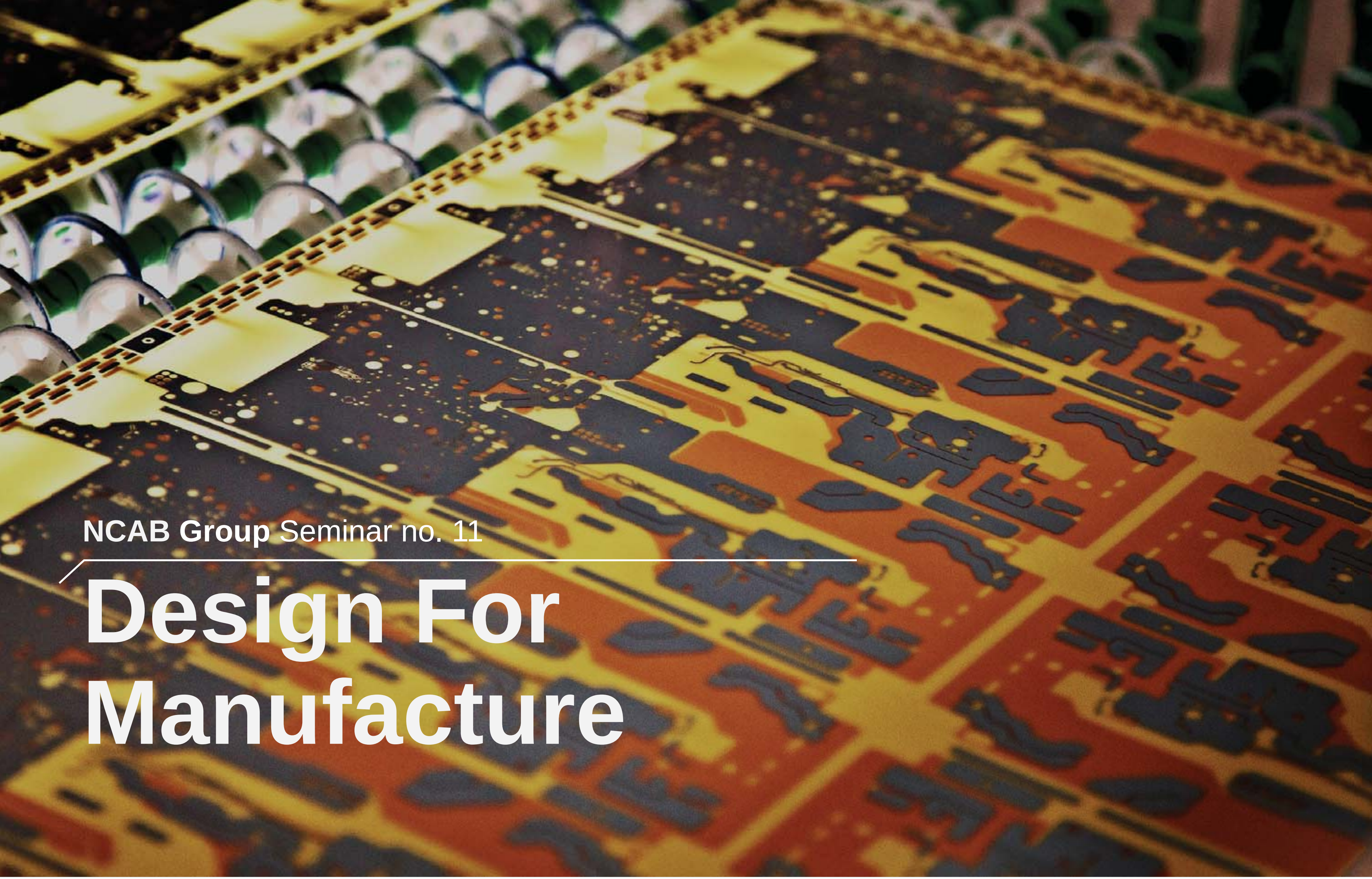


NCAB Group Seminar no. 11

Design For Manufacture

Design for manufacture (DFM)

What areas does DFM give consideration to?

- Common errors in the documentation
- Good design
- Required tolerances

Design for manufacture (DFM)

These areas include, but are not limited to, the following points:

1. Procurement documentation problems
2. Data generation problems
3. Solder mask openings / bridges
4. Annular ring / clearance
5. Copper balance / design
6. Copper thickness/ track width
7. Drill diameter / aspect ratio
8. Material yield

1

PROCUREMENT DOCUMENTATION PROBLEMS

'Misinformation'

The **most common errors** that occur relate to missing/ambiguous/incomprehensible/conflicting information.

Our experience shows that this **occurs in about 30%** of all the new articles which are handled by NCAB Group.

This leads to engineering questions (EQ's) being raised that sometimes take time to clarify and can actually **affect delivery dates**.

These EQ's <u>have</u> to be asked
or the PCB can be "perfectly wrong"

1

PROCUREMENT DOCUMENTATION PROBLEMS

Examples of missing information

- Outline information / data / drawings
- Plated or non plated hole identification
- Surface finish within the data package
- Copper thickness
- Material details
- Color of solder mask/legend print
- Thickness of finished board
- Missing Gerber or drill files
- Etc

1

PROCUREMENT DOCUMENTATION PROBLEMS

Examples of ambiguous / incomprehensible / conflicting information

- The given board thickness does not match the specified build.
- Legend print is included in the documentation but shall not be printed.
- Dimensions listed on the drawing do not match with the Gerber outline.
- Number of holes in drill drawing does not match with the number of holes detailed in the supplied drill file.
- The hole sizes in drill drawing does not match the sizes in drill file.
- Copper thickness in specification is different to the stated build.
- Specified impedance requirements cannot be achieved based upon the stated build.

1

PROCUREMENT DOCUMENTATION PROBLEMS

Examples of ‘impossible’ information

References are made to all kinds of available standards in the world without any explanation to support:

- The PCB shall fulfill relevant parts from IEC-60255
- Press-fit shall be according to IEC 60352-5
- The PWB shall fulfill the safety standard EN-50178

! The specification from the customer contains 52 pages, and ‘hidden’ on page 43 it says that **“if there are 1.3mm holes within the data, then they should in fact be produced as 1.4mm with a tolerance of +0.05/-0.10mm”.**

1

PROCUREMENT DOCUMENTATION PROBLEMS

Conclusions

As far as is possible, send information that is **relevant** for the PCB manufacturing.

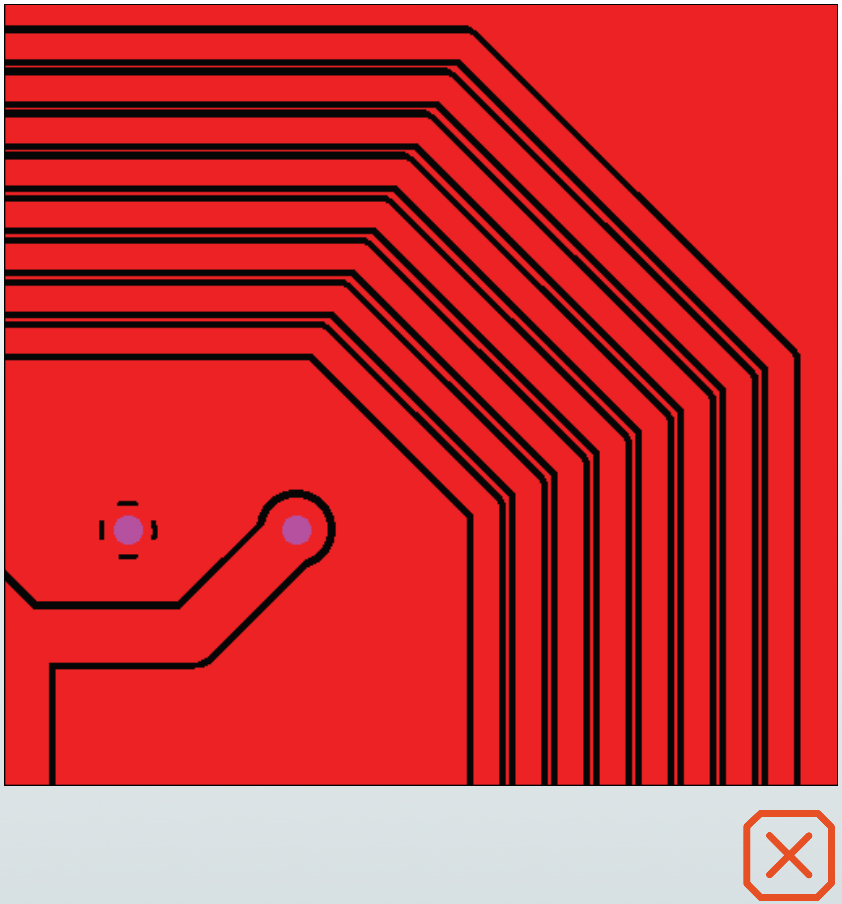
Avoid **too much** information as this almost always leads to some sort of “double information”.

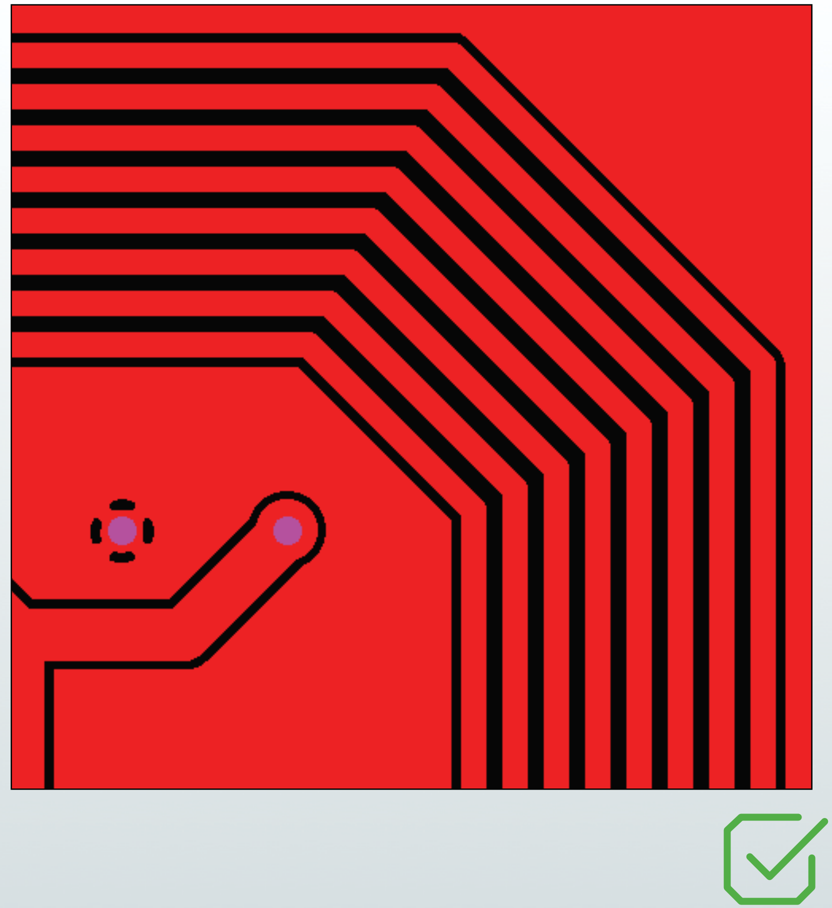
Always refer to **internationally recognised** specifications (IPC). Otherwise, the specific demand must be extracted and provided in detail.

2

DATA GENERATION PROBLEMS

Copper slivers

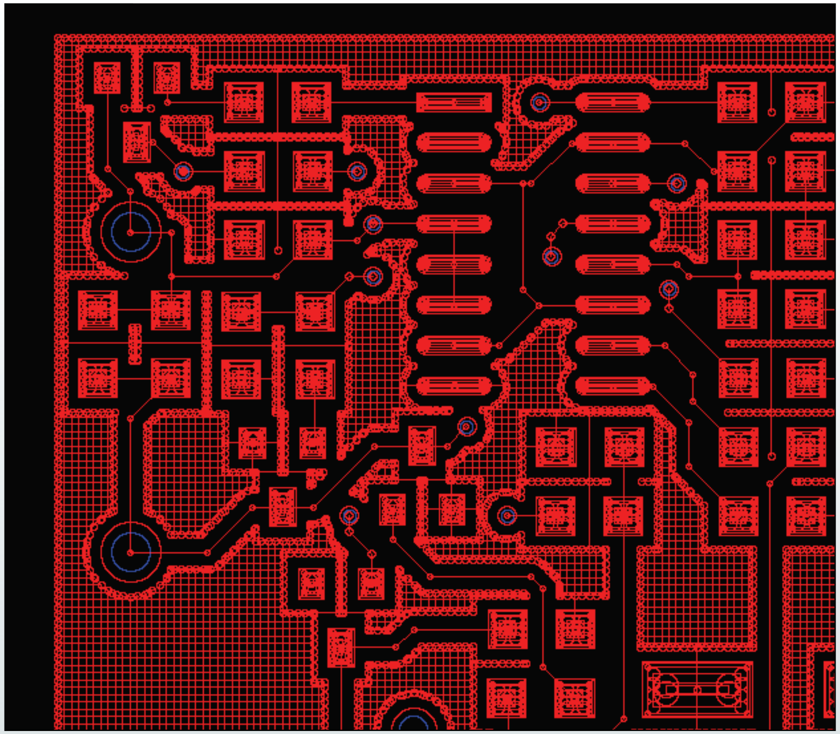


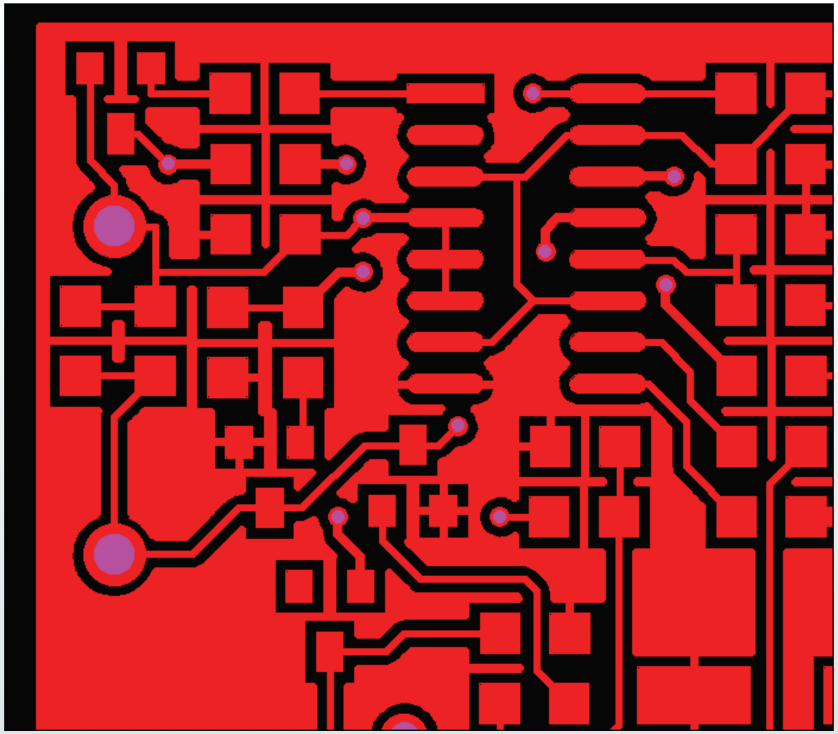


2

DATA GENERATION PROBLEMS

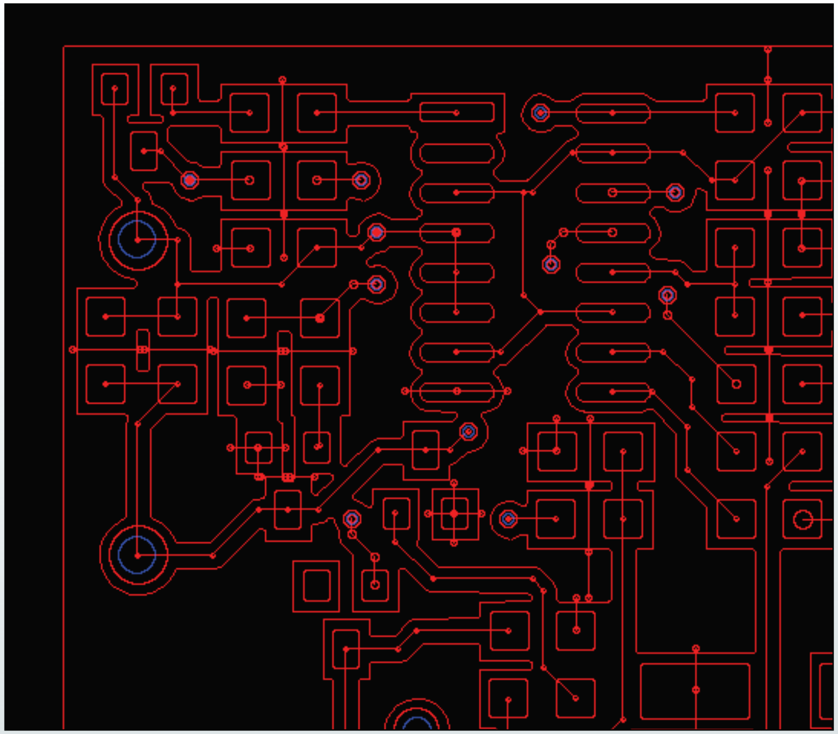
Unflashed pads & drawn surfaces

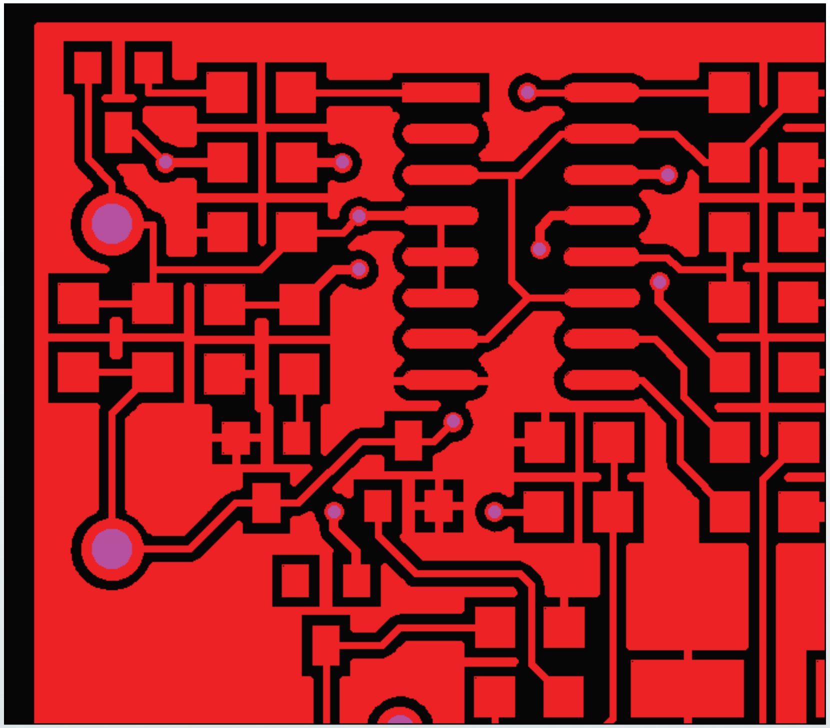




2 DATA GENERATION PROBLEMS

Flashed Pads & Surface

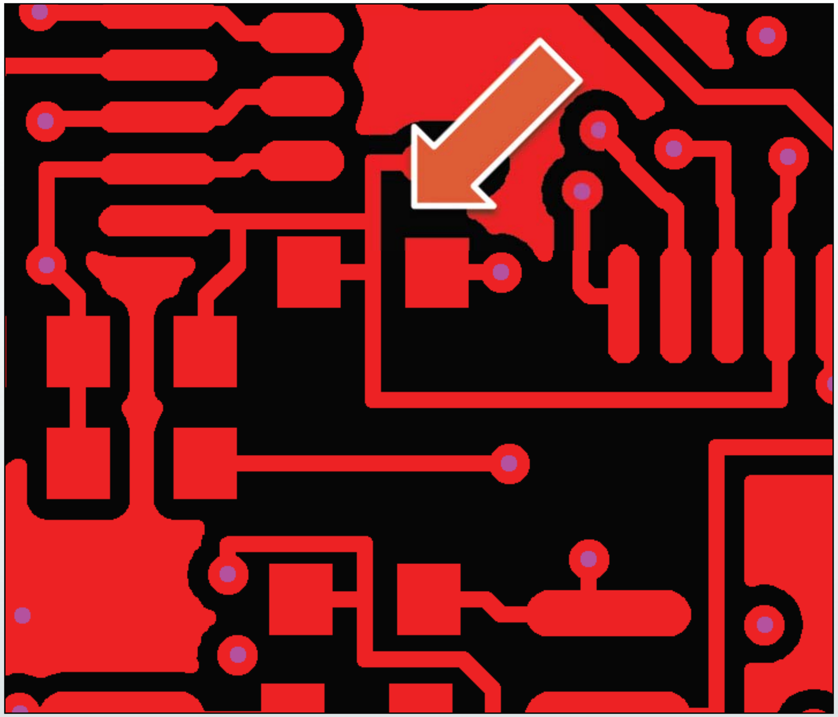


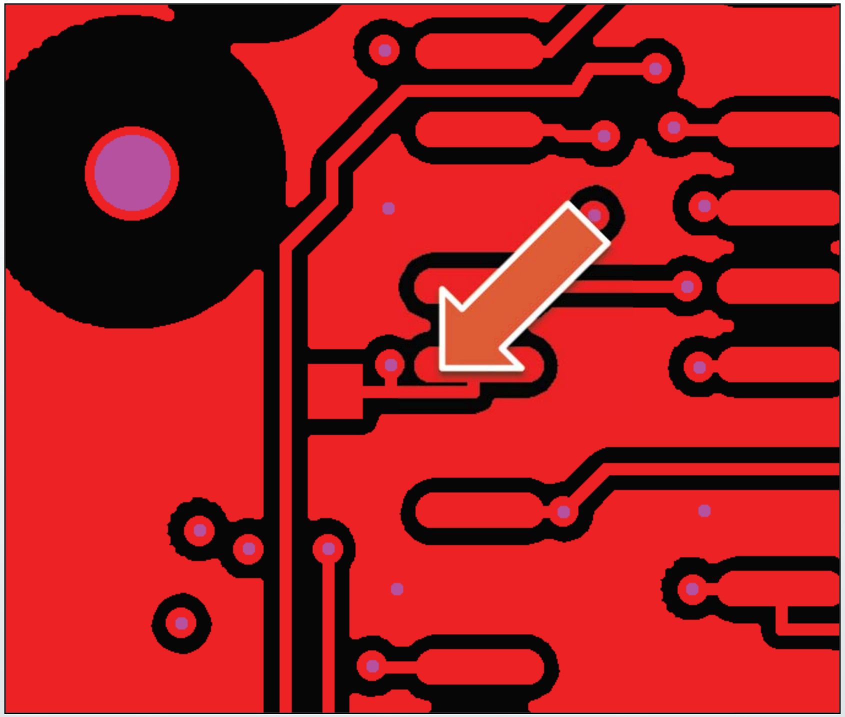


2

DATA GENERATION PROBLEMS

Same net spacing – examples

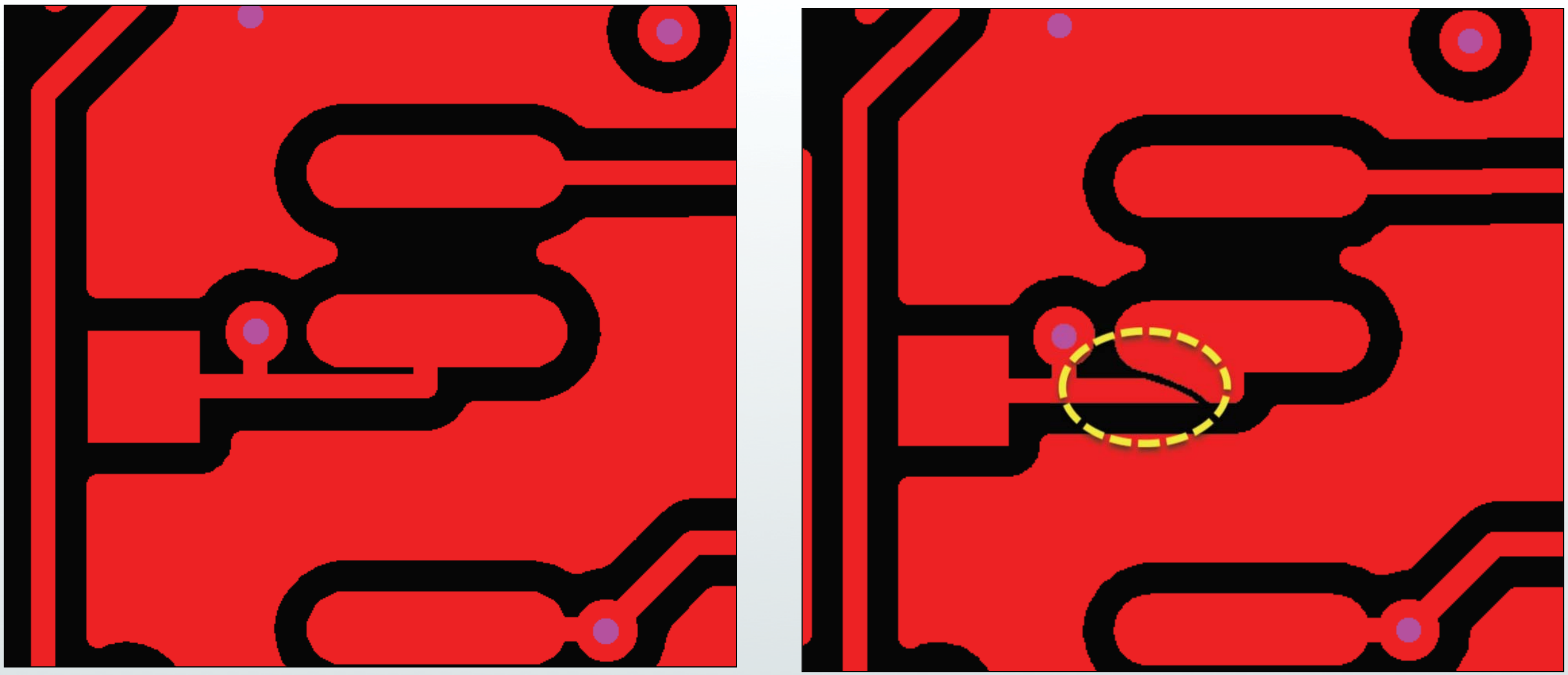




2

DATA GENERATION PROBLEMS

Same net spacing – what's the risk?



In this example…. the risk of open circuits.

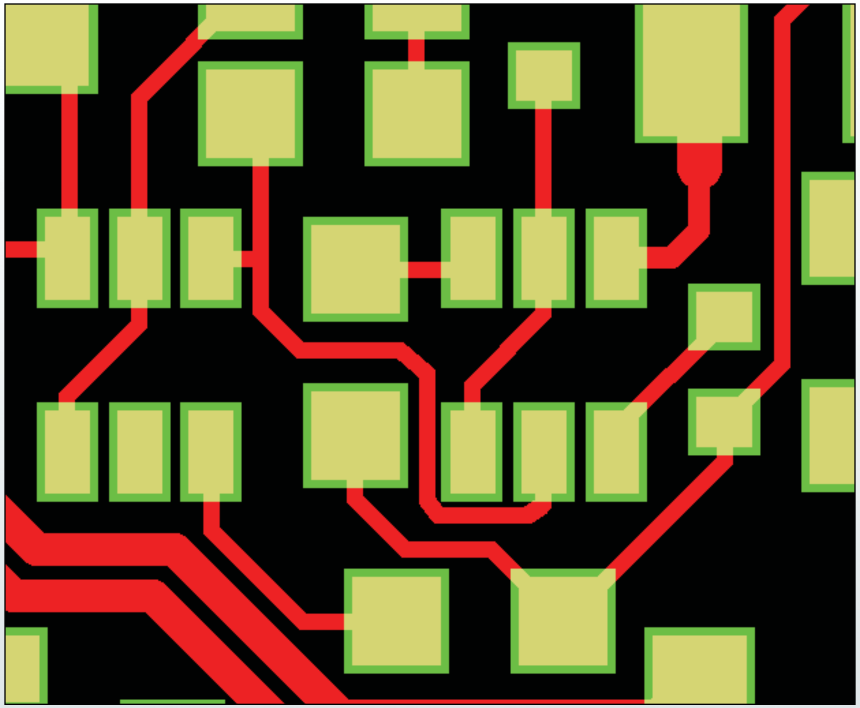
3

SOLDERMASK OPENINGS / BRIDGES

Soldermask enlargement

Example of a soldermask enlargement of

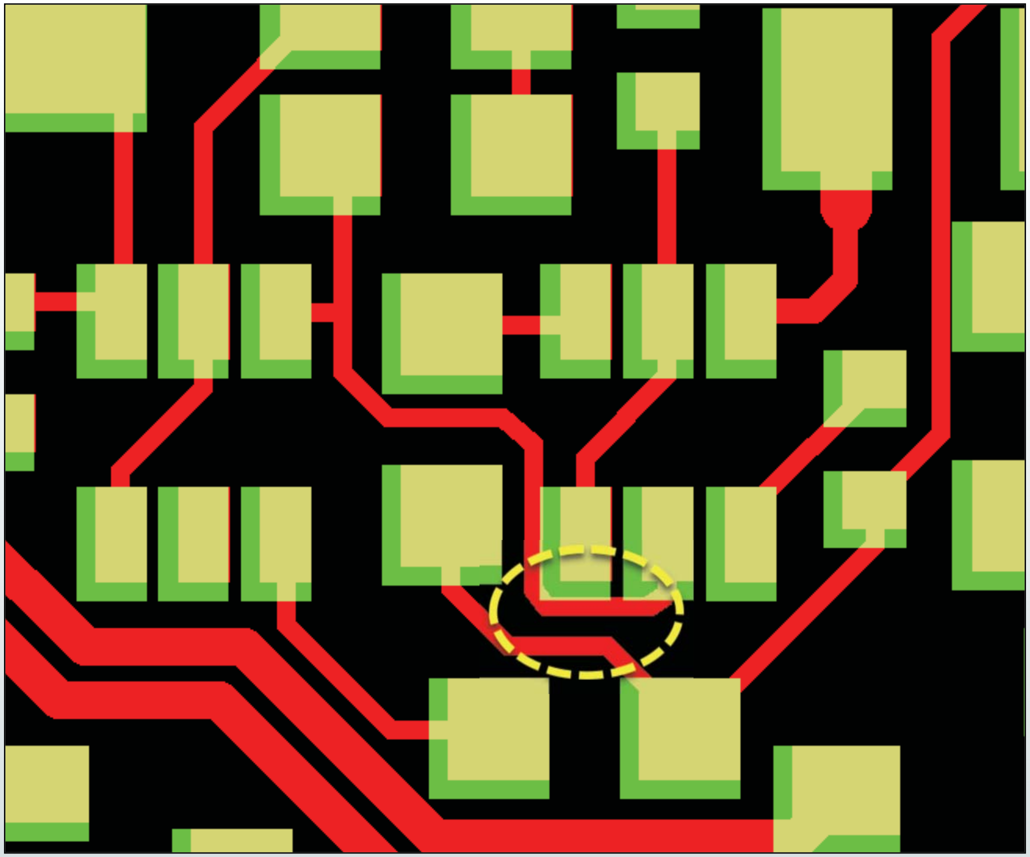
0.075mm (3mil)



3

SOLDERMASK OPENINGS / BRIDGES

Soldermask with maximum displacement



3

SOLDERMASK OPENINGS / BRIDGES

Recommendation

General

A = 160µm

B = 230µm

C = 65µm

D = 100µm

Moderate

A = 125µm

B = 200µm

C = 50µm

D = 100µm

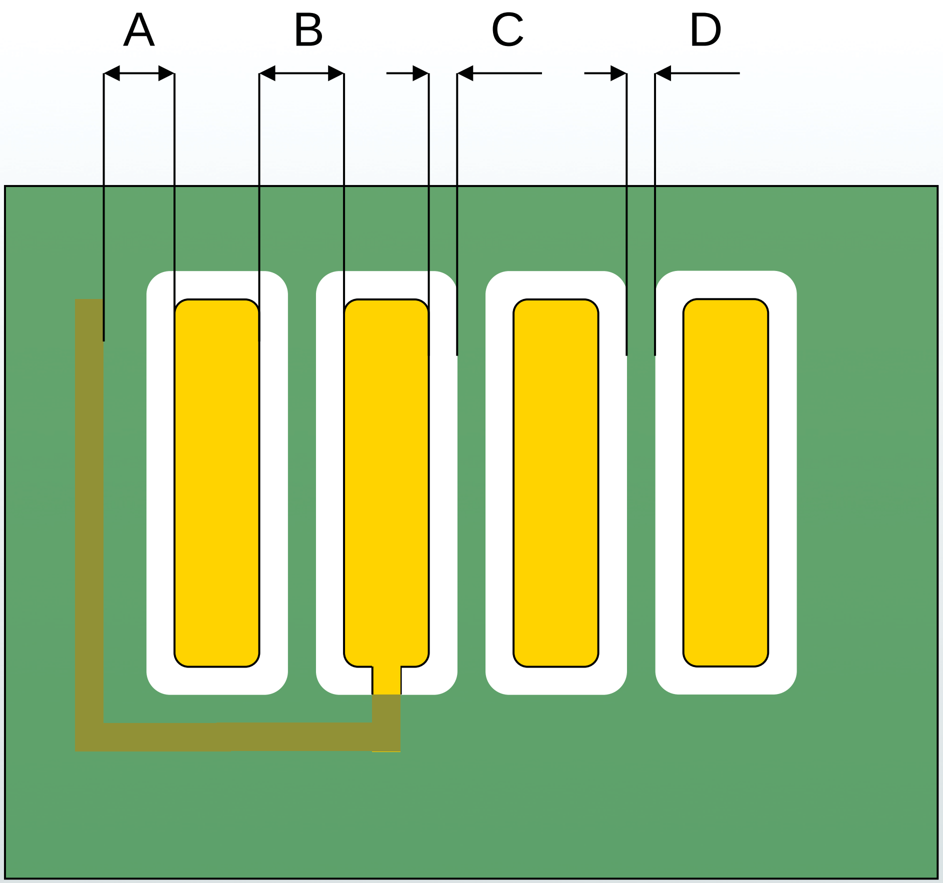
Advanced

A = 100µm

B = 150µm

C = 37µm

D = 80µm

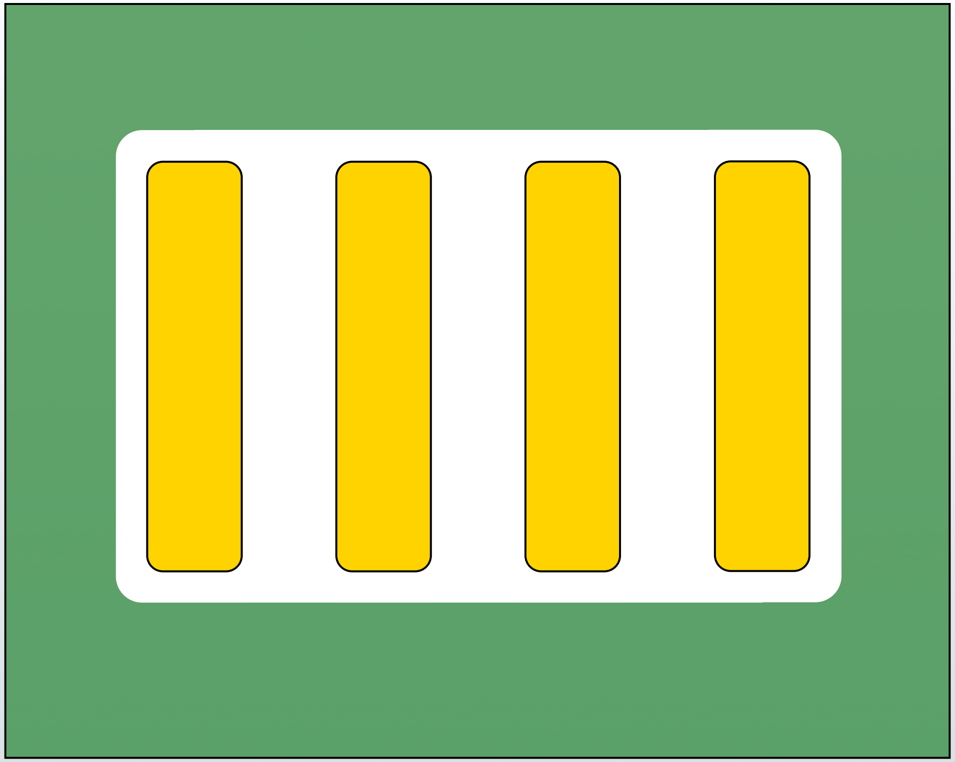


Note: Cu thickness ≤ 35um.

3

SOLDERMASK OPENINGS / BRIDGES

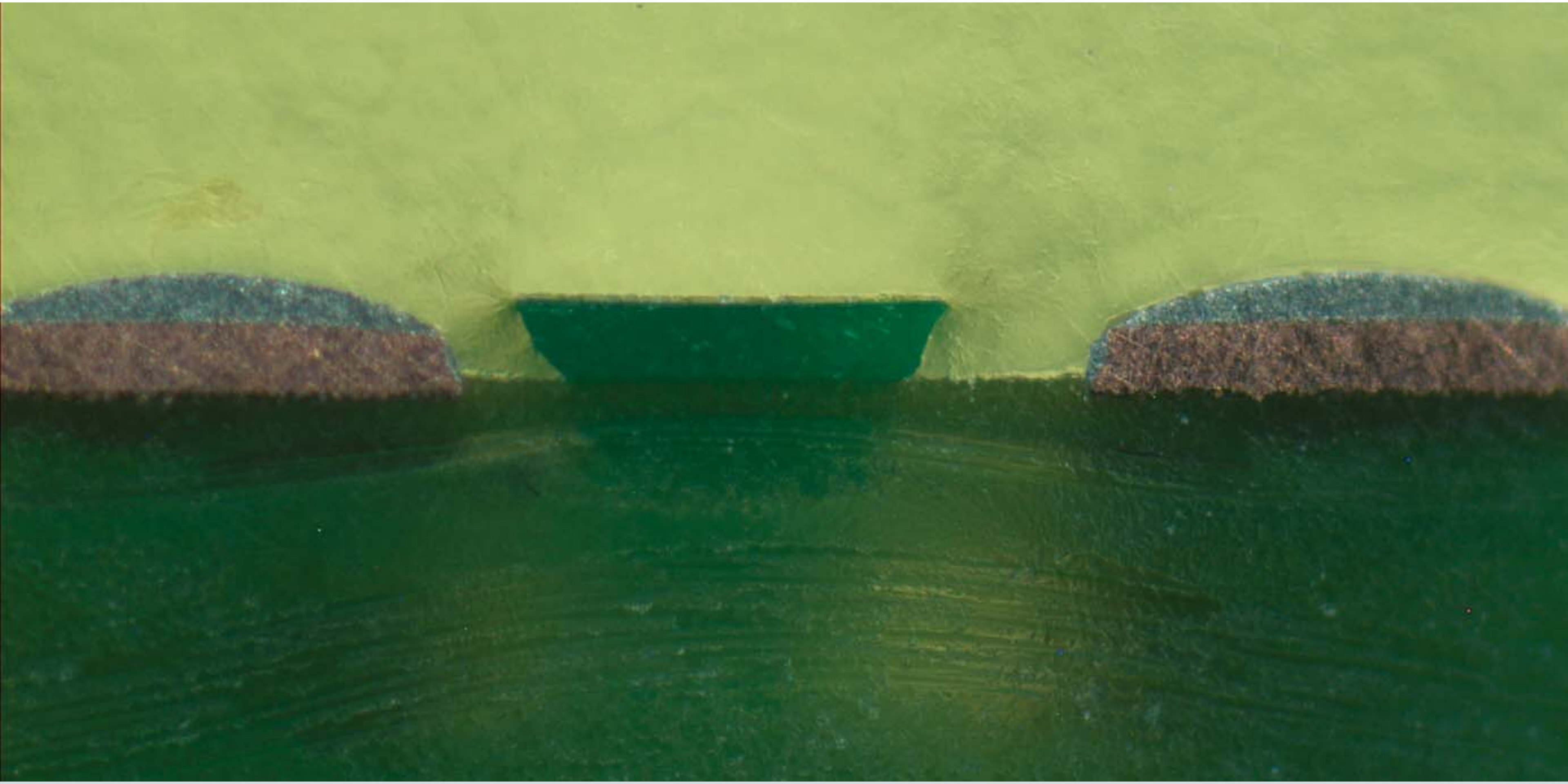
Remove soldermask bridges when the pitch is too small



3

SOLDERMASK OPENINGS / BRIDGES

Soldermask web



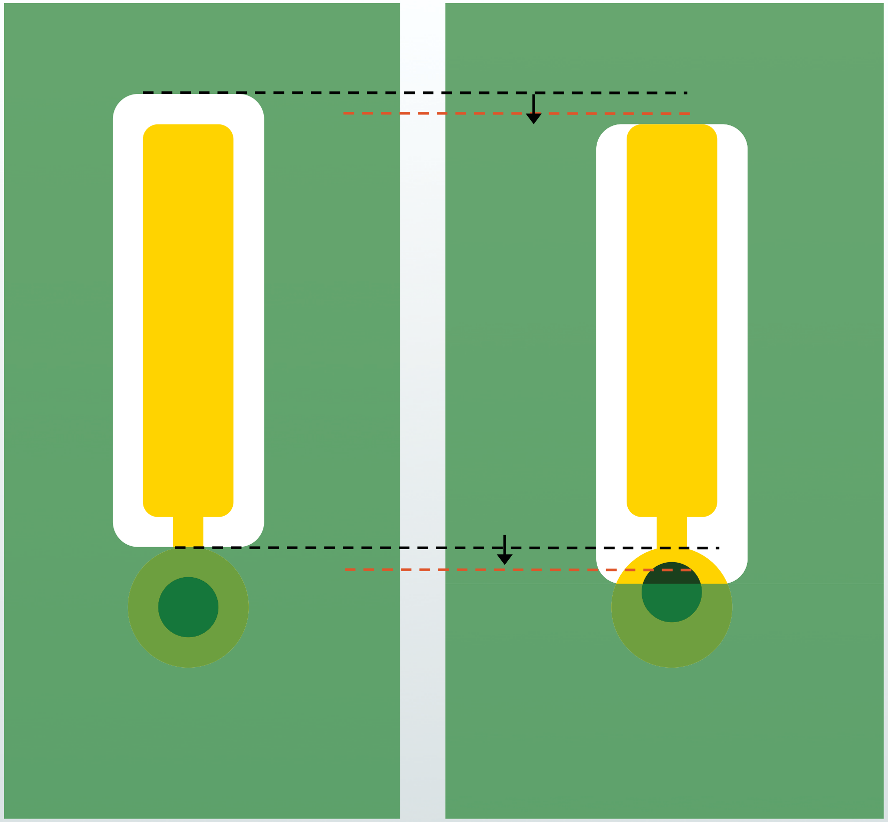
3 SOLDERMASK OPENINGS / BRIDGES

Placement of via holes

There is an obvious risk when placing a via hole too close to a SMD pad.

If the soldermask moves (acceptable alignment) and via hole moves (acceptable drilling registration), towards each other, there is a risk that the via hole can be exposed.

Result can be that the solder applied during the assembly process may creep down into the hole during the soldering process and provide a bad soldering result.



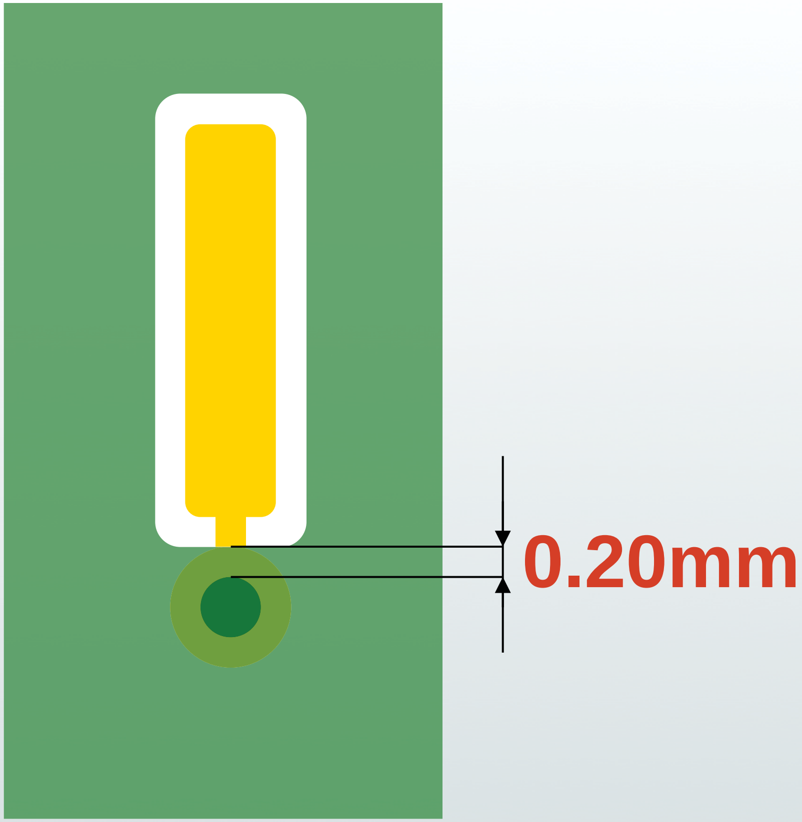
3

SOLDERMASK OPENINGS / BRIDGES

Recommendation

The distance between the soldermask opening and hole edge should be at least 0.20mm to ensure that the hole remains protected by soldermask.

With this design the solder will **<u>not</u>** creep down the hole in the soldering process and provide a bad soldering result.



4 ANNULAR RING / CLEARANCES

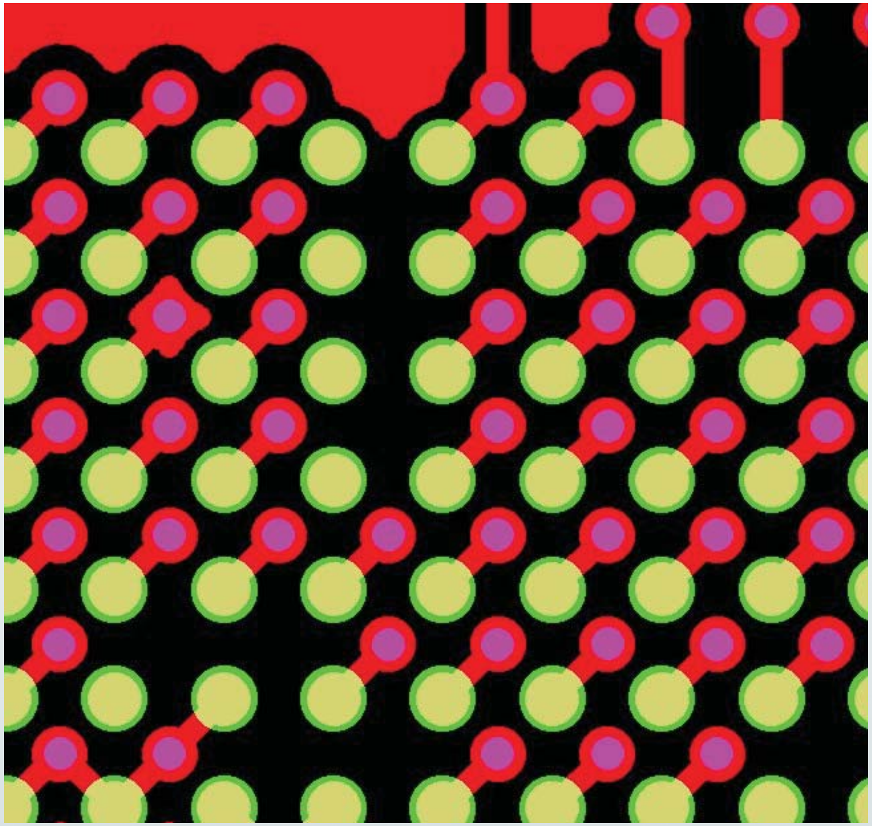
Design impact

In this example the pad is

0.50mm

and the drill hole

0.30mm



4

ANNULAR RING / CLEARANCES

Design impact

In this example the pad is

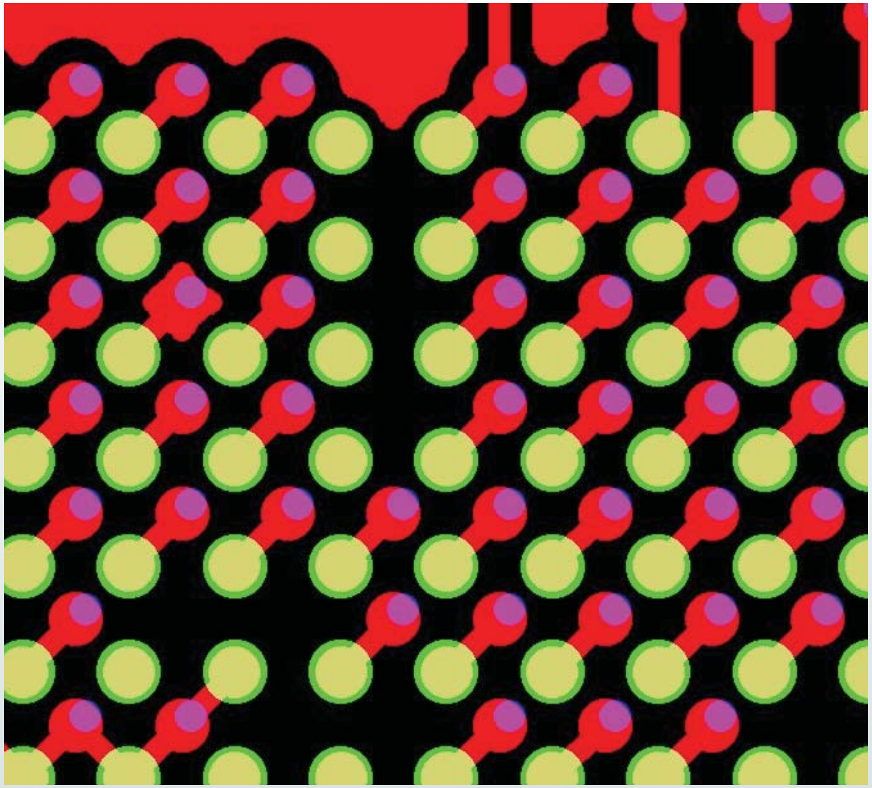
0.50mm

and the drill hole

0.30mm

yet we have drill movement (pink) causing < 90^o breakout.

OK as per IPC class 2



4 ANNULAR RING / CLEARANCES
Design impact

In this example the pad is

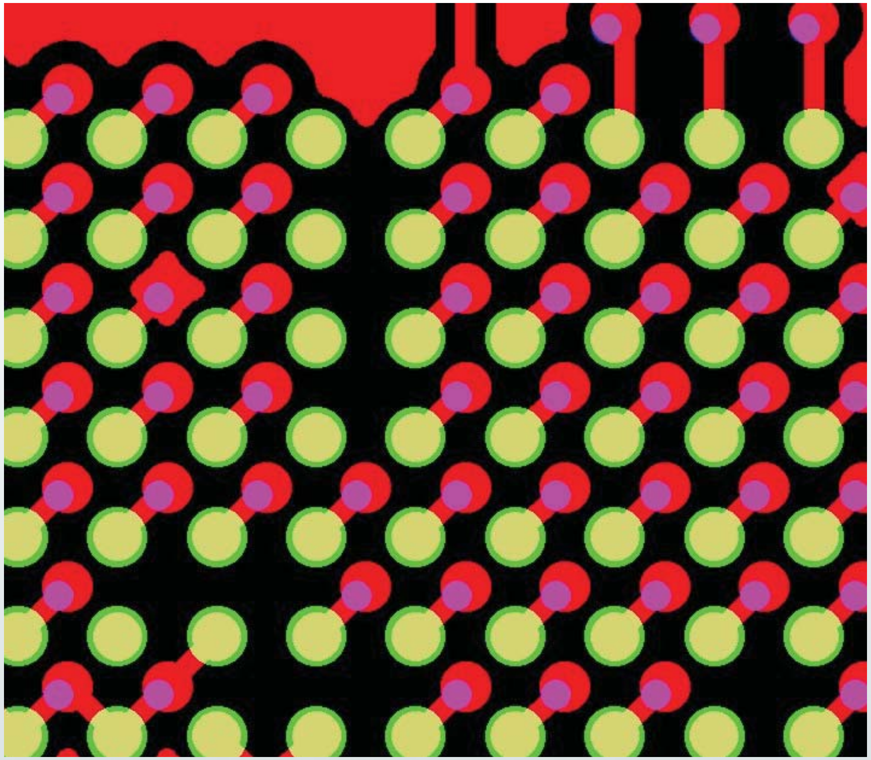
0.50mm

and the drill hole

0.30mm

drill movement (pink) in the opposite direction reduces land/conductor junction >20%.

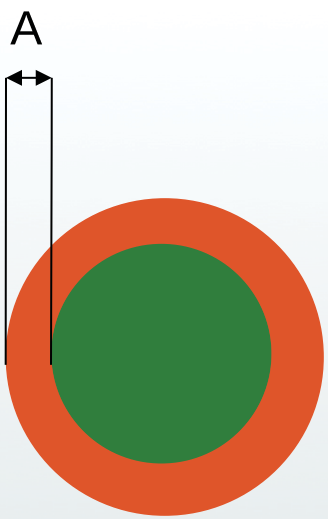
Reject as per IPC class 2

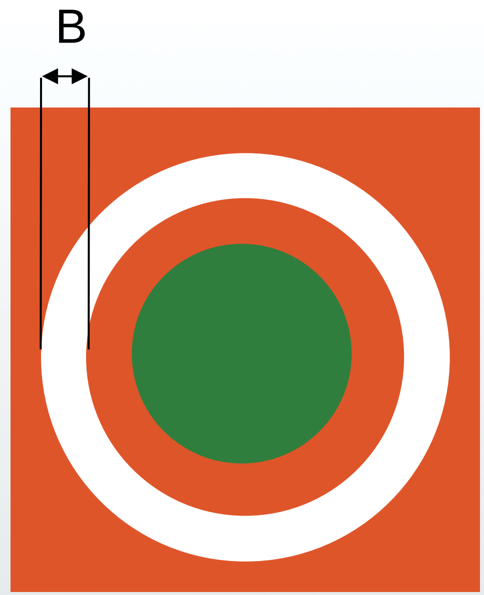


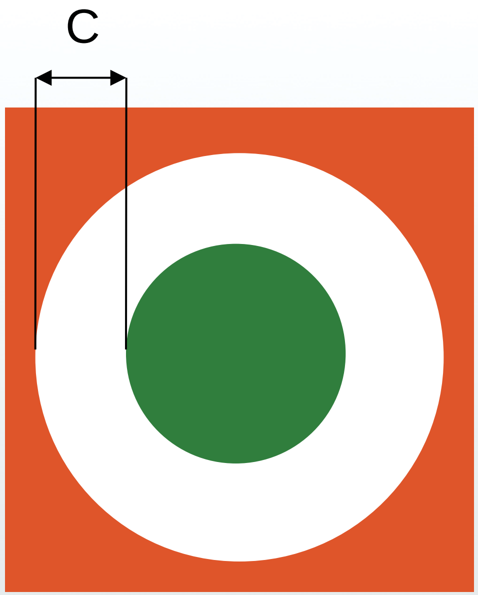
4

ANNULAR RING / CLEARANCES

Recommendation







General

A = 150µm

B = 150µm

C = 300µm

Moderate

A = 125µm

B = 125µm

C = 250µm

Advanced

A = 100µm

B = 100µm

C = 200µm

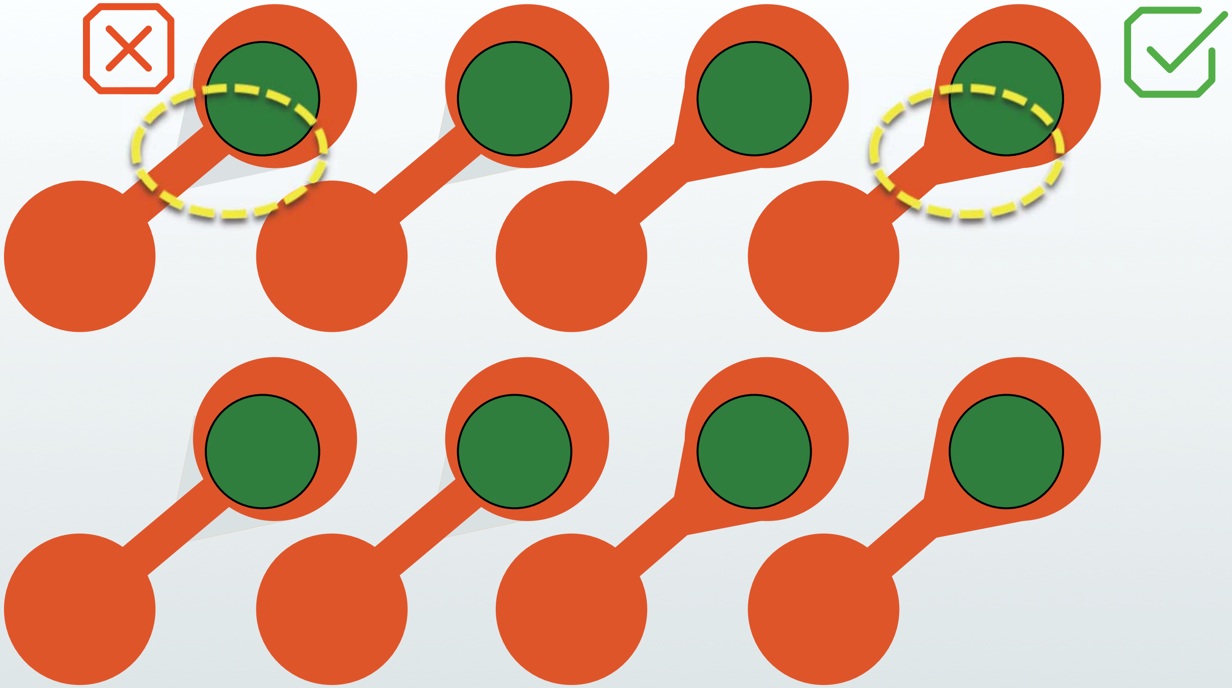
4

ANNULAR RING / CLEARANCES

Tear drops

Tip!

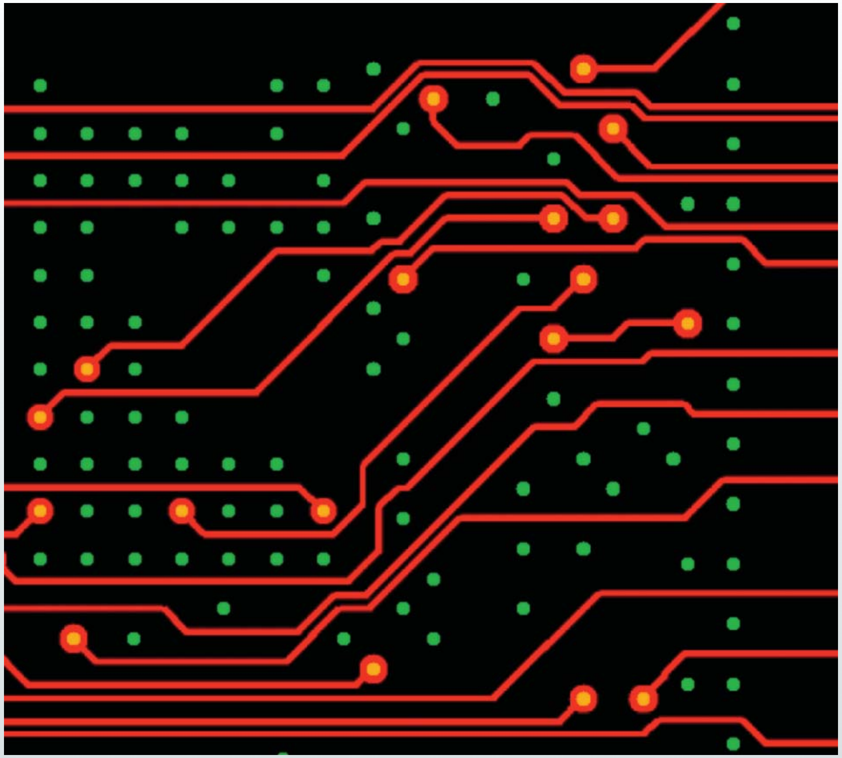
Design with tear drops or give acceptance to factory to add tear drops

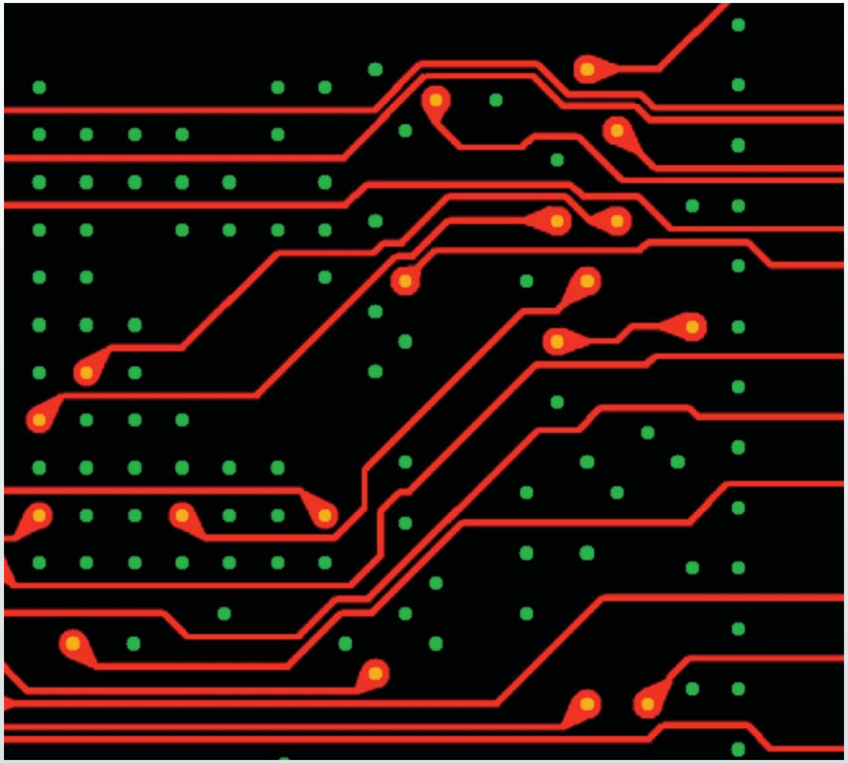


4

ANNULAR RING / CLEARANCES

Tear drops
– available space is necessary!

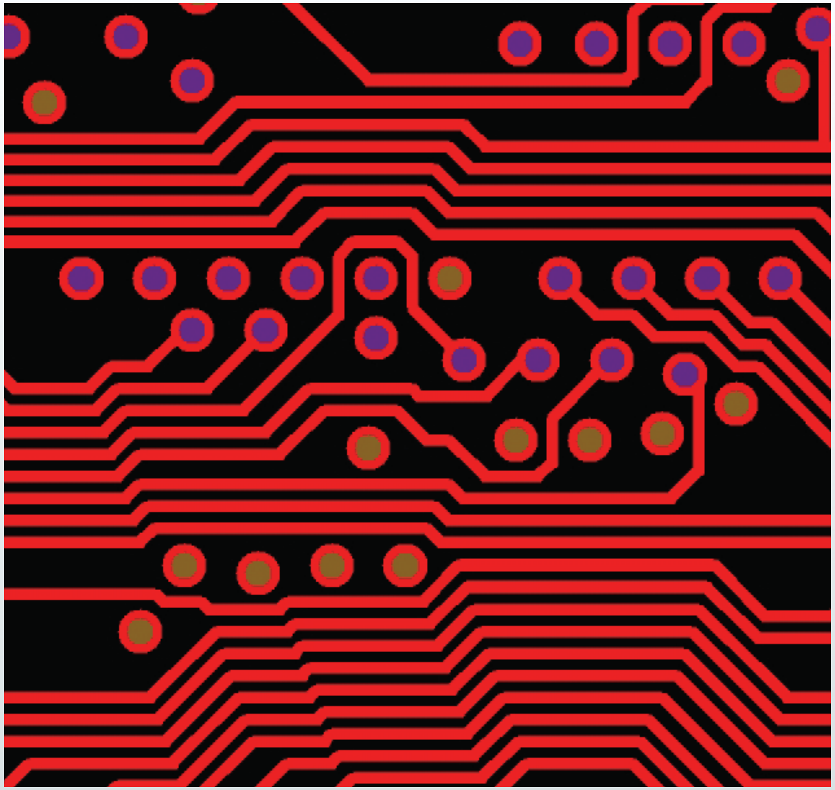


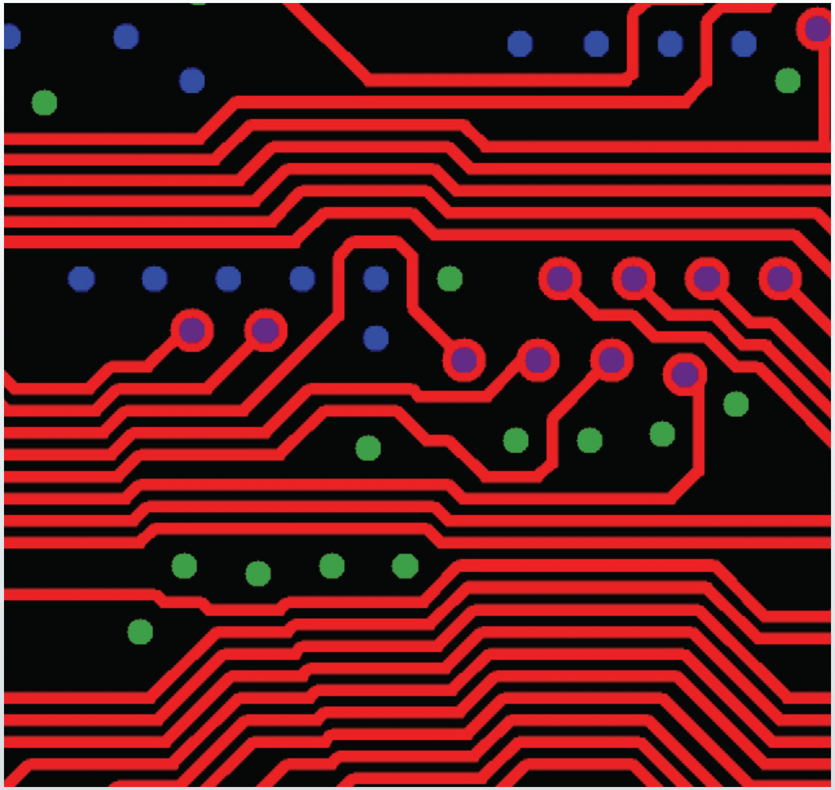


5

COPPER LAYOUT

Redundant pads on inner layers – why?



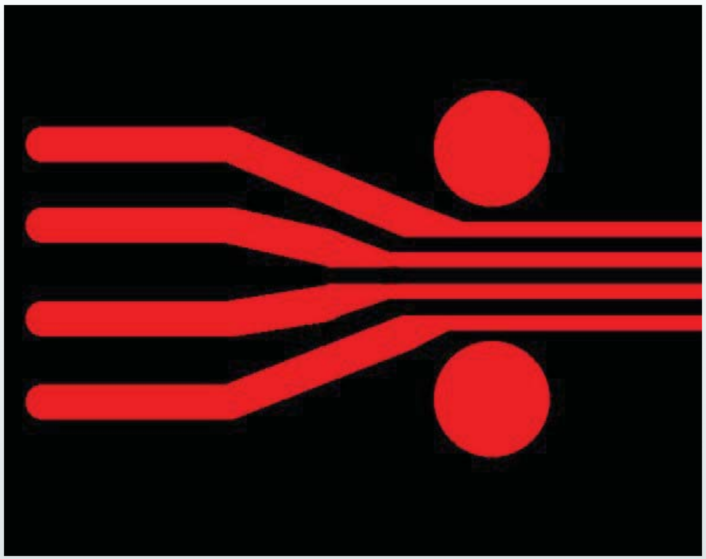


5

COPPER LAYOUT

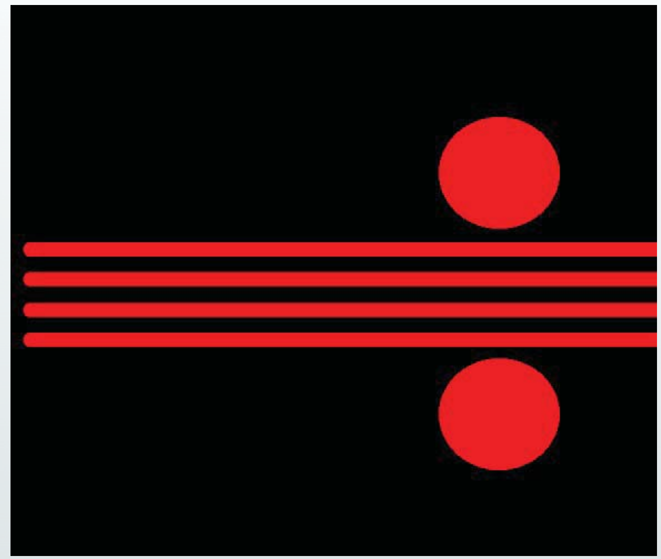
Thin track and gap

The outcome / yield at a manufacturer is influenced
by the distance that thin track and gaps run side by side.



Good design

- thin tracks are used
ONLY where required.



Poor design

- thin tracks are used on the
WHOLE board (auto routing).

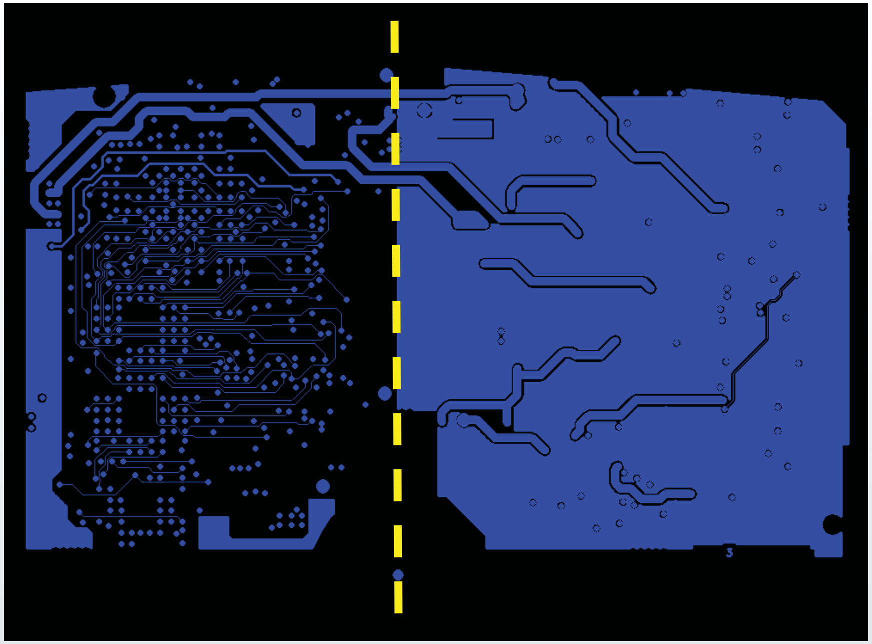
5

COPPER LAYOUT

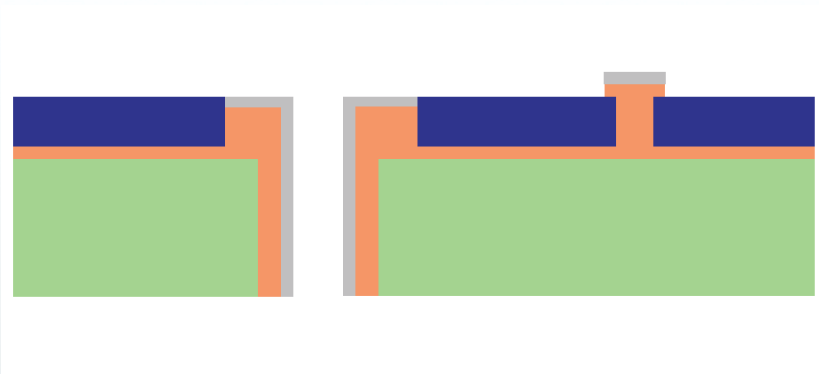
Copper balance

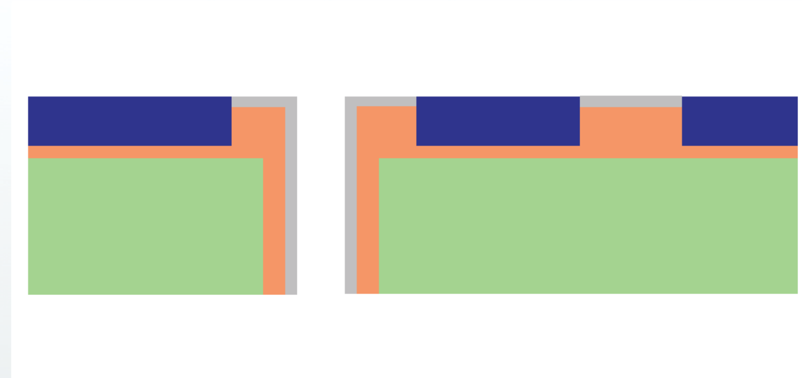
Example of poor copper balance

– two very different sides to one layer!



5 COPPER LAYOUT
Poor copper balance





Poor
copper balance leads to excessive copper plating

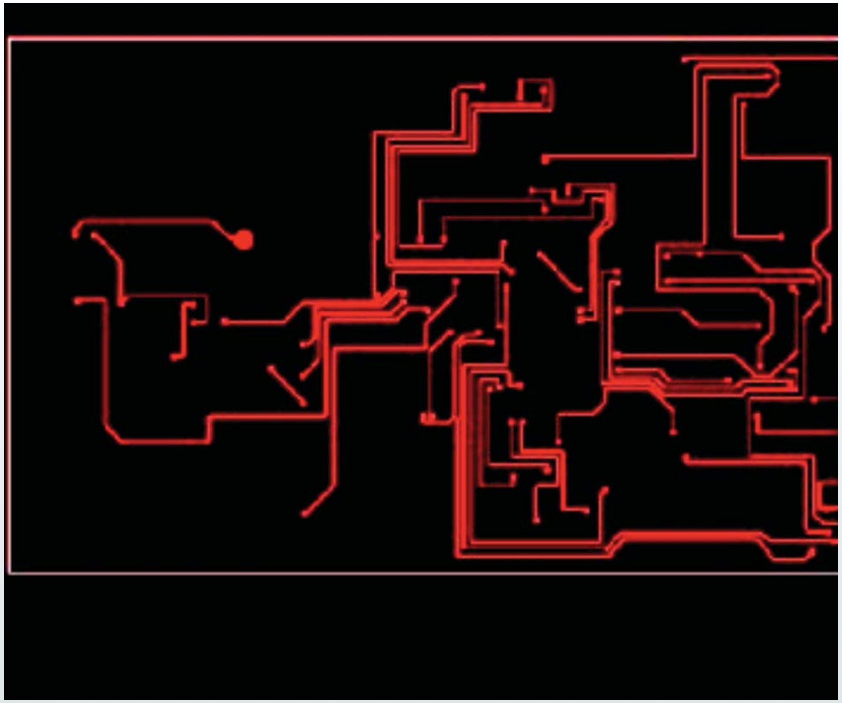
Good
copper balance leads to an even copper plating

Tip! Additional copper should be used to balance sparse areas.

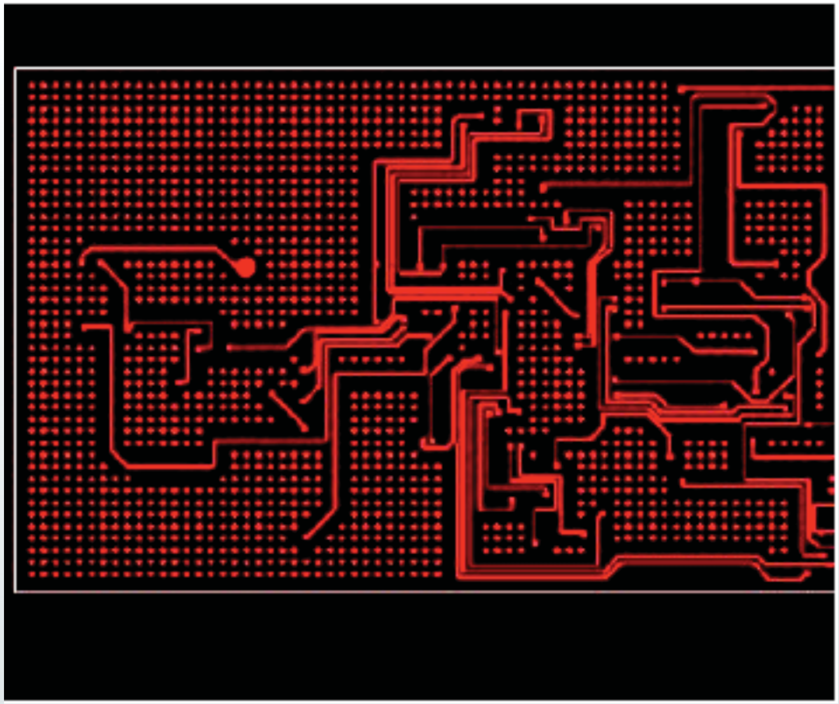
5

COPPER LAYOUT

Improving copper balance



Before

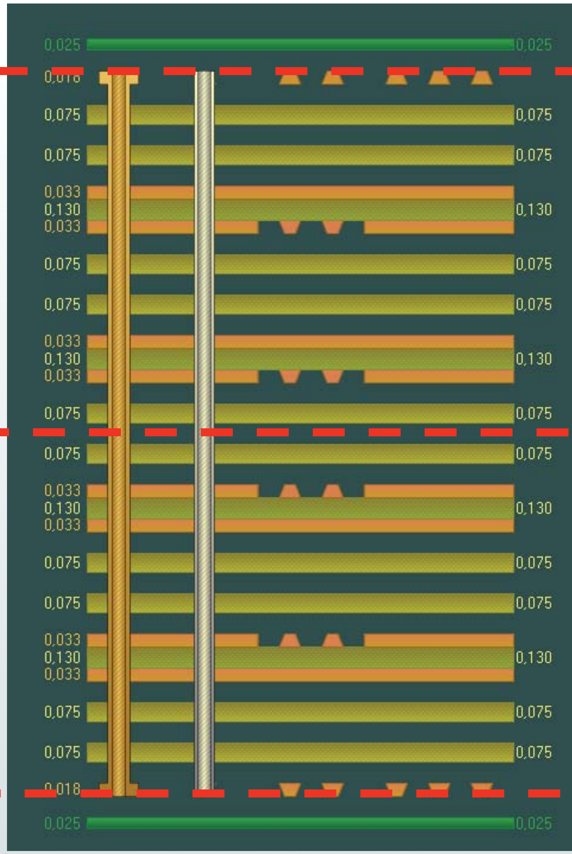


After

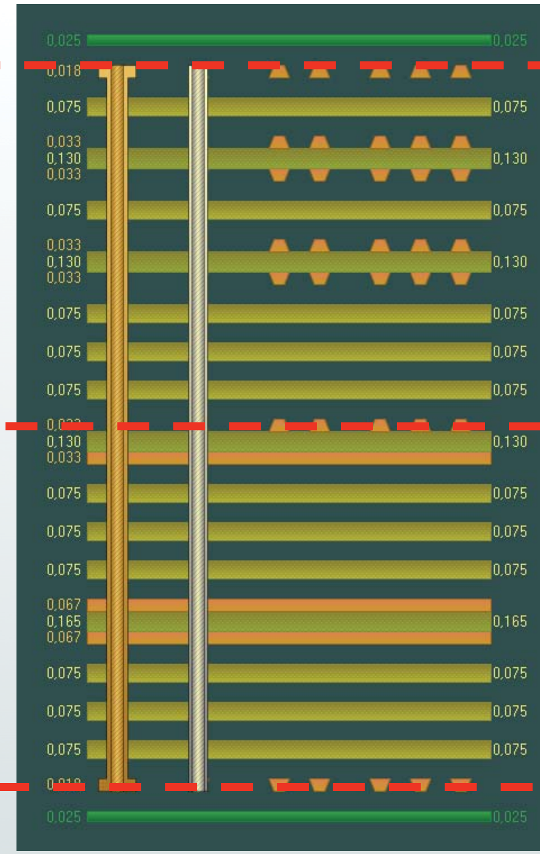
5

COPPER LAYOUT / DESIGN

Symmetry of builds



Symmetrical build up



Unsymmetrical build up

NCAB GROUP

6 COPPER THICKNESS / TRACK WIDTH

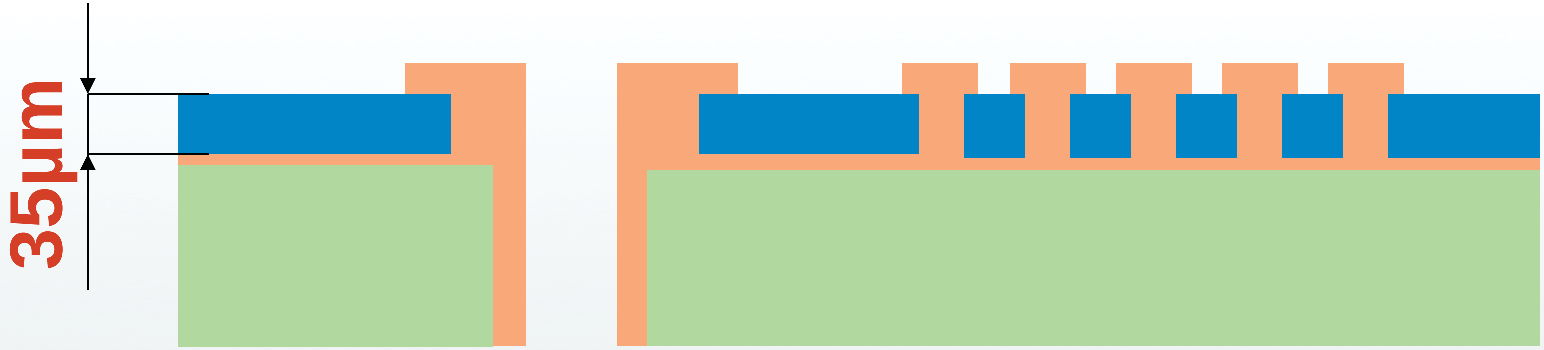
General surface thickness

Normally, the aim during is to achieve an average copper thickness, within the hole, of 25µm*. The distribution / thickness on the surface depends upon the copper balancing and normally a plating thickness between 15-35µm is achieved. At 18µm base copper this provides a final copper thickness in the region of 30-50µm.

* NCAB Group works to **IPC class 3** requirements for through hole copper plating.

6 COPPER THICKNESS / TRACK WIDTH

Base copper



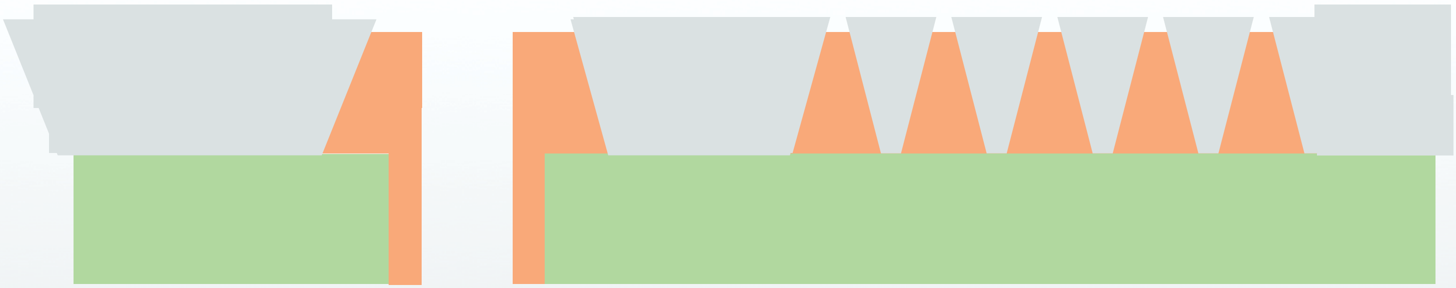
In order to obtain a thicker track you have to start with a thicker base copper, as the photoresist is typically 35µm thick and therefore attempts to plate more than this will result in over plating as shown in the above graphic.

Normal base copper thickness are 18, 35, 70, 105, etc.

6

COPPER THICKNESS / TRACK WIDTH

Thick base copper - limits



Thick base copper can, however, lead to difficulties or challenges when etching.

Because of this there are limits in terms of how thick/thin the track and gaps can be within the design the design.

Generally the thicker the copper then the greater the track and gap.

Normally the manufacturer will add an etch compensation as long as the isolation distance allows it.

6 COPPER THICKNESS / TRACK WIDTH

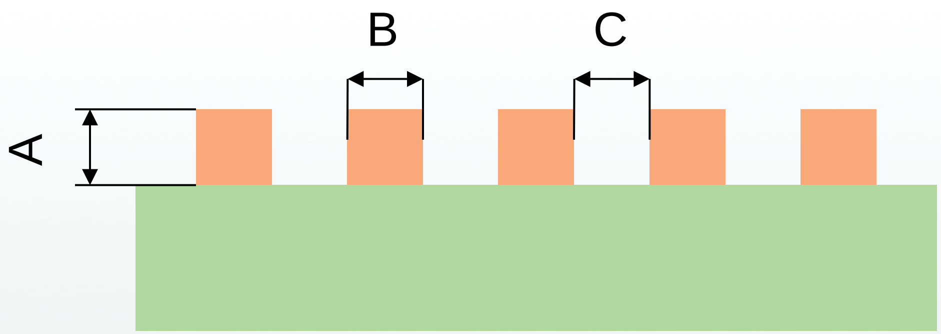
Poor design

This is an example of poor design when specifying 105µm copper. The design includes 6/6 mil track/clearance in the highlighted section, even though there is plenty of space to increase this.

6

COPPER THICKNESS / TRACK WIDTH

Outer layer recommendations

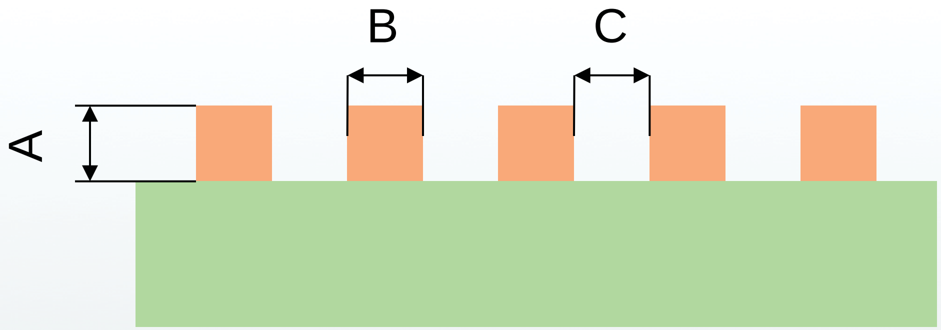


	General		Moderate		Advanced	
A	B	C	B	C	B	C
18µm	125µm	125µm	100µm	100µm	75µm	75µm
35µm	150µm	150µm	125µm	125µm	125µm	125µm
70µm	225µm	225µm	200µm	200µm	175µm	175µm
105µm	300µm	300µm	250µm	250µm	225µm	225µm

6

COPPER THICKNESS / TRACK WIDTH

Inner layer recommendations



	General		Moderate		Advanced	
A	B	C	B	C	B	C
18µm	125µm	125µm	100µm	100µm	75µm	75µm
35µm	150µm	150µm	125µm	125µm	100µm	100µm
70µm	200µm	200µm	175µm	175µm	140µm	150µm
105µm	250µm	250µm	225µm	225µm	200µm	200µm

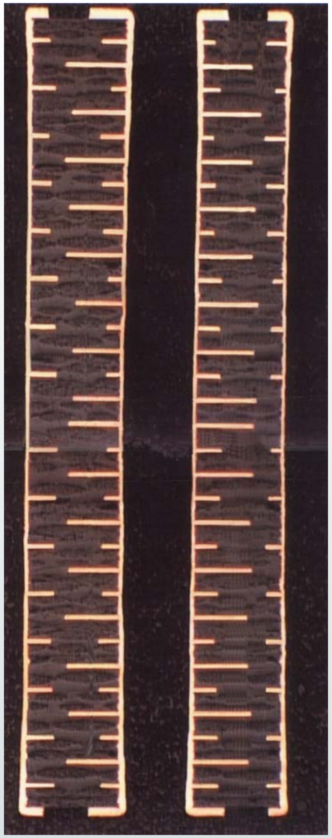
7 DRILL DIAMETER / ASPECT RATIO

What's the relationship?

Aspect ratio is the ratio between the minimum hole diameter and overall thickness of the board.

For example; if the board thickness is 1.60mm and the minimum hole size is 0.40mm, then the aspect ratio is said to be 1:4.

Higher aspect ratios are more difficult to produce.



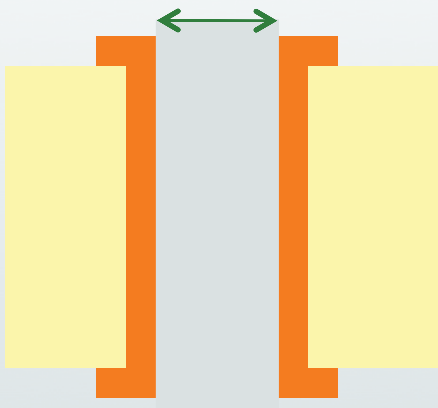
Example of 1:15

7

DRILL DIAMETER / ASPECT RATIO

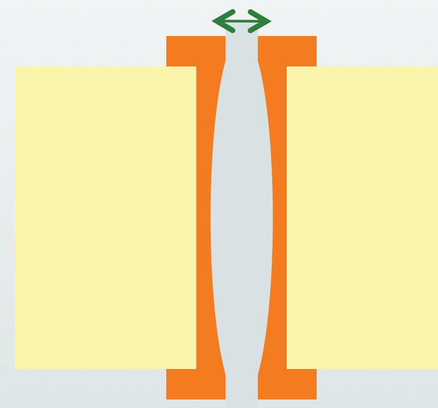
Small hole size / Higher aspect ratio

When the holes are small it is difficult for the plating solution to flow through the holes and plate evenly.
This can lead to very thin plating in the middle of the hole (if you are using the wrong equipment). It is more common to now see that manufacturers have an aspect ratio of 1:8.



Large hole

= better solution flow



Small hole

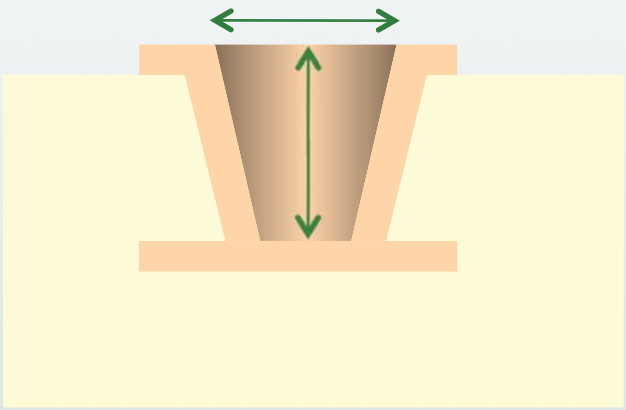
= ‘tougher’ solution flow

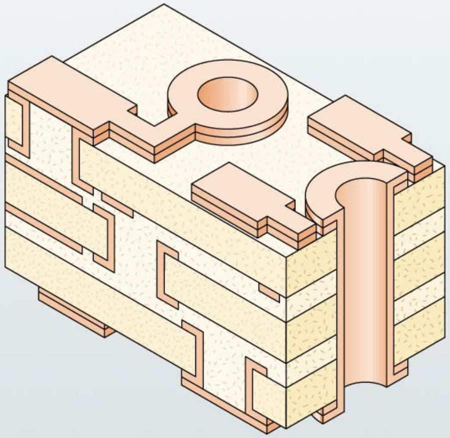
7 DRILL DIAMETER / ASPECT RATIO

Blind via holes

Blind holes that are drilled with laser or depth controlled drilling must have an aspect ratio of less than 1:1. It is preferable if the aspect ratio is 0.7:1.

If the holes are built in sequence it is possible to have the same aspect ratio as for plated through holes.



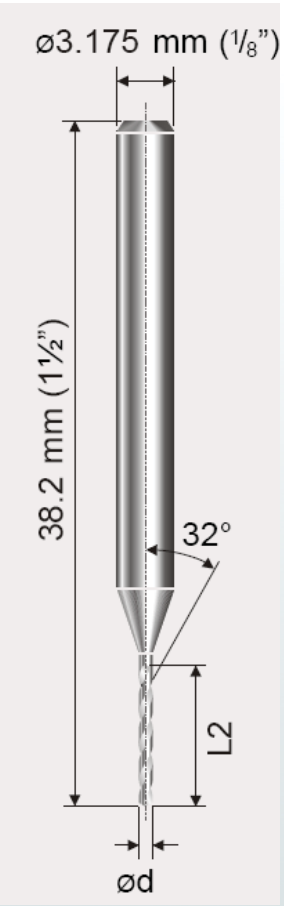


7 DRILL DIAMETER / ASPECT RATIO

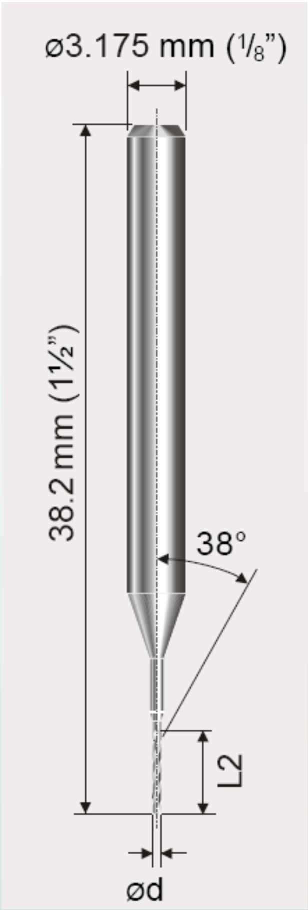
Drilling

Drilling of small holes sets higher demands on the equipment and also reduces the number of boards you can drill in the stack.

If the smallest drill is 0.20mm you can only drill a PCB which is 1.60mm thick.



0.50-3.10mm



0.05-0.50mm

KEMMER PRÄZISION

DRILL DIAMETER / ASPECT RATIO

Smallest hole size / aspect ratio recommendations

General

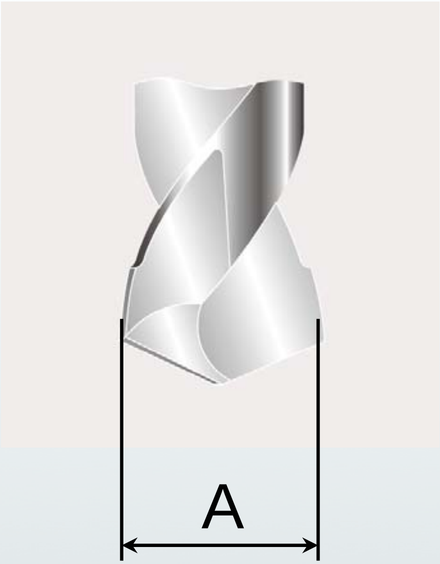
A = 300µm
B = 6-8:1

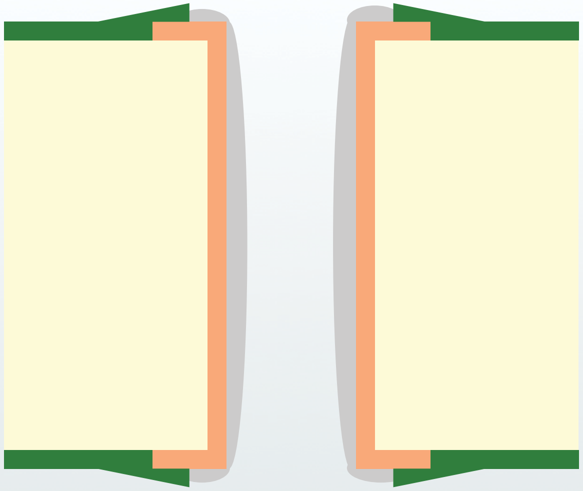
Moderate

A = 250µm
B = 8-10:1

Advanced

A = 200µm
B = 12-20:1





B = Aspect ratio

8

MATERIAL YIELD

Panel size

Since a large part of the cost is related to the raw material (see NCAB Group presentation on **PCB cost drivers** for more information!), it is therefore important for the manufacturer to have a good material yield to avoid scraping unused processed material.

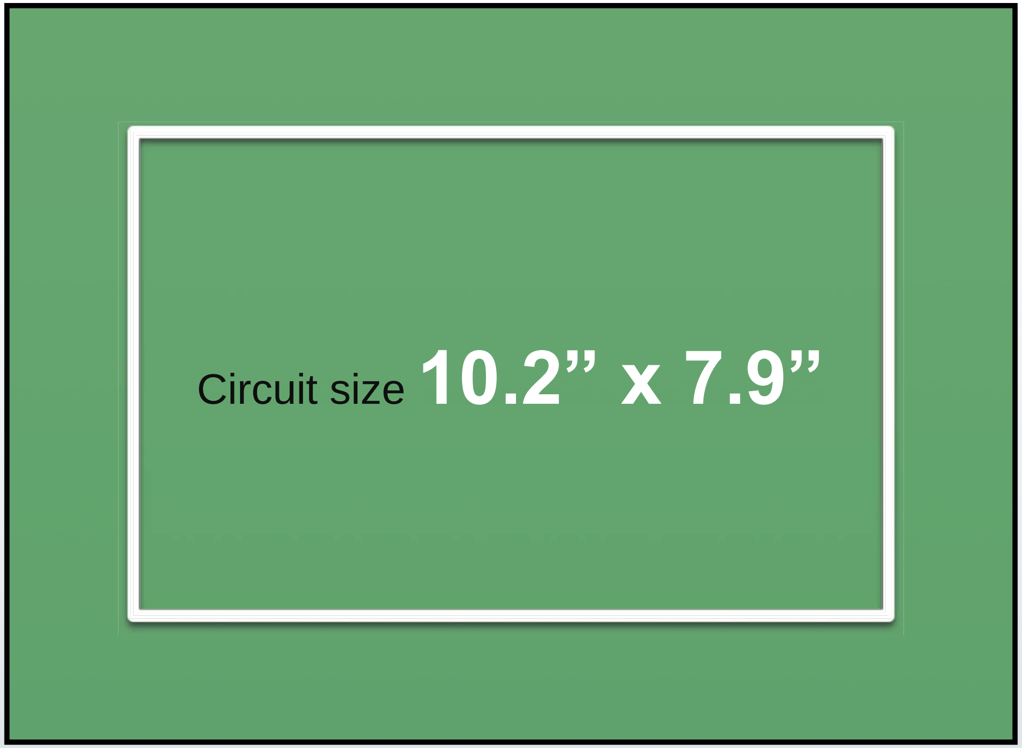
In Asia the production panel size can be adapted to the board design (more panel size options), however in Europe it is more common to use fewer standard panel sizes.

However, this does not mean that material utilisation within Asian factories is a factor which can be 'ignored'.

8

MATERIAL YIELD

Example – customer panel



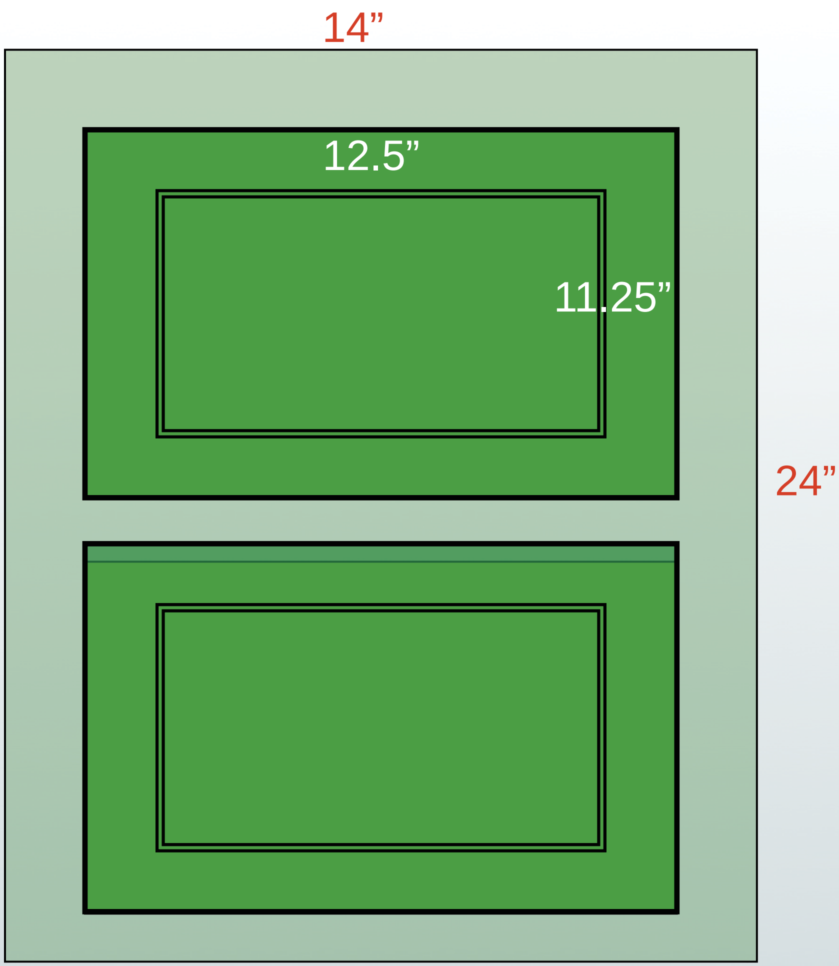
Customer panel **12.5”x 11.25”**

8

MATERIAL YIELD

Example – production panel

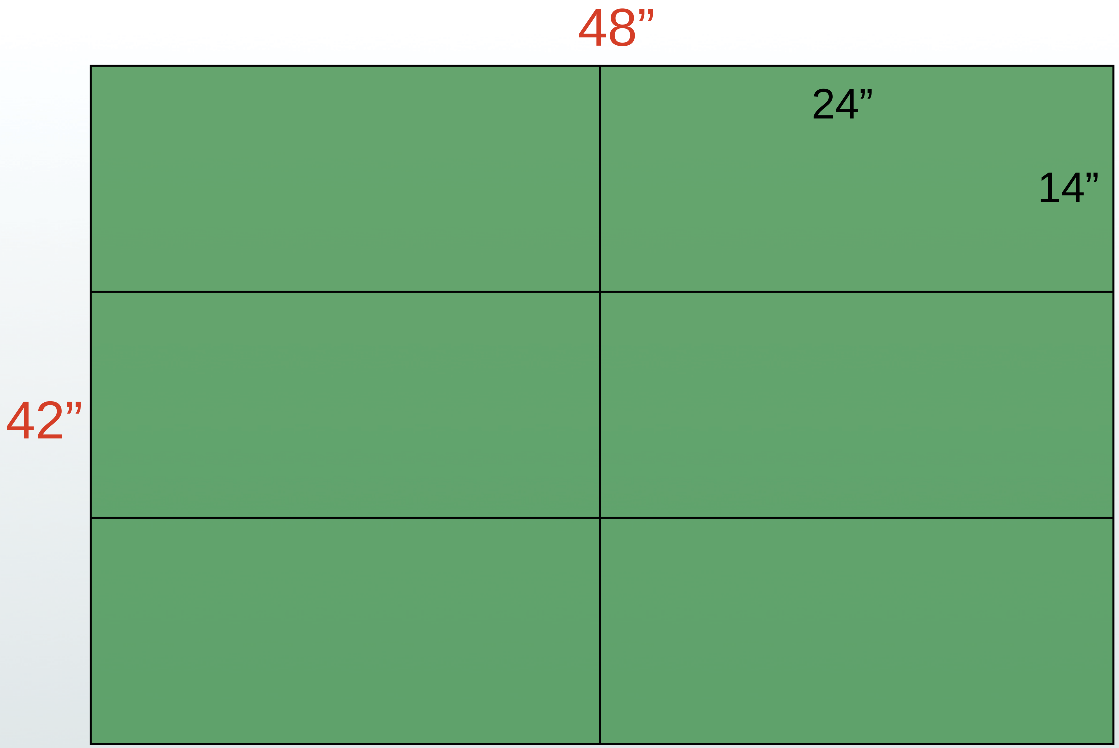
This example shows 2 x customer panels in one production panel



MATERIAL YIELD

Example – Raw material sheets

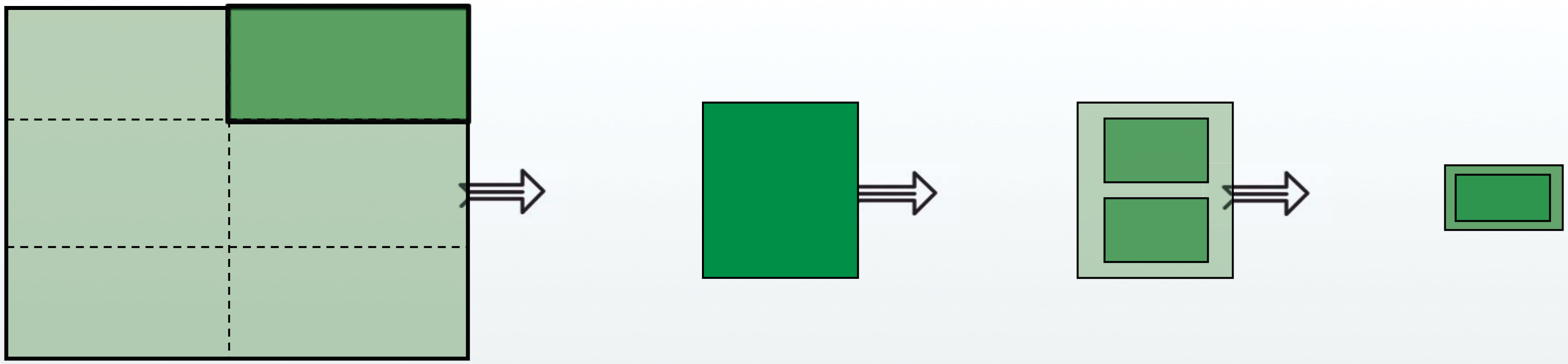
6x production panels can be cut from one sheet of raw material



8

MATERIAL YIELD

Utilisation / material yield



1 x sheet of raw material (48"x42") = 2,016 sqin = 6 x working panels

1 x working panel = 2 arrays

1 cust. panel = 1 PCB (10.7"x7.9") = 84.53 sqin

So… 1 sheet of raw material contains 12 PCB's.
Looking at the material yield is this example we can calculate as:

Yield = (12 * 84.53) / 2,106 = 50.3%

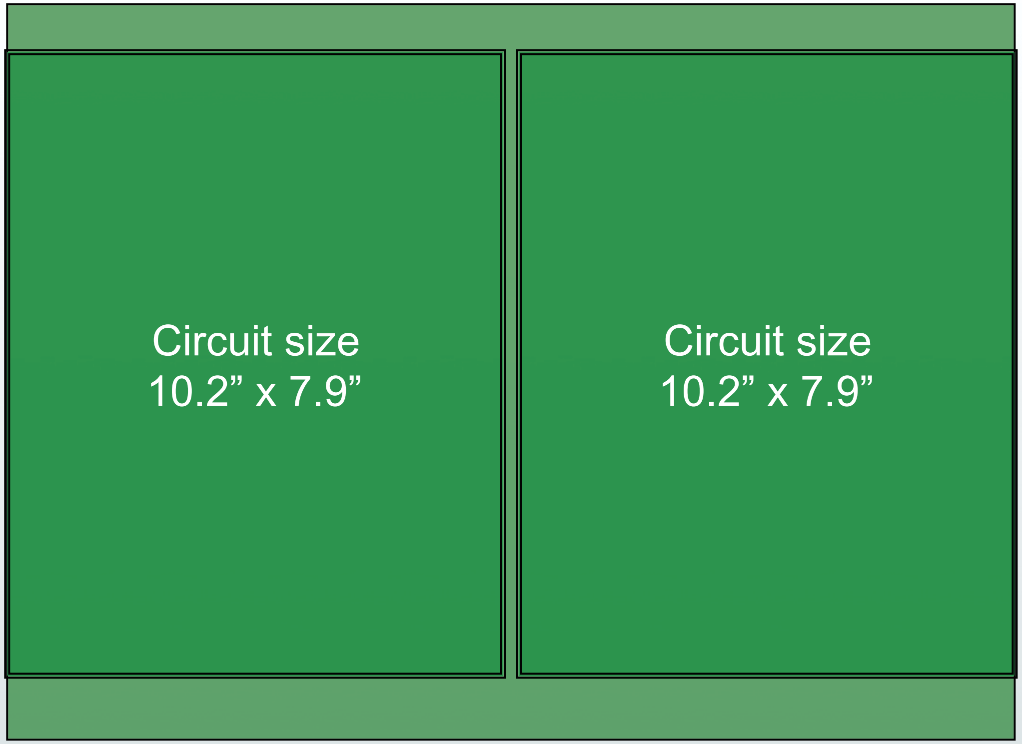
This is far from being efficient and can result in the factory wishing to revise prices (upwards!) on back of poor utilisation.

8

MATERIAL YIELD

How to improve the yield?

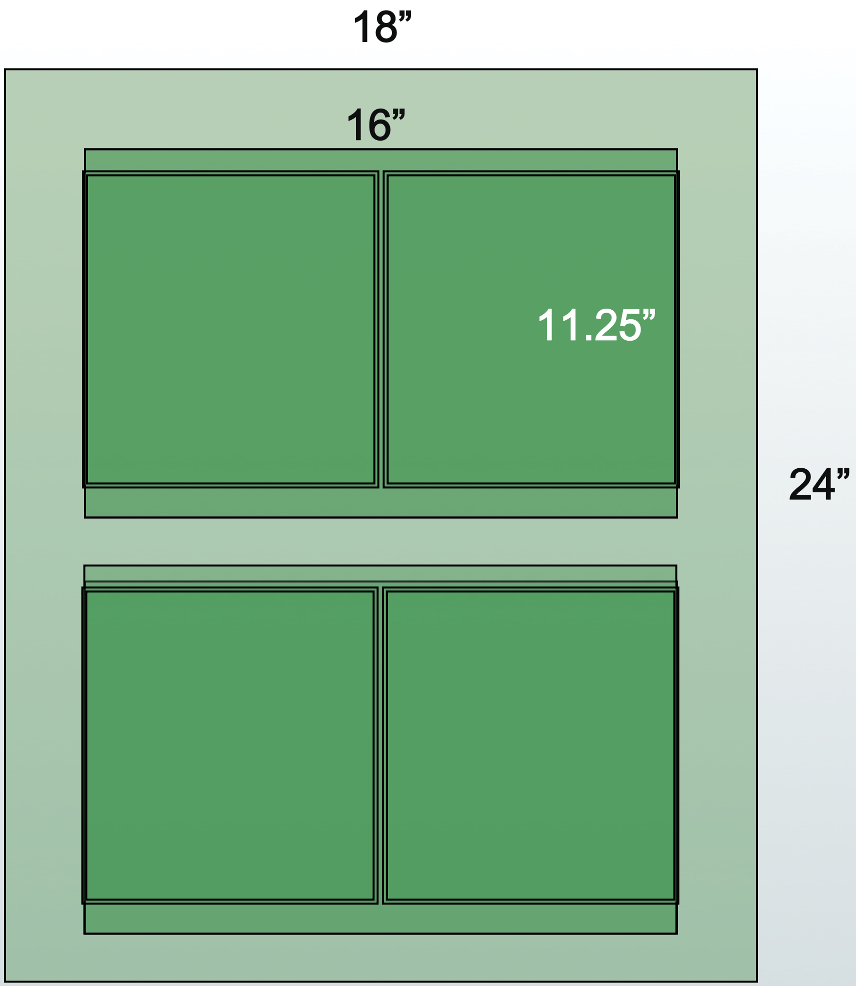
Re-design the customer panel to 16”x 11.25” - **without** any impact on PCB circuit size or function (was 1up, 12.5” x 11.25”)



8 MATERIAL YIELD

How to improve the yield

Re-design example shows 2x arrays / 4 circuits in one production panel

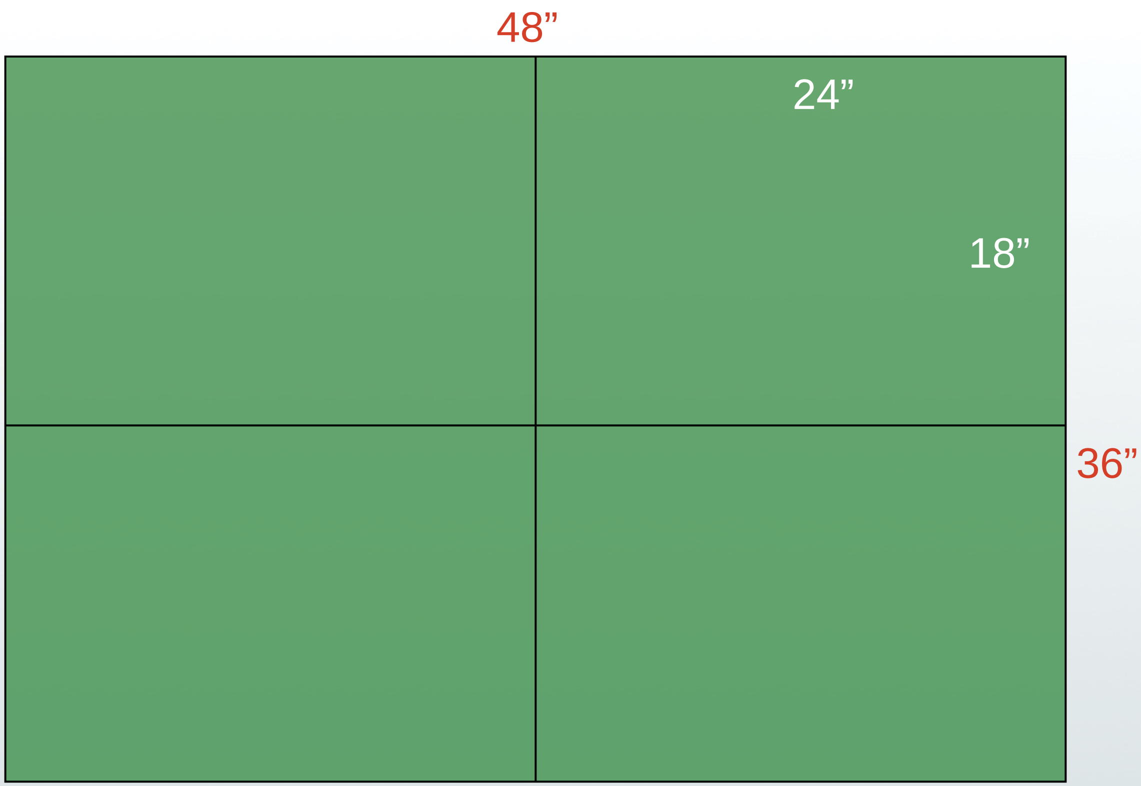


8

MATERIAL YIELD

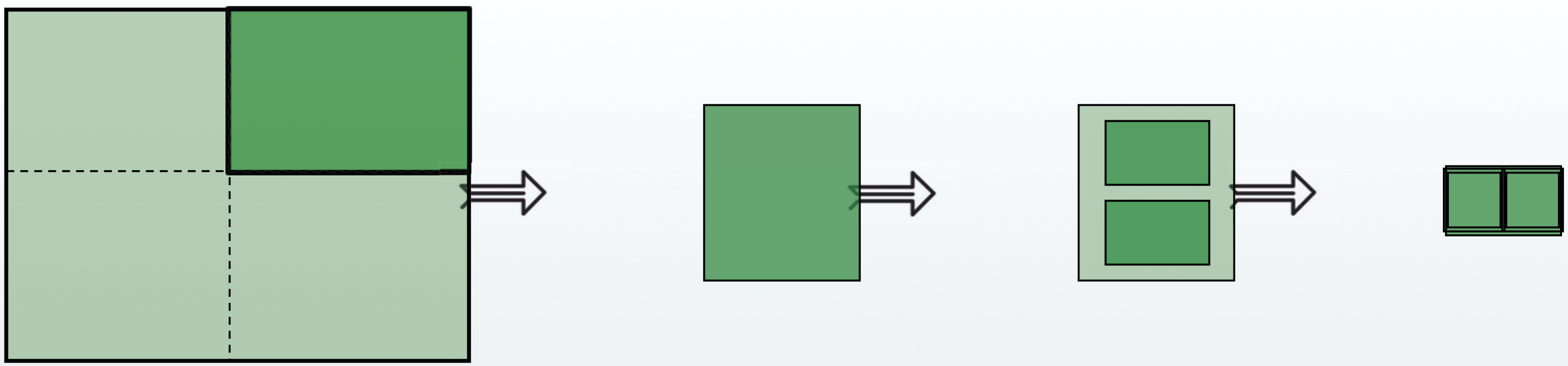
How to improve the yield

Re-design example shows 4x production panels can be cut from one smaller sheet of raw material.



8 MATERIAL YIELD

Improved utilisation / material yield



1 x sheet of raw material (48”x36”)
= 1,728 sqin = 4 x working panels

1 x working panel = 2 arrays

1 array = 2 PCB
(10.7” x 7.9”) = 2 x 84.53 sqin

So, 1 sheet smaller of raw material now contains 16 PCB’s (was 12). Looking at material yield we can conclude that this is calculated to be:

Yield = (16 * 84.53) / 1,728 = 78.2%

This is approximately a **55%** increase in material yield or material efficiency compared to the original design.

8

MATERIAL YIELD

Conclusion

The conclusion is that not only can material yields be improved, but costs can be optimised if the re-designs are welcomed – we all benefit from a good material yield.

This apply to all types of materials and especially the higher grades of base material that may cost many times more than a standard FR4 for example.

The NCAB Group welcomes discussions on how we can optimise material yields and this becomes even more critical when we consider high running / high volume boards which can have a long life cycle.


RoHS
Questions?

NCAB GROUP SEMINARS

Improve your knowledge about PCBs – participate in our seminars

- Technical trends in the global PCB industry
- How to produce a printed circuit boards
- New technologies
- Cost drivers in PCB production
- Surface finishes
- HDI - High Density Interconnect
- IMS - Insulated Metal Substrate
- Rigid-flex
- NCAB Group PCB Specification
- Impedance controlled boards
- DFM – Design For Manufacturing
- IPC vs. Perfag
- Reliability, IPC & NCAB
- Material for lead-free production
- Technical advice
- NCAB Group Laboratory