International Journal For Technological Research In Engineering
Volume 2, Issue 7, March-2015 ISSN (Online): 2347 - 4718

REVERSE ENGINEERING FOR ERROR DETECTIONS OF PRINTED CIRCUIT BOARD (PCB)

J. Akku Naidu[1], N. Sowjanya[2], T. A. S. Bhargav[3], R. Ganesh Sainath[4], M. Sujatha[5]
[1,2,3,4]Student, [5]Assistant Professor
Department of ECE, Lendi Institute of Engineering and Technology, Vizianagaram, India.

Abstract: Old PCBs are sometimes difficult to support Increasing costs of spare parts and Non-existent procurement sources. For that purpose in this project, a program was written which can automatically reverse-engineer a net list of a one- or two-layer printed circuit board (PCBs) from photos of the board. Reverse-engineering a printed circuit board (PCB) is useful for purposes such as repairing equipment for which component-level documentation is not available, integrating a poorly-documented board into a system, and for identifying obsolete parts and represent Missing manufacturing files. The program is written in Java. It is an offline, non-interactive process, but various intermediate images are presented for testing and development purposes.

I. INTRODUCTION

A. scope and assumptions:

The program takes as input a photograph of each side of board (or copper bearing side in the case of one layer board). The program produces as output a net list which describes the connections between the components and to find the missing manufacturing files.

B.pcb structure:

PCB consists of printed conductive wires (called PCB tracks) and component layouts attached to an insulator sheet. The insulator sheet is a sub-states and it is made up of glass epoxy or pertinax. The finished PCB layout looks exactly the same as physical PCB with all it's layers representing the layers physically present in the PCB. To avoid a chance for corrosion and abrasion occurs in copper tracks, a plastic layer that is solder mask required. This protects the all copper track and spreading from exposure to surrounding atmosphere. The commonly used solder mask is green coloured and that's why PCB are green coloured in appearance. Solder mask is applied to the top and bottom layer of PCB depending on the presence of copper components. The part reference information for various components on top of solder mask represented by silk screen printing. This silk screen provides readable information on component part number and placement of components. The original solder mask was pattern-printed, using screen printing, but this technique has limitations on definition, with the result that photo image able materials are used for most applications. As with any resist process, there is a choice between dry film materials, where a known thickness of mask is laid over the board, and liquid processes, where a deposit of approximately even thickness is attempted, typically in the case of solder resist by curtain coating. The gang solder mask window will normally be able to be manufactured by screen printing, using 0.38 mm spacing; however, the pocket solder mask type will require clearances of 75–125 µm, and must therefore be made with photo image able resist. When solder resist is applied over tracks whose coating will melt when soldered (that is, not SMOBC), the tracks should be no more than 1.3 mm wide, or else have holes in them so that the coating can bond through to the underlying laminate. The need for reverse engineering of PCB's comes about for a variety of reasons chief among them is the need to replace an obsolete board that is no longer available from the original manufacturer (OEM) [2]. Often companies employ other enterprise to reverse engineer their own designs, because they have little or no documentation for their own product or their current design firm or manufacturer refuses to release design details and manufacturing files, hence a nuisance for most of organizations. The PCB manufacturer itself conduct a preliminary test after fabricating the PCB to ensure that All the connection in gerber files supplied by the end user get inter connected according to the gerber files. Checking the internal connections with in a multilayered board are very difficult. However the end user can conduct test like visual inspection on receiving the PCB.

II. PROCESS

A. IMAGE INPUT

The images should be cropped manually so that only board is visible. The photos should be taken with in the board on a dark background so that any background visible through holes does not look like a copper. An example cropped image is shown in fig1.Later processing steps start from a binarized image. To produce this image, the colour input image is converted to gray scale and then threshold. To convert to gray scale, each pixel's colour value is projected onto a line in RGB space. The position along this line becomes the pixel's new gray level. The direction of the line is found by applying the k-means algorithm to the RGB pixel values to classify them into two groups corresponding to board and copper colours. Since there may be lighting variation, the classification of pixels is not used to threshold the image. The centers of the clusters found by this classification are the two points which define the line onto which pixel values are projected to find gray scale values. This results in a high contrast gray scale image regardless of the actual colours present in the image.

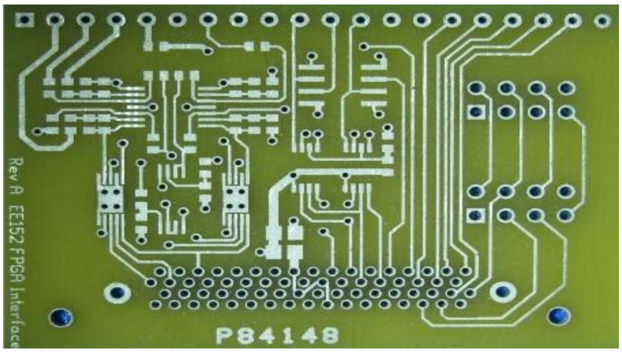

Fig: 1 Input image

After gaining the raw PCB image, the image has to go through the image processing stage. Even though, the high resolution of scanner is used, there are still some minor defects in the image, due to data lost during transmission and improper reflection or absorption of light from the uneven or wavy surfaces. Image which captured in Stage 2 is in truecolor type as shown in Fig. 2. True color or PCB image is single or double whose pixel values specify intensity values.

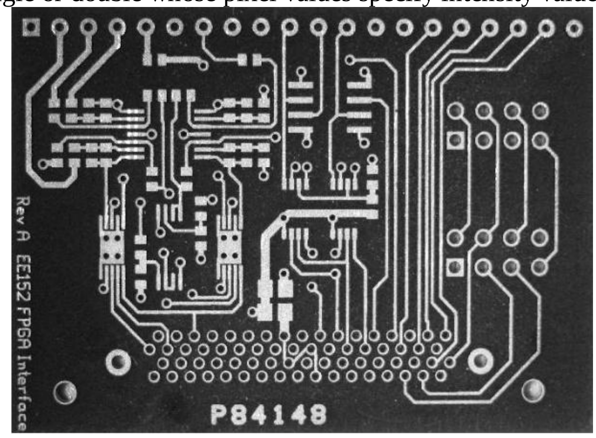

Fig: 2 Gray scale images

Illumination may vary across the image. Note that in figure1,the upper left corner is darker than the lower right corner. To correct for this, the lighting is estimated by applying a median filter with a large window. The resulting lighting image is substrated from the gray scale image to produce an unlit image which has less variation than the original image, and which is more suited to a global threshold in figure2. Even though the image has converted to grayscale type but still it has some defects that need to be eliminated.

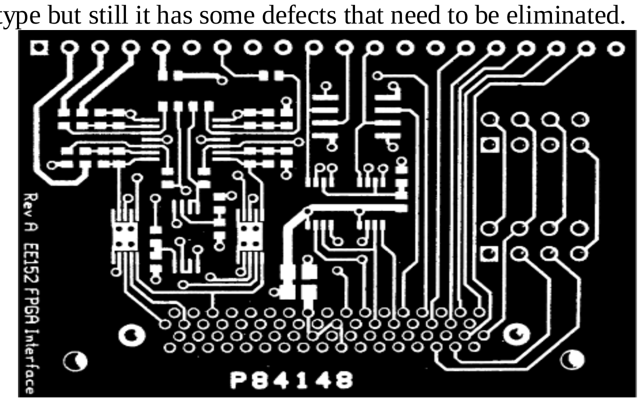

Fig: 3 Threshold image

Digital image which is capture from Stage 2 is prone to be affected by noise. Noise is the result of errors that occur during image acquisition process, which result in pixel values. It does not reflect the true intensities of the real scene. Filtering is a preliminary process in image processing applications. It is a fundamental operation in low level computer vision, aiming to restore noisy image to its noiseless counterpart. Otsu method is used for finding the thershold value by minimizing if intra class variance and as well asinter class variance of intensity levels of image.

.inter-class variance

$\sigma^2 w(t)=w1(t)\sigma1^2(t)+w2(t)\sigma2^2(t)$.

Intra class variance

$\sigma^2 b(t)=\sigma^2-\sigma^2 w(t)=w1(t).w2(t)[\mu1(t)-\mu2(t)]^2$.

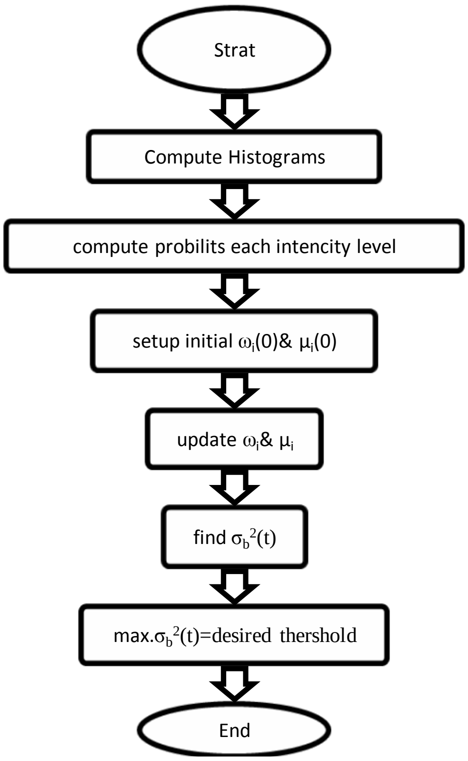

Fig: 4 Flow chart of Otsu method for threshold.

Finally Otsu's method is used to find a global threshold for the image to classify each pixel as PCB substrate or copper. Holes in the board will appear primarily black because of the dark background. Opening and closing morphological operations are performed to reduce noise in the threshold. image.

B. Finding nets

Each connected region in the threshold image is considered a net. For two-layer boards, nets are found on each layer independently and merged in a later step. The Open CV function cv : : find contours is used to find the boundaries of connected regions and any holes within them. Each hole is likely to correspond to a drilled hole in the PCB, so these holes are stored for later use when merging to-layer boards.

Nets are sequentially numbered so that they can be identified in the output netlist. Net numbers are unique across all layers.

C. Finding components

Components are found by comparing each footprint in a library of component templates to the threshold image. Each template is a gray scale image with the Sam e scale as the board image. A black pixel indicates a location where no copper is expected, a white pixel indicates a location where copper is expected, and a gray pixel is ignored. A minimum fraction of the white and a minimum fraction of the black pixels must both be found with the template in a particular location for the component to be detected in that location.

D. Generating a netlist

Identify all of the electrical connections between components on the board, a netlist is required. Once nets and components are found and component pad locations are determined, a netlist can be generated. For each pad on each component, the net containing the pad's center point is found and the component and pad are added to a list

E. Gerber files

Gerber data is a simple, generic means of transferring PCB information to a wide variety of devices that convert the electronic PCB data to artwork produced by photoplotter. Gerber data contains aperture information, which defines the shapes and sizes of lines, holes and other features. However, in this research PCB is fabricated and drills manually, thus the amount of PCB that can be produces is small. For mass production in future, gerber data is the solution.

III. RESULT

The program was tested with two layer board. Although this research was successful in achieving the objective, which is producing a new framework for PCB reverse engineering using of java.

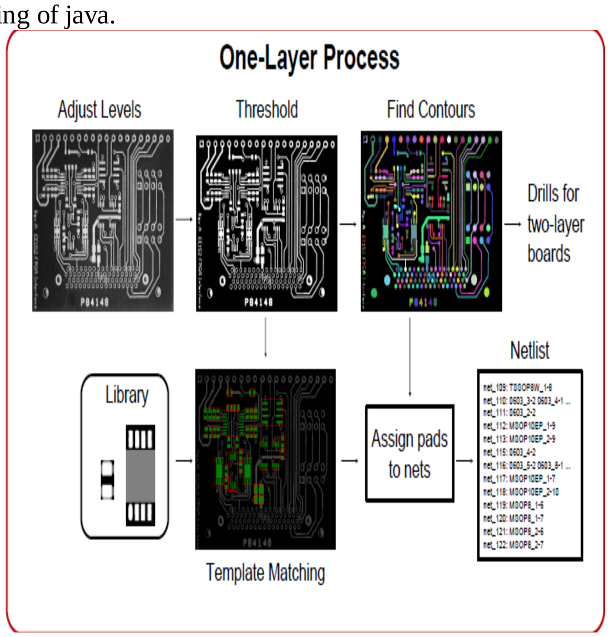

Fig: 5 one layering process

Fig: 6 selected input

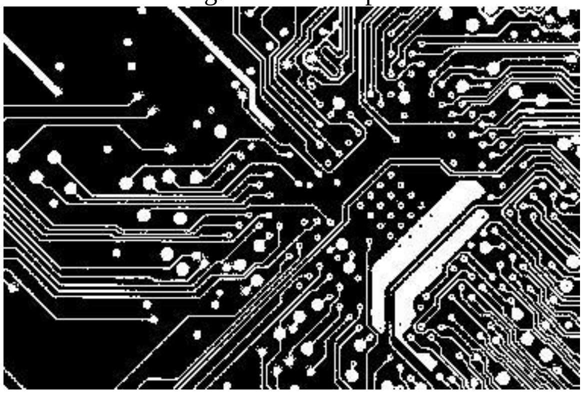

FIG: 7 reference input

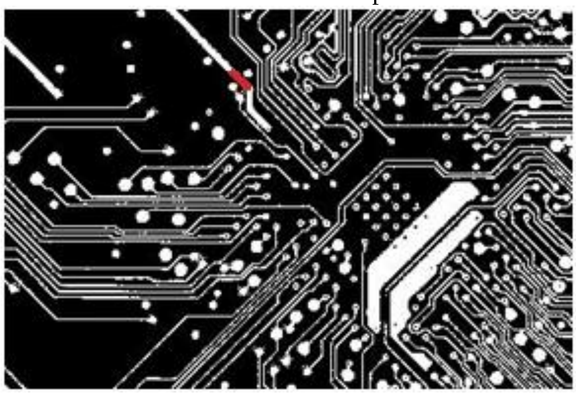

FIG: 8 error detected

IV. FUTURE WORK

Like open cv there is a best alternative open source tools for development of image processing and computer vision algorithms such as VXL, BOOF CV. The program would benefit greatly from user interface. There are a few parameters such as minimum areas which should be easily modified by the user. CEDA PCB reverse engineering services can take an existing pcb and create a complete documentation package required for manufacturing. The documentation package includes fabrication drawings ,bill of materials ,gerber files, cad design and drill data. Some open source image processing softwares such as GIMP ,INKSCAPE and dia,all of which are widely available for reverse engineering PCB's.

REFERENCES

[1] Ruzinoor Che Mat, Shahrul Azmi, Ruslizam Daud Abdul Nasir Zulkifli & Farzana Kabir Ahmad – "Reverse Engineering for Obsolete Single Layer Printed Circuit Board (PCB)" 1-4244-0220-4/$20.00©2006 IEEE.

[2] K. Tombre and S. Tabbone, "Vectorization in Graphics Recognition: To Thin or not to Thin". IEEE 2 (2000) 91-96.

[3] Malay K. Pakhira – "A Modified k-means Algorithm to Avoid Empty Clusters" International Journal of Recent Trends in Engineering, Vol 1, No. 1, May 2009

[4] otsu, nobuyuku. "A threshold selection Method from Gray-Level histograms." Systems,Man and cybernetics, IEEE Traanctions on vol.9, no.1,pp.62,66,jan. 1979

[5] L. Wenyin and D. Dori, "A Fast Vectorization Algorithm Applied to Engineering Drawings" in Proceedings of the 13th International Conference on Pattern Recognition Volume III (Robotics and Application). Vienna, Austria (1996).

[6] LSP Corporation 2005, Reverse Engineering Solutions. Retrieved 25 November 2005, from http://www.lspcorp.com/pcb002.html

