

Predictability for PCB Layout Density

Ruth Kastner and Eliahu Moshe,
Adcom Ltd., ISRAEL

Abstract

The trend towards increasingly complex designs with smaller physical sizes has been translated into ever-increasing pressure on system developers to pack more functions and options into a given area. In addition, cost needs to be driven down as much as possible. As a result, the design process has become more extensive in terms of resources, complexity, choice of PCB technology and cost reduction. In order to handle this challenge effectively, one would like to predict the efforts involved in the layout design a-priori. This capability is now available in the form of "Predictability Calculator", described below.

The Predictability Calculator is a tool that provides the designer with the necessary trade-off analysis performed at the feasibility stage, given the constraints of the assigned area. It takes advantage of the fact that all designs are done today using CAD systems, hence data analysis is possible given the electrical schematics that is available at an early stage of the PCB layout design. This initial data include the number of components and their type and characteristics that are known once they have been selected. The number of connections is also available based on the interconnections and busses.

Although it is recognized that the PCB technology may be selected independently of the designer, it is nonetheless a part of the tradeoffs supplied by the Predictability Calculator with the objective of providing maximum performance at a minimal cost.

This tool has been utilized so far in the feasibility stage of over 40 complex boards, and also in post mortem analysis of other boards. It has thus demonstrated a proven capability of providing feasibility data for the routing complexity with a high level of confidence. The next step in the tool development will include an in-depth placement feasibility and the trade offs with embedded resistors and capacitors.

I. Introduction

Proper PCB design, being an integral part of the Electronic-Mechanical design of many high – speed, analog PCBs, can make or break the operation of the electrical performance of the design. Complex physical and electrical designs, densely packed boards and faster signal requirements are examples of factors that add complexity to today's PCB designs. Consequently, designers should be able to easily define, manage, evaluate and validate physical/spacing constraints that apply to critical high-speed signals. This should be done at an early stage of the design process. At the same time, the designer must ensure that the final PCB layout design meets performance, manufacturing, and test specifications goals.

The capability to perform this analysis at an early stage of the PCB design is now available in the form of Adcom's "Predictability Calculator". This is a tool that provides the designers with the possible trades offs for determining whether a given PCB design layout is feasible, given the constraints mentioned above, within the assigned area and net list. To make effective use of this tool, several parameters should be made known prior to this feasibility analysis run. These include the following:

1. Understanding of the measurement units, that can be imperial or metric, and their impact on the trace/space width at the routing phase. Also important at this stage is the understanding of the metric structure of the current BGA and MicroBGA components relative to the via holes that go through those components.
2. The trace width will be dictated by the level of the current flow and the maximal tolerable temperature rise.
3. Component placement strategy has an impact on the electrical performance. Therefore, the designer should keep in mind the board electrical flow during the entire design phase.
4. In high-speed, high-frequency design, with "controlled impedance" analysis of the trace is required, where the trace is now considered a transmission line.
5. Finally, the board technology and stack-up structure, i.e., the PCB thickness, the number of signal and non-signal layers, need to be taken into account, as is the HDI technology that

would be required for the completion of the PCB layout routing, given all the above-mentioned constraints.

II. The Predictability Calculator Algorithm

II.a Density Definition

A board density can be measured in several methods. One definition is based on: the number connections per sq. Inch. For this definition, any number between 65 to 120 connections per sq inch is considered dense today. Another definition includes the number of components per sq. Inch and yet another one uses the pad count per sq. Inch.

Take, for example two board layouts for the same product, designed in around 1990 and 2000, respectively. The 2000 board has more functions compared with the one from 1990; however it is denser in terms of aspects as size, less numbers of layers, components and connections, and assembly density. These two boards are shown in Fig. 1.

(a) Board from 1990

Size: 11.75"x 8.75"

Thickness: 0.092"

Layers: 18

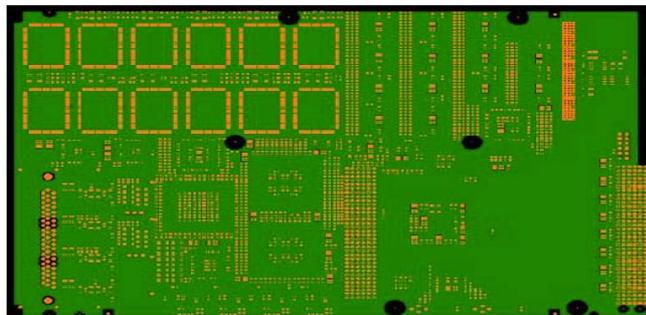
No. of components: 1410

No. of connection: 10530

Assembly density:

102 leads/in²

Design Rules: 5/5, 13/25



(b) Board from 2000

Size: 9.2"x 6.3"

Thickness: 0.072"

Layers: 10

No. of components: 1621

No. of connection: 12456

Assembly density:

216 leads/in²

Design Rules: 5/7, 6/12, 13/25

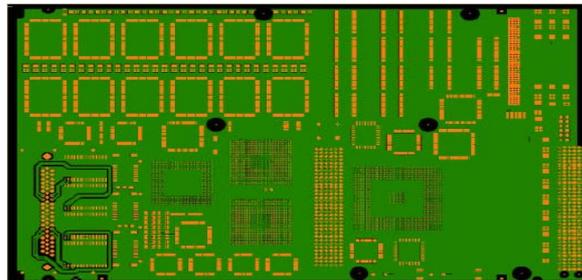


Figure 1. Two PCBs designed in (a) 1990 and (b) 2000.

For the same product, the newer board is denser while utilizing a smaller area and fewer layers.

The user can define a maximal density factor for a number of PCBs, say in a given product. Then all boards would have the same or lower density factor. Obviously this definition can change as the technology moves forward and new design rules are implemented.

II.b Inputs: Demand and Capacity

Demand

in order to gain confidence in the feasibility of the board design, we define the following terms, used as inputs:

- **Demand** - all available design data requirements
- **Capacity** - all available design resources.

Given these two values we run the program to get the density result defined as the ratio between the Demands to the Capacity.

In the **Demand** part we enter all known CAD design data (see Figure. 2). Usually this data is available once the electronic design has been completed. Mechanical design leads to the board size and its outline structure. After importing the mechanical data into the CAD design work desk, the available area will show, expressed either in terms of sq inches or sq mm.

The design technology is a key parameter in our prediction calculator. Many of the hi-speed designs have electrical signal constrains such as controlled impedance signals, differential signals, fast clocks and tuning requirements. This data is mostly known to the electrical designer.

Numbers of connection	11961	
Number of components	3275	
Available area	83.78	inch?
Design technology	Digital	
Differential busses +	<input type="checkbox"/>	
Tuning +	<input type="checkbox"/>	
Wiring Demand	68.5035	

Figure 2. Demand dialog box in the Predictability Calculator

The program calculates the Wiring Demand that appears at the bottom of the dialog box. The higher the number, the more complex is the board. As a rule of thumb, if the Wiring Demand number exceeds the value of 80-100, one should consider HDI.

In the **Capacity** dialog box (Figure. 3), we enter initial concepts such as BGA pitch, typical trace width, typical via hole, number of signal layer expected and type of design. There is always a limit on the amount of routing each board can accommodate. The main contributors are

- Pitch/distance between vias or holes in the substrate
- Number of wires that can be routed between the vias
- Number of signal layers required
- Design type
- Design versatility

Technology efficiency	Mixed both sides	
Pitch	1	(mm.)
Pad Diameter	0.4	(mm.)
Trace Width	0.1	(mm.)
Space Width	0.1	(mm.)
Number of signal layers	8	
Units	mm	
Wiring Capacity	162.5600	

Figure 3. Capacity dialog box in the Predictability Calculator

As a result of the calculation, the wiring capacity appears at the bottom of the dialog box. Again, the higher the number the more complex is the board.

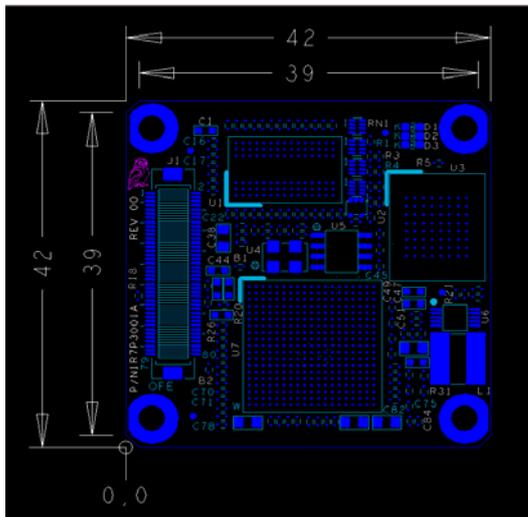
III. "What If" Analysis

If the **Capacity** value is higher than **Demand** value then the chosen design rules and technology are sufficient, and cost reduction may be an option.

If the **Capacity** value is approximately equal to the **Demand** value, the chosen design rules and technology are sufficient, however, either some effort may be required to finish the layout, or a compromise on the constraints will be required during the design, or future changes will be difficult to implement. Finally, if the **Demand** value is higher than **Capacity** value, the chosen design rules and technology are insufficient, and a set of check ups could take place. These can include the available layout area, the technology used for the PCB manufacturing and the number of layers.

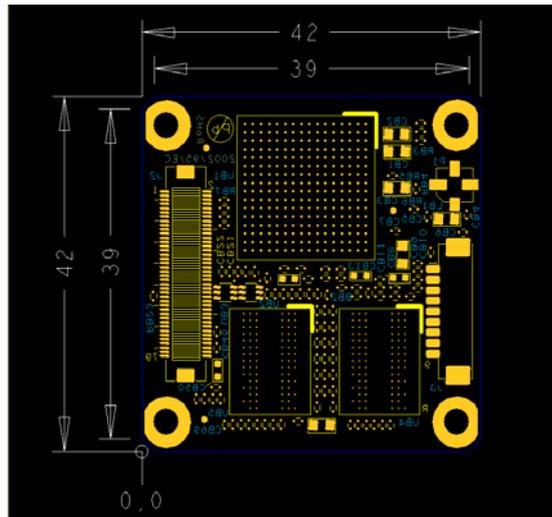
IV. Examples

Example A



Demand data:

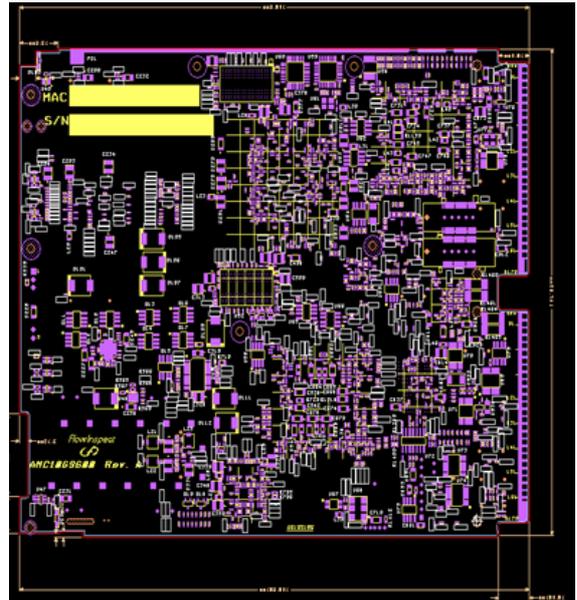
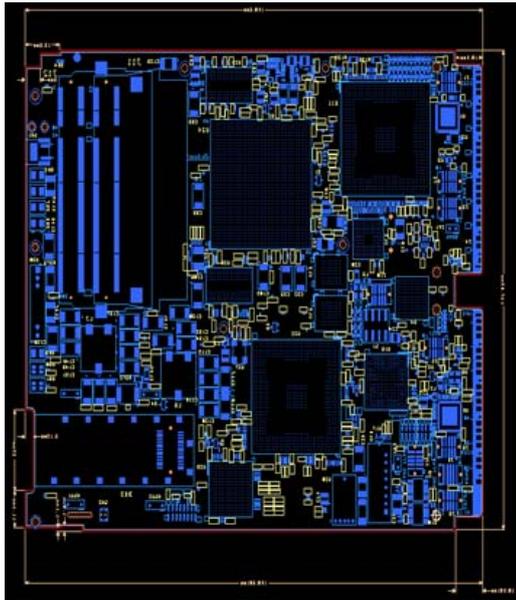
Number of connection 1174
 Number of components 262
 Available area 1422 mm²
 Design technology Hi-End
 Differential busses
 Tuning
Wiring Demand 158.78



Capacity Data:

technology efficiency 2 HDI structure
 pitch 0.8
 pad diameter 0.45
 trace width 0.1
 space width 0.1
 number of signal layers 8
Wiring Capacity 165.1

Example B



Demand Data:

*Number of connection 9060
 Number of components 2727
 Available area 26103mm²
 Design technology digital*

Wiring Demand,81.82

Capacity Data:

*technology efficiency mixed both sides
 pitch 1
 pad diameter 0.45
 trace width 0.1
 Space width 0.1
 number of signal layers 6
Wiring Capacity 121.92*

V. Summary

Since all the electronic designs are done today with CAD tools, smooth integration is available between the schematic and the layout. All of the schematic symbols, pins, connections and constrains are imported to the PCB layout tool in the form of the netlist.

The netlist file brings the schematic and the PCB database together. The data embedded therein includes, firstly, all the available design resources that fit into the definition of **Capacity** of the specific PCB layout. Secondly, all the requirements for that specific design are known as the **Demand**. The demand data includes, as a minimum,

- The number of components
- The number of connections
- The available area,
- The design type.

The Capacity data includes, as a minimum,

- The parts pin definitions (pads sizes, pitch in mm)
- The specific connection names (differential, controlled impedance, power, RF, analog)
- The PCB route directives (Stripline, Microstrip, trace/space width, via's)
- The PCB stack-up and technology (HDI, no of signal layers, no of plane layers)

The result of the ratio between the **Capacity** and the **Demand** provides us with a relative number the **Density**.

VI. Conclusion

If the **Capacity** value is higher than the **Demand** value, the designer can proceed with the layout with a high confidence that it will be successfully completed, without extra effort and future changes or updates may take place. Nevertheless, cost reduction may be considered already at this stage, by playing "what-if" scenarios with the tool.

If the **Capacity** value is approximately equal to the **Demand** value, the designer should review his strategy and design rules. If the board is a new design, changes and add-ons will be very difficult to be implemented in the next phase even if completed successfully at the first stage, or design factors and constrains would be compromised due to the density. The designer should change the parameters to achieve a more relaxed design, where future changes and update may also take place.

If the **Capacity** value is smaller than the **Demand** value, the designer needs to change the parameters or the design rules defined by a "what-if" simulation available by the tool, and optimize the parameters to ensure a PCB layout design which will be completed successfully with no constraining compromises, and without extra effort.

Organization that have a PCB data base information center can normalize their boards to a relative board density factor and keep the boards records by comparing the **Density Factor** data and achieve better electronic performance and PCB cost reduction.

VII. Future Plans

Adcom is working now on an additional PCB feasibility layout tool which will take into consideration more placement factors such as component placement restriction due to height limitations and heat dissipation constrains. This tool will be utilized at an early design stage....

Acknowledgement

This work was funded by Elbit System; ISRAEL