

Community Home

Thread

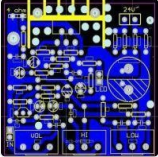
Featured

Image

Video

Search In

Enter Keywords for Search



Gerber File

Group Administrators: 1 | Group Member: 38 | Group Threads: 38

Gerber is used in PCB fabrication data. PCBs are designed on a specialized electronic design automation (EDA) or a computer-aided design (CAD) system. The CAD systems output PCB fabrication data to allow manufacturing. This data typically contains a Gerber file for each image layer (copper layers, solder mask, legend or silk...).

Thread

Reply

First


Previous

General Introduction of Gerber File

This Author Only

Favorite

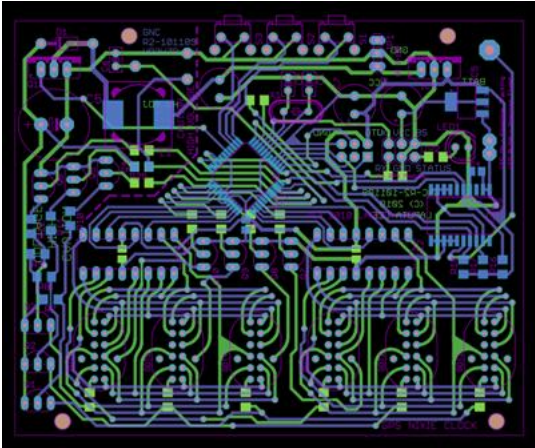
Reply



Dick

[Leave A Message](#)

[Follow](#)



The Gerber format is an open ASCII vector format for 2D binary images. It is the de facto standard used by printed circuit board (PCB) industry software. During bare board fabrication Gerber is the standard input format for photoplotters and all other fabrication equipment needing image data, such as imagers or automated optical inspection (AOI) machines or for viewing reference data in different departments. Gerber files also contains a 'stencil' and the central locations of components to allow the PCB assembler to create the stencil and place and bond the components.

There are three major generations of Gerber format:

Gerber X2. This is the current Gerber format.

Extended Gerber, or RS-274X.

Standard Gerber, or RS-274-D. This obsolete format was revoked.

The standard file extension is .GBR or .gbr though other extensions are also used. Some abbreviations that you often see:

GTL=Top copper layer

GBL=Bottom copper layer

GTS=Top solder mask layer

GBS=Bottom solder mask layer

GTO=Top silk screen layer

GBO=Bottom silk screen layer

GTP=Top paste file (top stencil file)

Community Home

Thread

Featured

Image

Video

Search In

Enter Keywords for Search

GKO=Outline layer


DRL=Drill layer

147

likes

Likes

Statement: This post is only the personal view of the author and does not represent the opinions of ALLPCB.com.



Miguel Angel

[Leave A Message](#)

[Follow](#)

Appreciate your sharing.Please keep up your updates.

[Reply](#)

Thread

Reply

First

Previous

- About Us

[About ALLPCB](#)

[Why Us](#)

[News](#)

Products & Services

[PCB Capabilities](#)

[PCB Instant Quote](#)

Beginners' Guides

[PCB FAQs](#)

[Operation Manual](#)

[Earn ALLPCB Points](#)

Official Bulletin

[Disclaimer](#)

[Terms of Service](#)

[Privacy Policy](#)

[PCB Projects Sponsorship](#)

[Electronic Promotion Activity](#)

Follow Us

[Facebook](#)

[Twitter](#)

[LinkedIn](#)

[Google+](#)

[YouTube](#)

Our Preferred Partners:

















Follow Us:











Contact Us: Please send Gerbers to service@ALLPCB.com for quotation

© ALLPCB.com,All Rights Reserved [Privacy Policy](#) [Sitemap](#)