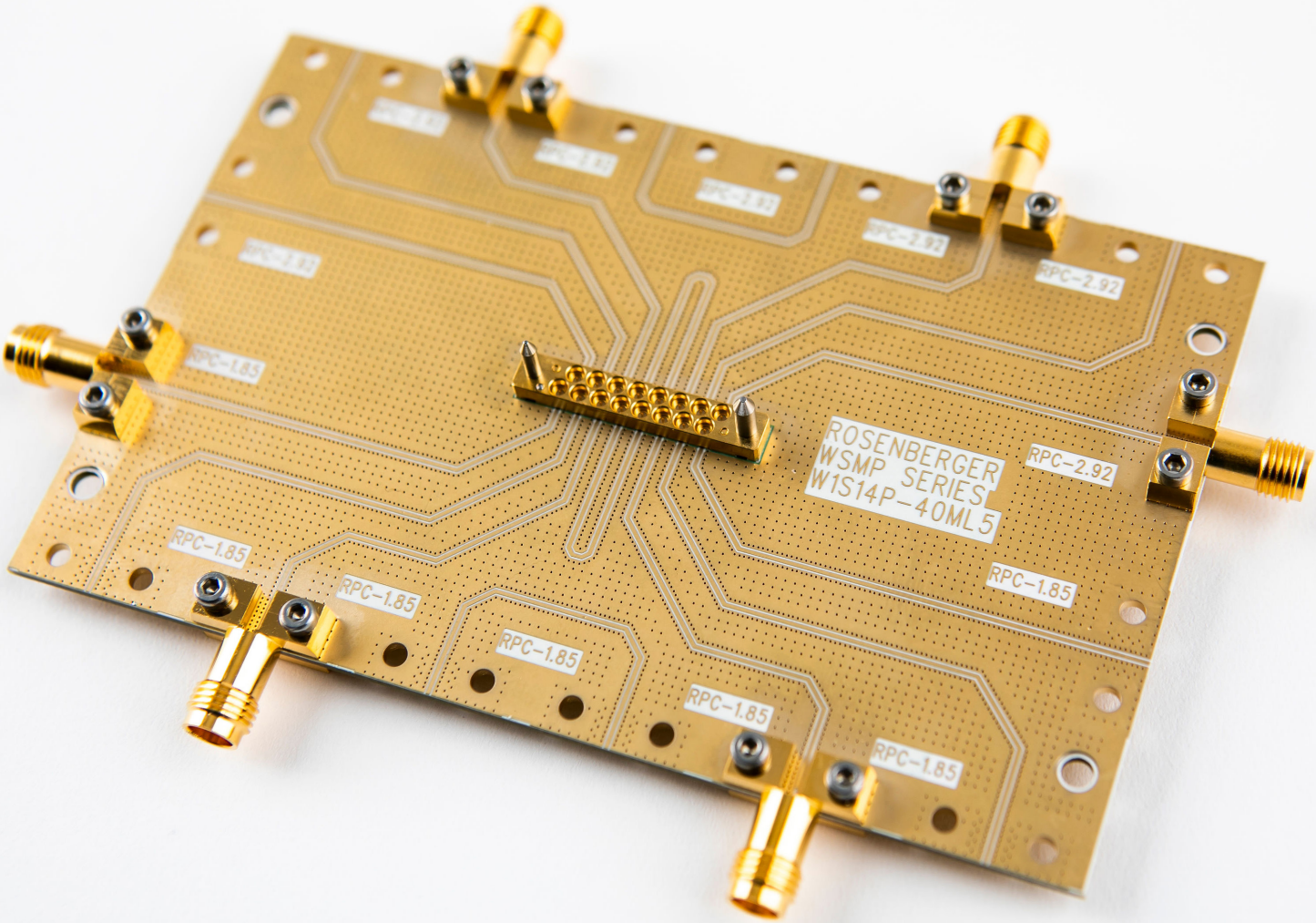
Rosenberger

Printed Circuit Board (PCB) Footprint Optimization

Rosenberger North America



SUMMARY & PURPOSE:

This application note describes the Printed Circuit Board (PCB) footprint optimization service that Rosenberger offers. The PCB footprint optimization service has been offered by Rosenberger for many years and has demonstrated itself to be a valuable service for our customers. The purpose of this application note is to assist our customers when footprint optimization is needed and need information for the application.

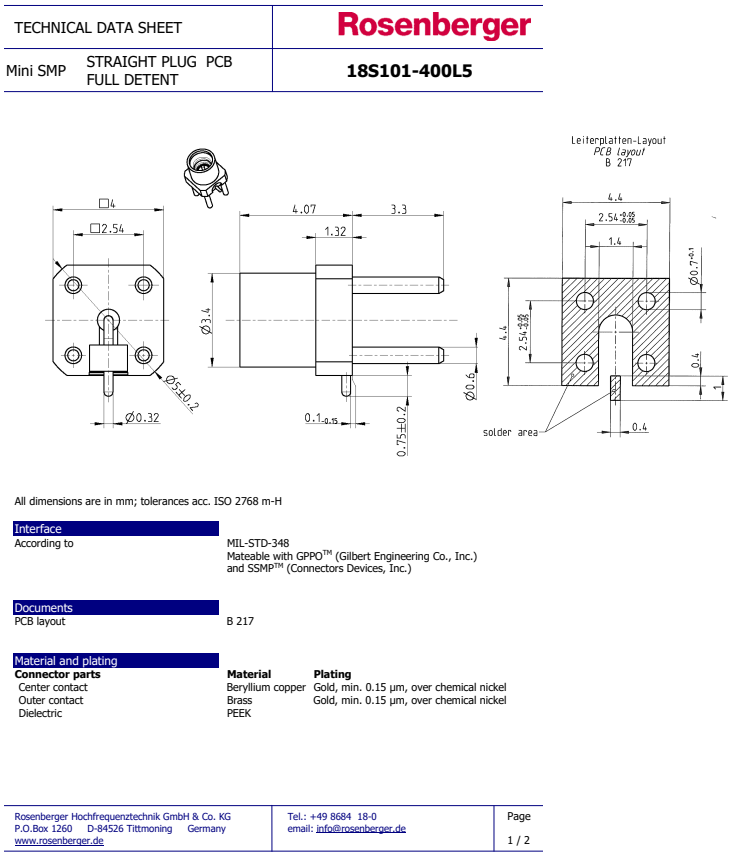
SECTION 1 - INTRODUCTION:

Rosenberger North America offers a Printed Circuit Board (PCB) footprint optimization service to its customers. The PCB footprints are used to optimize S-parameter performance of our PCB connectors for a specific PCB stackup. Rosenberger may justify waiving the service fee if the business case can be made and typically requires 2 to 3 weeks to complete, depending on the workload of our signal integrity group and the complexity of the design.

This article concentrates on clarifying the following:

- What a standard connector footprint is, where and when it may be used, and the standard footprint's potential limitations.
- Typical assumptions that could be made by a connector manufacturer when designing a new RF connector.
- Basic characteristics of an optimized footprint, what data is needed from the customer to generate the footprint, and the overall process Rosenberger uses for generating a custom footprint.
- A brief glossary of terms that are commonly used in custom footprint and simulation discussions.

At the end of this applications note, four (4) of the common PCB transmission line layout styles are listed.



TECHNICAL DATA SHEET | Rosenberger

Mini SMP STRAIGHT PLUG PCB FULL DETENT | 18S101-400L5

Leiterplatten-Layout
PCB layout
B 217

All dimensions are in mm; tolerances acc. ISO 2768 m-H

Interface
According to: MIL-STD-348
Mateable with GPPO™ (Gilbert Engineering Co., Inc.) and SSMP™ (Connectors Devices, Inc.)

Documents
PCB layout: B 217

Material and plating

Connector parts	Material	Plating
Center contact	Beryllium copper	Gold, min. 0.15 µm, over chemical nickel
Outer contact	Brass	Gold, min. 0.15 µm, over chemical nickel
Dielectric	PEEK	

Rosenberger Hochfrequenztechnik GmbH & Co. KG
P.O.Box 1260 D-84526 Tittmoning Germany
www.rosenberger.de
Tel.: +49 8684 18-0
email: info@rosenberger.de
Page 1 / 2

Figure 1 - Page one of the popular and high performing mini SMP 18S101-400L5 and the standard footprint MB 217

SECTION 2 - STANDARD FOOTPRINT

With all of the connectors Rosenberger manufactures, best practice includes a standard PCB layout. This is what a basic PCB layout could potentially be, or at least is a starting point for a printed circuit board design with respect to a given connector. Typically, the standard footprint will mirror the bottom ground surface of the connector, along with a signal launch pad. Please refer to Figure 1.

Note in Figure 1 how the bottom surface of the 18S101-400L5 mini SMP mirrors the ground surface along with a signal launch pad. The launch pad is the most significant section of the optimization, which can include antipads in the ground layers underneath if necessary for impedance matching. The pad location and dimensions, along with possible antipads, is dependent on the PCB stackup including dielectric permittivity and dielectric thicknesses. Also note, the connector data sheet (Figure 1) that "PCB layout B 217" is identified as the standard footprint.

However, standard footprint may have their limitations. Standard footprints may not present a constant impedance and/or be extremely lossy over a microwave frequency band, due to geometry or material limitations. Please see Table 1 for a few common PCB materials and their corresponding loss characteristics.

In summary, while a standard footprint has its place, and is often very useful, many high-performance PCB applications require a custom footprint. Rosenberger has the experience and depth-of-knowledge to help you obtain the highest performance possible with our connectors in your design.

PCB Material	Dielectric Constant	Dissipation Factor
FR4	3.58 to 4.8	.0195[1]
RO/Duroid5880[2]	2.33	.0005-.0012
RO4350B[2]	3.66	.0037-.0331

Table 1
[1] Depends on variety of FR4 material.
[2] RO/Duroid® 5880 and RO4350B are registered trademarks of Rogers Corporation.

SECTION 3 - CONNECTOR DESIGN:

When Rosenberger designs a new microwave connector, they include the 50 Ohms impedance transition to a 50 Ohms PCB transmission line, including microstrip and grounded coplanar waveguide (GCPW). Designing performance from a coaxial connector through the mode conversion to a PCB is a vital step of the design.

With the GCPW, a low-loss material will be used, such as those shown in Table 1, above. Testing is performed in a similar manner and with multiple connectors on very carefully designed PCBs.

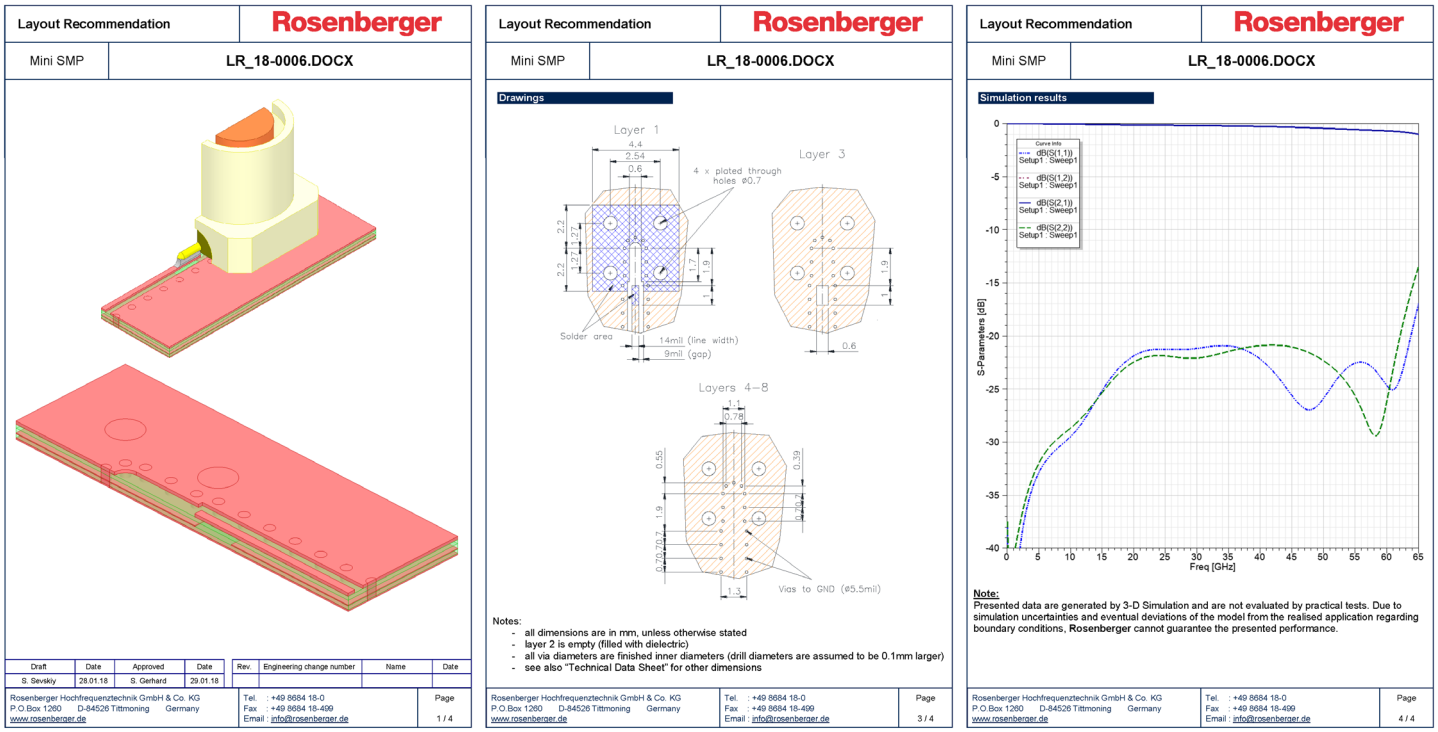


Figure 2 - Optimized Footprint results

SECTION 4 - CUSTOM FOOTPRINTS:

When a design requires a high-performance PCB design, for use with Rosenberger connectors, Rosenberger can assist with an optimized footprint. Rosenberger can take the design and simulate it using Ansys HFSS or Simulia CST.

Rosenberger is the right choice for high-performance RF connector needs and corresponding PCB optimization. Rosenberger has:

- Many years of experience with PCB/footprint optimization.
- Can make footprints for virtually any connector Rosenberger manufactures.
- Enormous simulation capability.
- Exceptional turnaround time.
- A large library of PCB materials.

For Rosenberger to properly simulate a design, please provide the following via a footprint request form:

a) Company name, address, contact name, email and phone.

b) The Rosenberger connector(s) being considered.

c) PCB stack-up information: Usually filled out in the footprint request form but may also be provided in a spreadsheet. The PCB stack-up information is critical because it will provide Rosenberger with the details of the PCB's materials electrical characteristics.

When one receives a custom footprint from Rosenberger, they will typically receive a multi-page document that shows:

1) The connector on the PCB in a perspective view with a cut-away/cross sectional view.

2) Drawings of the PCB pad layout, with the relevant layers being shown.

3) Predicted performance with S11, S12 and S22, S21 (Forward & Reverse) graphs/waveforms. Returning to the 18S101-400L5, as an example, please see Figure 2 above. Note how this connector performs in this application, 21 dB or better, return loss through 55 GHz.

Written by Bruce Cardwell, Rosenberger North America, Akron

GLOSSARY OF TERMS USED IN FOOTPRINT OPTIMIZATION & SIMULATION

Aperture: A hole, or gap. In the context of PCB footprints, an opening below a signal carrying trace, a cut-out below the signal carrying trace. Apertures can occasionally be found directly below the center contact of a connector on some optimized designs.

Coplanar Waveguide: A signal trace, similar to microstrip except that ground trace will be adjacent and on each side of the signal trace.

CST: Three-Dimensional Electromagnetic (3-D) Simulation software developed and sold by Simula, a division of Dassault Systems.

Dielectric: The property of an insulating material to transmit electromotive force, or permit the transmission of electromotive force. Typically associated with capacitance and the ability of the material to transfer electric force is called permittivity.

Encrypted Model: A model, in the Rosenberger context, that typically is a connector, and the encrypted model represents the electrical characteristics of the connector without revealing the internal mechanical characteristics, many of which are proprietary.

Footprint (PCB): The actual trace, or series of traces on a printed circuit board (PCB) that the connector is soldered to.

Grounded Coplanar Waveguide: Similar to Coplanar Waveguide except that there is a ground plane below the signal and adjacent ground traces.

Ground plane: A layer on a PCB which may be on the top, bottom or within the PCB that is usually at ground potential.

HFSS: Three-Dimensional (3-D) Electromagnetic Simulation software, developed and sold by Ansys Corporation.

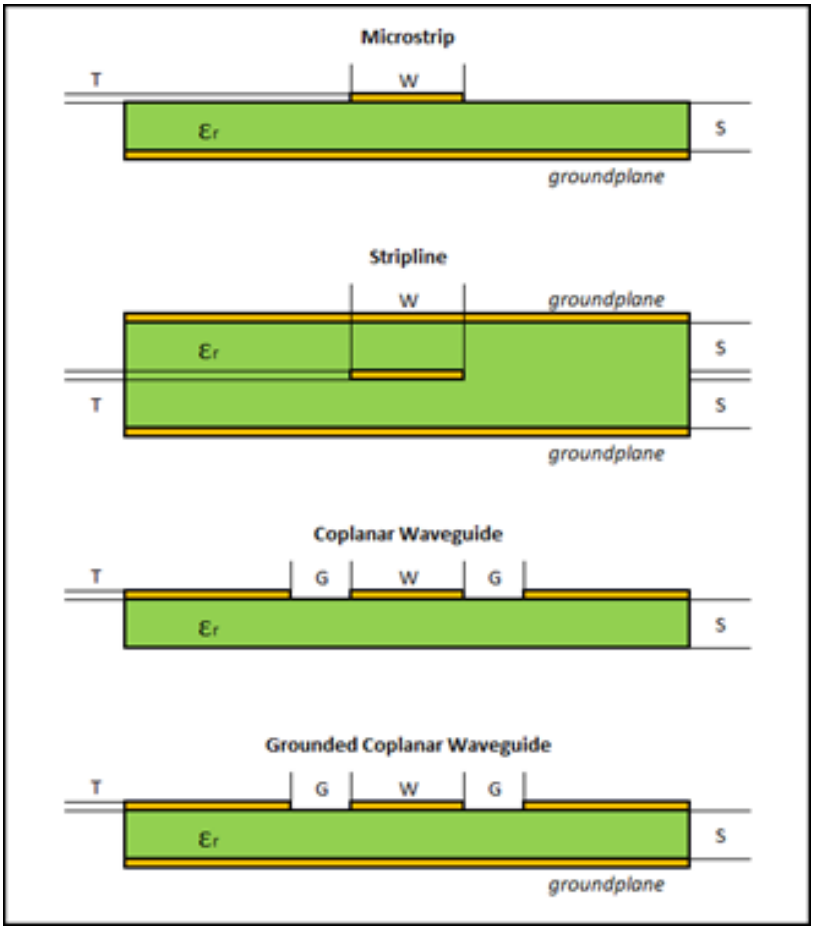
Microstrip: A small 50-ohm PCB trace, on the surface of a PCB with a solid or ground plane beneath it.

Shrink Wrapped model: A 3-D model, that is a Step file (.stp) that only displays the outer envelope or features of the object; internal features and dimensions are not visible or displayed. Generally, we use this to protect some of the proprietary features of our designs.

Step File (.stp): A 3-D Computer Aided Design (CAD) file that represents an object.

Stripline: A small 50-ohm PCB trace that is contained within a printed circuit board.

SOME COMMON PCB LAYOUT STYLES:



Pride in Precision.

Rosenberger

Rosenberger North America
P.O. Box 309
Akron, PA 17501 USA Phone: +1 717-859-8900
Email: salesinfo@rosenbergerna.com
Web: www.rosenbergerna.com
AS9100:D and ISO 9001:2015 Certified

Rosenberger

Rosenberger Hochfrequenztechnik GmbH & Co. KG
Hauptstrasse 1 | 83413 Fridolfing
P.O. Box 1260 | 84526 Tittmoning
Germany
Phone: +49 8684 18-0
Email: info@rosenberger.de
Web: www.rosenberger.com
Certified by ISO/TS 16949 DIN EN 9100 ISO14001