



Understanding the TR Footprint

Purpose:

The purpose of this Application Note is to go into detail about the TR footprint. This document will cover topics such as acceptable plating materials, proper dimensions and choosing the correct footprint.

The Footprint:

The TR footprint is a piece of artwork implemented on the top layer of the PCB such that the compression mount TR mounts to the board on the footprint, applying signal and ground connections in the correct locations.

In a standard footprint .pdf there will be several individual footprints on different pages. These footprints consist of both microstrip and stripline versions of the different TR mechanical configurations such as straight mount and right angle. While a majority of the footprint remains the same through the different configurations, there are important differences such as keep-out areas and extra holes in the PCB. For implementation on your PCB find the page with the correct TR mechanical configuration and routing method (microstrip vs. stripline) and refer to that image for dimensions.

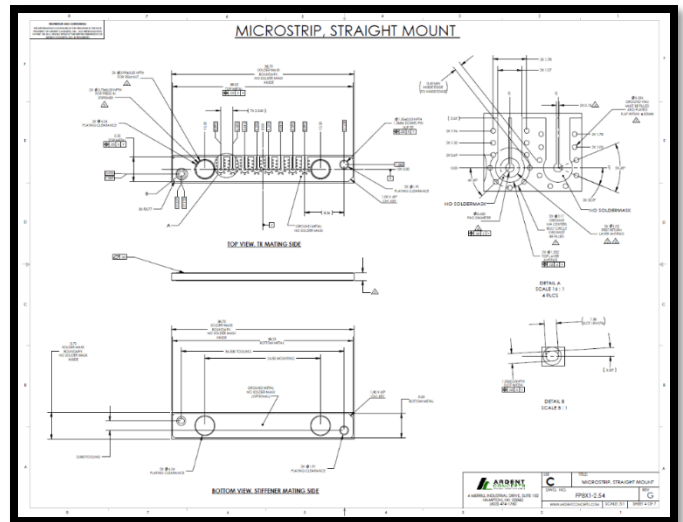


Figure 1

A standard microstrip footprint is shown in Figure 1. This footprint is optimized for up to 20 GHz applications. For anything over 20 GHz, we recommend the performance of a Footprint Optimization, which is a service that Ardent offers.

Plating:

In the footprint document, the notes call out for hard gold over nickel plating. It is important to note that this is not the only acceptable form of plating and that ENIG, ENIPIG or any other form of noble metal plating is acceptable. Tin or lead plating **prohibited**.



Important Notes When Reviewing Footprint:

There are a few important notes to pay attention to when implementing the TR footprint on your PCB that are not pulled front and center when looking at the TR footprint document. Those key components are as follows:

- The TR footprint requires **NO SOLDERMASK** within the artwork. Soldermask is often uneven and causes planarity issues, which is problematic for the TR's compression mount technology as it relies on a flat surface to make the proper connection.
- There are two different size callouts for the TR's mounting holes:
 - 3.99 mm diameter mounting holes for PCBs that have a thickness of greater than 93 mils (2.36 mm)
 - 3.73 mm diameter mounting holes for PCBs that have a thickness of less than 93 mils (2.36 mm)
- The TR footprint has an alternating ground via bolt pattern, which is shown in the footprint. This is important to implement as it severely cuts down on crosstalk between TR channels.

Who is Amphenol Ardent Concepts?

Amphenol Ardent Concepts is a leading designer and manufacturer of high performance multicoax and coaxial assemblies, connectors, and sockets used in the development of next generation semiconductors and electronics systems. Our core technology is the smallest, fastest, most electrically efficient compression mount connector technology worldwide. As data rate requirements increase and devices and systems shrink, Ardent's products deliver superior signal integrity in a dense footprint that can be reusable across programs to maximize cost savings.

[Amphenol Ardent Concepts](http://www.amphenolardentconcepts.com)

4 Merrill Industrial Drive Hampton, NH 03842

(603) 474-1760

Sales: info@ardentconcepts.com Technical: Support@ardentconcepts.com