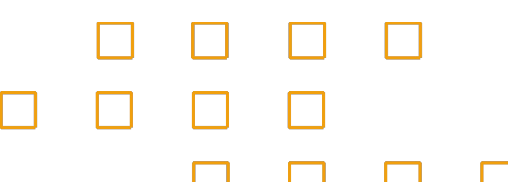


Applications of Semi-Additive Process Technology to PCB Design and Production

Authors:

Paul A. Dennig, Ph.D. and Mike Vinson

Averatek Corporation

Santa Clara, California

Technical Conference January 25–27, 2022

THREE QUESTIONS

1. Are there benefits to shrinking circuits on Printed Circuit Boards?
2. Can you give an example?
3. What are the effects?

THREE QUESTIONS

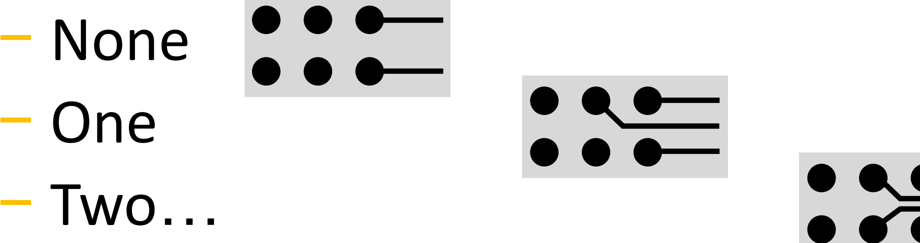
1. Are there benefits to shrinking circuits on Printed Circuit Boards?

2. Can you give an example?

3. What are the effects?

Shrinking Circuits: Where Do We Start?

- What challenges face every layout specialist?
  - Ball Grid Arrays (BGA), routing traces outside of BGA, joining both, etc.
- Which process to use?: Semi-Additive Process (SAP) vs. Subtractive Etch (SE) Process
- General design: look at # of signal lines passing between BGA pads
  - None
  - One
  - Two…
  - Many



- Specific case: circuit layout example
- Look at some electrical effects of shrinking circuits

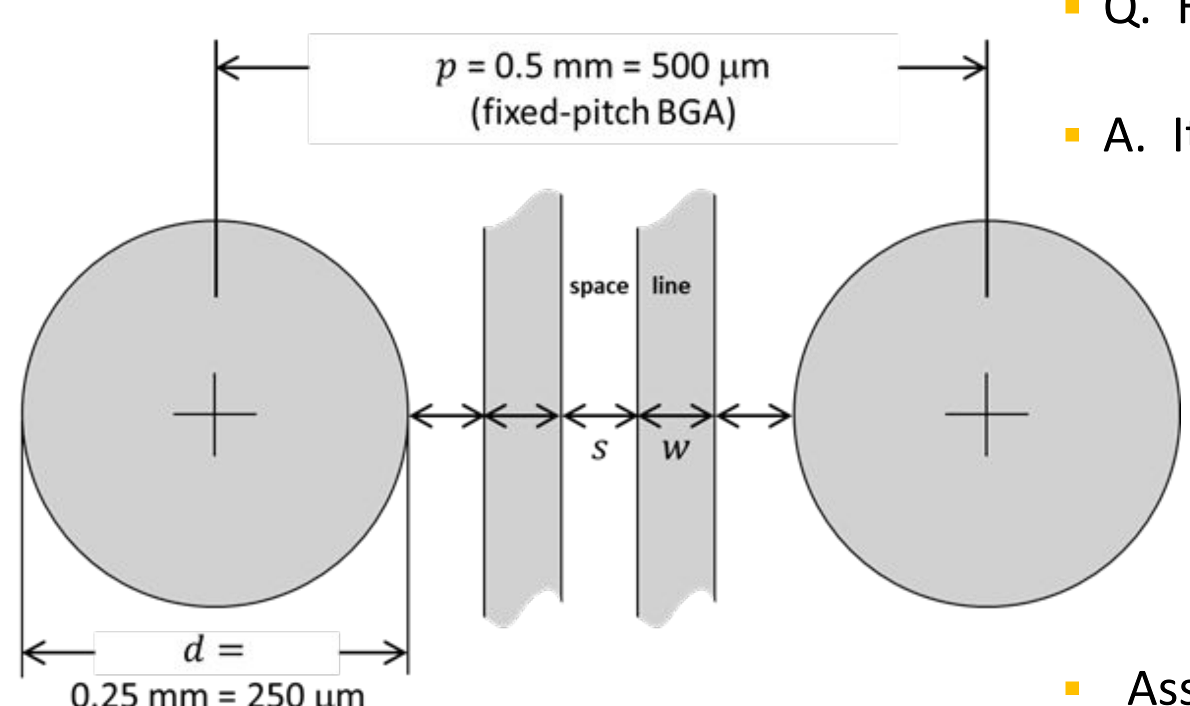
LOOK FOR THESE ON NEXT SLIDES:

- Increase # of traces through BGA

- Reduce # of:
  - layers in PCB
  - laminations
  - microvias

- Combined, these promote:
  - decreased board size
  - higher reliability
  - lower costs
  - smaller environmental footprint

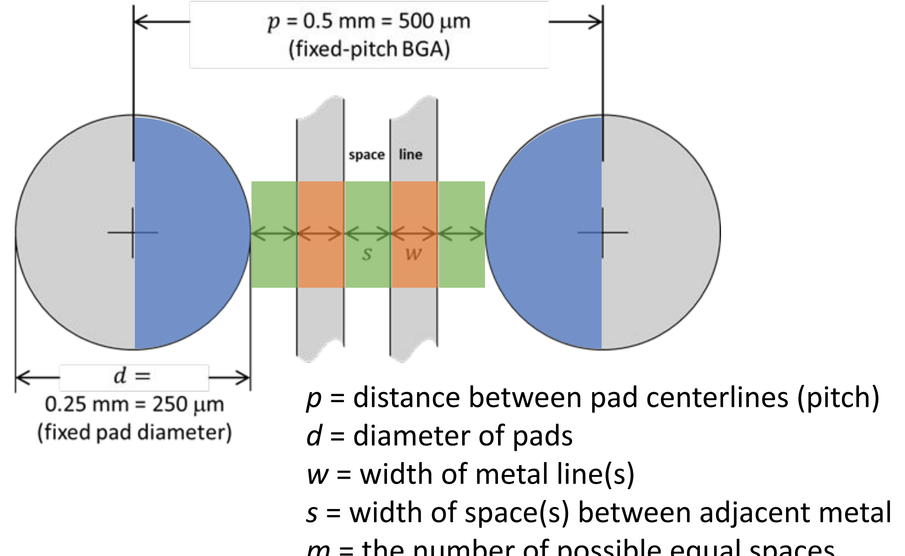
General Case: Ball Grid Array (BGA) Land Pattern



- Q. How many Lines can fit between two BGA Pads?
- A. It depends on the Line width. First, define:
  - p = distance between pad centerlines (pitch)
  - d = diameter of pads
  - w = width of metal line(s)
  - s = width of space(s) between adjacent metal
  - m = the number of possible equal spaces
  - n = the number of possible equal lines
- Assume:
  1. All pads are for signals (worst case for routing)
  2. Line width = space between lines

Calculating the number of Lines, n

- $m = n + 1$ [Eq. 1]
- $p = d/2 + (m*s) + (n*w) + d/2$ [Eq. 2]
- $n = \text{Int}[(p - d - s) / (w + s)]$ [Eq. 3]



p = distance between pad centerlines (pitch)
d = diameter of pads
w = width of metal line(s)
s = width of space(s) between adjacent metal
m = the number of possible equal spaces
n = the number of possible equal lines

Table of "n" for Different Minimum Feature Sizes

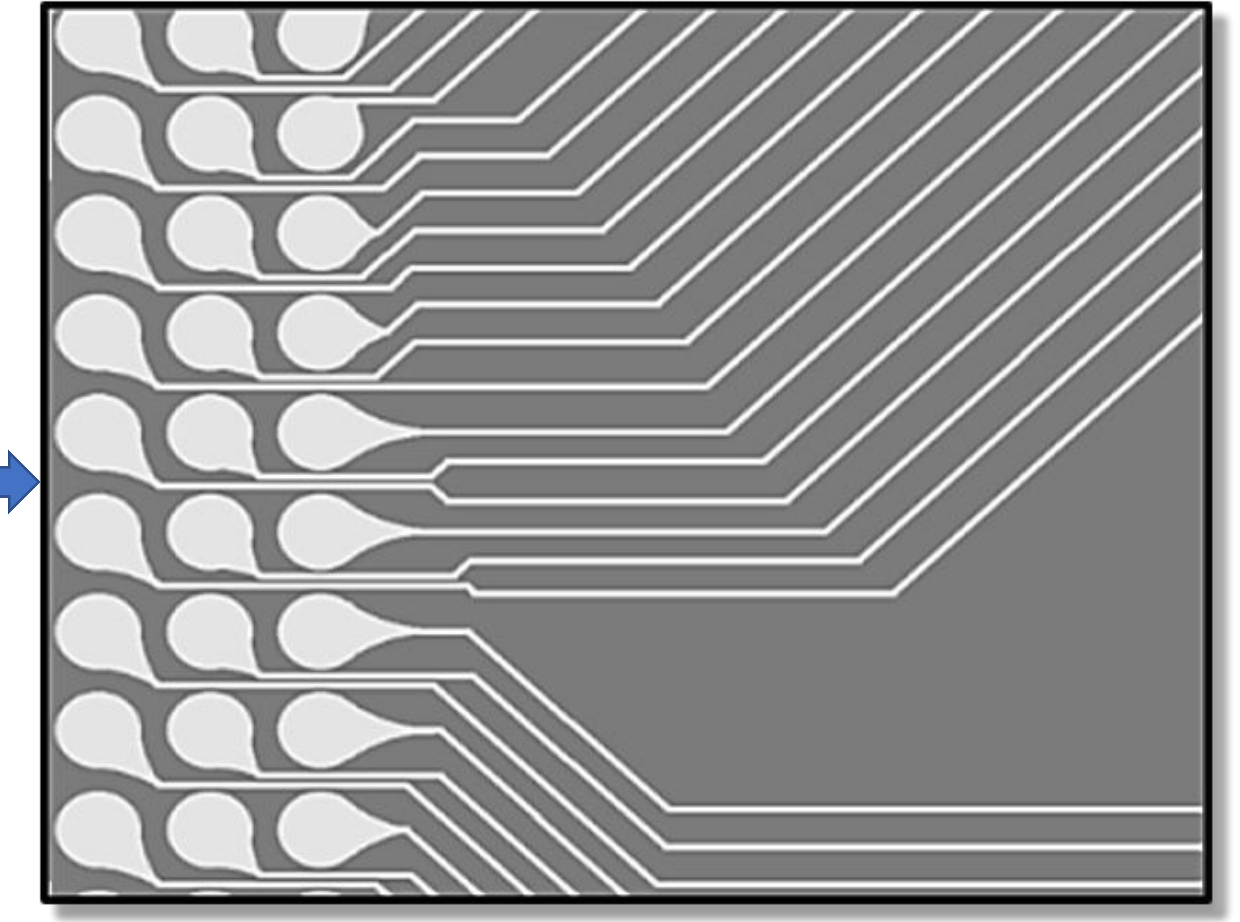
Ball Pitch	Line & Space (minimum feature size, s and w)	Routing Layers Required	# of Lines (between pads, n)
0.5 mm	75 µm	12*	1
	50µm	6	2
	35µm	4	3
	28µm	3	4
	19µm	2	6
	10µm	1	12

SE
SAP

Notes:
1) * = starting assumption
2) ∴ [R.L.R. ∗ (# Lines)] = constant; 12 in this example
3) white rows – see next pages

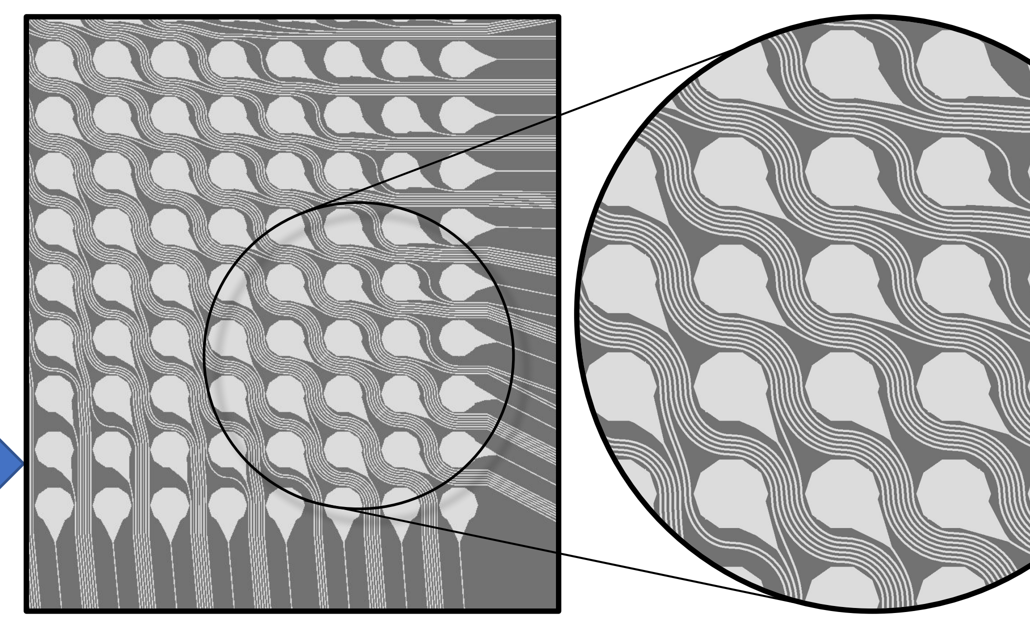
BGA Land Pattern, 50x50 BGA Grid, 100 μm-Line Pitch

Ball Pitch	Trace & Space (minimum feature size, s and w)	Routing Layers Required	# of Traces (between pads, n)
0.5 mm	75 μm	12	1
	50μm	6	2
	35μm	4	3
	28μm	3	4
	19μm	2	6
	10μm	1	12



BGA Land Pattern, 50x50 BGA Grid, 38 µm-Line Pitch

Ball Pitch	Trace & Space (minimum feature size, s and w)	Routing Layers Required	# of Traces (between pads, n)
0.5 mm	75 µm	12	1
	50µm	6	2
	35µm	4	3
	28µm	3	4
	19µm	2	6
	10µm	1	12



Routing More Lines Between Pads

By decreasing linewidths:

- Can route more traces through a BGA
- This reduces the # of routing layers
- Compare SAP and SE for
  # of lines between pads
  - SE upper limit is about 1 trace
  - SAP upper limit: photolithography limited

Benefits of using SAP Technology in PCB Designs

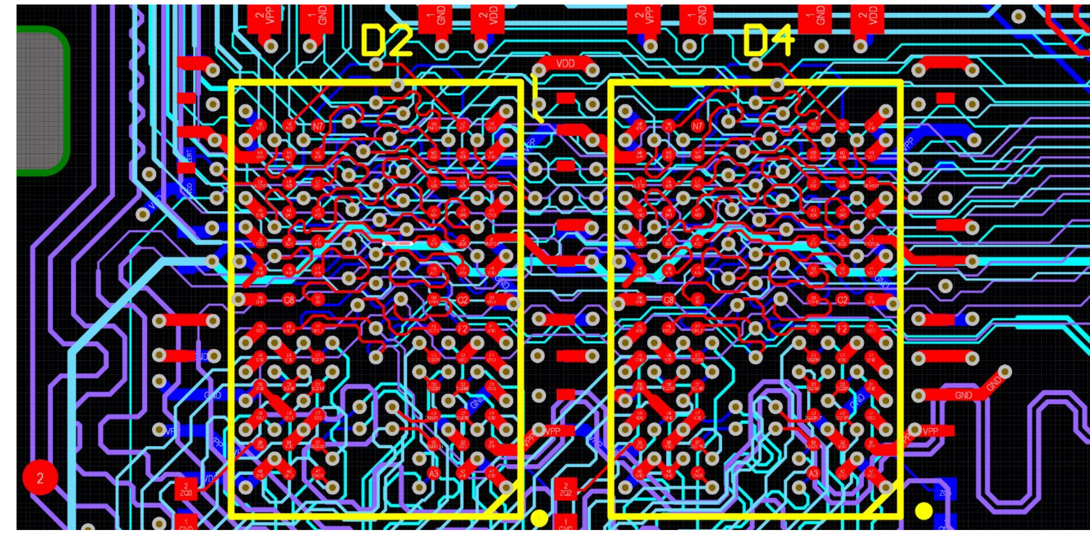
- Increase # of traces through BGA
- Reduce # of:
  - layers in PCB
  - laminations
  - microvias
- Combined, these promote:
  - decreased board size
  - higher reliability
  - lower costs
  - smaller environmental footprint

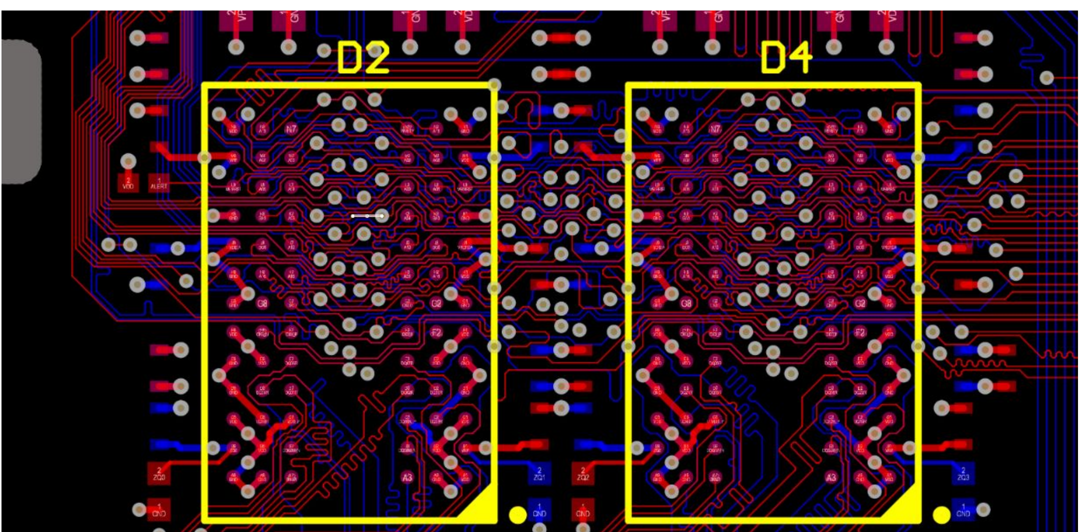
- Can design with much:
  - shorter line lengths
  - thinner dielectrics
  - thicker dielectrics (& use tightly-coupled differential conductor pairs)
- Could combine low-density SE layers with high-density SAP layers in one board design

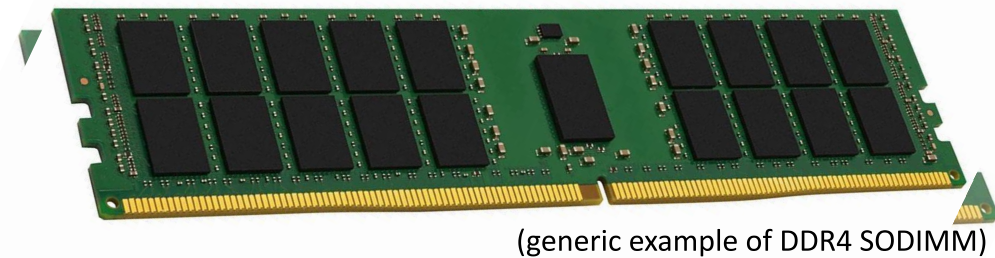
THREE QUESTIONS

1. Are there benefits to shrinking circuits on Printed Circuit Boards?
2. Can you give an example?
3. What are the effects?

Specific Circuit Application Example: DDR4 S







(generic example of DDR4 SODIMM)

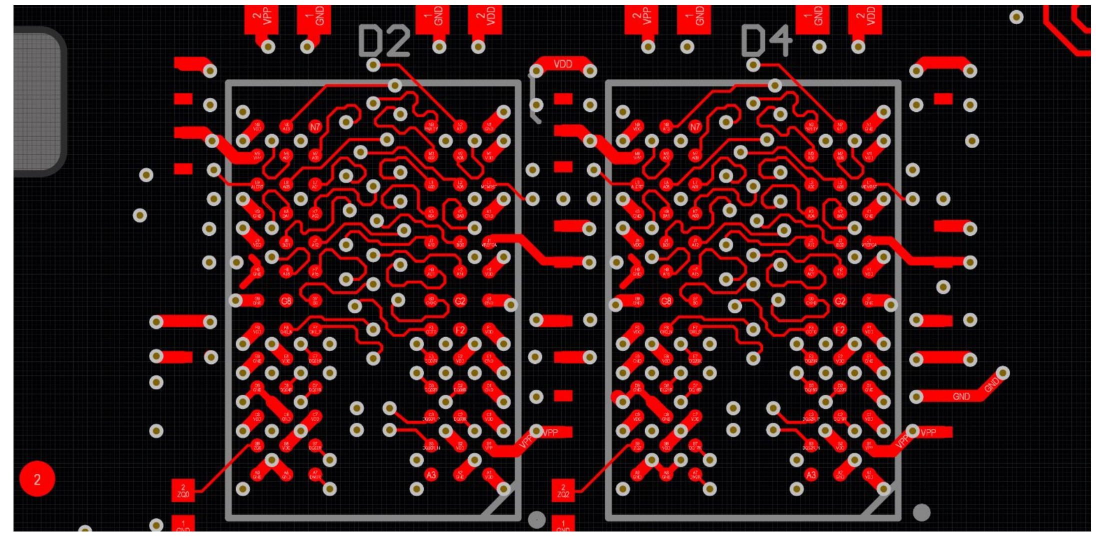
- DDR4 SODIMM reference design (above)
- Multi-layer CAD drawings (at left)
- *Upper drawing*: original reference design, 75 um lines and spaces for SE Process
- *Lower drawing*: re-design, using SAP design guidelines, 35 um line & 52.5 um space

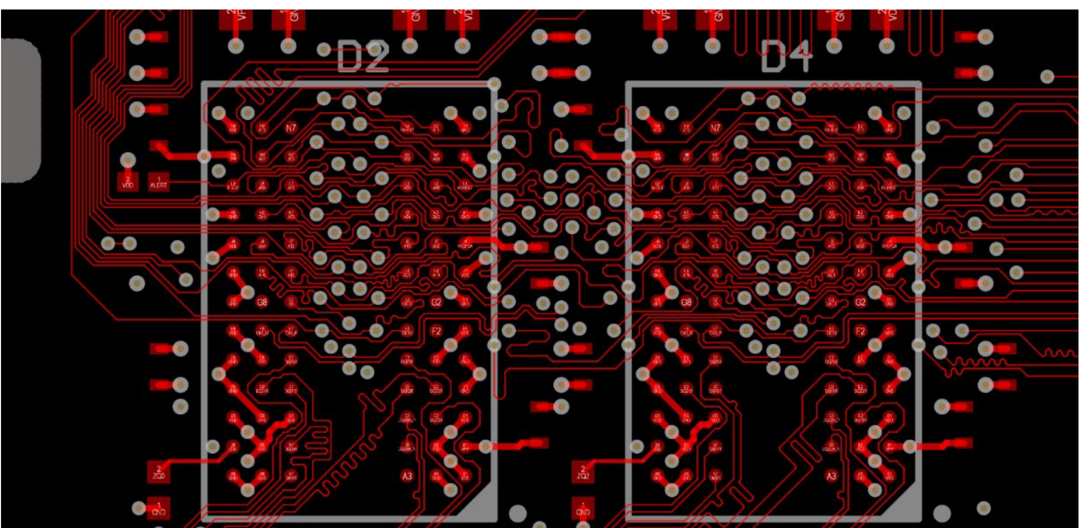
(Courtesy, T. Chester, Ref. [3] & Altium)

Notes:

(1) SODIMM (or SO-DIMM): Small Outline Dual In-Line Memory Module
(2) DDR4: Double Data Rate 4
(3) e.g., https://resources.altium.com/p/fly-topology-routing-ddr3-and-ddr4-memory

Specific Example: DDR4 SODIMM (cont'd.)





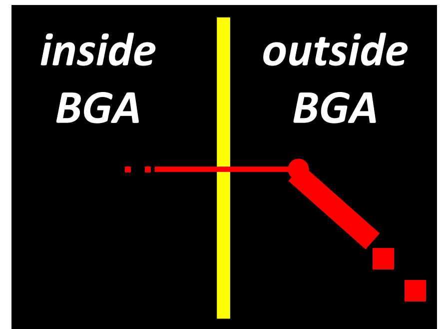
- Same circuit
- Focus: Signal Layer around BGA land pattern, single metal layer views
- *Upper drawing*: original design 75 um lines and spaces for SE. One or two lines fit.
- *Lower drawing*: SAP version of the same circuit. Up to four lines run between pads.

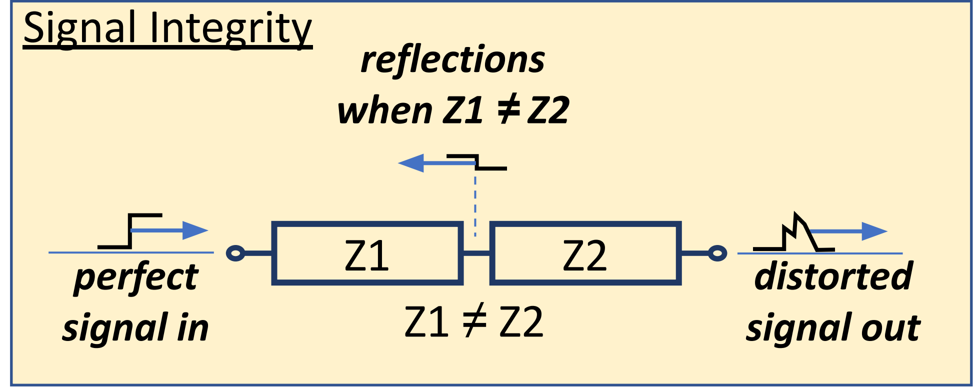
(Courtesy, T. Chester, Ref. [3] & Altium)

THREE QUESTIONS

1. Are there benefits to shrinking circuits on Printed Circuit Boards?
2. Can you give an example?
3. What are the effects?

Considerations





- Look at joining finer lines at the BGA to wider lines outside
- Board designs typically use a Characteristic Impedance of Z_0 = 50 Ohm for signal lines
- However, when signals meet mismatched impedances, reflections result & can distort the signal
- Signal Integrity (SI): potential signal distortion = ***f*(**impedance, line length, slew rate, etc.**)**

Refs. [5 – 7]

Impedance Variation

- Q1. Why Z_0 = 50 Ohms?
- Q2. Where does $Z_0 \neq$ 50 Ohms?
- Q3. Why?
- Q4. When does it matter?
- Q5. How much?

A1. This is a practical but arbitrarily-chosen standard value. Often, Integrated Circuit I/O's are designed to it.

A2. Generally, where a metal line cross-section changes*

A3. By (i) design or (ii) through manufacturing process variation

A4. When good Signal Integrity (SI) becomes critical

A5. Today, +/- 10% Z_0 tolerance is accepted. Too often, it can't be achieved. Moreover, we see this spec tightening into the future.

Note: * – other factors also apply. See, e.g. Refs. [5 – 6]

Two Sources of Impedance Variations due to Widths

- Manufacturing Process Variations in width
  - From process variation: across a panel, within a batch, batch-to-batch, etc.
  - Impedance (Z_0) change: Z_0 ↑ as line width ↓ (see next slide)
  - Possible Signal Integrity (SI) degradation if I.C.'s Z_0 not matched to line's Z_0
  - However, there is less process variation with SAP compared to SE

- Intentional Design Changes in width
  - Uneven-size metal lines joined by design (e.g. route lines away from BGA)
  - Signal reflections may occur any time there is impedance mismatch
  - Reflection magnitude depends on many factors
    - Q. Are the reflections significant?
    - A. Must do SI analysis (line length, signal slew rate, etc.)

Impedance Changes from Process Variations

EXAMPLE: DIELECTRIC
THICKNESS ≈ 5 mils (127 µm)
Dk = 4.0

METAL TRACE
WIDTH ≈ 10 mils (254 µm)

SAP variance
Tight impedance control with SAP will even benefit wide lines (10 mil nom)

Typical SE variance
$Z_0 \approx 50$ +/- 5 [Ω]
= nom +/- 10 %

$Z_0 \approx 50.2$ +/- 0.25 [Ω]
= nom +/- 0.5 %

□ *getting narrower* || *getting wider* □

CHARACTERISTIC IMPEDANCE, Z_0 [Ω]: 55.0, 54.0, 53.0, 52.0, 51.0, 50.0, 49.0, 48.0, 47.0, 46.0, 45.0

-30, -20, -10, 0, 10, 20, 30

DEVIATION, ACTUAL WIDTH vs. NOMINAL DESIGN LINE WIDTH [microns, µm]

- Microstrip transmission line
- Assume:

  - one 10 mil-wide trace (250 um)
  - 5 mil-thick dielectric
  - Dk = 4.0
  - ground plane beneath
- Variations:
  Processing □ Width
- Calculate the impedance, Z_0, as line width changes

[7]

Effects of Line Width Changes (cont'd.)

- Tighter manufacturing tolerance on linewidth with SAP gives tighter Z_0 than with SE
- Intentional width changes by design, such as during PCB layout (e.g., routing out of BGA)
  - Even more dramatic DZ_0 , compared to manufacturing process
- Impedance changes will result in reflections. The lengths of the lines also play a role
- Therefore, signal integrity must consider both effects

  (i) design

  (ii) process

Conclusions

- We've demonstrated a SAP PCB layout on the signal layer of a DDR4 SODIMM circuit

- SAP
  - minimizes line widths and their processing variations
    - allows more lines between BGA for many benefits
    - fewer layers, laminations, and microvias
    - …resulting in lower production costs and higher reliability
    - worth pursuing, especially for designs with large BGA pad counts
  - SAP layers may be combined with SE layers in the same board

- Proper routing requires careful considerations of signal integrity

Acknowledgements

The authors would like to thank:

- Steve Iketani and Haris Basit/Averatek: line routing
- Tara Dunn and John Johnson /Averatek: PCB manufacturing costs
- Tomas Chester/Chester Electronic Design: DDR4 SODIMM re-design using SAP
- Eric Bogatin and students / CU Boulder: signal integrity

Please Look for This Upcoming Paper

E. Bogatin, C. Suresh, M. Piket-May, Univ. of Colorado,

and H. Basit, P. Dennig, Averatek Corp., "Utilizing Fine Line PCBs with High Density BGAs,"

Signal Integrity Journal [manuscript submitted for publication]. To be available:

https://www.signalintegrityjournal.com

References

[1] https://www.jedec.org/standards-documents/docs/dg-414h, see, for example, "DESIGN REQUIREMENTS FOR OUTLINES OF SOLID STATE AND RELATED PRODUCTS / Ball Grid Array Package and Interstitial Ball Grid Array Package," publication JEP95.

[2] Mike Vinson, "Semi-Additive PCB Processing: Process, Reliability Testing and Applications," IPC APEX 2021 conference paper, virtual.

[3] Tomas Chester, personal communications with Averatek, June – September 2021, www.chesterelectronicdesign.com, and tchester@chesterelectronicdesign.com.

[4] E. Bogatin, "New Electroless Process Promises Finer PCB Features," Signal Integrity Journal, May 11, 2021 [Online]. Available: www.signalintegrityjournal.com/blogs/4-eric-bogatin-signal-integrity-journal-technical-editor/post/2122-new-electroless-process-promises-finer-pcb-features .

[5] E. Bogatin, Signal and Power Integrity Simplified, 2nd ed., Boston, MA: Pearson Education, Inc./Prentice Hall, 2010.

[6] E. Bogatin, Bogatin's Practical Guide to Transmission Line Design and Characterization for Signal Integrity Applications, Boston, MA: Artech House, 2020.

[7] D. M. Pozar, Microwave Engineering, 4th ed., Hoboken, NJ: J. Wiley & Sons, Ch. 3, 2012.