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
- Contributions 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.

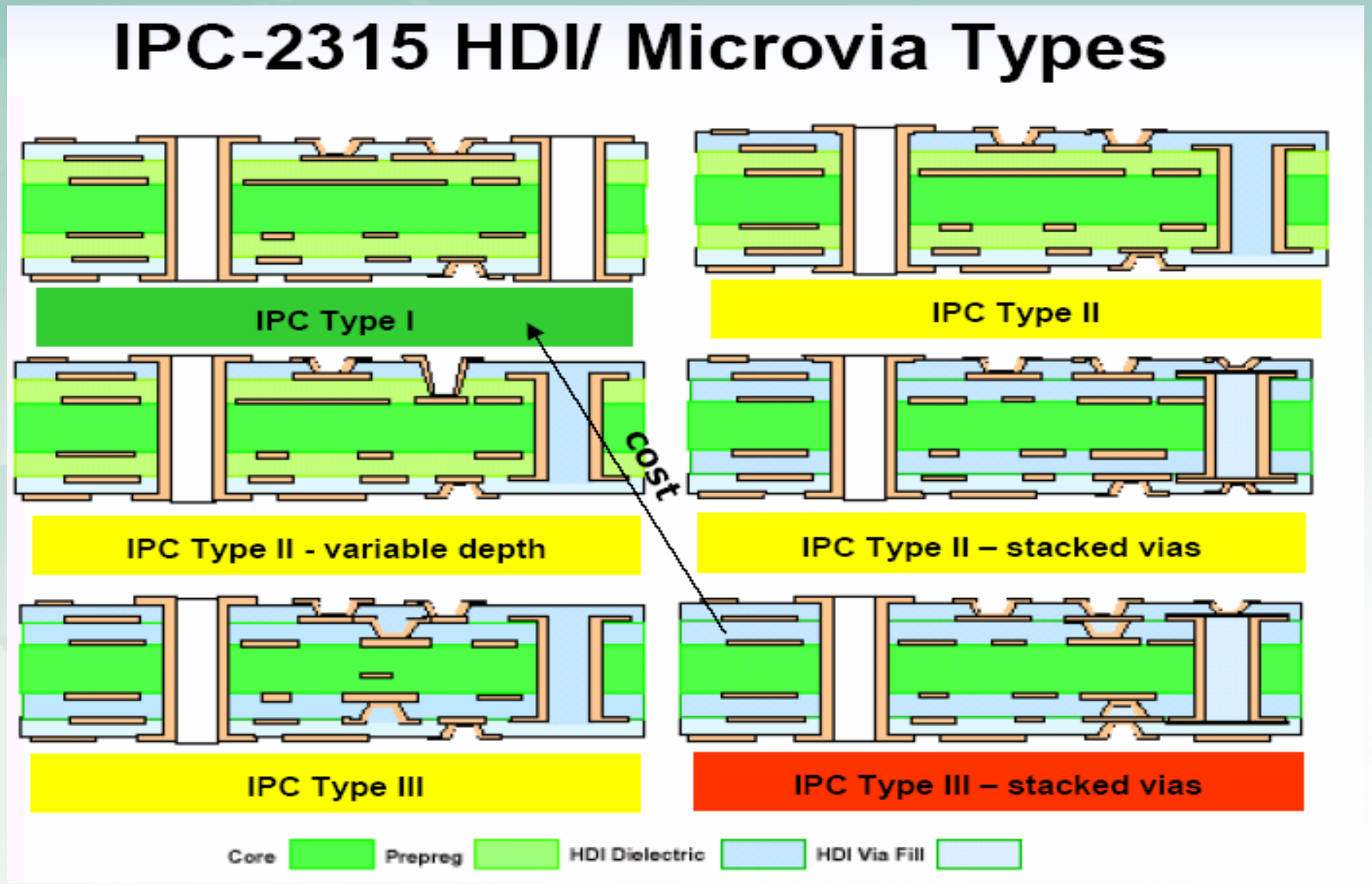
- In order to gain confidence in the algorithm, we bench-marked 6 of Elbit Systems communication boards, 2 Opgal boards and 2 Adcom boards

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
cost
IPC Type II - variable depth
IPC Type II – stacked vias
IPC Type III
IPC Type III – stacked vias
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

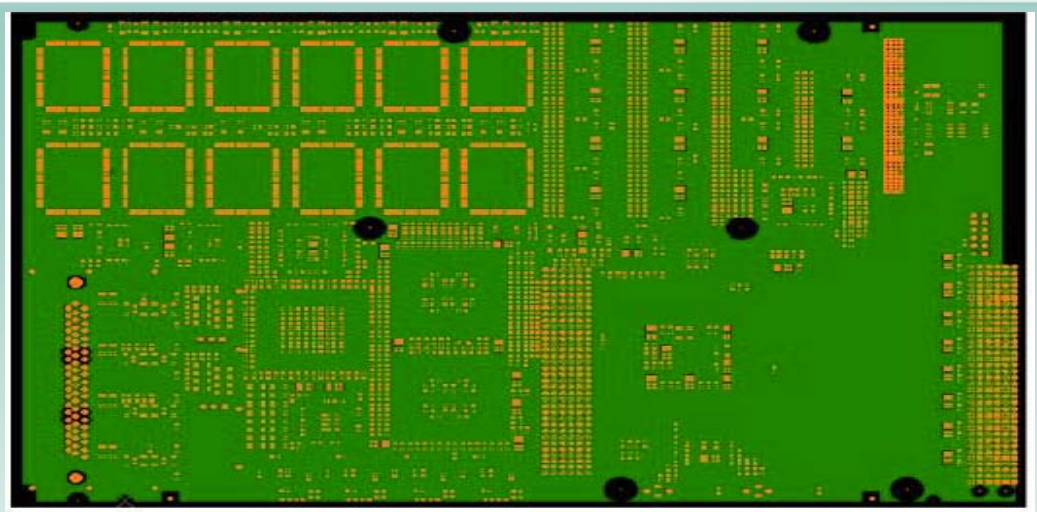
Definitions

Density - IPC defines higher component density in HDI

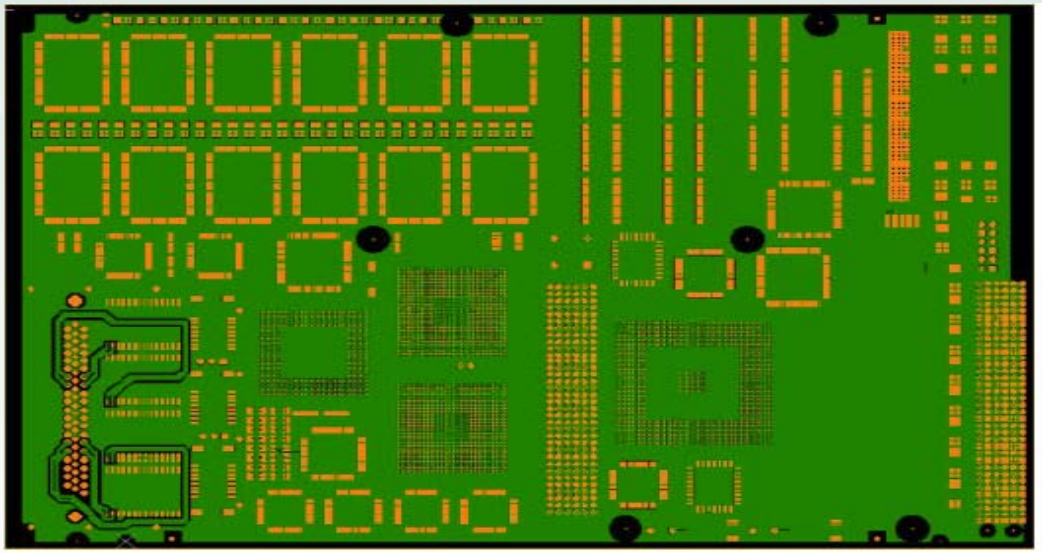
- The component connections number increases from 65 to 120 connection per sq inch
- Higher pad density (>20 pads/cm²)

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/in²
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 available

Density - The ratio between the Demand to the Capacity.

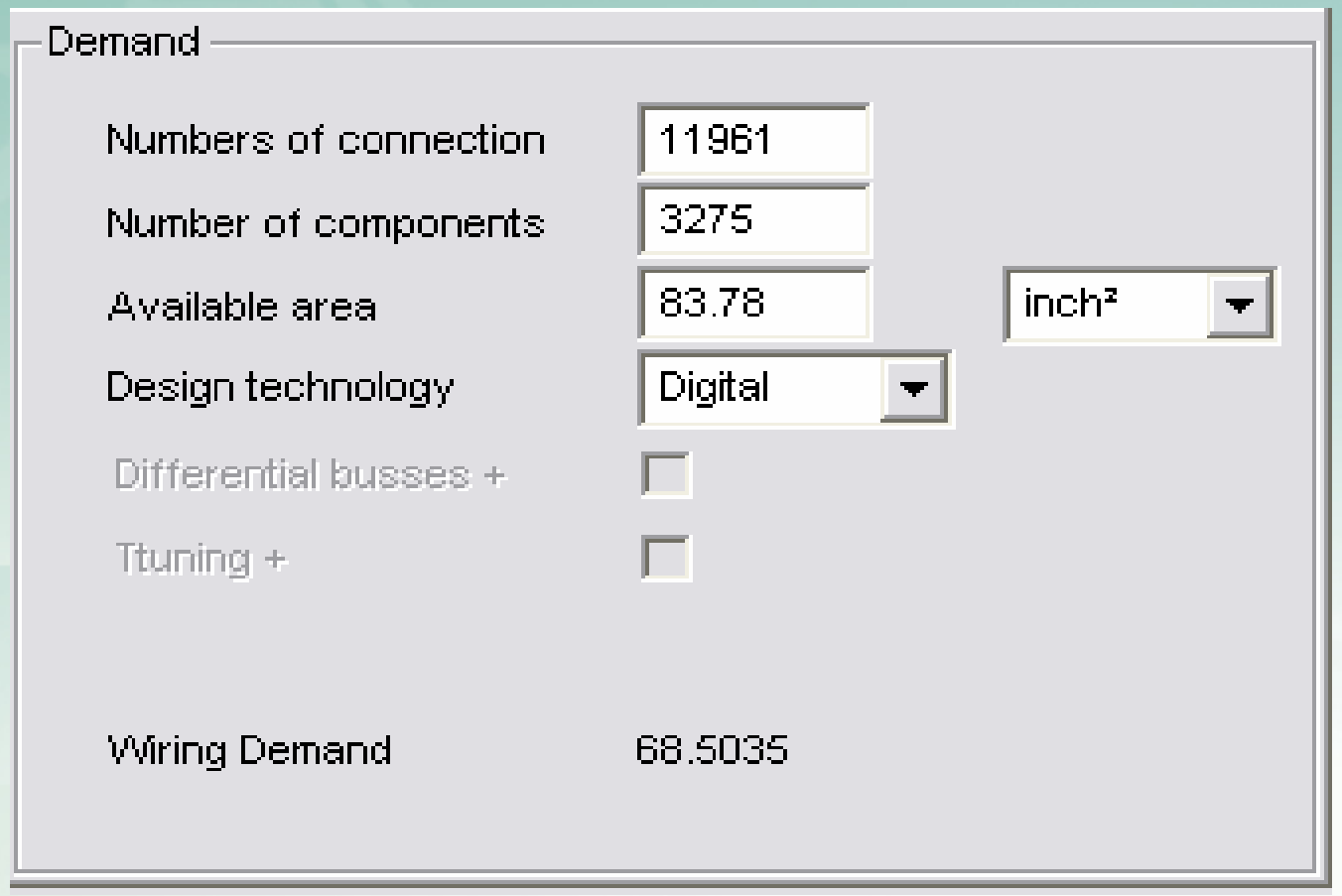
Demand

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

Number of Signal Layers

Wiring Capacity

Capability

Pitch 1
Pad Diameter 0.4

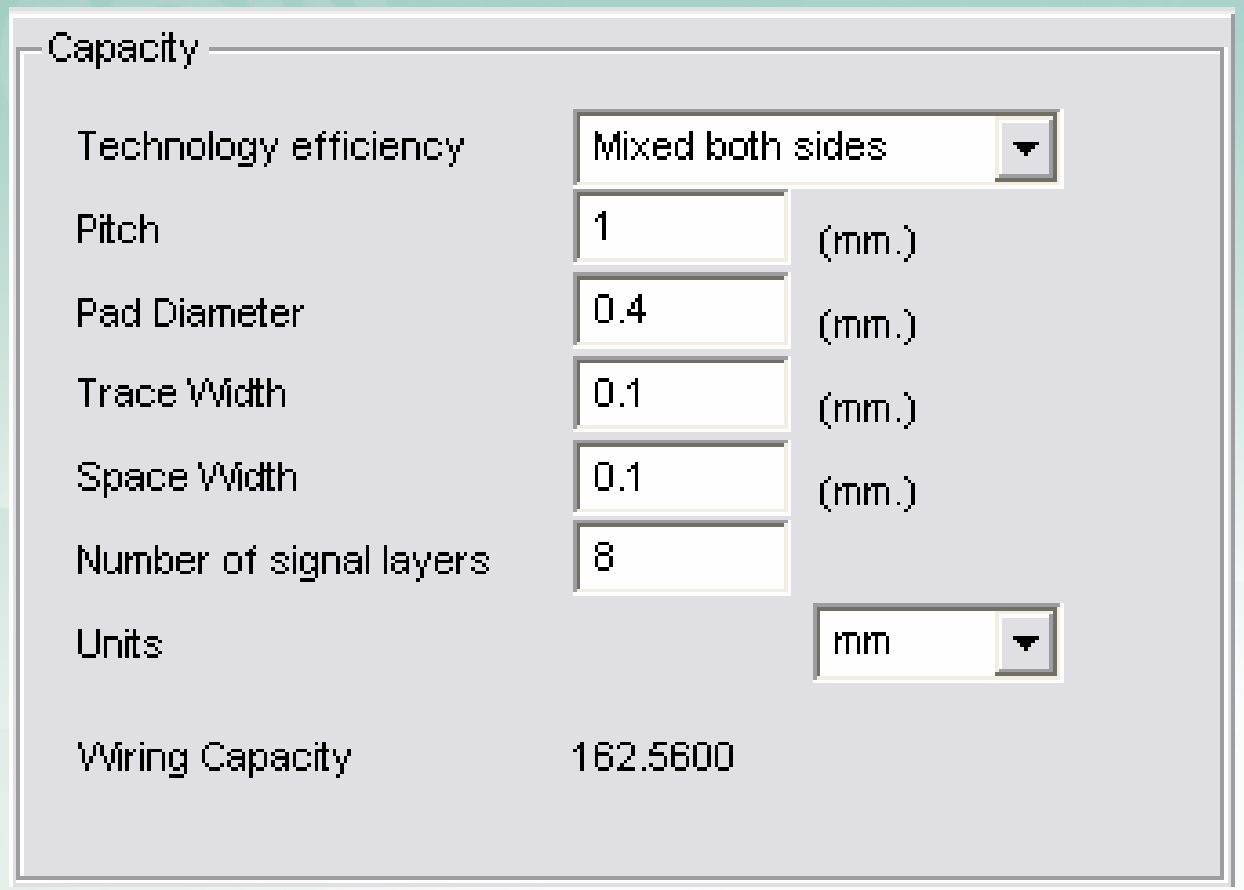
Trace Width 0.1

Space Width 0.1

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

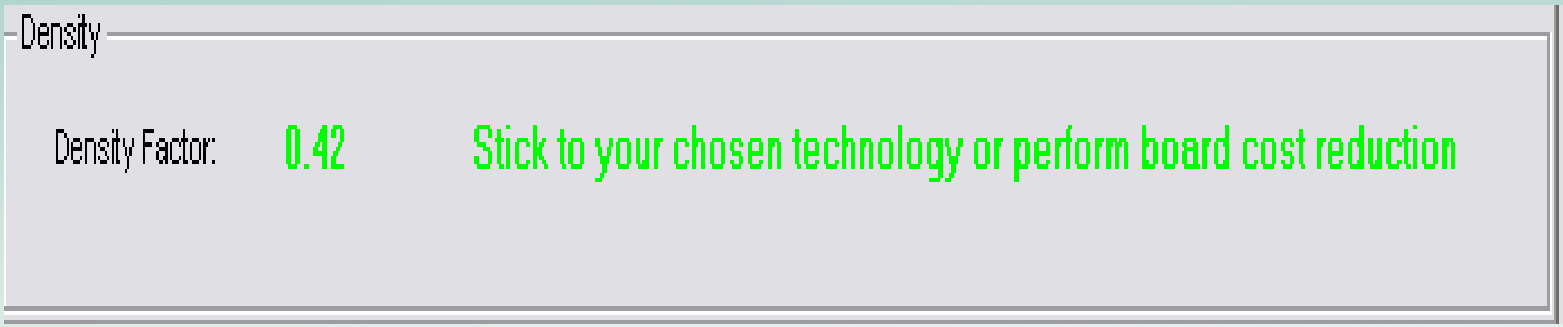
Density

If the wiring Demand exceeds 80-120 - consider HDI

Density

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.

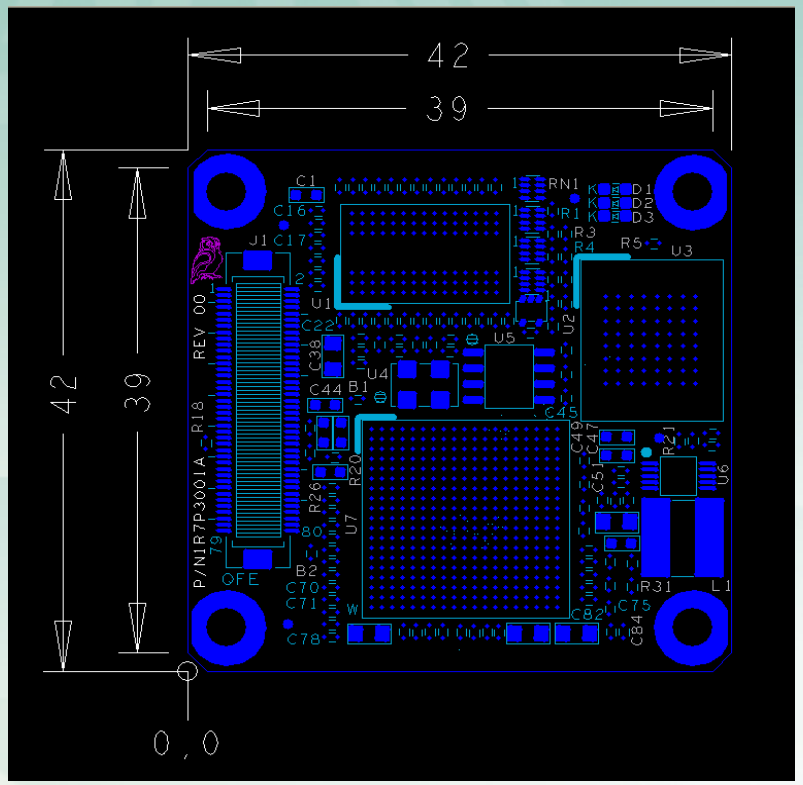
“What If”

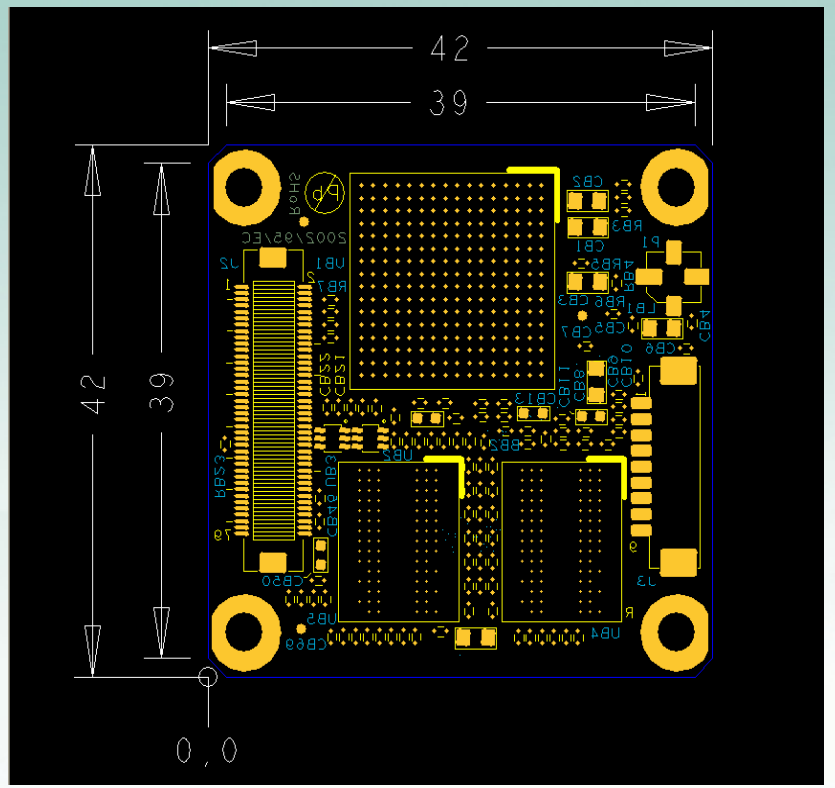
There are many parameters to juggle, e.g.,

- 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	mm²
design technology =	Hi-end	
differential busses		
tuning	+	
wiring demand =	**158.776**	

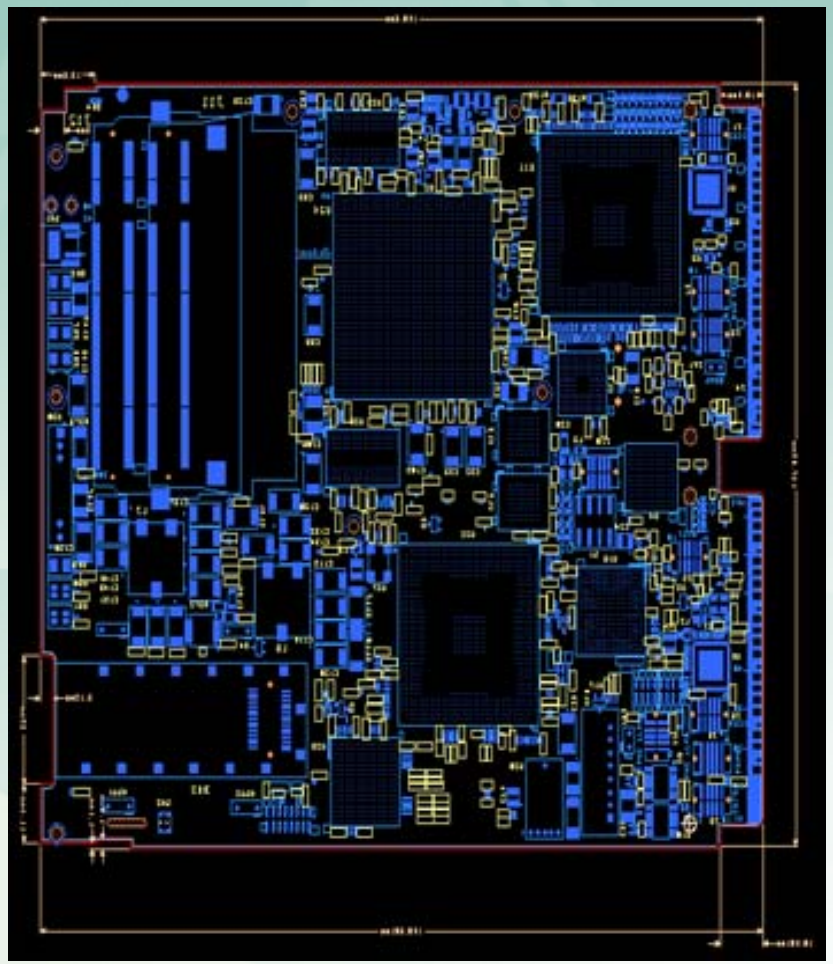
Example 1

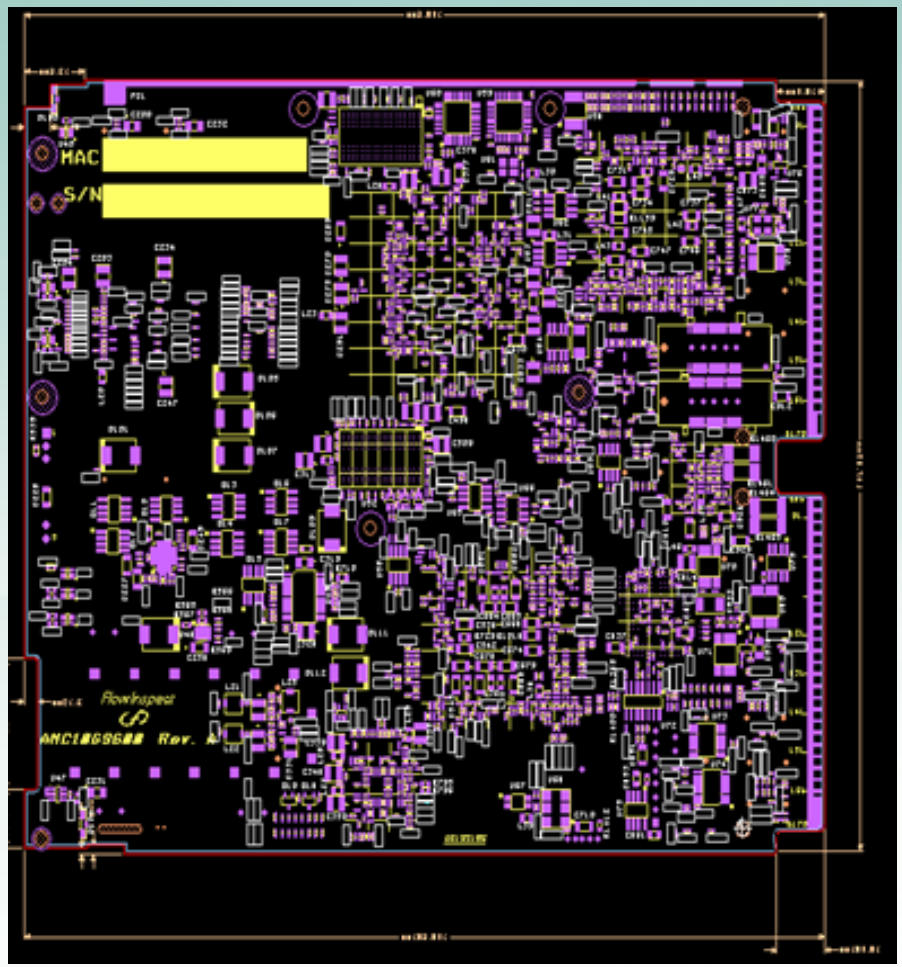
capacity			
technology efficiency	=	2 HDI structure	
pitch	=	0.8	
pad diameter	=	0.45	
trace width	=	0.1	
space width	=	0.1	
number of signal layers	=	8	
units	=	mm	
wiring capacity	**=**	**165.1**	

Example 1

Density factor = 0.96

Example 2





Example 2

demand			
number of connection	=	9060	
number of components	=	2727	
available area	=	26103	mm²
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**

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;