Altium®

ALTIUMLIVE 2018:

The Benefits Gained by Using HDI Technology

Susy Webb
Design Science
Sr PCB Designer

San Diego
Oct, 2018

HDI/ Microvia – What’s the difference

- IPC: uVias are laser vias of generally 6 mil drill (150 microns) or smaller
- HDI is the technology used with microvias:
- Smaller holes and padstacks than TH
- Smaller traces and clearances
- Thinner Dielectrics
- Finer aspect ratios
- Etc.

When to make the jump to HDI:

- IC packages may need it – either high pin count or fine pitch uBGAs (.65mm and below)
- Large parts with lots of connections
- Lack of room - Small physical board size
- Using minimum holes and line widths/spaces and still not enough room for all
- Amount of time needed/allowed to design the board
- When uVias will add other needed benefits

Altium

What HDI can Offer

- Increased design flexibility
- Theoretical Cost equality
- Improved reliability – HDI tested as the most reliable for organic substrates*
- Better for EMI and signal integrity*
- More creative fanout possibilities for all parts
- Efficient signal transition from layer to layer
- Often can be manufactured worldwide – large market for fab

*Happy Holden – “HDI/Microvia Technologies”, PCB East 2009

Possible cost equality - Price/Density Comparison

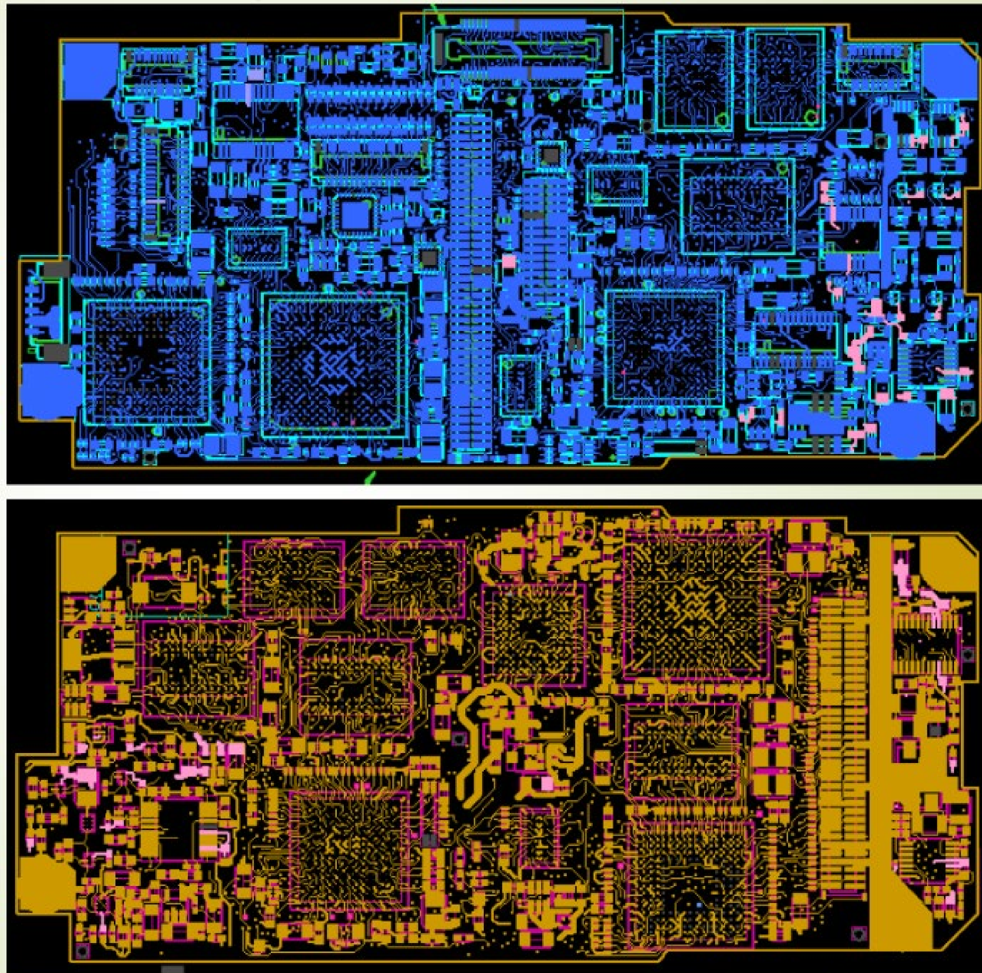
N Layers	A		B		C		D		E		F		G	
	THRU-HOLE		HDI BLIND		HDI BL/BU		1BU BLIND		2BU BLIND		2BU BL.BU		2BU BL.BU	
	N		1+N+1		1+bN+1		1+N+1		2+N+2		2+bN+2		2+bN+2	
	blind via*		L1-L2		L1-L2		skip via L1-L3		staggered L1-L2, L2-L		skip via L1-L3		staggered L1-L2, L2-L	
	buried via		none		L2-L(N-1)		L2-L(N-1)		none		L2-L(N-1)		L3-L(N-2)	
	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN
4L	0.62	20	0.83	40	1.05	80	1.26	135	1.38	135	- -	- -	- -	- -
6L	0.78	20	0.99	60	1.24	160	1.46	200	1.60	200	1.74	260	1.91	280
8L	1.00	30	1.21	120	1.49	180	1.74	240	1.90	240	2.06	300	2.25	320
10L	1.30	40	1.51	200	1.83	210	2.11	260	2.30	260	2.50	400	2.73	440
12L	1.70	60	1.92	210	2.31	230	2.62	300	2.85	300	3.10	600	3.37	650
14L	2.24	70	2.48	220	2.95	250	3.32	360	3.61	360	3.91	800	4.25	860
16L	2.97	80	3.22	260	3.81	300	4.25	420	4.61	420	5.00	1000	5.43	1100
18L	3.92	100	4.21	300	4.95	400	5.47	480	5.93	480	6.42	1250	6.96	1350
20L	5.14	105	5.48	360	6.41	500	7.04		7.62		8.23		8.90	
22L	6.67	110	7.08	400	8.23	600	8.99		9.70		10.45		11.27	
24L	8.53	125	9.03	460	10.41	700	11.32		12.17		13.07		14.04	
26L	10.68	130	11.30	500	12.92		13.96		14.96		16.00		17.11	
28L	13.09	135	13.83	540	15.65		16.82		17.93		19.09		20.32	
30L	15.63	140	16.50	580	18.47		19.73		20.94		22.18		23.48	
32L	18.17	145	19.17	620	21.21		22.53		23.79					
34L	20.59	150	21.69	660	23.73		25.09							
36L	22.79	160	23.96	700	25.94									
38L	24.68	180	25.91	740										
40L	26.26	200												

RCI = Relative cost index, DEN = Pins per sq. in.

Picture reprinted from Happy Holden

Altium

HDI may cost more, but fewer Bd Layers may make up for it

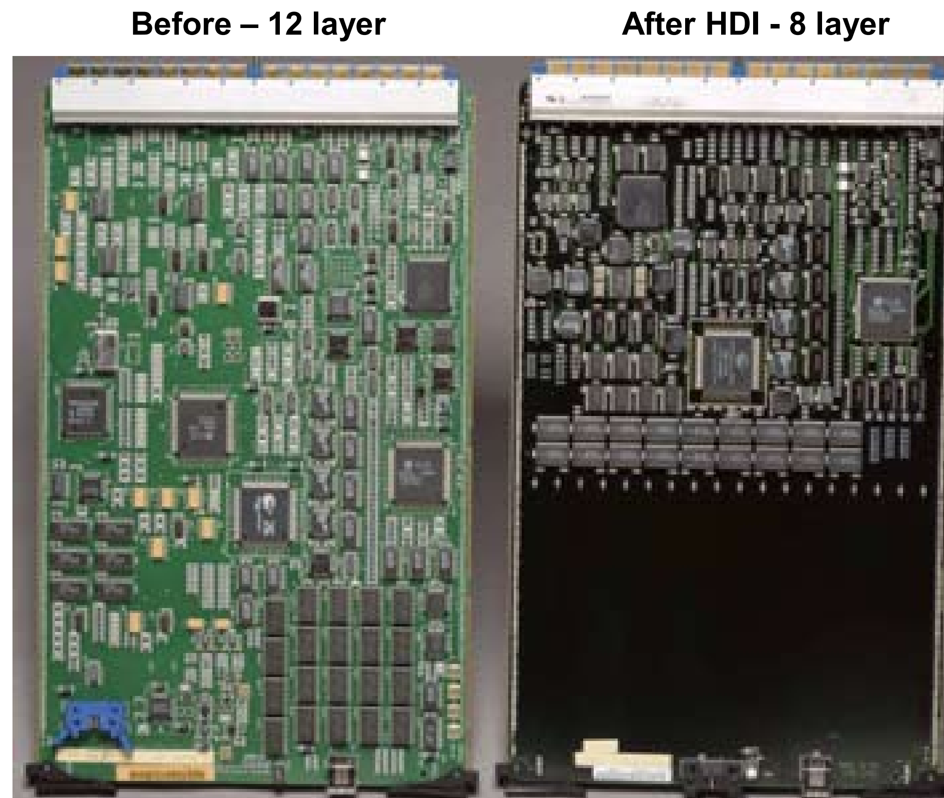


- Example goes from 16 layers TH and BB to 12 layers w/HDI
- Much better for hole aspect ratios

Part of a slide from *Happy Holden – "What you need to learn to gain 3X to 4X higher Layout Density"

Altium®

HDI may cost more, but smaller board size may make up for it



*Happy Holden – "HDI/Microvia Technologies", PCB East 2009

- Good if small physical board size needed
- Less board material needed
- Smaller board is easier to panelize

Altium

Routing Efficiencies per type of board

A measure of the total # of traces vs the total number possible (efficiency %)

Design Scenario	Conditions	Efficiency (ε)
Rigid Thru-Hole	Gridded CAD	30%
Rigid Thru-Hole and SMT	With or W/O Back Side Passives	35-50%
Rigid Thru-Hole and SMT	With Back Side Active Components	30-45%
Rigid SMT Only	With Back Side Active*	up to 55%
Rigid Thru-Hole and SMT	1 Sided Blind Vias*	up to 60%
Rigid Thru-Hole and SMT	2 Sided Blind Vias*	up to 65%
2 layer HDI Structure	*	up to 70%
4 layer HDI Structure	With Blind/Buried Vias*	up to 80%
6 Layer HDI Structure	With Blind/Buried Vias*	up to 75%

** = gridless CAD system*

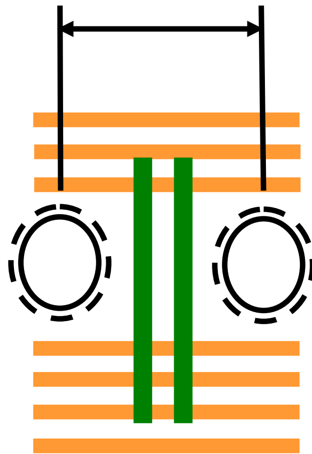
2 layer means 2 layers of Microvias

Reprinted from

HDI may cost more, but can improve routing on all routing layers

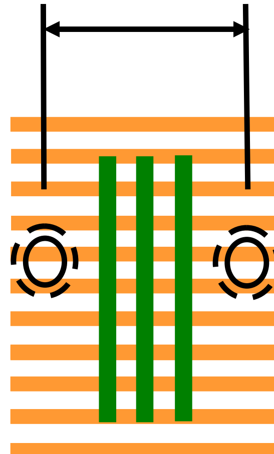
- uVias depth allows more internal signal routing, and uVia size allows for more external routing

TH 1mm (.0393”) pitch



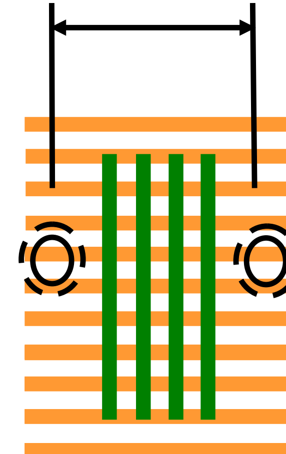
.5mm pad (@ .020”)
.25mm hole (@.010”)
.1mm lines and spaces (@.004”)

HDI 1mm (.0393”)



.25mm pad (@ .010”)
.125mm hole (@.005”)
.1mm lines and spaces (@.004”)

HDI 1mm (.0393”)

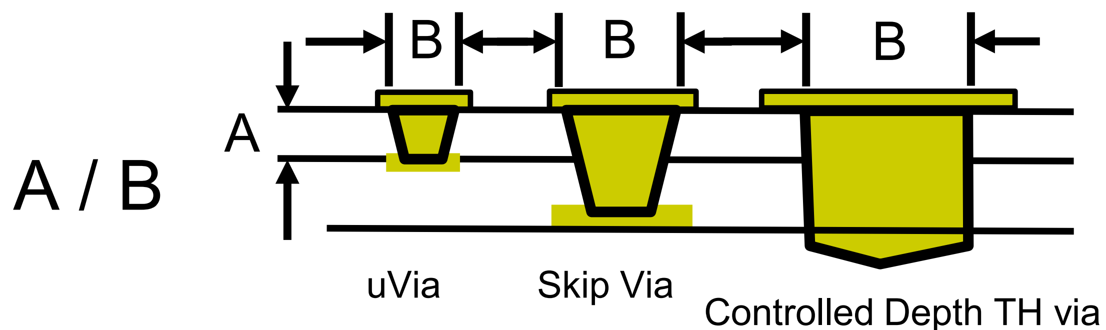


.25mm pad (@ .010”)
.125mm hole (@.005”)
.076mm lines and spaces (@.003”)

Altium

uVia Depth and Board Thickness

- HDI size works well with smaller aspect ratio for layer thicknesses
- Smaller pads/holes take up less room
- TH aspect ratio generally 10:1 up to 12:1
- HDI aspect ratio generally 0.5:1 up to 0.7:1



Aspect Ratio = A) Board (or layer) Thickness divided by B) Drill Diameter

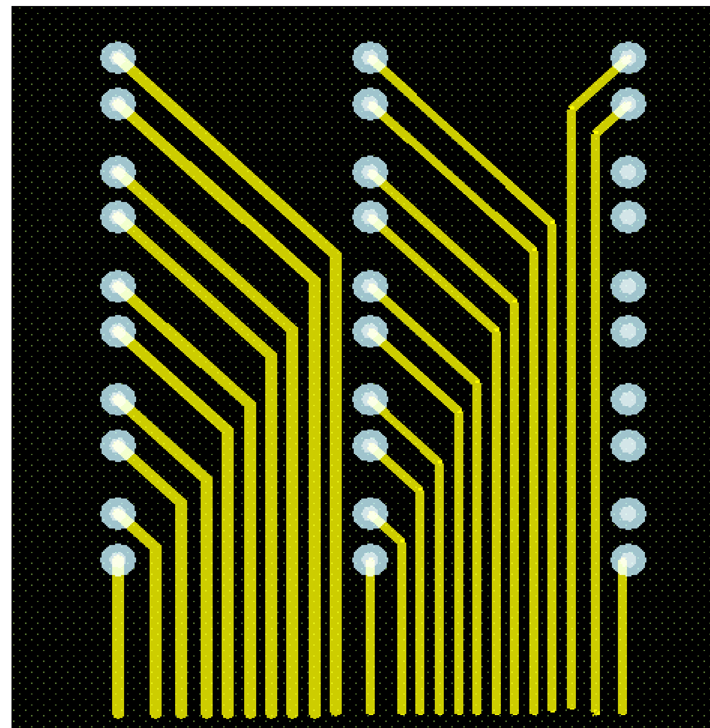
Thin boards – Thin layers – Thin copper

- The thinner dielectrics used with HDI can easily make thinner boards or many layers, if desired
- Even very thin dielectrics used (.002” or less)
- Thin dielectrics may lead to thin trace widths for impedance control
- Thinner copper thickness to start is recommended for trace width/spacing of below 3/4 due to etch compensation

Altium

Thinner copper will allow for finer spacing, meaning more routes in same area possible

- Better efficiency = more traces/ channels/ boulevards
- Fewer routing layers needed



4/4 = 10 traces 3/3 = 12 traces

Altium®

Various possibilities for signal flow layer to layer without blocking other busses

	BUS #1	BUS #2	BUS # 3	Critical Signal group #1	Critical Signal group #2	Power and Ground
layer 1 - Signal w/ poured gnd						
layer 2 - Gnd						
layer 3 - Signal						
layer 4 - Signal						
layer 5 - Gnd						
layer 6 - Pwr						
layer 7 - Signal						
layer 8 - Signal						
layer 9 - Gnd						
layer 10 - Signal w/ poured gnd						

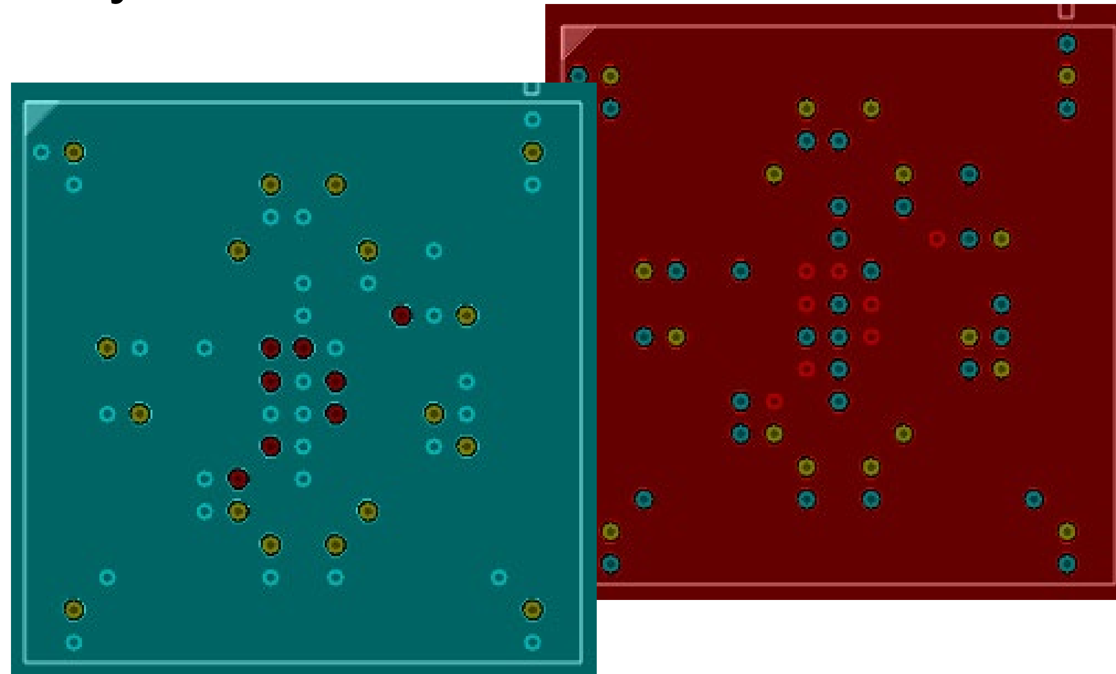
HDI Good for SI and EMI

- Because dielectric layers are thinner, traces closer to return planes
- Improved containment of energy fields
- Possibly better for amount of separation for differential pairs
- HDI external layers are often flooded plane - also good for EMI
- Allows planes to be close together
- Good for inter-plane capacitance

Altium

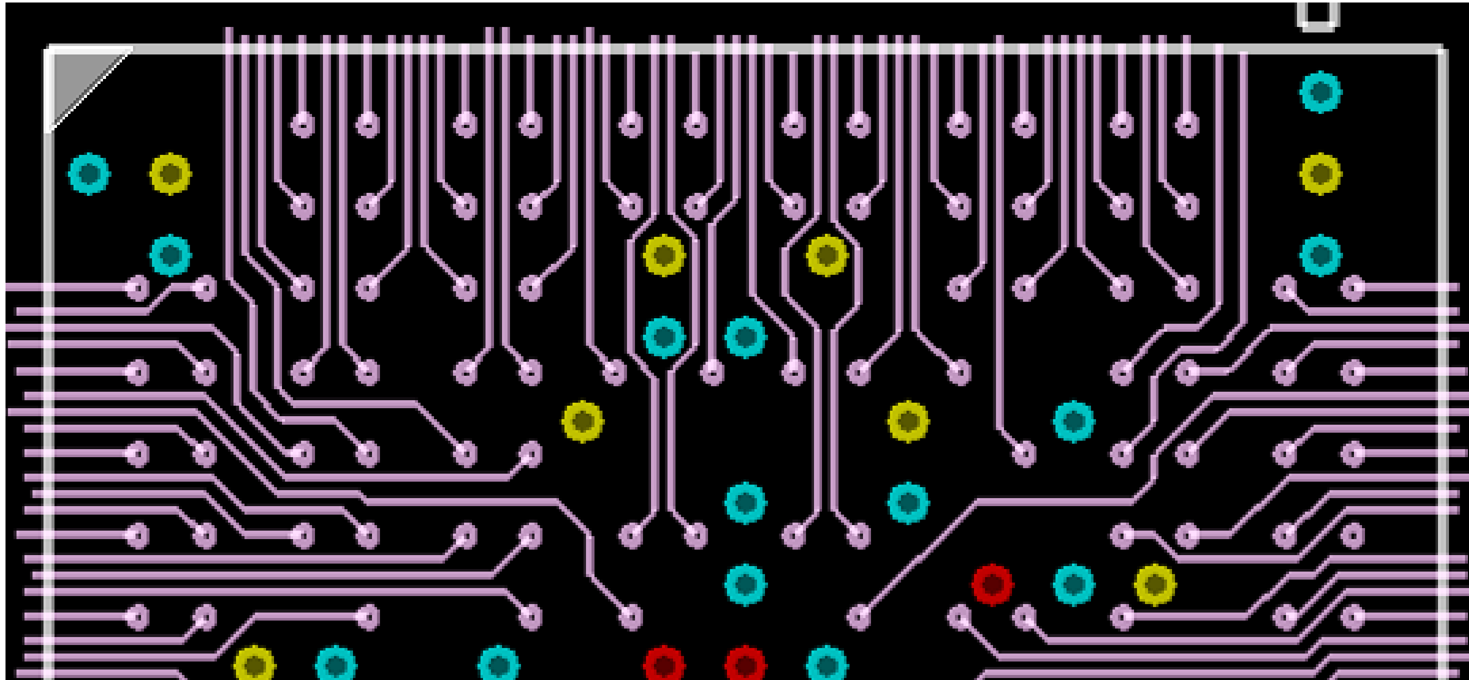
Power Delivery

- Larger power/ground copper area under BGAs with HDI also means better inter-plane capacitance and potentially better power delivery



More routing on internal layers

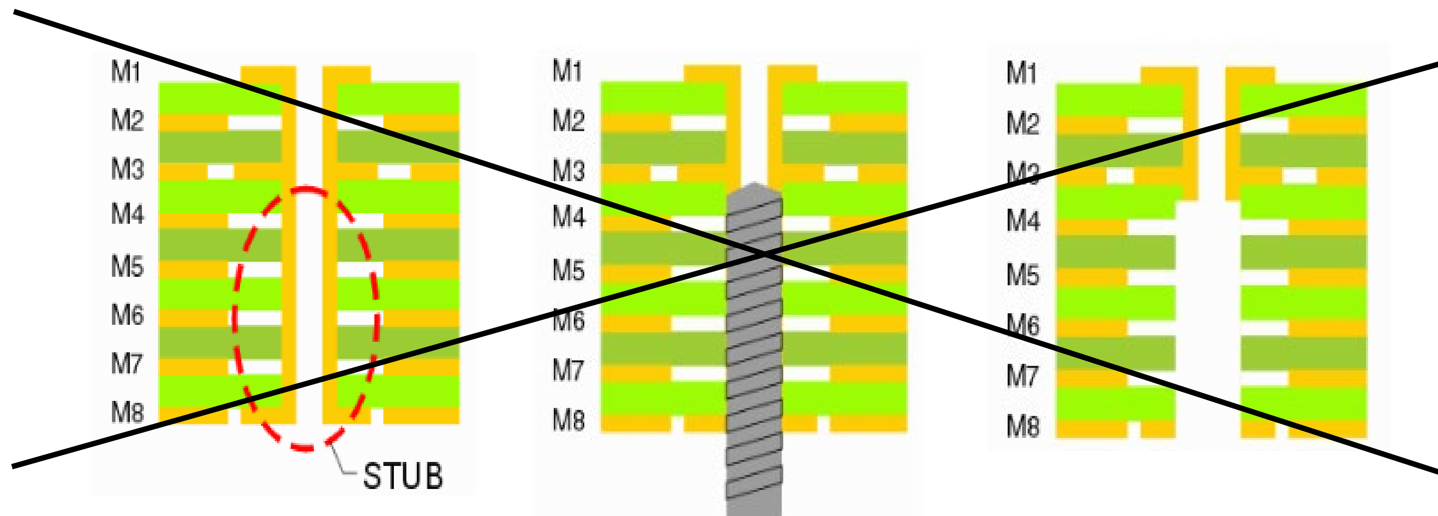
- More signals benefit from routing on inner layers – good for signal integrity, trace shape, impedance control



Altium

No Via Stubs

- uVias only go where needed
- No controlled depth/backdrilling necessary
- No backdrill obstructions/clearance
- Reduced inductance in via barrel

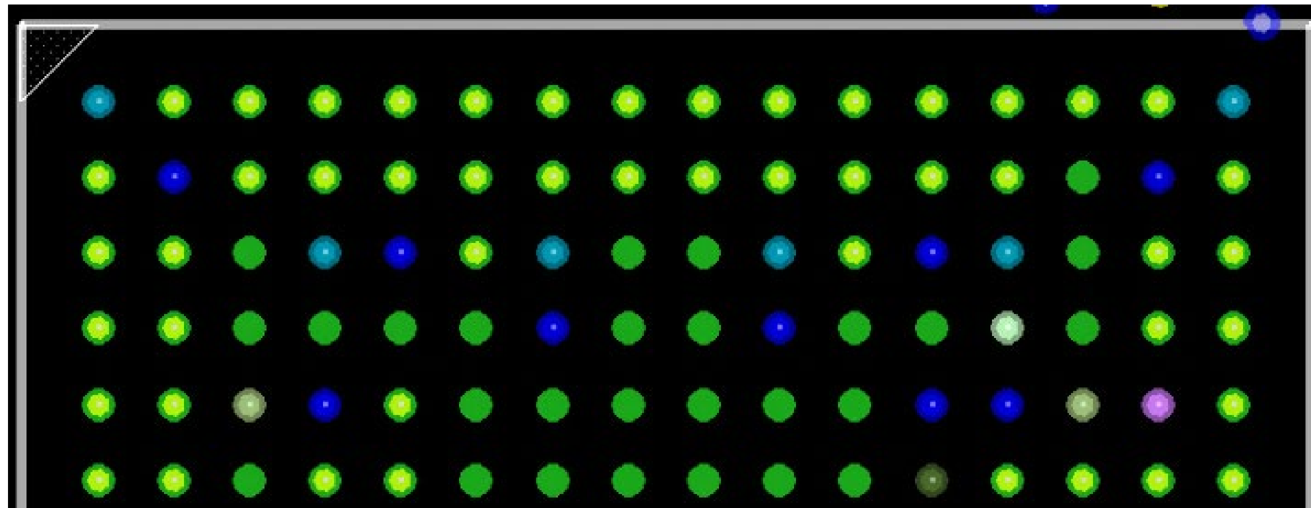


Pictures of ‘Backdrilled’ TH vias from Netex-0 “Extracting Geometry, Nets, and Components from ODB++”

Altium

uVIA in PAD better than TH Via in Pad

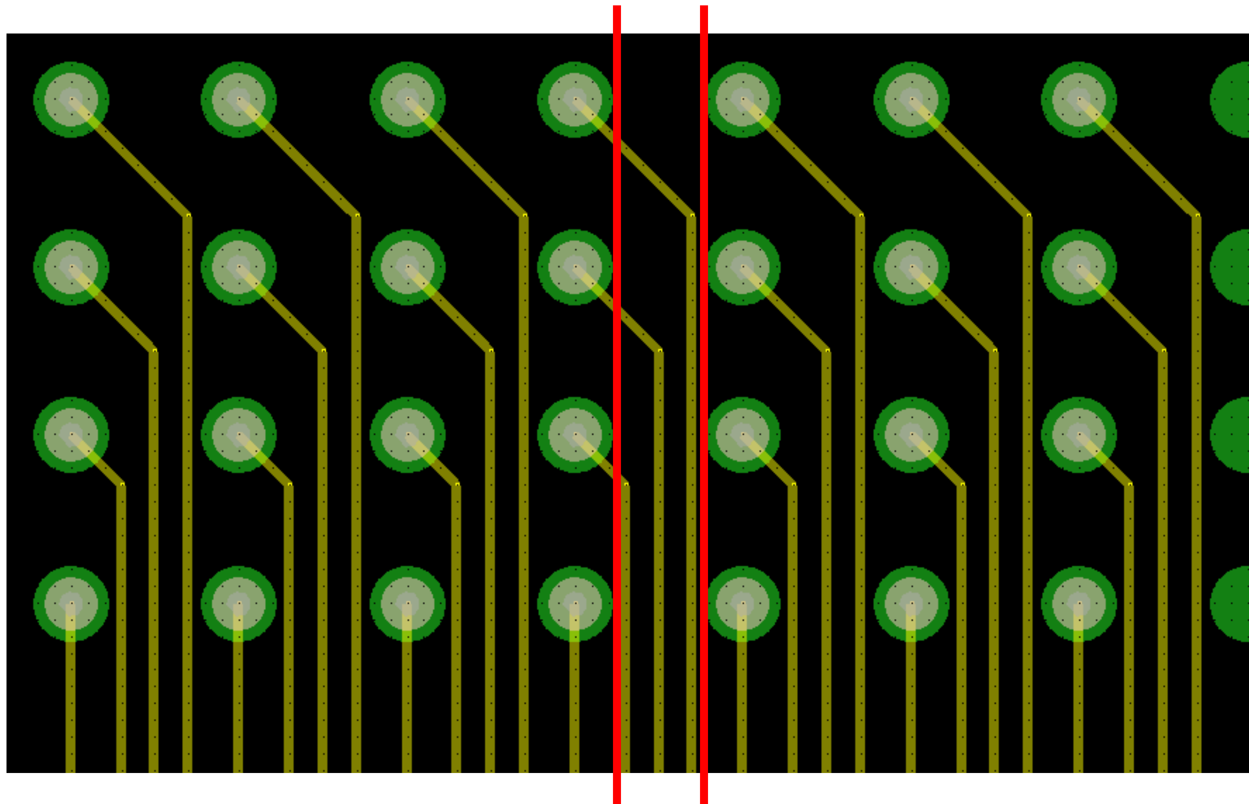
- SM for uVia is within the SM for pad so no extra SM opening needed as in dog bone
- Reduced inductance – connection is made right from ball to pad to via (w/o dog bone) and down to other layers.



Altium

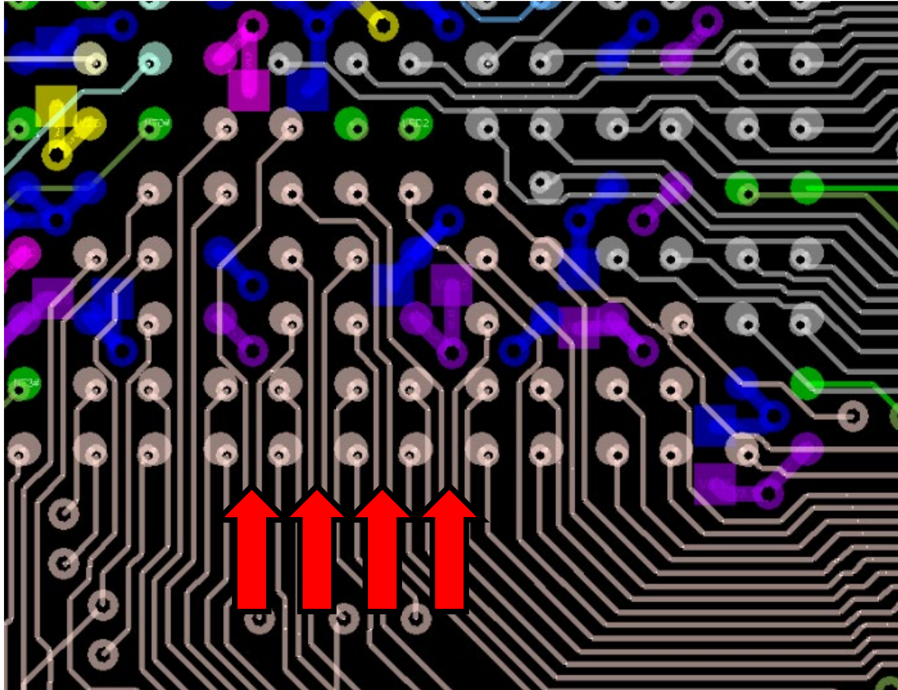
Via in Pad also means smaller antipad openings to avoid for routing return

- Signals can route slightly closer to pads



Benefits

Altium



An offset uVia grid can add extra routes

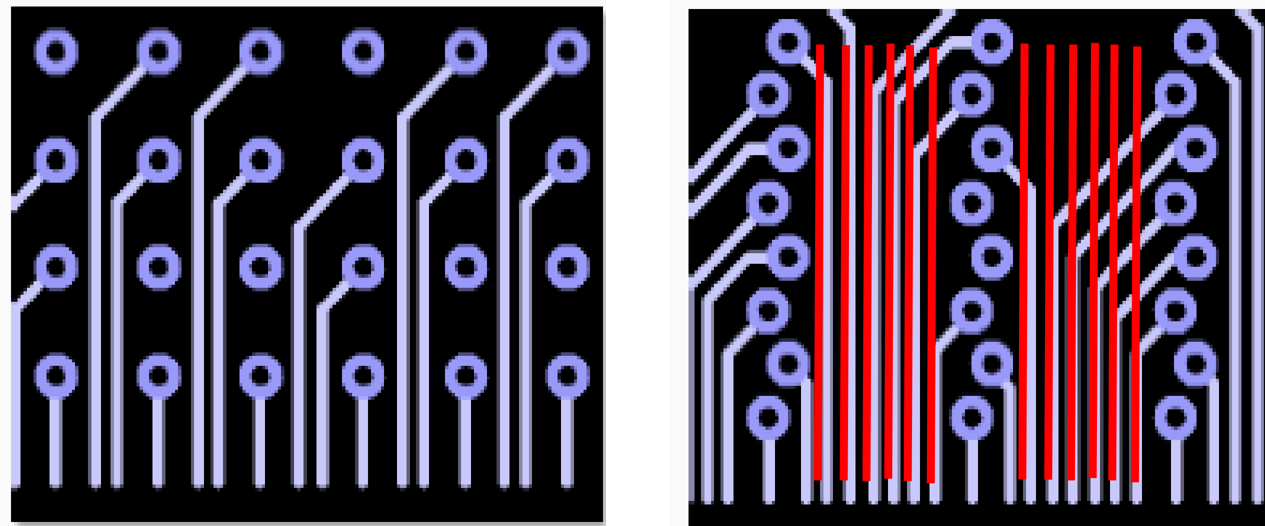
- Helps any size part be more routable

- HDI vias can be centered in, offset from, or tangent to surface mount pads to set up routing channels

Altium

HDI Routing Channels Improve Efficiency

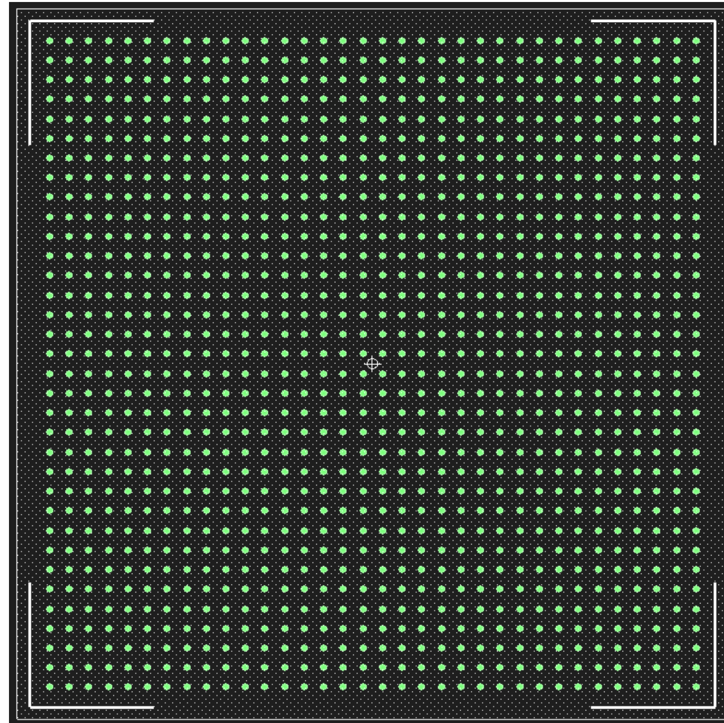
- Channels might be set up very differently to fan out a small, very fine pitch part vs a large or very fine pitch part
- Small parts may just need a path for all signals



Reprinted with permission from BGA Breakouts & Routing by Charles Pfeil

Altium®

HDI can make difficult parts routable



TI GTM (N2377) BGA with 2377 pins at 1mm pitch

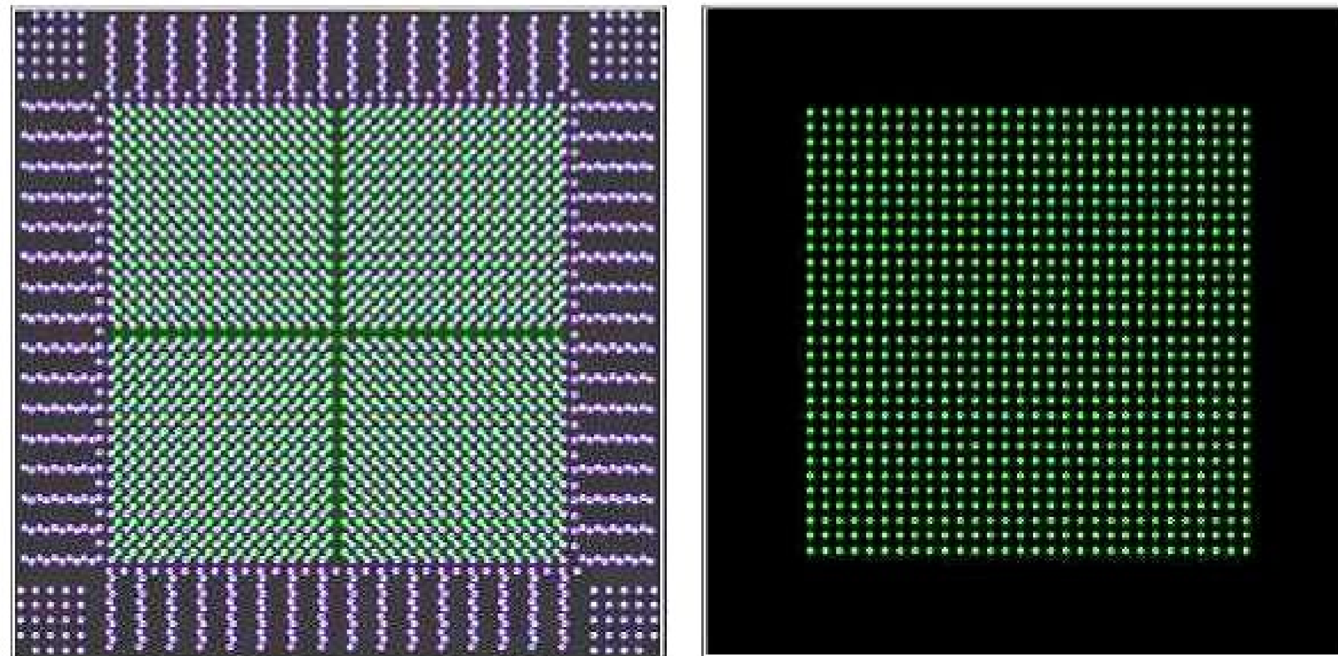
- Ability to fanout large, high pin count packages
- No way to route with through hole
- Or device would need too many TH routing layers

Altium

The Advantage of Channeled Fanouts

With good fanout patterns, you can effectively reduce the size of a large BGA array for routing

With HDI, 1760 pins effectively reduced 41% to 1024

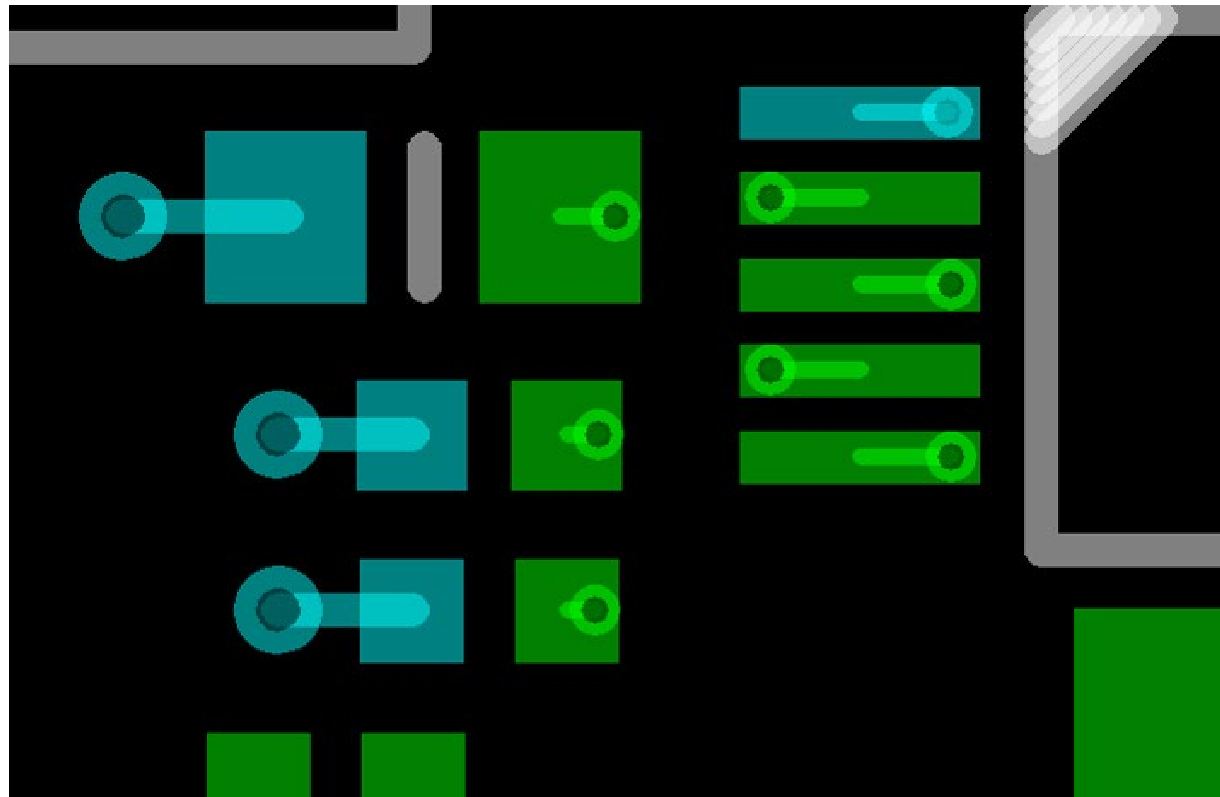


Reprinted with permission from BGA Breakouts & Routing by Charles Pfeil

Altium

uVias are not just for BGAs

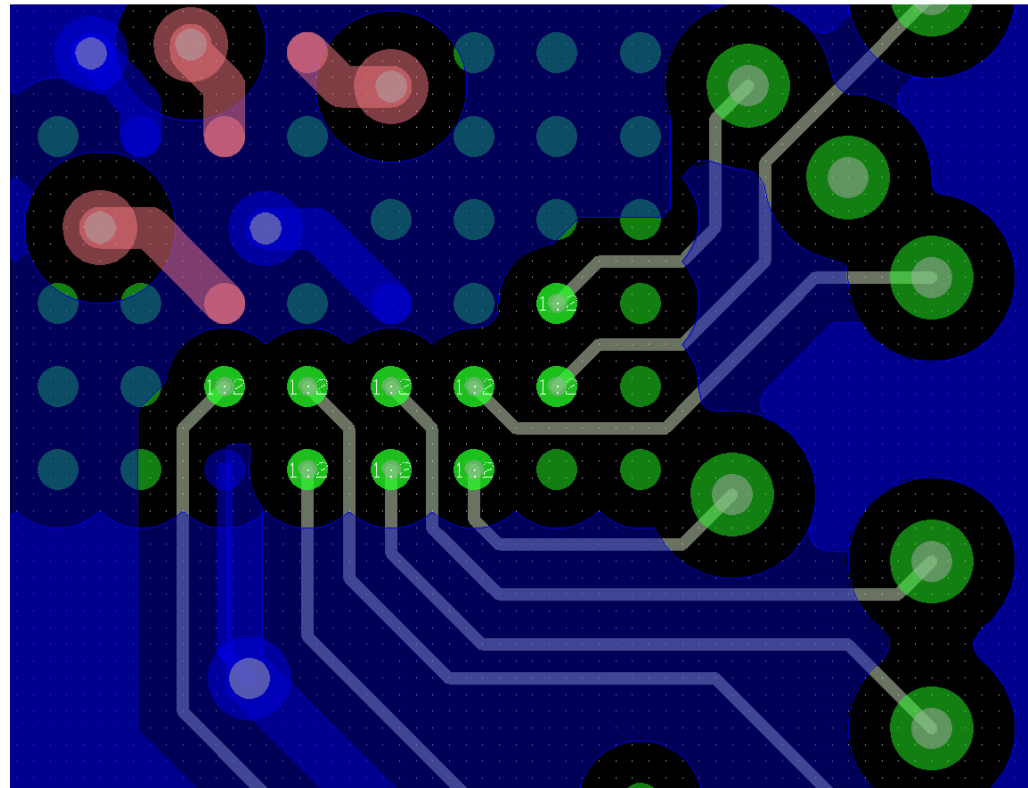
- Via in pad can also help move parts close together
  - Signals may be shorter and timing better
    - Possibly smaller board as well



Benefits

Altium

When already on the board, uVias can be used in congested areas to lead signals out of BGA to an open area, and from there to TH or buried vias

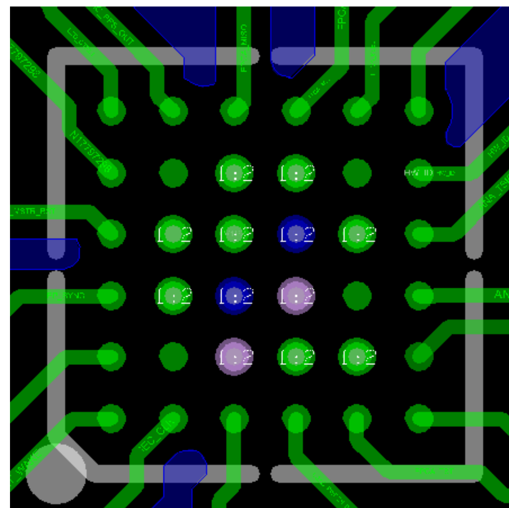


(.5mm part)

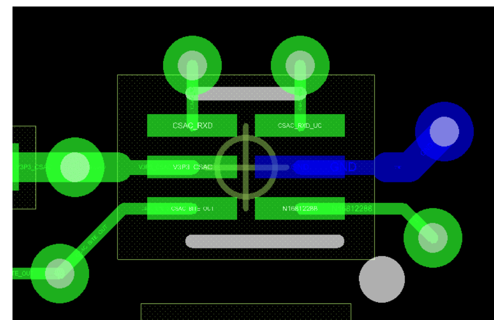
Altium®

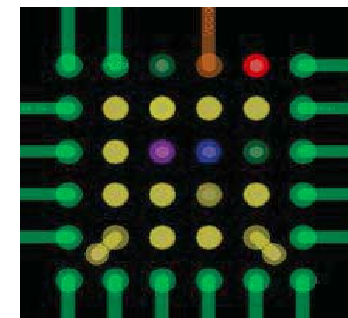
Finer pitched parts can be used with HDI

- Some of the new parts are only available in small BGA packages
- Other fine pitch devices have very little room for all the large TH vias needed nearby



0.4mm BGA

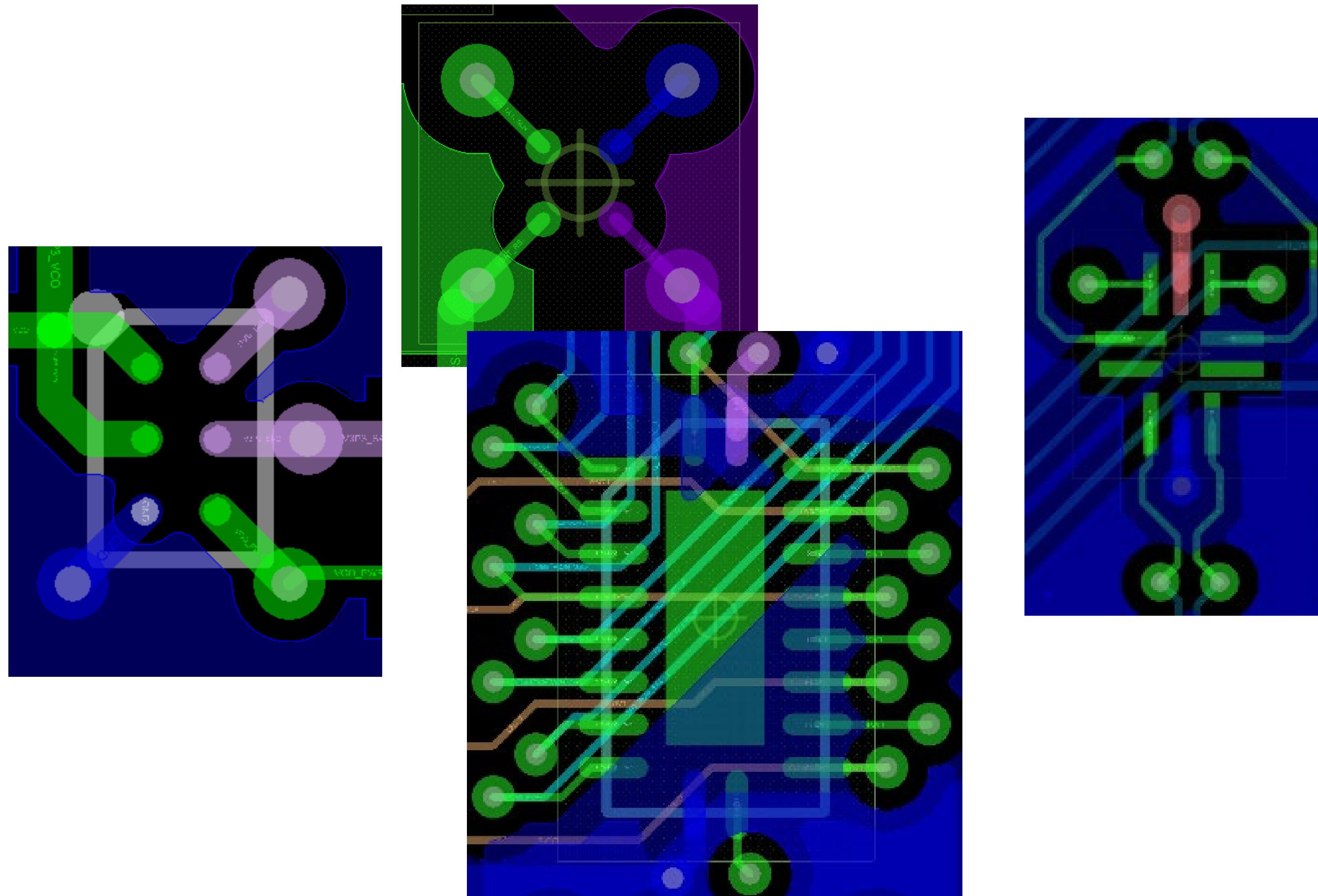




0.35mm BGA

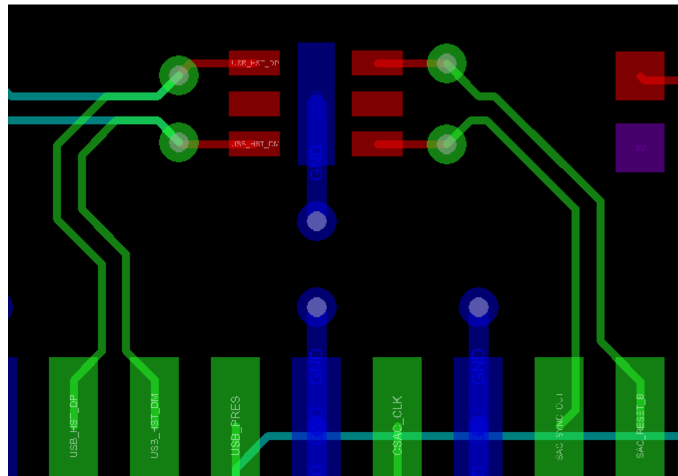
Altium

Some parts are just complex to route with TH vias



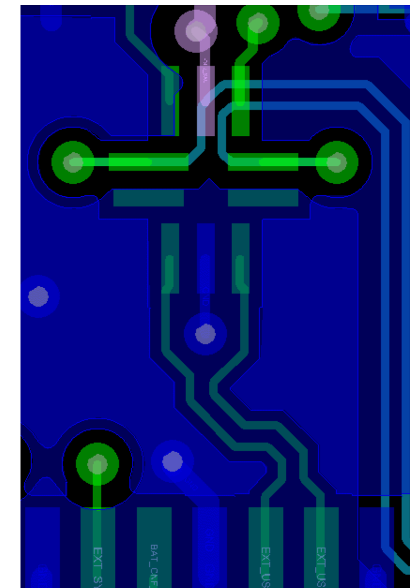
Altium

Some parts are just complex to route with TH vias



- uVias would help with ESD routing

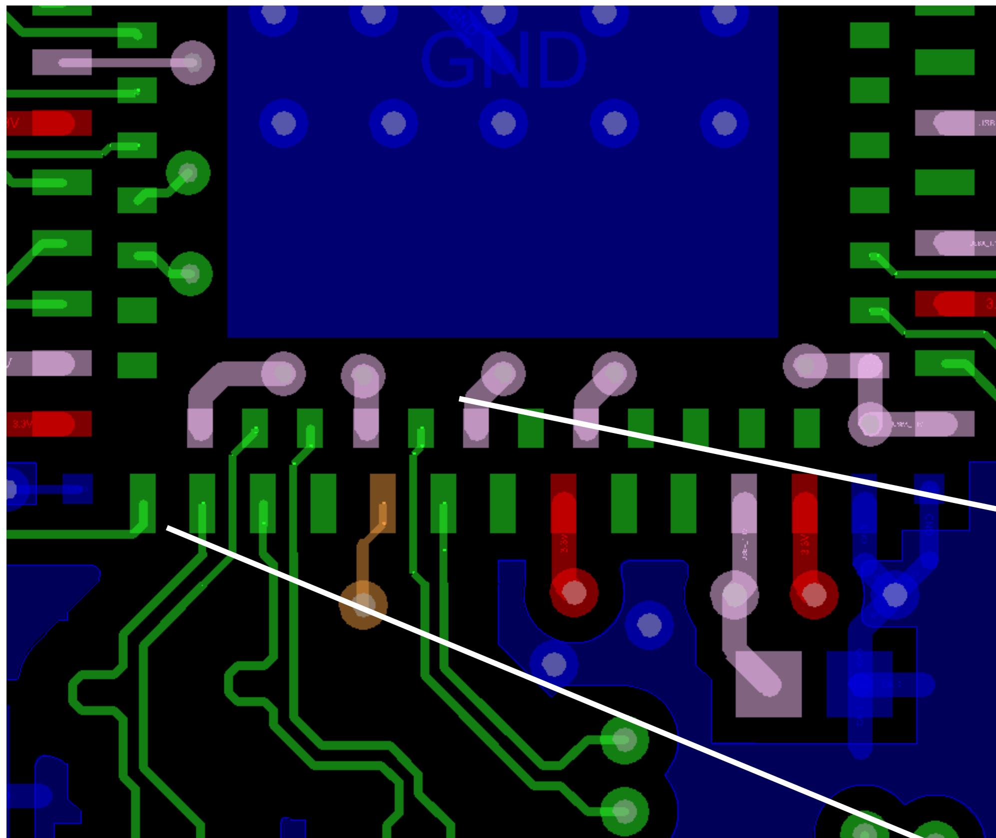
- uVias would help with differential routing on newer style ICs



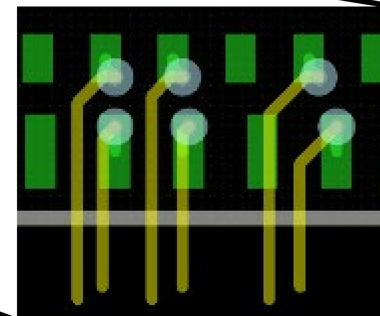
TI_TS3USB30ERSWR

Altium

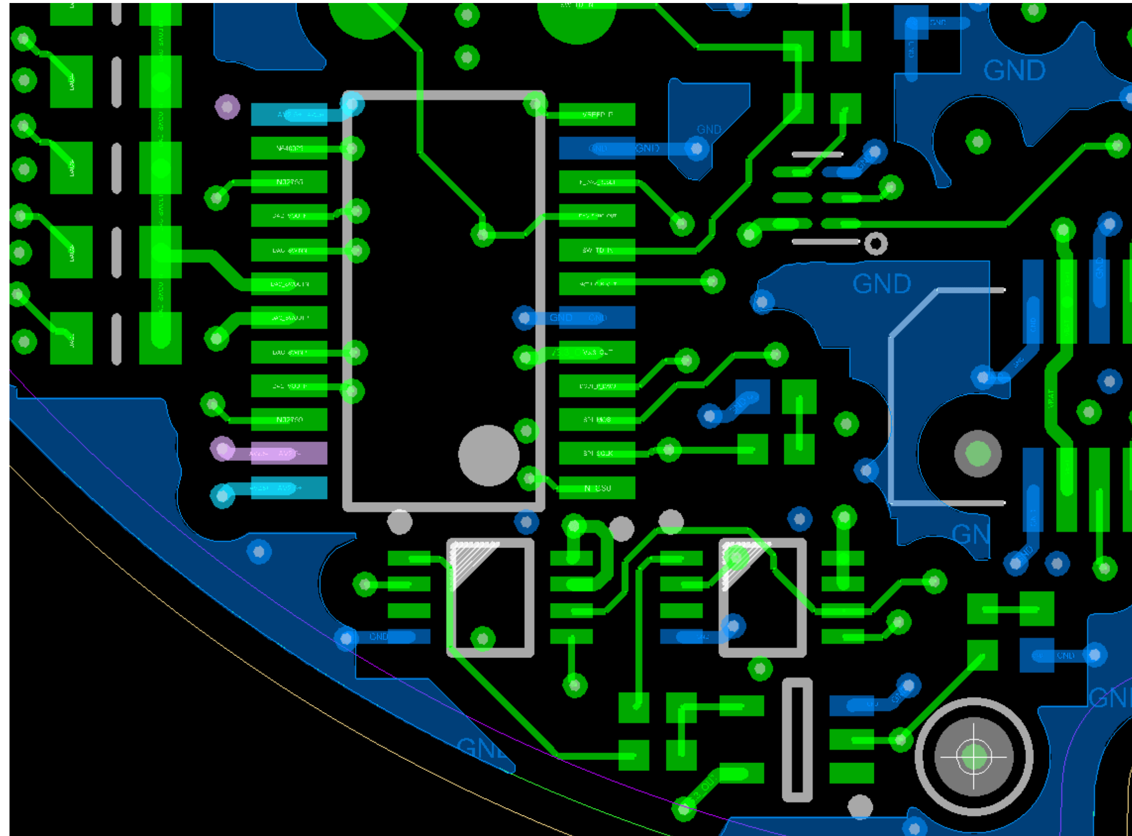
Some parts would benefit from using uVias



- Here, uVias help diff pairs move to inner layer, closer, & away from tab



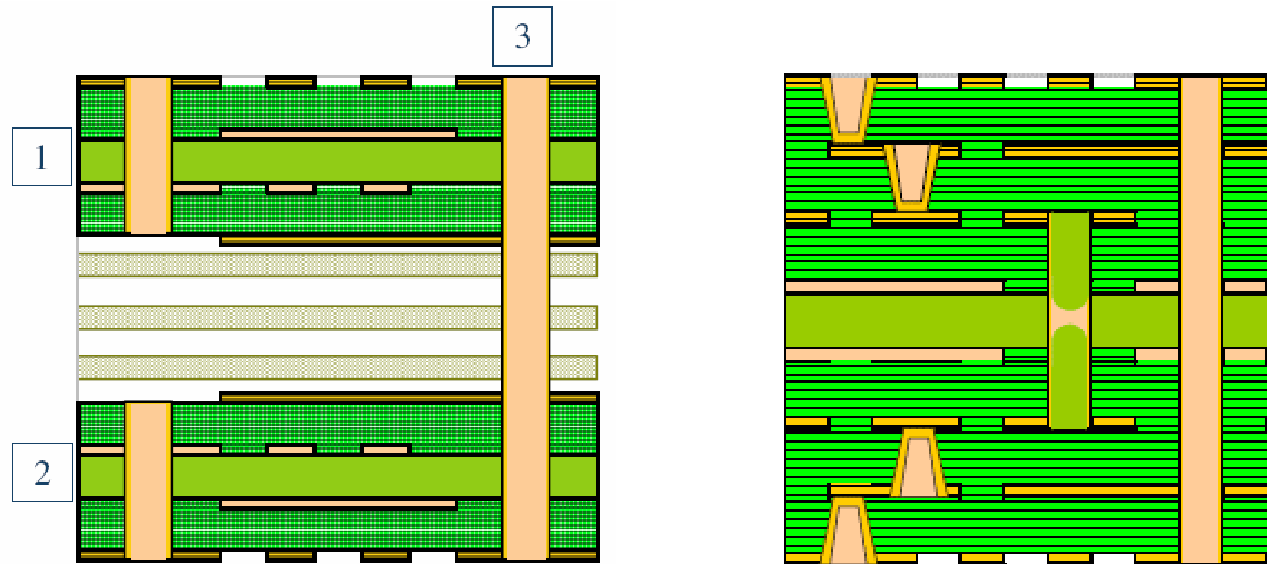
Some boards or areas would benefit from uVias because of P&R density



Altium

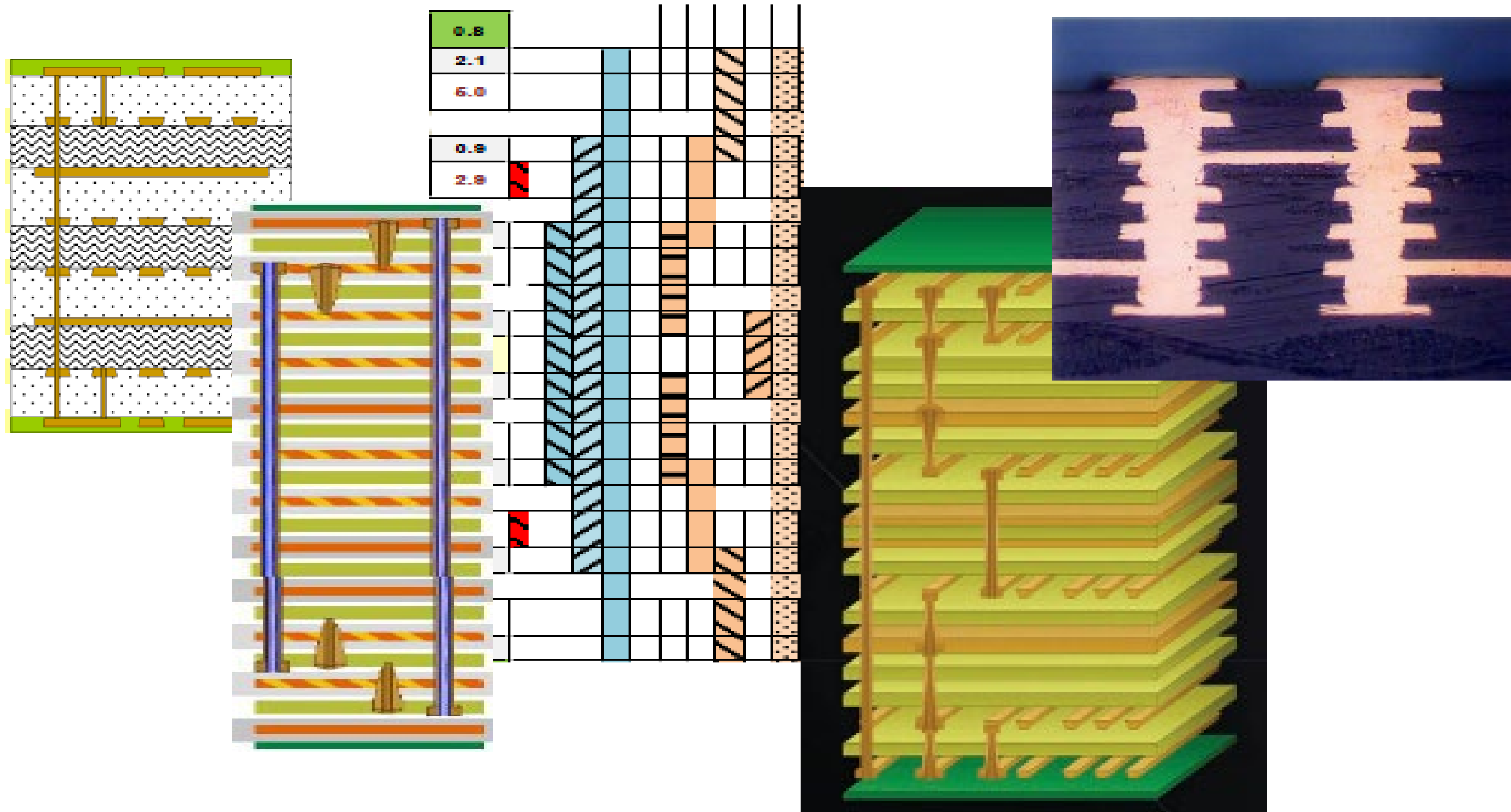
Copper filled/planarized uVias allow for active parts to easily be placed on both sides of the board

- Easy to fan out parts on their ‘own’ layers



Altium

HDI Provides the largest variety or stackup possibilities



- Via patterns can stack and stagger through many layers

Altium

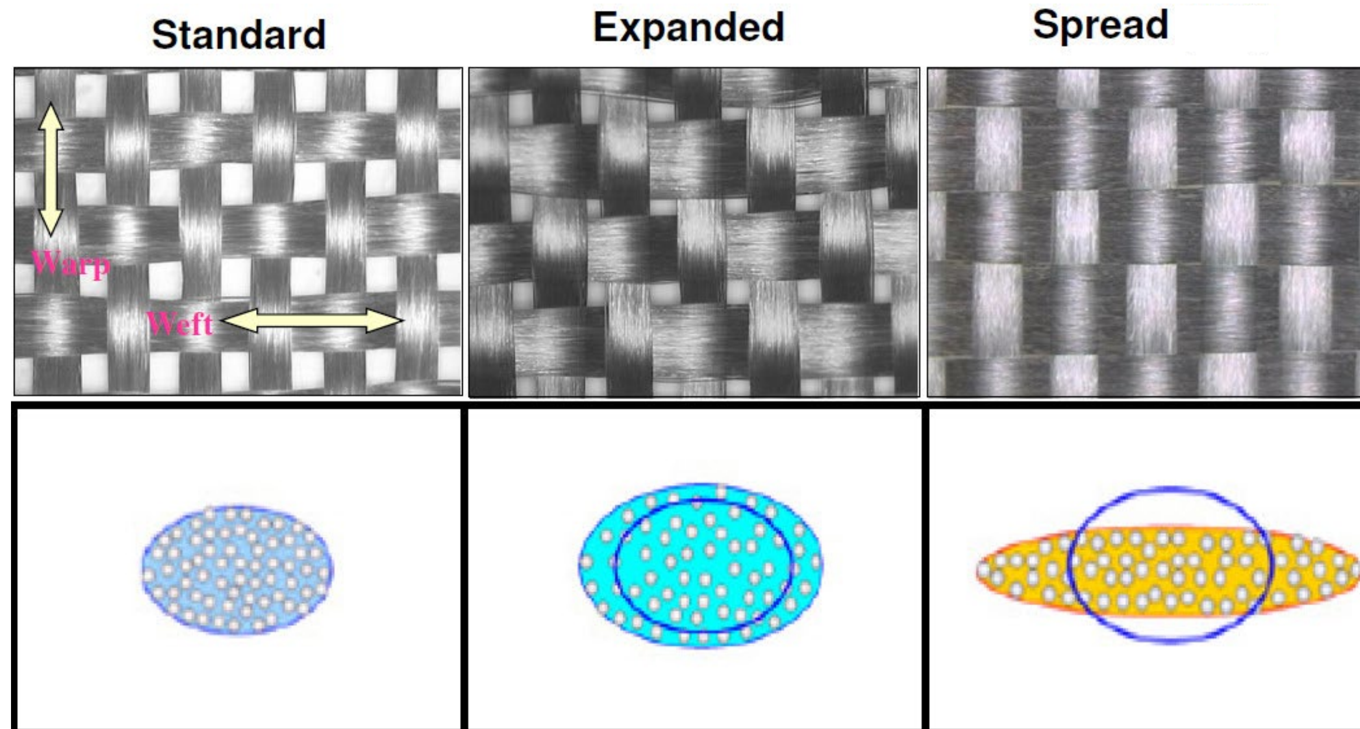
HDI can affect Thermal Management

- Consider thermal transfer when building stack
- The primary heat transfer is accomplished through **conduction**
- The tighter we place parts, the more heat
- The smaller parts often produce more heat
- Solid copper fill in uVias helps to improve heat transfer layer to layer
- Newer thin materials may also improve thermal performance*

*Happy Holden “HDI’s Beneficial Influence on High Frequency Signal Integrity”

Flatter Weave Materials Available for HDI

- HDI's spread material may be helpful to all signals' quality – particularly HS or differential pairs



Design for low cost

- Keep to Type I or Type II, if possible – fewest laminations
- Use least depth of uVias – no skip vias
- When changing layers, staggered uVias are easier to produce than stacked
- If HDI is used on the board, it usually does not cost extra to use in many other places
- You can add microvias to a through hole board…. But it's still a TH board, just more expensive*

*Happy Holden in "Current PCB Cost Adders"

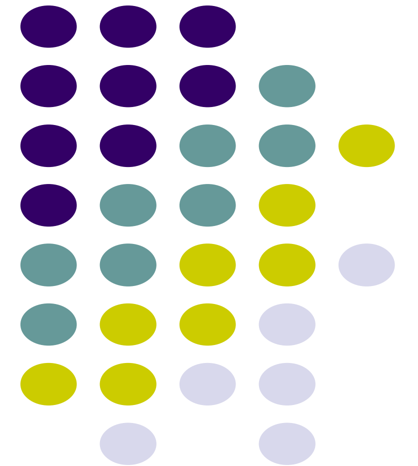
Fabrication Issues

- Get fabrication involved early!*
- **Check with your fabricator** for his norms before starting a board - capabilities, up-charges, turn times, etc.
- Think about fabrication yield… HDI may cost a bit more, but that may beat the cost of a TH board that is difficult for the designer to design and/or for the fabricator to build!
- **Consider what is most cost effective from a DFM point of view**

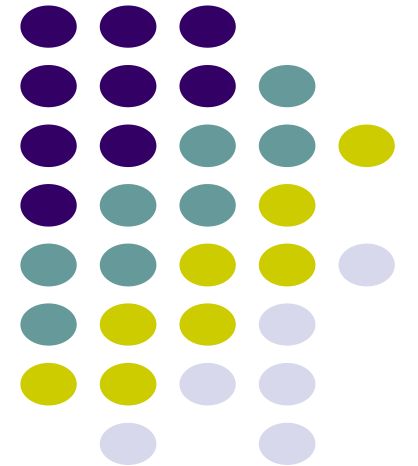
Thank you!

Susy Webb
DesignScience@ymail.com

Benefits to Board

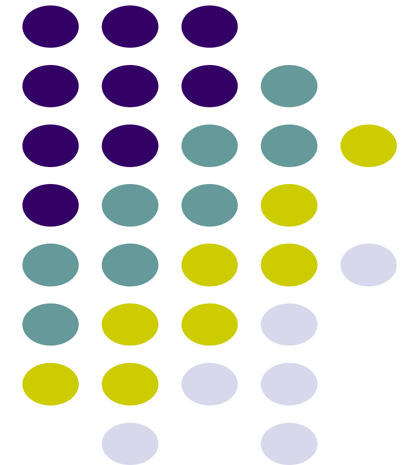


Benefits to Board



Benefits to Board

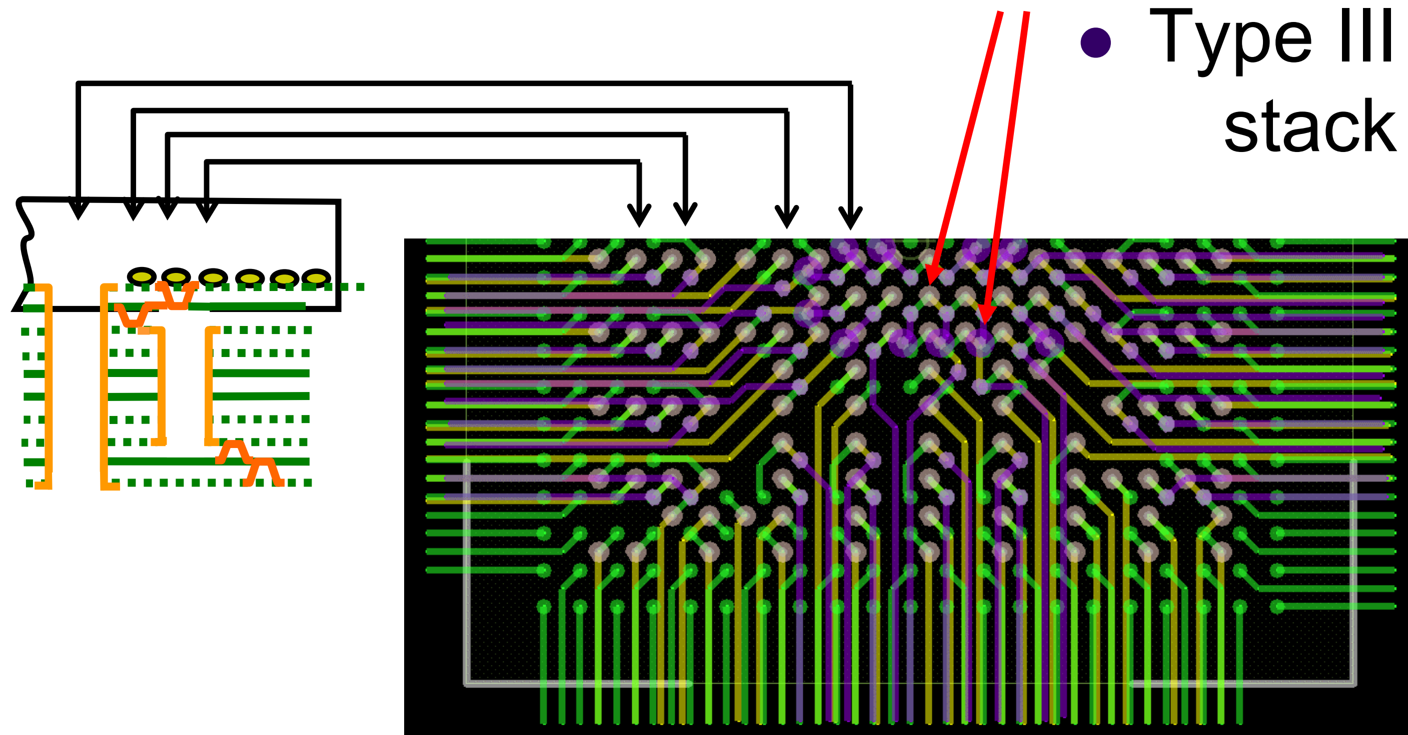
Many designers understand the help that HDI technology can give to the fanout of a BGA, but there are many other benefits to consider as well. Some may feel like uVias may be cost prohibitive, so we will talk about how the technology can affect the general efficiency of a board, counteracting that cost. The routing and stackup possibilities expand rapidly with HDI providing new resources, and we will discuss how it provides some true benefits for signal integrity and EMI too. And while some parts are fairly awkward to route with through hole vias, they can be much easier to design with uVias. All this and a few helpful manufacturing tips will be discussed in this presentation.

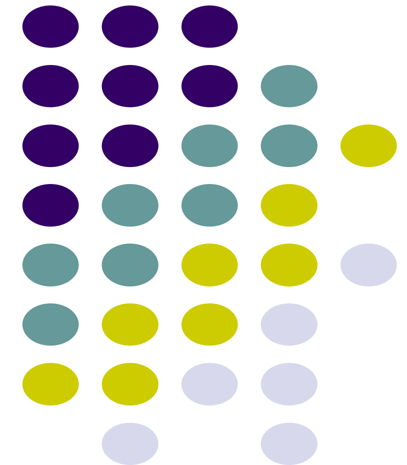


Efficient signal transition from layer to layer

Here, a Plan with 'rough-in' routing for a BGA with several types of vias

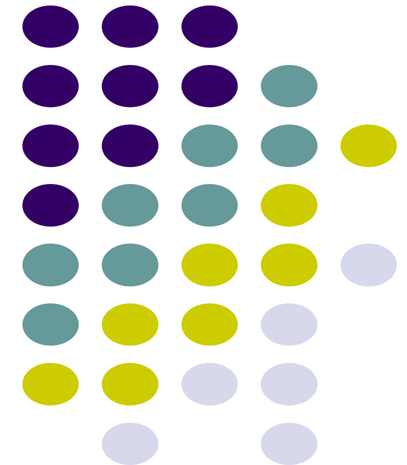
- Type III stack





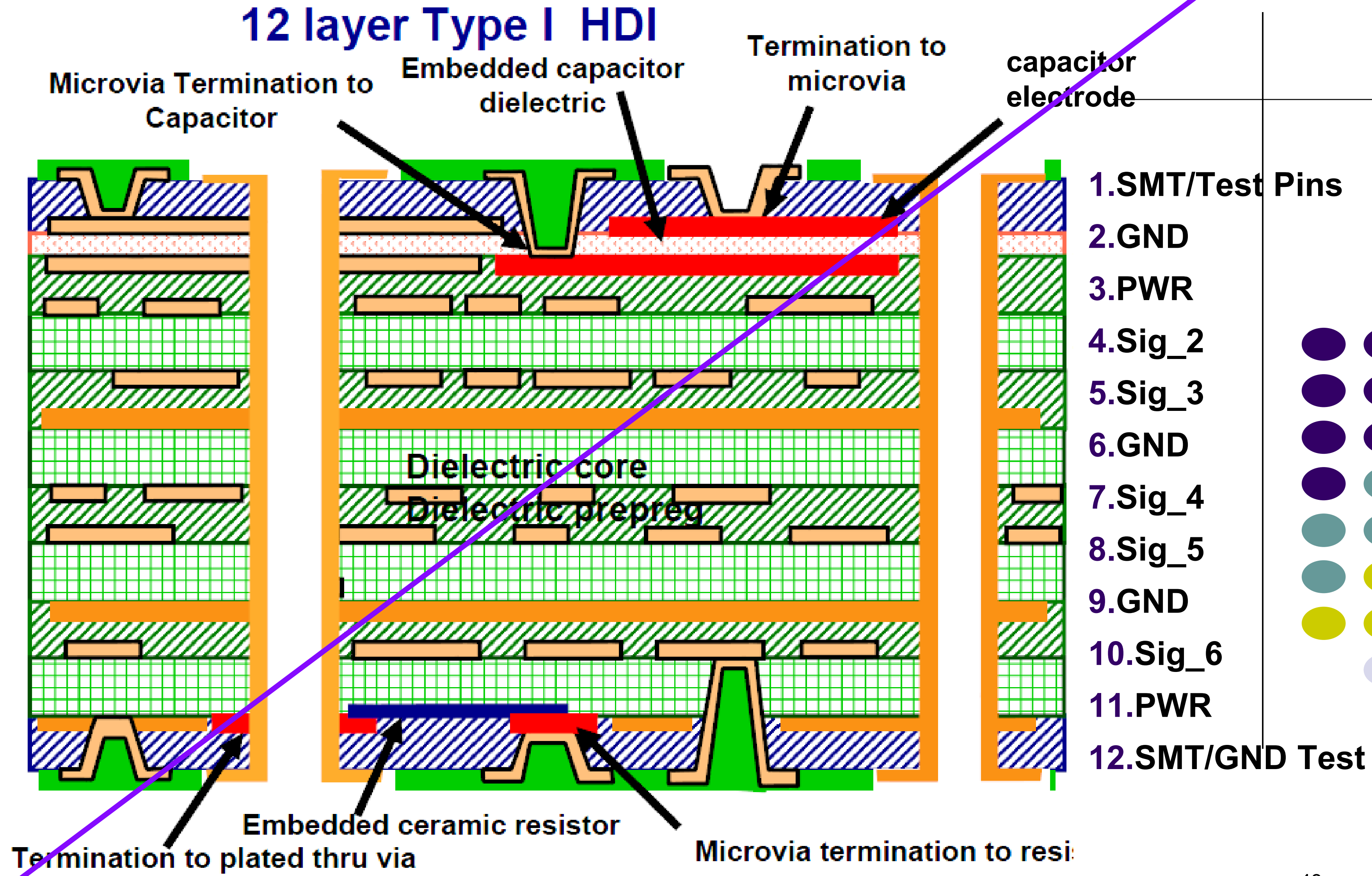
Design Priorities – What is most needed on the board. Will HDI help with that?

- Major amounts of routing
- SI and EMI control
- Good power delivery
- Signals, power and ground flowing throughout board
- Low Cost
- Board thickness issues
- ALL of the above



Benefits

HDI can affect Embedded Passives



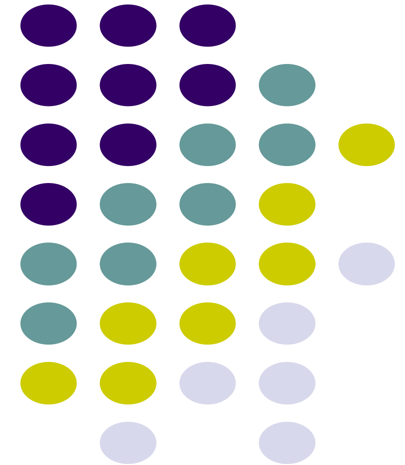
*Happy Holden – “HDI/Microvia Technologies”, PCB East 2009

Cost equality

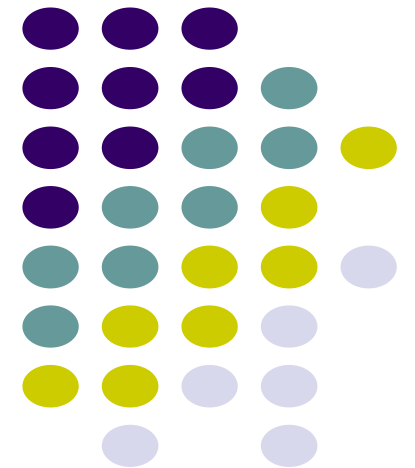
Benefits to Board

Price/Density Comparison

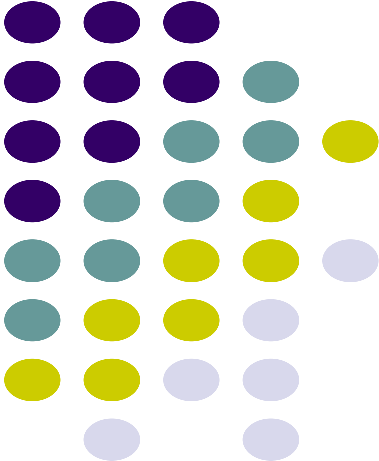
- Possible cost equality - talk with **your** vendor about everything that your board will need
- 8 layer TH board is the base for comparison
  - An 8 layer TH board may reduce to a 4 layer Type I board for **lower** RCI
  - A 14 layer TH board may reduce to a 8 layer Type II board for **lower** RCI



Benefits to Board



Benefits to Board



READY for Altium 2018 –
except convert to their slide template
37 slides for 45 min in 2017

The Benefits Gained by Using HDI Technology on Boards

Susy Webb, CID

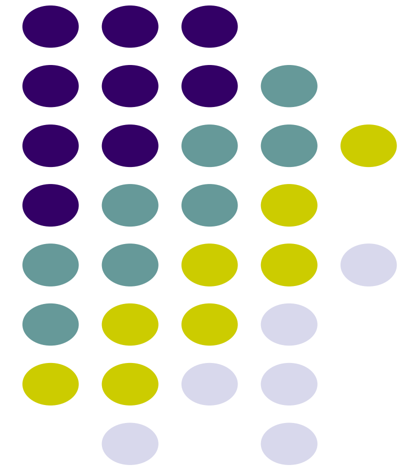
Sr. PCB Designer

Fairfield Geotechnologies

Houston, Texas

© Design Science 2018

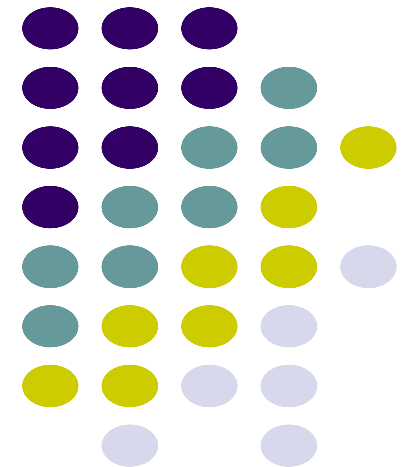
Email DesignScience@Ymail.com



The information herein is partially the opinion of the presenter and others, and is not considered applicable in every PCB situation

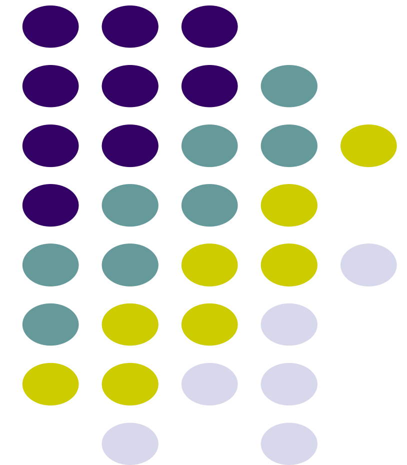
HDI/ Microvia – What's the difference

- IPC: **uVias** are laser vias of generally 6 mil drill (150 microns) or smaller
- **HDI** is the **technology** used with microvias:
  - Smaller holes and padstacks than TH
  - Smaller traces and clearances
  - Thinner Dielectrics
  - Finer aspect ratios
  - Etc.



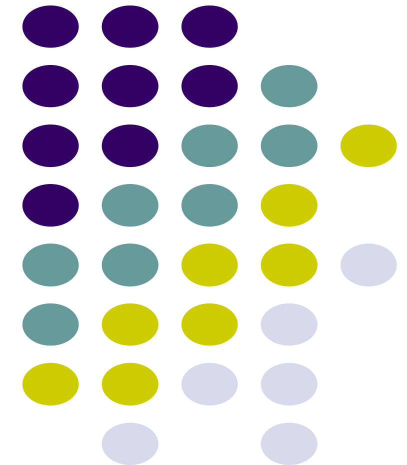
When to make the jump to HDI:

- IC packages may need it – either high pin count or fine pitch uBGAs (.65mm and below)
- Large parts with lots of connections
- Lack of room - Small physical board size
- Using minimum holes and line widths/spaces and still not enough room for all
- Amount of time needed/allowed to design the board
- When uVias will add other needed benefits



What HDI can Offer

- Increased design flexibility
- Theoretical Cost equality
- Improved reliability – HDI tested as the most reliable for organic substrates*
- Better for EMI and signal integrity*
- More creative fanout possibilities for all parts
- Efficient signal transition from layer to layer
- Often can be manufactured worldwide – large market for fab



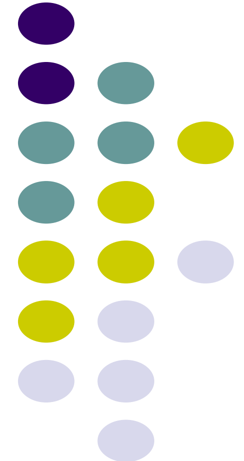
*Happy Holden – “HDI/Microvia Technologies”, PCB East 2009

Possible cost equality - Price/Density Comparison

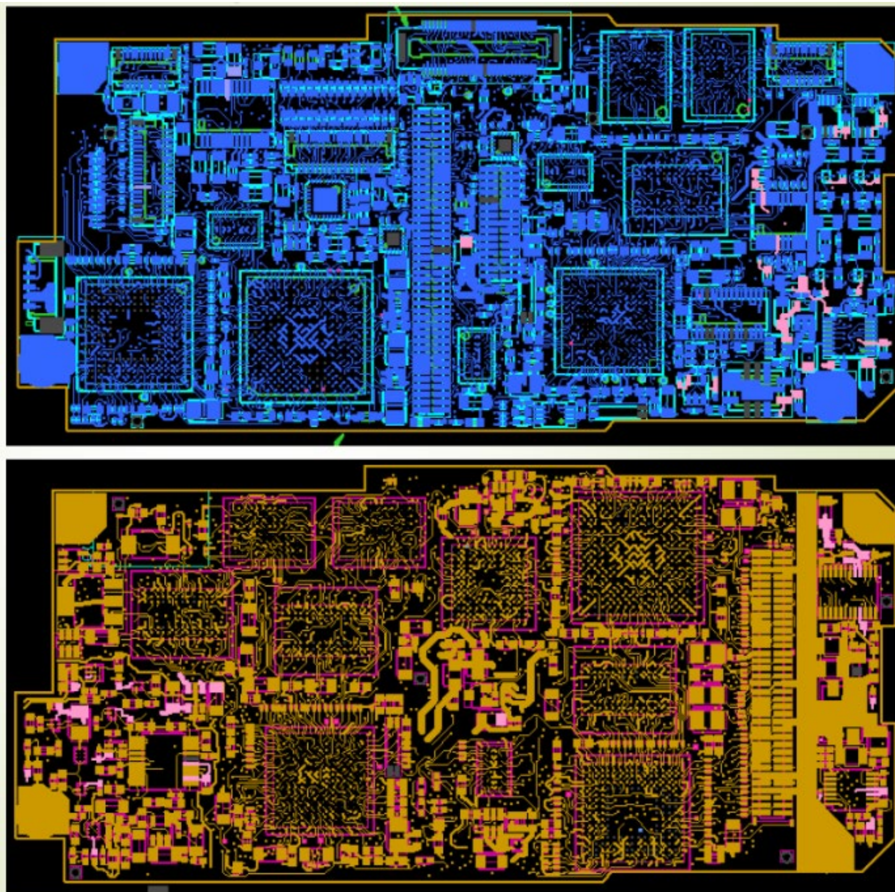
N Layers	A THRU-HOLE N blind via* buried via		B HDI BLIND 1+N+1 L1-L2 none		C HDI BL/BU 1+bN+1 L1-L2 L2-L(N-1)		D 1BU BLIND 1+N+1 skip via L1-L3 L2-L(N-1)		E 2BU BLIND 2+N+2 staggered L1-L2, L2-L none		F 2BU BL.BU 2+bN+2 skip via L1-L3 L2-L(N-1)		G 2BU BL.BU 2+bN+2 staggered L1-L2, L2-L L3-L(N-2)	
	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN	RCI	DEN
4L	0.62	20	0.83	40	1.05	80	1.26	135	1.38	135	- -	- -	- -	- -
6L	0.78	20	0.99	60	1.24	160	1.46	200	1.60	200	1.74	260	1.91	280
8L	1.00	30	1.21	120	1.49	180	1.74	240	1.90	240	2.06	300	2.25	320
10L	1.30	40	1.51	200	1.83	210	2.11	260	2.30	260	2.50	400	2.73	440
12L	1.70	60	1.92	210	2.31	230	2.62	300	2.85	300	3.10	600	3.37	650
14L	2.24	70	2.48	220	2.95	250	3.32	360	3.61	360	3.91	800	4.25	860
16L	2.97	80	3.22	260	3.81	300	4.25	420	4.61	420	5.00	1000	5.43	1100
18L	3.92	100	4.21	300	4.95	400	5.47	480	5.93	480	6.42	1250	6.96	1350
20L	5.14	105	5.48	360	6.41	500	7.04		7.62		8.23		8.90	
22L	6.67	110	7.08	400	8.23	600	8.99		9.70		10.45		11.27	
24L	8.53	125	9.03	460	10.41	700	11.32		12.17		13.07		14.04	
26L	10.68	130	11.30	500	12.92		13.96		14.96		16.00		17.11	
28L	13.09	135	13.83	540	15.65		16.82		17.93		19.09		20.32	
30L	15.63	140	16.50	580	18.47		19.73		20.94		22.18		23.48	
32L	18.17	145	19.17	620	21.21		22.53		23.79					
34L	20.59	150	21.69	660	23.73		25.09							
36L	22.79	160	23.96	700	25.94									
38L	24.68	180	25.91	740										
40L	26.26	200												

RCI = Relative cost index, DEN = Pins per sq. in.

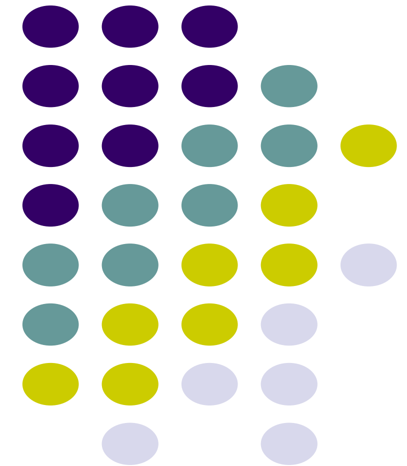
Picture reprinted from Happy Holden



HDI may cost more, but fewer Bd Layers may make up for it

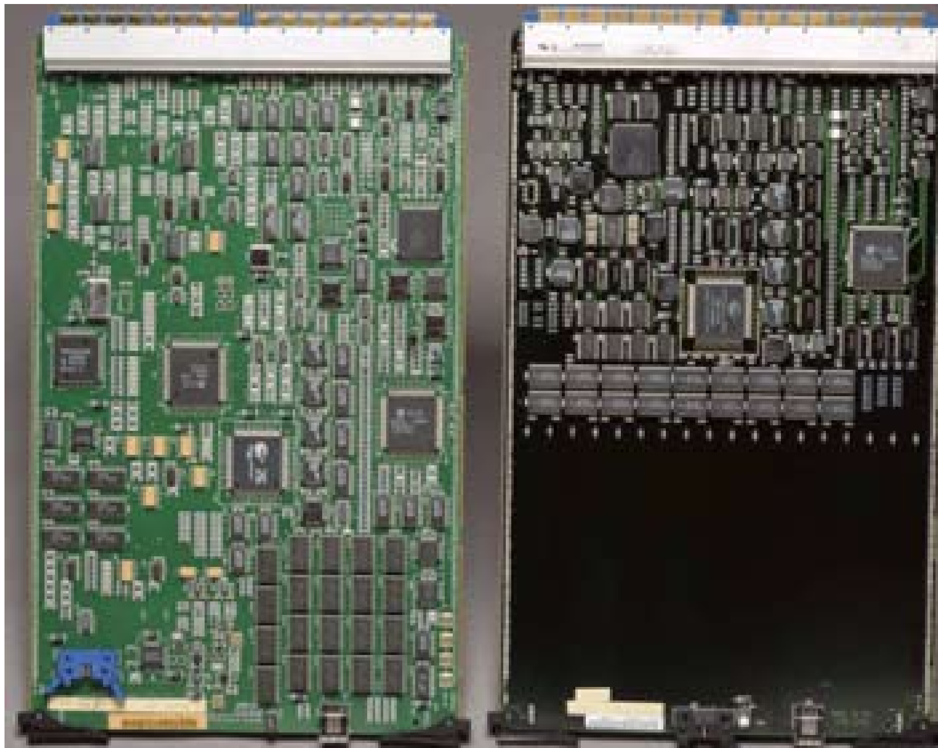


- Example goes from 16 layers TH and BB to 12 layers w/HDI
- Much better for hole aspect ratios

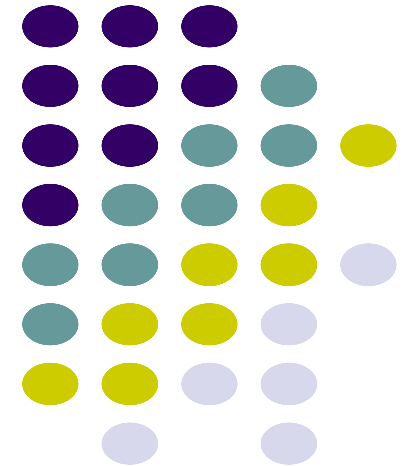


Part of a slide from *Happy Holden – “What you need to learn to gain 3X to 4X higher Layout Density”

HDI may cost more, but smaller board size may make up for it



- Good if small physical board size needed
- Less board material needed
- Smaller board is easier to panelize



*Happy Holden – "HDI/Microvia Technologies", PCB East 2009

Routing Efficiencies per type of board

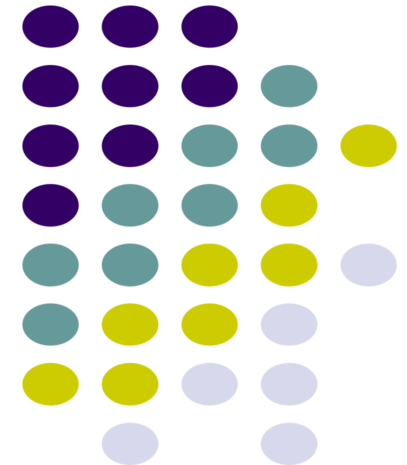
A measure of the total # of traces vs the total number possible (efficiency %)

Design Scenario	Conditions	Efficiency (ε)
Rigid Thru-Hole	Gridded CAD	30%
Rigid Thru-Hole and SMT	With or W/O Back Side Passives	35-50%
Rigid Thru-Hole and SMT	With Back Side Active Components	30-45%
Rigid SMT Only	With Back Side Active*	up to 55%
Rigid Thru-Hole and SMT	1 Sided Blind Vias*	up to 60%
Rigid Thru-Hole and SMT	2 Sided Blind Vias*	up to 65%
2 layer HDI Structure	*	up to 70%
4 layer HDI Structure	With Blind/Buried Vias*	up to 80%
6 Layer HDI Structure	With Blind/Buried Vias*	up to 75%

** = gridless CAD system*

2 layer means 2 layers of Microvias

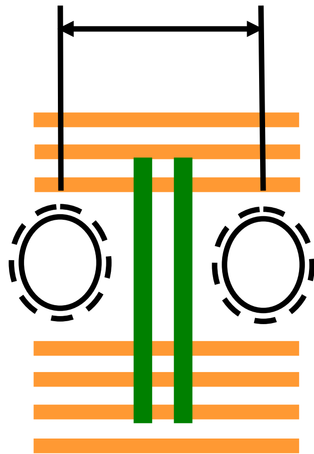
Reprinted from



HDI may cost more, but can improve routing on all routing layers

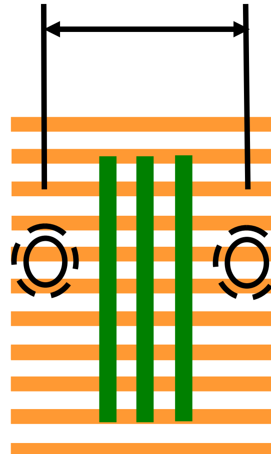
- uVias depth allows more internal signal routing, and uVia size allows for more external routing

TH 1mm (.0393”) pitch



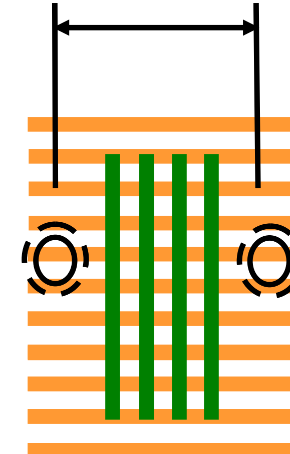
.5mm pad (@ .020”)
.25mm hole (@.010”)
.1mm lines and spaces (@.004”)

HDI 1mm (.0393”)

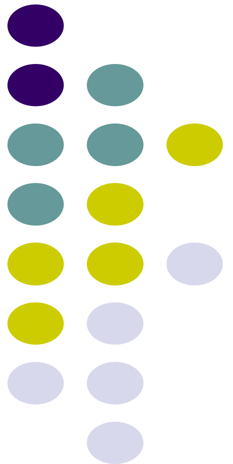


.25mm pad (@ .010”)
.125mm hole (@.005”)
.1mm lines and spaces (@.004”)

HDI 1mm (.0393”)

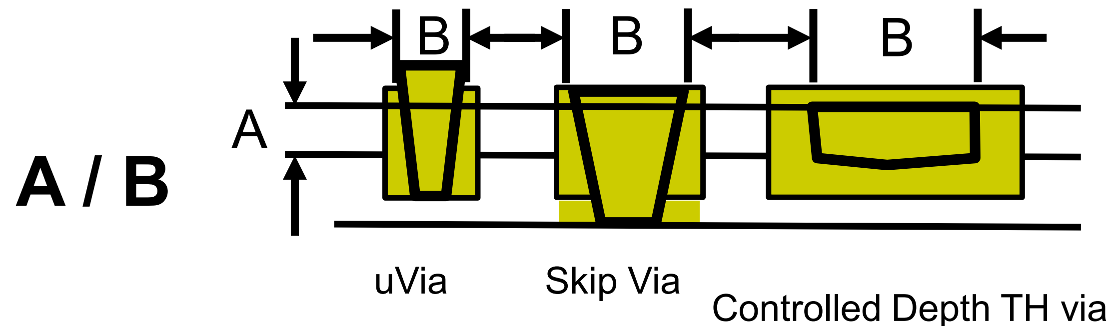


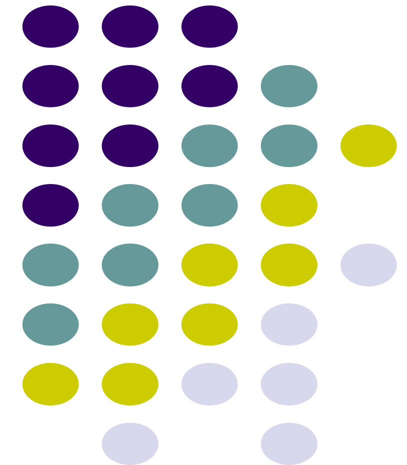
.25mm pad (@ .010”)
.125mm hole (@.005”)
.076mm lines and spaces (@.003”)



uVia Depth and Board Thickness

- HDI size works well with smaller aspect ratio for layer thicknesses
  - Smaller pads/holes take up less room
- TH aspect ratio generally 10:1 up to 12:1
- HDI aspect ratio generally 0.5:1 up to 0.7:1

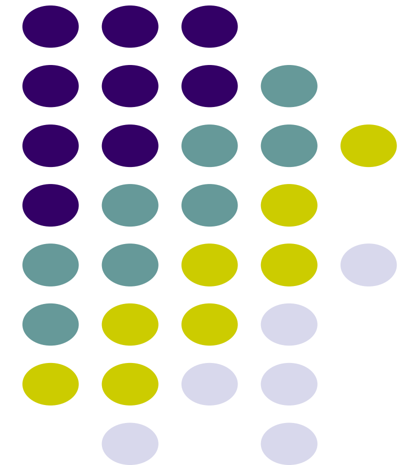




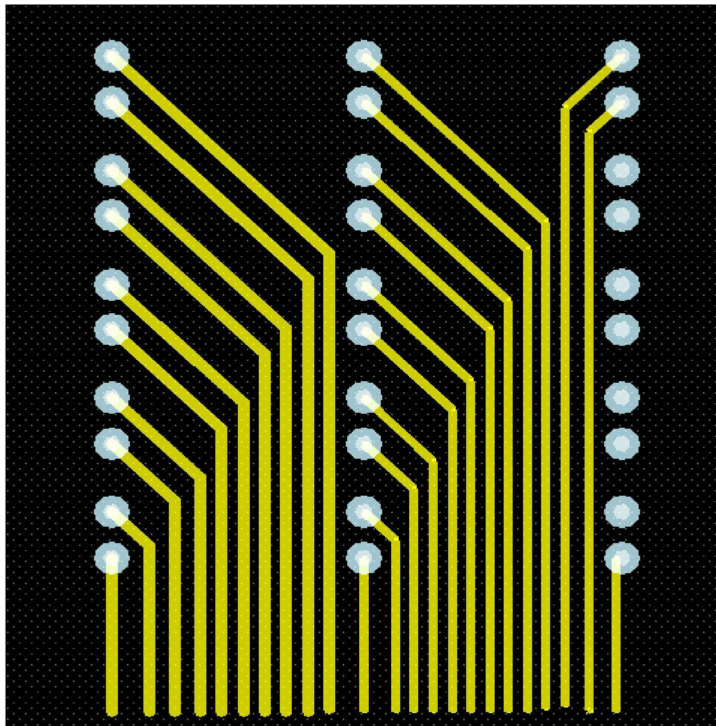
Aspect Ratio = A) Board (or layer) Thickness divided by B) Drill Diameter

Thin boards – Thin layers – Thin copper

- The thinner dielectrics used with HDI can easily make thinner boards or many layers, if desired
- Even very thin dielectrics used (.002” or less)
- Thin dielectrics may lead to thin trace widths for impedance control
- Thinner copper thickness to start is recommended for trace width/spacing of below 3/4 due to etch compensation

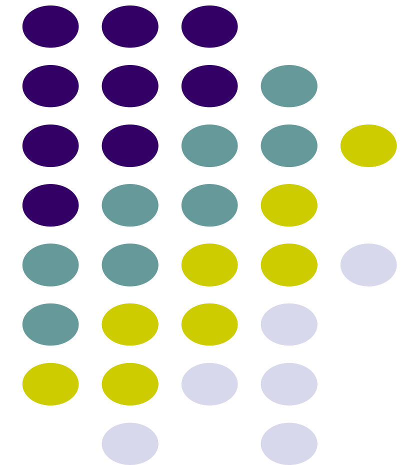


Thinner copper will allow for finer spacing, meaning more routes in same area possible



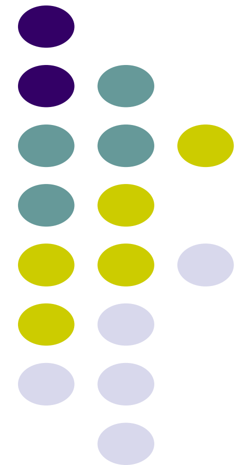
4/4 = 10 traces   3/3 = 12 traces

- Better efficiency = more traces/ channels/ boulevards
- Fewer routing layers needed



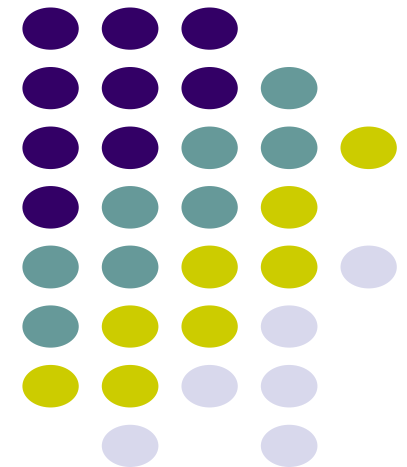
Various possibilities for signal flow layer to layer without blocking other busses

	BUS #1	BUS #2	BUS # 3	Critical Signal group #1	Critical Signal group #2	Power and Ground
layer 1 - Signal w/ poured gnd						
layer 2 - Gnd						
layer 3 - Signal						
layer 4 - Signal						
layer 5 - Gnd						
layer 6 - Pwr						
layer 7 - Signal						
layer 8 - Signal						
layer 9 - Gnd						
layer 10 - Signal w/ poured gnd						



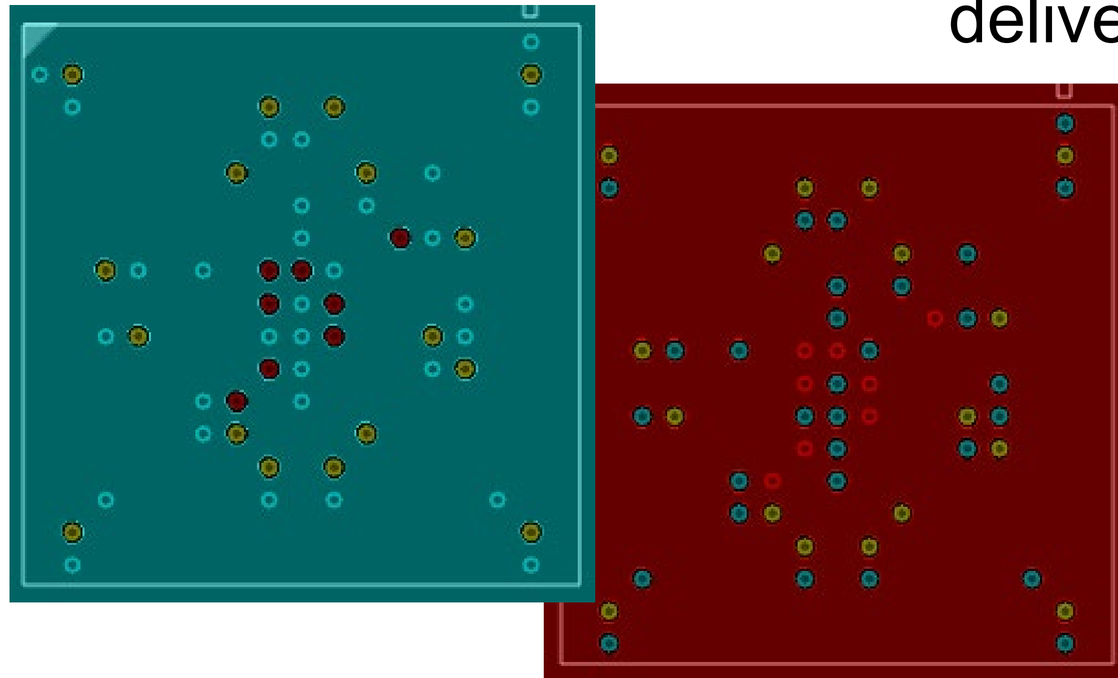
HDI Good for SI and EMI

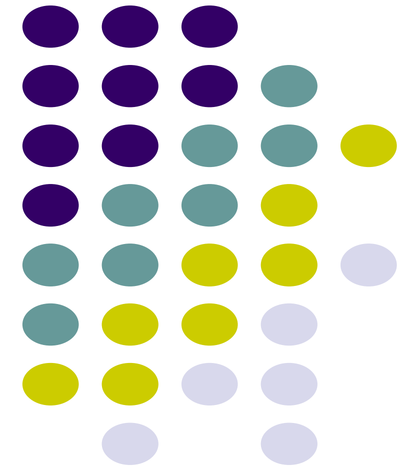
- Because dielectric layers are thinner, traces closer to return planes
- Improved containment of energy fields
- Possibly better for amount of separation for differential pairs
- HDI extenal layers are often flooded plane - also good for EMI
- Allows planes to be close together
- Good for inter-plane capacitance



Power Delivery

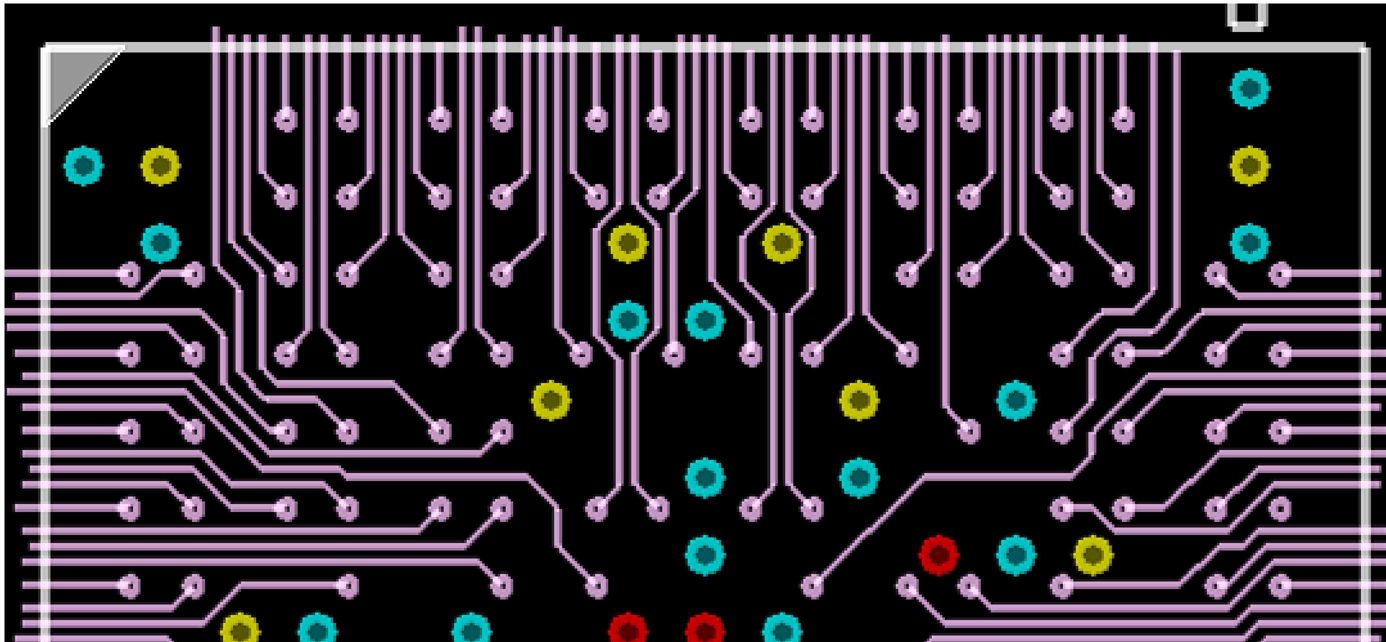
- Larger power/ground copper area under BGAs with HDI also means better inter-plane capacitance and potentially better power delivery

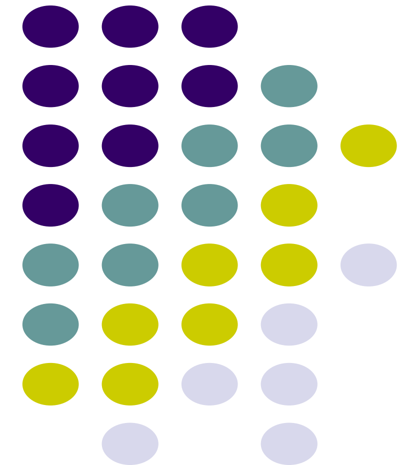




More routing on internal layers

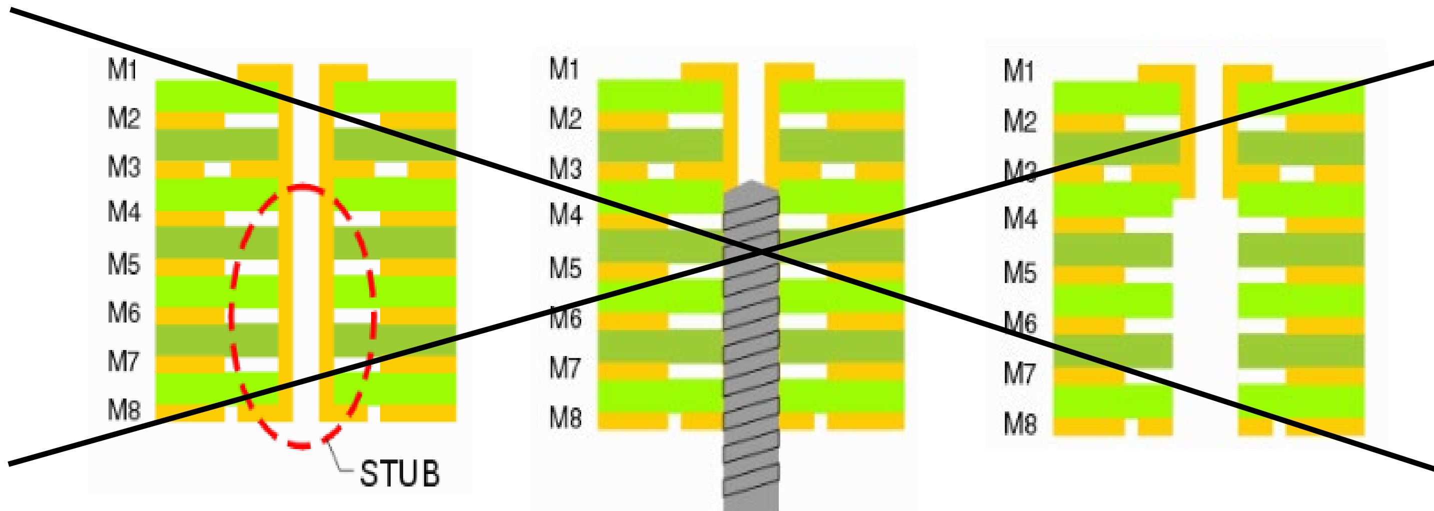
- More signals benefit from routing on inner layers – good for signal integrity, trace shape, impedance control



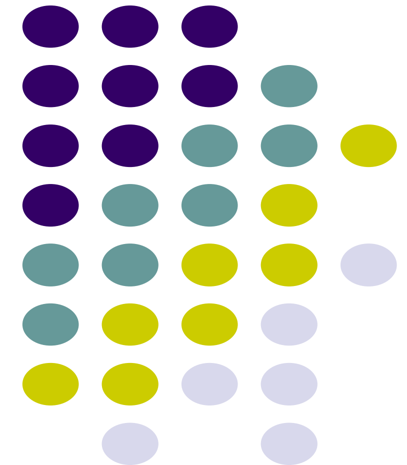


No Via Stubs

- uVias only go where needed
- No controlled depth/backdrilling necessary
- No backdrill obstructions/clearance
- Reduced inductance in via barrel

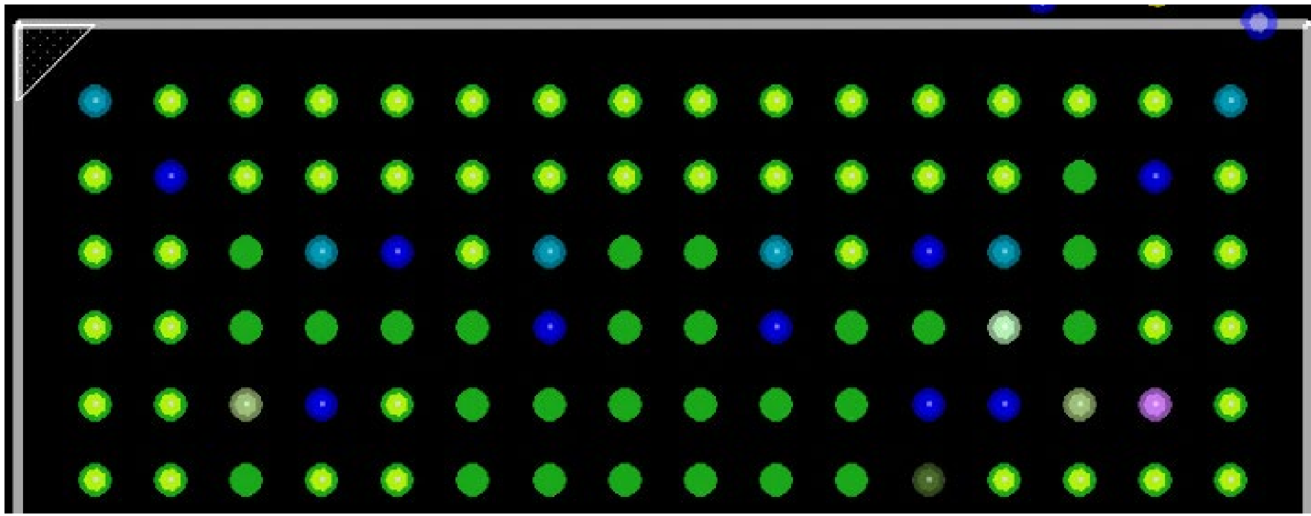


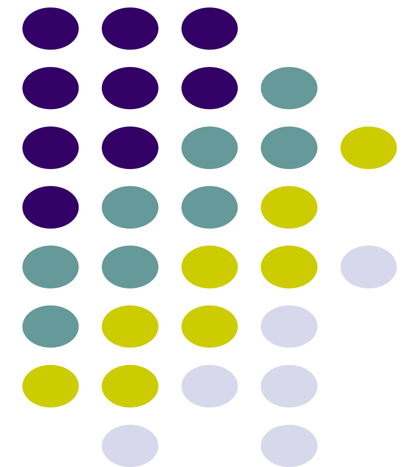
Pictures of ‘Backdrilled’ TH vias from Netex-0 “Extracting Geometry, Nets, and Components from ODB++”



uVIA in PAD better than TH Via in Pad

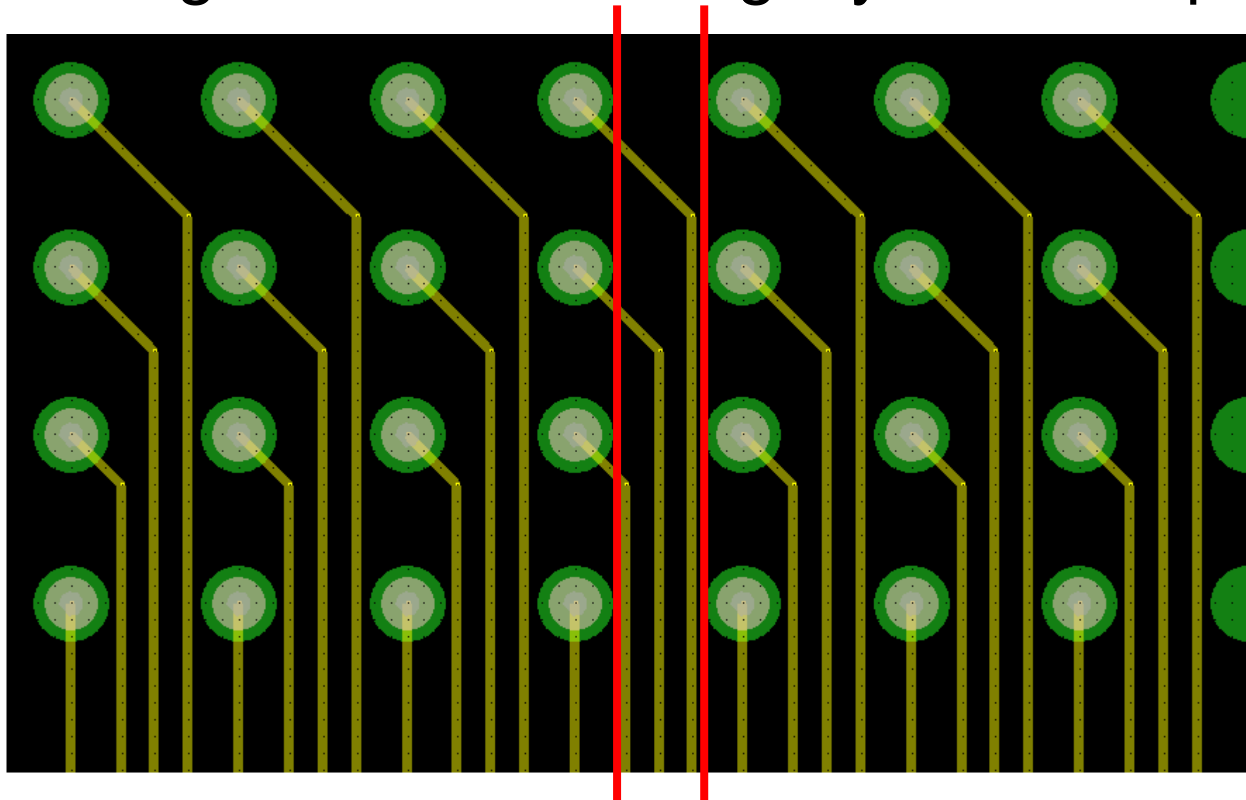
- SM for uVia is within the SM for pad so no extra SM opening needed as in dog bone
- Reduced inductance – connection is made right from ball to pad to via (w/o dog bone) and down to other layers.

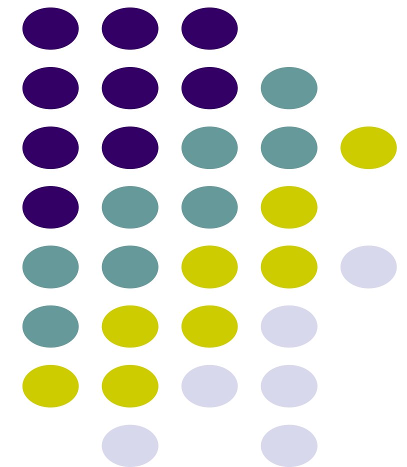




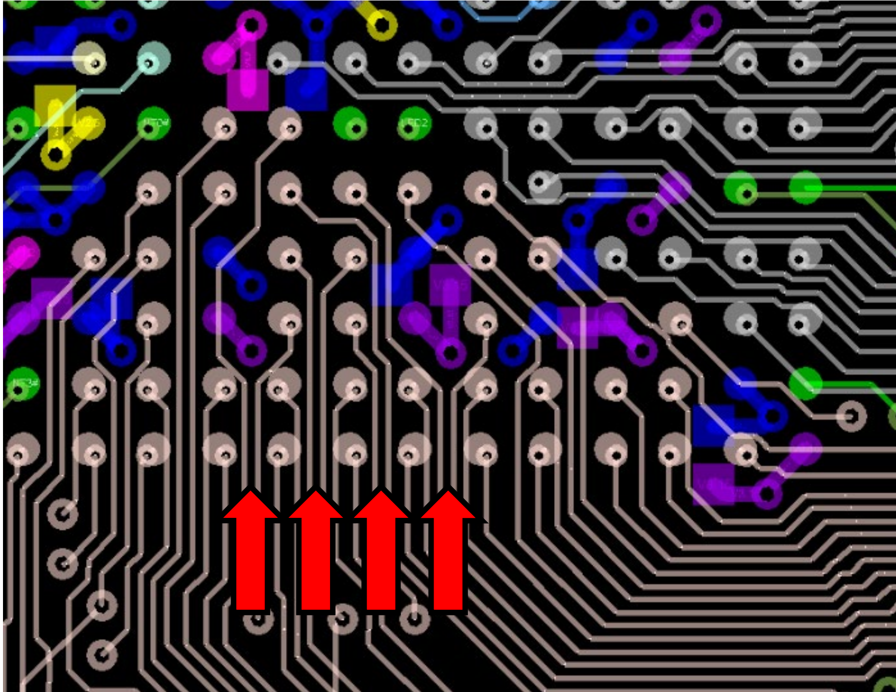
Via in Pad also means smaller antipad openings to avoid for routing return

- Signals can route slightly closer to pads



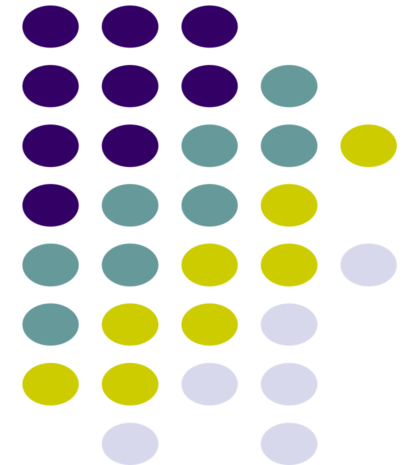


Benefits to Board



An offset uVia grid can add extra routes

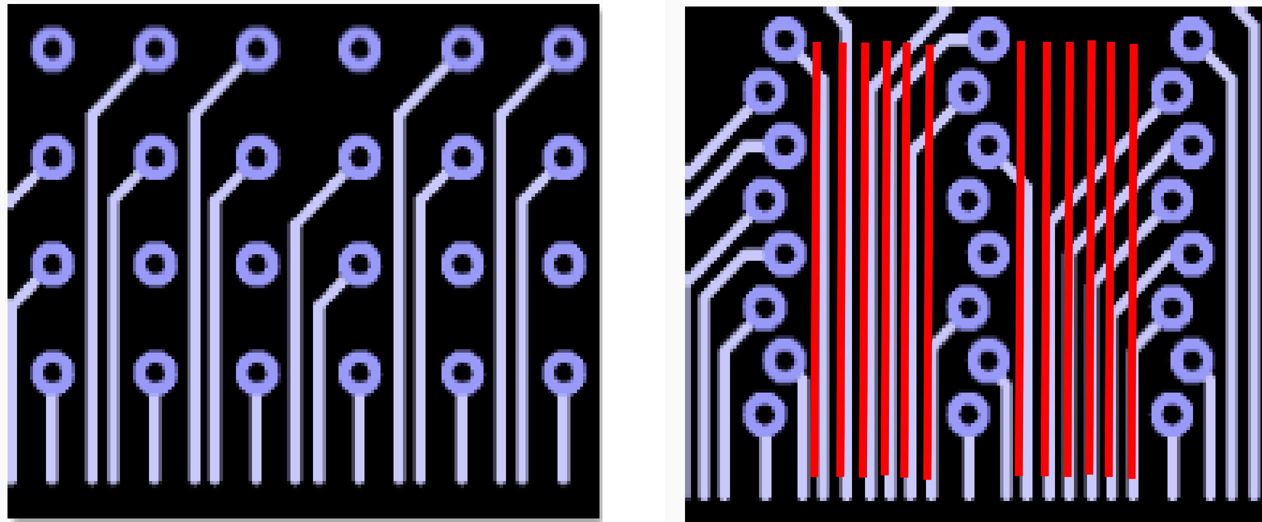
- Helps any size part be more routable

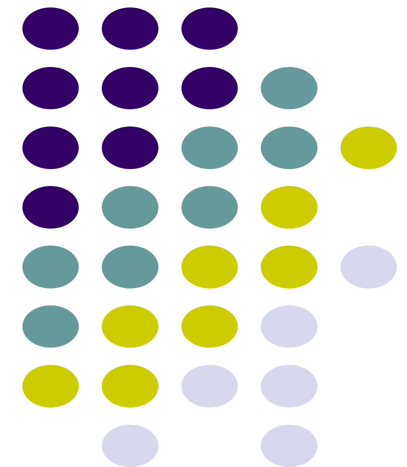


- HDI vias can be centered in, offset from, or tangent to surface mount pads to set up routing channels

HDI Routing Channels Improve Efficiency

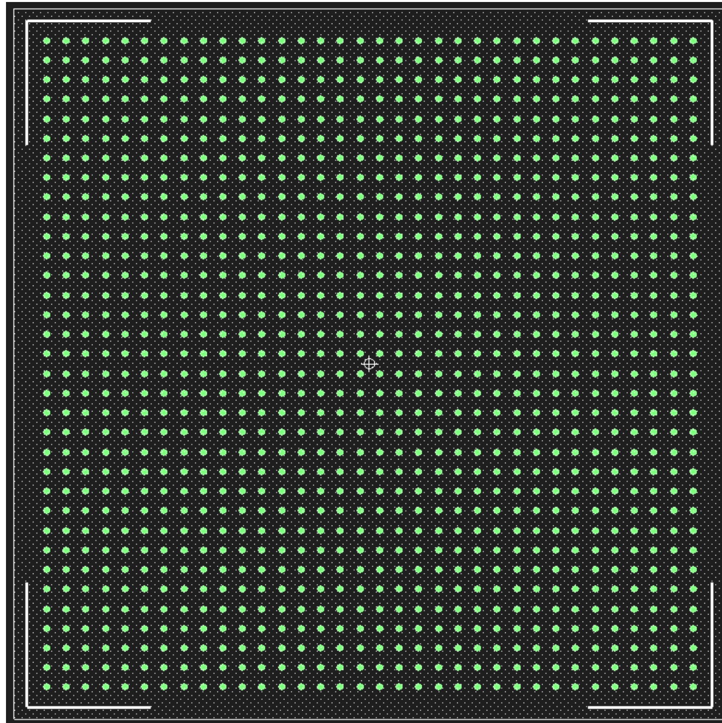
- Channels might be set up very differently to fan out a small, very fine pitch part vs a large or very fine pitch part
- Small parts may just need a path for all signals





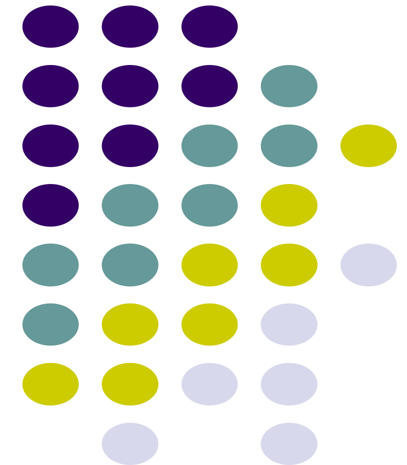
Reprinted with permission from BGA Breakouts & Routing by Charles Pfeil

HDI can make difficult parts routable



TI GTM (N2377) BGA with 2377 pins at 1mm pitch

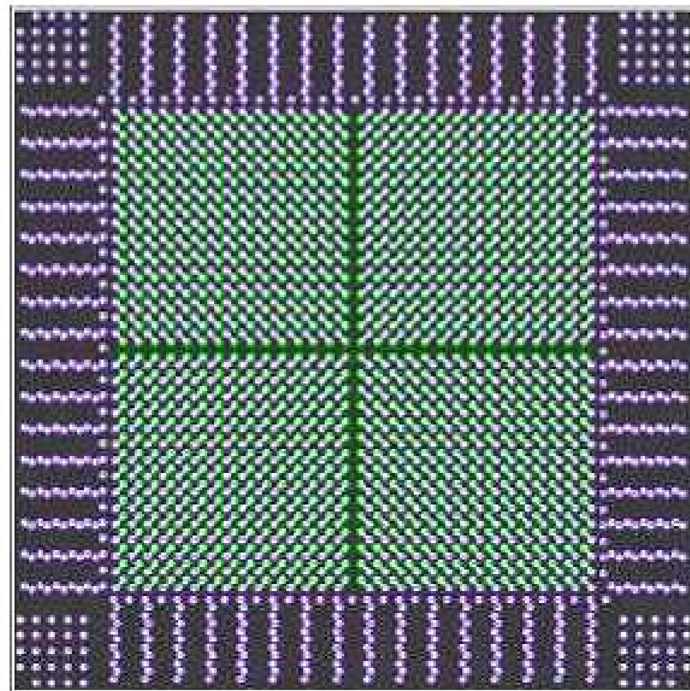
- Ability to fanout large, high pin count packages
- No way to route with through hole
- Or device would need too many TH routing layers

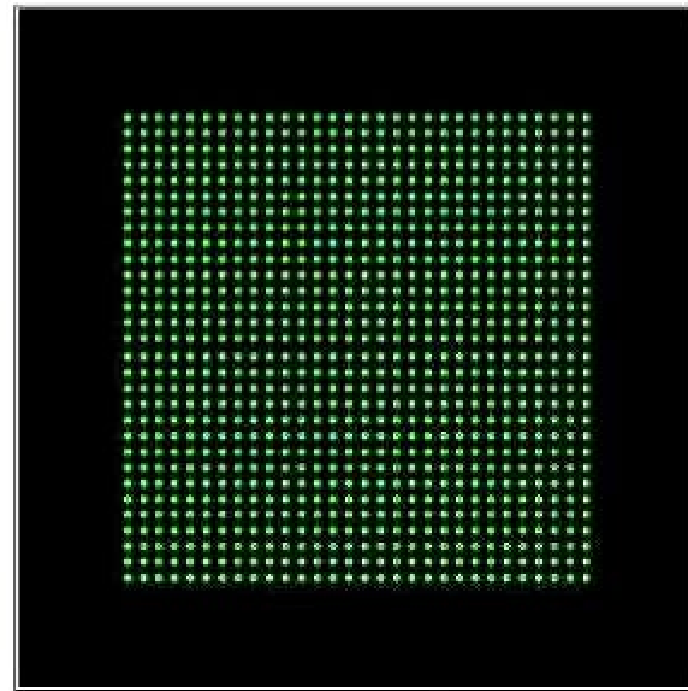


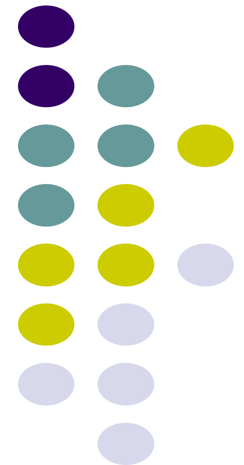
The Advantage of Channeled Fanouts

With good fanout patterns, you can effectively reduce the size of a large BGA array for routing

With HDI, 1760 pins effectively reduced 41% to 1024



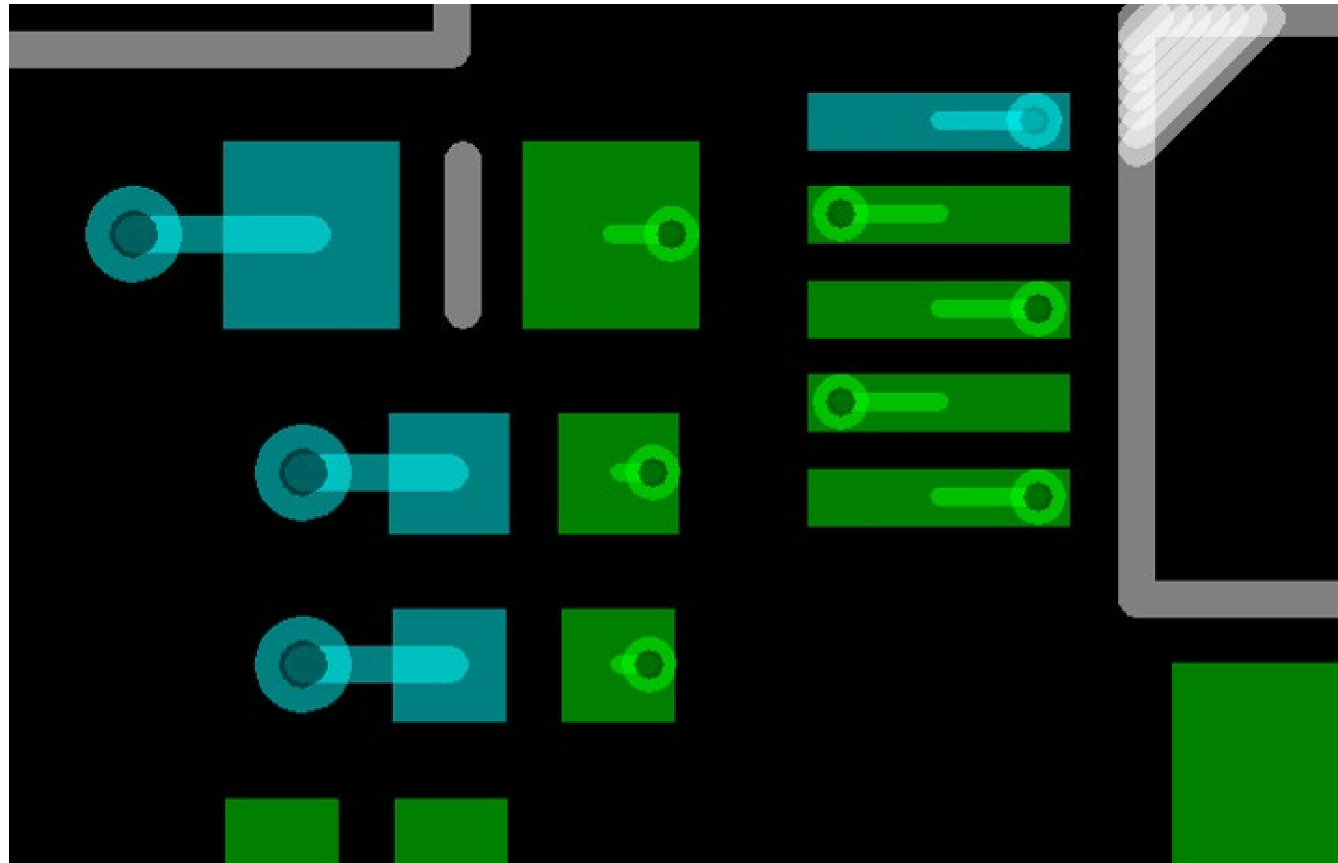


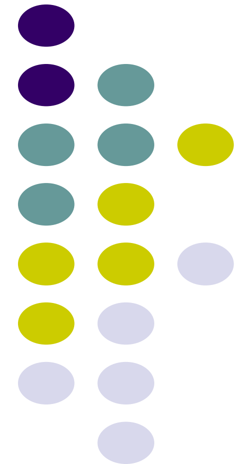


Reprinted with permission from BGA Breakouts & Routing by Charles Pfeil

uVias are not just for BGAs

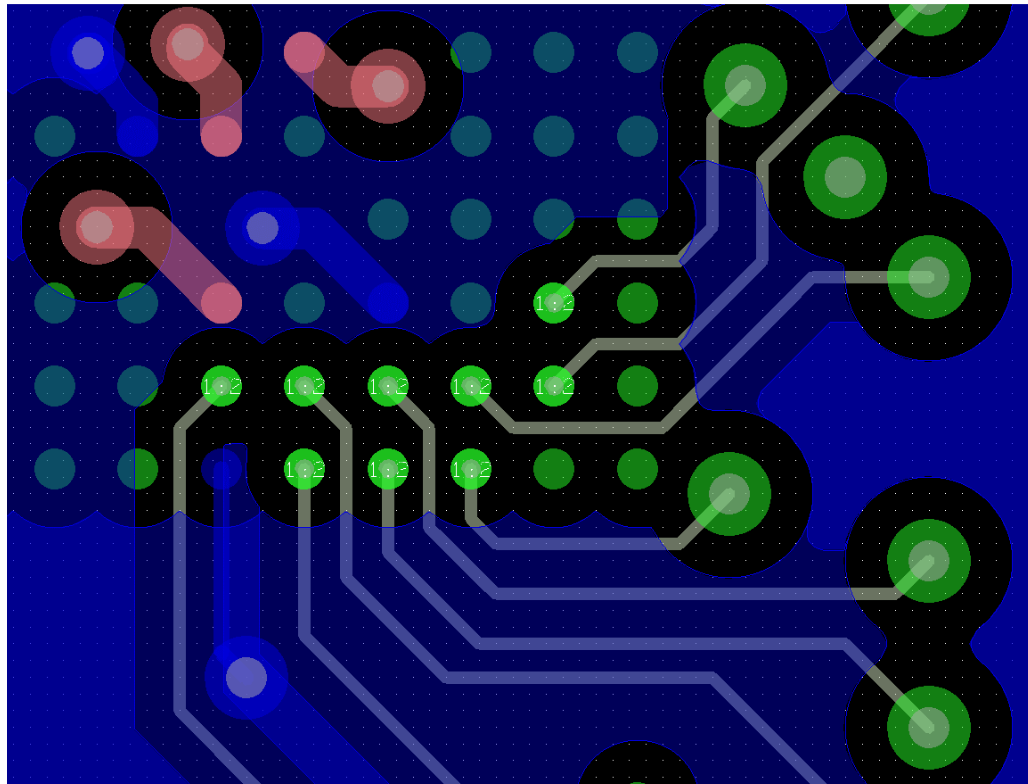
- Via in pad can also help move parts close together
  - Signals may be shorter and timing better
    - Possibly smaller board as well



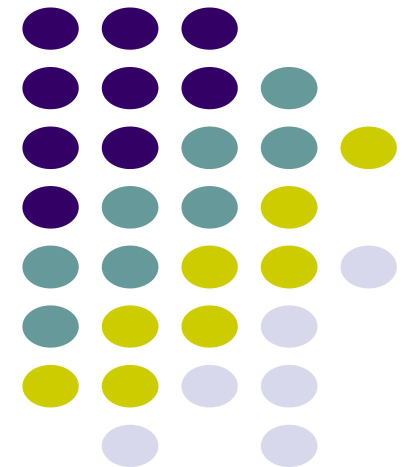


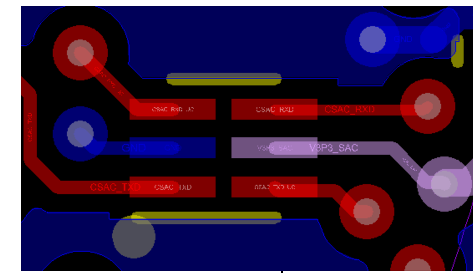
Benefits to Board

When already on the board, uVias can be used in congested areas to lead signals out of BGA to an open area, and from there to TH or buried vias



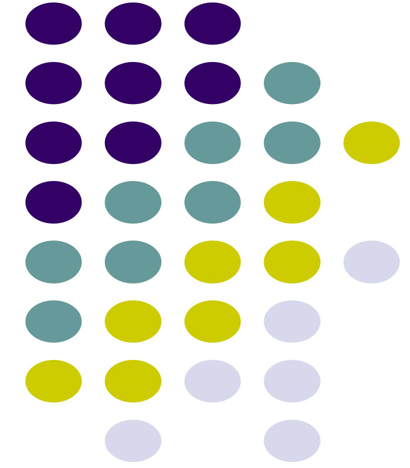
(.5mm part)

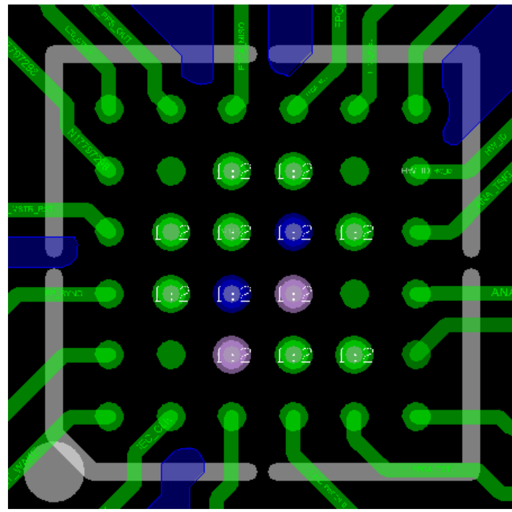




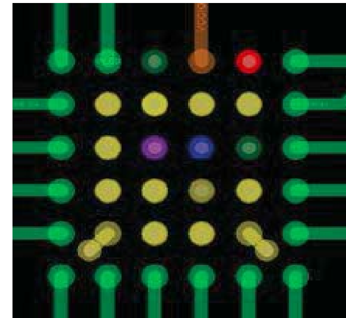
Finer pitched parts can be used with HDI

- Some of the new parts are only available in small BGA packages
- Other fine pitch devices have very little room for all the large TH vias needed nearby

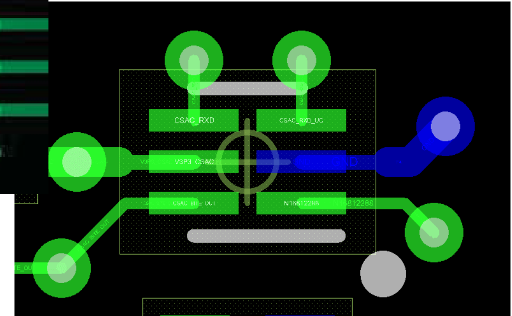




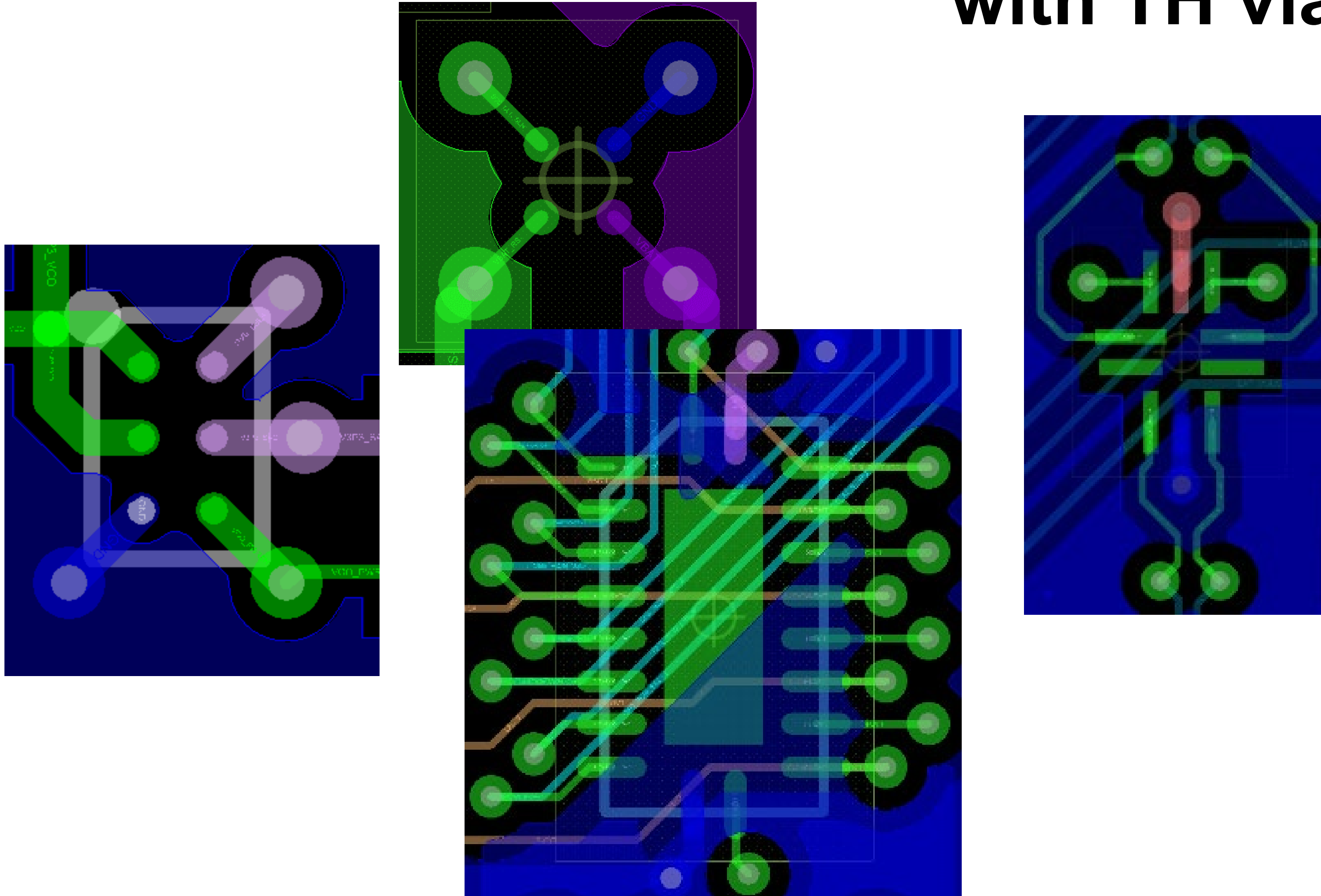
0.4mm BGA

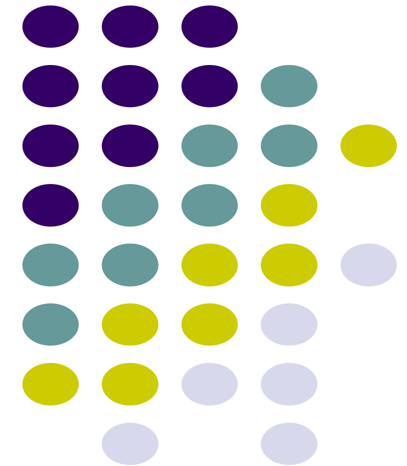


0.35mm BGA

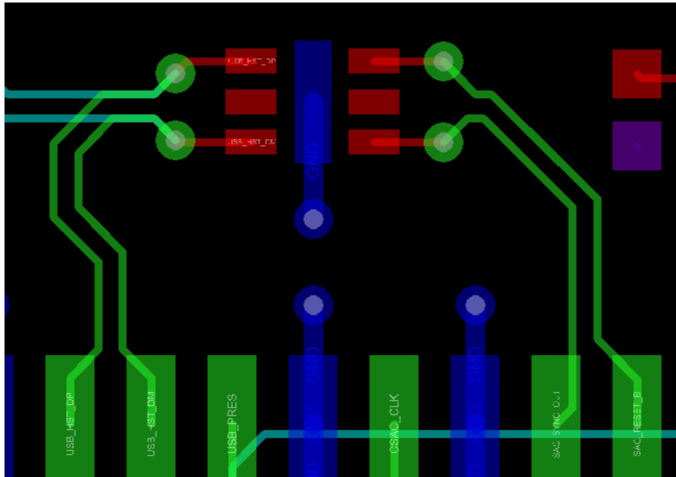


Some parts are just complex to route with TH vias



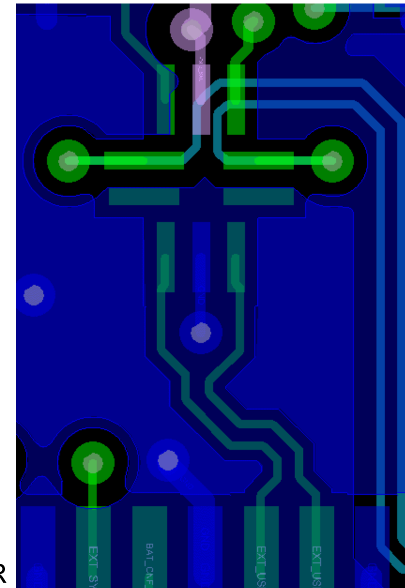


Some parts are just complex to route with TH vias

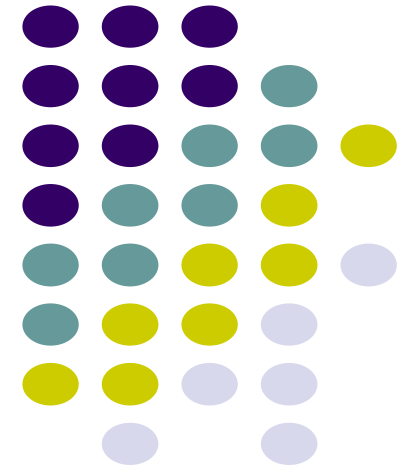


- uVias would help with ESD routing

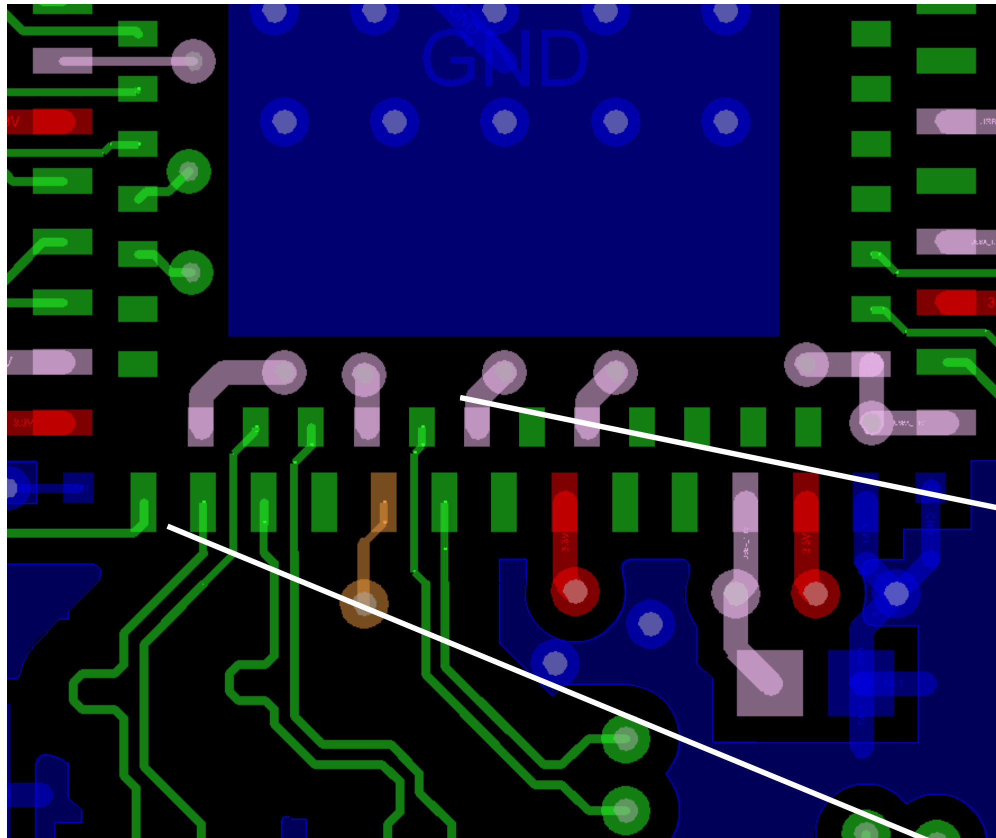
- uVias would help with differential routing on newer style ICs



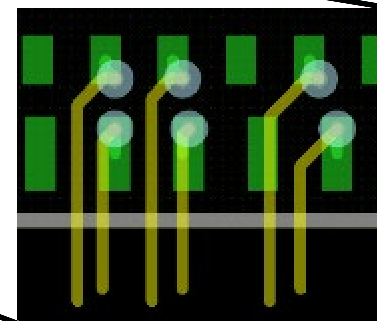
TI_TS3USB30ERSWR

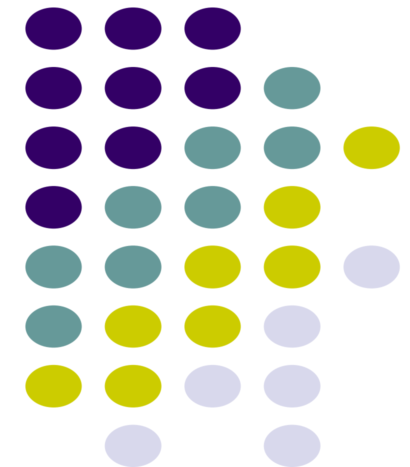


Some parts would benefit from using uVias



- Here, uVias help diff pairs move to inner layer, closer, & away from tab

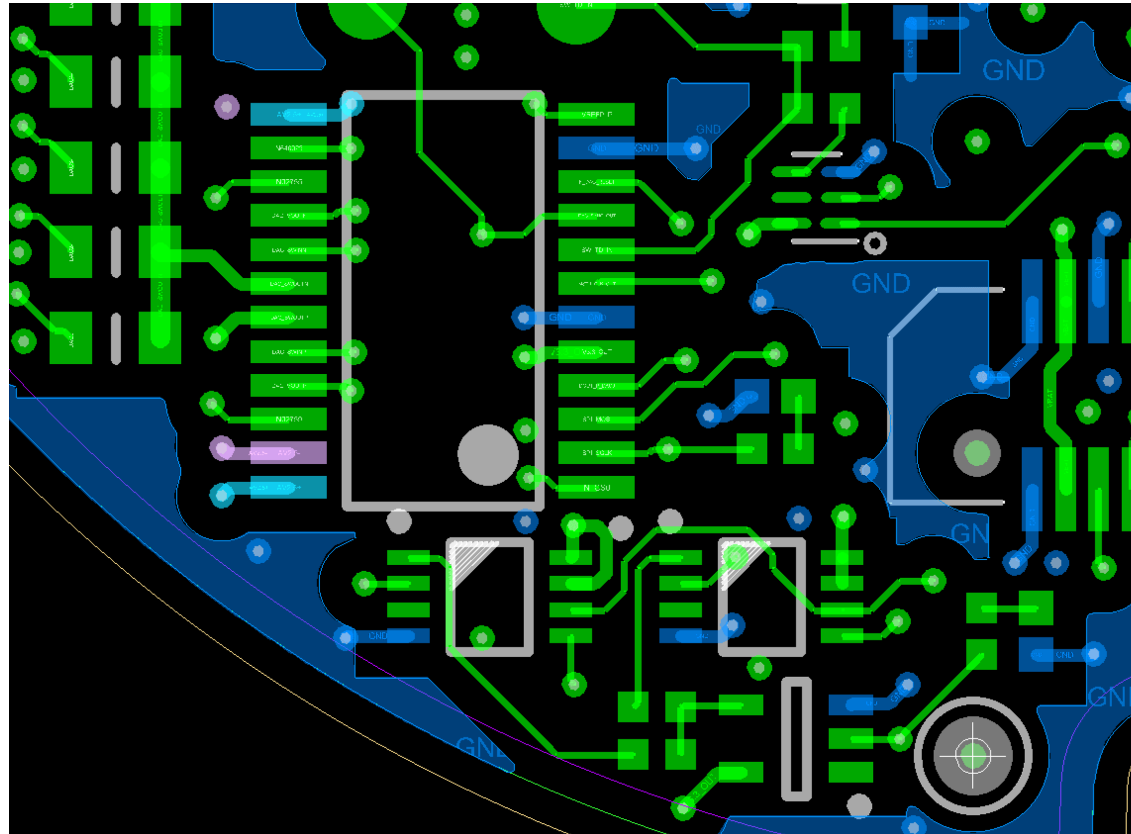


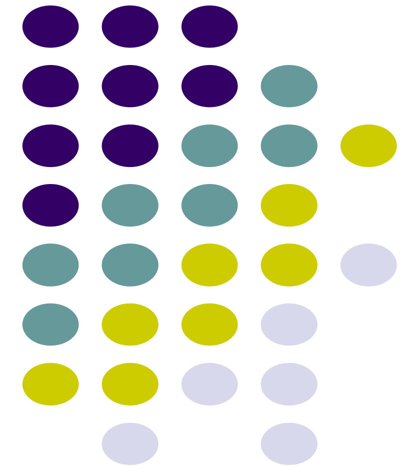


Better picture?

Benefits to Board

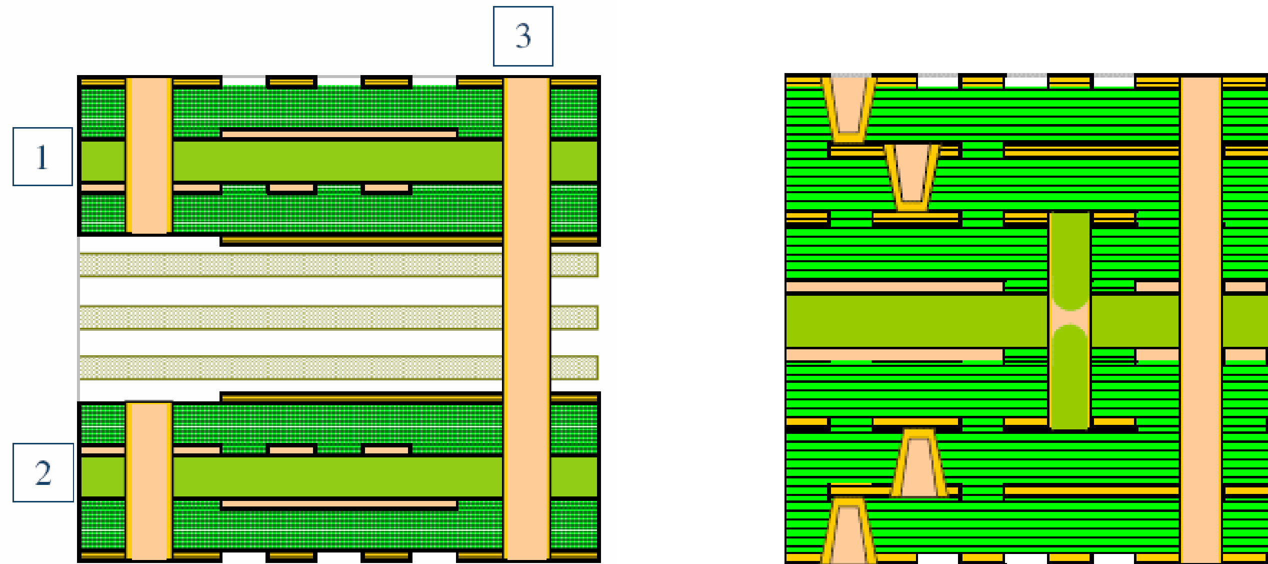
Some boards or areas would benefit from uVias because of P&R density

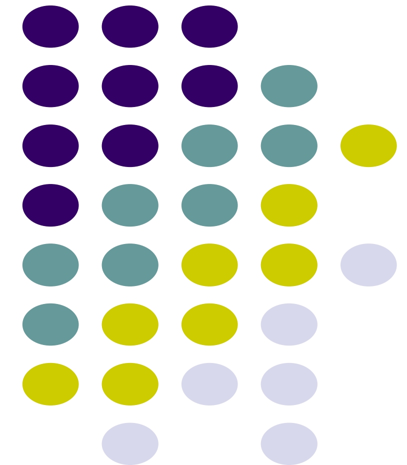




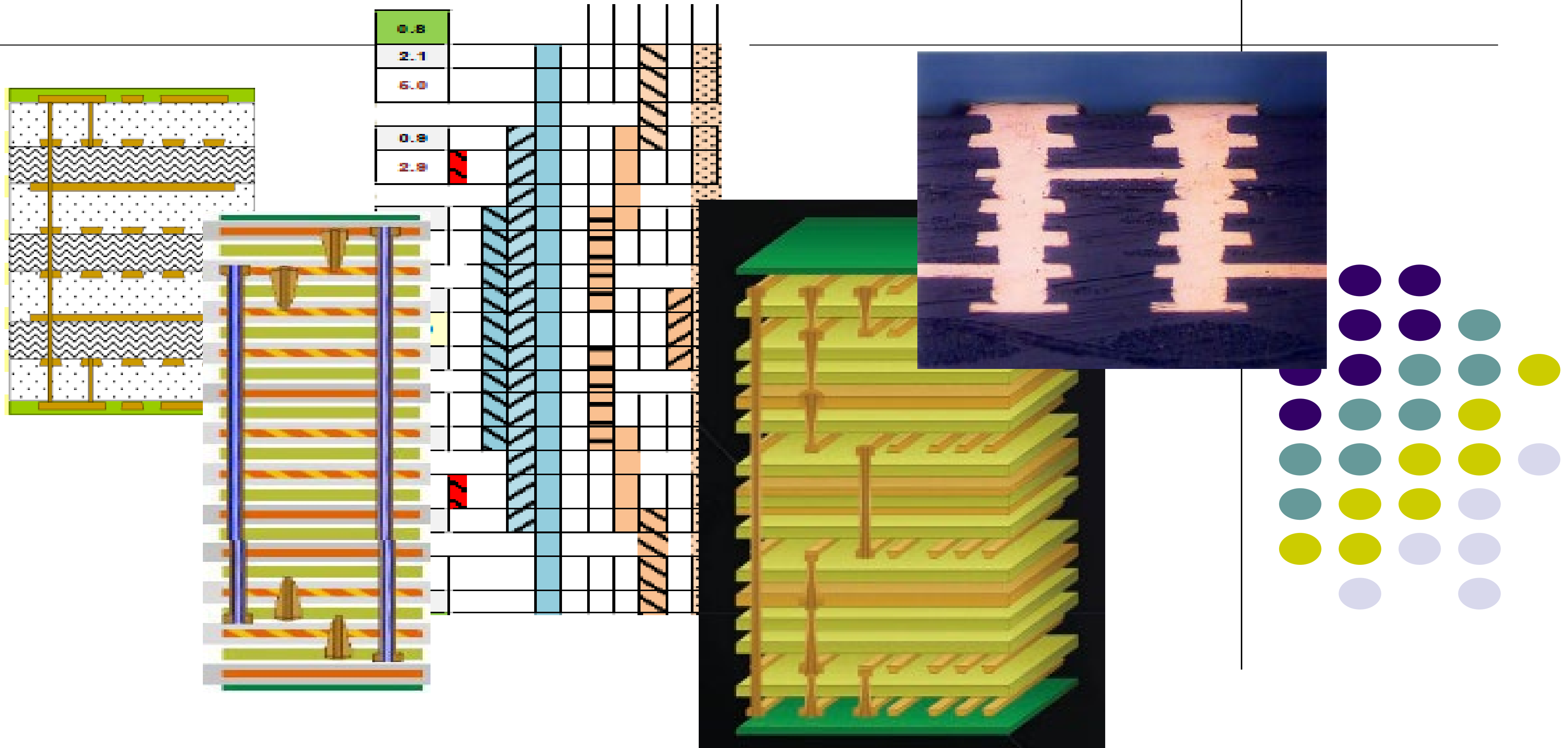
Copper filled/planarized uVias allow for active parts to easily be placed on both sides of the board

- Easy to fan out parts on their ‘own’ layers





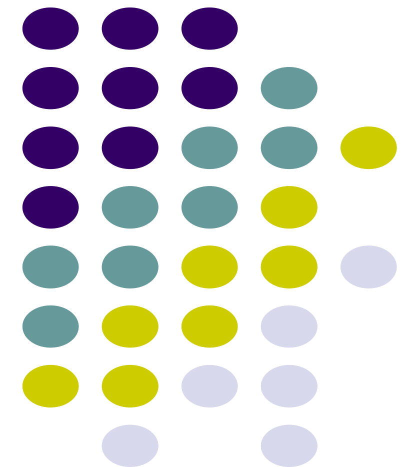
HDI Provides the largest variety or stackup possibilities



- Via patterns can stack and stagger through many layers

HDI can affect Thermal Mgt.

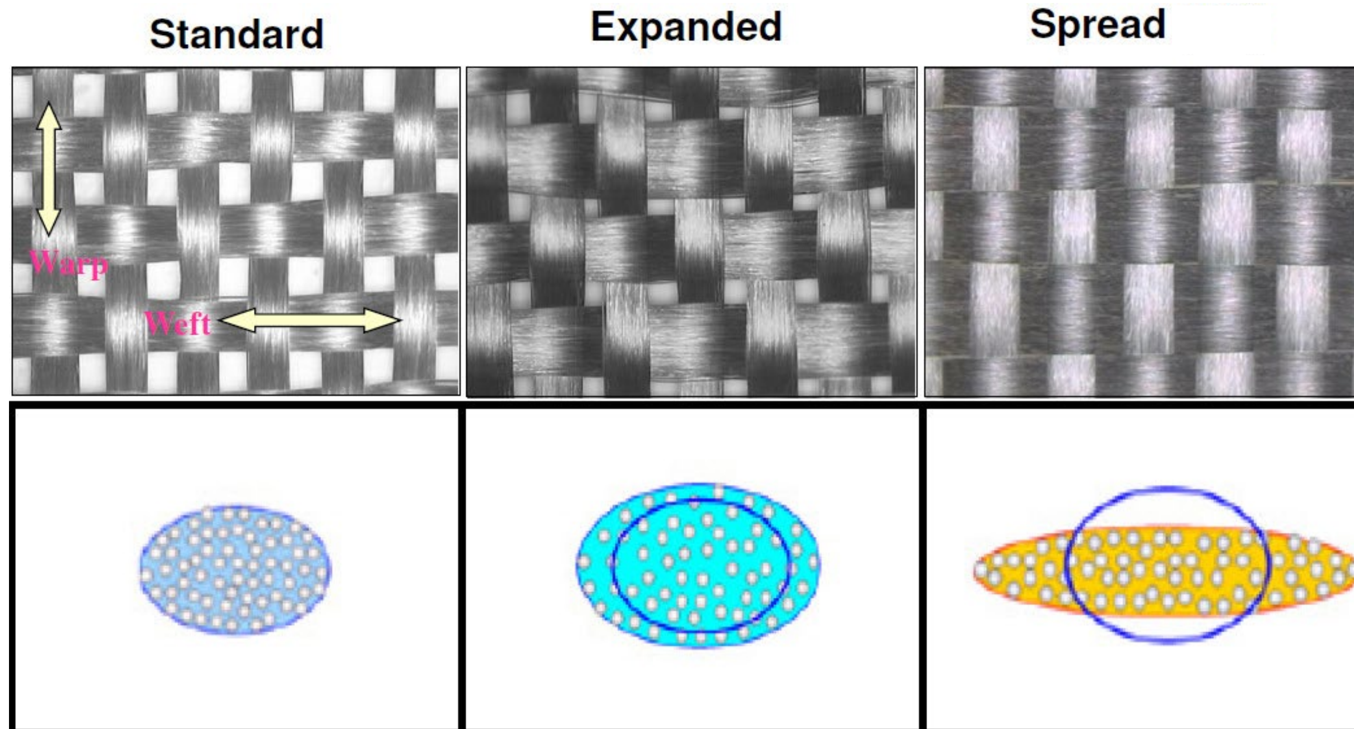
- Consider thermal transfer when building stack
  - The primary heat transfer is accomplished through **conduction**
  - The tighter we place parts, the more heat
  - The smaller parts often produce more heat
  - Solid copper fill in uVias helps to improve heat transfer layer to layer
  - Newer thin materials may also improve thermal performance*

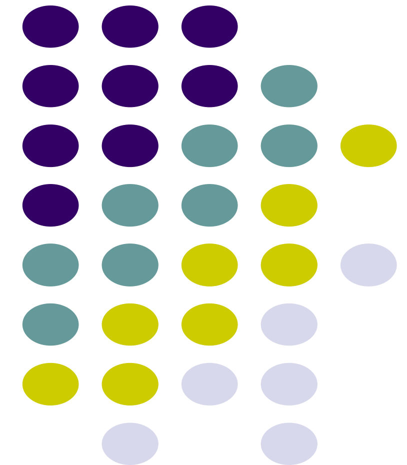


*Happy Holden "HDI's Beneficial Influence on High Frequency Signal Integrity"

Flatter Weave Materials Available for HDI

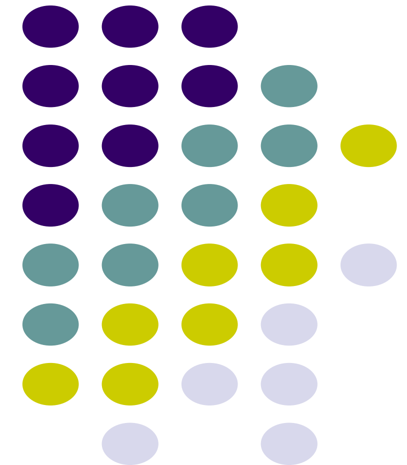
- HDI's spread material may be helpful to all signals' quality – particularly HS or differential pairs





Design for low cost

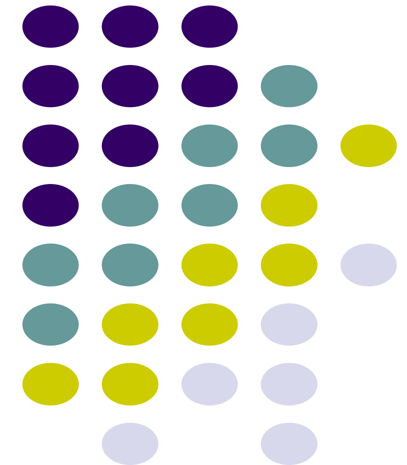
- Keep to Type I or Type II, if possible – fewest laminations
- Use least depth of uVias – no skip vias
- When changing layers, staggered uVias are easier to produce than stacked
- If HDI is used on the board, it usually does not cost extra to use in many other places
- You can add microvias to a through hole board…. But it's still a TH board, just more expensive*



*Happy Holden in "Current PCB Cost Adders"

Fabrication Issues

- Get fabrication involved early!*
- **Check with your fabricator** for his norms before starting a board - capabilities, up-charges, turn times, etc.
- Think about fabrication yield… HDI may cost a bit more, but that may beat the cost of a TH board that is difficult for the designer to design and/or for the fabricator to build!
- **Consider what is most cost effective from a DFM point of view**



Thank you!

Susy Webb
DesignScience@ymail.com