# Design Considerations for Surface Mount Devices



In many applications, Surface Mount Technology (SMT) has replaced traditional printed circuit board (PCB) construction where elements of the components were mounted in openings through the circuit board. In IPC-7351, SMT is defined as:

"*The electrical connection of components to the surface of a conductive pattern that does not utilize component holes*"

In addition, for the purposes of this discussion, a surface mount device (SMD) will have all of its interconnecting terminals, including ground(s), on the same plane. An example of a surface mount attenuator mounted to a PCB is shown in Figure 1.

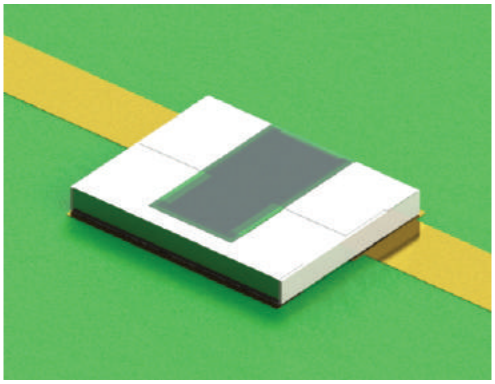


**Figure 1. SMD Attenuator on Board**

Use of SMT assembly offers a number of advantages including reduced size of components and the ability to mount components in almost any location on both sides of the board. This leads directly to significant decreases in size and weight of the resulting assembly. With these advantages, however, come additional complexities in the design. The main areas of concern are thermal dissipation and high frequency electrical performance. In addition, soldering requires careful consideration as interconnect density is generally greatly increased.

SMT assembly is generally accomplished using tin bearing solders. Proper soldering technique is paramount to the reliability. Barry Industries has designed their devices to be soldered in accordance with the Industry Standard IPC-A-610. This standard is quite extensive and is available for purchase at www.ipc.org, but for the purposes of this discussion, a few of the key elements will be highlighted.

IPC-A-610 defines an acceptable solder joint to have the following characteristics:

- A solder fillet will appear generally smooth and exhibits good wetting of the solder to the parts being joined
- The outline of the parts being soldered is easily determined
- Solder at the joint creates a feath ered or blended edge
- Fillet is either concave in shape or has a wetting angle *not exceeding* 90 degrees.

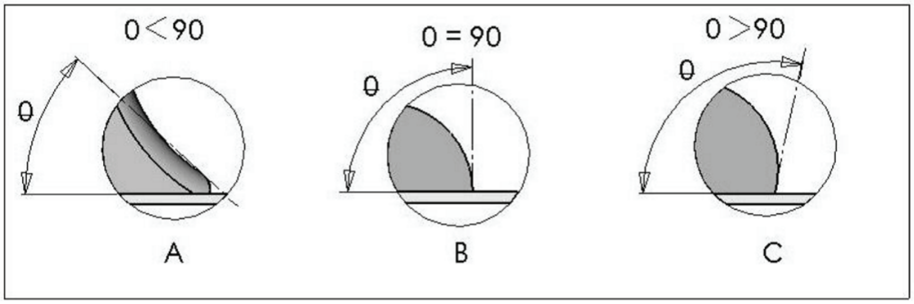



**Figure 2. IPC-A-610 Solder Wetting Angles**

Figure 2 illustrates the definition of the wetting angle as specified in IPC-A-610.

In addition, the land pattern design must take into account both the requirements for forming a reliable solder joint and the performance concerns discussed later.


60 Walton St. Attleboro, MA 02703 • T (508) 226-3350 • F (508) 226-3317 • sales@barryind.com

To achieve optimum solder joint characteristics, the guidance of IPC-7351, Generic Requirements for Surface Mount Design and Land Pattern Standard should be followed.

Thermal energy in a SMD must be carried away from the component at a rate sufficient to limit the temperature rise to a level that does not cause damage. Because the thermal resistance of a typical PCB is much higher than a heat sink, it is often a challenge to design a PCB capable of dissipating the full rated power of the component. Careful analysis and in application testing are recommended to ensure that the device is operating within the specified limits. Greater detail is available in Barry Industries' application note "*SMD Resistor Thermal Analysis*".

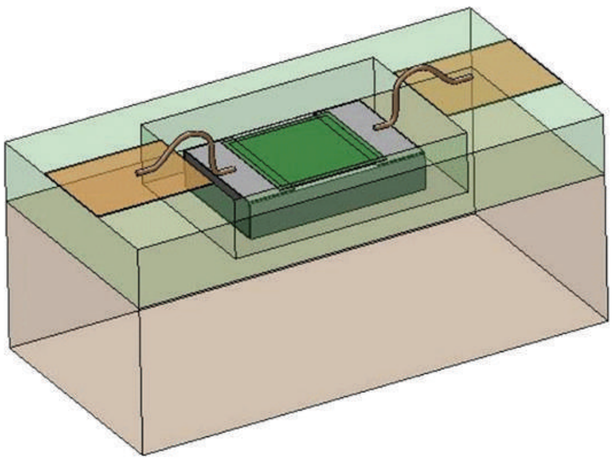


**Figure 3. Resistor Mounted on Groundplane Through PCB**

When a component is mounted such that its ground plane is in contact with the ground plane of the interconnecting circuitry as shown in Figure 3, the majority of the electromagnetic fields are contained within the device.

In this case, the effects of the PCB on the electromagnetic characteristics of the device are relatively minor and related to the parasitic reactance of the interconnects.

When a component is mounted on the surface of the PCB as shown in Figure 4, however, the electromagnetic fields are present in both the component and the PCB. The characteristics of the PCB, including the relative permittivity and distance to ground surfaces, become a primary factor in the overall performance of the component. The design of the PCB, including the interconnecting transmission lines, component land pattern, relative permittivity of the PCB, distance to ground surfaces and interconnecting vias must be taken into account in modeling the component performance.

Barry Industries offers attenuator, resistor and termination chip devices in a wide array of package styles. Some of these package styles are designed for SMT assembly while others require mounting through a PCB to operate as specified. The following table lists the available terminal configurations and the intended board configuration.

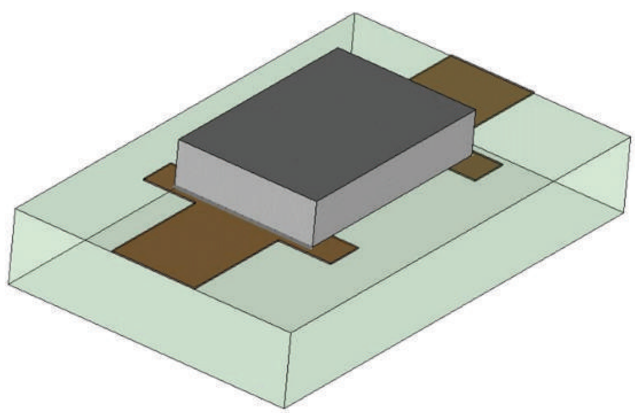
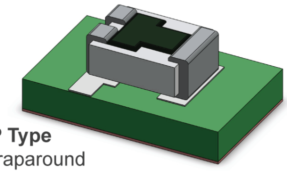


**Figure 4. SMT Resistor on PCB Surface**

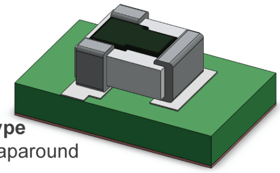
# Design Considerations for Surface Mount Devices



SMD ATTENUATORS

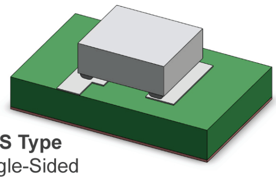


**AP Type**
Full Wraparound

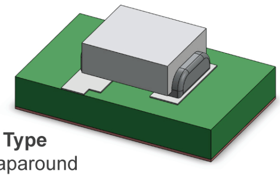


**AT Type**
3-Sided Wraparound

FLIP-CHIP SMD ATTENUATORS

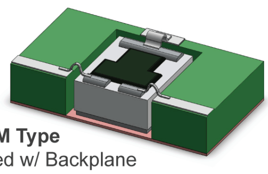


**AS Type**
Single-Sided

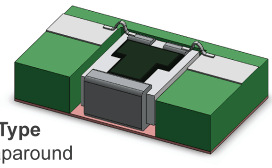


**AK Type**
1/4 Wraparound

PCB CAVITY ATTENUATORS

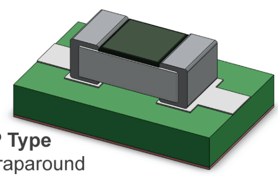


**AM Type**
Single-Sided w/ Backplane

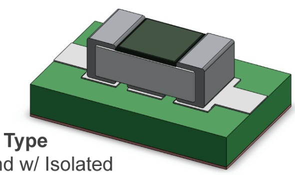


**AV Type**
1/2 Wraparound to Ground

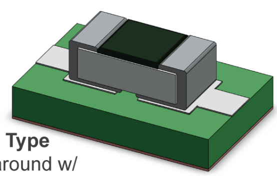
SMD RESISTORS



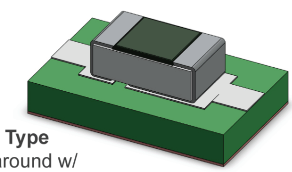
**RP Type**
Full Wraparound



**RY Type**
Wraparound w/ Isolated Center Pad

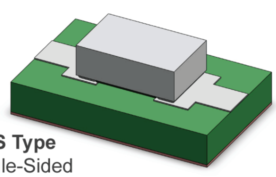


**RE Type**
Wraparound w/ Both Mounting Pads Extended

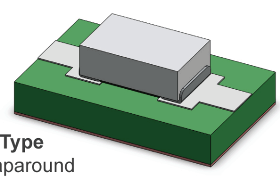


**RZ Type**
Wraparound w/ One Extended Mounting Pad

FLIP-CHIP SMD RESISTORS

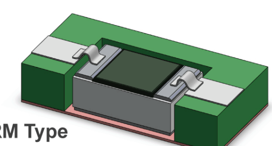


**RS Type**
Single-Sided



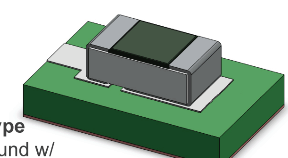
**RK Type**
1/4 Wraparound

PCB CAVITY RESISTOR



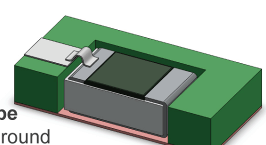
**RM Type**
Single-Sided w/ Backplane

SMD TERMINATION



**TZ Type**
Wraparound w/ Extended Mounting Pad Connected to Ground

PCB CAVITY TERMINATION



**TV Type**
1/2 Wraparound to Ground