AN10778

PCB layout guidelines for NXP MCUs in BGA packages

Rev. 01 — 22 January 2009 **Application note**

Document information

Info	Content
Keywords	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
Abstract	This application note is focused on Printed Circuit Board (PCB) layout issues when using (LF)(TF) BGA packages from the NXP LPC Microcontroller family.

founded by Philips

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

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.

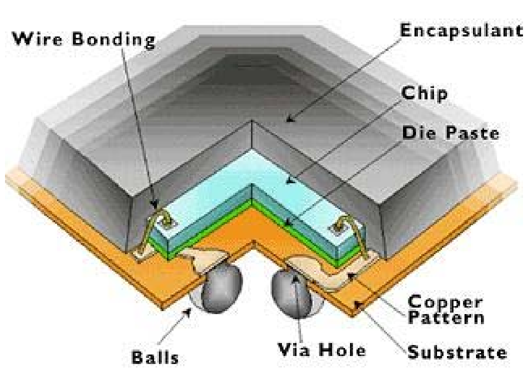


Fig 1. (LF)(TF)BGA Cross Section

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.

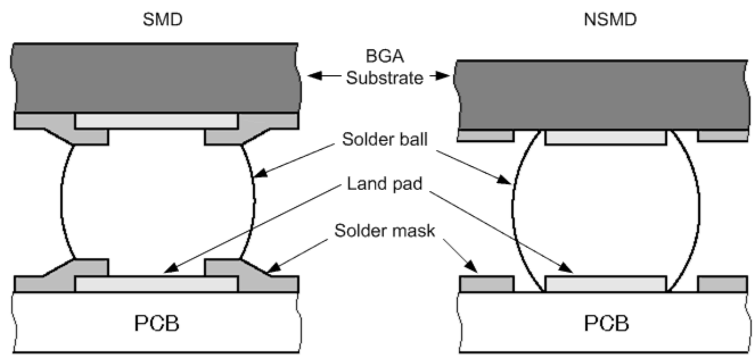


Fig 2. Solder mask vs non-solder mask defined land pad

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 wett 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.

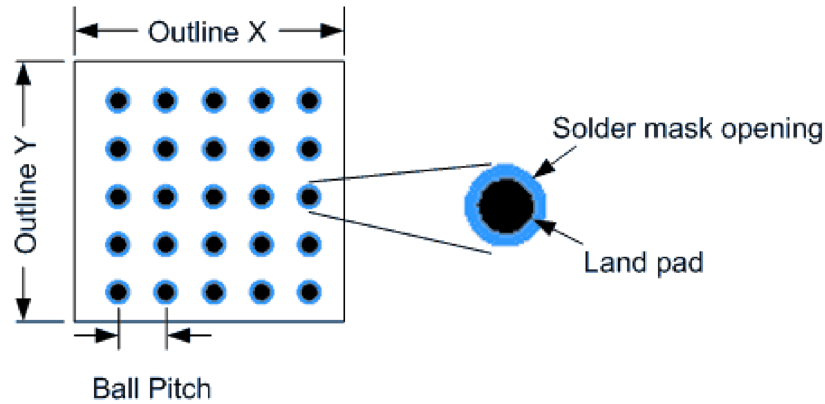


Fig 3. Generic BGA Footprint

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.

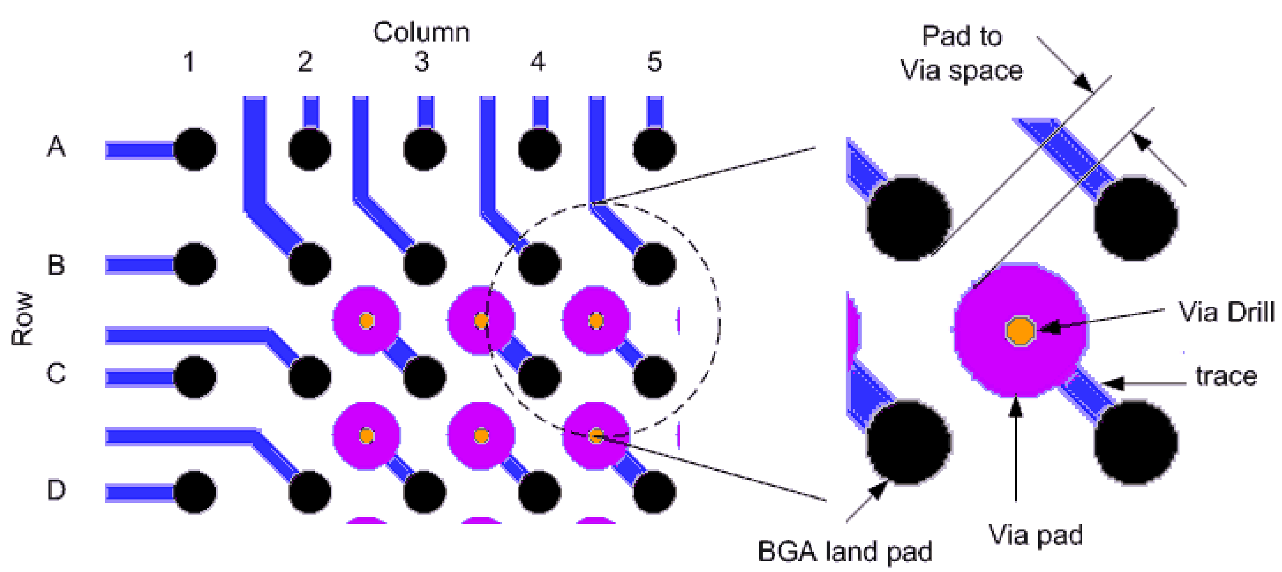


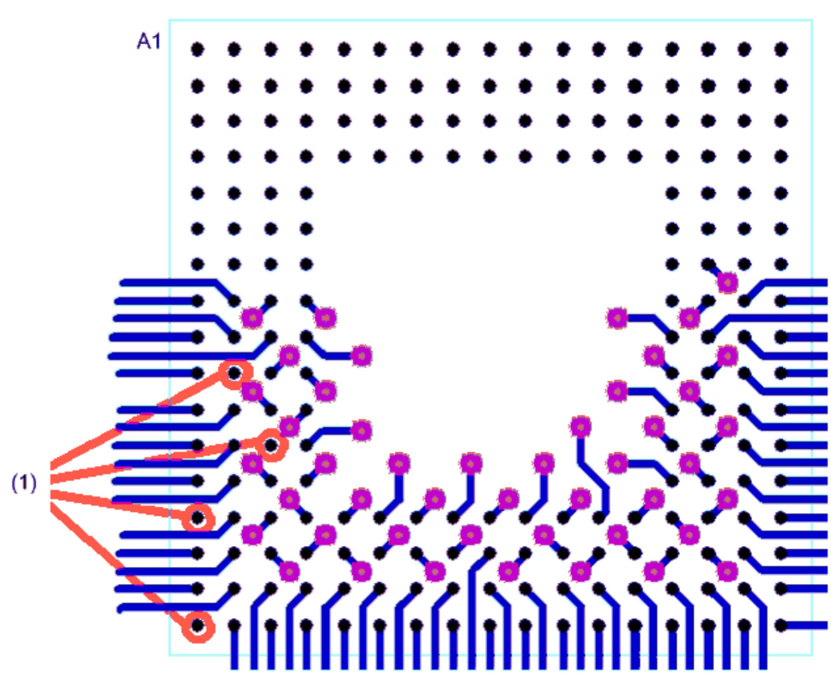
Fig 4. 1.0 and 0.8mm pitch BGA Via fan-out pattern

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.

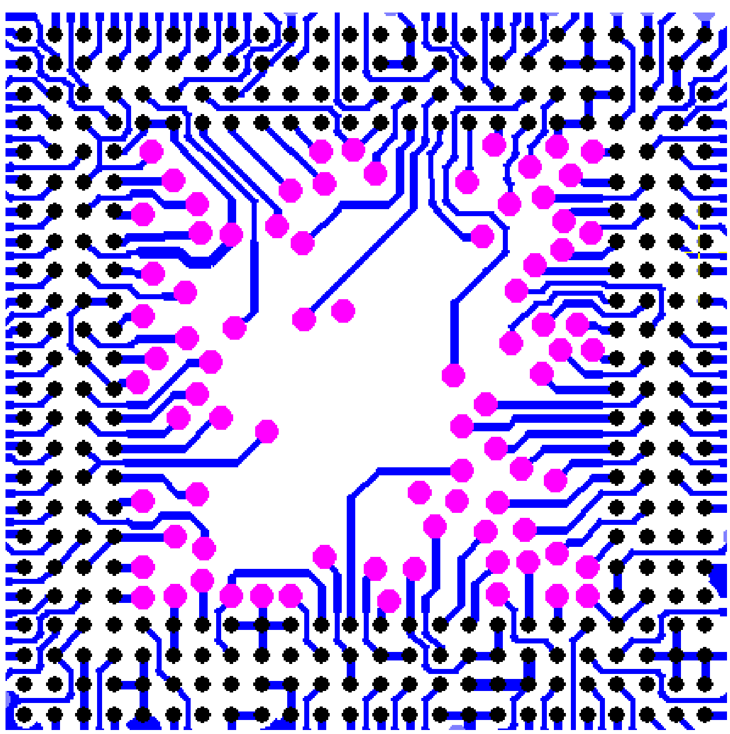


Fig 6. Example fan-out of LFBGA320

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 <= 0.15mm 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

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

For more information, please visit: http://www.nxp.com
For sales office addresses, email to: salesaddresses@nxp.com

Date of release: 22 January 2009
Document identifier: AN10778_1

founded by PHILIPS

射 频 和 天 线 设 计 培 训 课 程 推 荐

易迪拓培训(www.edatop.com)由数名来自于研发第一线的资深工程师发起成立，致力并专注于微波、射频、天线设计研发人才的培养；我们于 2006 年整合合并微波 EDA 网(www.mweda.com)，现已发展成为国内最大的微波射频和天线设计人才培养基地，成功推出多套微波射频以及天线设计经典培训课程和 ADS、HFSS 等专业软件使用培训课程，广受客户好评；并先后与人民邮电出版社、电子工业出版社合作出版了多本专业图书，帮助数万名工程师提升了专业技术能力。客户遍布中兴通讯、研通高频、埃威航电、国人通信等多家国内知名公司，以及台湾工业技术研究院、永业科技、全一电子等多家台湾地区企业。

易迪拓培训课程列表：http://www.edatop.com/peixun/rfe/129.html

射频工程师养成培训课程套装

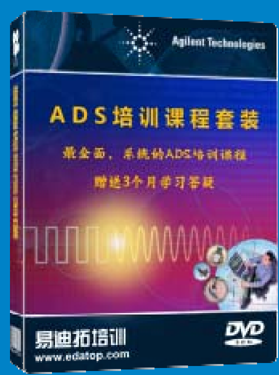
该套装精选了射频专业基础培训课程、射频仿真设计培训课程和射频电路测量培训课程三个类别共 30 门视频培训课程和 3 本图书教材；旨在引领学员全面学习一个射频工程师需要熟悉、理解和掌握的专业知识和研发设计能力。通过套装的学习，能够让学员完全达到和胜任一个合格的射频工程师的要求…

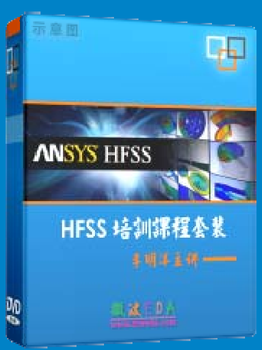
课程网址：http://www.edatop.com/peixun/rfe/110.html

ADS 学习培训课程套装

该套装是迄今国内最全面、最权威的 ADS 培训教程，共包含 10 门 ADS 学习培训课程。课程是由具有多年 ADS 使用经验的微波射频与通信系统设计领域资深专家讲解，并多结合设计实例，由浅入深、详细而又全面地讲解了 ADS 在微波射频电路设计、通信系统设计和电磁仿真设计方面的内容。能让您在最短的时间内学会使用 ADS，迅速提升个人技术能力，把 ADS 真正应用到实际研发工作中去，成为 ADS 设计专家...

课程网址： http://www.edatop.com/peixun/ads/13.html



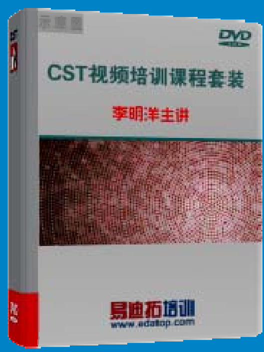


HFSS 学习培训课程套装

该套课程套装包含了本站全部 HFSS 培训课程，是迄今国内最全面、最专业的HFSS培训教程套装，可以帮助您从零开始，全面深入学习HFSS的各项功能和在多个方面的工程应用。购买套装，更可超值赠送 3 个月免费学习答疑，随时解答您学习过程中遇到的棘手问题，让您的 HFSS 学习更加轻松顺畅…

课程网址：http://www.edatop.com/peixun/hfss/11.html

CST 学习培训课程套装



该培训套装由易迪拓培训联合微波 EDA 网共同推出，是最全面、系统、专业的 CST 微波工作室培训课程套装，所有课程都由经验丰富的专家授课，视频教学，可以帮助您从零开始，全面系统地学习 CST 微波工作的各项功能及其在微波射频、天线设计等领域的设计应用。且购买该套装，还可超值赠送 3 个月免费学习答疑…

课程网址：http://www.edatop.com/peixun/cst/24.html

HFSS 天线设计培训课程套装

套装包含 6 门视频课程和 1 本图书，课程从基础讲起，内容由浅入深，理论介绍和实际操作讲解相结合，全面系统的讲解了 HFSS 天线设计的全过程。是国内最全面、最专业的 HFSS 天线设计课程，可以帮助您快速学习掌握如何使用 HFSS 设计天线，让天线设计不再难…

课程网址：http://www.edatop.com/peixun/hfss/122.html

13.56MHz NFC/RFID 线圈天线设计培训课程套装

套装包含 4 门视频培训课程，培训将 13.56MHz 线圈天线设计原理和仿真设计实践相结合，全面系统地讲解了 13.56MHz 线圈天线的工作原理、设计方法、设计考量以及使用 HFSS 和 CST 仿真分析线圈天线的具体操作，同时还介绍了 13.56MHz 线圈天线匹配电路的设计和调试。通过该套课程的学习，可以帮助您快速学习掌握 13.56MHz 线圈天线及其匹配电路的原理、设计和调试…

详情浏览：http://www.edatop.com/peixun/antenna/116.html

我们的课程优势：

※ 成立于 2004 年，10 多年丰富的行业经验，

※ 一直致力并专注于微波射频和天线设计工程师的培养，更了解该行业对人才的要求

※ 经验丰富的一线资深工程师讲授，结合实际工程案例，直观、实用、易学

联系我们：

※ 易迪拓培训官网：http://www.edatop.com

※ 微波 EDA 网：http://www.mweda.com

※ 官方淘宝店：http://shop36920890.taobao.com