

Gerber Expert

Gerber-to-CAD reverse engineering solution for PCB assembly

Benefits

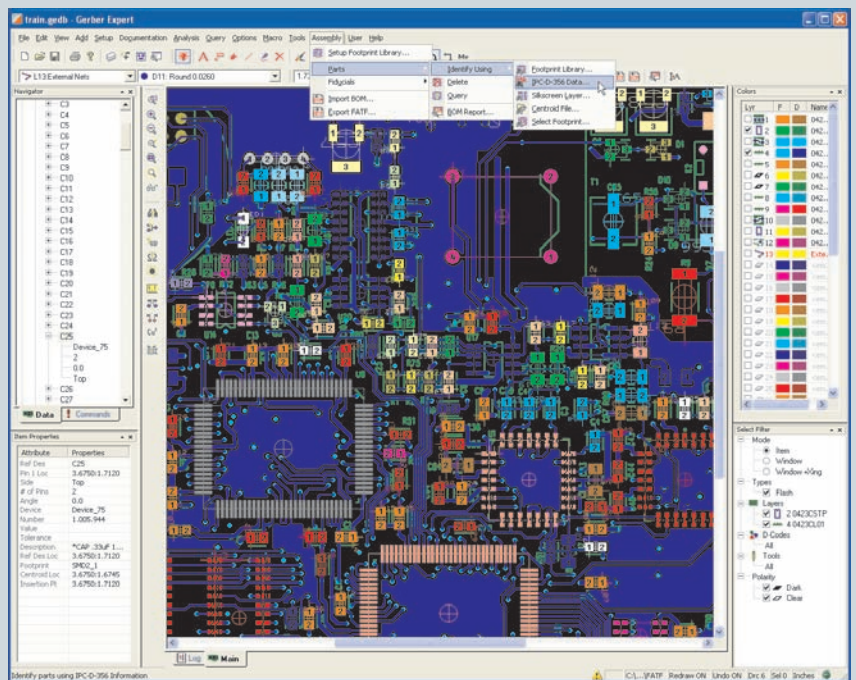
- Accelerate new product introduction efforts for PCB assembly and test
- Leverage a highly automated reverse engineering tool that extracts component and board information

Features

- Provides full support for traditional industry data formats (Gerber, Drill, Mill, IPC-D-356, HPGL, DXF and more).
- Assembly reverse engineering creates intelligent part information where none previously existed. Five unique levels of automated footprint/centroid extraction are available, allowing fast and easy processing of even high-density board designs.
- HyperNETLIST generation creates netlist information for even the most complex designs.
- Import wizard loads all industry formats (Gerber, Drill, Mill, HPGL, DXF and more).

Summary

Gerber Expert is a Gerber-to-CAD reverse engineering tool. By using its highly automated tools and processes, you are able to quickly reverse engineer a set of Gerber files into usable file formats including Fat-f. The Fat-f file is an industry standard, generic CAD format that is owned by Siemens PLM Software. It can be used to import data into any of the Tecnomatix® software for electronics manufacturing solutions such as Unicam FX, Assembly and Test Expert and MES.



The ability to reverse engineer Gerber data is critical since Gerber data is the most common format supplied to PCB manufacturers. All CAD designs are exported into Gerber files for production tooling and photo artwork generation. Often, this is the only data that users can readily obtain.

TECNOMATIX

www.siemens.com/tecnomatix

SIEMENS

Gerber Expert

Features *continued*

- Database navigator provides quick and easy access to important design data and frequently-used functions; no need to hunt through menus and toolbars.
- HyperNETLIST comparison provides true graphical verification; during comparison, the solution includes a pin-point error feature, not found in competitive systems, that quickly locates your shorts and opens.

Supported file formats for import

- Barco DPF
- DXF (AutoCAD 2000)
- Excellon and Excellon 2
- Fire 9XXX
- Gerber 274-D and 274-X
- HPGL and HPGL/2
- IPC-2581 (OffSpring)
- IPC-D-356 and IPC-D-356A
- ODB++
- ODB++ (X)
- Sieb & Meyer
- Takeuchi

System requirements

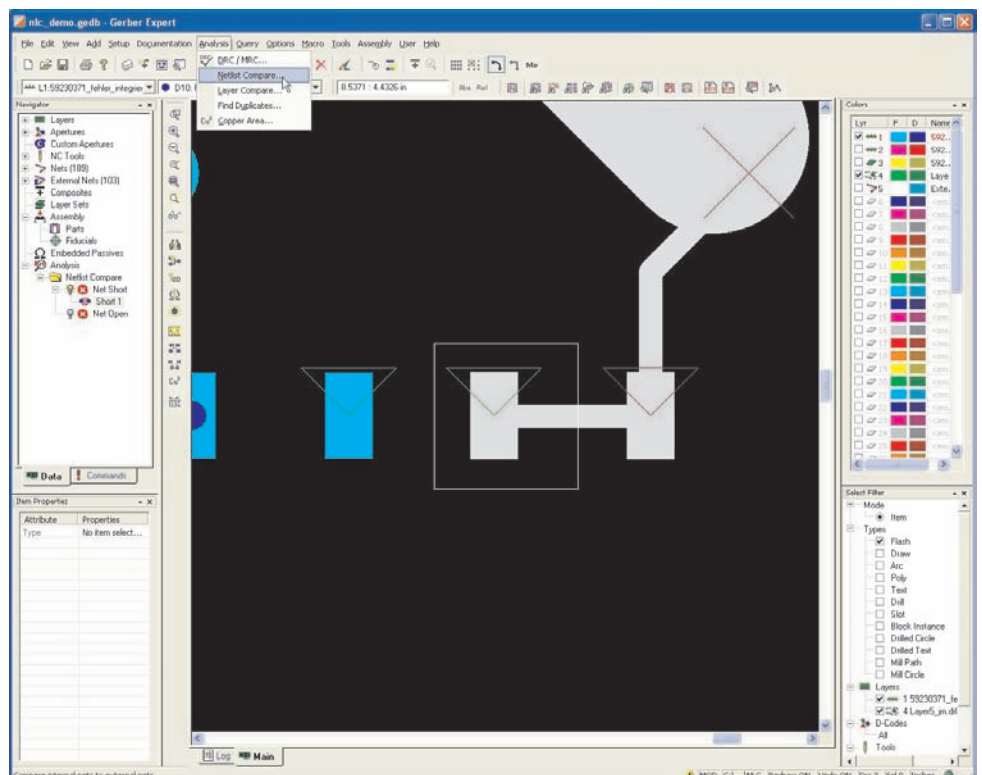
- IBM PC-compatible with at least a 1 GHz Pentium CPU
- For best graphics display, a 17" or larger monitor with at least 1280 x 1024 resolution is recommended
- At least 512 MB of RAM and 30 MB of hard disk space
- Operating systems supported: Windows 2000, Windows XP Home, Windows XP Professional

Gerber data is "non-intelligent". It contains only drawn lines and pad flashes, with no correlation to the parts, nets and routes in the circuit. Automatically extracting electrical information directly from the Gerber files to relate parts, pins and connectivity is virtually impossible without reverse engineering.

Gerber Expert solves this problem. It lets you quickly identify the lines and pads in your Gerber file that define components, part parameters and connectivity. Gerber Expert then automatically extracts the component and board information – including all nets, vias and test points – from the Gerber data and can subsequently export this data into many formats including the Fat-f format. Using any of the five unique levels of automated footprint/centroid extraction, you can easily reverse engineer your PCB. With the right data set, you can complete the entire process for even your most complex/high-density boards in a matter of minutes, not hours.

Basic features

- Single file, 32-bit "intelligent" database structure
- High-speed graphics
- 999 layers
- 9999 apertures
- Integrated printing
- Polygon support (intelligent raster fill, POEX/POIN, G36/G37)
- Compositing (pos and neg merging, paint and scratch; see also object polarity under advanced features)
- ODB++ import and export (support for valor and frontline solutions)
- ODB++XML import and export (support for valor and frontline solutions)
- OffSpring import and export (support for IPC-2581)
- Import wizard (detects Gerber, Drill, Mill, HPGL, DXF and other formats; converts aperture tables and reads automatically)
- Automatic aperture and NC tool list converter (rules-based, supports any system)
- Gerber 274D and 274X import and export



- MDA AutoPlot import and export (Fire 9XXX, symbolic sciences)
- Barco DPF import and export
- NC data import and export (support for excellon, Sieb & Meyer)
- IPC-D-356 and 356A netlist import and export
- IPC-D-350 export
- DXF import and export
- HPGL, HPGL/2 import and export
- PostScript export
- Bitmap output (Windows BMP, black and white, color, up to 1000 dpi)
- Editing capabilities (move, copy, delete, rotate, mirror, trim, join and more)
- Adding capabilities (flash, line, arc, circle, rectangle, polygon, text and more)

Information features

- Query data (objects, D-codes, nets, net length, user data)
- Measure data (point-to-point, center-to-center, edge-to-edge)
- Reports (apertures, drill, mill)
- Redline notes (document changes, problems and more)
- Dimensioning (create fabrication drawings, document board size and more)
- Drawing (drill charts, balloon notes and more)

Macro-scripting features

- Powerful macro developer (drag-and-drop capabilities and more)
- Load, run and record (support for startup, shutdown and onload macros)
- Support for "nested" macros (embed macros within macros)
- Integrated debugger (watch points, stops, step-through and more)

Advanced features

- Database navigator (allows quick and easy access to layers, apertures, netlists and more)
- Custom aperture editor (build special pads, logos and more)
- Object level polarity (work with pos and neg data on the fly; eliminate the need for composites)
- Interactive data grouping (group objects together for fast and easy editing)
- Merge multiple PCB files
- Layer scaling
- Layer sets (define layer sets for blind and buried vias and MCM stackup)
- Interactive draw-to-flash conversion (see automatic draw-to-flash below)
- Automatic draw-to-flash conversion (convert all layers at once; use mask layer as guide and more)
- Draw-to-custom conversion (convert drawn graphics into a custom aperture)
- Raster-to-vector conversion (convert raster fill polygons to vector fill for older plotters)
- Drawn text-to-true text conversion
- HyperNETLIST generation (supports SMT, through-hole, blind and buried and MCM)
- Silkscreen clipping (clips silkscreen away from pads)
- Composite-to-layer (flattens any composite down to a single positive layer)

- Pad removal (isolated or redundant)
- Tear-dropping (true-shape and snoman)
- Film spread (tile all layers to single sheet of film)
- Basic panelization (step and repeat your data fast)
- Intelligent step and repeat codes (Gerber and NC data)
- Automated stencil enhancement (quickly enhance generic paste stencils with IPC-7525 shape definitions)
- Automatic solder mask generation and optimization
- Automatic paste mask generation and optimization
- NC-Drill editing (create, add, edit, sort, multiple tool tables, advanced manufacturing features and more)

Assembly reverse engineering

- Automated component footprint identification (identify using: footprint library, IPC-D-356, silkscreen, centroid or manual ID)
- Integrated footprint library (control "master" footprints, device creation and association and pin numbering)
- BOM import (finalize reverse engineering with part numbers, device descriptions, values and tolerance data)
- Fat-f export (support for industry standard Tecnomatix/UNICAM)
- Generic ASCII Centroid/COM export (universal output support with full access to all data elements)

For more information, contact:

Acuity Electronics, Inc.
PO Box 1015
Belmont, CA 94002

Toll-Free: 855-642-2848
Email: info@acuityelectronics.com

www.acuityelectronics.com

© 2010 Siemens Product Lifecycle Management Software Inc. All rights reserved. Siemens and the Siemens logo are registered trademarks of Siemens AG. D-Cubed, Femap, Geolus, GO PLM, I-deas, Insight, JT, NX, Parasolid, Solid Edge, Teamcenter, Tecnomatix and Velocity Series are trademarks or registered trademarks of Siemens Product Lifecycle Management Software Inc. or its subsidiaries in the United States and in other countries. All other logos, trademarks, registered trademarks or service marks used herein are the property of their respective holders.
X3 5208 12/10 B