AND9867/D

Mounting Instructions for PIM Modules (Q0, Q1, Q2, F1, F2)

ON Semiconductor®

www.onsemi.com

APPLICATION NOTE

Introduction

This application note covers the mounting instructions for ON Semiconductor Power Integrated Modules (PIMs) using the following packages:

- Q0
- Q1
- Q2
- F1
- F2

This application note covers the following topics

- PCB hole sizes and plating
- PCB design
- Heatsink and Thermal Interface Material (TIM)
- Press−in process
- Soldering process
- Mounting module to the heatsink
- Mounting heatsink and module to PCB

ON Semiconductor family of Power Integrated Modules has package options using solder pins or press−fit pins for the connection of the module to a Printed Circuit Board (PCB). Figure 1 shows a Q1 module with press−fit pins as an example.

After mounting, press−fit pins provide a cold−welding connection between pins and plated through holes (PTH) of the PCB. This easy assembly method avoids extra heating, avoids contamination and provides good mechanical and electrical performance. Press−fit assembly is a well−established connection method for power semiconductor modules. The press−fit pins provide a gas−tight metal to metal contact between the press−fit pin and the plated through hole (Figure 2).

Modules having solder pins are soldered into the PCB. Modules having press−fit pins can also be soldered into the PCB, but with larger holes than used for the standard press−fit process.

The purpose of this application note is to provide recommendations for the PCB and to recommend mounting and dismounting methods with proper tools to achieve the required reliability and performance with either press−fit or solder connections.

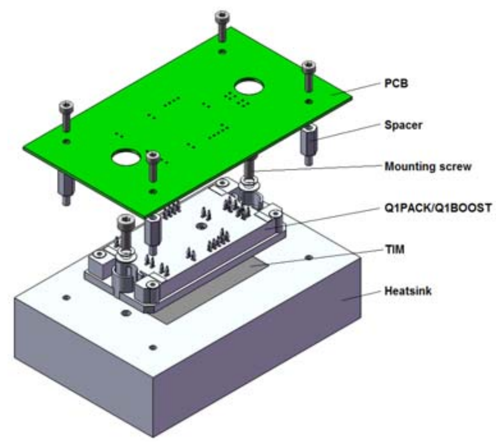


Figure 1. Q1 Module with PCB and Heatsink

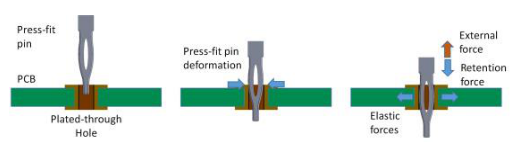


Figure 2. Press−fit Process

PCB SPECIFICATION

Correct design of the plated through holes (PTH) in the PCB is essential to obtain good quality press−fit or solder connections.

PCB Specification

The schematic structure of a plated through hole is shown in Figure 3. If the initial drilled hole diameter is too small, the press−in force into the PTH will be too high and mechanical damage on both press−fit pin and PTH will occur. If the final hole diameter is too small, damage to the pin and the hole may also occur. If the final hole diameter is too large, it may not provide a reliable connection between plated through hole and press−fit pin.

The effect of the pad size in reducing the creepage and clearance between pins should always be considered in minimum permissible spacing calculations.

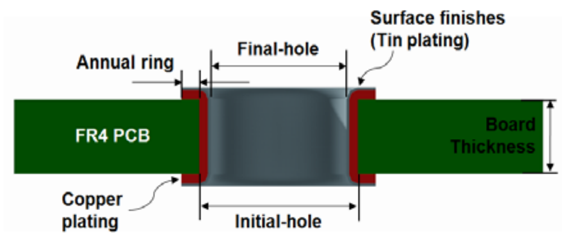


Figure 3. Cross−section of PCB

Tables 1−4 list the recommended PCB specification based on the evaluation of the press−fit technology according to the IEC 60352−5 standard.

Specification for Modules with Press−fit Pins, Soldered to the PCB

In cases where the press−fit pins are soldered instead of pressed into the PCB the recommended final PCB hole diameter is 1.85 mm when using 1.6 mm press−fit pins used in Q0, Q1 and Q2 modules; and 1.45 mm when using 1.2 mm press−fit pins used in Q0, F1 and F2 modules. Final hole diameter also depends on the manufacturing level of the PCB manufacturer as determined in different applicable IPC standards.

The pcb pad diameter is determined by the minimum annular ring size, the production alignment tolerance (level A, B or C) and the drilling tolerance as detailed in IPC−2222. The effect of the pad size in reducing the creepage and clearance between pins should always be considered in minimum permissible spacing calculations.

Specification for Modules with Solder Pins, Soldered to the PCB

The recommended final PCB hole diameter for solder pins is 1.2 mm when using 1 mm pins used in F1, F2, Q0, Q1 and Q2 modules.

The pcb pad diameter is determined by the minimum annular ring size, the production alignment tolerance (level A, B or C) and the drilling tolerance as detailed in IPC−2222. The effect of the pad size in reducing the creepage and clearance between pins should always be considered in minimum permissible spacing calculations.

For 1 mm diameter pin, the final pad diameter for level B manufacturing is calculated as 1.8 mm: allowing for 50 um absolute minimum annular ring, 0.5 mm alignment tolerance and 1.2 mm actual PCB hole size.

Table 1. PCB SPECIFICATIONS FOR F1 AND F2 MODULES WITH 1.2 MM PRESS−FIT PINS − IMMERSION OR GALVANIC TIN. Pins are IEC qualified for immersion tin plating.

	Min.	Typ.	Max.
Initial Drilled Hole Diameter Ø [mm]	1.12	1.15	
Cu Thickness in the Hole [μm]	25		50
Sn Thickness [μm] (Chemical Tin)			15
Final Hole Ø [mm]	0.98		1.09
Annular ring [μm]	200		
Thickness of Conductive Layer [μm]	35	70−105	400
Board Thickness [mm]	1.6		

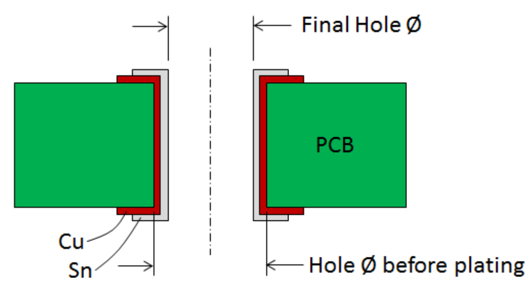


Figure 4.

Table 2. PCB SPECIFICATIONS FOR F1 AND F2 MODULES WITH 1.2 MM PRESS−FIT PINS − HAL PLATING. Pins are IEC qualified for HAL plating but immersion/galvanic tin is recommended for more consistent mounting.

	Min.	Typ.	Max.
Initial (Drill) Hole Ø [mm]	1.12	1.15	
Cu Thickness in the Hole [μm]	25		50
Sn Thickness [μm]			40
Final Hole Ø [mm]	0.94		1.09
Annular ring [μm]	200		
Thickness of Conductive Layer [μm]	35	70−105	400
Board Thickness [mm]	1.6		

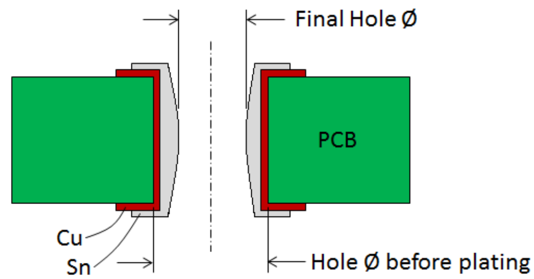


Figure 5.

Table 3. PCB SPECIFICATIONS FOR Q0, Q1 AND Q2 MODULES WITH 1.6 MM PRESS−FIT PINS − IMMERSION OR GALVANIC TIN. Pins are IEC qualified for immersion tin plating.

	Min.	Typ.	Max.
Initial Drilled Hole Diameter Ø [mm]	1.57	1.60	
Cu Thickness in the Hole [μm]	25		50
Sn Thickness [μm] (Chemical Tin)			15
Final Hole Ø [mm]	1.41		1.56
Annular ring [μm]	200		
Thickness of Conductive Layer [μm]	35	70−105	400
Board Thickness [mm]	1.6		

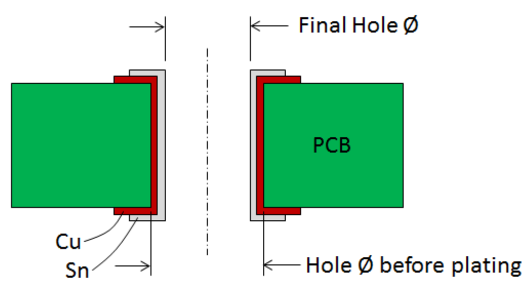


Figure 6.

Table 4. PCB SPECIFICATIONS FOR Q0, Q1 AND Q2 MODULES WITH 1.6 MM PRESS−FIT PINS − HAL PLATING. Pins are IEC qualified for HAL plating but immersion/galvanic tin is recommended for more consistent mounting.

	Min.	Typ.	Max.
Initial (Drill) Hole Ø [mm]	1.57	1.60	
Cu Thickness in the Hole [μm]	25		50
Sn Thickness [μm]			40
Final Hole Ø [mm]	1.41		1.56
Annular ring [μm]	200		
Thickness of Conductive Layer [μm]	35	70−105	400
Board Thickness [mm]	1.6		

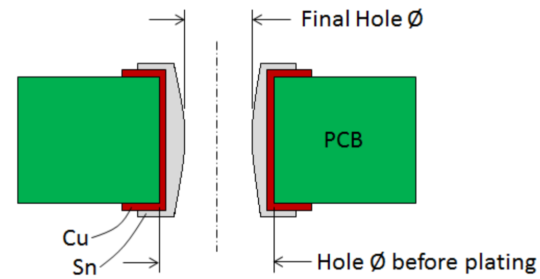


Figure 7.

Table 5. PCB SPECIFICATIONS FOR F2 MODULES WITH EON PRESS−FIT PINS − IMMERSION OR GALVANIC TIN. Pins are IEC qualified for immersion tin plating. For HAL tin coating, a thicker tin thickness of up to 40 um is permitted reducing the hole size to 0.94 mm but this requires additional approval from ON Semiconductor.

	Min.	Typ.	Max.
Initial Drilled Hole Diameter Ø [mm]	1.12	1.15	
Cu Thickness in the Hole [μm]	25		50
Sn Thickness [μm] (Chemical Tin)			15
Final Hole Ø [mm]	1.02		1.09
Annular ring [μm]	200		
Thickness of Conductive Layer [μm]	35	70−105	400
Board Thickness [mm]	1.6		

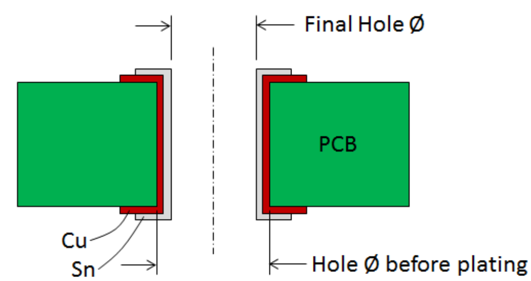


Figure 8.

PCB DESIGN

PCB Layout Restrictions

PCB bending during the press−in process causes mechanical stress to other PCB components, such as capacitors and resistors. Experiments to verify a safe minimum distance between passive components and the plated through hole were conducted with FR4 PCB. Various sizes (0603, 0805, 1206, 1210, 1812, and 2220) of mechanically sensitive components were evaluated. Based on experimental results, the recommended minimum space between center of the plated through hole and the edge of the component is 4 mm, as shown in Figure 9.

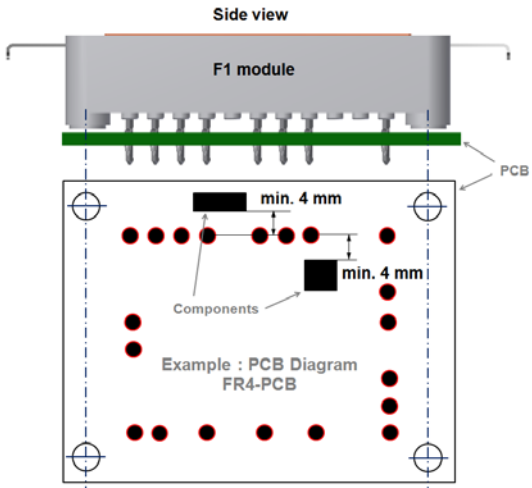


Figure 9. PCB Design Restriction: Distance from Center of PTH to Edge of Components

The minimum distance between the edge of the PCB and the centre of the pin hole must be more than 4mm.

The minimum distance between the center of the pin hole and a neighbored component on the PCB must be more than 4 mm.

Recommended PCB−thicknesses and Mounting Heights by Module Type

The distance between the top surface of the heatsink and the bottom plane of the PCB is defined by the module height of 12 mm for Q0, Q1, F1 and F2 packages or 17 mm for Q2 packages. PCB spacers can be used for fixing. The number and the position of the fixing points depend on the design of the circuit, location of different masses like capacitors or inductor and the environment of the system. General recommendations cannot be given. The recommended heights of these spacers are given in the following sections

Mounting with Distance of 12 mm

Option 1: without spacers

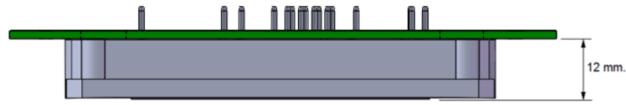


Figure 10. Mounting Height with Module as Spacer

Option 2: with spacers

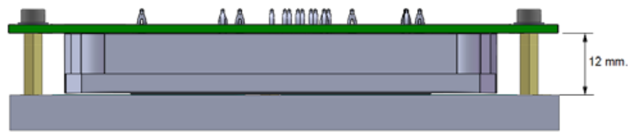


Figure 11. Mounting Height with Spacer in case of using Recommended Press−fit Tool

Example: Q1 Module with Press−fit Pins

Figure 12 shows an example of mounting a Q1 module.

The Q1 package has its power connectors/terminals distributed across the surface of the plastic case. Electrical connections are made by connecting these terminals to a print circuit board (PCB) by soldering or press−fit technology.

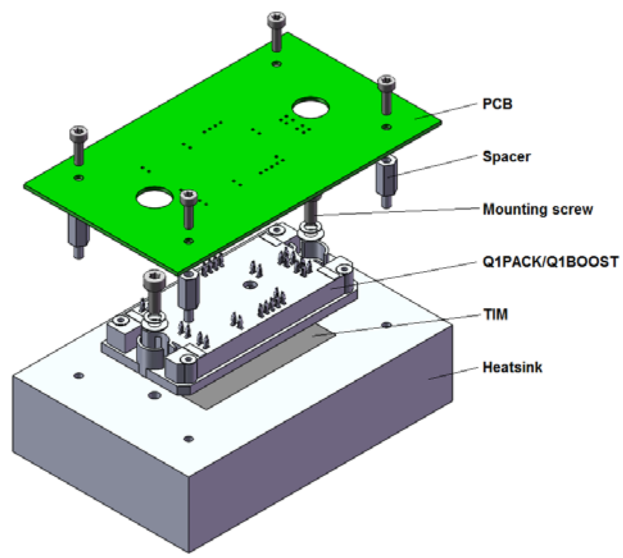


Figure 12. Q1 Package Mounting Example

The PCB has four mounting holes for the spacers, two cut−out holes for accessing the heatsink mounting screws through the PCB and the holes for the pin connections.

The dimensions and positions of the cut−out holes, the PTH holes and mounting holes are specified in the datasheet drawing for the specific module.

The Q0 solder pin modules have plastic mounting clips which require different shaped cut−outs in the PCB as specified in the datasheet.

HEATSINK AND THERMAL INTERFACE MATERIAL

Power semiconductor modules generate heat that needs to be dissipated to protect against overheating. In general, module operation temperature should not exceed the maximum allowable junction temperature (T_{Jmax}) specified in the datasheet. Thermally conductive metal heatsinks that absorb and disperse heat are commonly used for cooling high power electronics. The thermal performance of a module, in combination with a heatsink, can be characterized by the thermal resistance $R_{\theta ja}$, which is the sum of all thermal resistances in the thermal path: junction−to−case ($R_{\theta jc}$), case−to−heatsink ($R_{\theta cs}$), heatsink ($R_{\theta sink}$), and heatsink−to−ambient ($R_{\theta sa}$), as shown in Figure 13.

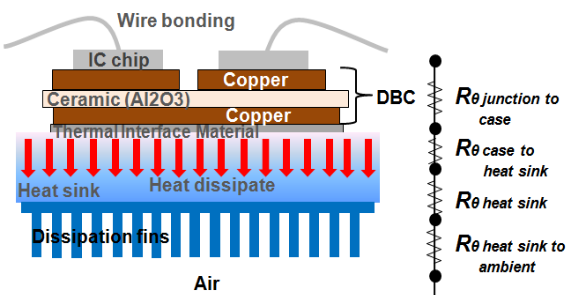


Figure 13. Thermal Model of Power Module & Heatsink

Generally, air convection is the dominant heat transfer mechanism in electronics. The heat transfer by air convection strongly depends on the air velocity and the area of the heat−transferring surface. Proper contact between module substrate and the surface of the heatsink is crucial for managing the overall thermal efficiency of the system. Thermal Interface Materials (TIMs) are thermally conductive materials used to achieve good mating of the two surfaces and improve heat transfer.

Heatsink Surface

The contact surface of a heatsink must be flat and clean to maximize heat transfer. Rough surfaces result in large voids between the substrate of the module and the surface of the heatsink. The following surface qualities are required for the heatsink to achieve a good thermal conductivity, according to DIN 4768−1. Roughness (Rz) should be 10 μm or less and flatness, based on a length of 100 mm, should be 50 μm or less. The heatsink should have no contamination, unevenness, and burrs on the surface contacting the module.

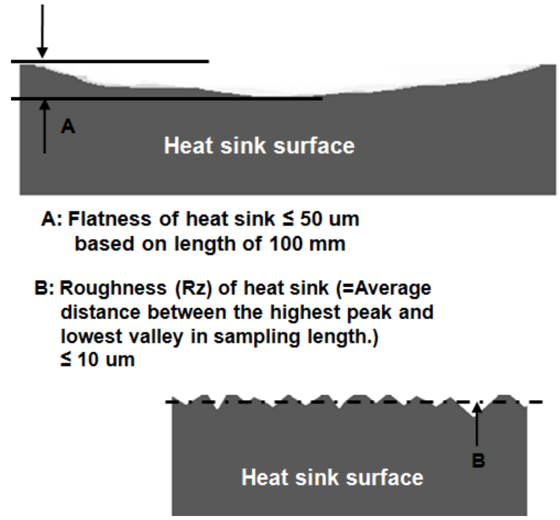


Figure 14. Microscopic View of Surfaces. A is Flatness and B is Roughness (Rz)

The interface surface of the heatsink must be free of particles and contamination. Avoid handling the heatsink surface with bare hands or contacting any foreign materials. If it is necessary to remove contamination from heatsink, cleaning can be accomplished using dry cloth soaked with solvent, such as isopropyl or ethylene alcohol.

Thermal Interface Material (TIM)

The backside of the module and the surface of the heatsink are not ideally smooth. TIM is used to prevent air cavities and help the thermal dissipation. Such TIM material may be a thermal pad, foil, grease, or any other similar material. The material selection should consider the thermal conductivity, drying out behavior during aging, and shape maintaining properties during power ON/OFF cycling.

The surfaces of the heatsink and the substrate of the module are not perfectly flat. After the module is mounted to a heatsink, air gaps can form between these two surfaces and the effective contact is limited to the area shown in Figure 15. Air is a poor heat conductor with 0.03 W/m·K thermal conductivity. It acts as a thermal barrier that limits the efficiency of heat transfer from the device.

(a)

(b)

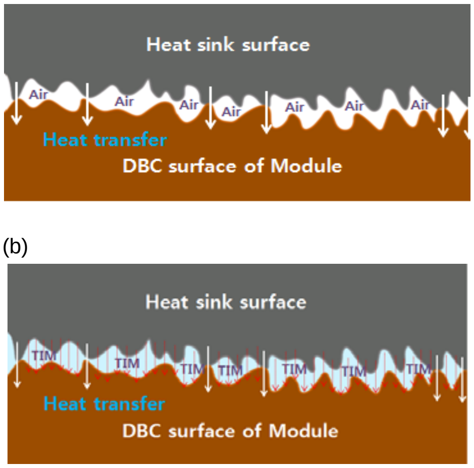


Figure 15. (a) Heat Transfer, Module to Heatsink, (b) Heat Transfer with Thermal Interface Material

Thermal interface materials are widely used in the industry to fill air gaps between contact surfaces. Thermal interface material provides better thermal performance than air and compensates for imperfect mating surfaces, such as roughness and flatness shown in Figure 14. There are various thermal interface materials available in the market. The right choice of material is an essential factor for the application. It should be selected considering the following features:

- High Thermal Conductivity
- Ease of Distribution with Low Contact Pressure
- Minimal Thickness
- Degradation of Characteristics Over Time
- Stability of Characteristics Over Time
- Toxicity (Non−Toxic Optimal)
- Ease in Handling during Application or Removal

Thermal Grease

The most common thermal interface materials are thermal greases. Thermal grease can be applied to the heatsink or the module substrate using a rubber roller or spatula or by screen printing. A rubber roller, as shown in Figure 16**,** is an easy and fast method for applying thermal grease.

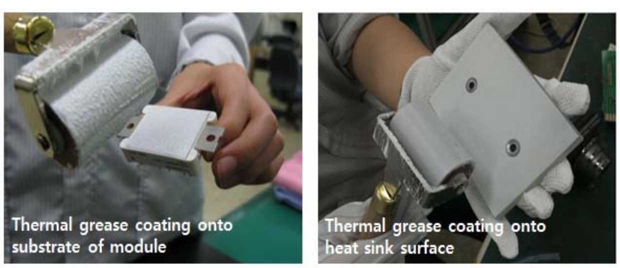


Figure 16. Applying Thermal Grease

1. Choose one side of the surface, module substrate, or heatsink to apply thermal grease.
2. Coat a rubber roller with thermal grease.
3. Paint the surface repeatedly using the rubber roller to create a uniform layer of thermal grease around 80 μm – 100 μm thick.

Since the thermal grease has the lowest thermal conductivity in the thermal path, a layer as thin as possible is necessary to keep the overall thermal resistance low. Recommended thickness of printing layer is 60 μm – 100 μm to fill the gap between two contact surfaces completely. Check the thermal grease thickness with thickness gauges, such as wet film combs or wet film wheels. Because manual control of the printing pressure and speed can be learned by experience, training is needed to achieve a technique for good quality printing layer in real application.

Alternatively, apply thermal paste by screen printing, for example using a honeycomb pattern. The recommended thermal paste thickness is 80−100 μm A thickness of the TIM layer in excess of this recommendation will unnecessarily increase thermal resistance.

When applying thermal grease, the material must be applied uniformly on the whole surface which is in contact to the module substrate surface. If the module is re−mounted, surfaces should be cleaned and TIM needs be applied again**.**

Pre−applied Thermal Interface Material

As an option the module may be prepared or provided with a pre−applied PCM layer. The recommended pattern for such PCM layer is shown in Figure 17.

Figure 17. Printing Pattern Example for TIM Material

Modules Shipped with Pre−applied Phase−change Material

Please refer to the application note referring to handling modules with pre−applied TIM material.

PRESS IN PROCESS OF MODULES WITH PRESS−FIT PINS

The press−fit process is a cost−effective way to assemble power modules without introducing additional thermal stress. The press−fit connection generates a good electrical, and also strong mechanical connection between the module and the PCB. This section deals with the mounting process to achieve suitable press−fit connections.

There are several types of presses available; from simple toggle presses to the automated pneumatic presses shown in Figure 18. If possible, monitor the press−in/press−out distance, speed, and force to achieve mechanical stability and high reliability of the press−fit connection. The travel distance during the press−in process should be controlled to ensure that the press−fit zone of the pins sits properly in the plated through hole. The speed also influences the quality of the press−fit connection; therefore, speed recommended by IEC standard should be applied.

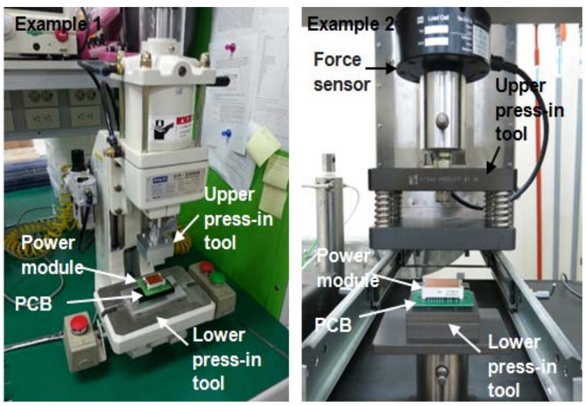


Figure 18. Press−in Presses

Generally, a module can be pressed in until the stand−offs on the four corners of the module touch the PCB. If, for example, more than one module is mounted on a PCB at short distances or assemblies are subjected to mechanical shocks in the application, the press−in method as described in Figure 23 is recommended.

Press−in Tool Construction

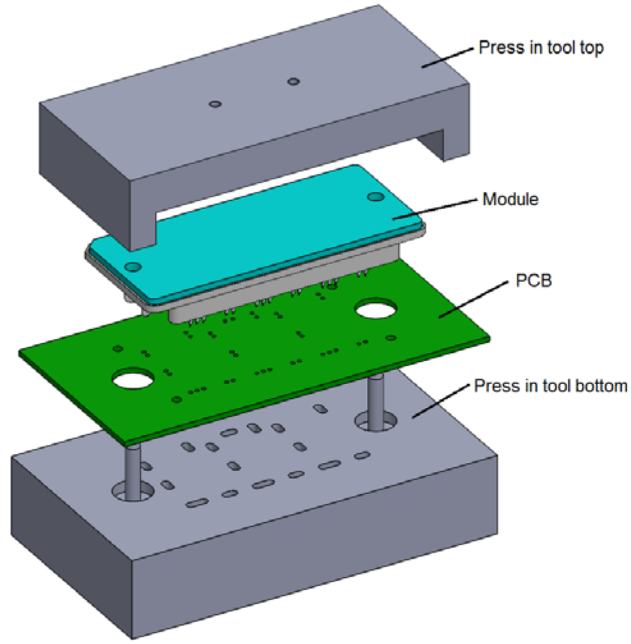


Figure 19. Recommended Tooling Construction for the Press−in Process

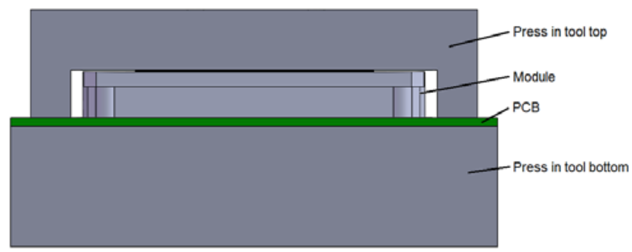


Figure 20. Press−in Tools Closed, Module is Pressed in

The preferred method is to press the module into the PCB from the top. Pressing multiple modules into one PCB can be done one by one; or all at once.

Pressing multiple modules at the same time requires a press−in tool according to the above detailed single tool. The tool has to ensure the correct leveling of the modules and the PCB to avoid any mechanical stress.

Press−in Process Parameters

The total press−in force is the result of the number of pins in a module, multiplied with the force required for a single pin. Press−in forces lower than 60 N/pin mean that press−fit pin may have a less secure connection in the plated through hole. The primary reason for the low press−fit force is that the diameter of plated through hole is too large for the press−fit pins. Press−in forces higher than 150 N/pin can cause mechanical damage to the press−fit terminal, the PTH, or to the tracks on the PCB. The recommended press−in speed ranges from 25 mm/min to 50 mm/min in accordance with the recommendations in IEC 60352−5.

The press−fit pins have to be pressed into the holes of the PCB to the correct depth. The center of the press−fit zone has to be at least 0.5 mm below the top surface and at least 0.5 mm above the bottom surface of the PCB (Figure 21).

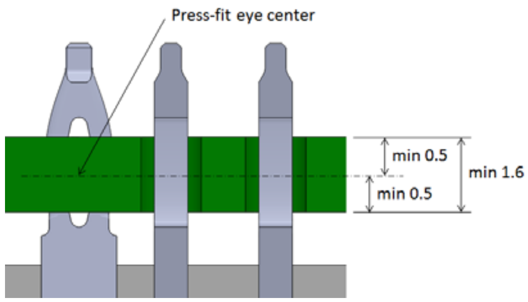


Figure 21. Press−in Depth in PCB

General Press−in Process

Figure 22 shows the general sequence of the press−in procedure. The press−in tool is comprised of two parts: the upper press−in tool is flat to contact with the module backside evenly and the lower press−in tool has engraved spaces to accommodate the press−fit pins.

The two parts of the tool need to be aligned to each other (a). In the first step of the assembly, the printed circuit board is placed on the alignment pins of the lower part of the press−in tool (b).

Then, the module is placed on top of the printed circuit board using the alignment pins. It is necessary to check if the module and the printed circuit board are in alignment (c). In the next step, the press−in force is applied via the upper part of the press−in tool to the backside of the module evenly. The module should be pressed−in with a speed of 25~50 mm/min until the stand−offs on the four corners of the module touch the surface of the printed circuit board while press−in distance and force are monitored at the same time (d).

It is required to adjust the traveling distance of the press to avoid damages to the module case due to pressure being applied. A simple manual press does not use a distance sensing system, so a distance keeper should be designed on the press−in tool to terminate press−in process appropriately. The next section describes the function of the distance keeper more in detail.

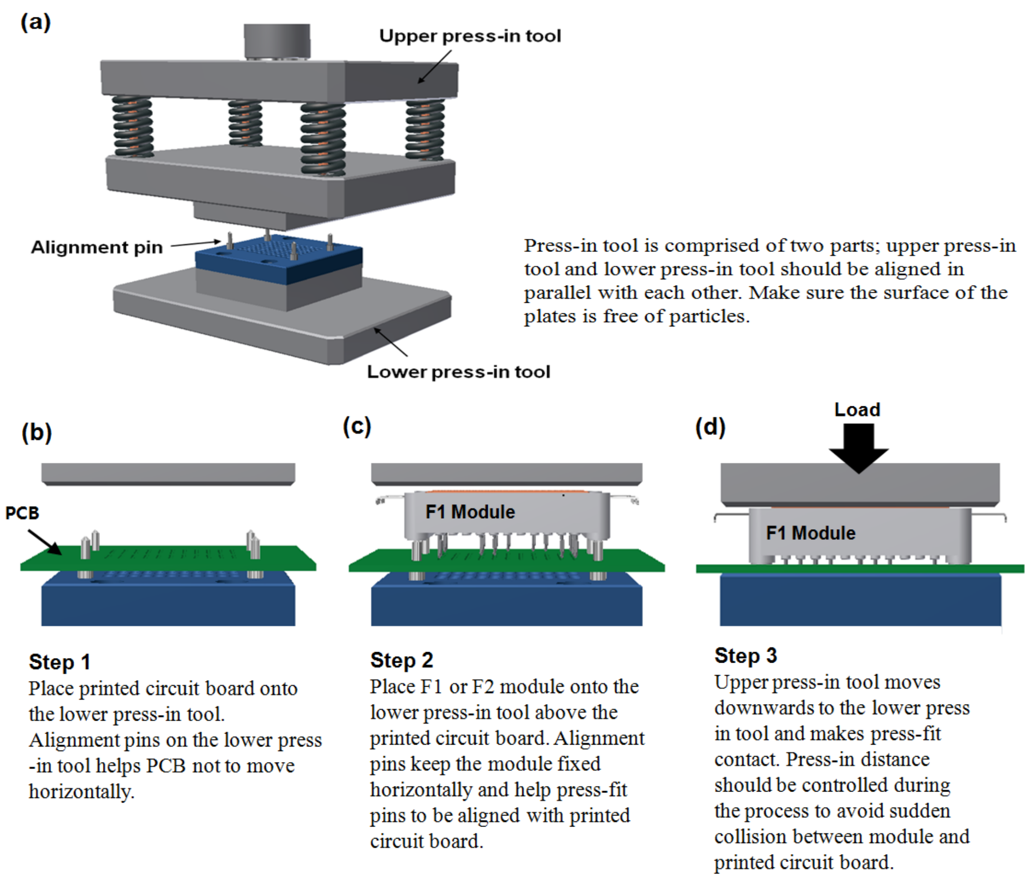


Figure 22. Press−in Process

Press−in Process for Multiple Modules

In case multiple modules are assembled to the same PCB and heatsink, height tolerances can result in unintended bending of the PCB or inappropriate heatsink contact.

Therefore, if more than one module is mounted on the same PCB, it is required to minimize the height tolerances between those modules. This section presents a modified press−in process related to that.

Figure 23 shows a press−in tool including distance keepers. The distance keeper terminates the press−in process and limits the press−in depth.

By contacting the printed circuit board ahead of the module case, it prevents direct contact between the case and the PCB. If the distance keeper contacts the surface, press−in force rises sharply and the press−in process can be terminated by reaching the limit of the press−in force. The distance keeper should be designed to avoid the collision with other PCB components.

The press−in process using the distance keeper is described in Figure 23. First, the PCB should be placed on the lower press−in tool (a). Then, a module is placed on the lower press−in tool and aligned with the PCB (b). The press−in stroke is applied to the backside of the module until the distance keeper touches the surface of the PCB (c).

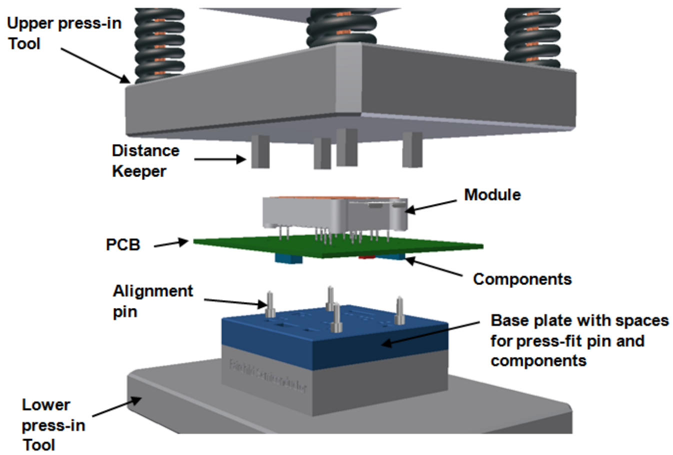


Figure 23. Press−in Process with Distance Keepers

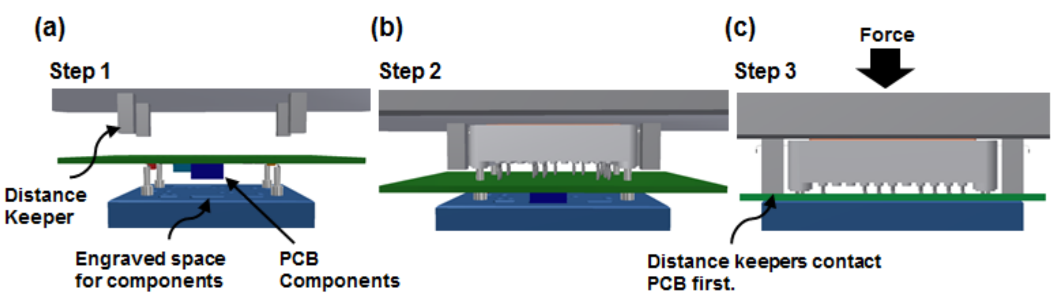
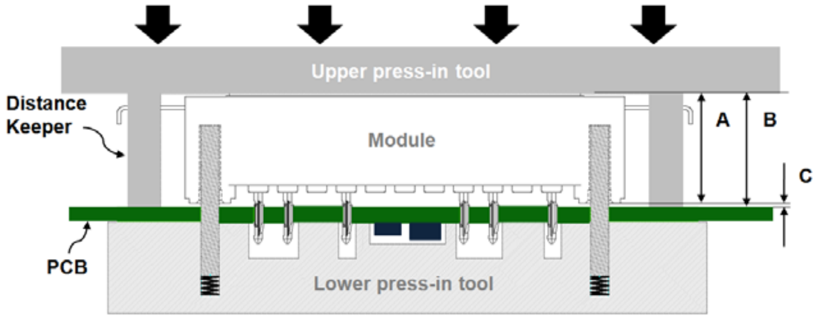


Figure 24. Schematic Description of the Press−in Process using Distance Keeper

As illustrated in Figure 25, depending on the length of the distance keeper, the contact length between press−fit pins and the plated through hole is determined. The total height of the module is 12.00 ±0.30 mm for Q0, Q1, F1 and F2 modules. It is recommended that the length of the distance keeper should be 12.40 (+0.05/−0) mm to achieve a stable contact length.

Consequently, an air gap between the top of the case and the surface of the PCB remains. Screwing the PCB down to the stand−offs of the module case, as shown in Section 5, is not allowable. Instead, the assembly should use space posts to support the PCB.



A: Package height (DBC bottom to case top) is 12.00 +0.30/−0.30 (mm)
B: Distance keeper height is 12.40+0.05/−0 (mm)
C: Air gap between case top and surface of the PCB

Figure 25. Press−in Depth in Accordance with Length of Distance Keeper

Press−in Tool Design Comments

Some design options should be considered to avoid press−in failure. First, the contact plate of the upper press−in tool must be larger than the module DBC substrate size. If the contact area of the upper press−in tool is smaller than the module substrate and pressure is applied to the center area of the module only, the module can be mechanically damaged during the press−in process. The press−in tool design should consider the size of the module.

If other components (capacitors, resistors) are assembled on the PCB next to the module mounting area, the press−in tool design should avoid collision during the press−in process. As shown in Figure 20, the lower press−in tool is designed with engraved spaces for the press−fit pin and other PCB Based on experimental results, components did not exhibit mechanical or electrical damages due to board bending during the press−in process when the distance between the center of the plated through hole and the edge of the component is 4 mm or more. It is also important to keep a certain area to provide support for the PCB.

Press−out Process

In some situations, it is necessary to remove power modules from the PCB. It is possible to disconnect the contact between module pins and PTH. The press−out process can be performed with the same equipment used in the press−in process.

Careful handling in the press−out process is essential to avoid mechanical damage to both the module and the PCB. PCB can be re−used once with a new module.

Please note: in case a module which was pressed out of a PCB should be used again, it is necessary to solder the module to the PCB; this is because the press−fit zone will remain deformed after the press−out process. An additional press−in cycle will result in low holding forces between the press−fit pin and PCB hole.

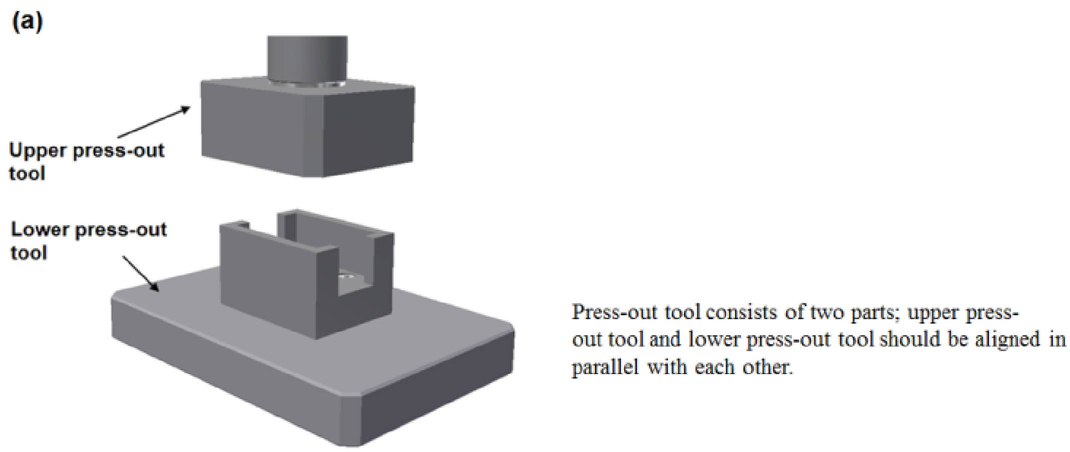


Figure 26. Press−out Tools

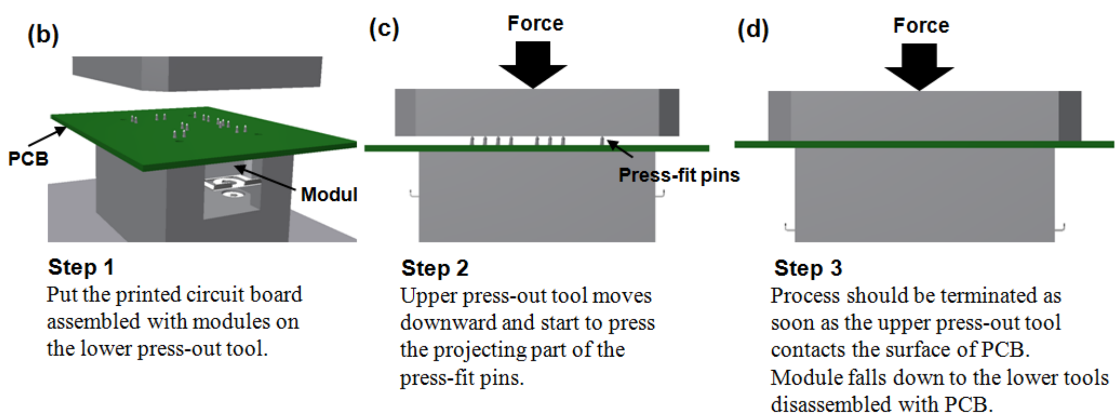


Figure 27. Press−out Process

The press−out tool consists of an upper and a lower press−out tool, as shown in Figure 26. The upper press−out tool should be parallel and aligned to the lower press−out tool (a). The assembled board should be placed on the lower press−in tool (b). The upper press−out tool moves down and contacts the projecting part of the pins. Press−out force should be applied to all pins evenly within 3−12 mm/min of press−out speed according to IEC 60352−5 (c). The module falls down to the lower tool as it is disassembled from the PCB. The press−out distance should be monitored to ensure the upper press−out tool does not apply unnecessary pressure on the PCB after the module is fully pressed−out (d).

Press−out Tool Design

The upper and lower press−out tools should be parallel to each other to prevent stress on the module during press−out process. Some pins may have contact with the upper press−out tool while other pins have no contact with tool. Pulling force can be exerted on the press−fit pins where upper the press−out tool does not contact.

The press−out tool design should consider other components assembled on the PCB next to the module mounting area to avoid collision during the press−out process. The specific tool to disassemble the modules from the PCBs has two parts similar to the press−in tool. The lower part serves as a support for the PCB. It has a cavity and supporting pins. It is important that the size of the cavity is close to the dimensions of the module. The supporting pins are in line with the fixing pins of the upper part. There may be components mounted to the PCB in the area of the tool, in such case it is possible to cut out the nest.

The upper part consists of two pressing plates that are connected by springs. The pins on the lower plate serve to fix the PCB. With the aid of these parts, the bending of the PCB can be prevented during pressing.

The ram fastened to the upper plate is designed according to the positions of the parts on the PCB and the layout of the pins.

It is not possible to press the module out with a flat plate if the PCB is thicker than 2.5 mm due to the overhang of the pins. In this case, pressing sticks positioned according to the pin layout are necessary.

Recommended spring force: 25N/mm
Recommended number of springs: 4pcs

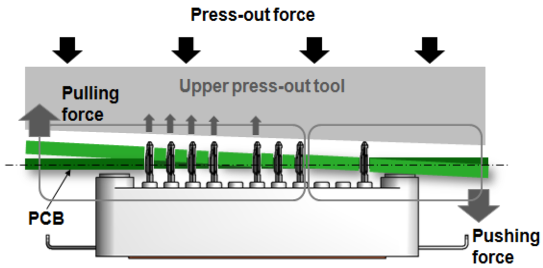


Figure 28. Press−out Failure Mode: Upper and Lower Press−out Tools Not Parallel

Due to the stress relaxation between contact partners, the initial contact force tends to reduce over time. In addition, the press−out force varies in relation to the contact ratio between press−fit pin and plated through hole. As shown in Table 6, the press−out force should be higher than 40 N per pin.

Table 6. PRESS−OUT FORCE AND SPEED SPECIFICATION

	Min.	Typ.	Max.
Press−Out Force (per Pin)	40 N		
Press−Out Speed	3 mm/min – 12 mm/min		

This condition is automatically fulfilled with the use of the recommended press−in tool.

SOLDERING TO PCB

PCB holes with fully plated through−holes will enable 100% wetting and fillets between pin and both sides of PCB.

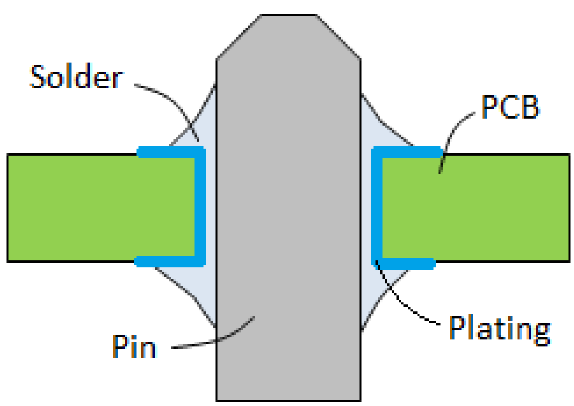


Figure 29. Solder Wetting of PCB Through Hole

Wave soldering profile

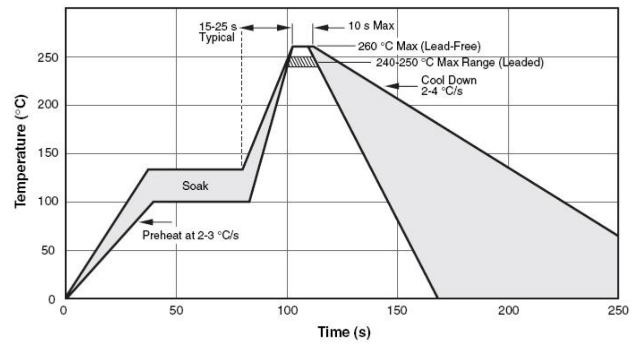


Figure 30. Wave Solder Profile

MODULE MOUNTING TO HEATSINK

This section describes how the modules are mounted to the heatsink. The different module family members have different sized mounting holes and recommendations.

Methods of Screw Clamping

There are two recommended screw clamping methods which apply to all modules. The F1 module is used as an example. Figure 31 describes one method for fastening the module to the heatsink. Fasten two screws simultaneously to prevent tilting or rising of one side of module during fastening. Electric screwdrivers can tighten the screws with the specified torque. Additional flat or spring washers are permissible, considering clearance and creepage distances specified later in this section.

Screw holes on heatsink need to be countersunk.

If method 1 cannot be applied, the method as described in Figure 32 is also acceptable. Fasten the first screw loosely to prevent tilting or rising of the module (step 1). Then insert the second screw with final torque so as to be fully tightened with the heatsink (step 2). Finally, apply full torque to the first screw for solid tightening with the heatsink.

For F1/F2 packages using metal clips, the torque is between 2.0−2.4 Nm using M4 screws.

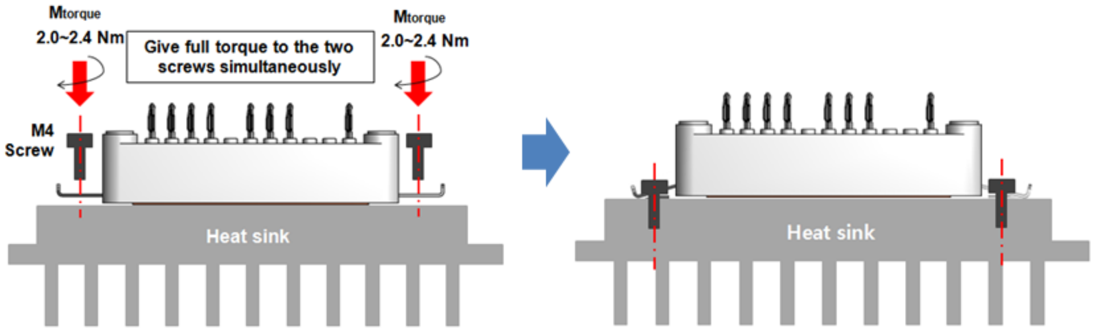


Figure 31. Illustration of Screw Clamping with Heatsink (Method 1)
Torque values apply to F1/F2 packages. See later diagrams for other packages.

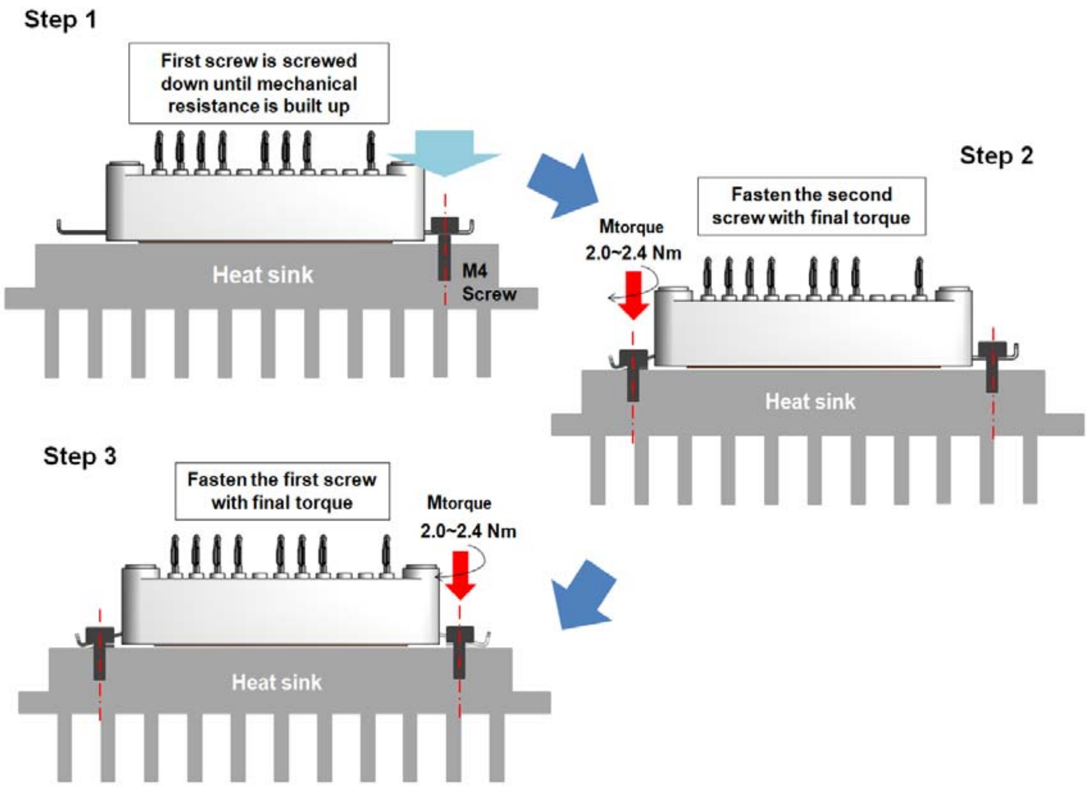


Figure 32. Screw Clamping Steps (Method 2)
Torque values apply to F1/F2 packages. See later diagrams for other packages.

Q0/Q1 Modules

When using screws with flat washers:

Metric screw: M4 (recommended screw type DIN7984)
Flat washer: D = 8 mm ISO 7092 (DIN 433)
Spring washer: D = 8 mm DIN 127 or DIN 128
Mounting torque: 1.6 − 2.0 Nm
Screw holes on heatsink need to be countersunk.

A torque wrench shall be used to tighten the mounting screws at the specified torque. Excessive torque may result in damage or degradation of the device. The inaccuracy of torque wrench tightening method can range up to ±12%. This has to be taken into account to prevent over−tightening the fastener.

Due to excessive temperature fluctuations washers should be used to prevent the loosening of the screws. After accurate tightening of the screws the spring washer exerts a constant force on the joint. The flat washer distributes this force on the plastic surface.

When using screws with pre−assembled washers:

Screws with pre−assembled washers (SEMS or kombi screws) combine the screw and the washers into a single component. These screws eliminate the need to slip the washers into place by hand, boosting the speed and efficiency of the assembly process. The specifications of these screws are provided below:

Screw size: M4 according to DIN 6900 (ISO 10644; JIS B1188)
Flat washer: According to DIN 6902 Type C (ISO 10673 Type S; JIS B1256)
Washer outer diameter:
8 mm diameter can be fitted onto the module
Split lock spring washer:
According to DIN 6905(JIS B1251)
Mounting torque range: 1.6−2.0 Nm
See Figure 33.

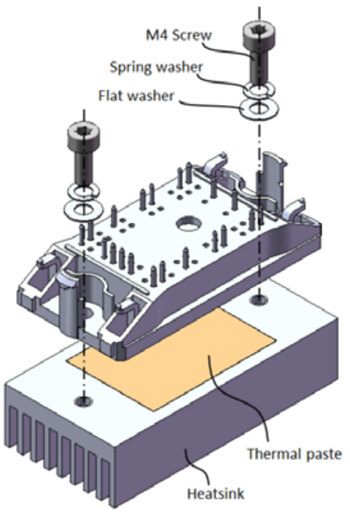


Figure 33. Q0/Q1 Modules − Mounting to Heatsink

Q2 Modules

Use M5 screws with torque of 3.0 – 5.0 Nm. See Figure 34.

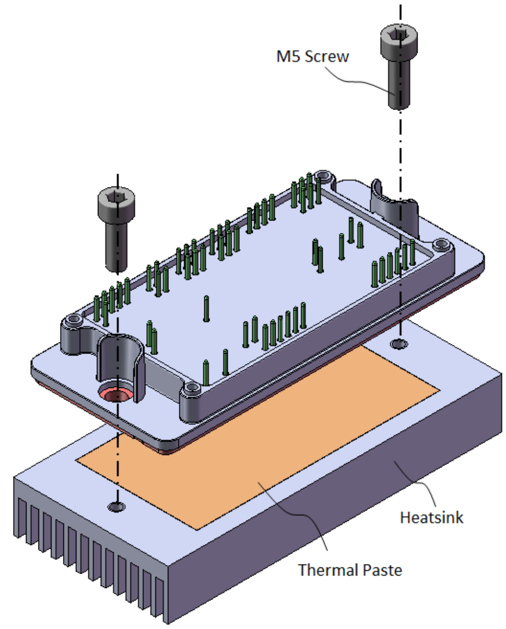


Figure 34. Q2 Modules − Mounting to Heatsink

MOUNTING HEATSINK AND MODULE TO PCB

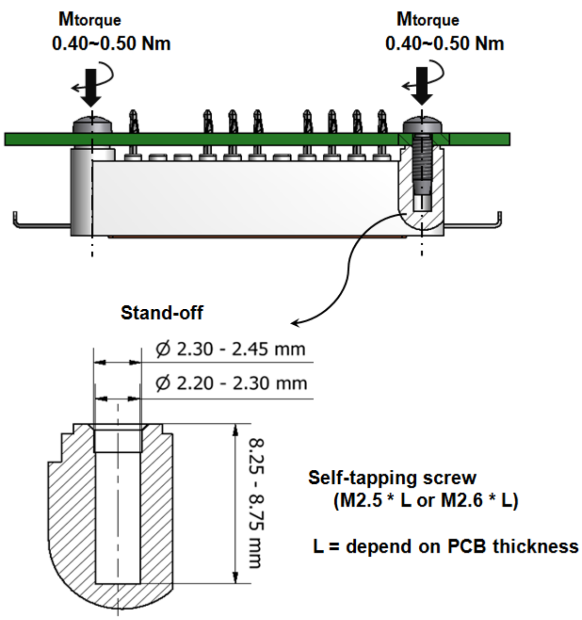
Mounting screws for attaching module to PCB

Screwing the PCB to the stand−offs on the module is one of the assembly methods. By adding this screw connection to the stand−offs, securing the assembly of module and PCB can be expected. Figure 35 shows the key dimensions of the stand−off. Self−tapping screws are recommended so that the screws form the thread in the hole.

For F1 and F2 modules, metric screws, self−tapping, with dimensions of 2.5xL or 2.6xL have been verified. The length of the screws (L) may differ depending on the thickness of the PCB. Typically, an 8 mm long screw (M2.5X8) can be used with 1.6 mm−thick PCB. Recommended mounting torque is 0.4~0.5 Nm for each screw. See Figure 30. This shows the recommended sequence for mounting the PCB to the module. Straight inserting avoids mechanical damage to the module case. An electric screwdriver helps achieve uniform force and speed for inserting the screws.

For Q2 modules, metric screws, self−tapping, with dimensions of 2.5xL have been verified. The length of the screws (L) may differ depending on the thickness of the PCB. Typically, an 8 mm long screw (M2.5X8) can be used with 1.6 mm−thick PCB. Recommended mounting torque is 0.3~0.5 Nm for each screw. See Figure 36. This shows the recommended sequence for mounting the PCB to the module. Straight inserting avoids mechanical damage to the module case. An electric screwdriver helps achieve uniform force and speed for inserting the screws.

Q0 and Q1 modules do not have screw holes for mounting onto the PCB.



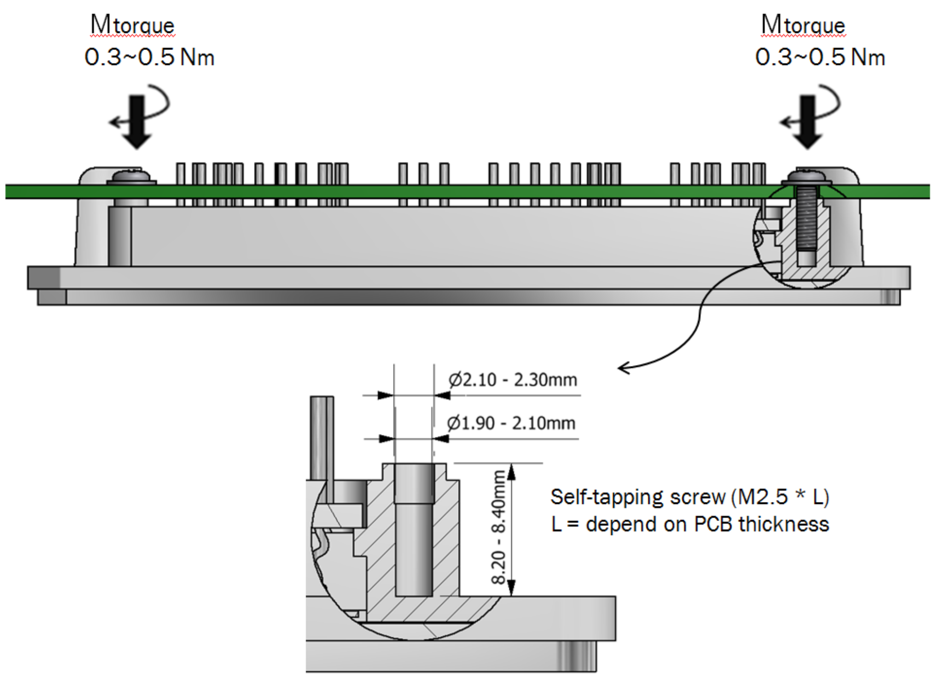


Figure 35. Q2 Screw Clamping on Stand−offs and Key Dimensions of Screw Hole

NOTE: Note: Do not screw the PCB down to the stand−offs of the module if an air gap remains between case top and PCB after press−in. This can lead to deformation of the PCB or other mechanical damage.

Assembly of the PCB and Heatsink

The overall structure of the mounted module should be considered.

If the PCB is large and heavy with other components assembled to it, there is some risk the PCB can bend, creating mechanical stress to the module and the PCB. When multiple modules are applied to the same PCB, height tolerance between modules can result in the mechanical stresses on the board and modules. To reduce stress, space posts should be added on the heatsink, as illustrated in to prevent movement of the PCB.

The recommended height of the space posts is 12.4 (+0/−0.1) mm. The effective distance between center of stand−off and the space post (= X) is 50 mm minimum. If distance keepers are used during the press−in process, resulting in tighter height tolerances; distances between the stand−off of the case and the space post (= X) smaller than 50 mm can be used.

Figure 36 shows the assembly procedure when space posts are used and the overall assembly structure:

Modules are first pressed into the PCB following the recommendations introduced in Section "Heatsink Surface" before heatsink mounting. Maintaining tight height tolerances between module and PCB is important. Next, the thermal interface material is applied. Then the modules and the PCB are placed on the heatsink (a).

Then the module is mounted onto the heatsink via the module's metal or plastic clamp. Refer to Section "Q2 Modules" for instructions of screw clamping (b). Finally, the PCB needs to be fixed on the space posts, as described in (c).

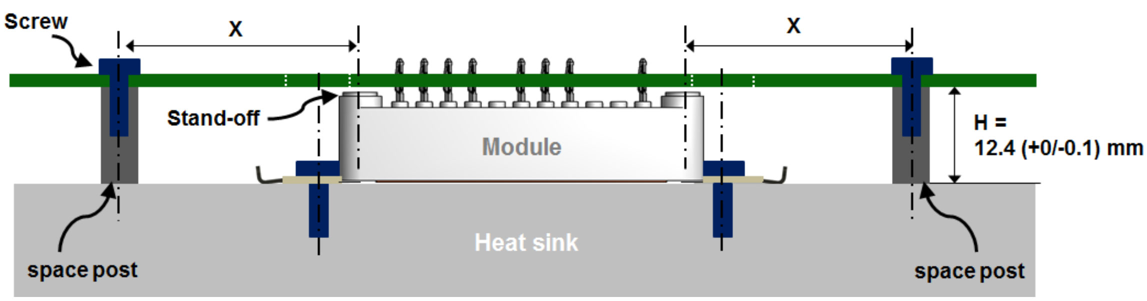


Figure 36. Space Posts for Assembly of PCB to Heat Sink

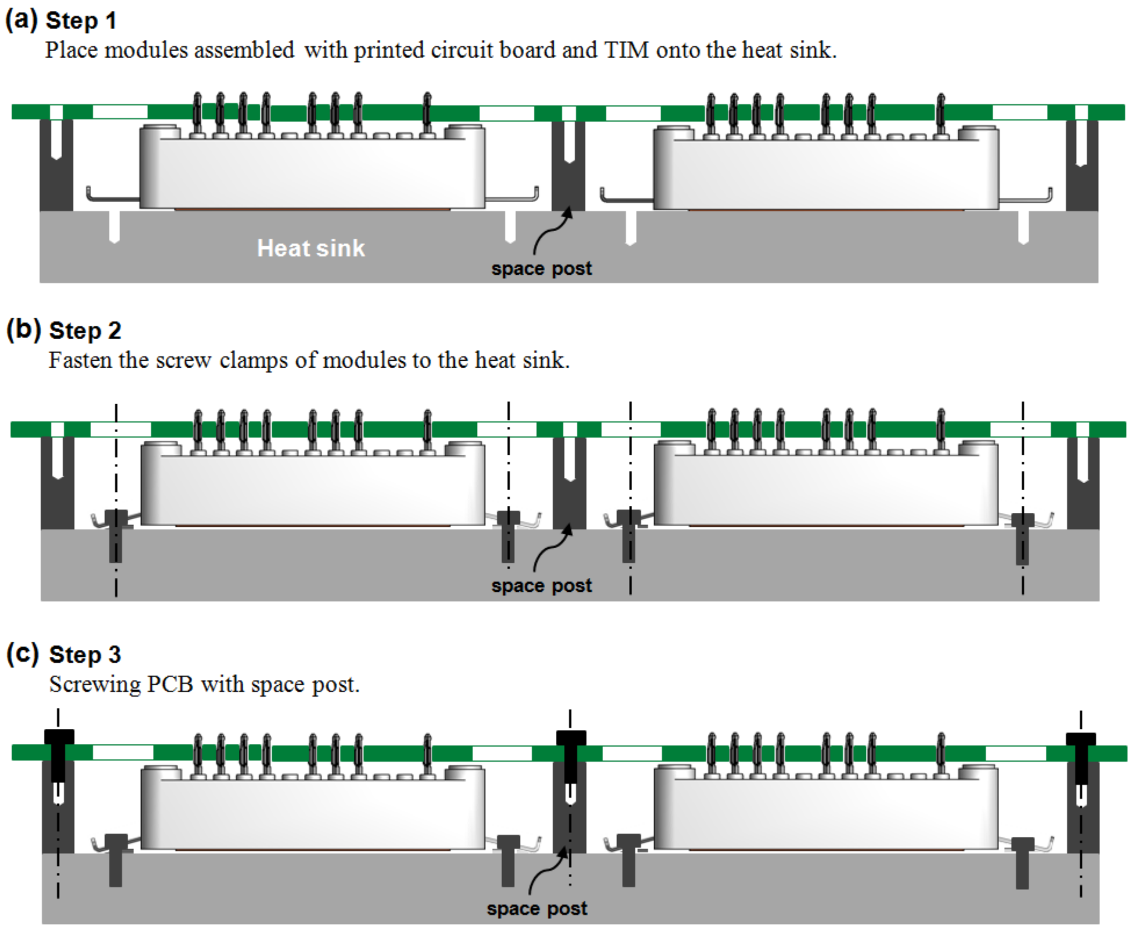


Figure 37. Example of Whole Assembly Process for PCB and Heatsink with Space Posts

During the assembly process, a single pin is not allowed to be drawn or pushed inwards or outwards from substrate more than ±0.2 mm or loaded with a force greater than 35 N (except during pressing−in of press−fit pins). The special design of press−fit pins prevents a deformation of pins greater than 0.1 mm during the press−in process. The tension of the pin must not exceed +/−5 N at a maximum substrate temperature of 100 °C.

Creepage and Clearance Considerations

The spacing of the assembly between the module and PCB must meet the clearance and creepage distance required by the relevant standards. After F1 or F2 modules are mounted to the PCB and heatsink, the minimum clearance is the distance between the screw head and the bottom surface of the PCB. The size (height) of the screw head and potential use of an additional washer, as well as the air gap between PCB and top side of the module, influence the creepage distance between the screw and the PCB and the module pins. F1 and F2 modules are mounted on the heatsink using M4 hexagon socket head screws, according to ISO 4762. Additional washers, according to ISO 7089, can be used. A distance of 6.8 mm between the screw and the PCB, as shown in Figure 38, can be maintained.

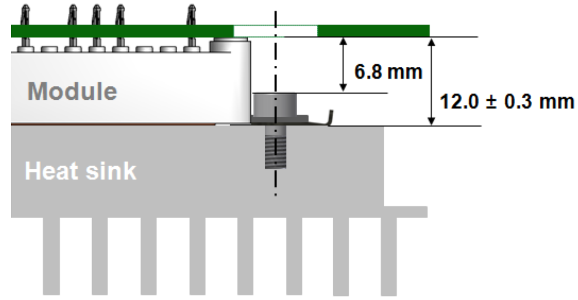


Figure 38. Distance between PCB and Screw

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
19521 E. 32nd Pkwy, Aurora, Colorado 80011 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative