Reverse Engineering: Printed Circuit Boards

Dr. Tarek A. Tutunji
Philadelphia University, Jordan
Reference

- PCB Reverse Engineering by John Armistead
- A Rapid Prototyping Methodology for Reverse Engineering of Legacy Electronic Systems by Deno, Landis, Hulina, and Balasubramanian
The Need for PCB RE

- Replace an obsolete PCB that is no longer available from the original manufacturer.

- Old PCBs are sometimes difficult to support:
  - Increasing costs of spare parts
  - Non-existent procurement sources.
The Need for PCB RE

- The technical documentation needed to re-manufacture replacement parts is often insufficient or non-existent.

- Missing manufacturing files such as Gerber files are often cited as the main reason.
The Need for PCB RE

- Re-engineering a PCB assembly can provide an improved or added performance to an old process.

- New and improved materials and techniques may be utilized improving operations, maintenance and support.

- Reverse engineer your PCB first as a base set of data and then modify that base to obtain a more modern PCB that is both less expensive to build and less expensive to service.
The Need for PCB RE

• Rapid prototyping and reverse engineering procedures offer a cost-effective method to replicate the missing part(s)
Needed Documentation

- PCB layer count and physical board dimensions.
- Padstack definitions and coordinate locations of the pads on the board.
- Component definitions and functional component specifications.
- Netlist information defining inter-component connections.
PCB RE Steps

1. Prescreening
2. Observation
3. Disassembly
4. Identify Connections
5. Capture the schematic in software
6. Generate Gerber files for the board
1. Prescreening

- The selection of potential candidates for the reverse engineering cycle depends on economics (return on investment), technical complexity and the amount of existing data.

- Components that have high usage and high per-unit cost but relatively low technical complexity are good candidates for the reverse engineering process.
1. Prescreening

- Drawings and technical manuals that provide information about the PCB assembly are collected.

- Usage and maintenance data are also obtained along with performance specifications.
2. Observation

- Obtain at least two samples of the board
- Evaluate existing data
- The physical unit is visually inspected and discrepancies between the available data and the actual PCB are noted.
- Identify all components and obtain specifications for each.
2. Observation

- Formulate a management plan including minimum requirements of an acceptable Technical Data Package (TDP).

- The objective of this stage is to complete the TDP, which will be sufficient for fabrication and procurement of the PCB assembly.
3. Disassemble

- The PCB is disassembled.

- A list characterizing each piece and the order of disassembly is maintained.

- Component availability is verified and exceptions are researched for substitutes.
3. Disassemble

- This process will render one sample board unusable again. One board should be retained intact and utilized as a reference board. That board should remain usable again.

- The identification of all components is required for the bill of materials (BOM).
4. Identify Connections

- Identify all of the electrical connections between components on the board
  - Node list, sometimes called a net list

- This process requires an experienced electrical engineer to be able to render the emerging node list into a readable schematic.

- Accuracy here is most important.
3. Capture the Schematic

- Build the schematic

- Capture the schematic in software including building component images for all components that are not in a parts library.

- Check the schematic connections to the board.
4. Generate Gerber Files

- The **Gerber format** is a file format used by PCB industry software to describe the images of a printed circuit board (copper layers, solder mask, legend, etc.) as well as the drilling and milling data.

- The Gerber format is the de-facto industry standard for printed circuit board image transfer.
Example Gerber layers, showing the top overlay (legend), top solder resist (protective film), top layer copper traces, and bottom layer copper traces of a printed circuit board.

%AMMACRO19*
21,1,.0512,.0512,0.0,0.0,0.45.*%
%ADD19MACRO19*%
%LPD*%
G75*
G54D10*
X176250Y117500D03*
Y130000D03*
Y163750D03*
...

...
Re-engineering: Design Verification

• Prior to prototype fabrication and testing, the technical data package is reviewed for accuracy and completeness.

• Prototypes are built and tested.

• Deficiencies in the results from prototype testing follow traditional debugging procedures and lead to final modifications and improvements in the TDP.
Patent / Copyright Issues

- If a board has a copyright mark, then you should not reverse engineer the traces verbatim. Rerouting the board using any one of the auto router software packages will generally render a board that is unlike the existing board although still electrically and schematically identical.

- The majority of the patented boards are patented for onboard firmware or highly specialized digital circuits. Unless you have very deep pockets, this is one PCB reverse engineering area that it is advisable to stay away from altogether. Of course, if your company owns the patent then there is no problem.
PCB RE Results

- Complete schematic diagrams. Includes any on board, point to point, wiring diagrams.
- Complete bill of materials including individual data sheets on each component.
- Complete Gerber files for the production of the PCB.
- A prototype PCB assembled with components for testing and evaluation.
Conclusions

- Reverse engineering PCB's is a necessary process to obtain lost manufacturing files.

- Sometimes, reverse engineering combined with re-engineering can revitalize old circuits to save time and money.

- Reverse engineering, redesign or re-engineering your PCB assemblies can be done quickly and economically. Original drawings can be provided with a pre-production prototype for testing and evaluation.