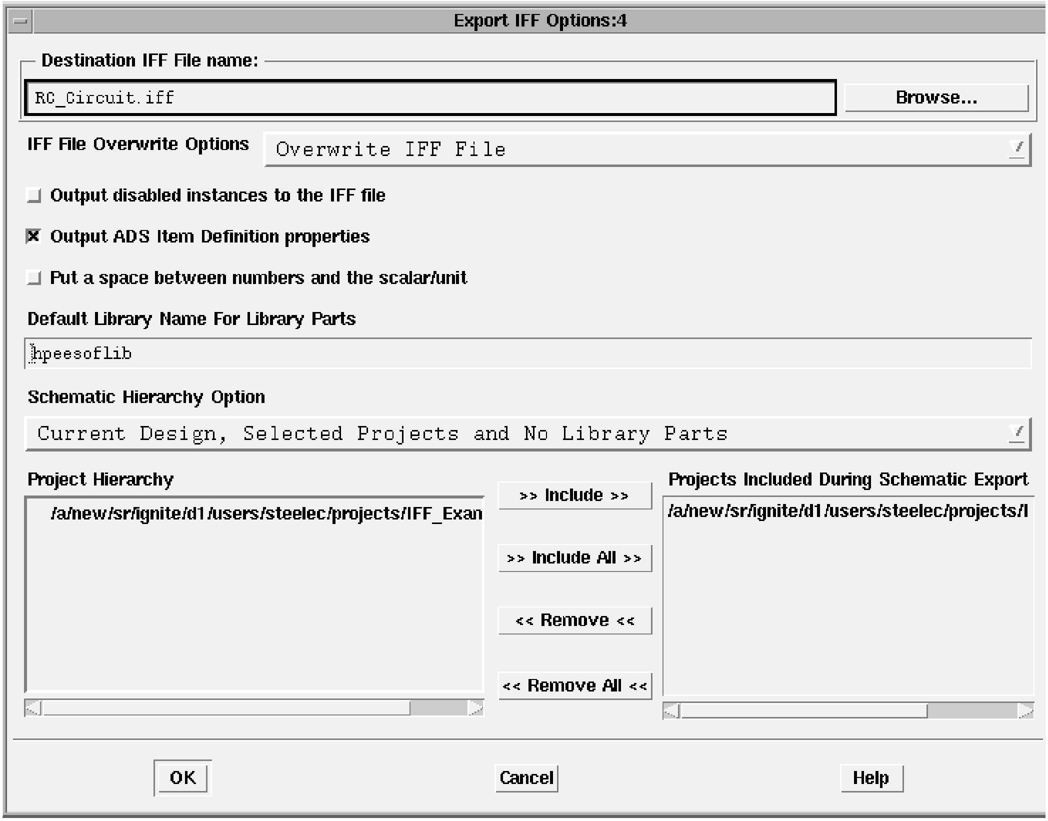


Review the log, searching for any warnings or error messages generated during export.

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

4. 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.

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**.

Chapter 10: 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" on page 10-2 and "Export IGES Options" on page 10-3 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" on page 2-3. For specific import options related to importing IGES Files, refer to "Import IGES Options" on page 10-2.

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" on page 2-11.

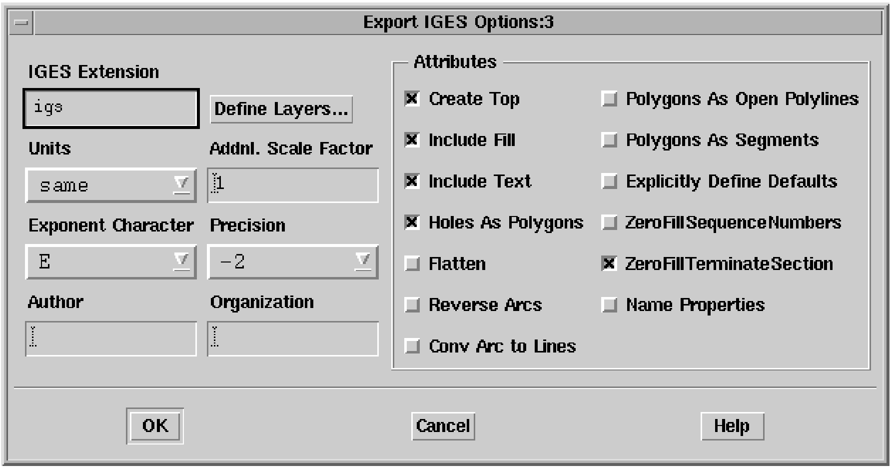
Note that the layer-to-IGES level number is controlled in the Layer Editor *IGES Layer* entry field for each program layer.

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" on page 2-7. For specific export options related to exporting IGES Files, refer to "Export IGES Options" on page 10-3.

Export IGES Options



IGES Extension

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

Units

These are the units that the IGES file will be written in. You may select from the following options: *same*, *mil*, *inch*, *um*, *mm*, *cm*. The default is *same*. When *same* is selected, the design is written in the same units that are stored in the design file. For more information on choosing layout units, refer to "*Setting Units/Scale Factors*" in the Advanced Design System *Layout* 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" on page 2-11.

Addnl. Scale Factor

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

Precision

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

Organization

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

Create Top

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

Include Fill

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

Include Text

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

Holes As Polygons

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

Flatten

When *Flatten* is selected, all levels of hierarchy are automatically removed and a single flat design is translated. There will be no references from the top level structure to any other structure. This option is useful when your post-processor does not support or correctly translate hierarchy in IGES files. But beware: if a substructure was initiated 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

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

You may encounter problems when attempting to read IGES files produced by the program into other systems. Some of the more common problems and their solutions are listed in the following table (Table 10-1). These solutions are implemented through the IGES Export Options dialog box in the Advanced Design System.

Table 10-1. Third-Party IGES Reader Problems and Advanced Design System Export Solutions

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

Chapter 11: 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.

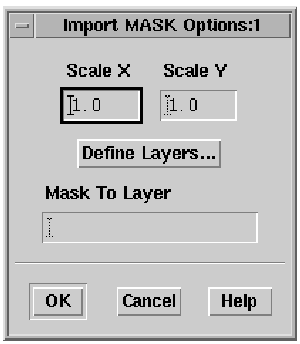
The Gerber and DXF translators use the mask file format as an intermediate file when converting data to Gerber and DXF. The mask format is also used as an intermediate file when translating other formats such as HPGL/2. In addition, the simplicity of the mask file format makes it an appealing option 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" on page 2-3.

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” on page 2-11.

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:

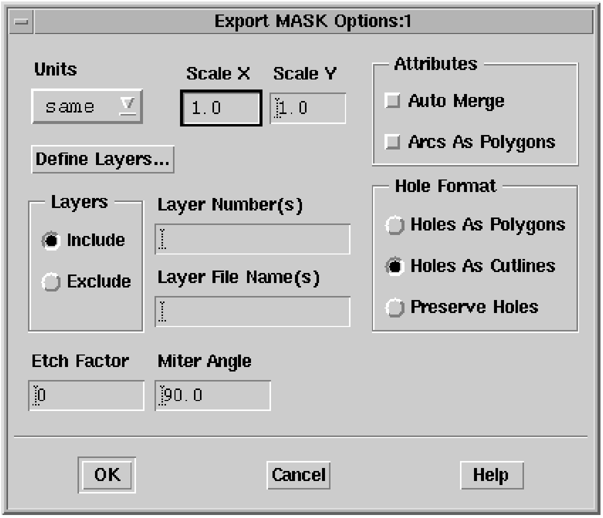
<layer_number> <file_name> <layer_number> <file_name>...

where all entries are separated by spaces.

Exporting Mask Graphics Files

For a step-by-step tutorial, see “Exporting a Layout” on page 2-7.

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 Units/Scale Factors*" in the Advanced Design System *Layout* 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" on page 2-11.

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:

<*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 converted into polygons. 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

This option is not available for DXF (hierarchical) Export. This option is desensitized as the default.

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.

Chapter 12: 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 export 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 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.*

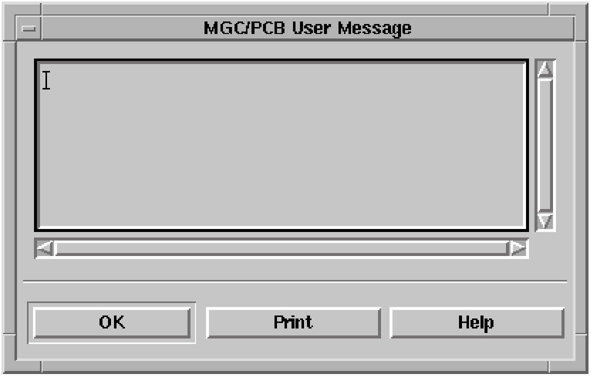
Mentor PCB products do not accept layout hierarchy, so the entire layout is flattened prior to building the Mentor layout.

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" on page 2-7. For available options (accessed via *More Options* in the Export dialog box), see "Export MGC/PCB Options" on page 12-4.
2. If the option *Prompt For User Message* was selected (see), a "Export MGC/PCB Options" on page 12-4 message dialog appears:



Enter any messages to be included with the design transfer. This information is used only by the *import_hpeesof* utility (see Chapter 12, MGC/PCB Files) and is not kept with the design.

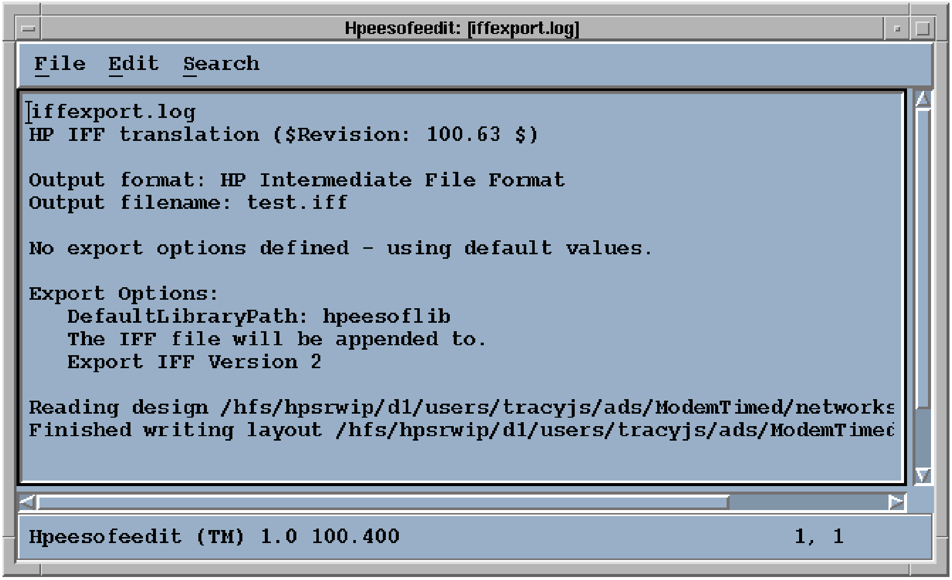
If you do not wish to include a message in the file, you may leave this window empty. If you want to print the message, click **Print**.

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:

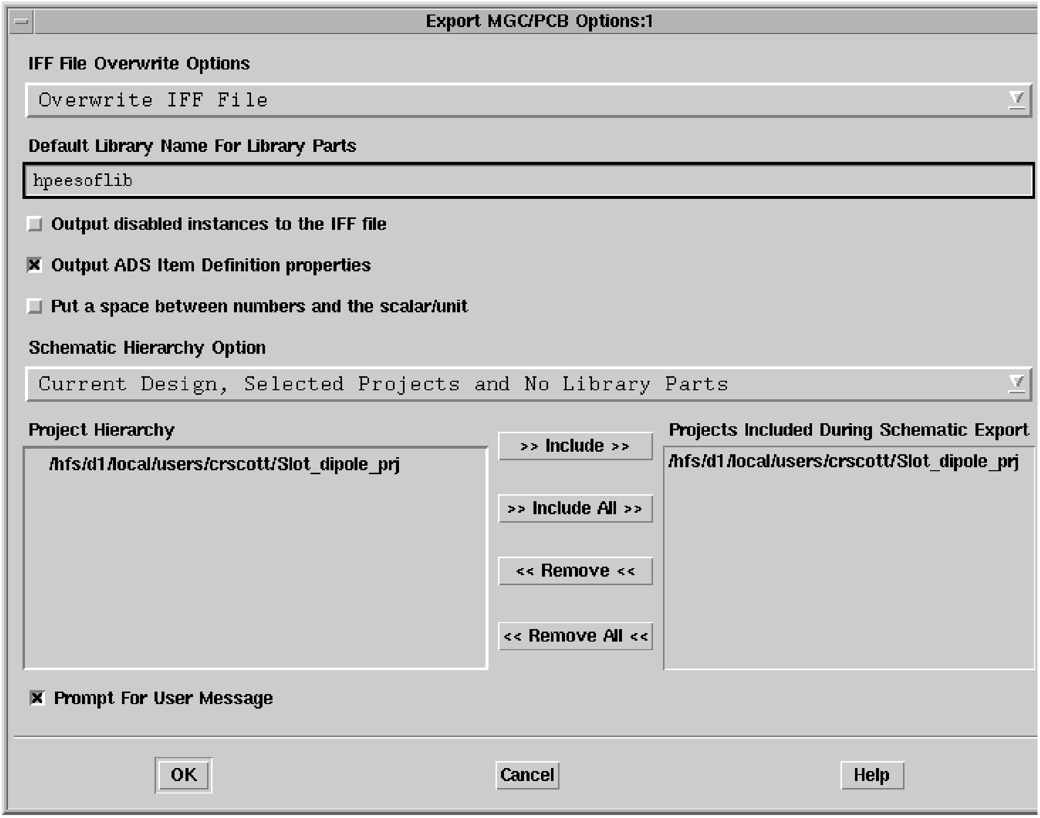


Review the log, searching for any warnings or error messages generated during export.

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

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

Export MGC/PCB Options



IFF File Overwrite Options

- *Overwrite IFF File*

When writing to an existing file, the contents of that file are overwritten. This is the default setting.

- *Append to IFF File*

When writing to an existing file, the new data is appended to the existing file.

Default Library Name For Library Parts

The name of the library to which the library parts are written. Design objectives are stored in a group that uses the same name as the 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

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

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" on page 12-4.

This option is selected as the default.

Index

U

V

Z