# PCB Mounting Practices to Obtain Optimum Performance for High-Voltage BGA and LGA Packages

*Author: Rohit Tirumala*
*Microchip Technology Inc.*

## INTRODUCTION

The plastic, or over-molded, Ball Grid Array (BGA) is a lead-less, low-profile surface-mount package. It offers a high number of potential input and output connections. Lead-less packages provide short, heavy copper (Cu) paths with low inductance (L) and resistance (R) connections from packages to Printed Circuit Board (PCB), enabling high-speed operation with low switching and conduction losses. Furthermore, the short vertical structure provides very low $R_{JC}$ (junction-to-case or die-to-ball thermal impedance) for high-power dissipation. The BGA package was developed for high density and high-speed digital devices, but has been increasingly used for packaging power devices, both single and multi-chip. Figure 1 shows an example of a 36-terminal BGA package outline.

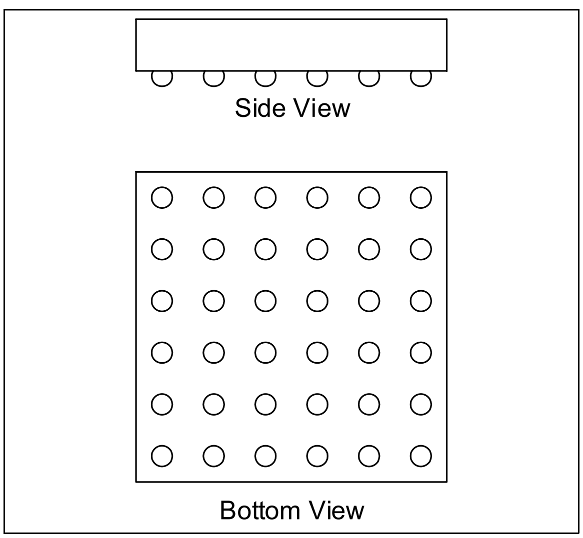


***FIGURE 1:*** *36-Terminal BGA Package Outline Drawing Example.*

Power packages typically require fewer connections, but do require higher current connections. Therefore, power BGA packages have larger copper land areas with multiple solder balls for higher current connections and lower $R_{JC}$. Even lower package $R_{JC}$, L and R are achieved with Land Grid Array (LGA) packages. The LGA replaces solder balls/pads with larger area solder pads. BGA ball arrays are usually standard ball sizes and pitch, whereas LGA pads are almost always custom to maximize package-to-PCB contact areas for each product. Figure 2 shows an example of a 3-terminal LGA package outline.

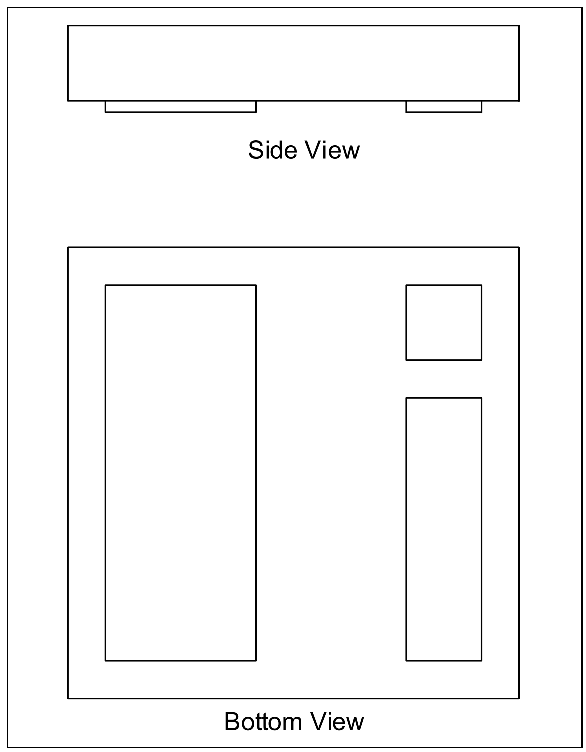


***FIGURE 2:*** *3-Terminal LGA Package Outline Drawing Example.*

Plastic BGA and LGA devices are molded packages with an internal PCB substrate for die/component assembly. The PCB substrate is typically multilayer, with copper trace layers and interconnecting copper vias optimized for electrical and thermal performance by the BGA and LGA device manufacturers. Figure 3 shows an example of a 36-terminal BGA Digital IC package structure, and Figure 4 shows an example of a 3-terminal LGA power transistor package.

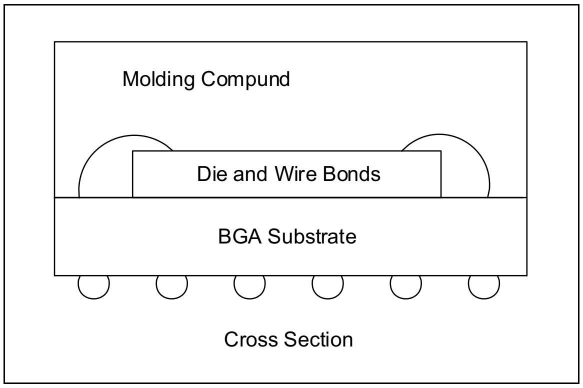


***FIGURE 3:*** *36-Terminal BGA Digital IC Package Structure Example.*

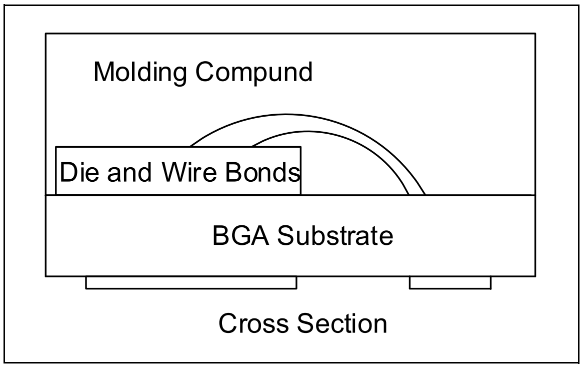


***FIGURE 4:*** *3-Terminal LGA Power Transistor Package Structure Example.*

BGA and LGA users must build on these electrical and thermal advantages with suitable Solder Mask Defined (SMD) solder connections to the PCBs and optimum electrical and thermal PCB design. This application note will provide recommendations for BGA and LGA PCB design and assembly techniques.

## BGA AND LGA ASSEMBLY MATERIALS, TECHNIQUES AND CONFIGURATIONS

The BGA package can accommodate many solder ball materials, size and pitch, but in practice there are only a few common/standard ball solders, sizes and pitches.

Solder balls are usually Lead-Free (Tin/Silver or Tin/Silver/Copper) and Eutectic Tin/Lead. Pb-Free solders are generally preferred, but Sn/Pb solders are still common for compatibility with older processes and components and for some lower temperature devices. The same or similar type solder pastes are typically used for BGA and LGA assembly to the PCB.

Standard ball pitch/diameters and associated ball separations are given in Table 1.

**TABLE 1: STANDARD BALL PITCH, DIAMETERS AND SEPARATIONS FOR BGA PACKAGES**

| Ball Pitch (mm) | Ball Diameter (mm) | Ball Separation (mm) |
|---|---|---|
| 0.75 | 0.40 | 0.35 |
| 1.00 | 0.40 | 0.60 |
| 1.00 | 0.50 | 0.50 |
| 1.00 | 0.60 | 0.40 |
| 1.27 (50 mil) | 0.51 (20 mil) | 0.76 (30 mil) |
| 1.27 (50 mil) | 0.64 (25 mil) | 0.63 (~25 mil) |

The larger LGA pads are usually coated with the same solders (Pb-free or Sn/Pb), but the lower profile solder on the pads does not contain solder balls. LGA pads do not have standard sizes, and are designed/optimized to accommodate specific products and applications.

## PCB Solder Pads for BGA and LGA

Solder and terminal pads located on BGA/LGA PCB are either SMD or Non-Solder Mask Defined (NSMD), and sometimes both SMD and NSMD on the same PCB. NSMD are generally preferred and more accurate/reliable, but SMD accommodate larger land areas for multi-ball connections, and can accommodate higher currents and power dissipation. Figure 5 shows a PCB SMD Terminal Pad/Resist layout and cross-section. Figure 6 shows a PCB NSMD Terminal Pad/Resist layout and cross section.

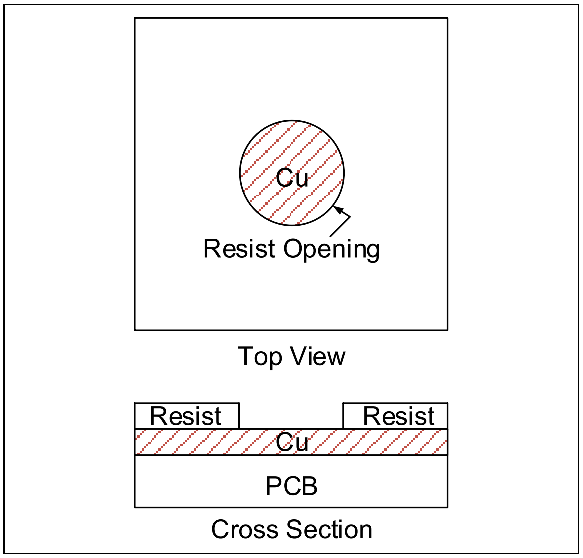


***FIGURE 5: PCB SMD Terminal Pad/Resist Layout.***

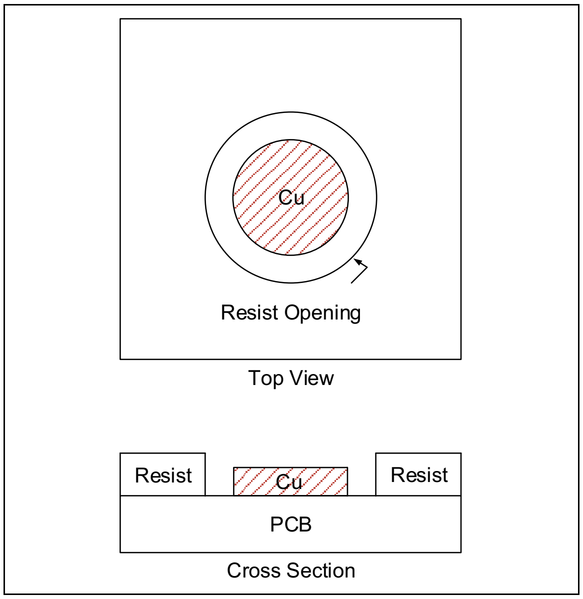


***FIGURE 6: PCB NSMD Terminal Pad Resist Layout.***

- Recommended SMD PCB resist opening diameters are 0.075 – 0.10 mm less than ball diameters.
- Recommended NSMD PCB pad and resist openings are given in Table 2.
- Recommended LGA solder resists opening are 0.05 mm larger than the LGA pads, but custom pad sizes and shapes for LGA often require different rules, which are usually provided by the LGA device manufacturer.

**TABLE 2: RECOMMENDED NSMD PCB PAD AND RESIST OPENINGS**

| Ball Pitch/Diameter (mm) | Solder Resist Opening Diameter (mm) | Solder Pad Diameter (mm) |
|---|---|---|
| 0.75/0.40 | 0.45 | 0.30 |
| 1.00/0.40 | 0.45 | 0.30 |
| 1.00/0.50 | 0.50 | 0.35 |
| 1.00/0.60 | 0.60 | 0.40 |
| 1.27/0.51 | 0.75 | 0.60 |
| 1.27/0.64 | 0.80 | 0.65 |

Standard recommendations are generally suited for lower voltages between pads, and larger spacing may be required at higher voltages. Higher voltage (HV) minimum pad spacing guidelines should be compliant with UL Pollution degree 1 minimum creepage paths, assuming a suitable HV underfill (see Table 3 for a condensed list of creepage distances). The HV device manufacturer will typically recommend optimum PCB pads for attaching BGA and LGA products, but these may vary for some user applications.

**TABLE 3: CREEPAGE DISTANCES FOR POLLUTION LEVEL 1 FROM UL840**

| Operating Voltage (Volts, AC RMS or DC) | Minimum Creepage Distance (mm) |
|---|---|
| 80 | 0.22 |
| 100 | 0.25 |
| 160 | 0.32 |
| 200 | 0.42 |
| 250 | 0.56 |
| 320 | 0.75 |
| 400 | 1.00 |

Example of underfill materials are:

- Loctite® 3153™ for BGA packages
- Hysol® E1926 for LGA packages

## Printed Circuit Boards for BGA and LGA

PCBs are typically multilayer to accommodate interconnects for higher density balls and pads. The copper pads and traces on different layers are interconnected with copper vias. These vias create both electrical and thermal connections between the layers. Vias at pads and under balls must be filled or tented to provide flat and solderable pad surfaces. This is especially important for BGA pads, which must ensure accurate ball locations.

High glass transition ($T_g$) FR-4 laminate materials are most common and economical, but other higher performance, high-$T_g$ laminates may be used to meet special electrical, mechanical or thermal requirements. In general, the laminate $T_g$ should be greater than +170°C.

The BGA and LGA solder pads are copper, and a number of different finishes are available with differences in cost, shelf life and long-term reliability. Common finishes are Organic Solderability Preservatives (OSP), Hot Air Leveling, Sn/Pb (HASL), Electroless Sn, Electroless Ag and Electroless Ni/Au. The choice of pad finish must be made based on specific products, applications and the PCB fabricators capabilities.

## Soldering BGA and LGA Packages to Printed Circuit Boards

The key processes are:

1. Stencil solder paste onto PCB pads
2. Place BGA/LGA into solder paste
3. Reflow the solder
4. Clean and remove flux
5. Add underfill.

There are variations and options for specific materials, equipment, devices and assemblers, but these processes are used for almost all BGA and LGA assembly.

### STENCIL AND SOLDER PASTE

Solder pastes are usually Pb-Free (Sn/Ag/Cu), are Eutectic Sn/Pb, and it is recommended that Pb-Free solder paste be used with Pb-Free solder balls and that Sn/Pb solder paste be used with Sn/Pb solder balls. Mixing paste and ball solder types is possible, but should be done cautiously.

Solder pastes should be stencil grade with high solids/solder (typically 90% by weight). No-clean solder pastes are preferred, but flux removal/cleaning is possible. There are many types of flux systems including aqueous, semi-aqueous and solvent based. New and complex cleaning chemistries are very specialized, and the cleaning chemistry recommended by the solder paste manufacturers is usually the best option.

Stainless steel stencils are recommended, with the stencil apertures and thicknesses shown in Table 4. These may vary based on the solder paste and print parameters.

**TABLE 4: STENCIL APERTURES AND THICKNESSES FOR BGA PACKAGES**

| Ball Pitch/Diameter (mm) | Stencil Aperture Diameter (mm) | Stencil Thickness (mm) |
|---|---|---|
| 0.75/0.40 | 0.35 | 0.125 |
| 1.00/0.40 | 0.35 | 0.125 |
| 1.00/0.50 | 0.40 | 0.150 |
| 1.00/0.60 | 0.45 | 0.150 |
| 1.27/0.51 | 0.55 | 0.150 |
| 1.27/0.64 | 0.60 | 0.150 |

### PLACEMENT OF BGA/LGA INTO SOLDER PASTE

The placement of BGA and LGA must be automatic to ensure the typical requirements of ±0.10 mm, which is within the ±0.05 mm accuracy of common pick and place equipment.

REFLOW SOLDERING

The recommended solder reflow equipment is Forced Air Convection (FAC), or a combination of FAC and Infrared (IR). Other types of equipment can achieve the required profile, but are less common and require special care to control.

There are two standard profiles solder reflow profiles: one for Pb-Free (Sn/Ag and Sn/Ag/Cu) solder, and the other for Eutectic Sn/Pb solder. The recommended profiles are shown in Figure 7 and Figure 8, respectively. In general, it is recommended to begin with the solder paste manufacturers' recommended profile, which addresses their solder composition, solder particle size/shape, and organic system. Any recommended profiles are a starting point, and must be adjusted to match the requirements of specific products and assembly processes. Factors to consider include the product size and mass, the type of assembly equipment, carriers and more. The nominal peak temperatures are +245°C for Pb-Free solder and +225°C for Eutectic Sn/Pb solder.

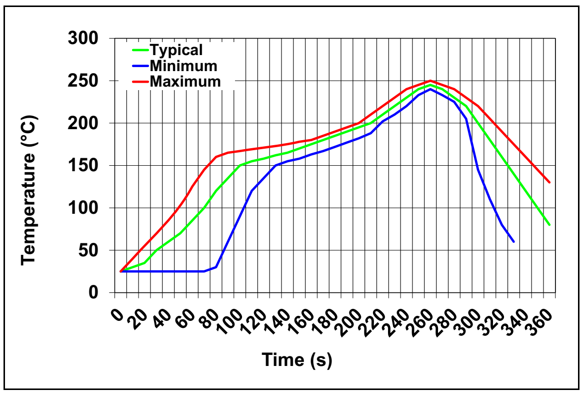


***FIGURE 7:*** *Typical Pb-Free (Sn/Ag and Sn/Ag/Cu) Solder Reflow Profile.*

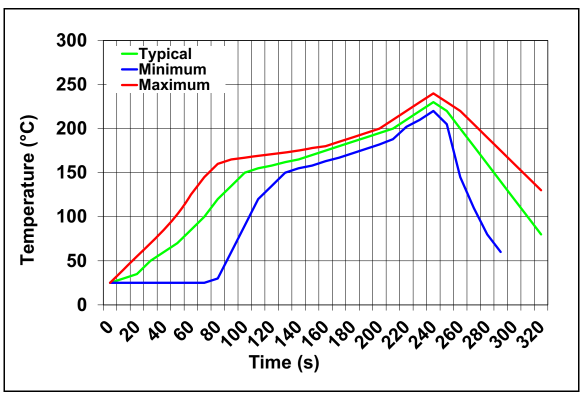


***FIGURE 8:*** *Typical Eutectic Sn/Pb Solder Reflow Profile.*

### CLEAN AND REMOVE FLUX

Cleaning and baking are recommended, and may be advantageous even for no-clean solder pastes. Solder pastes containing fluxes require more aggressive cleaning materials and processes. The specific cleaning chemistry and equipment required is dependent on the solder paste flux and organic chemistry, and the solder paste manufacturers' recommendations are the best starting point for determining cleaning chemistry and associated cleaning equipment. Again, any cleaning process will need to be optimized for specific solder reflow processes and products.

### UNDERFILL FOR BGA AND LGA

Underfill is recommended for all BGA and LGA assemblies, and it should be used automatically unless there is a good reason not to use it. It is even more important with larger packages, higher power dissipation and higher voltages. It can:

- Provide mechanical strength against vibration, coefficient of thermal expansion (CTE) mismatch and other stresses
- Improve heat transfer to the PCB
- Eliminate dirt, moisture and other contaminates that can collect under the BGA and LGA packages initially and over the life of the product
- Eliminate high-voltage arcing or leakage between pads, and allow reduced spacing between pads. The minimum spacing with underfill should meet UL Pollution degree 1 minimum creepage, but specific products and application conditions may allow tighter spacing or require larger spacing.

Underfills are very specialized with specific formulations for BGA and LGA. Their viscosity is adjusted for specific space height to make underfill application easy and virtually void free. It is generally applied along one or two adjacent edges of the BGA and LGA package, where the surface tension pulls the underfill under the package while pushing air out the opposite edges.

### INSPECTION OF BGA/LGA PACKAGES

The BGA and LGA solder pads are hidden under the packages and cannot be visually inspected. However, current solders pastes, stencils and placement equipment are well controlled and can provide very high assembly yields. The yields are generally verified with 100% testing, and sometimes supported by X-ray inspection. X-rays are well suited for sample checking and analysis of solder defects.

### BGA AND LGA SOLDERING REWORK

Although rework should not be required in a well-controlled assembly process, it is possible to remove and replace BGA and LGA packages. However, it is not possible to rework individual solder joints. Removal is usually accomplished by heating locally below the device to be removed and removing the device after the solder melts. Before replacing the device with a new BGA/LGA, the pads must be clean and flat. The replacement device will typically need added flux to hold during automatic placement, and should also be heated locally from the back of the PCB. Any rework should be performed before underfill.

## BGA AND LGA RECOMMENDATIONS DISCLAIMER

The BGA and LGA recommendations are based on typical products, materials, applications and assembly processes/equipment. Although these recommendations will often be sufficient, they should always be validated for the specific products and product requirements. This is especially important for high-power and high-voltage products, which can have unique requirements.

ISBN: 978-1-5224-0128-5

# Worldwide Sales and Service

## AMERICAS

**Corporate Office**
2355 West Chandler Blvd.
Chandler, AZ 85224-6199
Tel: 480-792-7200
Fax: 480-792-7277
Technical Support:
http://www.microchip.com/support
Web Address:
www.microchip.com

**Atlanta**
Duluth, GA
Tel: 678-957-9614
Fax: 678-957-1455

**Austin, TX**
Tel: 512-257-3370

**Boston**
Westborough, MA
Tel: 774-760-0087
Fax: 774-760-0088

**Chicago**
Itasca, IL
Tel: 630-285-0071
Fax: 630-285-0075

**Cleveland**
Independence, OH
Tel: 216-447-0464
Fax: 216-447-0643

**Dallas**
Addison, TX
Tel: 972-818-7423
Fax: 972-818-2924

**Detroit**
Novi, MI
Tel: 248-848-4000

**Houston, TX**
Tel: 281-894-5983

**Indianapolis**
Noblesville, IN
Tel: 317-773-8323
Fax: 317-773-5453

**Los Angeles**
Mission Viejo, CA
Tel: 949-462-9523
Fax: 949-462-9608

**New York, NY**
Tel: 631-435-6000

**San Jose, CA**
Tel: 408-735-9110

**Canada - Toronto**
Tel: 905-673-0699
Fax: 905-673-6509

## ASIA/PACIFIC

**Asia Pacific Office**
Suites 3707-14, 37th Floor
Tower 6, The Gateway
Harbour City, Kowloon

**Hong Kong**
Tel: 852-2943-5100
Fax: 852-2401-3431

**Australia - Sydney**
Tel: 61-2-9868-6733
Fax: 61-2-9868-6755

**China - Beijing**
Tel: 86-10-8569-7000
Fax: 86-10-8528-2104

**China - Chengdu**
Tel: 86-28-8665-5511
Fax: 86-28-8665-7889

**China - Chongqing**
Tel: 86-23-8980-9588
Fax: 86-23-8980-9500

**China - Dongguan**
Tel: 86-769-8702-9880

**China - Hangzhou**
Tel: 86-571-8792-8115
Fax: 86-571-8792-8116

**China - Hong Kong SAR**
Tel: 852-2943-5100
Fax: 852-2401-3431

**China - Nanjing**
Tel: 86-25-8473-2460
Fax: 86-25-8473-2470

**China - Qingdao**
Tel: 86-532-8502-7355
Fax: 86-532-8502-7205

**China - Shanghai**
Tel: 86-21-5407-5533
Fax: 86-21-5407-5066

**China - Shenyang**
Tel: 86-24-2334-2829
Fax: 86-24-2334-2393

**China - Shenzhen**
Tel: 86-755-8864-2200
Fax: 86-755-8203-1760

**China - Wuhan**
Tel: 86-27-5980-5300
Fax: 86-27-5980-5118

**China - Xian**
Tel: 86-29-8833-7252
Fax: 86-29-8833-7256

## ASIA/PACIFIC

**China - Xiamen**
Tel: 86-592-2388138
Fax: 86-592-2388130

**China - Zhuhai**
Tel: 86-756-3210040
Fax: 86-756-3210049

**India - Bangalore**
Tel: 91-80-3090-4444
Fax: 91-80-3090-4123

**India - New Delhi**
Tel: 91-11-4160-8631
Fax: 91-11-4160-8632

**India - Pune**
Tel: 91-20-3019-1500

**Japan - Osaka**
Tel: 81-6-6152-7160
Fax: 81-6-6152-9310

**Japan - Tokyo**
Tel: 81-3-6880- 3770
Fax: 81-3-6880-3771

**Korea - Daegu**
Tel: 82-53-744-4301
Fax: 82-53-744-4302

**Korea - Seoul**
Tel: 82-2-554-7200
Fax: 82-2-558-5932 or
82-2-558-5934

**Malaysia - Kuala Lumpur**
Tel: 60-3-6201-9857
Fax: 60-3-6201-9859

**Malaysia - Penang**
Tel: 60-4-227-8870
Fax: 60-4-227-4068

**Philippines - Manila**
Tel: 63-2-634-9065
Fax: 63-2-634-9069

**Singapore**
Tel: 65-6334-8870
Fax: 65-6334-8850

**Taiwan - Hsin Chu**
Tel: 886-3-5778-366
Fax: 886-3-5770-955

**Taiwan - Kaohsiung**
Tel: 886-7-213-7828

**Taiwan - Taipei**
Tel: 886-2-2508-8600
Fax: 886-2-2508-0102

**Thailand - Bangkok**
Tel: 66-2-694-1351
Fax: 66-2-694-1350

## EUROPE

**Austria - Wels**
Tel: 43-7242-2244-39
Fax: 43-7242-2244-393

**Denmark - Copenhagen**
Tel: 45-4450-2828
Fax: 45-4485-2829

**France - Paris**
Tel: 33-1-69-53-63-20
Fax: 33-1-69-30-90-79

**Germany - Dusseldorf**
Tel: 49-2129-3766400

**Germany - Karlsruhe**
Tel: 49-721-625370

**Germany - Munich**
Tel: 49-89-627-144-0
Fax: 49-89-627-144-44

**Italy - Milan**
Tel: 39-0331-742611
Fax: 39-0331-466781

**Italy - Venice**
Tel: 39-049-7625286

**Netherlands - Drunen**
Tel: 31-416-690399
Fax: 31-416-690340

**Poland - Warsaw**
Tel: 48-22-3325737

**Spain - Madrid**
Tel: 34-91-708-08-90
Fax: 34-91-708-08-91

**Sweden - Stockholm**
Tel: 46-8-5090-4654

**UK - Wokingham**
Tel: 44-118-921-5800
Fax: 44-118-921-5820

07/14/15