

© Norlinvest Ltd, BVI. Visionics is a trade name of Norlinvest Ltd. All Rights Reserved.

No part of the document can be reproduced in any form or by any means without the prior written permission of Visionics. This document is subjected to change without notice. Visionics will make changes in a manner that will not affect dependent systems.

Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Visionics.

Visionics, EDWinXP, Docone, EDComX, SimWinXP and Mixed Mode Simulator and their respective logos are trademarks or registered trademarks of Visionics. Unauthorized duplication of this work may also be prohibited by local statute.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Visionics. The information contained herein is the proprietary and confidential information of Visionics or its licensors, and is supplied subject to, and may be used only by Visionics's customer in accordance with, a written agreement between Visionics and its customer. Except as may be explicitly set forth in such agreement, Visionics does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Visionics does not warrant that use of such information will not infringe any third party rights, nor does Visionics assume any liability for damages or costs of any kind that may result from use of such information.

Reverse Engineering

Introduction

Here it describes the sequence of operation of reconstructing a complete Project Database.

The number of steps required to reconstruct a complete project database is determined by the complexity, structure and contents of imported graphic files. Most complete and successful reconstruction can be achieved using full set of artworks in Gerber ASCII format because of their straightforward connection to PCB layout. Import of DXF files are provided mainly for reconstruction of geometrically complicated board outlines. Much depends on purpose of reconstruction. In many cases imported data will allow after edition to generate revised set of artworks without necessity of reconstructing complete Project Database.

Note: No net-list or schematics information will be there but so created simple PCB layout database allows generating revised set of artworks in Gerber ASCII format as well as NC Drill output.

Here are a few examples where graphics imports may be used

- a. Complicated PCB outline with or without cutouts have been drawn using AutoCAD. Resulting graphics can be imported and used to reconstruct same PCB outline in the Project Database.
- b. User has old Gerber ASCII files for a PCB that requires changes in the artwork(s) these files can be imported and updated to obtain new corrected set of artworks.
- c. User has old Gerber ASCII files for a PCB but needs matching NC Drill data. The files can be imported. Graphic Import Editor has tools that allow reconstruction of drilling data.

Old Gerber ASCII files may be imported, edited and used as templates to reconstruct complete Project Database.

The import category for a double sided board should possess artwork of

- ? component layer
- ? solder layer
- ? component print and
- ? Solder mask.

The via position and thru pad position need to take input from the solder masks layer only in which the Dcode filter should be applied as per the via and pad requirements.

The via pads and thru pads require information from all the layers except *component print*

Obtain the required gerber files and for the import process, select Fabrication manager -> File -> Gerber/ Excellon /DXF/ HPGL Viewer. A file import dialog box will be open to import files as shown below. Select the *Files of type* as *Gerber ASCII files (*.GBR)*. Select the required gerber file. Fig : 1

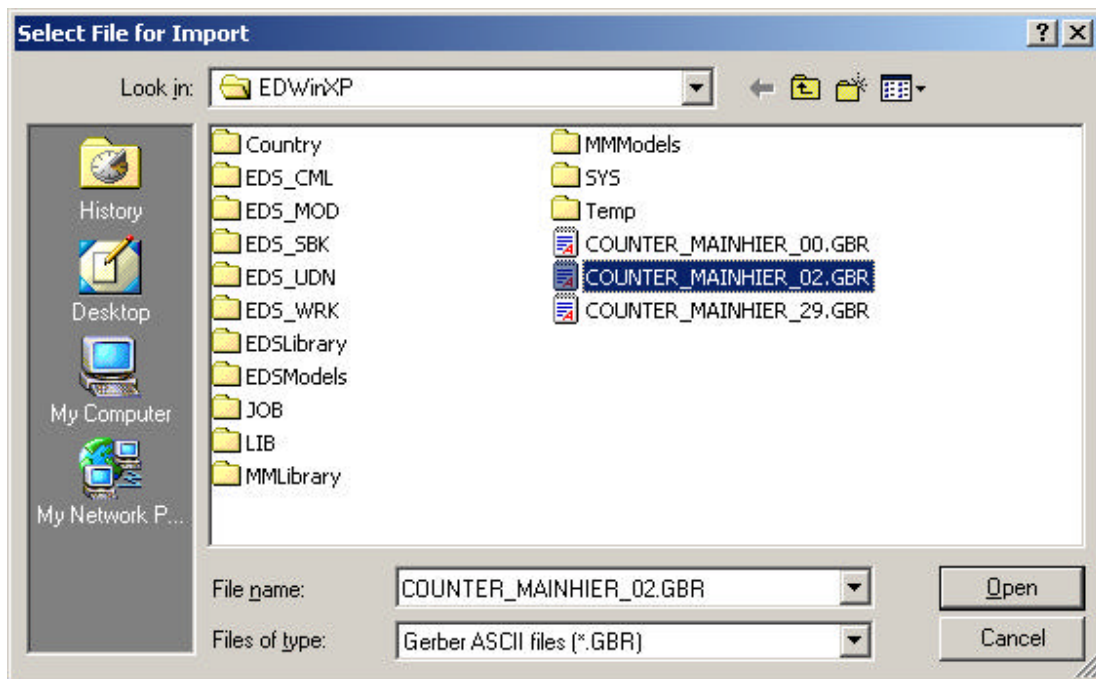


Fig: 1

Click open in the above in the above window top open the *Import Parameters* dialog as below in Fig: 2

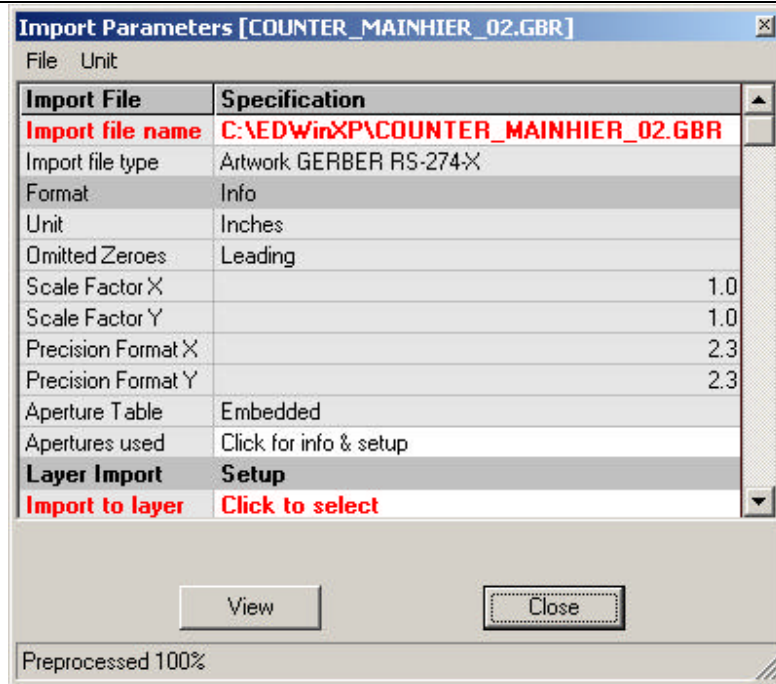


Fig:2

Select the appropriate layer for the layer which is selected for importing. Here Artwork Top layer is selected. Fig: 3

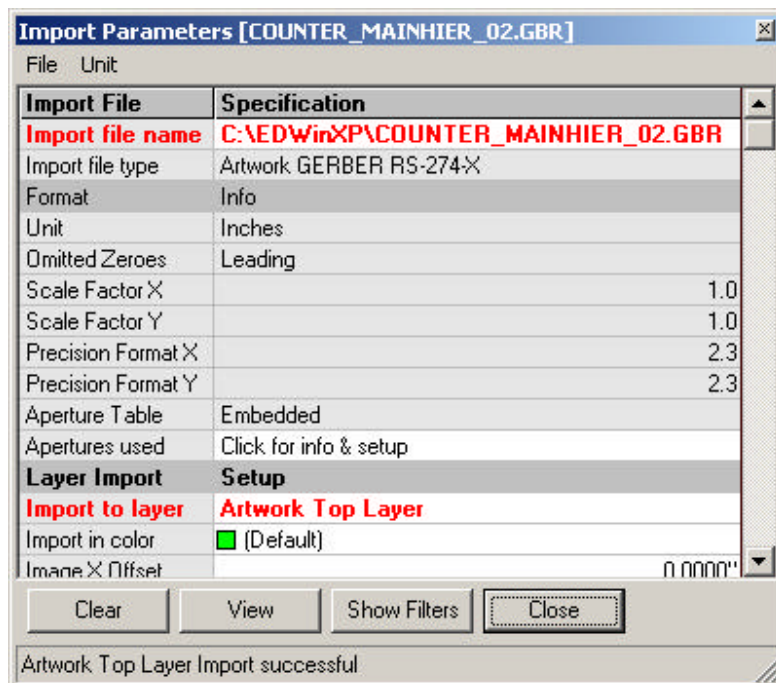


Fig: 3

Select the *Show Filters* option so that the unwanted items can be filtered off Fig : 4

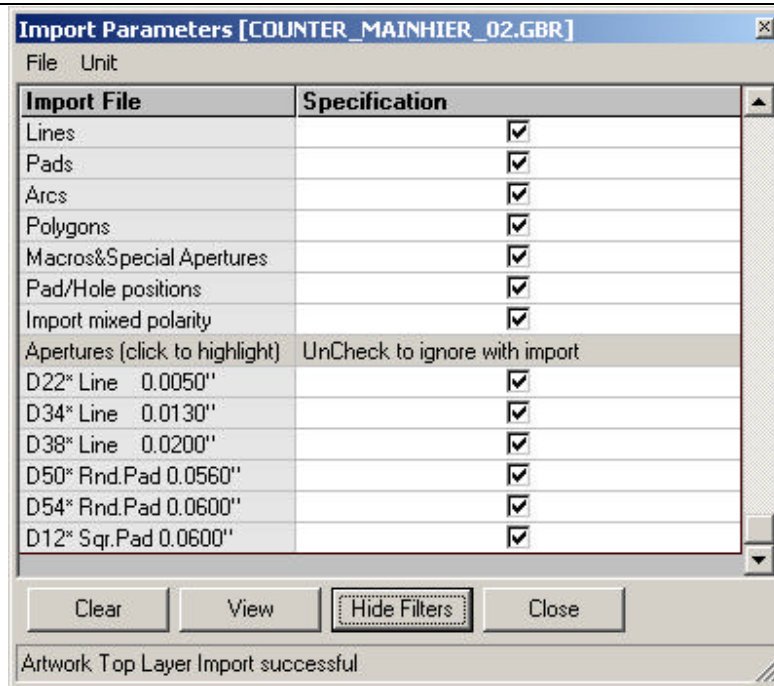


Fig : 4

Uncheck the Imported elements items which are not required. Since here the artwork top layer is imported, the board items are excluded by unchecking D22* line 0.0050" Fig : 5

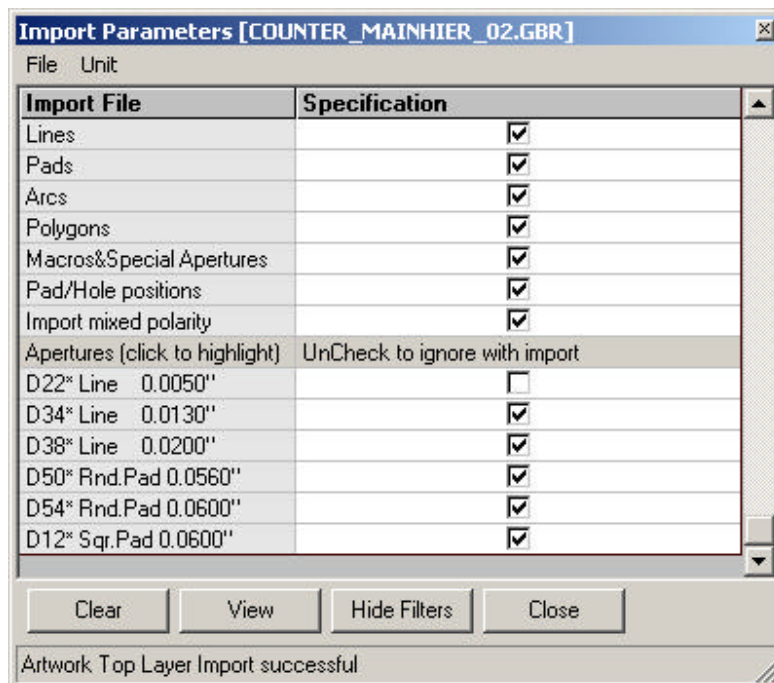


Fig: 5

Click *Close* to exit.

Note: For Multi layer PCB inner layers can be added by clicking “Add inner Layer” in Layer tool Bar (Fabrication Graphic viewer and import editor)

Repeat the procedure to import the other layers also in the Graphics Import editor.

After importing all gerber files to the corresponding layers, select Tools -> Import Categories for editing from the Graphics Import Editor. Fig : 6

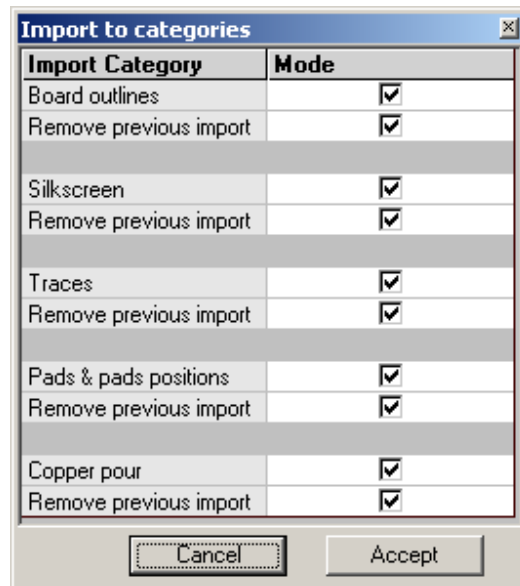


Fig: 6

Click *Accept* so that the changes are effected in the Fabrication Manager. In Fabrication Manager, the board appears as a template.

Reconstruction of Project

Reconstruction of complete Project Database involves additional operation sequenced as follows:

- a. Reconstruction of board outline
- b. Reconstruction of board cutouts (if applies)
- c. Reconstruction of template outlines (if applies)
- d. Reconstruction of components
- e. Reconstruction of via holes
- f. Reconstruction of traces
- g. Reconstruction of net-list
- h. Reconstruction of copper planes and copper pour areas.

Executing above-mentioned steps allows to create complete Project Database in such shape that further edition is possible using Schematic and Layout Editors.

This can be availed from Tools -> Reconstruction from Graphics. Fig: 7

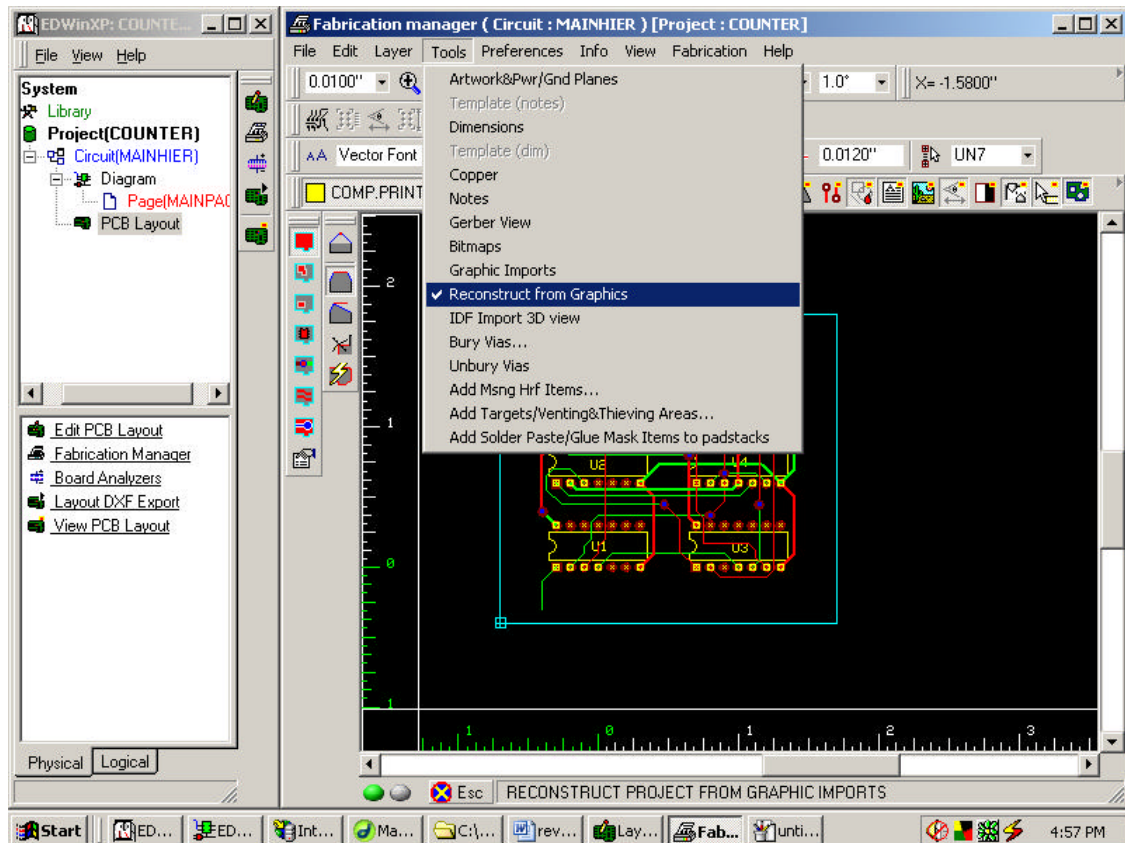


Fig : 7

Reconstruct board outline

Board outline may be reconstructed automatically by pointing one of the elements in the polygon. Other way of reconstruction is by dragging selected vertex of already existing outline and anchoring it on selected element in the reconstruction template.

Select the vertex of already existing outline and anchor it on the selected element in the reconstruction template. This action immediately replaces the reconstruction template which is the final board.

Reconstructing Components

Before reconstructing the components, select Fabrication Manager -> Tools -> Graphic imports-> Select the Active category as Thrupads Position-> Create graphic item (Second function tool) -> Assign Holes (F2) , Give the required Hole diameter (eg : .0350") , with some hole category. Fig:

8

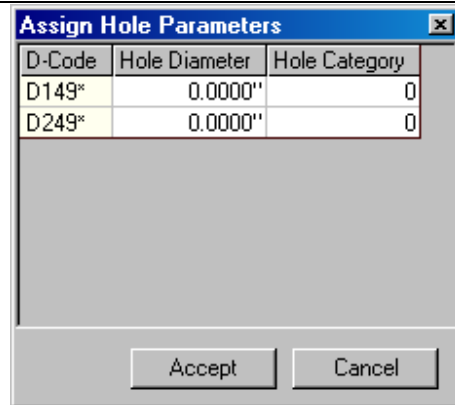


Fig: 8

Then select Fabrication Manager -> Tools -> Reconstruct from graphics -> Reconstruct components -> Reconstruct components/ Packages from graphics (F1) -> Block select the required component -> Select Finish reconstruction -> and give the *Value* for the properties given.

Fig: 9

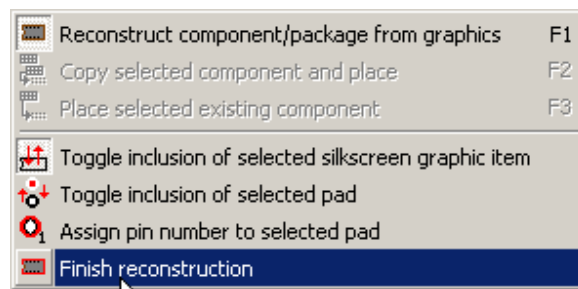


Fig : 9

Scratch part creates a new part in the project library or an existing library part (Assign -> Library Part) can also be used for replicating the components

If Library Part is selected (Assign -> Library Part), the corresponding part will be loaded from the library and will be placed in the position of the package template. Fig: 10

Component Name: Name of the component

Assign: Library Part and Scratch Part

Part Name: Name of the part to be loaded

Do not remove package template: If checked the template will remain on the board.

Package Name: The name of the package

Reconstruct component from package template	
Property	Value
Component Name	FLIP
Assign	Scratch part
Part Name	74
Do not remove package template	<input checked="" type="checkbox"/>
Package Name	DIP

Accept Cancel

Fig: 10

Hence reconstruct the traces, vias and copper automatically.

Refresh the Layout Editor to view the complete board that has been reconstructed.