

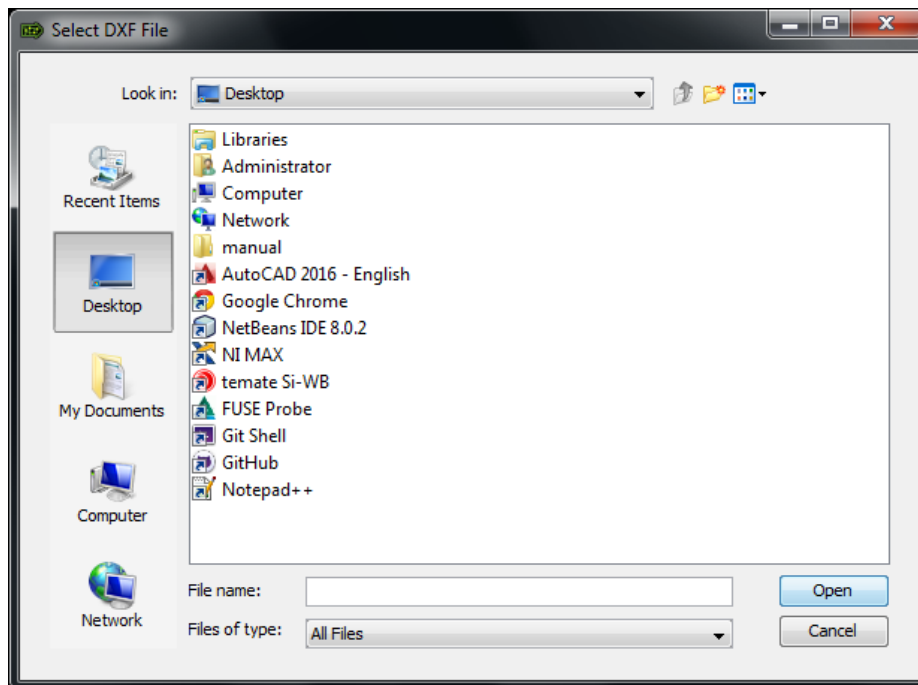
# DXF to Gerber Conversion: User Guide

## Introduction

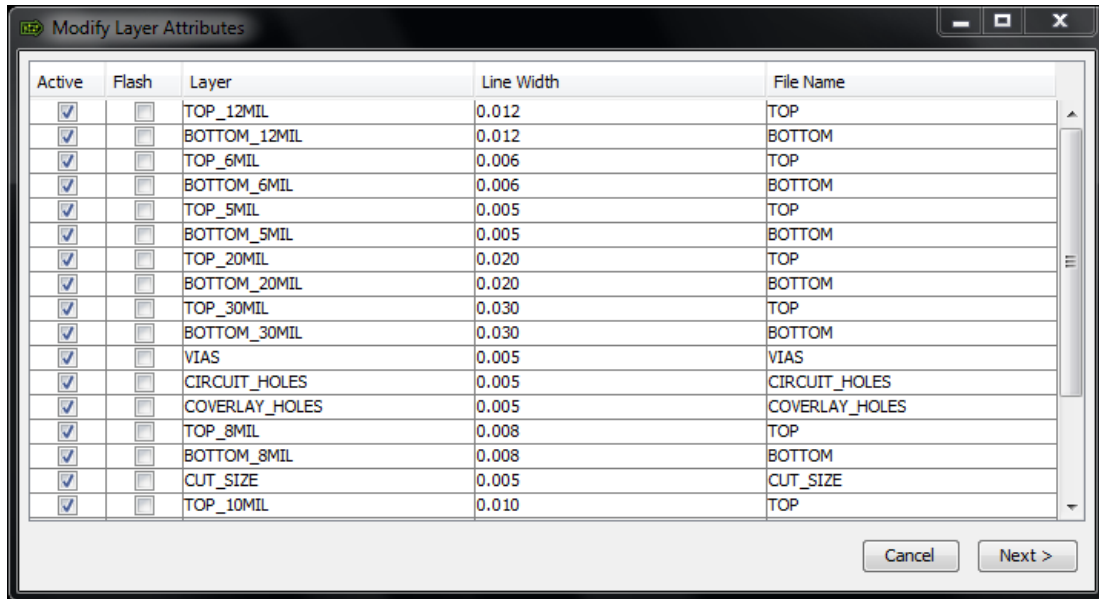
The DXF to Gerber Conversion program converts a DXF file to a Gerber RS-274-X file. To run the application, select the “DXF to Gerber Conversion.jar” file in the main directory.

## Step-by-Step

1. Select the DXF file to be converted to the Gerber format.



2. Modify the attributes of each layer.



### 2.1 Active

Determines whether the layer will be written to a Gerber file.

### 2.2 Flash

Determines whether all the circles in the layer will be “flashed” (filled-in). The line width for flashed circles is zero; non-circle entities will be assigned the specified line width.

### 2.3 Layer

The name of the layer in the DXF file. This field cannot be modified.

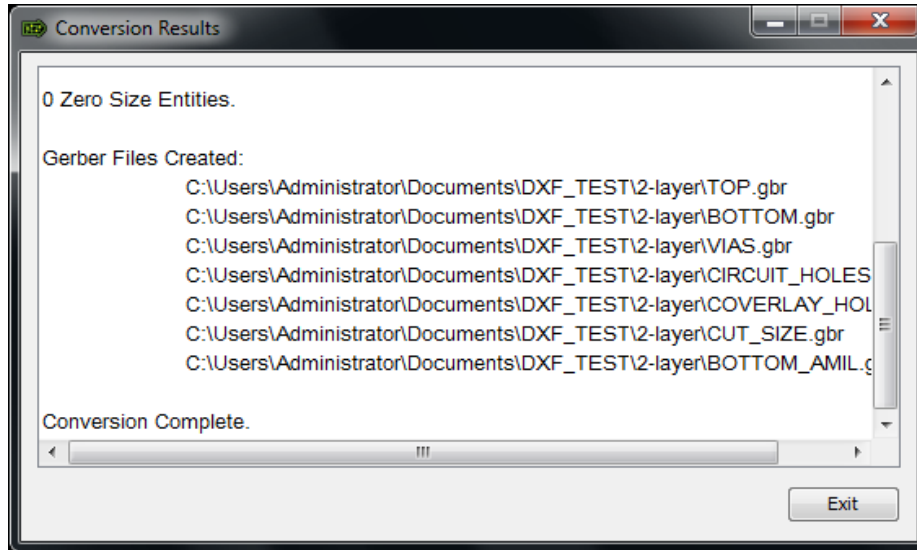
### 2.4 Line Width

The line width (in inches) to be assigned to a particular layer.

### 2.5 File Name

The file name of the Gerber output file. Geometry from all layers with the same Gerber output file name are merged into a single Gerber file.

3. Lastly, the status of the conversion process is displayed. Any deviation from CAD rules checking is reported.



## CAD Design Rules

**Important:** Although not described here, the designer should also abide by printed circuit design standards and practices.

1. The DXF to Gerber Conversion program supports AutoCAD DXF format CAD files.
2. DXF file geometry represents line centerlines. Endpoints are rounded. Geometry is processed using round apertures.
3. Line width is assigned by DXF file layer. All non-flashed entities within each DXF layer are assigned a single line width.
4. The DXF to Gerber Conversion program provides the option to flash (fill) all circles on a DXF layer. The line width is zero for flashed circles. The assigned line width is applied to all non-circles on a flashed layer.
5. Line width for the contents of each DXF file layer can be assigned within the DXF to Gerber Conversion program. Alternatively, the layer line width can be pre-assigned using the layer naming scheme (below).
6. The default Gerber output file name is x.gbr and the default line width is y mils, for DXF layers named in the format x\_yMIL. Otherwise, the default output file name is x.gbr, where x is the DXF layer name.
7. Geometry from all layers with the same Gerber output file name are merged into a single Gerber file.
8. Gerber coordinates are processed with absolute coordinates relative to the DXF 0,0 file origin.
9. Only DXF line, circle and arc entities are processed.
10. Only DXF geometry that reside entirely within the +x, +y quadrant are processed.
11. Zero length lines and zero radius circle and arcs are not processed.

## Gerber Format Specifications

- Gerber Type: Extended Gerber RS-274-X
- Output Units: Inches (English)
- Coordinates: Absolute
- Numeric Format (M.N):
  - Integer Digits (M): 2
  - Decimal Places (N): 5
- Image Mode: Positive (Normal)
- Arc/Circle Representation:
  - Circular Interpolation: Yes
  - Use Lines: No
  - Precision: Not Applicable
- Base Fill Line Width: 0.005
- Maximum Aperture: No Limit
- Suppression:
  - Suppress Leading Zeros: Yes
  - Suppress Trailing Zeros: No
  - Suppress Header: No
  - Suppress Trailer: No
- Gerber File Format:
  - End-of-Block (EOB) Character: \*
  - Parameter Delimiter: %
  - Line Breaks: <LF><CR>
- File Extension: .gbr

## Contact

Developer: Zachary Geier  
Email: [zdgeier@gmail.com](mailto:zdgeier@gmail.com)  
Cell: 434-426-1753