Printed Circuit Board Prototyping Guide

Shiva

03/12/08

This document has referenced and extracted certain contents *in verbatim from "Capture CIS Tutorial, Ekarat Laohavalesson", "Layout Plus Tutorial, Ekarat Laohavalesson", and "PCB Creation Guide, Whitney O'Meara"*

Contents

List of Figures

1 Introduction

This document provides step by step instruction to design and build PCBs. Prototyping PCB involves:

- Capturing schematic in Cadence Capture CIS tool
- Developing layout from the schematic netlist with Cadence Layout Plus tool
- Generate film from the physical layout files
- Fabricate the board
- Drill and solder the fabricated board

Bottom Line: This document focuses on "How" to build a board rather than "Why's" and "What's"

2 Schematic Capture

Capture CIS is an EDA (Electronic Design Automation) tool. It is used to create a schematic design for PCB. This process basically involves

- Place components
- Connect components using wires
- Update each component's property with Footprint
- Run DRC for errors
- Generate netlist for Layout Plus

Note

- Refer to "Capture CIS Tutorial, Ekarat Laohavaless" to get an overview of developing schematic.
- It is easier to specify a footprint associated with a particular part before import the netlist into PCB tool. These footprints tell the PCB tool about the dimension, number of pins and package type of the part (ex. PDIP-20 is a Plastic Dual In-line Package with 20 pins) that are critical during place and route processes of PCB design
- The footprint library is needed for specify a footprint for the part. Existing footprints on the local library can be used for some package types. However, if the right footprint for the part is not found, a new ones needs to be made by looking at the package drawing of the part and developing a new footprint using Layout Plus Library Manager.

Once the schematic is completed, the next step is to import the schematic into Layout Plus to develop the layout.

3 PCB Layout Development

Layout Plus is a circuit board layout tool that accepts a layout-compatible circuit netlist from Capture CIS and generates an output layout files that suitable for PCB fabrication. Generally, developing a layout involves:

- Import netlist
- Add Board Outline
- Update layer information
- Update Nets for track width and routing on the layer
- Place the components
- Typically, auto placement is sufficient. For pre placement, place and lock the components at their position
- Route for Ground in the bottom layer. This routing will generate Vias and Pad Stacks
- Disable routing for Ground pins.
- Auto route in the top layer for the remaining components
- DRC for errors
- Generate Gerber file.

Note

Refer to "Layout Plus Tutorial, Ekarat Laohavaless" to get an overview of the Layout Plus tool in developing the PCB layouts

4 Layout Film

Films are generated by printing the Layout files on the transparency. These files are required in exposing the PCB pre-sensitised boards There are two parts to this step:

- Import layout information file to print the physical layout
- Setup the doc print and the printer options

4.1 Printer settings

Choose,
Printer->Menu->Print Quality Menu
Printer->Item->Resolution
Printer->Value->Prores1200
Select
Set a value of 1200 Professional DPI. This high resolution is necessary to capture thin tracks and pads on the board.

4.2 Importing Layouts

Pentalogix's Viewmate tool is used in printing the layout on the transparency. Viewmate imports *.BOT and *.TOP files to generate the printouts. These files are generated along with Gerber files by the layout plus. Steps involved in printing are:

- Select *Program->Pentalogix->Viewmate*
- Select *File->New. See Figure 1*
- Import the layout files. *See Figure 2. 3*
- *File->Print*
- Select *Setup*
  - Select *Advanced*
  - Update *Print Quality = 1200 DPI*

5 PCB Fabrication

PCB development is conducted in stages. Steps involved are:

- Expose the Presensitized Copper Clad boards using film and ultra violet light
- Develop the exposed board
- Etch the developed board to remove copper from the board except for the tracks‘
- Tin the Etched boards for protection and soldering

Stage	Materials	Instruction	Comments

Setup	• Transparency Film • Presentisized Copper Clad Board with positive photo resist • Gloves • 4 Trays • Chemicals • Waste disposal containers • Paper towels	• It is important to wear gloves from the beginning • Select the board with right dimension and thickness • Check for the availability of the chemicals • Allot the trays as follows: – Developer – Water – Etchant – Tinning • Add 1 part developer to 10 part cold water in the tray • Pour the etchant solution in the tray • Tinning solution should *NOT* be placed in the tray	
Expose	• Transparency Film - Layout printout • Presentisized Copper Clad Board with positive photo resist • UV Exposer	• Peel protective cover on the board • Place the film and the board in the exposer • Exposure time – 1/16' - 80 Seconds – 1/32' - 100 Seconds – 1/64' - 120 Seconds	• UV Exposer has facility for 2-layer boards • If 2-layer boards are fabricated, it will be better to make pouch with film to place the boards

Develop	Developer solution	• Add 1 part developer to 10 part cold water in the tray. • Typically, couple of mins are sufficient • It is time to stop when the Green part of the board dissolves and layout outline is clear • Place the board in the water tray and rinse • After the removing the remaining etchant, dry the board with the paper towel	When the board is over exposed, even the tracks will dissolve. Developer can be disposed in the drain
Etch	Ferric Chloride Solution	• Pour solution in the tray • Submerge the board in the solution and agitate • Etching time varies from 20 - 40 mins depending on the board size. • Etching is complete, when the board turns into light green and the copper remains only on the tracks • Clean the Etched board with paper towels	• Solution can be reused until it turns into dark black color • Etching time depends on the board dimension and the amount of copper to be removed. E.g. It will take around 30 mins to etch 6x6 board with widely spaced tracks • If the solution comes in contact with the skin, it usually leaves a yellow stain and may cause irritation. • If the etchant turns into dark black color, add more solution • Etchant solution should be disposed in special disposal containers.

Tinning	• Acetone (Nail polish remover) • Liquid Tin (Friendly suggestion, hold the breadth for a minute)	• Gently pour small quantity of acetone on the board to remove the photoresist material on the tracks. Clear golden color tracks can be seen. • Clean the board with tinning solution. Tinning solution will turn the tracks to silvery white color	Etched boards can be cleaned with paper towels soaked in tinning solution, but it will be better to pour the solution and then gently rub on the tracks.

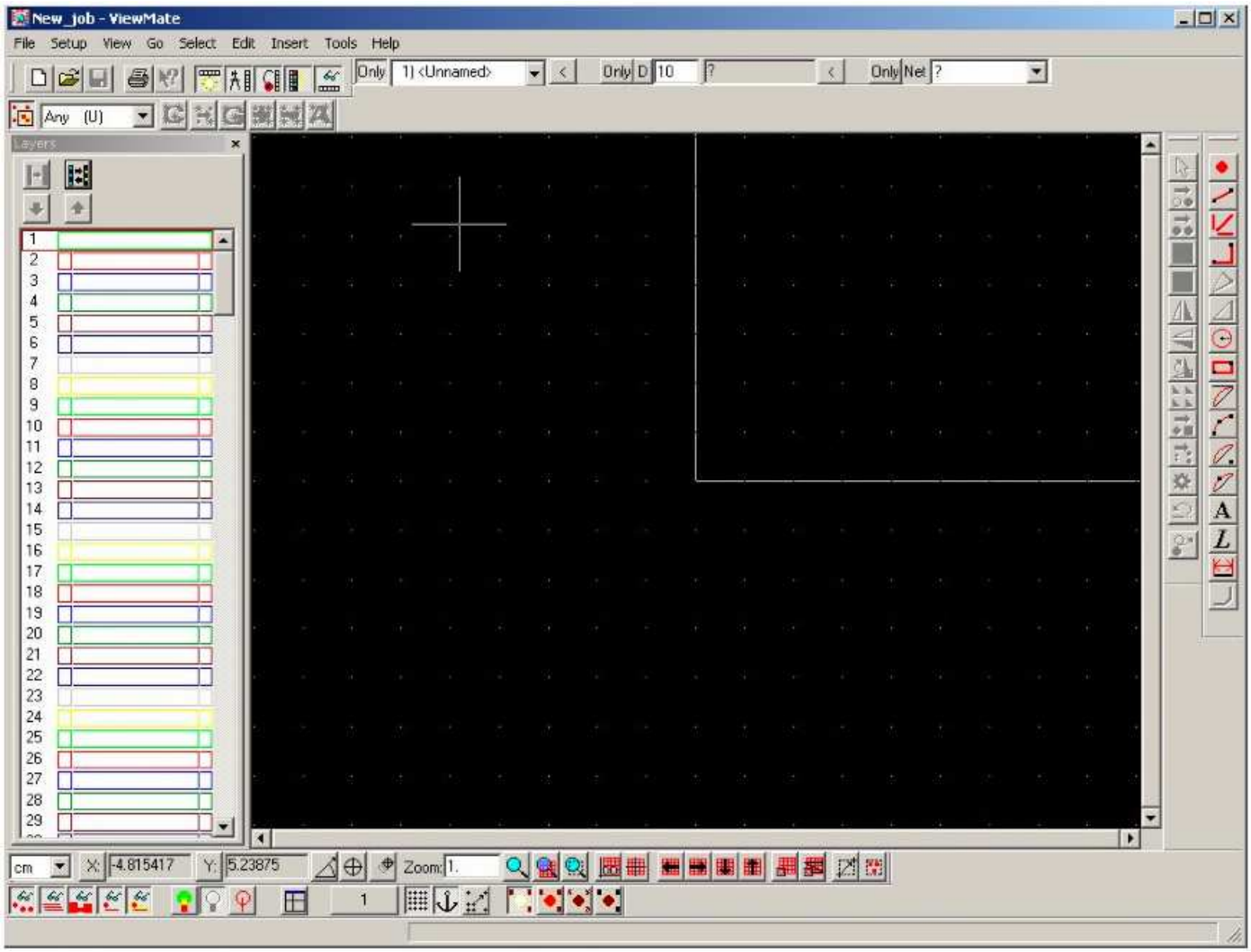

Figure 1: Viewmate New Window

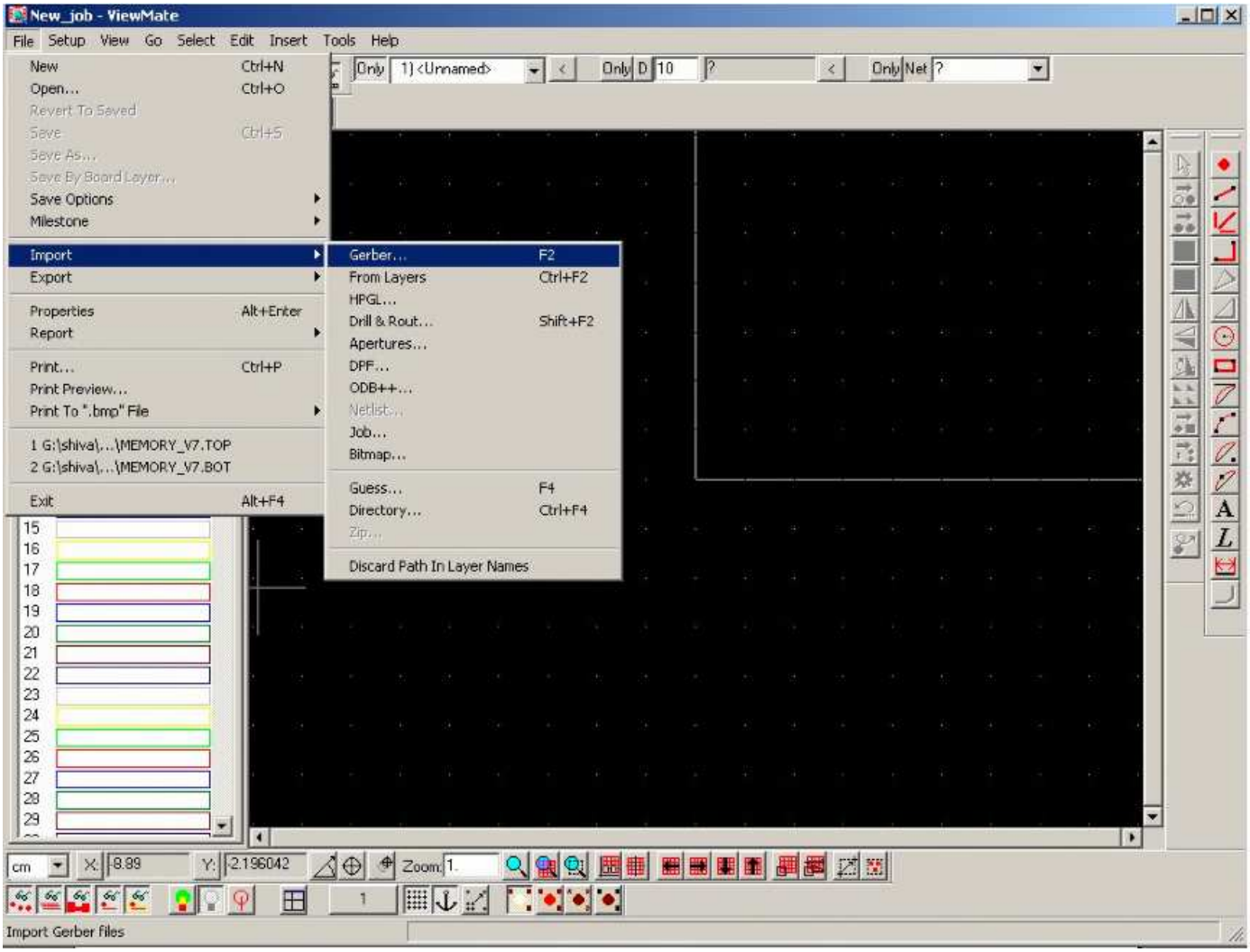

Figure 2: Importing Layout in Viewmate

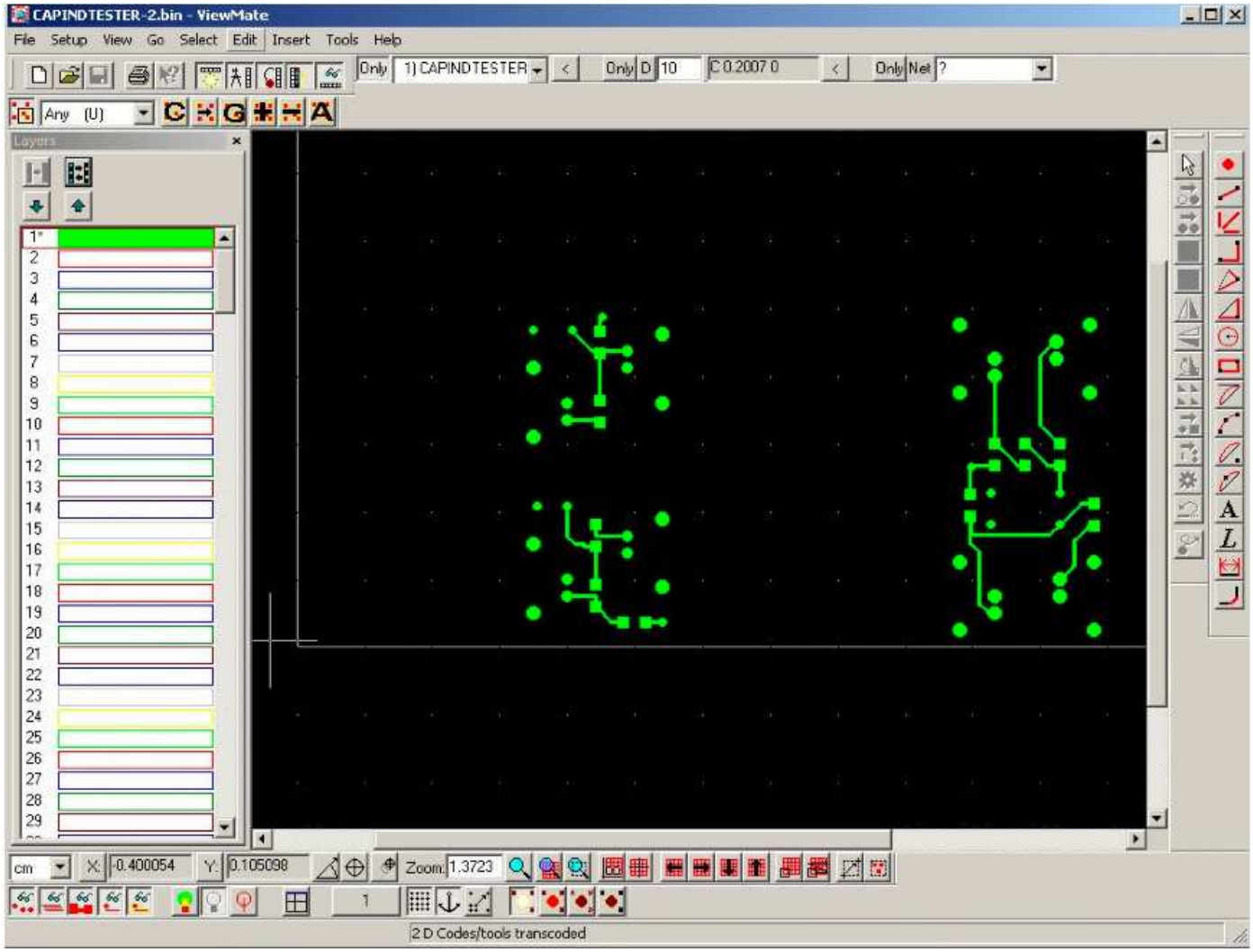

Figure 3: After importing in Viewmate