CCC

Suppressing ICs with BGA packages and multiple DC rails

Keith Armstrong
CEng, EurIng, FIET, Senior MIEEE, ACGI

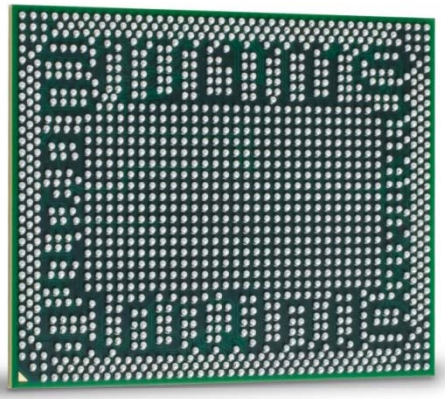
interference technology ITEM™

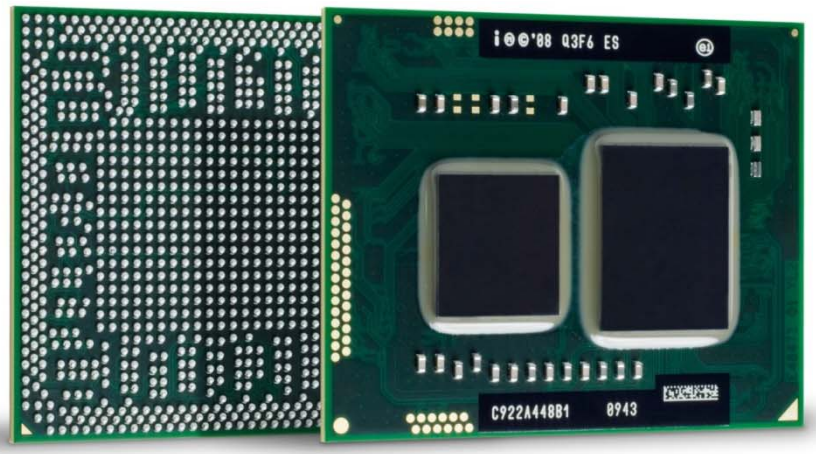
CHERRY CLOUGH CONSULTANTS LTD

Presenter Contact Info
email: keith.armstrong@cherryclough.com
website: www.cherryclough.com

CCC

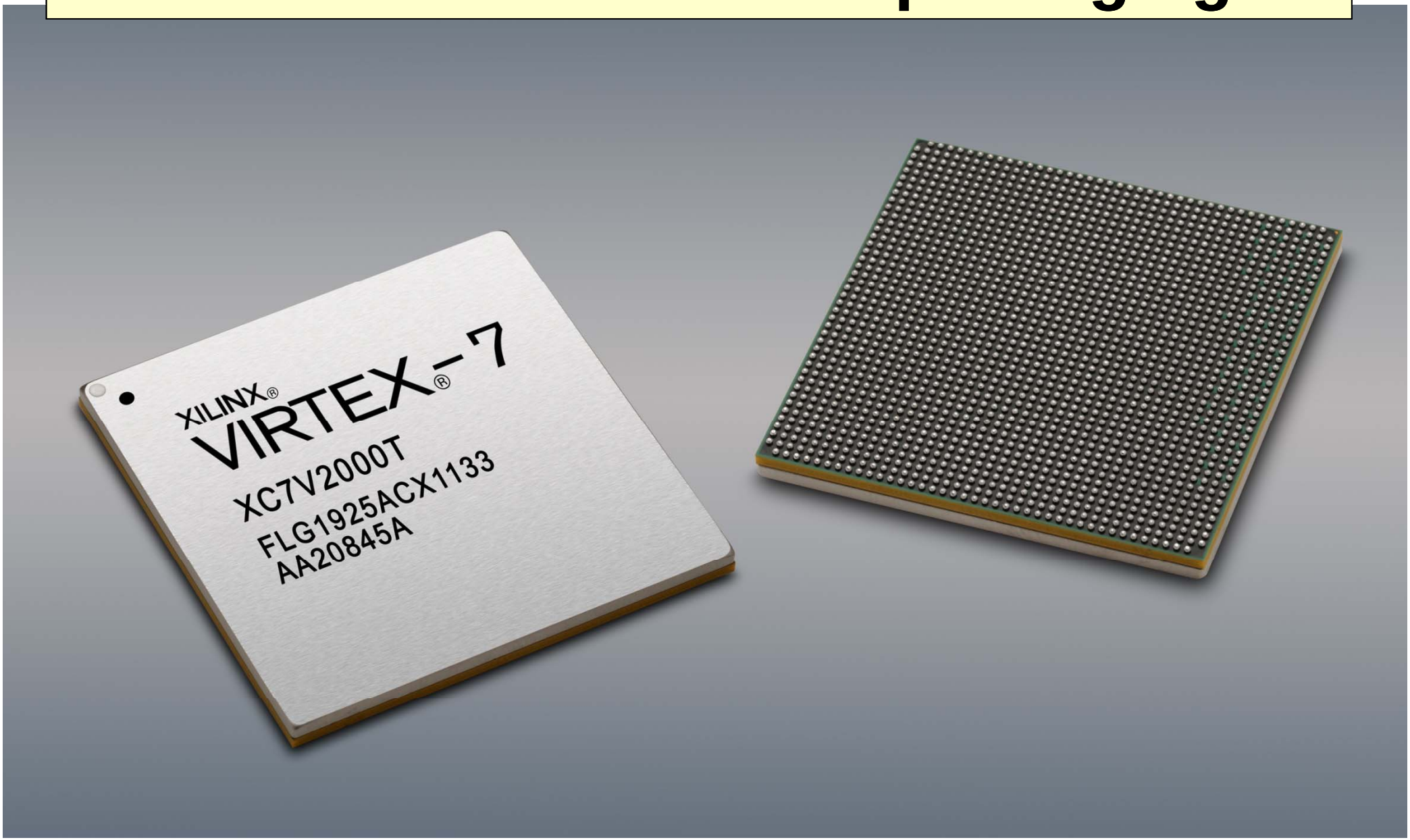
Some Intel Core™ *i5* BGA packages

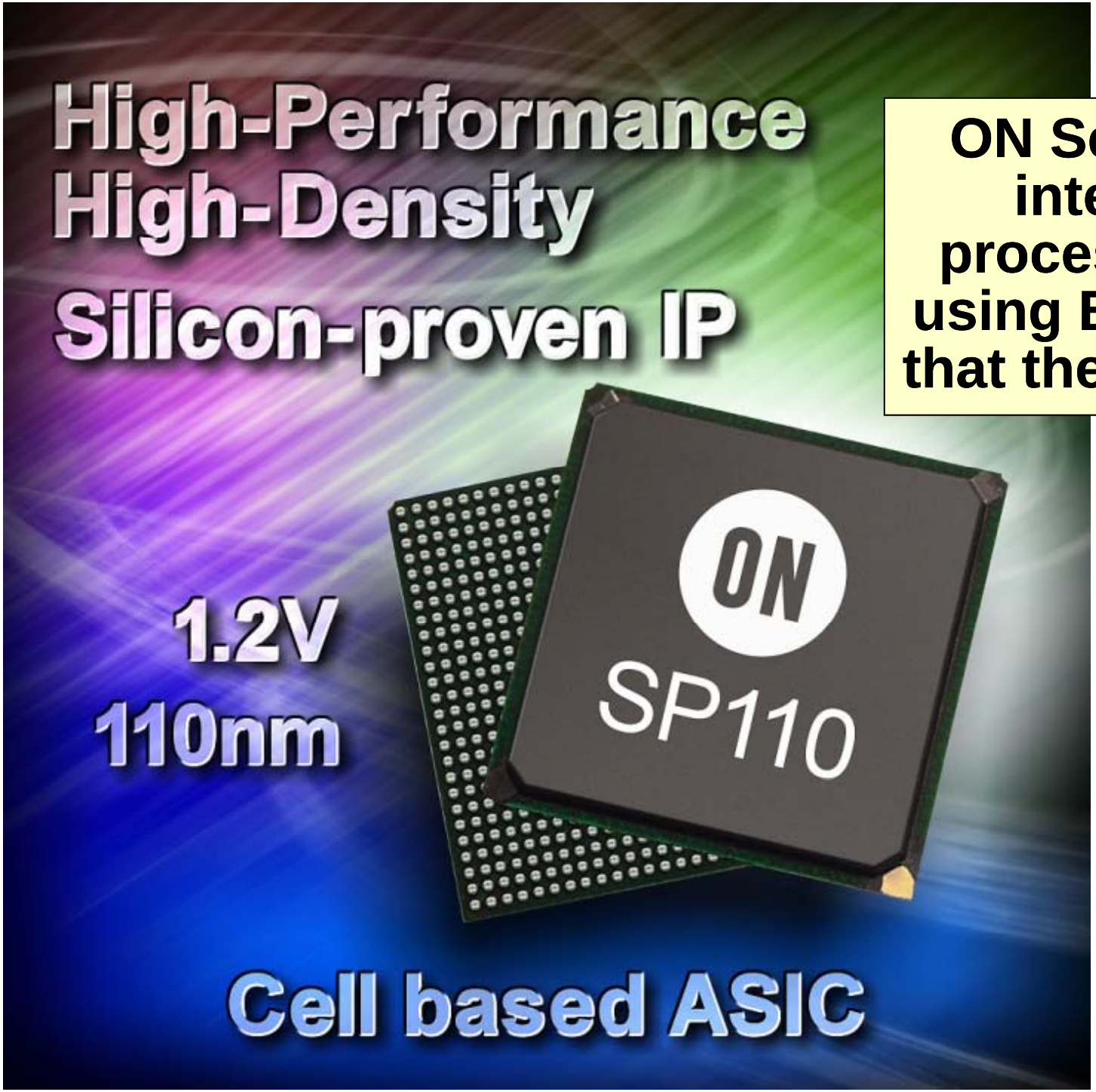




CCC

A Xilinx FPGA in BGA packaging





ON Semiconductor's internal 110nm process technology, using BGA packaging, that they use for ASICs

CCC

Creating good planes under BGAs

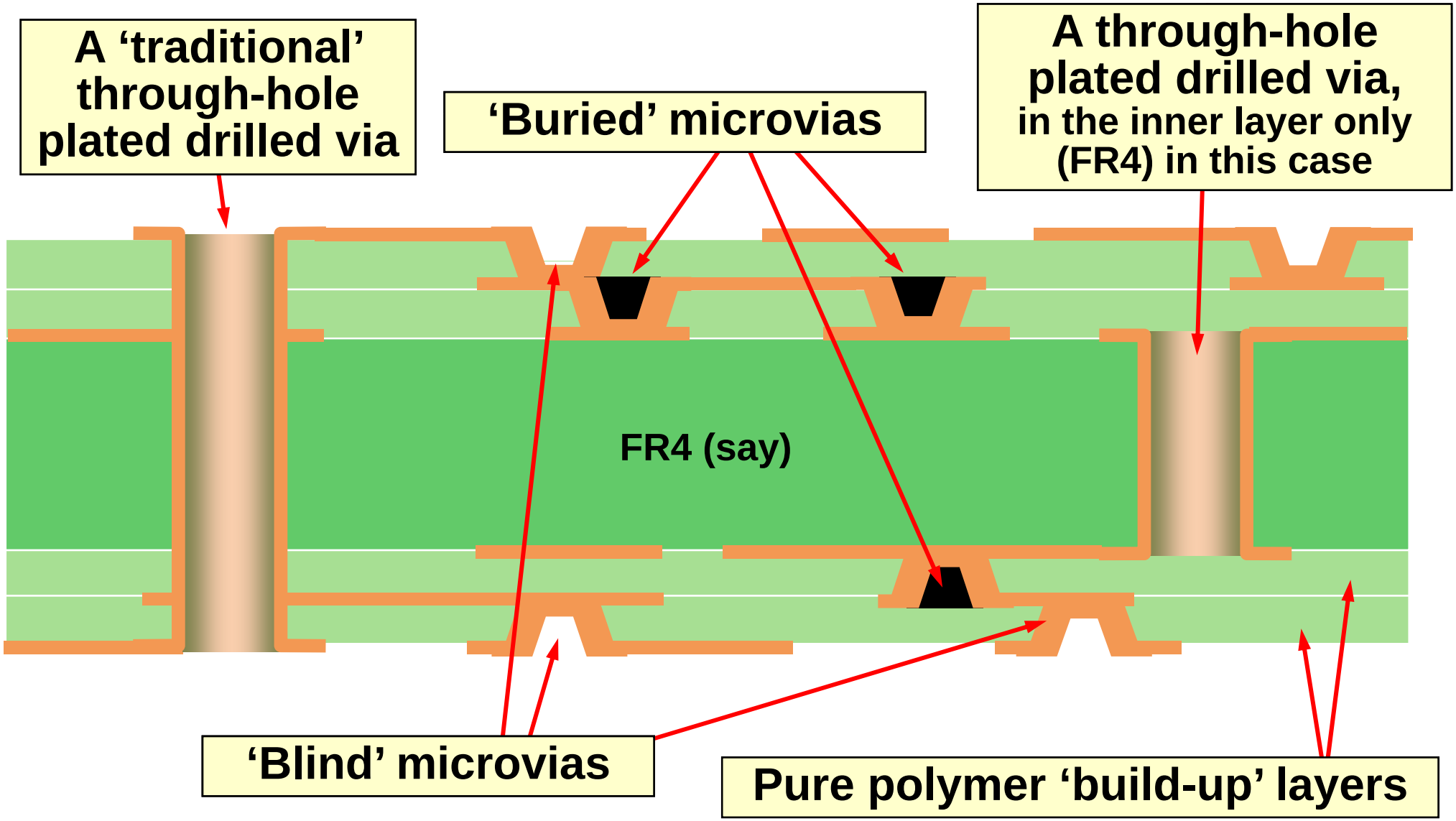
- For cost-effective SI, PI and EMC, it is important to have solid PCB planes *underneath the ICs…*
  - unfortunately, through-hole-plate (THP) PCB technology means an array of via holes under all BGAs…
  - which dramatically perforate the planes and significantly reduce their effectiveness for EMC…
    - and also harm signal integrity (SI) and power integrity (PI)
- Two solutions:
  - High Density Interconnect technology (HDI, = microvia)…
  - fine-line THP techniques, to create proper meshes

CCC

High Density Interconnect (HDI) PCB manufacturing technology

- **Also known as microvia technology, or Sequential Build-Up technology (or simply 'Build-Up')**
- **Based on 'microvias' of 0.15mm (6 thou) diameter or less, which are only as long as needed…**
  - **and don't steal solder during reflow, so allow *via-in-pad* layouts, very good for RF and EMC…**
  - **can achieve twice the number of pins/area than THP…**
  - **can significantly reduce the number of PCB layers *especially* where THP would require 10 or more layers**

Example of a 6-layer HDI board using '*original microvia technology*'



HDI benefits

- **HDI techniques help to make the smallest, lightest, and least power-hungry products…**
  - **and can be found in a wide variety of common products (including some toys)**
- **HDI makes it possible to use the smallest IC package styles, e.g…**
  - **Miniature or Micro BGA (especially with ball pitch <1mm)**
  - **DCA (direct chip attach)**
  - **Flip-chip**
  - **CSP (chip scale packaging)**
  - **TAB (tape automated bonding)**

CCC

HDI benefits continued...

- These small ICs, and the smaller PCBs they allow, are generally excellent for SI and EMC…
  - via-in-pad reduces decoupling inductance and pushes resonant frequencies higher…
  - shorter traces make less efficient ‘accidental antennas’…
  - smaller PCBs resonate at higher frequencies…
  - the much smaller sizes of the ICs and their close proximity to a solid 0V plane means they emit less…
  - shorter traces may not need to be transmission lines

HDI's planes aren't perforated

- So they have lower impedances…
  - hence lower emissions and better immunity...
  - and they have constant return path inductance…
    - for improved Z_0 control of transmission-lines...
  - and they achieve better shielding between the circuits on the top and bottom sides…
    - e.g. digital on top, analogue/RF on the bottom...
  - and they create solid, continuous planes under BGAs...
    - so help reduce the specs of filtering and shielding

HDI suppliers and technologies

- In May 2000 there were 62 manufacturers of HDI boards worldwide, and in May 2008 there were 32 manufacturers just in the UK…
  - their manufacturing techniques can vary, and may need different layout techniques, so always check with chosen manufacturer *before* starting board layout
- Basic standard: IPC-2315 (from www.ipc.org) …
  - but HDI requires a different approach to PCB layout…
  - and – depending on the supplier – some PCB EMC techniques might not be able to be used

CCC

People seem to think that HDI is costly, *but it should cost less than THP!*

- An IPC survey in 2000 found HDI boards could be purchased for the same cost as THP…
  - and not using buried vias helps reduce costs further
- Latest advice (Mentor Graphics) is that boards needing > 8-10 layers should cost less if made in HDI…
  - e.g. a high-density 18 layer THP would only need 10 layers if made using HDI…
  - but even lower densities and with fewer layers, the EMC (and SI) advantages of HDI make it more cost-effective than THP

Modern HDI PCB technology

– has many advantages over ‘original’ HDI…

– and is is the topic of my next webinar, on September 17

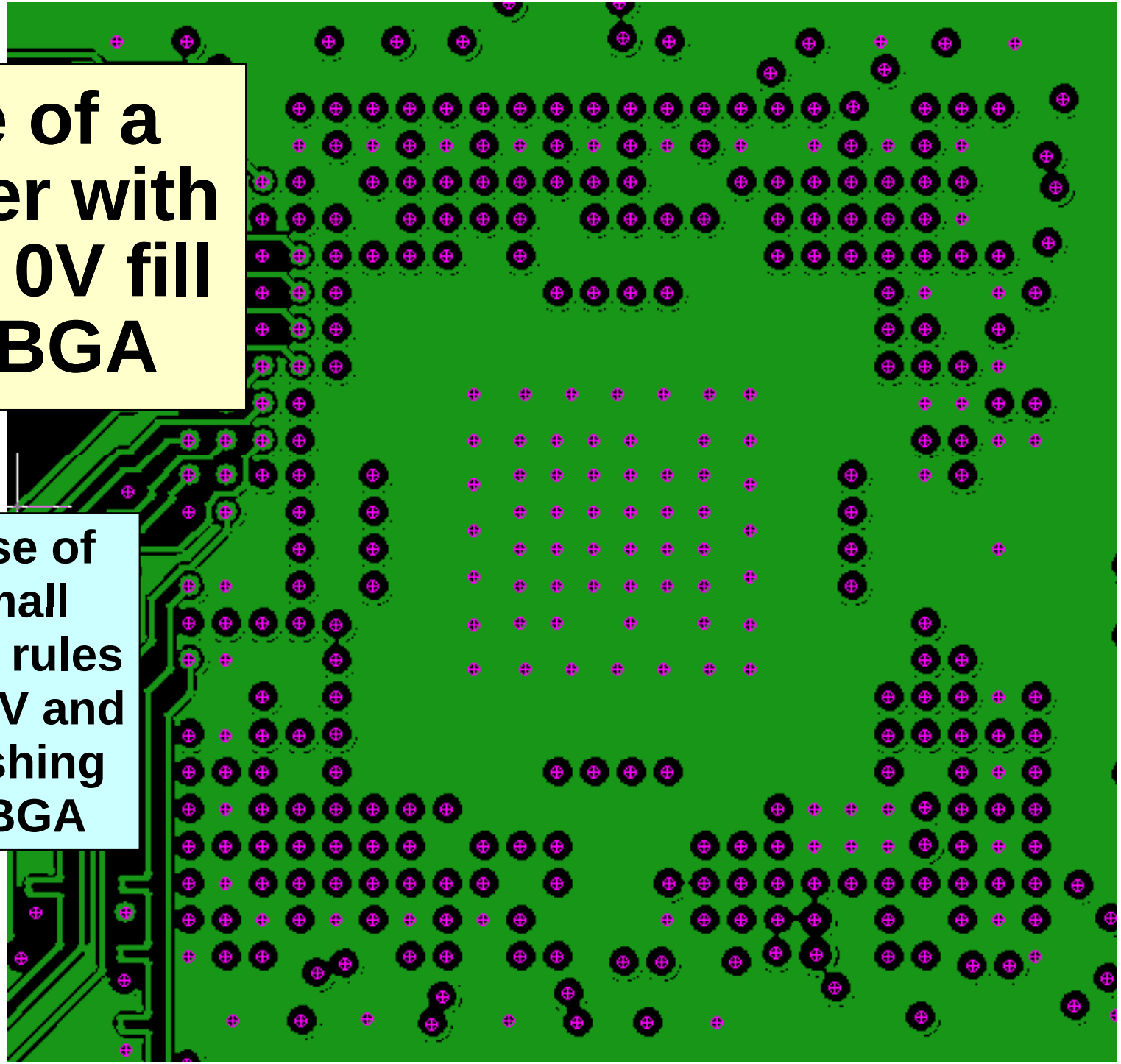
Fine-line THP techniques

- We must ***at least*** achieve a continuous mesh (grid) of 0V and power traces under every BGA...
  - to connect all their 0V and PWR power pins to their respective decaps...
  - and to the 0V and PWR planes on the rest of the board…
  - and to help control emissions
- It won't be half as good for EMC as a solid plane (e.g. using HDI)…
  - but it will be the best we can do in THP

emc13bga

Example of a signal layer with a meshed 0V fill under a BGA

Showing the use of adequately small track-and-space rules to create good 0V and PWR plane meshing underneath a BGA

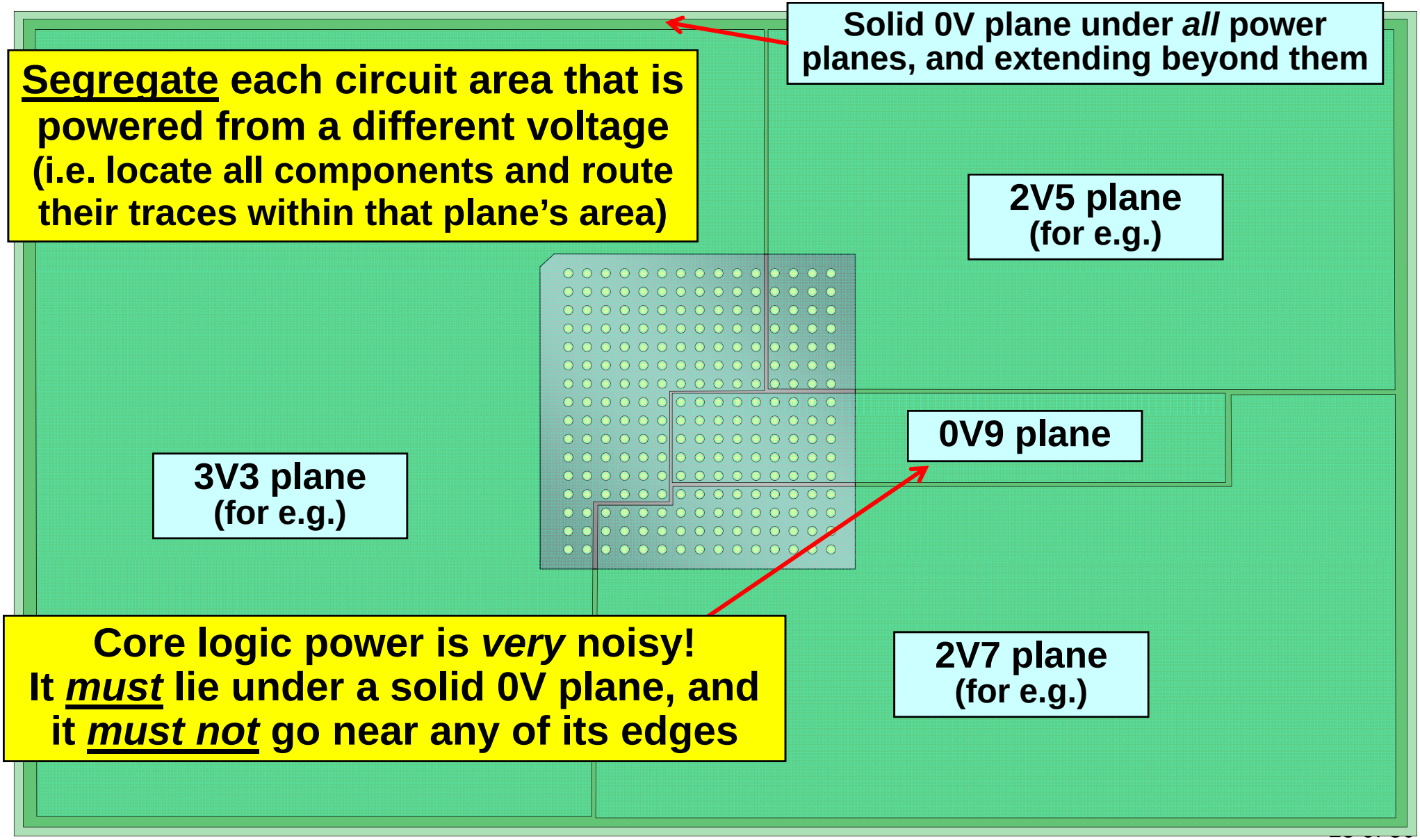


Planes under BGA devices continued…

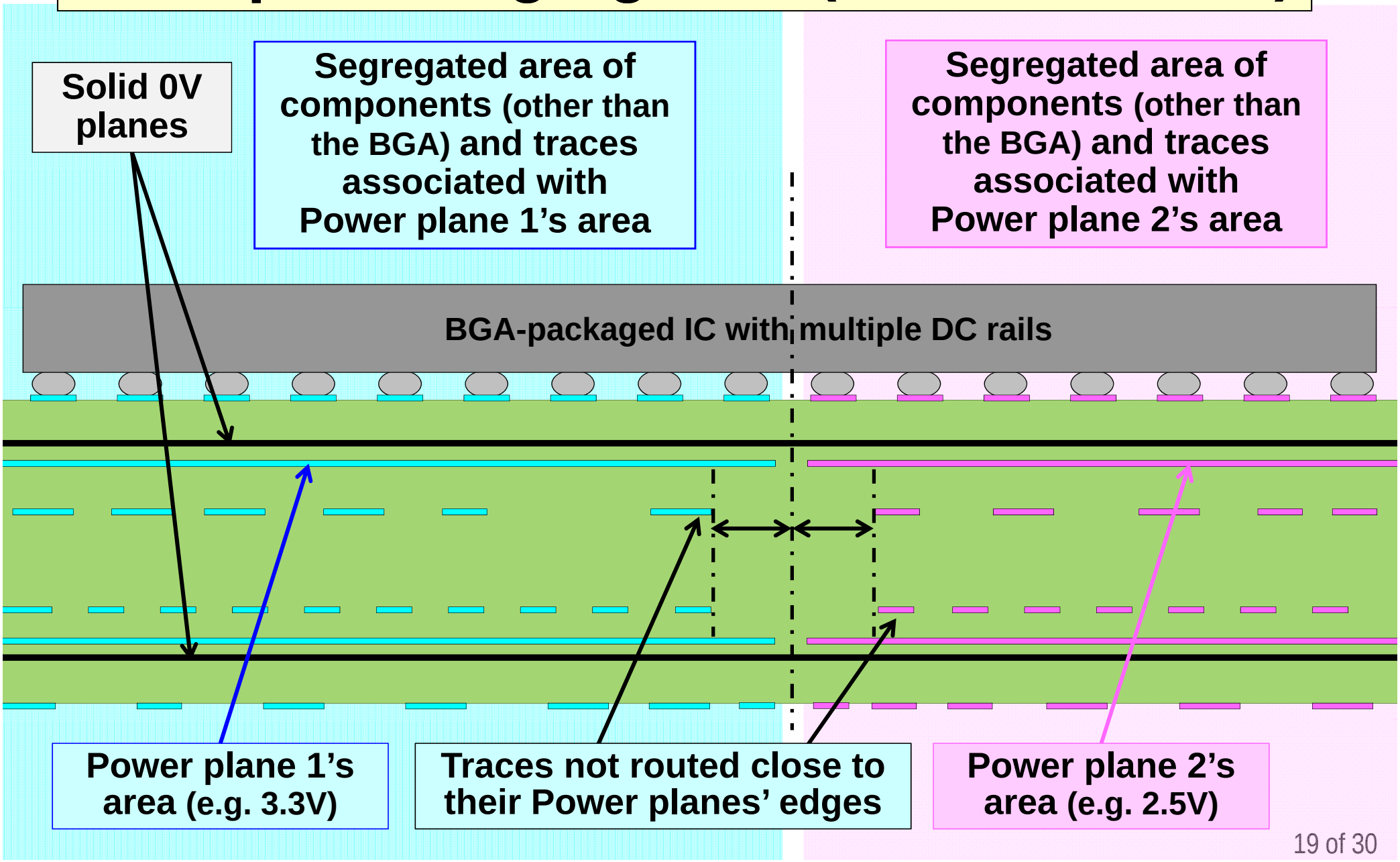
- **BGAs with ball pitch ≥ 1 mm can create continuous meshes underneath…**
  - **using 175µm (7 thou) track-and-space (track-and-gap)**
- **BGAs with pitches 0.8 - 1.0 mm need 100µm (4 thou) track-and-space to create a mesh…**
  - ***NOTE:* for several years now, 100µm offshore volume-manufacture now costs no more than 175µm**
- **And BGA pitches 0.5 - 0.8 mm need 50µm (2 thou)…**
  - **some UK PCB manufacturers can do this (e.g. Merlin Circuit Technology)**

Poll Questions

Where an IC has multiple DC rails
try to get all of its power planes on one board layer



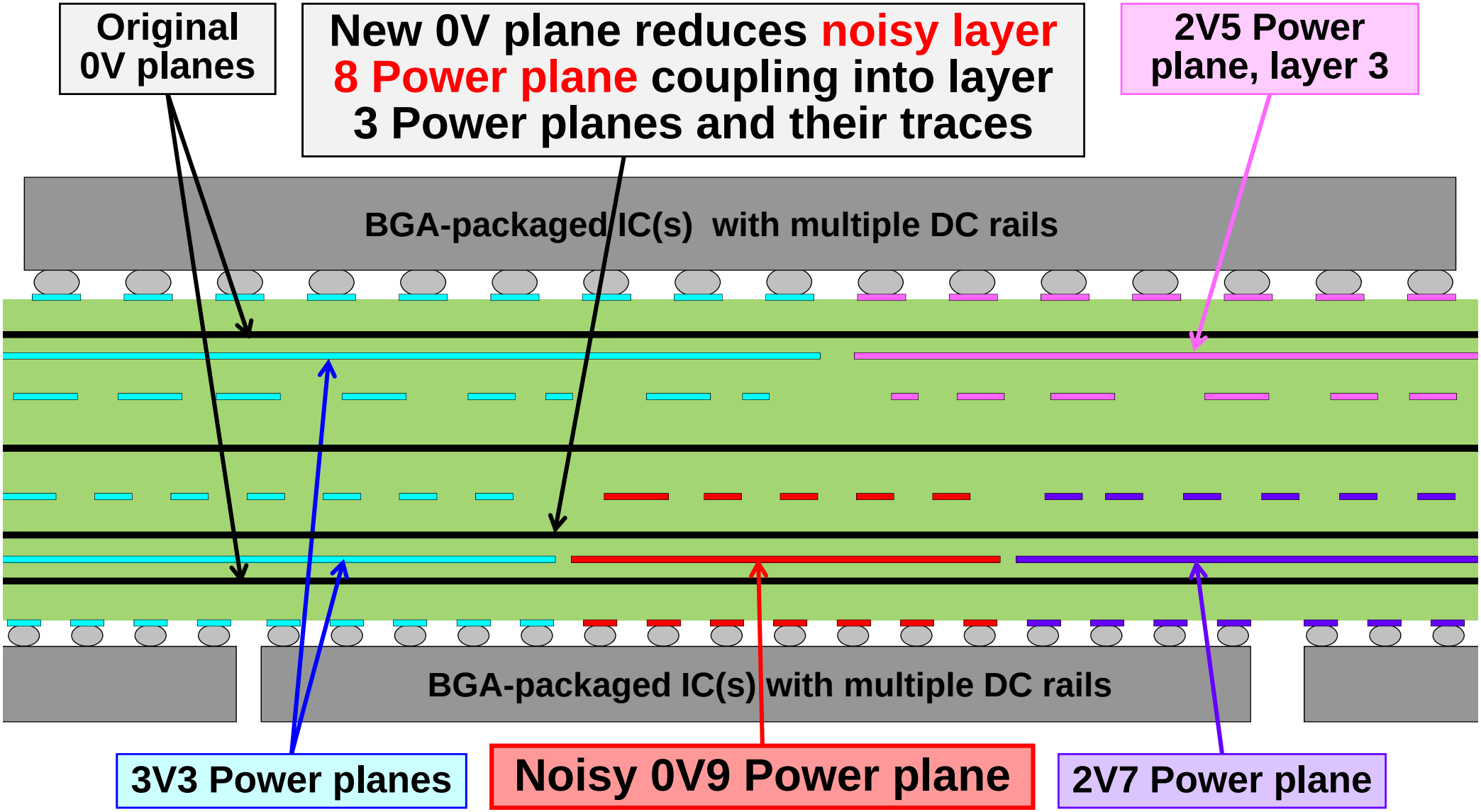
Example of segregation (in cross-section)



Reducing the crosstalk between PWR planes on parallel layers

- To help prevent RF noise in a ‘noisy’ plane…
  - e.g. processor core logic supply, typically 0.9 – 1.2V
  - from coupling its noise into parallel power planes and spreading more widely around a board…
    - increasing emissions…
  - place a new 0V plane between them in the stack-up

Example of adding a 0V plane to reduce noise coupling between Power plane layers

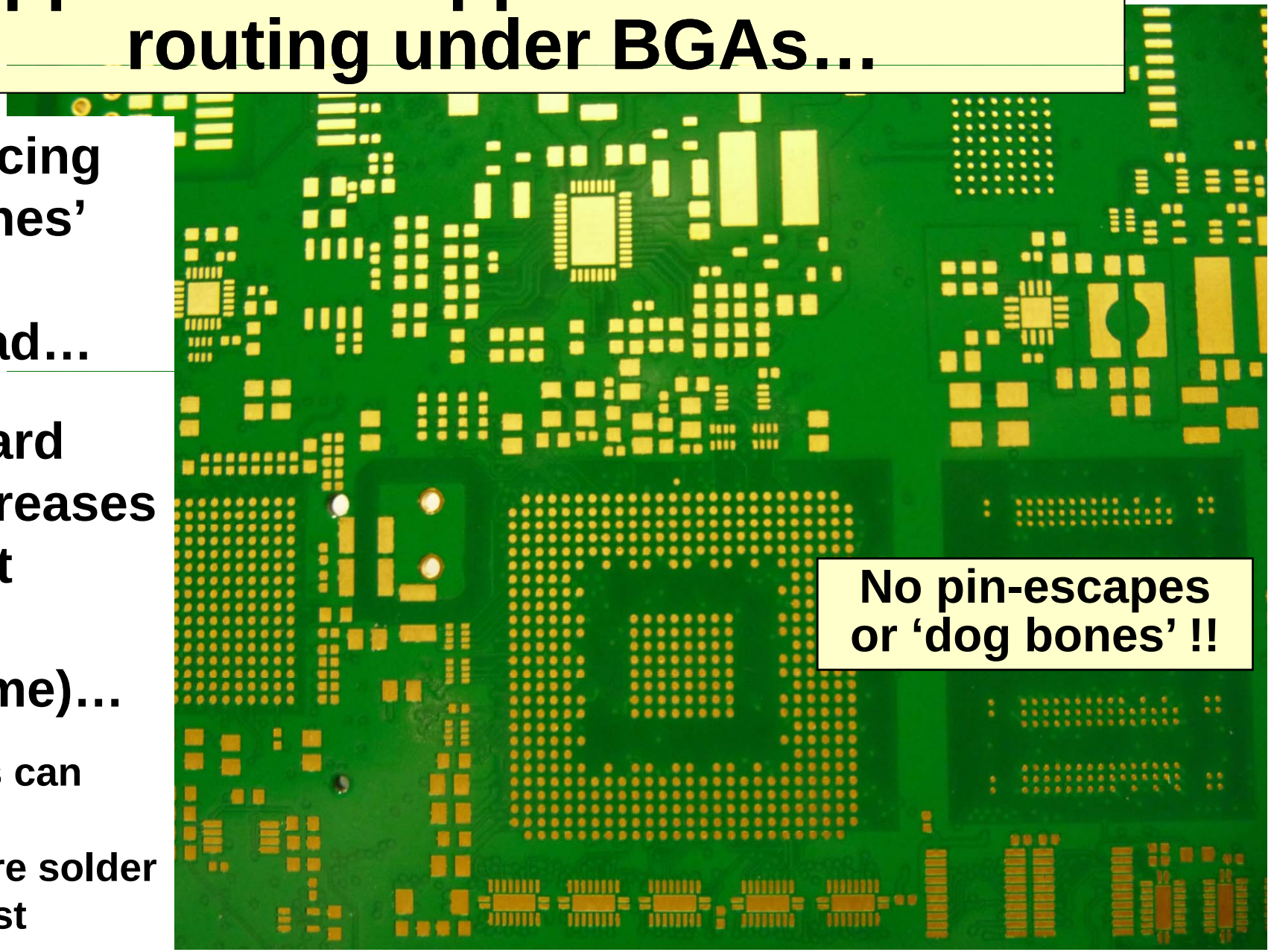


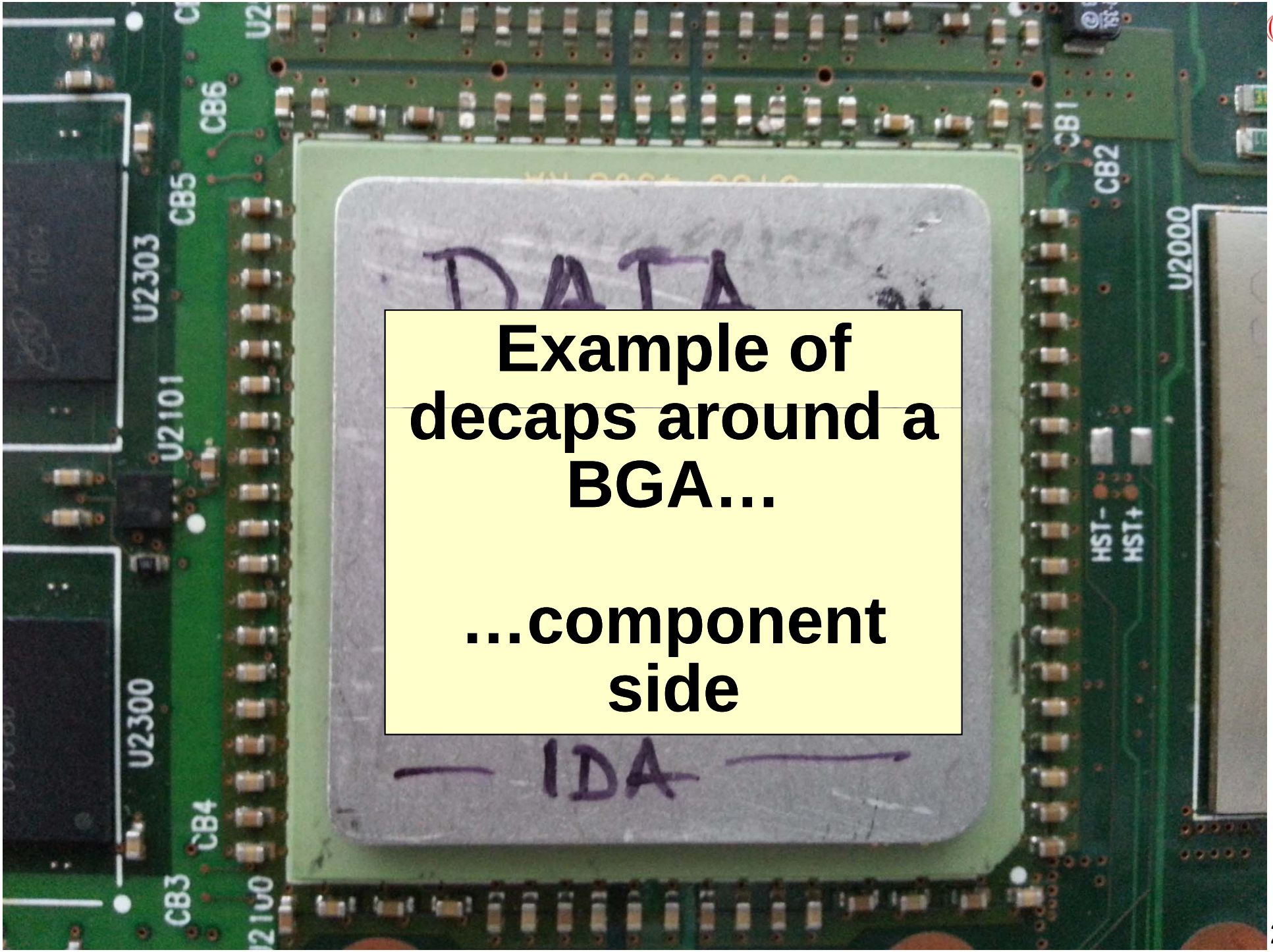
Reducing crosstalk between PWR planes on parallel layers continued…

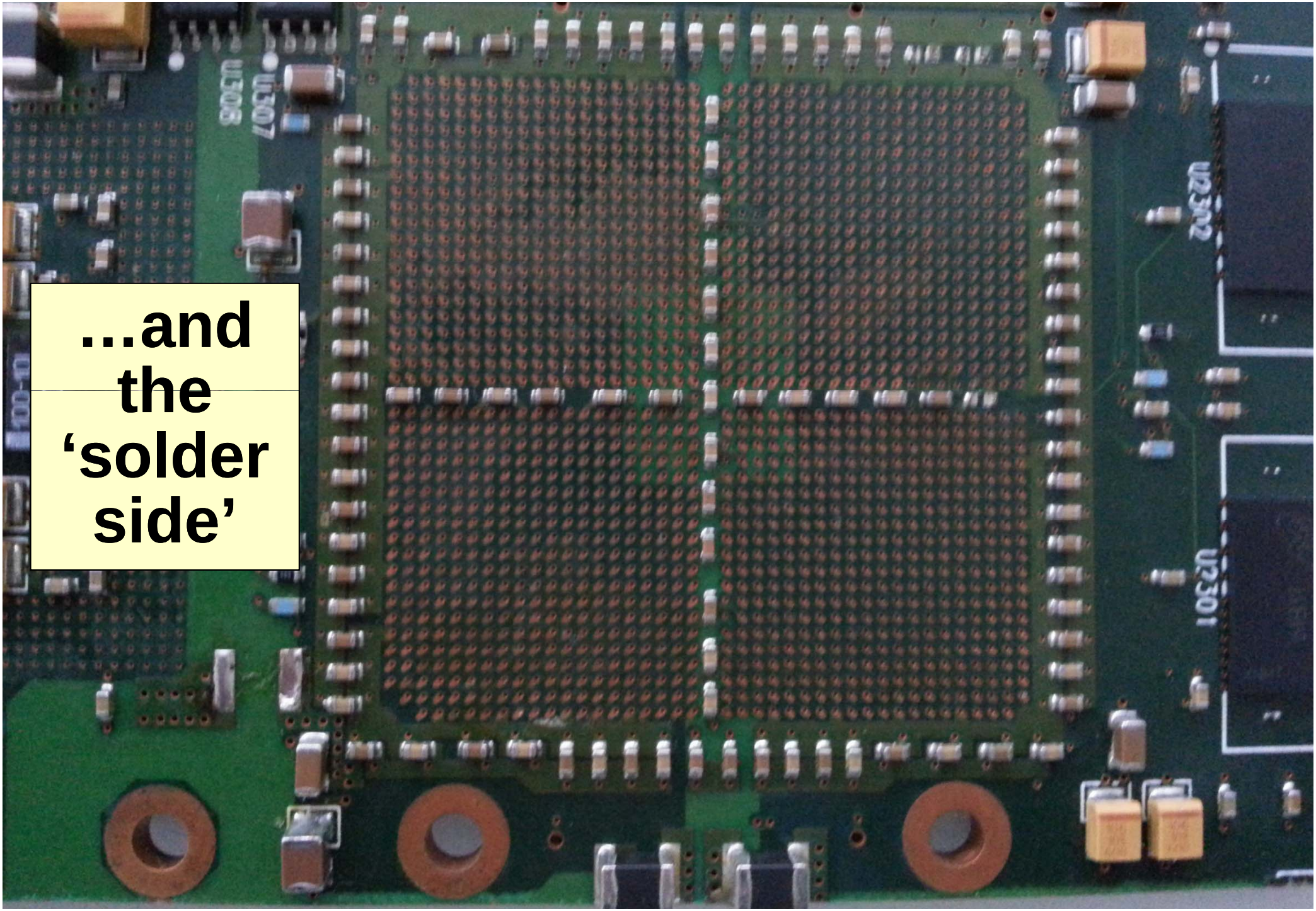
- ‘Sandwiching’ a power plane layer between two solid 0V planes as in the previous graphic…
  - also means no trace routing has to worry about crossing any splits between different power plane areas…
  - and can double the power planes’ buried decoupling capacitance…
    - which helps suppress emissions at >300MHz

Copper-filled/capped vias can ease routing under BGAs…

- by replacing ‘dog bones’ with via-in-pad…
- bare board cost increases by about +10% (in volume)…
- prototypes can use high-temperature solder to save cost




Example of decaps around a BGA…
…component side



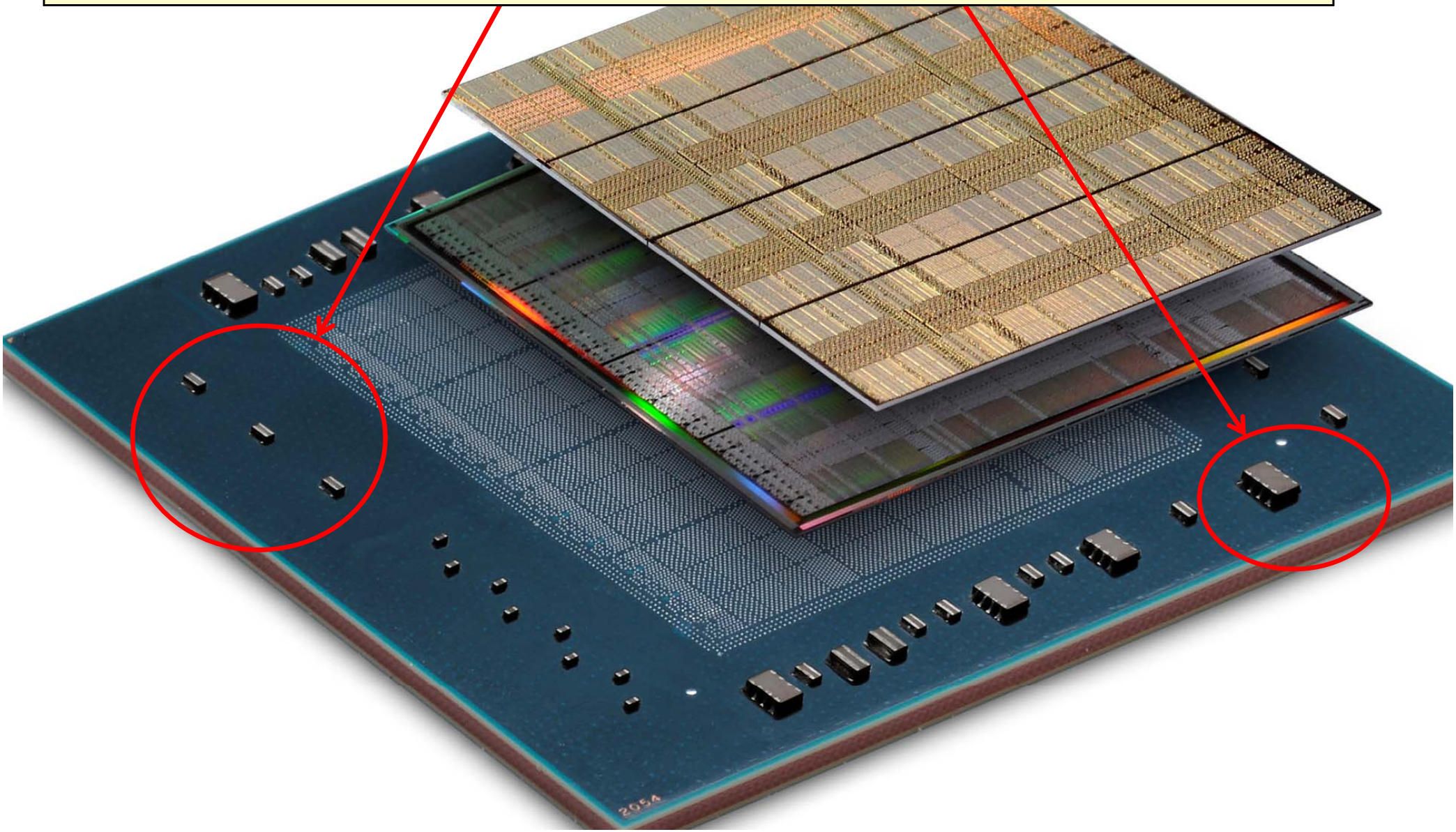
…and the ‘solder side’

Special capacitors with reduced ESL can improve Power Integrity, e.g…

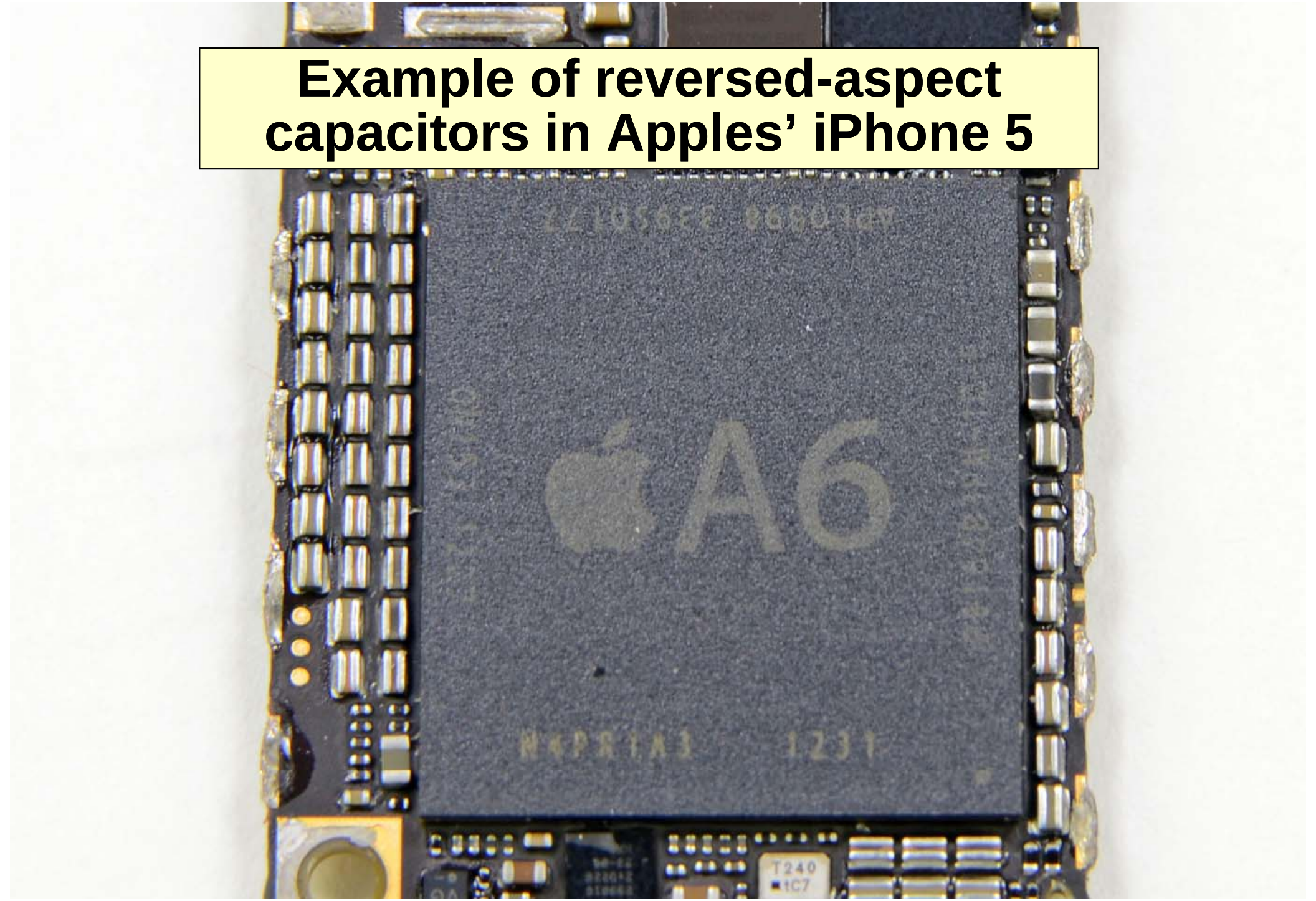
- ‘reversed-aspect’, e.g. 0204, 0508, Murata LLL, etc…
- ‘interdigitated’, e.g. AVX IDC, LICA; Murata LLA, LLM…
- balanced X2Y® capacitors used as decouplers…
- lossy capacitors dampen resonances, e.g. Murata LLR
- buried capacitors, e.g. Murata GRU…
- distributed ‘embedded’ capacitance using proprietary board laminates for one or more pairs of PWR/GND planes, e.g…
  - Faradflex (Oak Mitsui); ECM (3M); Interra™ HK 04J (Dupont); EmCap® (Sanmina), and others

Exploded view of the Xilinx Virtex-7 2000T

Notice the reversed-aspect and interdigitated decaps?



Example of reversed-aspect capacitors in Apples' iPhone 5



CCC

Suppressing ICs with BGA packages and multiple DC rails

the end

interference technology ITEM™

CHERRY CLOUGH CONSULTANTS LTD

Presenter Contact Info
email: keith.armstrong@cherryclough.com
website: www.cherryclough.com

Poll Questions