

# AN10778

## PCB layout guidelines for NXP MCUs in BGA packages

Rev. 01 — 22 January 2009

Application note

### Document information

Info	Content
<b>Keywords</b>	LPC2220, LPC2292, LPC2364, LPC2368, LPC2458, LPC2468, LPC2470, LPC2478, LPC2880, LPC2888, LPC3130, LPC3131, LPC3151, LPC3152, LPC3153, LPC3154, LPC3180/10, LPC3220, LPC3230, LPC3240, LPC3250, LH79524, LH7A400, LH7A404, TFBGA100, TFBGA144, TFBGA208, TFBGA180, TFBGA296, LFBGA208, BGA256, LFBGA256, LFBGA324, LFBGA320, Layout Guidelines, BGA, PCB, Fan-out
<b>Abstract</b>	This application note is focused on Printed Circuit Board (PCB) layout issues when using (LF)(TF) BGA packages from the NXP LPC Microcontroller family.

**Revision history**

Rev	Date	Description
01	20090122	Initial release

**Contact information**

For additional information, please visit: <http://www.nxp.com>

For sales office addresses, please send an email to: [salesaddresses@nxp.com](mailto:salesaddresses@nxp.com)

## 1. Introduction

The plastic Ball Grid Array (BGA), including Low profile Fine pitch BGA (LFBGA) and Thin profile Fine pitch BGA (TFBGA), packages have become, for many applications, the first choice for designers requiring medium to high pin-count IC packaging. For this reason many of the LPC Family of Microcontrollers are available in the LFBGA or TFBGA package.

When comparing it to other common alternative packages, such as the Quad Flat Pack (QFP), the (LF)(TF)BGA device has many advantages. Such as:

- The (LF)(TF)BGA has no easy-to-bend leads that can cause deviation from coplanarity.
- The (LF)(TF)BGA is typically 20% to 25% smaller than an equivalently functional QFP.
- Resolution and smearing problems with respect to the stencil-print process are less because the pitch is larger, and the apertures are circular.
- The self-alignment property of the component results in a large process window for automatic placement.
- The (LF) (TF)BGA is compatible with today's assembly techniques, which means that no adjustments are necessary to standard machines or materials.

### 1.1 Scope

The scope of this application note is focused on Printed Circuit Board (PCB) layout issues when using (LF)(TF) BGA packages from the NXP LPC Microcontroller family. Including:

- Recommended footprint patterns for the TFBGA180, TFBGA208, TFBGA296 and LFBGA320 pin packages.
- Recommended trace, space and via size for fan-out routing of the TFBGA180, TFBGA208, TFBGA296 and LFBGA320 pin packages

It is recommended that other assembly topics such as the solder paste chemistry, reflow solder profile and solder paste stencil etching, which are affected by all components on the board level assembly and not limited to the Microcontroller BGA alone, be a collaborative effort between the system designer and the assembly contractor.

## 2. BGA Package Description

A cross section of the typical (LF)(TF)BGA is shown in [Fig 1](#).

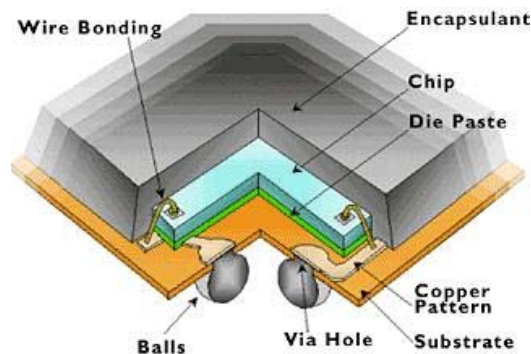


Fig 1. (LF)(TF)BGA Cross Section

This application note applies to BGA packages listed in [Table 1](#).

**Table 1. BGA Packages**

Package Name	NXP Outline Code	Outline Dimensions	Ball Pitch	Ball Diam	Ball Configuration
BGA256	SOT1018-1	[4] 17 x 17 x 1.35mm	1.0mm	0.50mm	16 x 16; full matrix
TFBGA100	SOT926-1	[2] 9 x 9 x 0.7mm	0.8mm	0.45mm	10 x 10; full matrix
TFBGA144	SOT569-2	[2] 12 x 12 x 0.7mm	0.8mm	0.45mm	13 x 13; partial matrix
TFBGA180	SOT570-2	[2] 12 x 12 x 0.8mm	0.8mm	0.45mm	14 x 14; partial matrix
TFBGA208	SOT950-1	[2] 15 x 15 x 0.7mm	0.8mm	0.45mm	17 x 17; partial matrix
LFBGA208	SOT1019-1	[5] 14 x 14 x 1.27mm	0.8mm	0.45mm	16 x 16; partial matrix
LFBGA256	SOT1020-1	[5] 14 x 14 x 1.25mm	0.8mm	0.45mm	16 x 16; full matrix
TFBGA296	SOT1048-1	[1] 15 x 15 x 0.7mm	0.8mm	0.45mm	18 x 18; partial matrix
LFBGA324	SOT1021-1	[5] 17 x 17 x 1.25mm	0.8mm	0.45mm	20 x 20; partial matrix
TFBGA208	SOT930-1	[2] 12 x 12 x 0.7mm	0.65mm	0.40mm	17 x 17; partial matrix
TFBGA180	SOT640-1	[3] 10 x 10 x 0.8mm	0.5mm	0.30mm	18 x 18; partial matrix
LFBGA320	SOT824-1	[2] 13 x 13 x 0.9mm	0.5mm	0.30mm	24 x 24; partial matrix

[1] Reference JEDEC MO-216

[2] Reference JEDEC MO-275

[3] Reference JEDEC MO-195

[4] Reference JEDEC MS-034

[5] Reference JEDEC MO-205

### 3. BGA Footprints

When building a BGA footprint the number one consideration is ensuring the ball pattern and outline matches the device package. This includes correct orientation of ball A1, matching all ball column x row locations, and the ball-to-ball pitch. Solder joint reliability is also of primary concern. For cost sensitive applications, minimizing the number of PCB layers required to route the BGA is a consideration. The BGA land pattern footprint plays a key role in solder joint reliability, and the number of PCB layers required to route the balls.

#### 3.1 Land Pad Design

The PCB BGA land pads have to be designed to ensure solder joint reliability and provide optimum manufacturability. The two basic types of BGA land pad design are:

- The Solder mask defined land pad (SMD)
- The Non-solder mask defined land pad (NSMD); recommended type for PCB

### 3.1.1 Solder mask defined land pad (SMD)

The SMD type of BGA land pad design is characterized by the copper pad being larger than the solder mask opening above this pad. Thus the solder joint area of the land pad is defined by the opening in the solder mask.

### 3.1.2 Non-solder mask defined land pad (NSMD)

The NSMD type of BGA land pad design is characterized by the copper pad being smaller than the solder mask opening. Thus the solder joint area of the land pad is defined by the size of the land pad. The solder mask clearance around the land pad must be large enough to ensure that no solder mask overlaps the land pad. Typical solder mask to land pad clearance is in the range 0.06 – 0.075mm, depending on the PCB manufacturer's solder mask alignment tolerance.

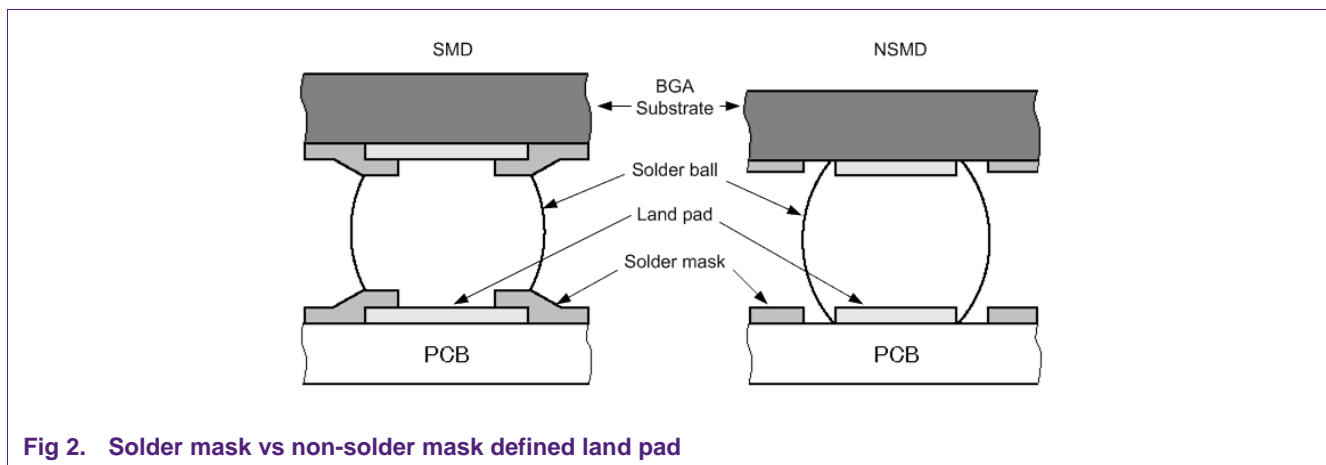


Fig 2. Solder mask vs non-solder mask defined land pad

## 3.2 Recommended BGA Footprint

The NSMD type land pad is recommended for the PCB BGA footprint. In addition to the top surface of the land pad, the reflowed solder paste will wet to the side wall making a mechanically stronger solder joint than the SMD type pad. The smaller NSMD land pad also leaves more space for routing traces between the land pads. It has been shown that matching the solder joint area of the PCB land pad to that on the BGA package substrate equalizes the ball solder joint stress between the BGA package and PCB land pad thereby reducing the chance of a solder joint stress crack. All of the BGA packages referenced in this application note use SMD type pads. The NSMD type pads on the PCB should be approximately 10 - 15% smaller than the SMD pads on the BGA to achieve equalized stress. This difference between the BGA package SMD pad and recommended PCB NSMD pad for each BGA package is reflected in [Table 2](#). A generic BGA footprint is shown in [Fig 3](#), and the specific dimensions for each BGA package are listed in [Table 2](#).

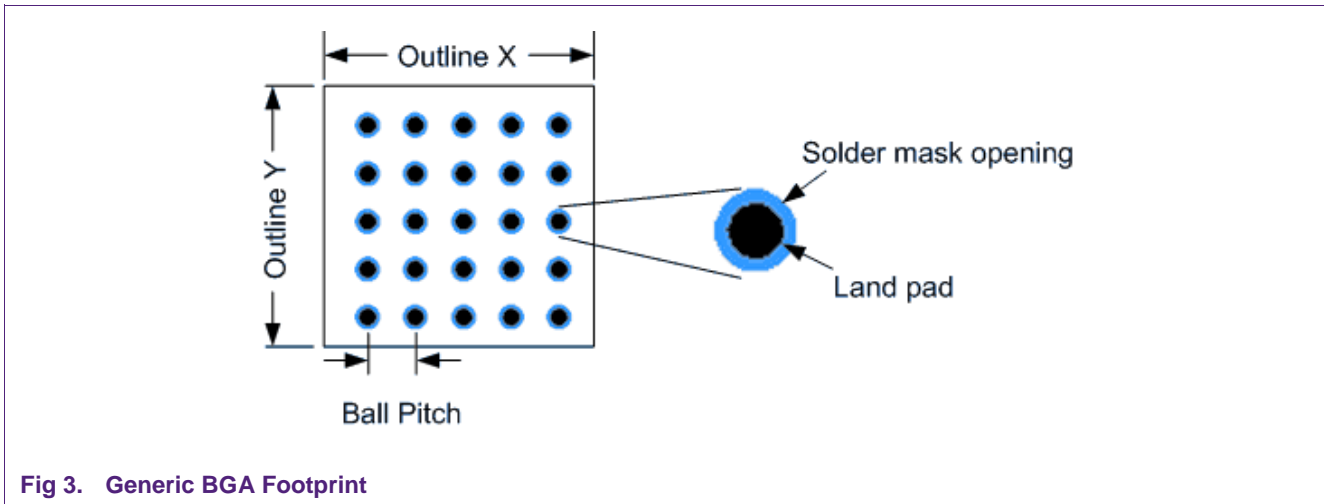


Fig 3. Generic BGA Footprint

Table 2. Recommended BGA Footprints

Package Name	Ball Pitch	Ball diameter	BGA substrate Land diameter	PCB land pad diameter	Solder mask diameter	Outline X & Y
BGA256	1.0	0.50	0.45	0.45	[4] 0.6	17.6
TFBGA100	0.8	0.45	0.4	0.35	[4] 0.5	9.6
TFBGA144	0.8	0.45	0.4	0.35	[4] 0.5	12.6
TFBGA180	0.8	0.45	0.4	0.35	[4] 0.5	12.6
TFBGA208 (SOT950-1)	0.8	0.45	0.4	0.35	[4] 0.5	15.6
LFBGA208	0.8	0.45	0.4	0.30	[4] 0.42	14.6
LFBGA256	0.8	0.45	0.4	0.30	[4] 0.42	14.6
TFBGA296	0.8	0.45	0.4	0.35	[4] [6] 0.5	15.6
TFBGA296	0.8	0.45	0.4	0.30	[4] [7] 0.42	15.6
LFBGA324	0.8	0.45	0.4	0.30	[4] 0.42	17.6
TFBGA208 (SOT930-1)	0.65	0.4	0.26	0.25	[5] 0.37	12.4
TFBGA180 (SOT640-1)	0.5	0.3	n/a	0.25	[5] 0.36	10.4
LFBGA320	0.5	0.3	0.25	0.25	[5] 0.36	13.4

Notes:

- [1] All dimensions are in millimeters
- [2] All BGA substrate land pads are SMD type
- [3] All PCB land pads are NSMD type
- [4] The recommended solder paste diameter is the same as the PCB land pad
- [5] The recommended solder paste diameter is 0.02mm larger than the PCB land pad
- [6] Used for routing 1 trace between land pads
- [7] Used for routing 2 traces between land pads

## 4. Recommended Fan-out Trace / Space guidelines

The small pitch between BGA balls and their matrix arrangement makes it impractical to route all of the BGA balls away from the BGA on a single layer. Fan-out vias (also called escape vias) are required to route the balls to other layers on the PCB. There are several via technologies used on PCB's. They are: Through-via, Blind via, Buried via, Micro via and In-pad via. Through-vias, where the drilled via hole goes through all layers on the PCB, cost considerably less than Blind, Buried, Micro and In-pad vias. Through-vias are generally larger than the other types of vias as well. All recommended fan-out examples in this application note use the through-via exclusively.

### 4.1 Recommended 1.0 and 0.8mm pitch BGA via fan-out pattern

For 1mm and 0.8mm pitch BGA's, the recommended via fan-out pattern centers each via within the space between four adjacent BGA land pads as shown in Fig 4. Generally, a single trace is routed between adjacent BGA land pads, allowing the two outer rows of balls to be routed without a fan-out via. For BGA's with larger than 0.8mm ball pitch one or two traces may be routed between adjacent BGA land pads, allowing the three outer most rows of balls to be routed without a fan out via. By reducing the BGA land pad, trace width and trace-to-pad space design rules for the 0.8mm ball pitch TFBGA296, two traces may be routed between the BGA land pads. See Table 3 for the layout tool design rules for 1.0 and 0.8mm pitch BGA via fan-out.

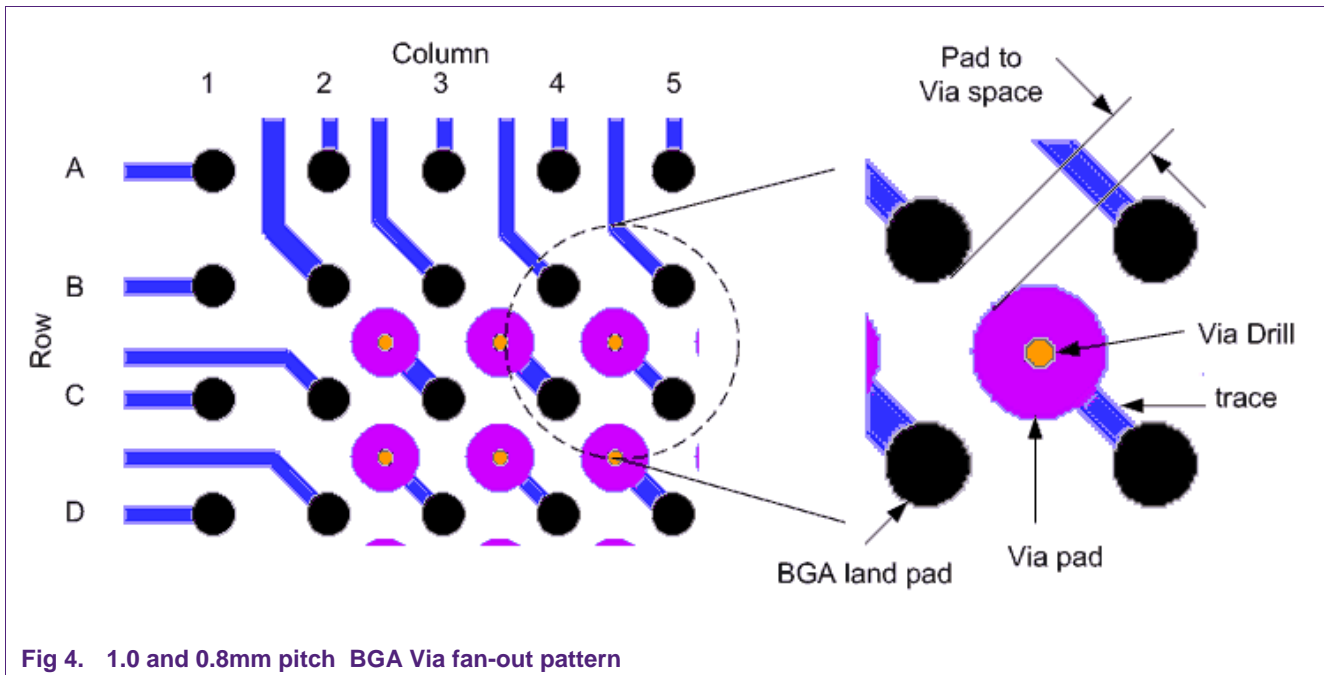


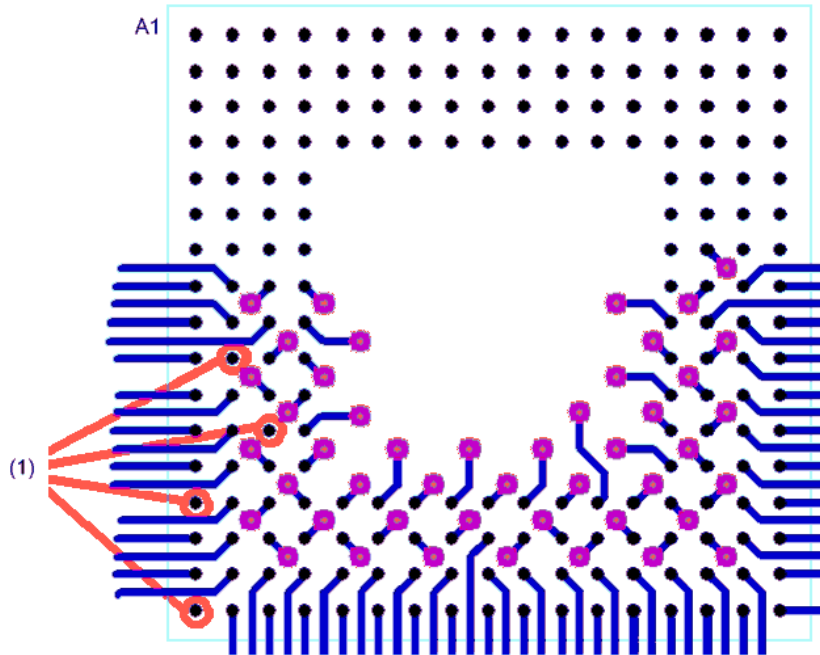
Fig 4. 1.0 and 0.8mm pitch BGA Via fan-out pattern

Table 3. 1.0 and 0.8mm pitch BGA layout design rules

BGA Pitch	BGA land pad	Via			Land pad to via space	Between vias		Between Land pads	
		Pad	Drill size / finished hole size	Inner plane layer anti-pad		Trace / space	# of traces	Trace / space	# of traces
1.0	0.45	0.55	0.3 / 0.18	0.800	0.2	0.15	1	0.18	1
1.0	0.45	0.485	0.25 / 0.1	0.695	0.24	0.1	2	0.11	2
0.8	0.35	0.485	0.25 / 0.1	0.695	0.148	0.105	1	0.15	1
0.8	0.30	0.485	0.25 / 0.1	0.695	0.173	0.105	1	0.1	2

### 4.2 Recommended 0.65mm pitch BGA via fan-out pattern

For 0.65mm pitch BGA's, the recommended via fan-out pattern centers each via within the space between four adjacent BGA land pads. Instead of placing the vias 0.65mm apart they are placed 1.3mm from each other, skipping every other location, and staggering them between adjacent rows, as the partial fan-out example is shown in Fig 5. With this pattern the TFGBA208 package can use 0.125mm (0.005") trace and space design rules. With a single trace routed between adjacent BGA land pads, the two outer rows of balls can be routed without a fan-out via. See Table 4 for the layout tool design rules for 0.65mm pitch BGA via fan-out.



(1) Note: no connect pins on the LPC3152/54

Fig 5. Recommended 0.65mm pitch BGA via fan-out pattern



Table 4. 0.65mm pitch BGA layout design rules

BGA Pitch	BGA land pad	Via			Land pad to via space	Between vias	Between Land pads	
		Pad	Drill size / finished hole size	Inner plane layer anti-pad		Trace / space	Trace / space	# of traces
0.65	0.25	0.425	0.2 / 0.05	0.6	0.122	0.125	0.125	1

### 4.3 Recommended 0.5mm pitch BGA via fan-out pattern

The pattern of centering the through-via within the four adjacent BGA land pads can not be used with 0.5mm pitch BGA's. This is due to the smallest through-via pad being too large to fit in the space available between the land pads. With a single trace routed between adjacent BGA land pads, the two outer rows of balls can be routed without a fan-out via. The two inner rows of balls must be routed to vias in the center area of the BGA and escape routed on other layers. An example fan-out of the LFBGA320 package is shown in Fig 6. See Table 5 for the layout tool design rules for 0.5mm pitch BGA via fan-out.

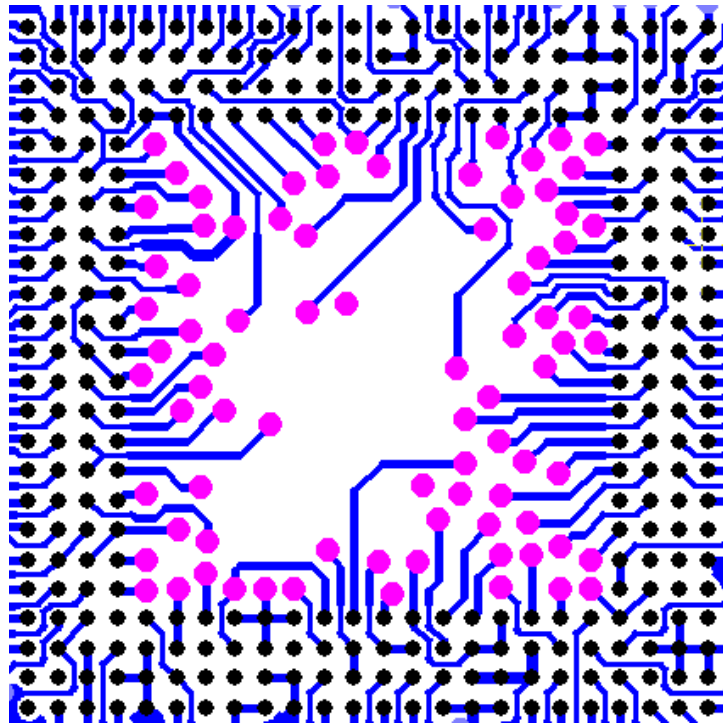


Fig 6. Example fan-out of LFBGA320

Table 5. 0.5mm pitch BGA layout design rules

BGA Pitch	BGA land pad	Via			Land pad to via space	Between vias	Between Land pads	
		Pad	Drill size / finished hole size	Inner plane layer anti-pad		Trace / space	Trace / space	# of traces
0.5	0.25	0.4	0.2 / 0.05	0.6	0.1	0.1	0.08	1

## 5. Board Cost considerations

PCB cost is affected by many factors, generally increasing in cost as:

1. Overall PCB area increases
2. As the number of layers increases
3. Using in-pad via, blind via, buried via, micro via
4. As the diameter of the through-via decreases
5. As the trace width decreases below 0.125mm (5 mils)
6. As the space between metal features decreases below 0.125mm (5 mils)

Therefore, selecting via size, trace width and spacing for fan-out routing of the BGA requires a balance between feature size, number of PCB layers and overall board area to get the most economical layout.

### 5.1 Area Rules

On many boards the design rules for via size, trace width and space for fan-out routing of the BGA may require smaller feature sizes than any other area on the board. If your layout tool is capable of defining multiple rule areas, it may be cost effective to limit the area around the BGA to the smaller feature sizes and use larger vias and larger trace widths and spacing for the balance of the board. In other words if all but the BGA fan-out can use 5 or 6 mil trace and space rules, then limiting 3 or 4 mil trace and space rules to the BGA fan-out area may have only a small cost premium.

### 5.2 How many PCB layers to fan-out the BGA

Generally one trace is routed between adjacent BGA land pads, enabling the two outer rows of BGA balls to be routed on the same layer as the BGA is mounted on. The next two rows in can be routed on the next signal layer, provided the vias are spaced far enough apart to allow one trace between them, as is the case for the 1.0mm, 0.8mm and 0.65mm recommended fan-out via patterns in [Fig 4](#) and [Fig 5](#). Each additional BGA row will take one additional PCB layer to fan-out. For example, the TFBGA296 has balls seven rows deep and will take 5 PCB layers to fan-out, including power and ground. Because PCB's are constructed in even number layers, a PCB using the TFBGA296 package would require a minimum of six layers, including one split power plane and one ground plane.

## 6. BGA Power and Ground

NXP LPC Microcontroller family devices have many power and ground pins. This is due to having multiple power domains, and the potential for large simultaneous switching currents when all 16-bit or 32-bit external data bus outputs change from all low to all high, or all high to all low, at the same time. It is recommended that the MCU VDD(core) and VDD(IO) power nets and VSSx be distributed on a plane layer of the PCB instead of routed by the thin traces, like those used for carrying other signals. BGA power and ground balls are typically routed to a near by fan-out via the same as any signal. It is recommended that the short trace between the BGA ball and fan-out via be no wider than 0.15mm (6mils). Although it is common to use a wider trace (0.5mm) to route power and ground from other IC packages (SOIC, QFP, TSOP, etc.), using larger than 0.15mm may begin to act like a heat sink that could adversely affect the solder joint. If a BGA power or ground pin must be routed more than 1mm to get to a fan-out via, the trace should be routed the first 1mm with a  $\leq 0.15\text{mm}$  trace then up sized for the balance of the route. It is recommended that all power and ground vias that tie into a plane do so as a solid 360 degree connection, as shown [Fig 7a](#). This provides a lower inductance connection to the plane and will provide a more solid ground plane throughout the BGA area. Avoid the use of thermal (4-point) connections, as shown in [Fig 7b](#).



a. Direct via connection to the plane

b. Thermal (4-point) connection to the plane

**Fig 7. Via to plane connection**

## 7. Legal information

### 7.1 Definitions

**Draft** — The document is a draft version only. The content is still under internal review and subject to formal approval, which may result in modifications or additions. NXP Semiconductors does not give any representations or warranties as to the accuracy or completeness of information included herein and shall have no liability for the consequences of use of such information.

### 7.2 Disclaimers

**General** — Information in this document is believed to be accurate and reliable. However, NXP Semiconductors does not give any representations or warranties, expressed or implied, as to the accuracy or completeness of such information and shall have no liability for the consequences of use of such information.

**Right to make changes** — NXP Semiconductors reserves the right to make changes to information published in this document, including without limitation specifications and product descriptions, at any time and without notice. This document supersedes and replaces all information supplied prior to the publication hereof.

**Suitability for use** — NXP Semiconductors products are not designed, authorized or warranted to be suitable for use in medical, military, aircraft, space or life support equipment, nor in applications where failure or malfunction of a NXP Semiconductors product can reasonably be expected to result in personal injury, death or severe property or environmental damage. NXP Semiconductors accepts no liability for inclusion and/or use of NXP Semiconductors products in such equipment or applications and therefore such inclusion and/or use is for the customer's own risk.

**Applications** — Applications that are described herein for any of these products are for illustrative purposes only. NXP Semiconductors makes no representation or warranty that such applications will be suitable for the specified use without further testing or modification.

### 7.3 Trademarks

Notice: All referenced brands, product names, service names and trademarks are property of their respective owners.

## 8. Contents

---

<b>1.</b>	<b>Introduction .....</b>	<b>3</b>
1.1	Scope .....	3
<b>2.</b>	<b>BGA Package Description .....</b>	<b>3</b>
<b>3.</b>	<b>BGA Footprints.....</b>	<b>4</b>
3.1	Land Pad Design.....	4
3.1.1	Solder mask defined land pad (SMD) .....	5
3.1.2	Non-solder mask defined land pad (NSMD).....	5
3.2	Recommended BGA Footprint .....	5
<b>4.</b>	<b>Recommended Fan-out Trace / Space</b>	
	<b>guidelines.....</b>	<b>7</b>
4.1	Recommended 1.0 and 0.8mm pitch BGA via	
	fan-out pattern .....	7
4.2	Recommended 0.65mm pitch BGA via fan-out	
	pattern .....	8
4.3	Recommended 0.5mm pitch BGA via fan-out	
	pattern .....	9
<b>5.</b>	<b>Board Cost considerations.....</b>	<b>10</b>
5.1	Area Rules .....	10
5.2	How many PCB layers to fan-out the BGA.....	10
<b>6.</b>	<b>BGA Power and Ground .....</b>	<b>11</b>
<b>7.</b>	<b>Legal information .....</b>	<b>12</b>
7.1	Definitions .....	12
7.2	Disclaimers.....	12
7.3	Trademarks .....	12
<b>8.</b>	<b>Contents.....</b>	<b>13</b>

---

Please be aware that important notices concerning this document and the product(s) described herein, have been included in the section 'Legal information'.

---

© NXP B.V. 2009. All rights reserved.

For more information, please visit: <http://www.nxp.com>  
 For sales office addresses, email to: [salesaddresses@nxp.com](mailto:salesaddresses@nxp.com)

Date of release: 22 January 2009  
 Document identifier: AN10778\_1

