

Printed Circuit Board Assembly Analysis

Introduction

The first step in the manufacture of the printed circuit board assembly is the fabrication of the bare circuit board, often called a printed wiring board. The bare boards can have circuits on one or both sides and in many cases are laminated from several layers with additional circuits in the internal layers. The circuits are generally produced by photo-chemical processing. Additional steps are needed for the machining of holes through the board, solder masking, legend printing and final PWB testing.

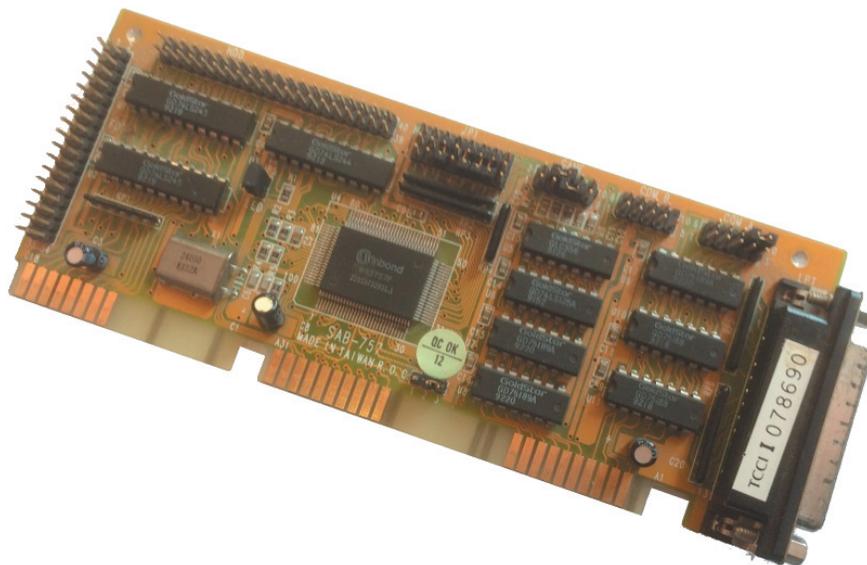
The PWB is then populated with electronic components using a combination of automatic and manual operations. The bare board is first partially populated using equipment for automatic placement or insertion of electronic components. Items that cannot be automatically inserted are then added manually at a workbench or at a semi-automatic machine, which presents items to the operator in the correct sequence for manual insertion. The boards are then soldered using either wave and/or reflow soldering methods. Following this, hand insertion and hand soldering may occur for some items, together with the addition of mechanical parts, such as screws, standoffs, and heat sinks.

The assembly of printed circuit boards can be analyzed using operations in the Operation Library, together with the similar operations for any mechanical items. The individual electronic components are not identified, only the number of items with the same package style and numbers of leads. The Operation Library contains time or cost equations for the assembly of various electronic packages by different assembly methods (automatic, robot, semi-automatic, and manual). Various secondary operations specific to printed circuit boards are also included in the library.

Analysis of a printed circuit board assembly is based on a list of operations required for its assembly. The figure on the next page shows the upper side of a small circuit board populated with through-hole electronic components. The electronic components on the upper side are inserted first, by either automatic or manual methods. This is followed by wave soldering and cleaning of the board. One connector is then inserted on the lower side of the board and for this reason must be hand inserted and soldered. Some mechanical parts are then added manually. Finally, in this design, the dual-in line-packages (DIPs) are manually inserted into sockets.

Electronic packages of the same type, with the same number of leads, and inserted using the same method, are grouped together. The table on the next page lists the assembly sequence for the sample board with the packages grouped in this way. You will need to refer to this table during the exercise that follows.

The subject of this tutorial is the small video/game card from a PC shown below. This is a double-side mixed surface mount/through hole board. The overall dimensions of the board are length 6.25 inches and width 2.5 inches. The board substrate thickness is 0.062 inches.



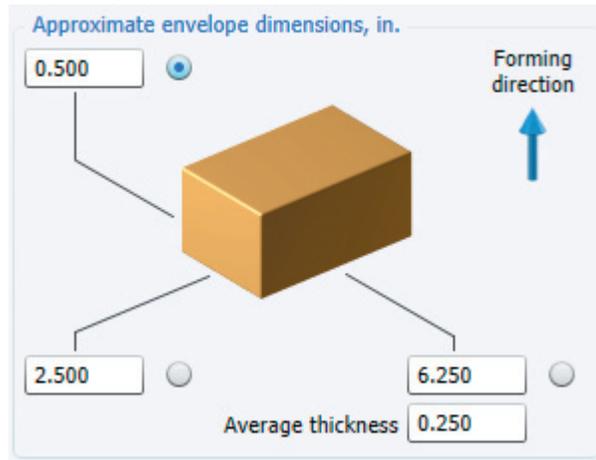
In addition to the electronic components there are 9 jumper caps to be added (snap fit) to the connector pins in various places on the board.

The board substrate material is the standard FR-4 used for the majority of boards produced. The bare board has solder masking applied to both sides, legend printing on one side of the board and has gold plated contacts along one edge of the board. The minimum conductor spacing is 0.006 inches and the minimum trace (conductor) width is 0.015 inches. There are 502 holes through the board.

Package Style	Number of leads, pins or pads	Quantity	Number of different components	Average cost
Through Hole				
Dual in-line package (DIP)	20	3	2	0.10
Dual in-line package (DIP)	14	7	4	0.10
Single in-line package (SIP)	8	4	3	0.05
Single in-line package (SIP)	6	1	1	0.05
Single in-line package (SIP)	5	1	1	0.05
Radial	2	3	1	0.05
Can	2	1	1	0.10
Connector	10	3	1	0.05
Connector	3	1	1	0.05
Connector	2	1	1	0.05
Connector	24	1	1	0.10
Connector	40	1	1	0.15
Connector	34	1	1	0.10
Connector	25	1	1	0.10
Surface Mount				
Chip	2	25	7	0.01
Quad Flat Pack	100	1	1	2.00

Analysis steps

1. Start the analysis by entering **PCB Sample** for the *Part name*. Use the default *Life volume* of 100,000. Select the solid block *Envelope shape* and enter the dimensions as shown below.



2. Now click the **Select process and material** button. In the left column of processes, choose *Printed circuit board assembly*. For material, choose *FR-4* in the Printed wiring board material category. Click the **OK** button to calculate some preliminary results.
3. Highlight the *Bare board process* entry on the Process Chart. We will replace some of the default information used in the original estimate to improve the accuracy of the results. Accept the default data for the *Basic data* group. In the *Board processing data* group, change the *Number of holes* to **502** and the *Minimum trace width* to **0.015** inches. Check the boxes for *Gold plated edge connectors*, *Solder mask required* and *Legend screen printing*. All other defaults can be used. Click the **Calculate** button and entries for *Solder mask* and *Legend printing* will be added to the Process Chart.

The screenshot displays the software interface for configuring PCB processes. On the left, a tree view shows the hierarchy: FR-4 printed circuit board > Bare board process. The 'Bare board process' is expanded, showing sub-items: Printed wiring board fabrication, Hole drilling, Solder mask, Legend printing, Testing, and Pcb Assembly. The 'Pcb Assembly' item is highlighted. On the right, configuration panels include 'Basic data' (Batch size: 12500, Overall plant efficiency, %: 85.00, Basic material cost, \$/in²: 0.0400), 'Board processing data' (Board type: Double sided, Board thickness, in.: 0.062, Number of holes: 502, Copper weight, oz/sq ft: 0.500, Minimum trace width, in.: 0.015, Minimum conductor spacing, in.: 0.006, Gold plated edge connectors: checked, Solder mask required: checked, Legend screen printing: checked, Circuit overlays, \$: 930), and 'Panel data' (Length of panel, in.: 24.000, Width of panel, in.: 18.000, Number of boards per panel: 18). At the bottom left, a table shows cost results.

Cost results, \$	Previous	Current
Material	0.9600	1.0560
Setup	0.0066	0.0127
Process	7.5857	9.7456
Rejects	0.5062	0.8182
Piece part	9.0584	11.6324
Tooling	0.0167	0.0251
Total	9.0751	11.6575
Tooling investment	1,665	2,505

4. We now select the operations for adding (insertion or placement) the electronic components to the PCB. A minimum of one operation for each package style and number or leads, pins or pads is required. To make it easier, an average cost for the components covered by each operation is used. These costs are included in the overall material costs for the analysis. If an exact component cost total is critical, then instead it is necessary to use an operation from the library for each individual component added, so that specific component costs can be entered. For this analysis, use the average component costs given in the previous table.
5. Highlight the *Pcb Assembly* entry on the Process Chart. Choose *Operation* from the *Insert* menu to open the *Insert Operation* dialog.
6. In the *Printed circuit board assembly* category, open the *Through hole insertion* subcategory. Highlight *DIP insertion* and click the **Insert** button twice to add two of these operations to the Process Chart. In the same way, add the indicated number of the following operations from the *Through hole insertion* subcategory:
 - SIP insertion* (3)
 - Radial insertion* (1)
 - Can insertion* (1)
 - PTH connector insertion* (7)
7. Expand the *Surface mount placement* subcategory and add one *Chip placement* and one *Quad flat pack placement* operation. Close the *Insert Operation* dialog.

Now that the operations have been added to the Process Chart, we will highlight each one and adjust the default values according to the information on the parts list table.

8. Select the first DIP insertion operation. Change the *Number of pins* to **20** and the *Number of components inserted* to **3**. After the 3 has been entered a new response for the *Number of component types* of this package style will appear. The default of 1 should be changed to **2**. Enter **.10** for the *Average component cost*.

Operation inputs	
Insertion method	Auto
Fault rate	0.002
Number of pins	20
Number of components inserted	3.000
Number of component types	2.000
Average component cost, \$	0.100
Setup cost per part type, \$	1.600
Component rework time, s	88.900
<input type="checkbox"/> Replacement component required?	

Highlight each additional PCB assembly operation on the Process Chart and adjust the inputs according to the parts list table.

9. Highlight *Can insertion* on the Process Chart and select **Manual** for the *Insertion method*. Since the can is bonded over for attachment to the outside of the board by hand soldering we will add a hand soldering operation. Open the Library Operations dialog and add the *Hand solder* operation from the *Miscellaneous operations* subcategory.
10. Highlight the *Quad flat pack placement* on the Process Chart and select **Robot** for the placement method. Also check *Replacement component required?* to indicate that during any rework a replacement component will be required.
11. The *Whole board operations* are now added from the operations library. With *Quad flat pack placement* still selected on the Process Chart, add the following operations:

Solder paste-manual
Reflow solder
Wave Solder
Cleaning
Functional testing
In-circuit testing

The default responses can be used for all operations in this case.

12. The final operation to be added is for the addition of the 9 jumper caps to some of the connectors on the board. These should be added before *Functional testing* in the sequence of processes, although the position in the process chart will not affect the results.

Highlight *Cleaning* on the Process Chart, then select *Part* from the *Insert* menu.

13. In the responses enter **jumper cap** for the *Part name*. Change the *Repeat count* to **9** and *Material cost per part* to **0.02**.
14. With the *jumper cap* highlighted on the Process Chart, select *Operation* from the *Insert* menu. Open the *Assembly* category and add *Acquire and insert parts* to the Process Chart. Close the Insert Operation dialog.
15. On the Responses Panel for the operation, accept the default of *Non rotational part*. Enter dimensions of *Part length* **0.4 inches**, *Part Width* **0.2** and *Part depth* **0.1**. Change the *Repeat count* to **9**. In the group *Symmetry 180 or less about* select *One axis*. For the *Securing process* select *Snap/push fitting*. Accept all other defaults.

You have now completed the PCB analysis. You can compare your results with the completed sample file *video board.dfm* included with your software in the `\data\samples` subdirectory.