

DXF / DWG to Gerber Conversion Guide

Revision Date: December 10th, 2013

Sponsored by



Authored by

Simon Garrison, Numerical Innovations



Notice Regarding Sharing of this Document

Changes to this document or any portion of it without prior written permission from Bay Area Circuits and Numerical Innovations is not allowed. Sharing of this report is allowed, however, the report must be shared in full unless quoted with attribution to the author (Simon Garrison) and a link to www.bayareacircuits.com and www.numericalinnovations.com.

Questions should be directed to Bay Area Circuits via email: info@bacircuits.com.

© 2013-2014 Bay Area Circuits, Inc and Simon Garrison. All Rights Reserved.

ALL RIGHTS RESERVED.

The information contained in this publication represents the views of the authors only. These views are the authors' opinion at the original time of publication. This information is for educational and informational purposes only. The authors reserve their right to change or update their opinions based upon new conditions. This publication is provided with the understanding that the publisher is not engaged in rendering legal, accounting, or other professional advice. If legal advice or other expert professional advice is required, the services of a competent professional person should be sought. While every attempt has been made to verify the information provided in this publication, neither the authors nor their affiliates/partners assume any responsibility for any errors, inaccuracies or omissions. Any and all slights of people and organizations are completely unintentional.

Introduction

When designing a printed circuit board (PCB) often times the only design software available may be mechanical CAD software such as AutoCAD. Mechanical CAD software packages are able to generate a DXF or DWG file which contains the required data to produce a PCB, however, the DXF or DWG format is not preferred by PCB manufacturers due to the extra time required to interpret these files and potential for errors to occur. Instead the Gerber file format is preferred and will ensure faster production time and eliminate errors.

As a solution, DXF/DWG data can be converted to a Gerber format and Simon Garrison from Numerical Innovations has put together an excellent guide explaining exactly how to do this. We hope you find it useful!

Peter Brissette
Bay Area Circuits, Inc.

By Simon Garrison, Numerical Innovations

Printed circuit board (PCB) fabricators prefer Gerber files and many will not accept DXF/DWG data, however, it is estimated that over 10% of all PCB/RF electronic layouts are still designed using AutoCAD or other similar CAD tools (ex. SolidWorks, ProE). Why?

The power of AutoCAD is that it supports a wide array of entity types which can be used to construct various electronic designs; from the simplest to the most complex designs including RF/Microwave circuits, MEMs designs, chemical milling artwork or other high-end electronic layouts. Designers generally choose to layout their designs in AutoCAD when existing EDA/CAD tools do not possess the capabilities to meet unique geometrical requirements.

The initial approach used by a designer attempting to layout an electronic drawing in AutoCAD typically involves:

1. Layout a design in AutoCAD (using desired entity types)
2. Export to a DXF file
3. Convert the DXF to Gerber data (using any DXF to Gerber converter)

Seems simple enough, right? Wrong! This approach usually results in an astonished look on the designer's face when the resulting Gerber files are nowhere near their appearance in AutoCAD and not worthy of sending out for manufacturing. This has been a mystery which has plagued PCB/RF designers for well over 20 years and the purpose of this document is to provide proper guidelines for DXF/DWG to Gerber conversion thereby eliminating PCB fabrication "holds" and production errors.

Gerber Basics

To figure out the problems and how to overcome them we'll first take a look at the Gerber format; the format we'll be converting the AutoCAD drawing to. The Gerber format has been in existence for over 35 years and is simply an ASCII text file containing X,Y coordinates and shapes (i.e. apertures/dcodes). For more information about the Gerber format visit the following link:

http://en.wikipedia.org/wiki/Gerber_format

DXF/DWG Basics

DXF is short for Drawing Exchange Format and it is a format used to transfer drawings between Computer Aided Design (CAD) systems. DXF is widely used as the de facto standard in the engineering and construction industries and is a tagged data representation of all the information contained in an AutoCAD drawing file. Tagged data means that each data element in the file is preceded by an integer number that is called a group code. A group code's value indicates what type of data element follows. This value also indicates the meaning of a data element for a given object (or record) type. Virtually all user-specified information in a drawing file can be represented using the DXF format. DWG is essentially a binary version of DXF which tends to be smaller and cause less parsing errors during conversions.

Unlike the older Gerber format, every year it is expected there will be a new release of AutoCAD which include changes to the entity types and DXF/DWG format which continuously pose compatibility issues and other problems. More recently cross-over from 32-bit to 64-bit versions has become prevalent – causing additional compatibility issues.

Keep Your Drawings Simple

To convert AutoCAD drawings to Gerber format with the best results, simplify the design. DXF/DWG files support hundreds of entity types (ex. Hatches, Blocks/Inserts, 3D objects, Tapered Polylines, TrueType fonts & Mtext, Splines, Regions, Model/Paper Space, etc.) so the challenge becomes converting these fancy entity types into a simple Gerber file which only contains X, Y coordinates and shapes (ex. apertures/dcodes). For best results and successful Gerber conversions create layouts using the following 5 fundamental DXF/DWG entities.

The 5 Fundamental DXF/DWG Entities

Polylines (or LW Polylines), Circles, Text, Blocks and Inserts. Believe it or not, you can create just about any PCB/RF layout using only these 5 entities and they most closely represent objects in the Gerber data.

- 1. Polylines (or LW Polylines)** A series of connected lines and arcs that may have varying widths. This is the most common entity used to construct a PCB/RF layout and is used in about 80% of the layout. Polylines with width are used to draw traces (i.e. tracks) to connect pads. Polylines with 0-width are commonly used to create filled boundaries. Polylines may also be used to created square and donut pads.
- 2. Circles** Most commonly used to draw round pads.
- 3. Text** Used to draw silkscreen and labels. Typically using stick fonts which are AutoCAD compiled fonts (SHX) provide the best results and are output as trace segments in the Gerber file. Many DXF to Gerber conversion tools may support TrueType fonts however be aware that internal shapes may appear.
- 4. Blocks** Typically used to create a hierarchy in a design and provide an optimized method to add repeated entities into a drawing. Blocks are essentially individual drawings that are inserted into a parent drawing. During the insertion blocks can be rotated, scaled, and transformed in any direction.
- 5. Inserts** Blocks can be inserted multiple times as an array using Minsert (Multiple Inserts) which allows an array of blocks to be either rectangular or polar.

How to Avoid the Most Common DXF to Gerber Conversion Pitfalls (Frequently Asked Questions)

Q: How do I create a filled area in my Gerber file?

A: To create a filled area first draw a closed, continuous boundary (with 0-width polyline) that does not self-intersect. Remember, Gerber has stringent fill rules: All boundaries must be continuous with no overlapping or self-intersections (like a bow-tie). The direction is not significant for filled boundaries (i.e clockwise or counter-clockwise).

Q: In the Gerber file my boundaries are still not filled. Why?

A: Most likely your boundaries are not closed & continuous (as described above). In many cases you may have a drawing which consists of individual line and arc segments – which are not continuous. We recommend that you convert those boundaries to Closed, 0-width polylines using the command “Pedit” and then select “Join”.

Q: How do I create a Clear Area (i.e Island) inside a filled Polygon?

A: In the Gerber format, there is no such thing as island/hole detection. Fortunately Gerber data supports Composites which allow you to explicitly define what to make clear or dark using the Gerber commands “LPD” (Layer Polarity Dark) and “LPC” (Layer Polarity Clear). The most reliable way to define a composite Gerber file is to create separate DXF layers for each polarity change. For example, if you require a dark rectangular boundary with many clear holes, you should create two layers: One layer which contains only the rectangular boundary and a second layer which only contains the holes. During DXF to Gerber conversion you can create a composite Gerber merge and assign the polarity for each layer.

Note: Some conversion tools will have a feature called “Polygon De-Embedding” where the conversion tool will decipher which objects are contained inside of other objects and logically try to determine what should be an island or hole and then explicitly output that information to the Gerber file.

Q: Should I use Solid Hatch to fill an area?

A: Whenever possible refrain from using solid hatches in your AutoCAD drawing. Solid hatches look good in AutoCAD because they are one of few entities in AutoCAD that appeared filled (on the AutoCAD screen). However, a solid hatch is drawn on top of your existing drawing causing duplicate data in the DXF and results from a Gerber conversion can be messy because of the duplicate objects. If you must use solid hatches please make sure your conversion software supports solid hatching and has the ability to detect islands/holes when necessary.

Q: How should I view the Gerber files after the DXF to Gerber conversion?

A: We recommend using a commercial grade Gerber Viewer.

Additional Resources

For questions about DXF/DWG to Gerber conversion contact Bay Area Circuits at info@bacircuits.com or 855-811-1975 or Numerical Innovations at sales@numericalinnovations.com or 866-528-9274. We would be glad to help!

PCB Creator by Bay Area Circuits; Free, easy to use PCB design software which allows users to create a schematic and convert into a PCB layout to generate required files to fabricate a printed circuit board. Real-time pricing and instant ordering options are included. Download at: www.pcbcreator.com

EasyGerb by Numerical Innovations; Allows the export of Gerber files from inside AutoCAD. Simply select any entities and create Gerber files – instantly! Download at: <http://www.numericalinnovations.com/collections/easygerb>

Questions? Comments?

Bay Area Circuits, Inc.

44358 Old Warm Springs Blvd

Fremont, CA 94538

Phone: 855-811-1975

General: info@bacircuits.com

Support: support@bacircuits.com

Website: www.bayareacircuits.com

About Bay Area Circuits

Bay Area Circuits has been serving the printed circuit board (PCB) manufacturing needs of high-tech electronics manufacturers, contract assemblers and design engineers for nearly 40 years. A focus on quick-turn prototyping and production leveraging innovative and high quality designs has made Bay Area Circuits the manufacturer of choice for customers around the world.

Numerical Innovations, a Division of Caneberra Technologies, Inc.

1888 Kalakaua Ave, Suite #C312

Honolulu, HI 96815

Phone: 866-528-9274

General: sales@numericalinnovations.com

Support: support@numericalinnovations.com

Website: www.numericalinnovations.com

About Numerical Innovations

Numerical Innovations (a division of Caneberra Technologies, Inc.) develops software tools for PCB, MEMs, IC, Microwave/RF, and optical components. We are proud to boast a customer base of over 1000 unique companies.