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/in^2
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/in^2
Design Rules: 5/7, 6/12, 13/25

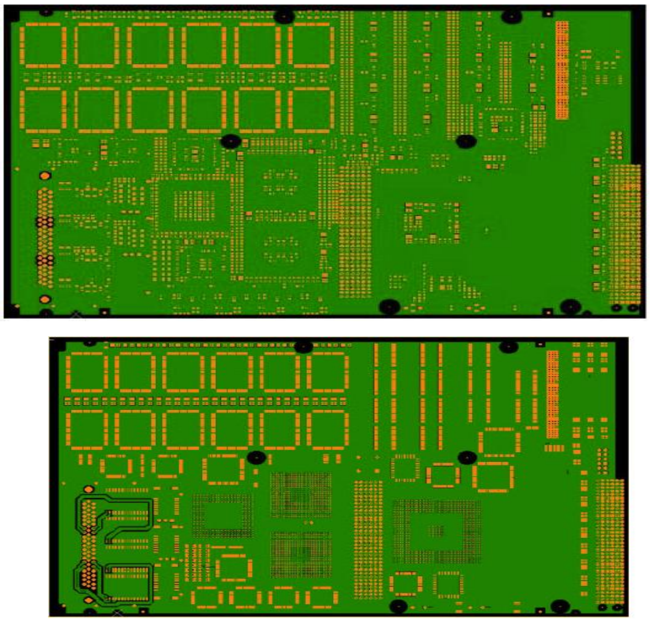


Figure 1. Two PCBs designed in (a) 1990 and (b) 2000.

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.

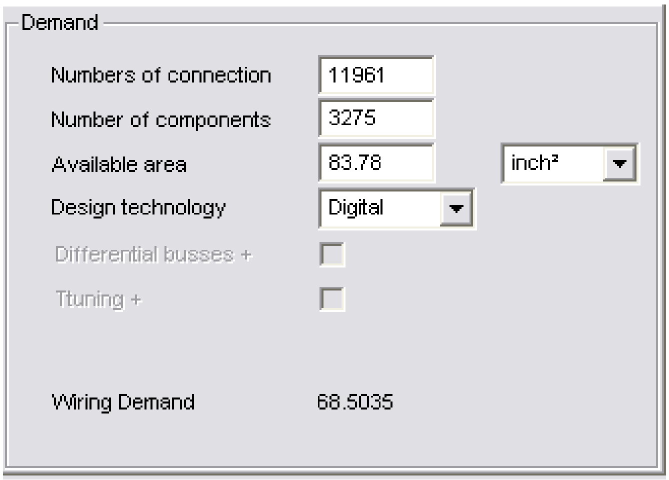


Figure 2. Demand dialog box in the Predictability Calculator

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

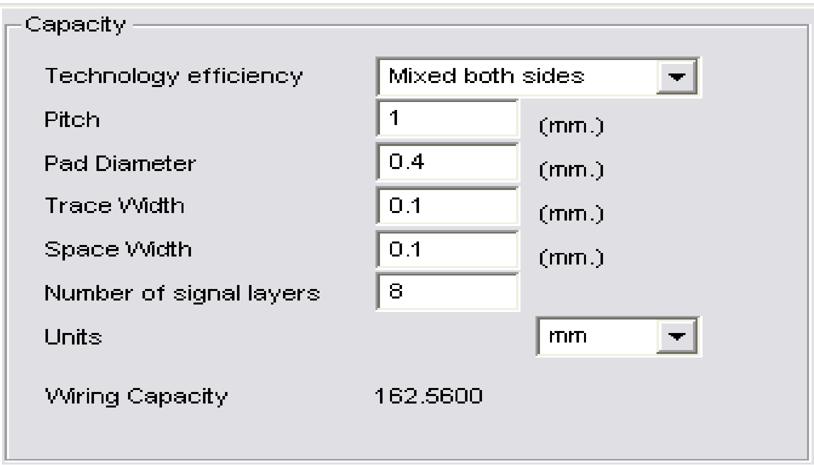


Figure 3. Capacity dialog box in the Predictability Calculator

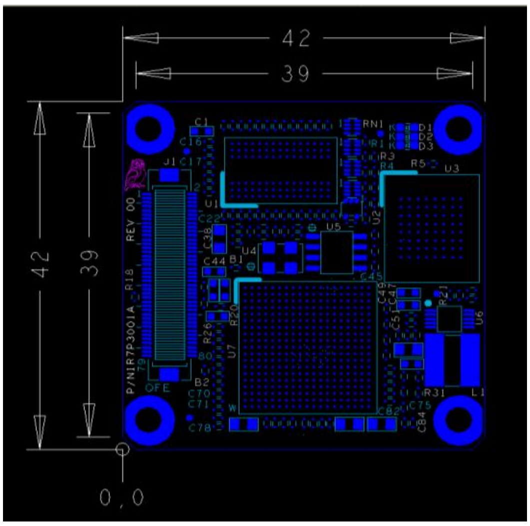
As a result of the calculation, the wiring capacity appears at the bottom of the dialog box. Again, the higher the number the more complex is the board.

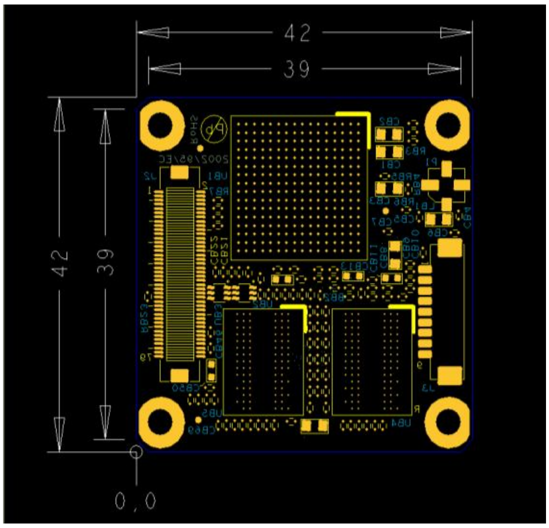
III. "What If" Analysis

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 constrains 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 sould take place. These can include the available laoyut area, the technology used for the PCB manufacturing and the number of layers.

IV. Examples

Example A

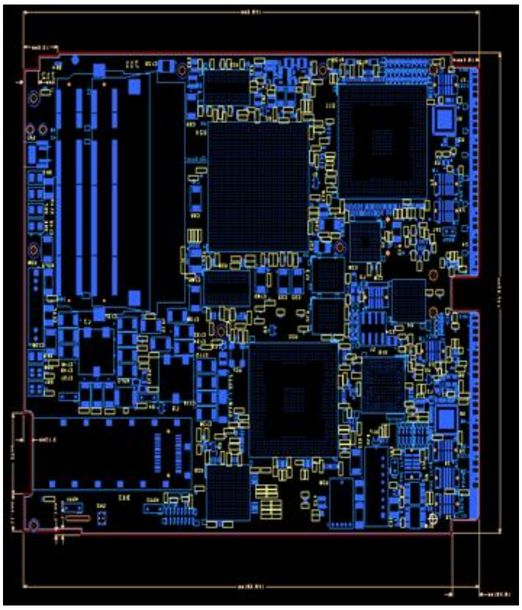


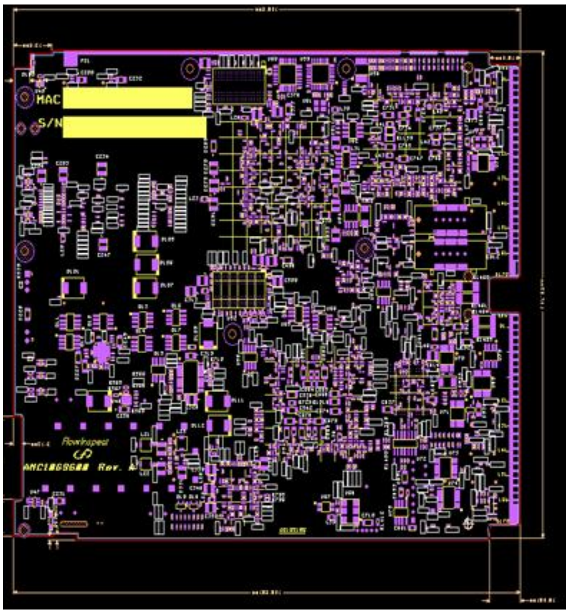


Demand data:
Number of connection 1174
Number of components 262
Available area 1422 mm²
Design technology Hi-End
Differential busses
Tuning
Wiring Demand 158.78

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

Predictability of PCB Layout Density

Ruth Kastner
Ruthk@adcom.co.il

Agenda

- Introduction
- Demands
- Capability
- “What If”
- Summary

Introduction

Facts:

- System design complexity is growing rapidly
- Development costs are increasing
- Estimations of effort, complexity, schedule and costs of new PCB layout designs are becoming increasingly difficult.

Introduction

Purpose:

Predictability of the PCB design effort by means of:

- Design Technology
- Contribution of the different component elements
- PCB Technology
- CAD tools capability

Introduction

- I will describe the methodology we have developed as a working tool for Elbit Systems.

- In order to gain confidence in the algorithm, we bench-marked 6 of Elbit Systems communication boards, 2 Opgal boards and 2 Adcom boards

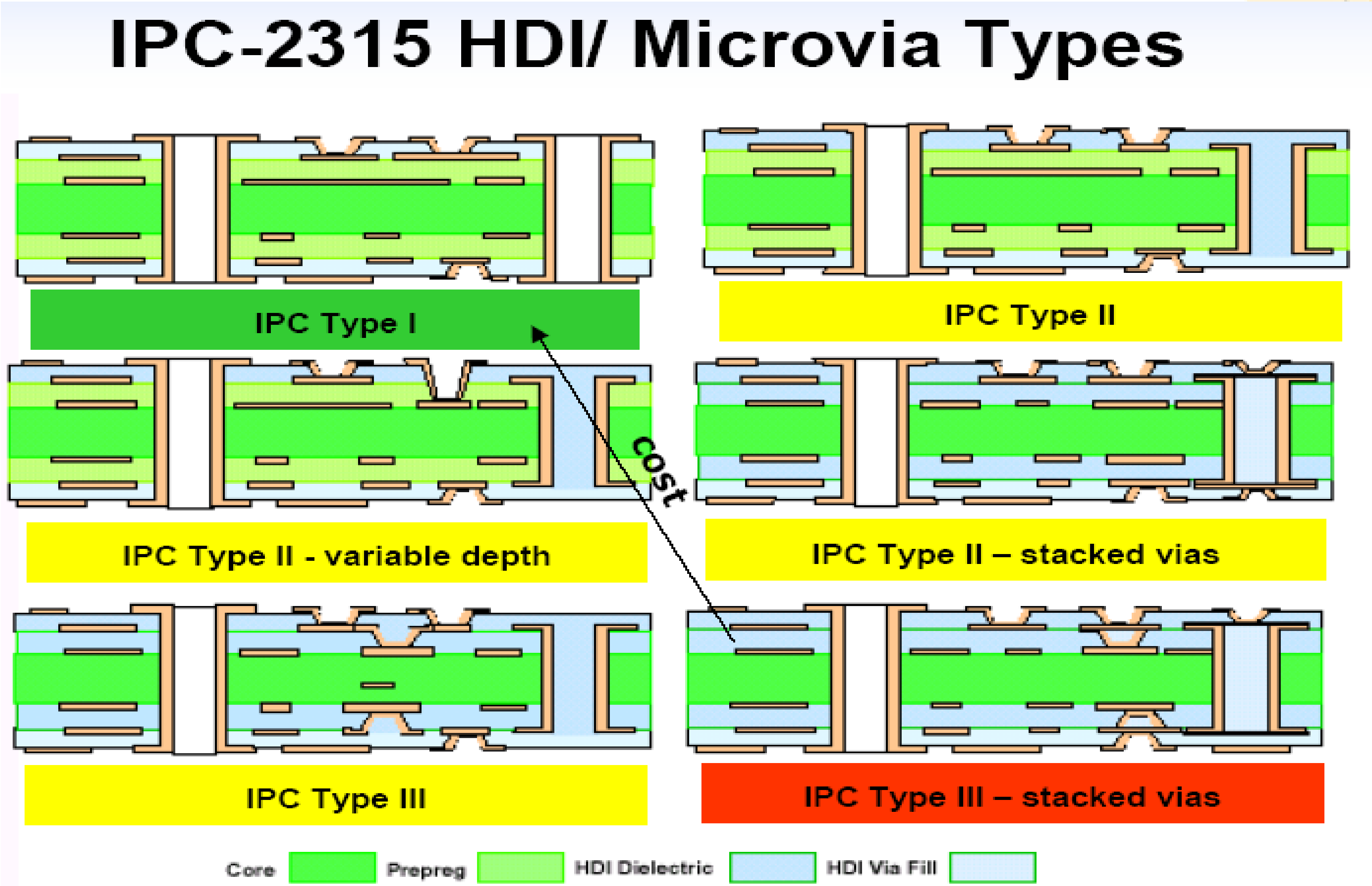
Definitions

HDI - High Density Interconnect

- Refers to substrates with vias of six mils (150μm) or less in diameter, made by any number of processes such as mechanical, plasma, laser, photo.

- Microvias are the principal feature of HDI, along with thinner dielectrics and smaller traces and spaces.

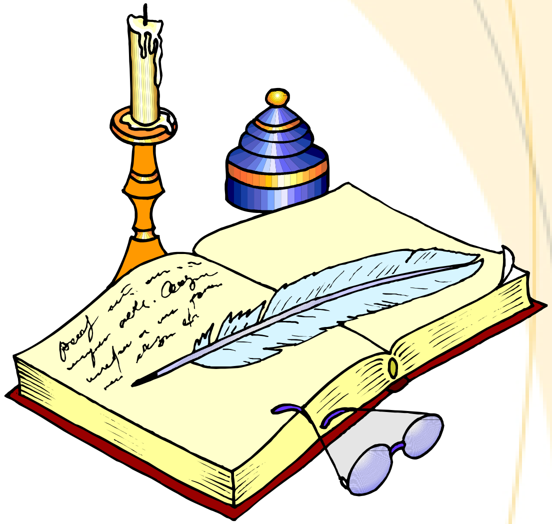
What is HDI



IPC-2315 HDI/ Microvia Types
IPC Type I
IPC Type II
IPC Type II - variable depth
IPC Type II – stacked vias
IPC Type III
IPC Type III – stacked vias
cost
Core
Prepreg
HDI Dielectric
HDI Via Fill

Definitions

HDI - High density interconnects (cont’d):



- Finer lines and spaces (<75 µm)
- Smaller vias (<150 µm)
- Pads (<400 µm)

- Board “build-up " and "sequential build-up (SBU)"

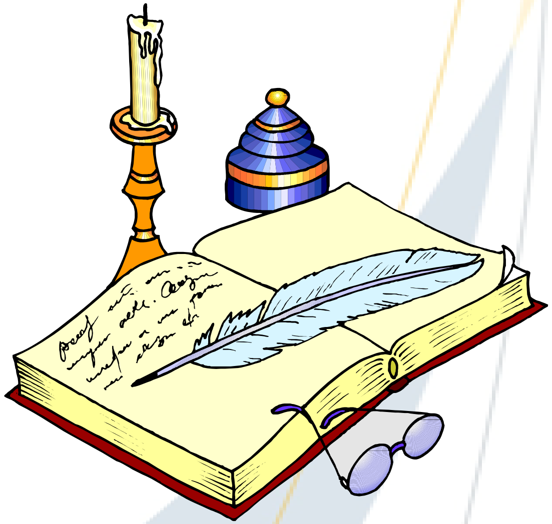
- Substrates or boards with a higher wiring density per unit area compared with conventional printed circuit boards

Definitions

Density - IPC defines higher component density in HDI

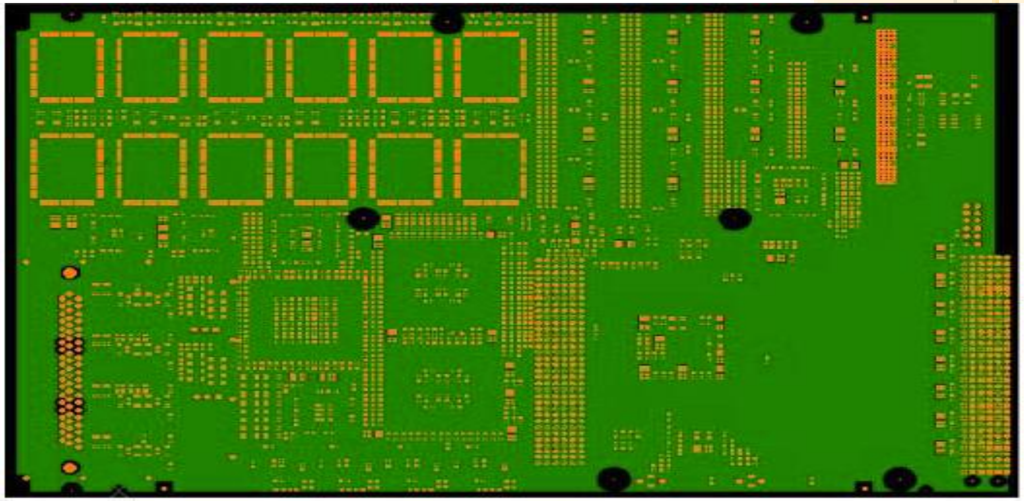
- The component connections number increases from 65 to 120 connection per sq inch

- Higher pad density increases above 20 pads/0.155 sq inch

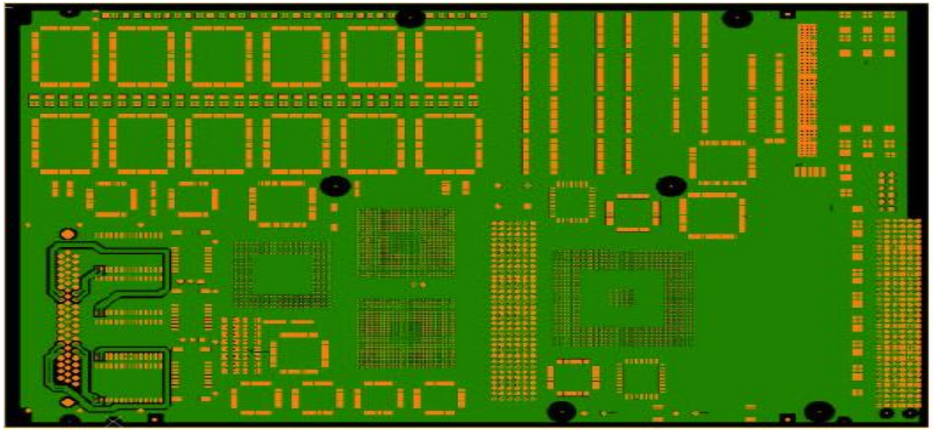


What is "Dense"

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



Size:9.2"x 6.3"
Thickness: 0.072"
Layers: 10
No. of components: 1621
No. of connection: 12456
Assembly density:
216 leads/in^2
Design Rules: 5/7, 6/12, 13/25



What is "Dense"

Critical PCB Design Parameters

- Board Dimensions
- Total Wiring Requirements
- Number of Layers
- Number of Embedded Resistors
- Number of Embedded Capacitors
- Active Component Types & Number
- Number of Discrete Resistors
- Number of Discrete Capacitors
- Design Type

What is Dense

The design task starts with three main phases

- Pre-design planning
- Constraint-based physical design
- Verification and iteration

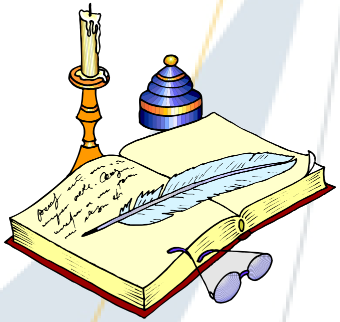
Definitions

in order to gain confidence that the capability meets the design demands, we define the following terms:

Demand - All available design data requirements

Capacity - All available design resources

Density - The ratio between the Demand to the Capacity.



Demand

Number of connection			
Number of components			
Available area			
Design technology		Analog	
		Mixed	
		Digital	
		Hi-end	
		Differential	
		Tuning	
Wiring Demand			

Demand

Number of connection			11961
Number of components			3275
Available area		Sqr Inch	83.78
Design technology			
		Mixed	2.5
		Digital	3
		Hi-end	3
		Differential	+ 0.25
		Tuning	+ 0.25
Wiring Demand			**68.5**

Demand

Demand

Numbers of connection	11961
Number of components	3275
Available area	83.78 inch²
Design technology	Digital
Differential busses +	☐
Ttuning +	☐
Wiring Demand	68.5035

Demand

The Demand data:

- The Demand data can be derived from every available CAD tool
- All the Demand data need to be analyzed by the designer

Demand

The Demand result:

- The wiring Demands calculation provides us with a number

- At the next phase, the board Capacity will be calculated

- Then, both results will create an inter-relationship between them.

Capability

There is always a limit on the amount of routing each board can accommodate.

The factors that define these limits 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

Capability

Technology efficiency			
		Mixed one side	
		Mixed both sides	
		SMT one side	
		SMT both sides	
		Mixed one side blind	
		Mixed both sides blind	
		2l HDI structure	
		4l HDI structure	
		6l HDI structure	
		8l HDI structure	

Capability

Technology efficiency			
		Mixed one side	0.35
		Mixed both sides	0.4
		SMT one side	0.45
		SMT both sides	0.5
		Mixed one side blind	0.55
		Mixed both sides blind	0.6
		2l HDI structure	0.65
		4l HDI structure	0.7
		6l HDI structure	0.75
		8l HDI structure	0.8

Capability

Pitch

Pad Diameter

Trace Width

Space Width

Units

Number of Signal Layers

Wiring Capacity

Capability

Pitch 1
Pad Diameter 0.4

Trace Width 0.1

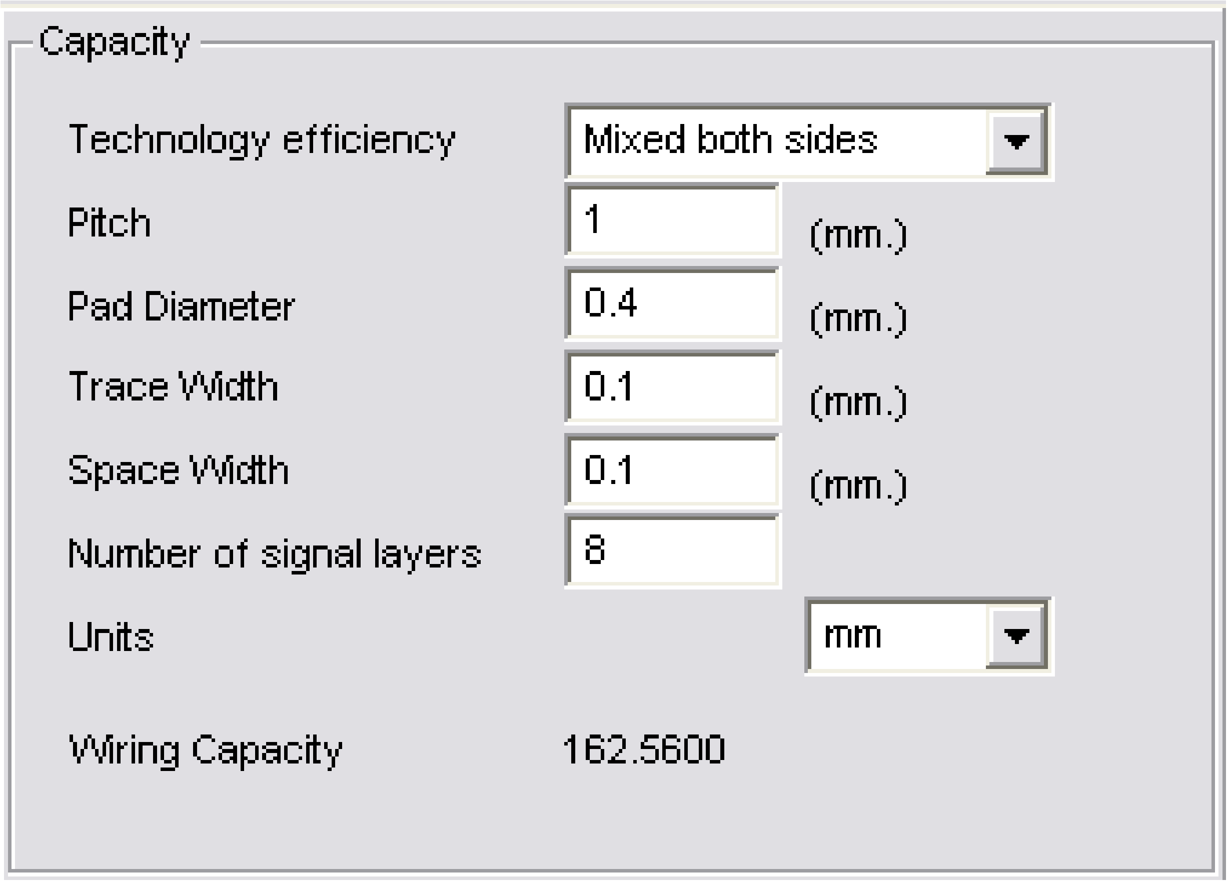
Space Width 0.1

Units: mm

Number of Signal Layers 8

Wiring Capacity 162

Capability


Capacity
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

Capability

The Capacity result:

- The Capacity wiring calculation provides us with a number
- Now we have on hand the wiring Demand number and the wiring Capacity number
- Both numbers create an inter-relationship between them

Density

The Density result:

We have inserted all the Demand requirements
And got a Wiring Demand number

If the wiring Demand exceeds 80-100 - consider HDI

Density

The Capability result:

If Capacity > = Demand — stick to your chosen technology

If Capacity < = Demand — change your design rules

Density

Density

Density Factor: 0.42 Stick to your chosen technology or perform board cost reduction

“What If”

If the result you get is almost “1”, then

Layout effort in design phase may result in more difficulties while doing changes/addition or reworks

- You can change some parameters and get a more “spacey” design

“What If”

If the result you get is higher than “1”, then

- The board layout with that ratio will require more PCB layout effort,
- will require more time,
- will require design compromises,
- will be difficult in future changes and
- will cost more in the manufacturing and assembly phases.
- You may change some parameters and get a more “spacey” design

“What If”

If the result you get is lower than “1”, then

- Your chosen design rules are good. Go ahead with the layout design.

- However, you may change some parameters to reduce cost.

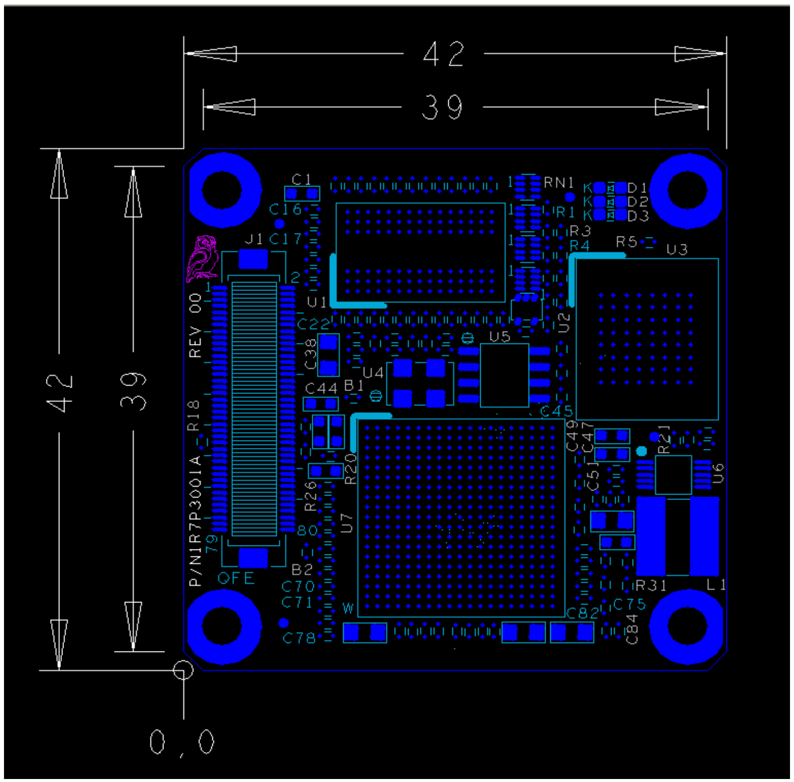
"What If"

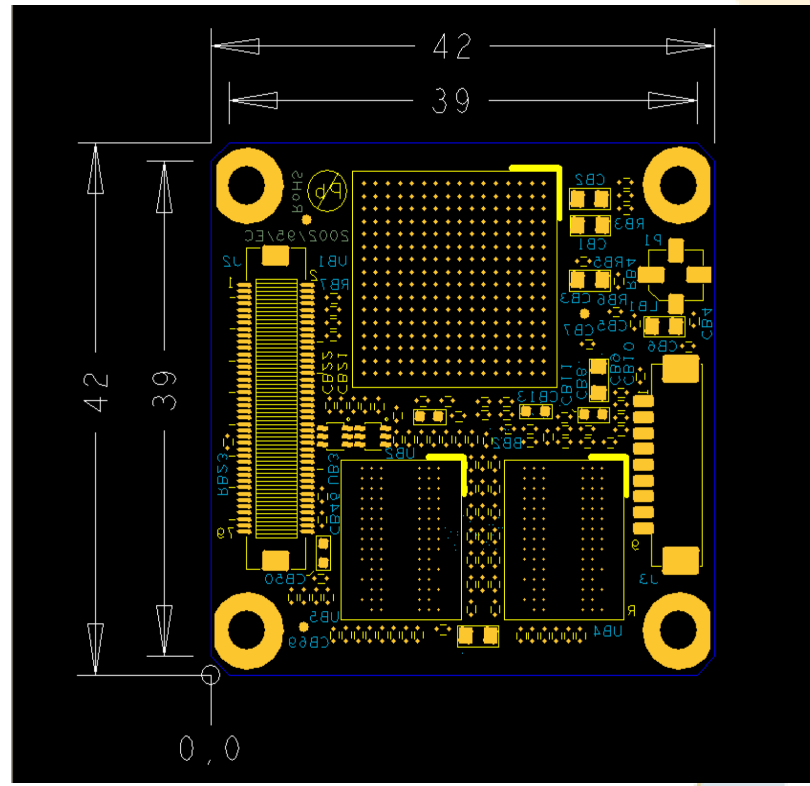
There are many parameters to juggle, e.g.,

- change technology,
- change trace/width options,
- change number of Signal layers.

- Once decided on a set of parameters, it is recommended to verify it with the PCB manufacturer.

Example 1





Example 1

demand		
number of connection =	1174	
number of components =	262	
available area =	1422	mm^2
design technology =	Hi-end	
differential busses tuning	+ 	
wiring demand =	**158.776**	

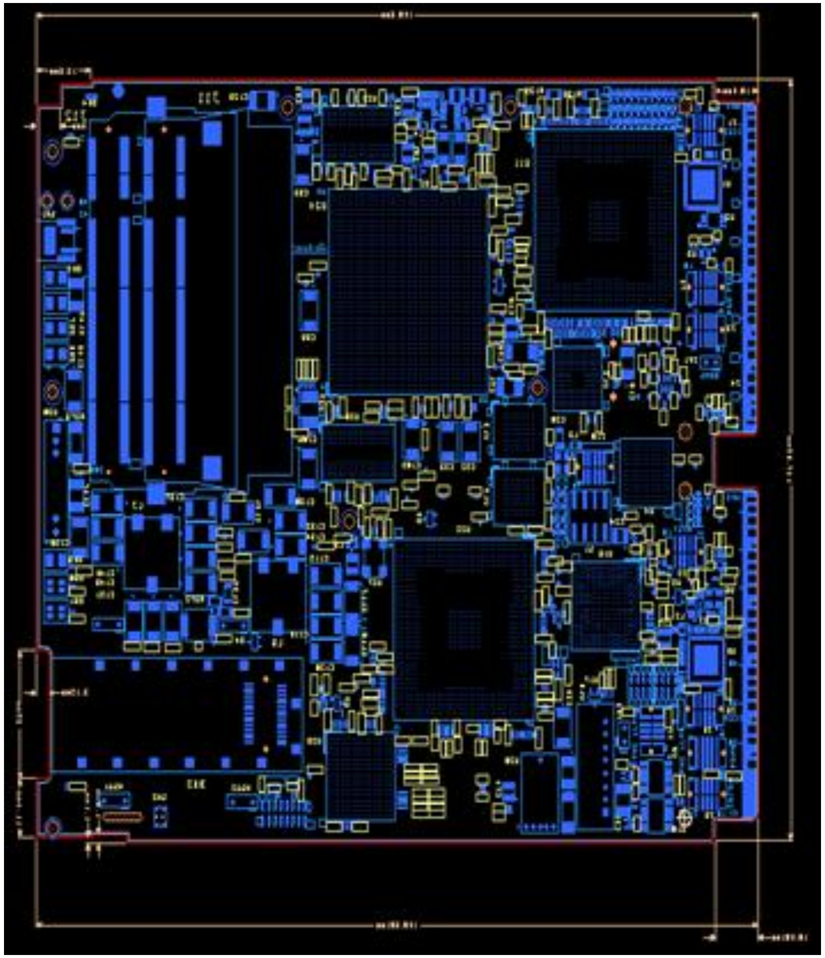
Example 1

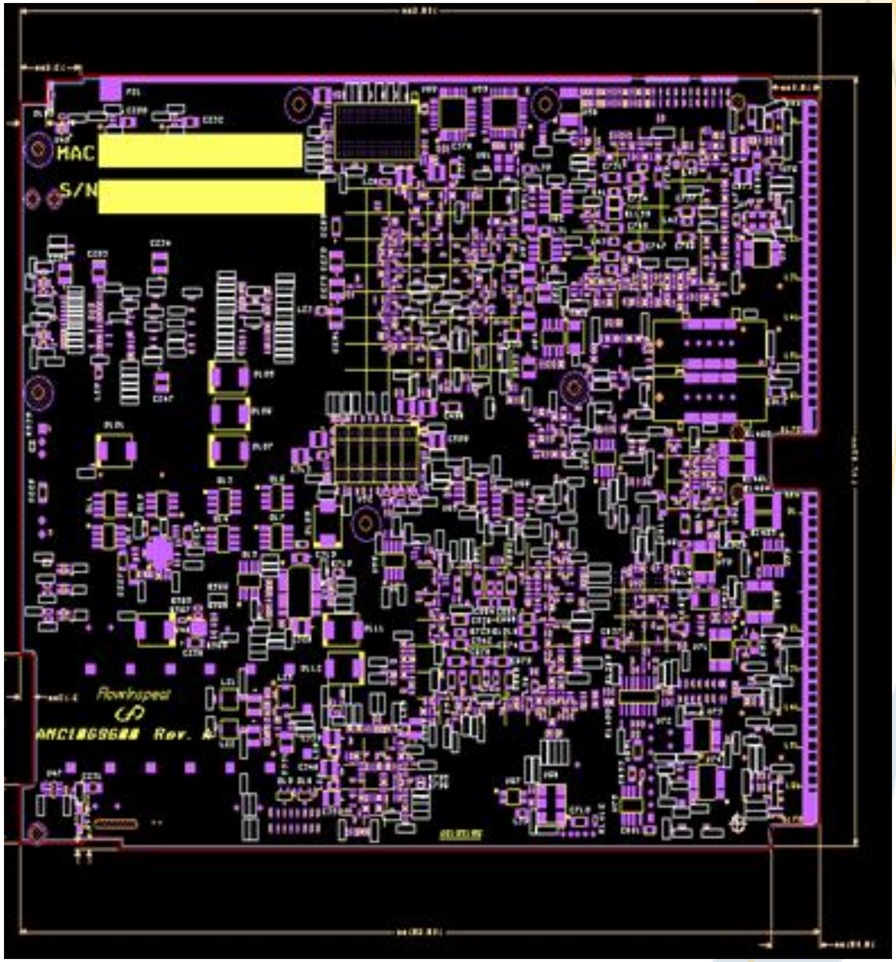
capacity		
technology efficiency =	2 HDI structure	
pitch =	0.8	
pad diameter =	0.45	
trace width =	0.1	
space width =	0.1	
number of signal layers =	8	
units =	mm	
wiring capacity =	**165.1**	

Example 1

Density factor =	**0.96**

Example 2





Example 2

demand		
number of connection =	9060	
number of components =	2727	
available area =	26103	mm^2
design technology =	digital	
wiring demand =	**81.82**	

Example 2

capacity		
technology efficiency =	Mixed both sides	
pitch =	1	
pad diameter =	0.45	
trace width =	0.1	
space width =	0.1	
number of signal layers =	6	
units =	mm	
wiring capacity =	**121.92**	

Example 2

Density factor =	**0.67**

Elbit Examples

Elbit Boards

- density_elbit_26.06.08.xls

Summary

The Demand number:

- Each company has its own nature of boards in accordance with its niche market.
- One needs to define the board nature.

Specification	Typical (mm)	Premium (mm) PCB Thickness > 1.5 mm	Premium (mm) PCB Thickness <= 1.5 mm
Trace & space width	0.1/0.1	0.076/0.076	0.076/0.076
Drilled hole diameter	0.305	0.254	0.15
Finished via diameter	0.254	0.203	0.1
Via capture pad	0.66	0.508	0.275
Aspect ratio	7:1	10:1	10:1

Summary

- Ones a normative wiring demand number has been defined all the boards should relate to it.

- Number above the normative number: go for HDI.
- Number below the normative number :stick with current technology or perform cost reduction.
- Create your own density reference data base.

Summary

The Density number

- Provides a solid measure for the capability of implementing a PCB layout at the very early stage of the design

- One needs to perform several "**what if**" simulation phases in order to get the best results

References

1. AN114 – Altera; Designing high density BGA packages for Altera
2. Printed Circuit Board Layout Time Estimation – C. Bazeghi & J. Renau U of California, Santa Cruz; funded by Sun Microsystems
3. Evaluating High – Density design Alternatives, by Merix;
4. PCB2EST- Quick Estimate Program for PCB Layout by Oztronix 2005;