Application Note AN-1168

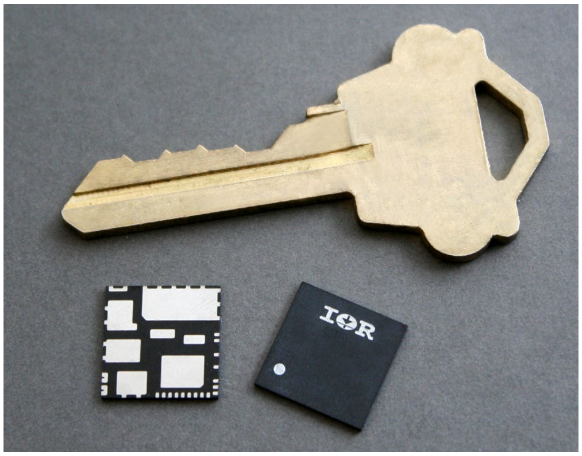
Integrated Power Module (µIPM™) Power Quad Flat No-Lead (PQFN) Board Mounting Application Note

Table of Contents

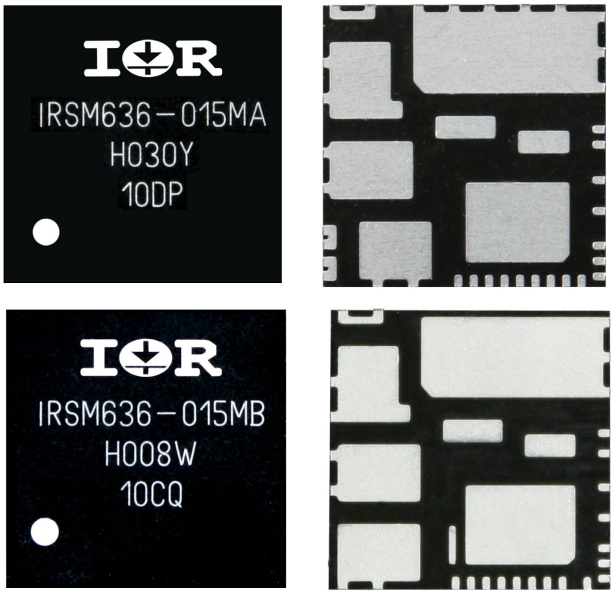
The PQFN package family comprises efficient devices with a wide range of input voltages, all of which are lead-free as indicated by the PbF suffix after the part number (for example, IRFH3800PbF). There are various sizes and outlines. The main text of this application note contains guidance applicable to the whole µIPM range of PQFN devices, while Appendix A contains device outlines, substrate layouts and stencil designs for each device. For more detail about individual devices, refer to the relevant product data sheet and package outline drawing. To simplify board mounting and improve reliability, International Rectifier manufactures µIPM PQFN devices to exacting standards. These high standards have evolved through evaluating many different materials and designs. Although such evaluations have yielded good results, the recommendations in this application note may need to be adjusted to suit specific production environments.

Introduction

Power Quad Flat No-Lead (PQFN) is a surface mount semiconductor technology designed primarily for board-mounted power applications. It eliminates unnecessary elements of packaging that contribute to higher inductance and resistance, both thermal and electrical, so that its power capabilities exceed those of comparably sized packages.



The PQFN package family includes various sizes and device outlines, some containing one chip (discrete devices) and others containing multiple chips.



To simplify board mounting and improve reliability, International Rectifier manufactures µIPM PQFN devices to exacting standards. These high standards have evolved through evaluating many different materials and designs. Although such evaluations have yielded good results, the recommendations in this application note may need to be adjusted to suit specific production environments.

For information about discrete PQFN devices, refer to AN-1136 and AN-1154. For information about PowIRstage PQFN devices, refer to AN-1169. For information about Audio PQFN devices, refer to AN-1170. For information about SupIRBuck™ PQFNs, refer to AN-1132 and AN-1133.

Device construction

µIPM PQFN devices are surface mounted and use current plastic-molding techniques with wire bond interconnects, as shown in Figure 1.

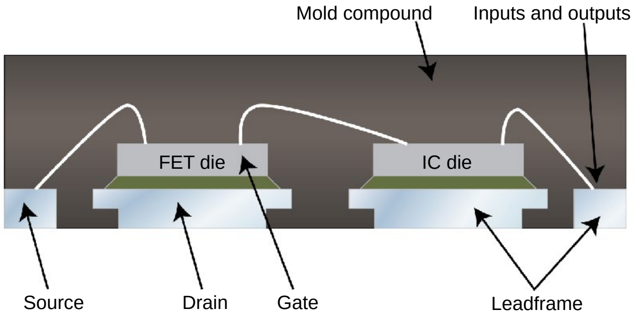


Figure 1 Sectional view

Figure 2 shows a sample contact configuration for a µIPM PQFN device. Specific pad assignments are shown in the data sheet for each product.

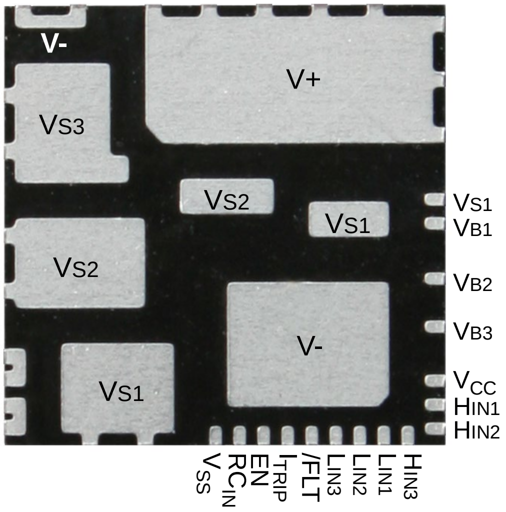


Figure 2 Sample contact pad configuration

Figure 3 shows how µIPM PQFN devices are labeled. Part number, batch number and date code are provided to support product traceability.

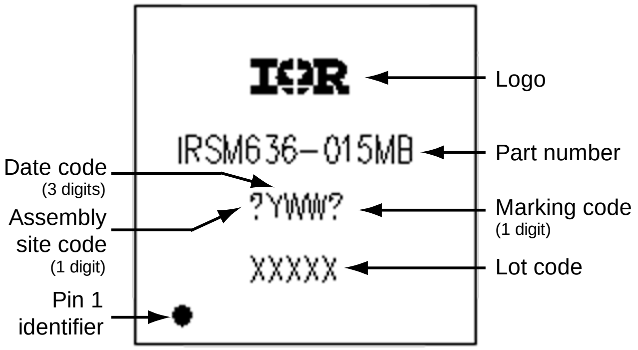


Figure 3 Device markings

The position of Pin 1 is indicated in two ways:

- A dot on the top side (Figure 4).
- A chamfer on a pad on the underside (Figure 5); this is on the pad area closest to Pin 1.

Figure 4 Pin 1 indicator on a µIPM PQFN device

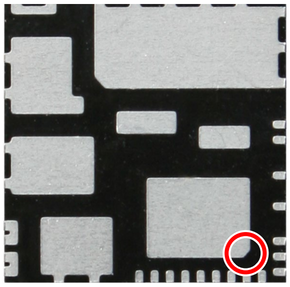


Figure 5 Pin 1 indicator on a µIPM PQFN device

Design considerations

Substrates

The PQFN was originally developed and evaluated for use with epoxy glass-woven substrates (FR-4). The test substrates were finished in Organic Solderability Preservative (OSP), but any of the numerous surface finishes available are suitable.

The substrate finish can affect the amount of energy required to make solder joints; this can in turn be a factor in solder quality issues such as solder balling, tombstoning (or tilt) and the formation of voids.

Substrate designs

To achieve low-loss track layouts, µIPM PQFN devices were designed for use with layouts that use solder-mask-defined (SMD) pad lands and non-solder-mask-defined (NSMD) lead lands. SMD pad lands allow the underlying copper traces to be as large as possible, which increases their ability to carry current and cool devices. NSMD (also known as copper-defined) lead lands maximize the tolerance of misaligned leads and the control of land pad dimensions.

However, µIPM PQFN devices have also been evaluated with entirely NSMD layouts. Using SMD pad lands is not essential and may be eliminated if doing so improves compatibility with existing processes.

When using SMD lands, the underlying copper traces should be at least 0.05mm larger on each edge than the openings in the solder mask. This allows for layers to be misaligned by up to 0.1mm on both axes.

When using NSMD lands, the openings in the solder mask should be at least 0.025mm larger on each edge than the copper pads. This allows for layers to be misaligned by up to 0.05mm on both axes.

The solder mask between lead lands, and between lead and pad lands, should be at least 0.15mm wide. Narrower strips may allow solder bridges to form.

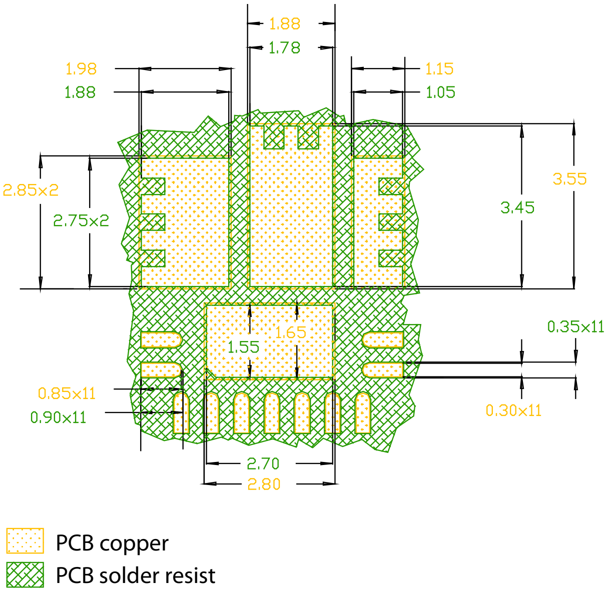


Figure 6 Sizing the areas of copper and solder mask

µIPM PQFN devices can be placed in parallel using simple layouts (Figure 7). International Rectifier recommends a minimum separation of 0.500mm (0.020"). The separation can be adjusted to reflect local process capabilities but should allow for rework. Micro-screen design and desoldering tool type may affect how closely devices are placed to each other and to other components.

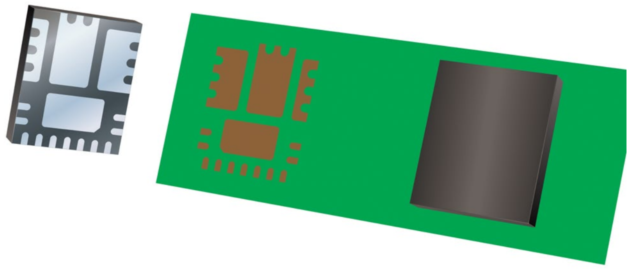


Figure 7 Placing µIPM PQFN devices in parallel

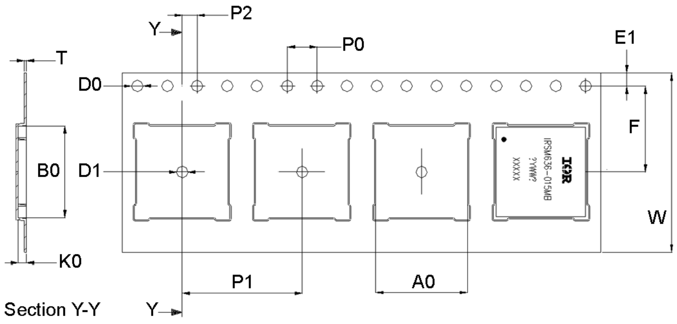
Refer to Appendix A for device outlines, substrate layouts and stencil designs for each package size and device outline in the PQFN range.

Assembly considerations

International Rectifier designed µIPM PQFN devices to be as easy as possible to assemble using standard surface mounting techniques. However, procedures and conditions can have a profound influence on assembly quality. It is therefore necessary to develop an effective process based on the individual requirements for the application.

Packaging

µIPM PQFN devices are supplied in tape and reel format (Figure 8).



	Dimensions (mm)					
	7x8		8x9		12x12	
Code	**Min**	**Max**	**Min**	**Max**	**Min**	**Max**
A0	7.15	7.35	8.20	8.40	12.20	12.40
B0	8.15	8.35	9.20	9.40	12.20	12.40
D0	1.50	1.60	1.50	1.60	1.50	1.60
D1	1.50	—	1.50	—	1.50	—
E1	1.65	1.85	1.65	1.85	1.65	1.85
F	7.40	7.60	7.40	7.60	11.40	11.60
K0	1.10	1.30	1.20	1.40	1.00	1.20
P0	3.90	4.10	3.90	4.10	3.90	4.10
P1	11.90	12.10	11.90	12.10	15.90	16.10
P2	1.90	2.10	1.90	2.10	1.90	2.10
T	0.25	0.35	0.25	0.35	0.25	0.35
W	15.70	16.30	15.70	16.30	23.70	24.30

Figure 8 Tape and reel packaging

Storage requirements

µIPM PQFN devices are packed in sealed, nitrogen-purged, antistatic bags. The sealed bags provide adequate protection against normal light levels but it is prudent to avoid prolonged exposure to bright light sources. The bags also provide protection from the ambient atmosphere. Devices in sealed, unopened bags have a shelf life of one year.

The package labeling shows whether devices should be treated as Moisture Sensitivity Level (MSL) 1, 2 or 3 after a bag has been opened. Appropriate storage is important to guarantee good solderability.

International Rectifier recommends that, when not in use, reels of devices should be resealed into the protective bags in which they were supplied.

Solder pastes

International Rectifier evaluated different types of solder paste from various manufacturers. The properties of pastes vary from manufacturer to manufacturer, meaning that some perform better than others. In general, high slumping pastes tend to suffer more from solder balling than slump-resistant pastes; solder balling is discussed in the next section on stencil design. In addition, some pastes appear to be more prone to voiding than others.

Solder alloys, metal contents and flux constituents all influence the rheology of the solder paste. This in turn influences how the paste reacts during processing. The assembly and board-level reliability of the PQFN package have only been evaluated using lead-free pastes (Sn96.5 Ag3.0 Cu0.5).

International Rectifier’s testing has also shown that halogen-free (or zero-halogen) solder pastes perform better than pastes that contain halogen. The flux residues after reflow are cleaner and more inert. There are fewer instances of voltage tracking.

International Rectifier qualifies devices using the reflow profile outlined in J STD 020C but the reflow profile that delivers the best results will depend on the solder paste used in board mounting. Different pastes from different suppliers behave in different ways. The technical datasheet for the solder paste in use may include advice on setting up the optimal reflow profile and should be consulted as a starting point.

Stencil design

The stencil design is instrumental in controlling the quality of the solder joint. Appendix A shows stencil designs that have given good results with the recommended substrate outlines. The reductions relative to the PCB pad size in these designs depend on the pad size itself. In general, larger pads require greater reduction. For example, the smaller single pads around the outer edges of a device typically have a reduction of 10-15%, while the larger central power pad has a reduction of up to 45%.

Designs are shown for stencil thicknesses of 0.127mm (0.005") and 0.150mm (0.006"). A thickness of 0.127mm (0.005") is optimal. International Rectifier does not recommend the use of stencils thinner than 0.100mm and thicker than 0.150mm.

As a general guideline, the reductions relative to the PCB pad sizes for stencils of different thicknesses are:

- 50% for a 0.127mm (0.005") stencil
- 45% for a 0.150mm (0.006") stencil

The aim when depositing solder paste is to produce a solder joint where voids account for less than 25% of the area. The solder joints on the small outer edge pads should be at least 0.05mm (0.002") thick. There is no minimum requirement for the larger pads.

Note: The thickness of the solder mask on the PCB can affect the thickness of the solder joints. This is because the solder mask can act as a gasket, with the effect of making the solder paste stencil thicker. This can cause the solder joints to be thicker than expected (or calculated). The designs in Appendix A assume a solder mask thickness of 0.025mm (0.001").

Solder joint voiding

The main causes of solder joint voiding are applying the incorrect volume of solder paste and/or using a reflow profile that is not optimized.

If an unacceptable amount of voiding is observed in solder joints, experimentation may be required to design a solder paste stencil that applies an appropriate amount of solder paste. Breaking up pads into smaller blocks can yield good results (Figure 9).

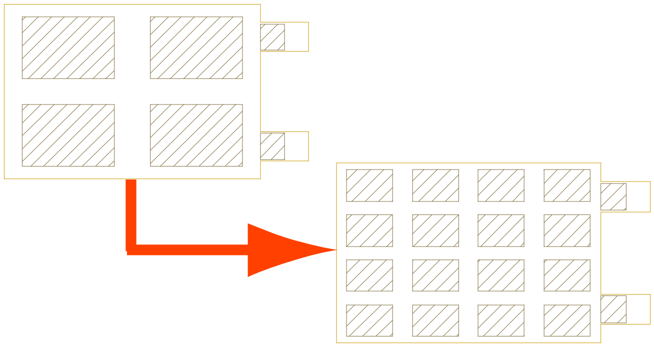


Figure 9 Dividing large pads into smaller blocks

Device placement

Inaccurate placement may result in poor solder joints or in devices being tilted and/or misaligned. Ideally, µIPM PQFN devices should be placed to an accuracy of 0.050mm on both X and Y axes but, during evaluations, devices centered themselves from placement inaccuracies of more than 0.300mm. Self-centering behavior is highly dependent on solders and processes, and experiments should be run to confirm the limits of self-centering on specific processes.

Reflow equipment

µIPM PQFN devices are suitable for assembly using surface mount technology reflowing equipment and are recommended for use with convection, vapor phase and infrared equipment. PbF qualified devices have a good resistance to short-term exposure to high temperatures, making them suitable for reflow profiles of up to 260°C (measured by attaching a thermocouple to a µIPM PQFN device).

There are no special requirements for successful assembly, but all reflow processes used in evaluation and qualification complied with the recommendations of solder paste suppliers. Using incorrect reflow profiles can cause solder quality issues such as solder balling, tombstoning (or tilt) and the formation of voids; if such problems arise, the reflow profile should be checked.

The PQFN package is designed to have superior thermal conductivity properties. For this reason, it is essential that the core of the substrate reaches thermal equilibrium during the pre-heating stage of the reflow profile to ensure that adequate thermal energy reaches the solder joint.

The designs for PCB layouts and solder paste stencils shown in Appendix A are suitable for all assembly styles, irrespective of size and complexity. The only caveat is that the reflow profile at the device must comply with the solder paste manufacturer's recommendations. If, for example, the PCB is large and/or heavy, the reflow profile may need to be slowed down to achieve good thermal equilibrium across the whole PCB. The same result may be achieved by making the pre-heat soak or plateau longer and/or hotter.

Failure to meet these requirements may result in unspent or non-volatilized flux residues being left underneath the device (Figure 10). This can result in current leakage and/or short circuits.

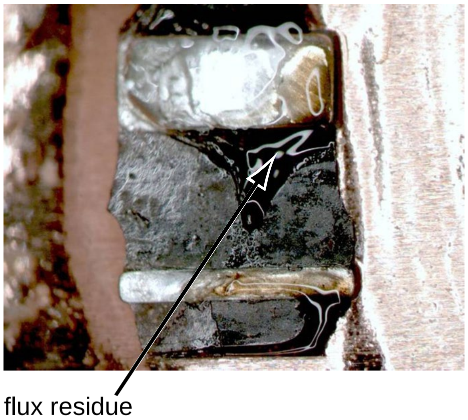


Figure 10 Flux residue compromising isolation

Inspection

For comprehensive information on inspecting board-mounted PQFN devices, refer to the SupIRBuck™ PQFN Inspection Application Note (AN-1133).

As with all QFN packaging, the best way to inspect devices after reflow is through a combination of visual inspection of the peripheral solder joints and X-ray imaging of the connections directly under the package.

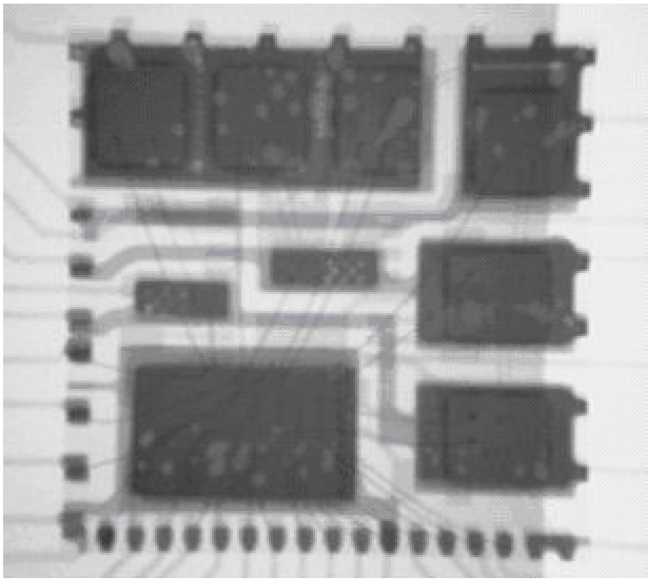


Figure 11 X-ray of a µIPM PQFN device

Figure 11 is a typical X-ray image of a board-mounted PQFN device, which shows the solder joints, device alignment and solder voiding level. Regarding solder joint voiding, most customers use 25–30% as the acceptable limit, often citing industry standards such as IPC-A-610 or IPC-7093. However, having tested board-mounted devices deliberately voided up to 45%, International Rectifier has been unable to detect any deterioration in electrical or thermal performance in application compared with devices voided to 5–10%.

Rework guidelines

Modern rework stations for ball grid array and leadless packages often use two heating stages:

- The first stage heats the substrate, either with a conventional hot-plate or a hot-air system. This reduces the amount of heating required from the hot-air de-soldering tool, which in turn reduces the risk of damaging either the substrate or surrounding components.
- The second stage uses a hot-air system for localized heating, often with the option of unheated air for faster cooling of the solder interconnections on the replaced device; this improves the solder grain structure.

The device placement mechanism or arm usually has a hot-air de-soldering gun as part of the pick head, equipped with a vacuum cup and thermocouple. Once the solder reflow temperature has been reached, the vacuum is automatically engaged to allow the device to be removed from the substrate. This reduces the risk of causing damage by premature removal.

Most rework stations have the facility to attach a micro-stencil supplied by the vendor, with the aperture design being supplied by the user. The apertures are aligned with the pads on the board before manually screening the solder paste. Alternatively, it is possible to use a standalone micro-stencil and squeegee to apply the paste.

The objective of rework is to remove a non-functional device and replace with a functional device. International Rectifier does not recommend reusing devices removed from a substrate. To permit subsequent failure analysis, take care when removing devices not to not exacerbate the existing failure.

For a demonstration of how to rework µIPM PQFN devices using basic soldering tools in development or test environments, visit www.irf.com/product-info/imotion/uipm-rework.html.

To replace a µIPM PQFN device:

Note: If you usually bake to remove residual moisture before rework, insert your normal procedure here.

1. Heat the site to approximately 100°C (150°C for lead-free assembly) using the substrate heating stage.

Note: Pb devices are qualified for a maximum reflow peak temperature of 240°C (260°C for PbF devices). To avoid overheating the device or substrate, adjust the settings on your equipment to achieve a maximum air temperature of 300°C.

2. Lower the placement arm to bring the de-soldering tool into contact with the device. When the device and the solder interconnects reach reflow temperature, lift the placement arm to remove the device from the substrate. Discard the device.
3. Clear residual solder from the site using a blade-type de-soldering tool and de-soldering braid. Clear residual flux using a flux-reducing agent. Take care in cleaning the site: damage to the solder mask may produce undesirable results.
4. When the site is ready, apply new solder paste with a micro-stencil and squeegee.
5. Position a new device on the vacuum tip of the placement head and lower the placement arm until the device is in contact with the solder paste.
6. Switch off the vacuum on the placement head and retract the placement arm, leaving the device in place.
7. Heat the site to approximately 100°C (150°C for lead-free assembly) using the substrate heating stage.
8. Use the de-soldering tool to heat both device and solder interconnects to reflow temperature, waiting until all the solder has reflowed.
9. Retract the arm, leaving the device in place. Cool as quickly as possible.

Sample process

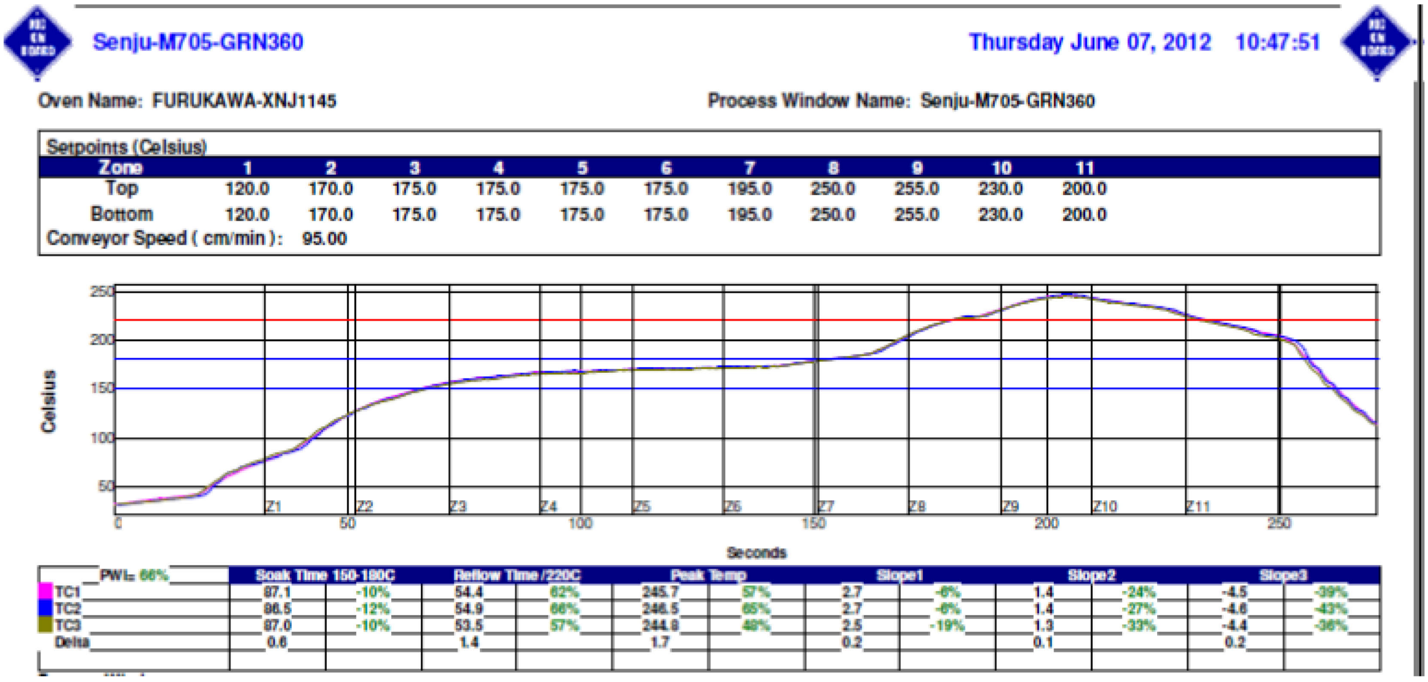
µIPM PQFN devices are suitable for a wide range of assembly conditions and are not considered to be process-sensitive. However, International Rectifier has used the following conditions successfully and offers them as an example of a suitable process. They do not constitute a recommendation: other processes may be more appropriate in different environments.

PCB

Material	High Tg FR4
Copper thickness	2 oz on outer layers
Pad surface finish	Electroless Nickel Immersion Gold (ENIG)

Solder paste

Stencil thickness	0.0127 mm / 5 mil / 0.005 inch
Solder paste	Senju M705-S101ZH-S4 Sn96.5 / Ag3.0 / Cu0.5



Senju-M705-GRN360 Thursday June 07, 2012 10:47:51

Oven Name: FURUKAWA-XNJ1145 Process Window Name: Senju-M705-GRN360

Setpoints (Celsius)

Zone	1	2	3	4	5	6	7	8	9	10	11
Top	120.0	170.0	175.0	175.0	175.0	175.0	195.0	250.0	255.0	230.0	200.0
Bottom	120.0	170.0	175.0	175.0	175.0	175.0	195.0	250.0	255.0	230.0	200.0

Conveyor Speed (cm/min): 95.00

PWI= 66%	Soak Time 150-180C		Reflow Time /220C		Peak Temp		Slope1		Slope2		Slope3	
TC1	87.1	-10%	54.4	62%	245.7	57%	2.7	-6%	1.4	-24%	-4.5	-39%
TC2	86.5	-12%	54.9	66%	246.5	65%	2.7	-6%	1.4	-27%	-4.6	-43%
TC3	87.0	-10%	53.5	57%	244.8	48%	2.5	-19%	1.3	-33%	-4.4	-36%
Delta	0.6		1.4		1.7		0.2		0.1		0.2	

Mechanical test results

International Rectifier has subjected board-mounted µIPM PQFN devices to extensive mechanical tests, conducted in accordance with industry standards and practices.

This section contains summarized results for shock tests, bend tests, vibration tests, cyclic temperature tests and warpage tests. Full reports are available on request.

Shock tests

Method

These tests were carried out in accordance with JEDEC JESD22-B111:

- Shock:1500G
- Duration: 0.5ms
- Quantity: 30 drops

$$A(t) = A_o \sin\left(\frac{\pi t}{t_w}\right)$$

$$\sqrt{2gH} = \frac{2A_o}{C}\frac{t_w}{\pi}$$

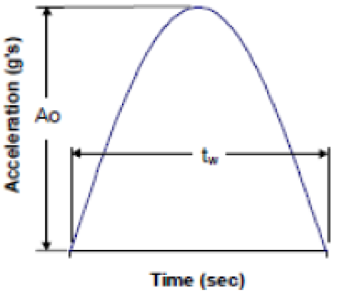


Figure 12 Typical shock test half-sine pulse

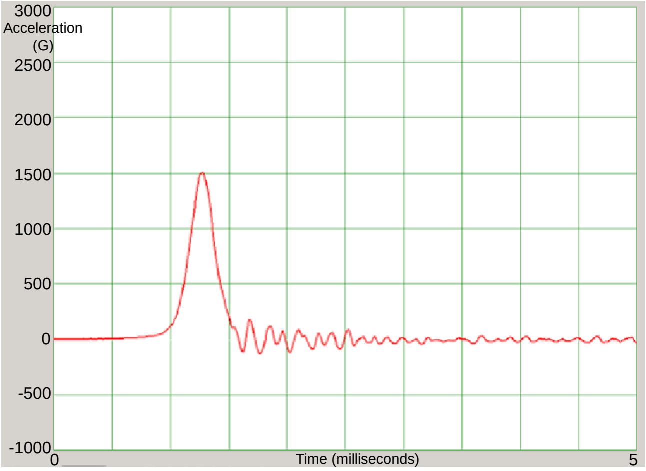


Figure 13 Actual shock pulse applied

Results

All devices passed.

Cyclic bend tests

Method

These tests were carried out in accordance with JEDEC JESD22-B113:

- 200,000 cycles
- 3Hz test rate
- 1mm deflection

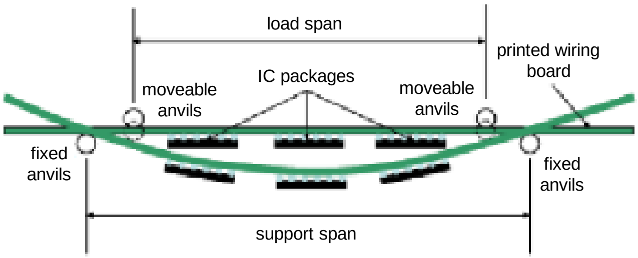


Figure 14 Four-point bend test set-up

Results

All devices passed.

Vibration tests

Method

These tests were carried out in accordance with MIL-STD-810 Vibration Testing (Method 514, Procedure 1, Category 24).

Devices were subjected to the vibration profile shown in Figure 15.

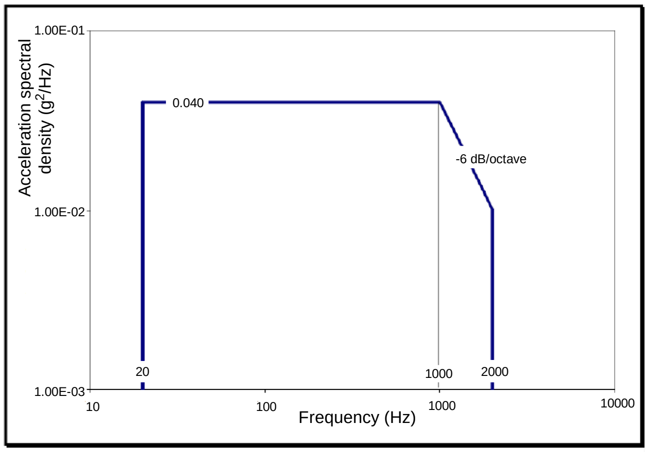


Figure 15 Vibration profile

Results

All devices passed.

Results

All devices passed.

Temperature cycle tests

Method

International Rectifier subjected board-mounted devices (often referred to as ‘second level testing’ or L2 testing) to temperature cycling from -40°C to +85°C.

Results

All devices passed 1000 cycles.

Figure 16 shows the Weibull plot of the characteristic life of devices mounted on boards using both 0.100mm (0.004") and 0.127mm (0.005") solder paste stencils.

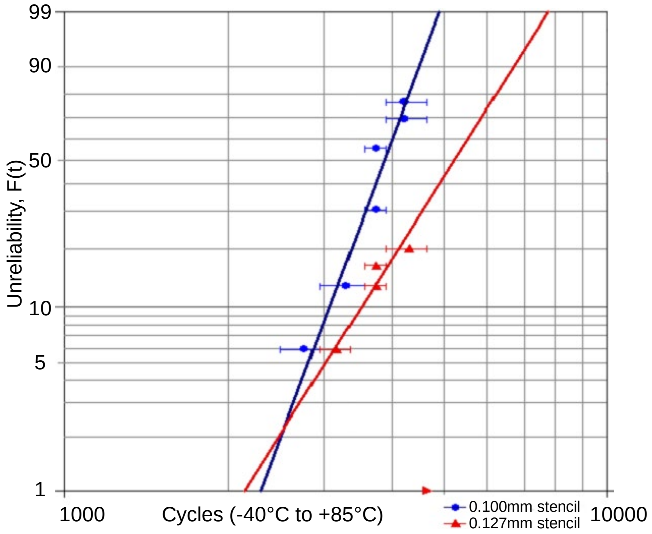


Figure 16 Temperature cycling test results

The devices were then left on test to determine their characteristic life. The results were:

- Stencil thickness of 0.127mm (0.005”) = 5640 cycles
- Stencil thickness of 0.100mm (0.004”) = 4053 cycles

Warpage tests

Method

These tests were carried out in accordance with JESD22-B112A.

The failure threshold was derived from IPC/JEDEC J-STD-020D, which states that warpage must not exceed JEDEC’s maximum dimensions for the coplanarity and stand-off of PQFN devices. Therefore, for 12x12 PQFN devices with a maximum coplanarity of 0.08mm and maximum stand-off of 0.05mm, the limit for warpage is 0.08mm (0.003").

Devices were measured for warpage after three reflow passes with a peak temperature of 260°C using the following procedure:

- Packages placed with leads uppermost (known as the ‘dead bug’ position)
- Samples of five units per assembly lot selected
- Five points measured (see Figure 17), using a Kunoh BK-02 non-contact measurement microscope.

Warpage was calculated for each sample device by subtracting the measurement at the lowest point from the measurement at the highest point.

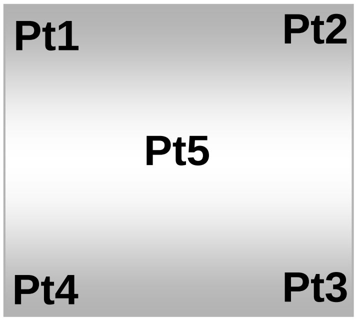


Figure 17 Five measurement points on device

Results

All devices passed.

Appendix A Model-specific data

This appendix contains the following information about various µIPM PQFN devices:

- Device outline drawing
- Recommended substrate/PCB layout
- Stencil designs for thicknesses of 0.127mm (0.005") and 0.150mm (0.006")

The footprint and stencil designs are recommendations only, and may need to be adjusted to specific requirements. During a study conducted on various package types, International Rectifier found the designs gave repeatable device alignment and proper solder connections.

For more details about individual devices, and to find out their size and outline, refer to the relevant product data sheet.

Interchangeability

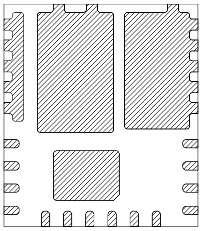
These devices are not interchangeable.

Acknowledgements

International Rectifier would like to thank DfR Solutions for providing the studies needed to develop the substrate/PCB layouts and stencil designs.

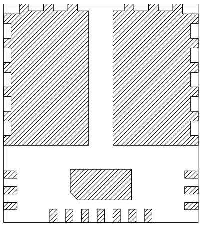
7x8mm outlines

PQFN



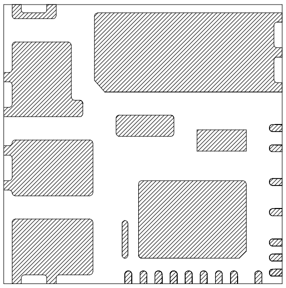
8x9mm outlines

PQFN

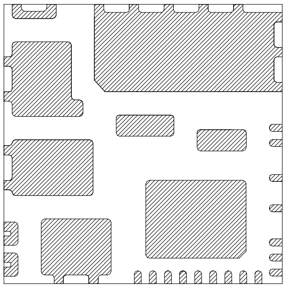


12x12mm outlines

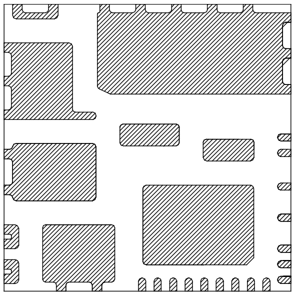
27L



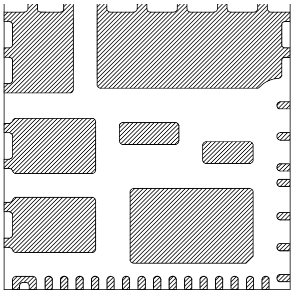
36L



37L



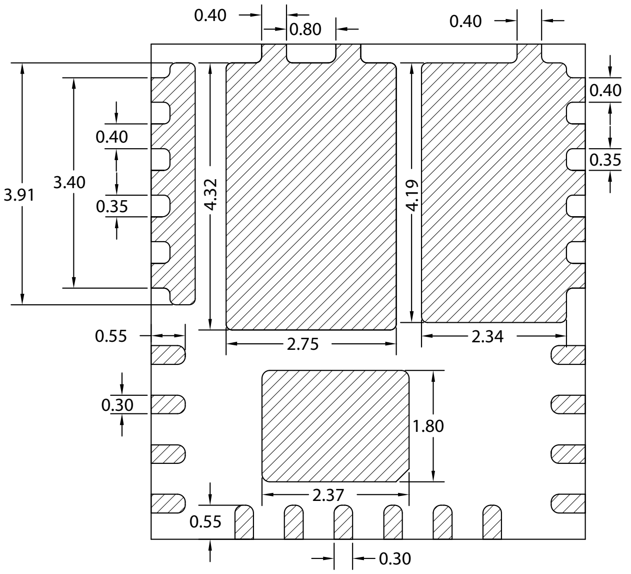
40L



Appendix A.1 7x8

Device outline

Figure A.1.1 shows the outline for these devices. The relative pad positions are controlled to an accuracy of ±0.050mm. For full dimensions and tolerances of each device, and to find out its size and outline, refer to the relevant product data sheet and package outline drawing.

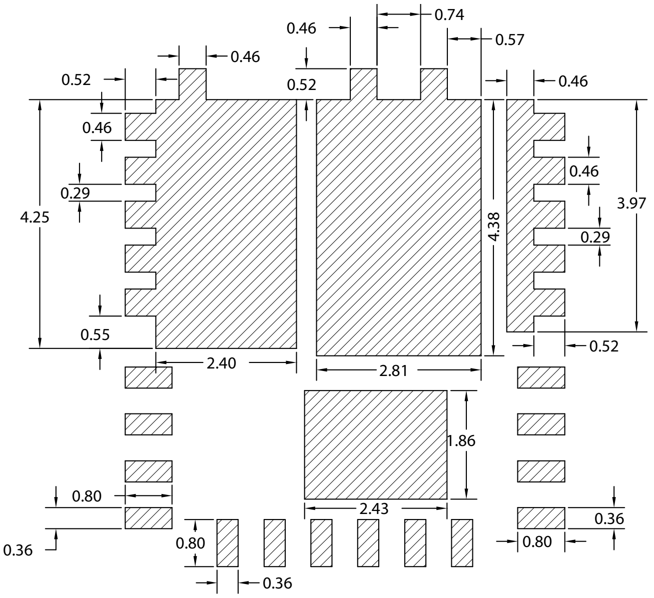


(dimensions in mm)

Figure A.1.1 7x8 PQFN device outline

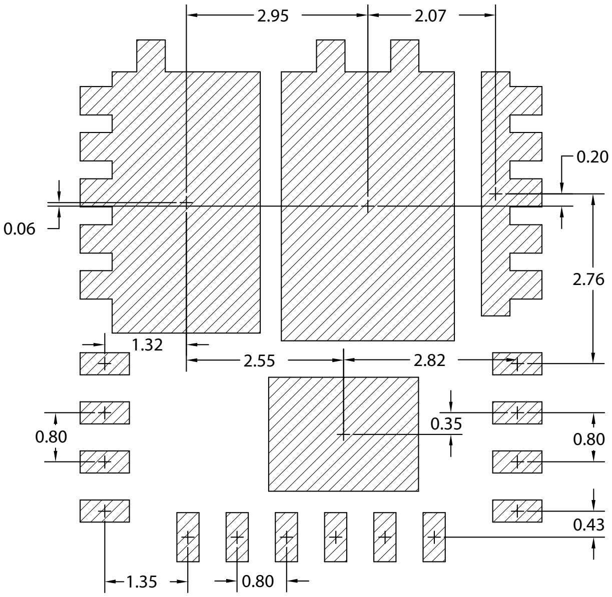
Substrate/PCB layout

Evaluations have shown that the best overall performance is achieved using the substrate/PCB layout shown in Figure A.1.2 (a and b).



(dimensions in mm)

Figure A.1.2(a) 7x8 PQFN substrate/PCB layout

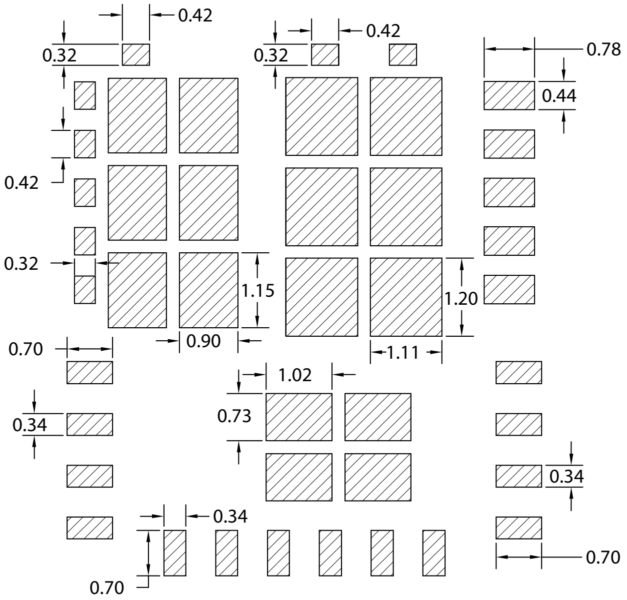


(dimensions in mm)

Figure A.1.2(b) 7x8 PQFN substrate/PCB layout

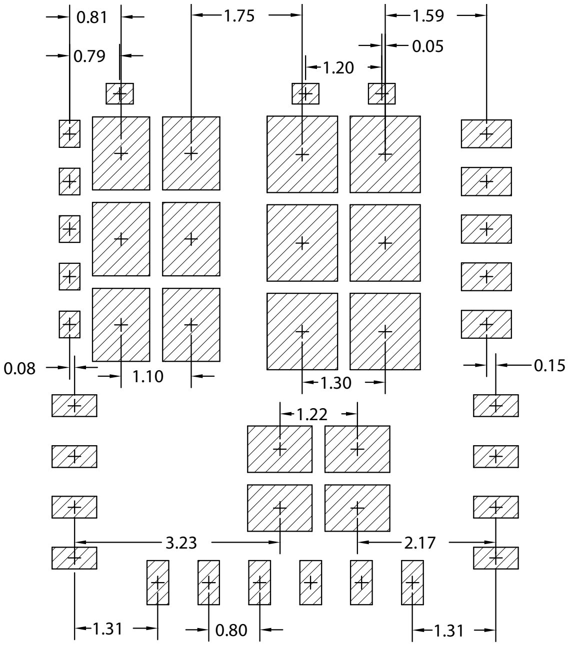
Stencil design

Evaluations have shown that the best overall performance is achieved using the stencil design shown in Figure A.1.3 (a, b and c). This design is for a stencil thickness of 0.127mm (0.005").



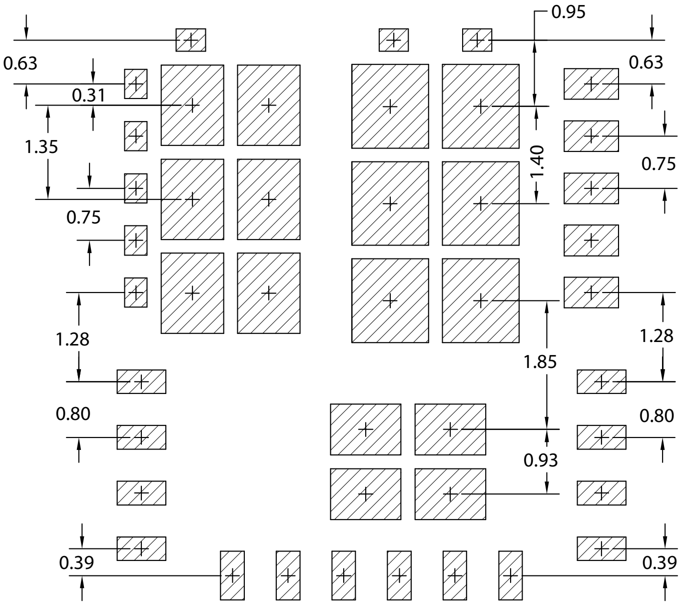
(dimensions in mm)

Figure A.1.3(a) 7x8 PQFN 0.127mm stencil design



(dimensions in mm)

Figure A.1.3(b) 7x8 PQFN 0.127mm stencil design



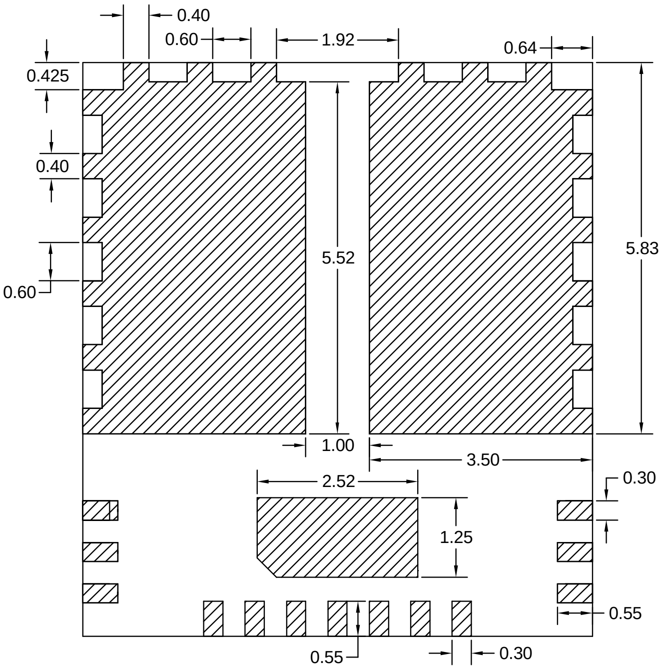
(dimensions in mm)

Figure A.1.3(c) 7x8 PQFN 0.127mm stencil design

Appendix A.2 8x9

Device outline

Figure A.2.1 shows the outline for these devices. The relative pad positions are controlled to an accuracy of ±0.050mm. For full dimensions and tolerances of each device, and to find out its size and outline, refer to the relevant product data sheet and package outline drawing.

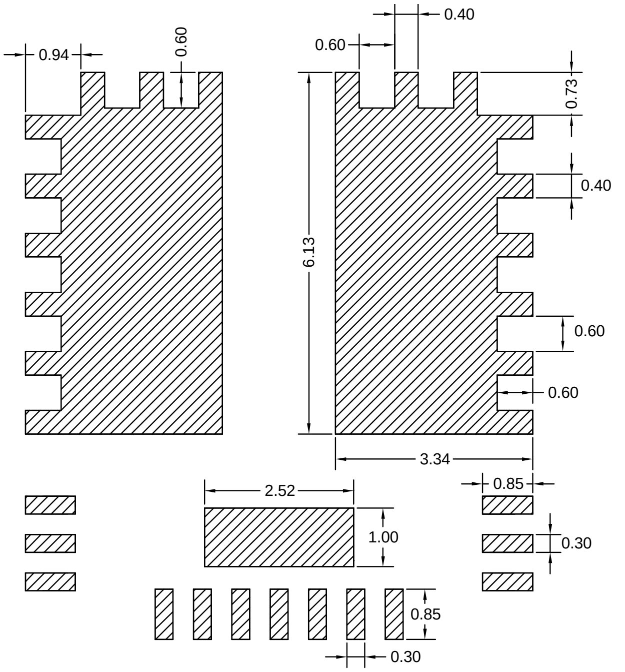


(dimensions in mm)

Figure A.2.1 8x9 PQFN device outline

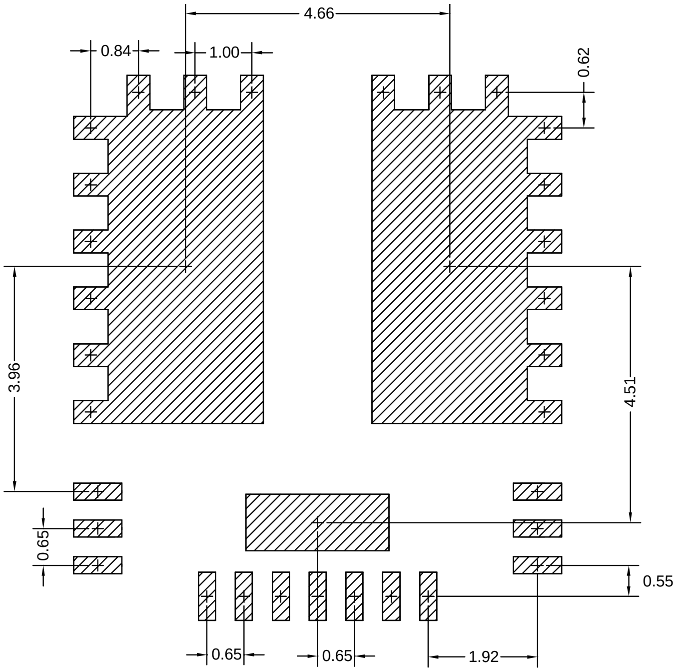
Substrate/PCB layout

Evaluations have shown that the best overall performance is achieved using the substrate/PCB layout shown in Figure A.2.2 (a and b).



(dimensions in mm)

Figure A.2.2(a) 8x9 PQFN substrate/PCB layout

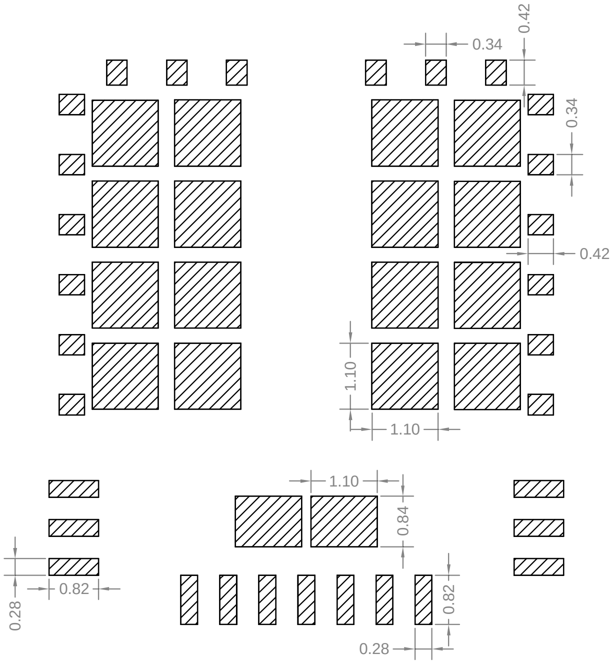


(dimensions in mm)

Figure A.2.2(b) 8x9 PQFN substrate/PCB layout

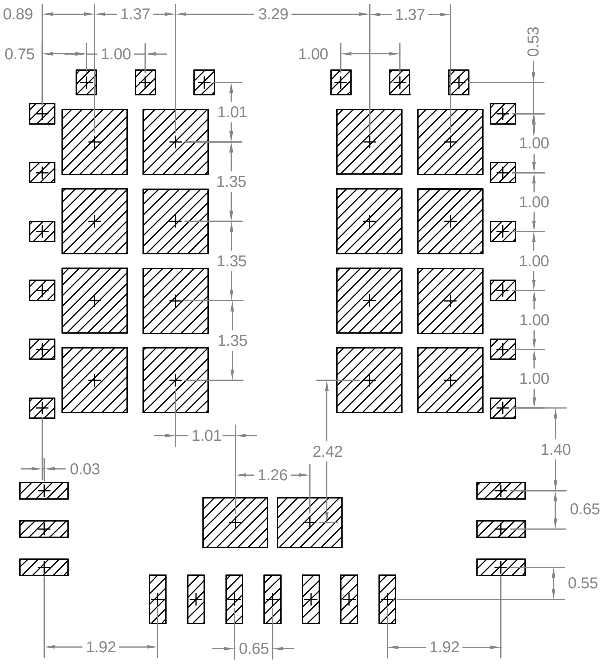
Stencil design

Evaluations have shown that the best overall performance is achieved using the stencil design shown in Figure A.2.3 (a and b). This design is for a stencil thickness of 0.127mm (0.005").



(dimensions in mm)

Figure A.2.3(a) 8x9 PQFN 0.127mm stencil design



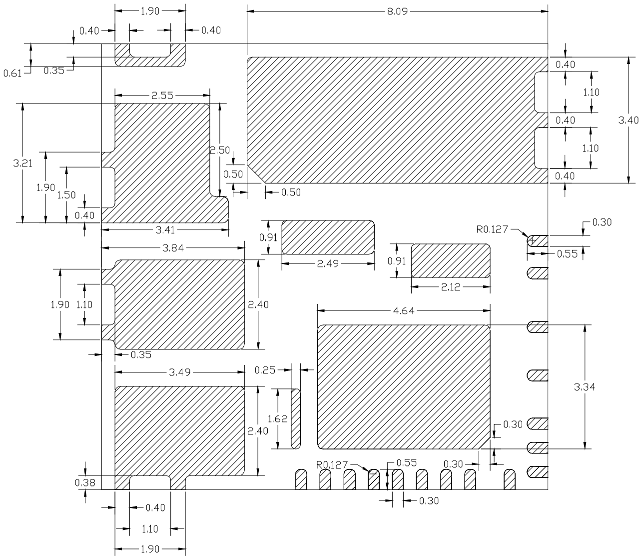
(dimensions in mm)

Figure A.2.3(b) 8x9 PQFN 0.127mm stencil design

Appendix A.3 12x12 (27L)

Device outline

Figure A.3.1(a) shows the outline for these devices. The relative pad positions are controlled to an accuracy of ±0.050mm. For full dimensions and tolerances of each device, and to find out its size and outline, refer to the relevant product data sheet and package outline drawing.



(dimensions in mm)

Figure A.3.1(a) 12x12 (27L) device outline

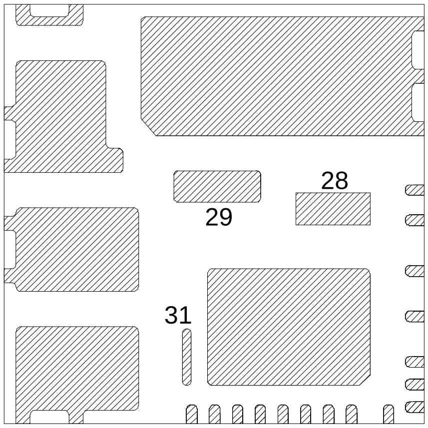
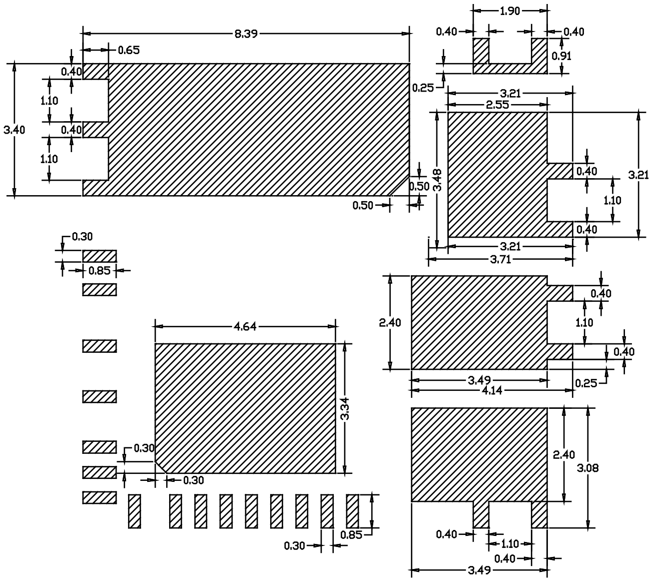


Figure A.3.1(b) 12x12 (27L) – optional pads

Pads 28, 29 and 31 on the underside of the device (Figure A.3.1(b)) are a result of the internal package construction and are not required for electrical or thermal connection. While soldering them to the PCB increases mechanical stability, it also traps more flux under the device and restricts the release of gas from the flux during reflow. This can increase the risk of board -level leakage in operation. Consequently, International Rectifier recommends that these pads are not soldered and excludes them from its recommended PCB layout.

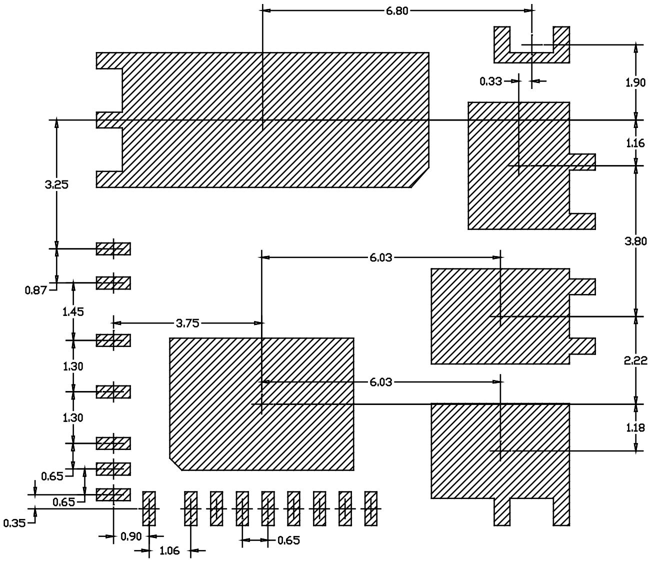
Substrate/PCB layout

Evaluations have shown that the best overall performance is achieved using the substrate/PCB layout shown in Figure A.3.2 (a and b).



(dimensions in mm)

Figure A.3.2(a) 12x12 (27L) substrate/PCB layout

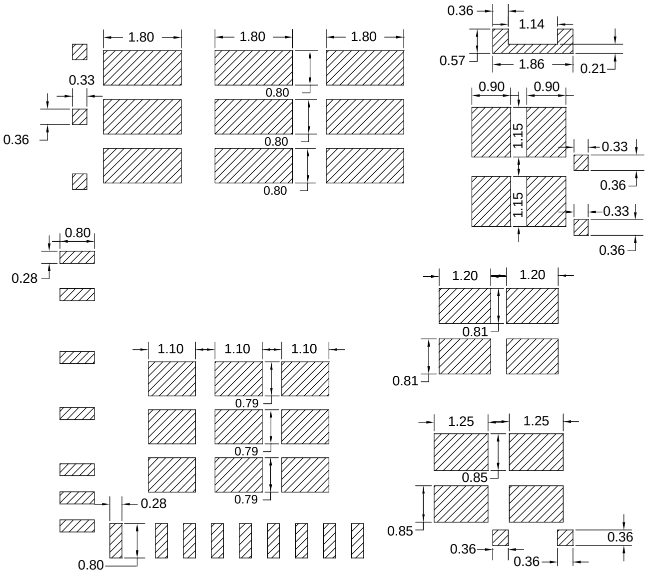


(dimensions in mm)

Figure A.3.2(b) 12x12 (27L) substrate/PCB layout

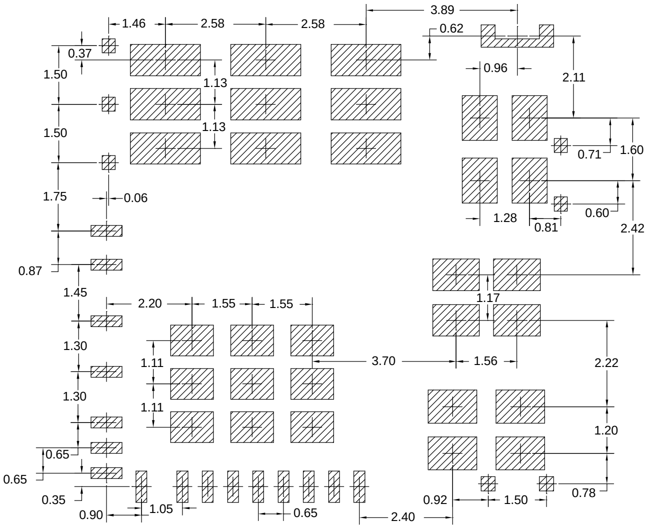
Stencil design

Evaluations have shown that the best overall performance is achieved using the stencil design shown in Figure A.3.3 (a and b). This design is for a stencil thickness of 0.127mm (0.005").



(dimensions in mm)

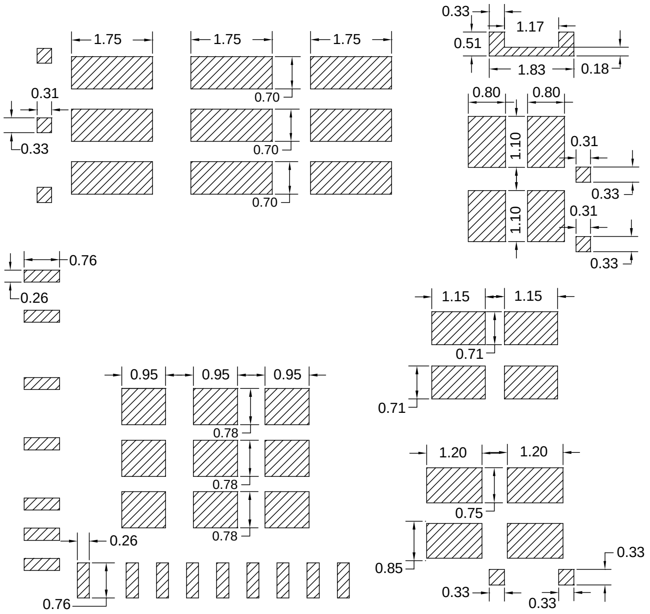
Figure A.3.3(a) 12x12 (27L) 0.127mm stencil design



(dimensions in mm)

Figure A.3.3(b) 12x12 (27L) 0.127mm stencil design

If you want to use a thicker stencil, 0.150mm (0.006"), use the pad sizes shown in Figure A.3.4 with the pad spacings shown in Figure A.3.3(b).



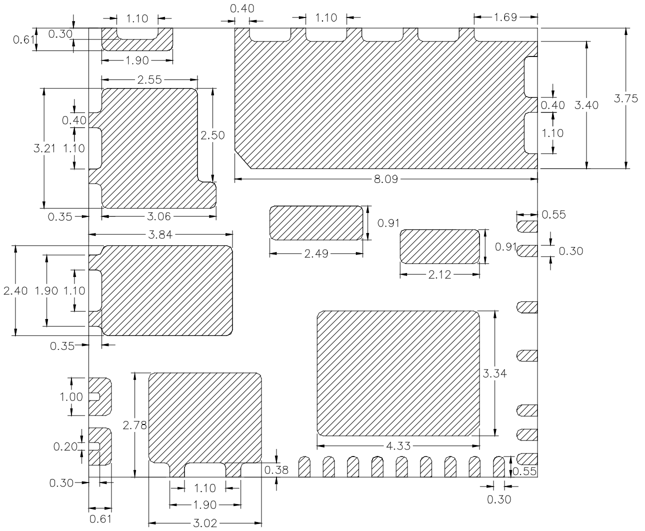
(dimensions in mm)

Figure A.2.4 12x12 (27L) 0.150mm stencil design

Appendix A.4 12x12 (36L)

Device outline

Figure A.4.1 shows the outline for these devices. The relative pad positions are controlled to an accuracy of ±0.050mm. For full dimensions and tolerances of each device, and to find out its size and outline, refer to the relevant product data sheet and package outline drawing.

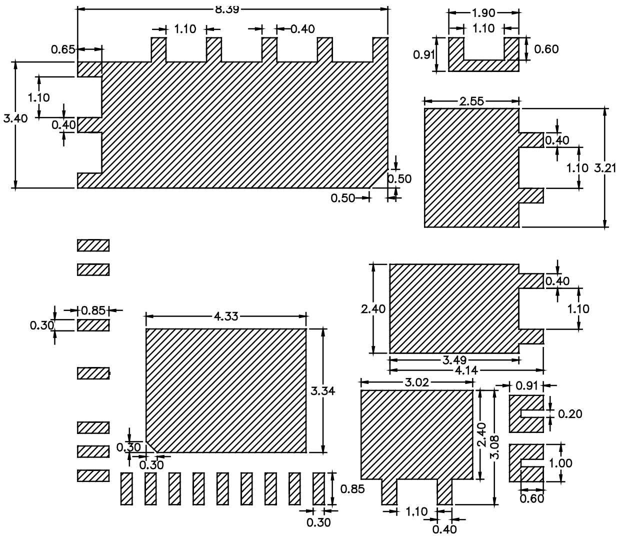


(dimensions in mm)

Figure A.4.1 12x12 (36L) device outline

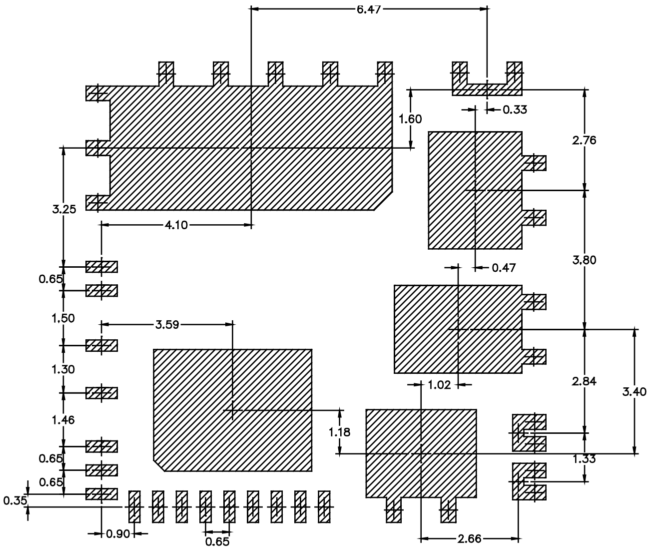
Substrate/PCB layout

Evaluations have shown that the best overall performance is achieved using the substrate/PCB layout shown in Figure A.4.2 (a, b and c).



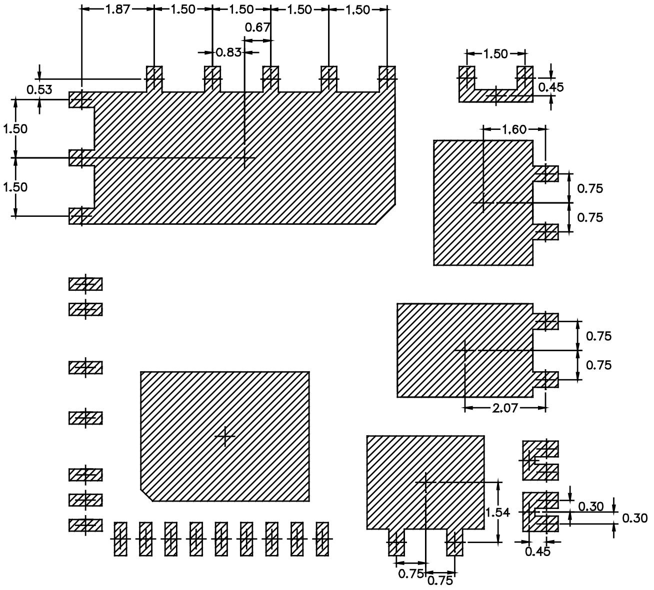
(dimensions in mm)

Figure A.4.2(a) 12x12 (36L) substrate/PCB layout



(dimensions in mm)

Figure A.4.2(b) 12x12 (36L) substrate/PCB layout

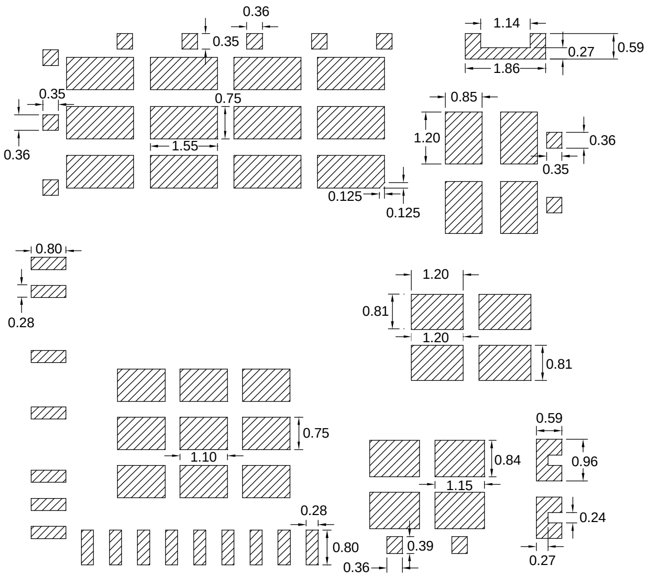


(dimensions in mm)

Figure A.4.2(c) 12x12 (36L) substrate/PCB layout

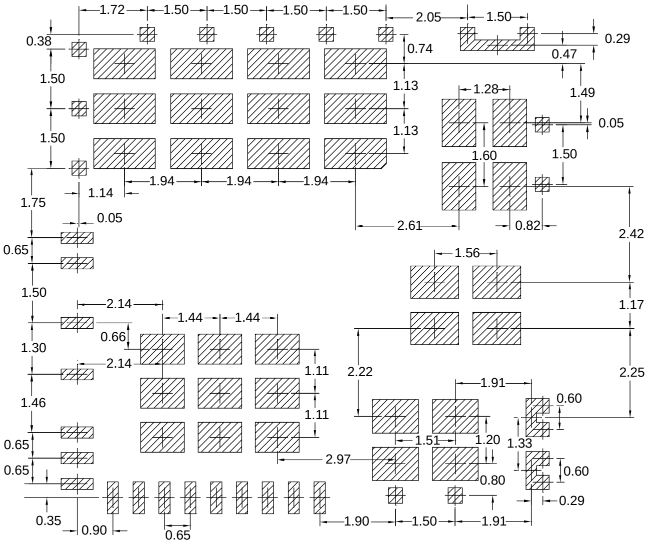
Stencil design

Evaluations have shown that the best overall performance is achieved using the stencil design shown in Figure A.4.3 (a and b). This design is for a stencil thickness of 0.127mm (0.005").



(dimensions in mm)

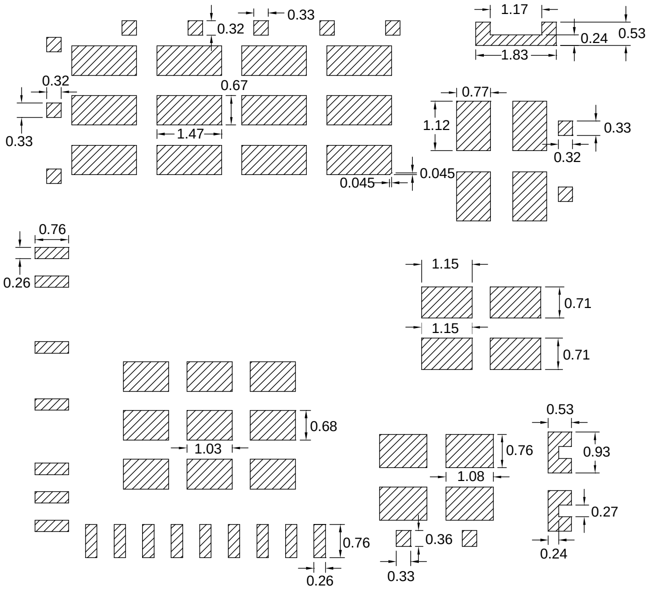
Figure A.4.3(a) 12x12 (36L) 0.127mm stencil design



(dimensions in mm)

Figure A.4.3(b) 12x12 (36L) 0.127mm stencil design

If you want to use a thicker stencil, 0.150mm (0.006"), use the pad sizes shown in Figure A.3.4 with the pad spacings shown in Figure A.3.3(b).



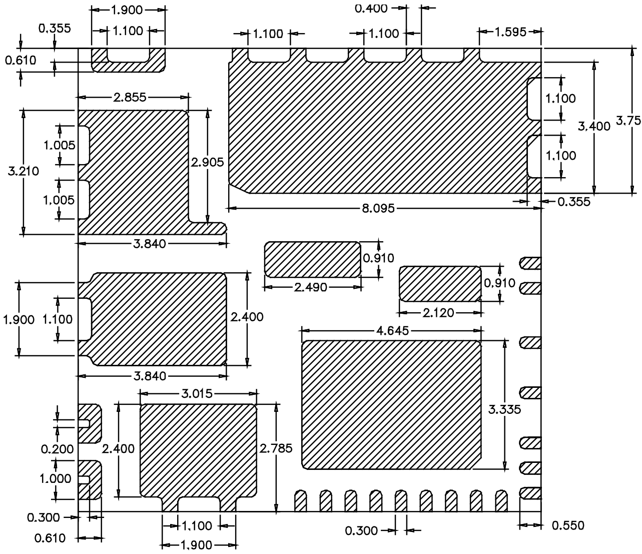
(dimensions in mm)

Figure A.3.4 12x12 (36L) 0.150mm stencil design

Appendix A.5 12x12 (37L)

Device outline

Figure A.5.1 shows the outline for these devices. The relative pad positions are controlled to an accuracy of ±0.050mm. For full dimensions and tolerances of each device, and to find out its size and outline, refer to the relevant product data sheet and package outline drawing.

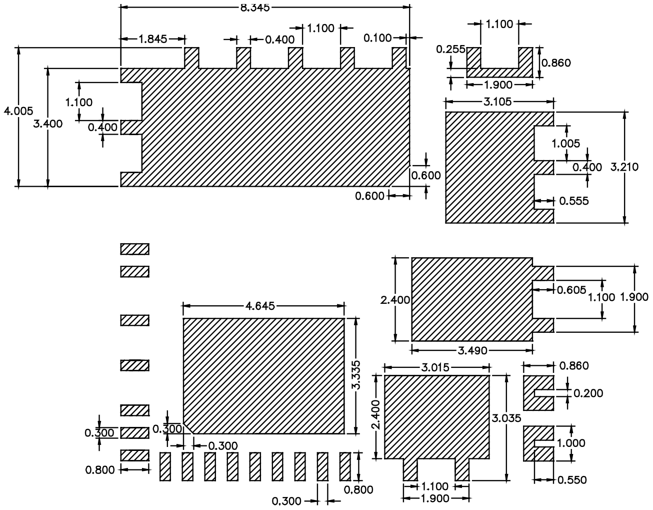


(dimensions in mm)

Figure A.5.1 12x12 (37L) device outline

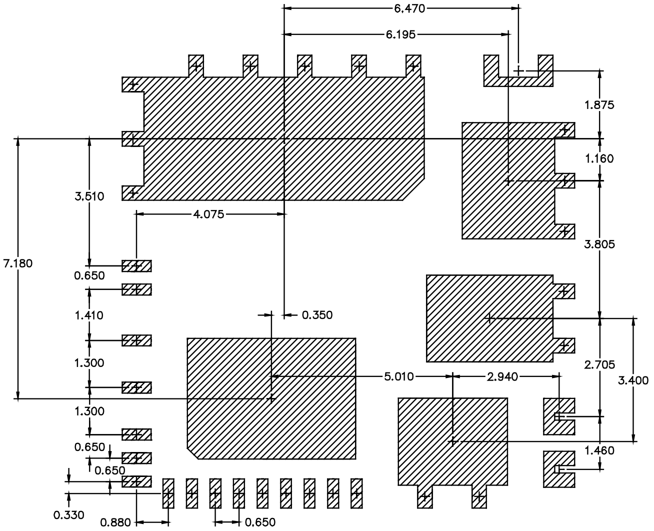
Substrate/PCB layout

Evaluations have shown that the best overall performance is achieved using the substrate/PCB layout shown in Figure A.5.2 (a, b and c).



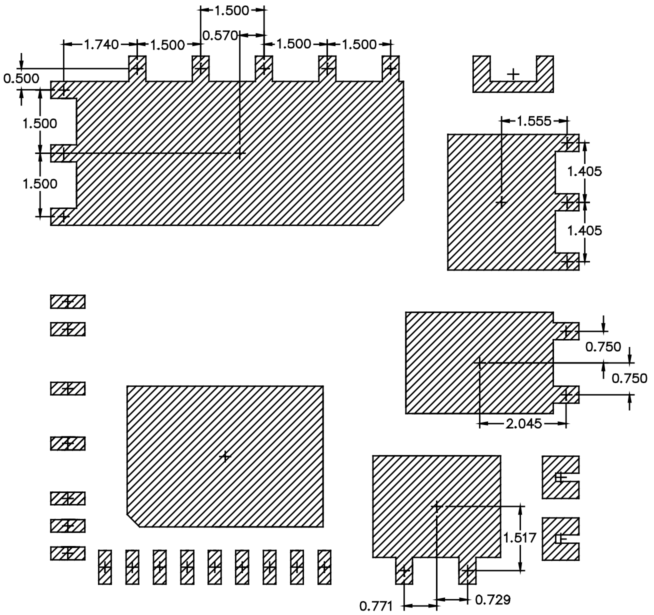
(dimensions in mm)

Figure A.5.2(a) 12x12 (37L) substrate/PCB layout



(dimensions in mm)

Figure A.5.2(b) 12x12 (37L) substrate/PCB layout

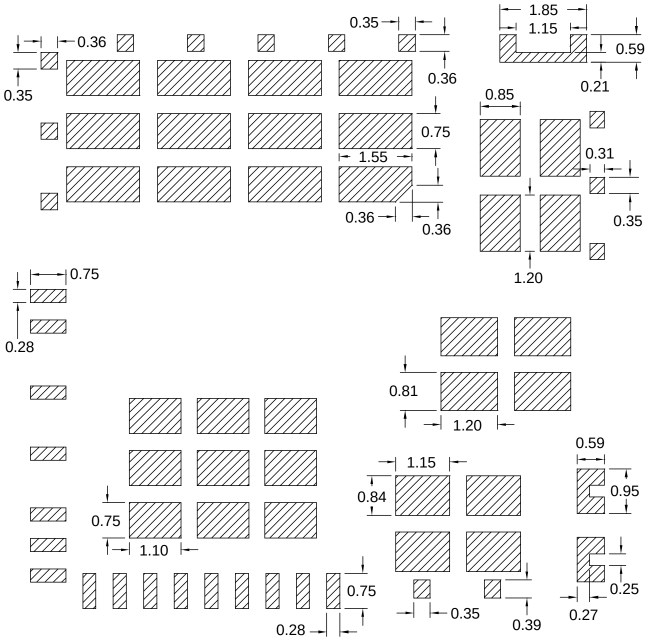


(dimensions in mm)

Figure A.5.2(c) 12x12 (37L) substrate/PCB layout

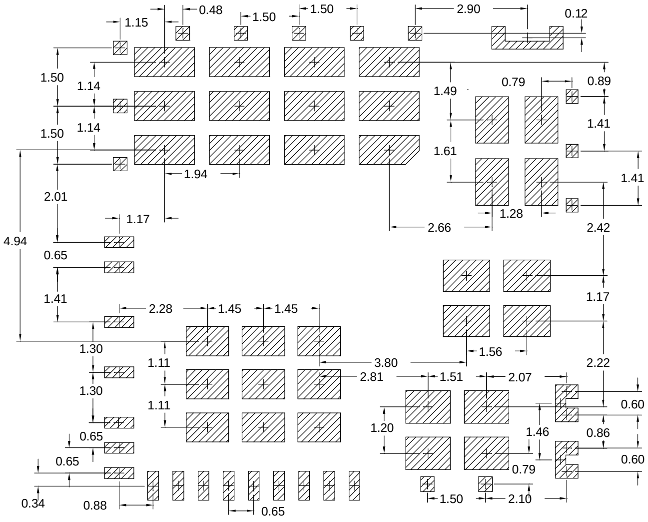
Stencil design

Evaluations have shown that the best overall performance is achieved using the stencil design shown in Figure A.5.3 (a and b). This design is for a stencil thickness of 0.127mm (0.005").



(dimensions in mm)

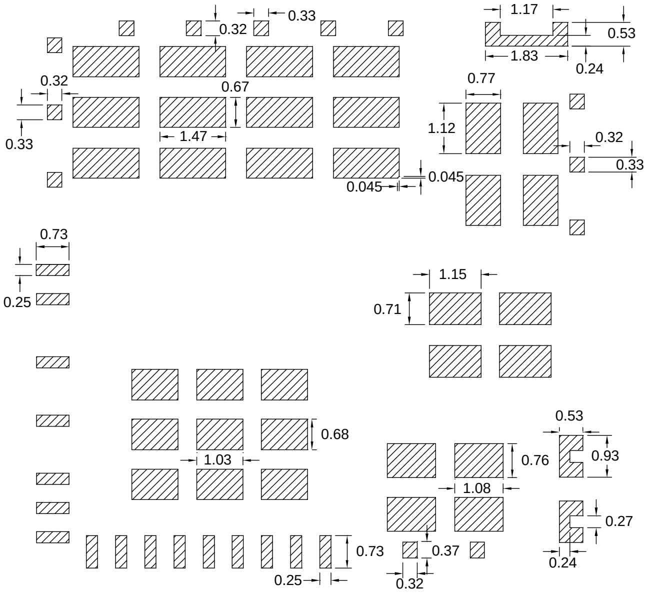
Figure A.5.3(a) 12x12 (37L) 0.127mm stencil design



(dimensions in mm)

Figure A.5.3(b) 12x12 (37L) 0.127mm stencil design

If you want to use a thicker stencil, 0.150mm (0.006"), use the pad sizes shown in Figure A.5.4 with the pad spacings shown in Figure A.5.3(b).



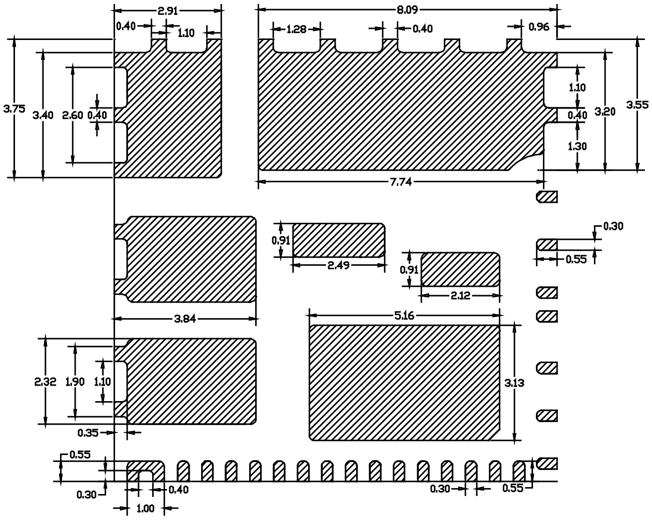
(dimensions in mm)

Figure A.5.4 12x12 (37L) 0.150mm stencil design

Appendix A.6 12x12 (40L)

Device outline

Figure A.6.1 shows the outline for these devices. The relative pad positions are controlled to an accuracy of ±0.050mm. For full dimensions and tolerances of each device, and to find out its size and outline, refer to the relevant product data sheet and package outline drawing.

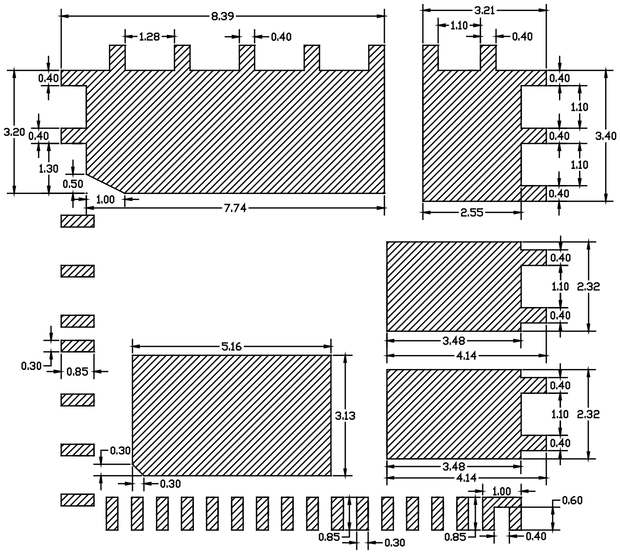


(dimensions in mm)

Figure A.6.1 12x12 (40L) device outline

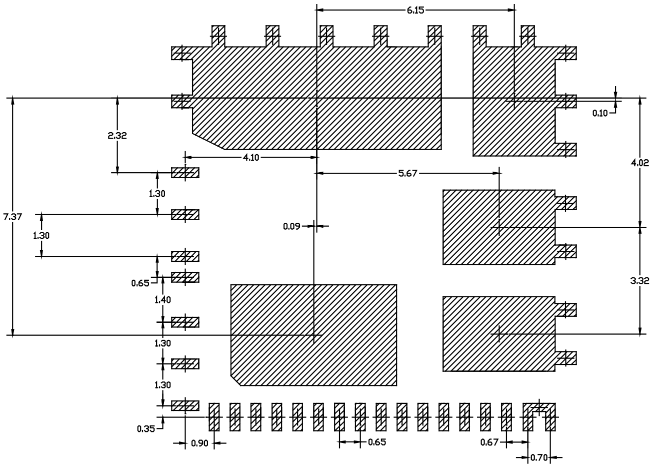
Substrate/PCB layout

Evaluations have shown that the best overall performance is achieved using the substrate/PCB layout shown in Figure A.6.2 (a, b and c).



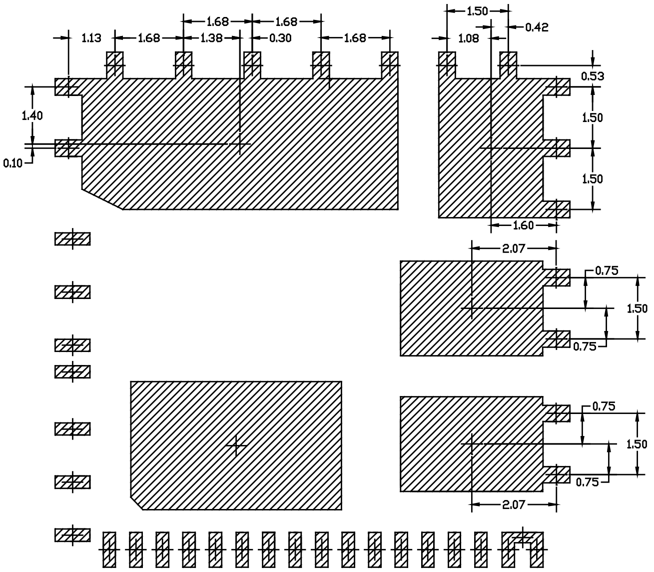
(dimensions in mm)

Figure A.6.2(a) 12x12 (40L) substrate/PCB layout



(dimensions in mm)

Figure A.6.2(b) 12x12 (40L) substrate/PCB layout

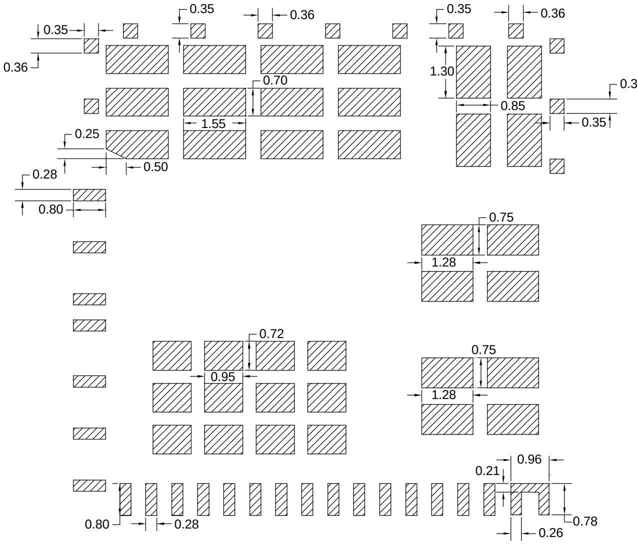


(dimensions in mm)

Figure A.6.2(c) 12x12 (40L) substrate/PCB layout

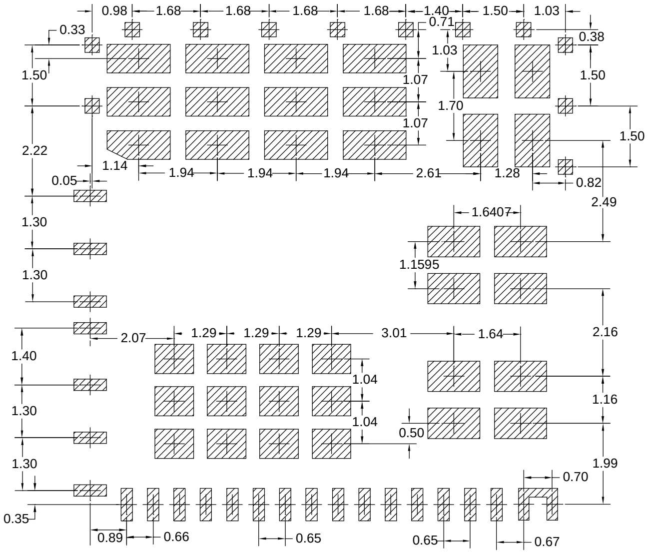
Stencil design

Evaluations have shown that the best overall performance is achieved using the stencil design shown in Figure A.6.3 (a and b). This design is for a stencil thickness of 0.127mm (0.005").



(dimensions in mm)

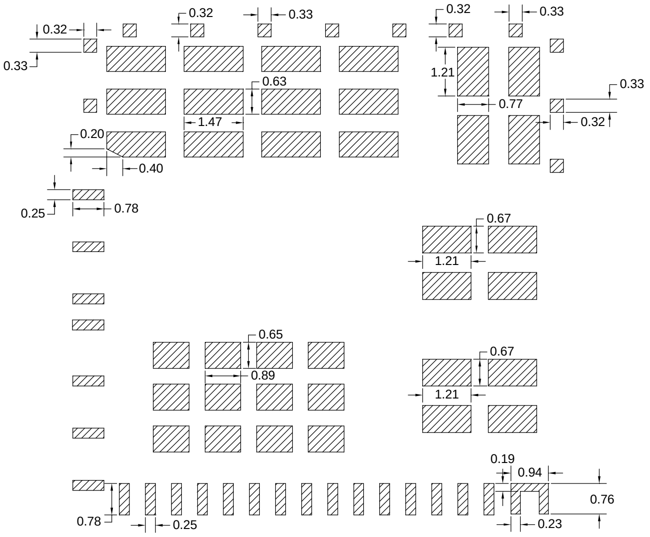
Figure A.6.3(a) 12x12 (40L) 0.127mm stencil design



(dimensions in mm)

Figure A.6.3(b) 12x12 (40L) 0.127mm stencil design

If you want to use a thicker stencil, 0.150mm (0.006"), use the pad sizes shown in Figure A.6.4 with the pad spacings shown in Figure A.6.3(b).



(dimensions in mm)

Figure A.6.4 12x12 (40L) 0.150mm stencil design