PCB design Support for Coaxial connector

Oct 2015

Hirose Electric Co., Ltd.

Outline

- *Introduction*
- *Mechanical performance*
- *Products*
- *Specifications*
- *Electrical performance*
- *PCB design considerations*
- *Appendix*

Introduction

◆ Purpose

Excellent performance connector for test port with screw mount up to 50GHz.
Realizes good electrical performance with stub-less structure and easy assembly with screw mount.
This design guide provides the information of product performance and PCB design in order to obtain full performance of the connectors.

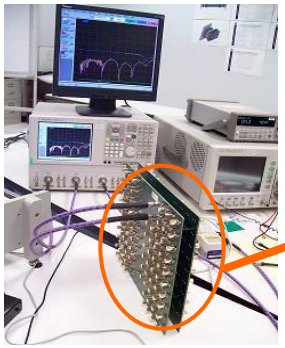
◆ Scope

Show the simulation and measurement results using 2.4mm,2.92mm vertical mount RF connector.

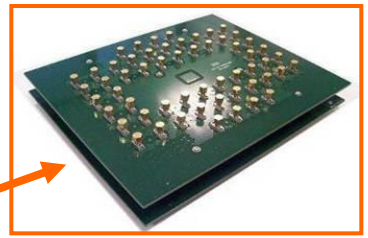
◆ Application and Interpretation

This connector allows high density test port layout due to its vertical mount feature ,which also provides flexible pattern design.

High Frequency Measurement Fixture



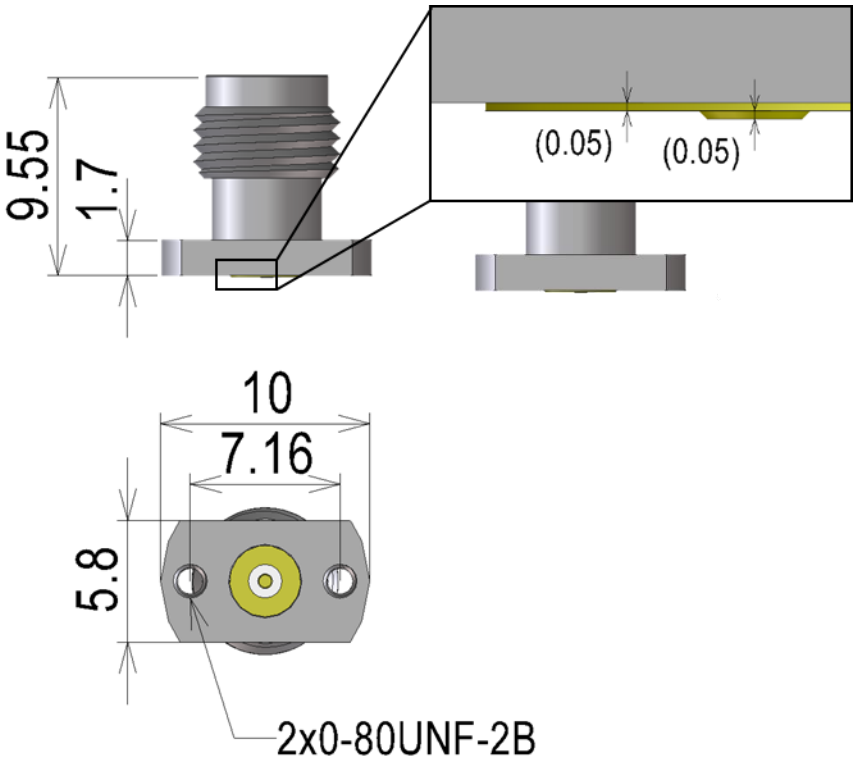
IT5 Demo board



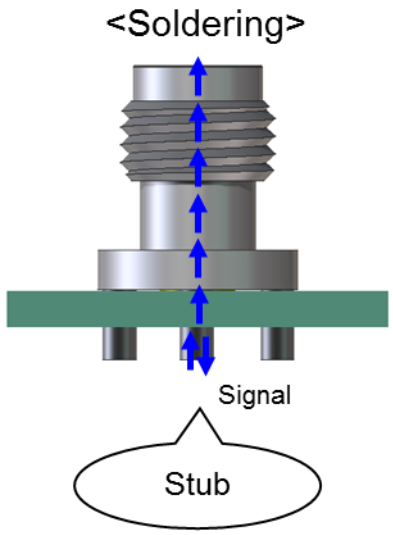
Used as test ports for high-speed, high-pin-count signals

Mechanical performance

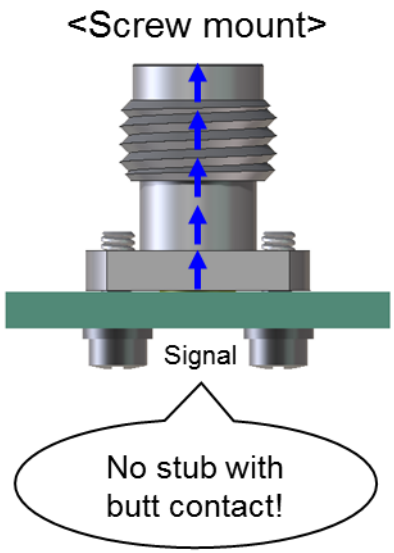
◆ Reliability



◆ Compliant pin design



Soldered center conductor could cause signal reflection by stub.



Center conductor with butt connection reduces signal reflection.

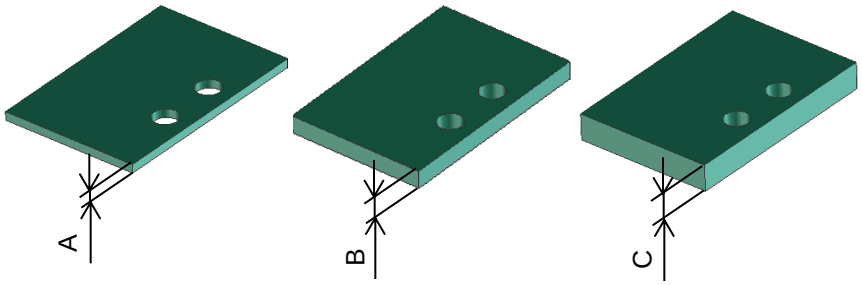
Excellent high frequency performance is achieved with stub-less design and stable assembly.

Mechanical performance

◆ Accepts various PCB thickness

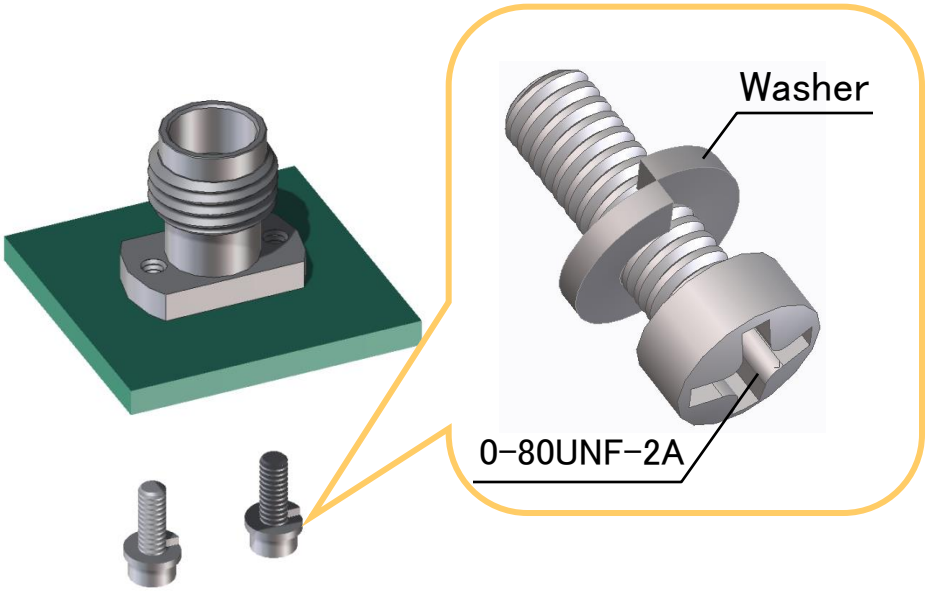
While through hole dip mount type does not accept various PCB thickness, screw compression mount type allows various PCB thickness with one specific connector type.

Flexible PCB thickness!!



NOTE: Accepts Min. 1.6 mm thickness PCB.

◆ Washers to attach by screws

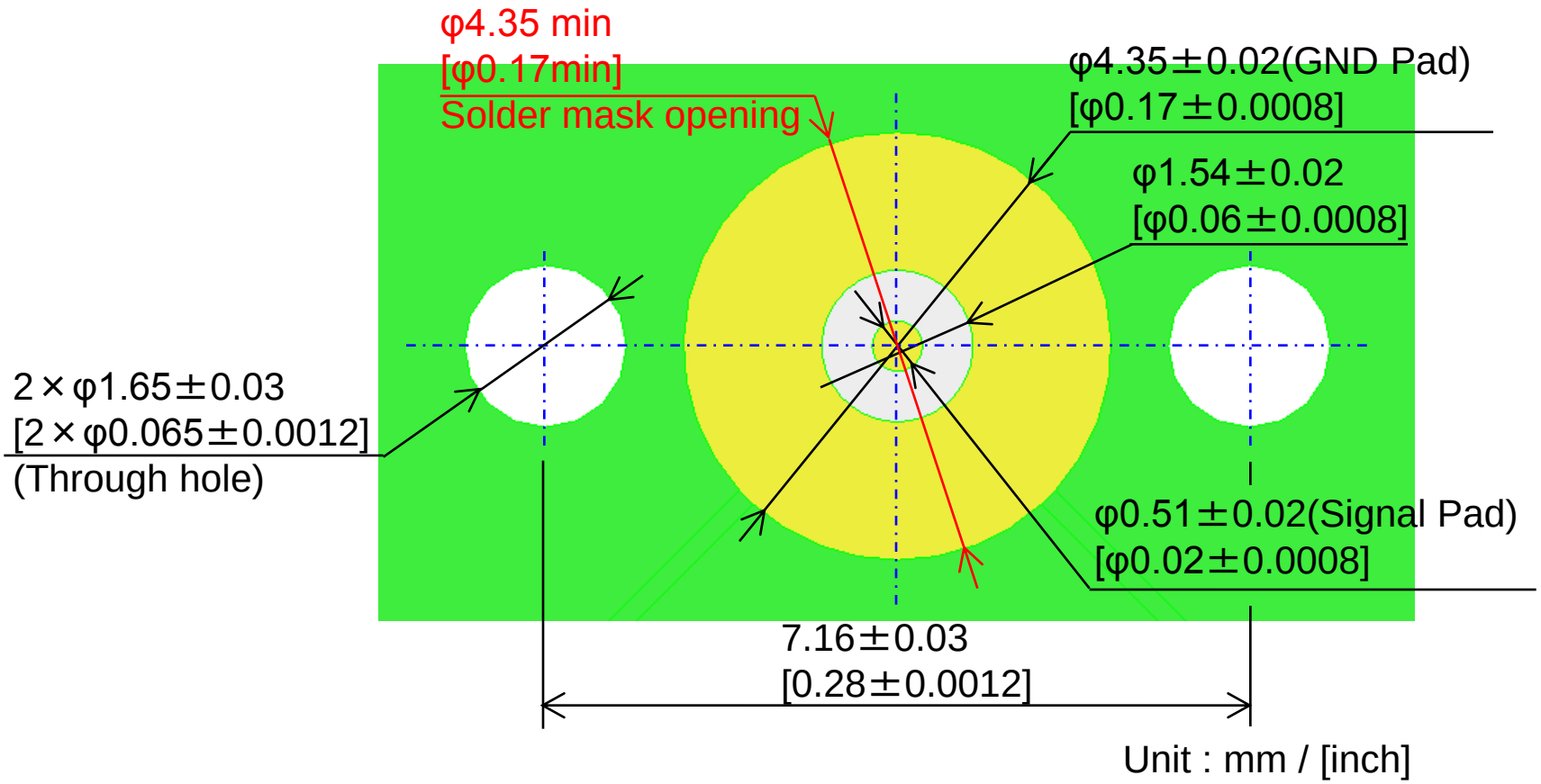


Locking structure !!

Screw mount streamlines assembly process.

Mechanical performance

◆ SMT footprint



Note

1. Please use gold plating on GND pad and signal pad.

Products

Performance of a 2.4mm and 2.92mm connector thru measurements to be shown.

2.4mm
Range: 0 to 50GHz
（H2.4-R-SR2）

2.92mm
Range: 0 to 40GHz
（HK-R-SR2-1）

Specifications

■ Material & Finish

COMPONENT	MATERIAL	FINISH
Shell	Steel stainless	Passivation
Center Conductor	Beryllium copper	Gold plated
Insulator	PTFE resin	-

■ Electrical Performance

	H2.4-R-SR2	**HK-R-SR2-1**
Contact Resistance	4 mΩ max. (Center and Outer) at 100 mA DC	4 mΩ max. (Center and Outer) at 100 mA DC
Withstanding Voltage	500 V AC for 1 minute	500 V AC for 1 minute
Insulation Resistance	5000 MΩ min. at 500 V DC	1000 MΩ min. at 500 V DC
Durability	500 cycles	500 cycles
Characteristic Impedance	50 Ω	50 Ω
V.S.W.R.	1.35 Max. [DC to 26.5 GHz]	1.35 Max. [DC to 26.5 GHz]
	1.40Max. [26.5 GHz to 40 GHz]	1.40Max. [26.5 GHz to 40 GHz]
	1.45Max. [40 GHz to 50 GHz]	

Electrical performance

- **Electrical performance of back-to-back connection**
- **PCB stackup/foot print/routing layer**
- **Electrical performance on PCB**

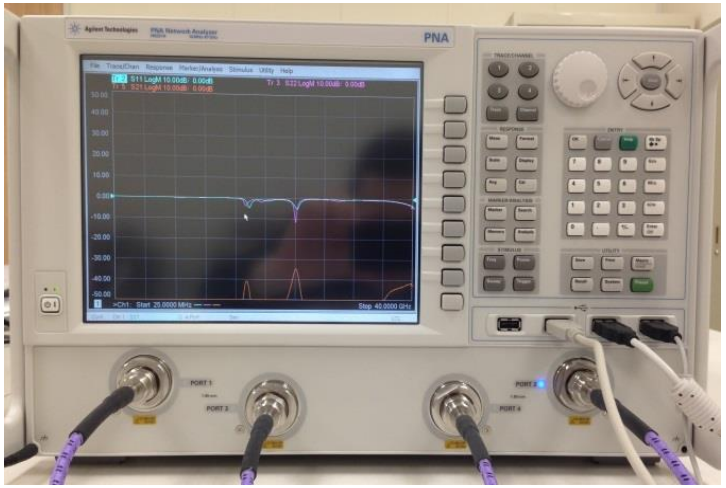
PNA set up

4 port Agilent 5227A PNA

Ecal : Agilent N4694-60003

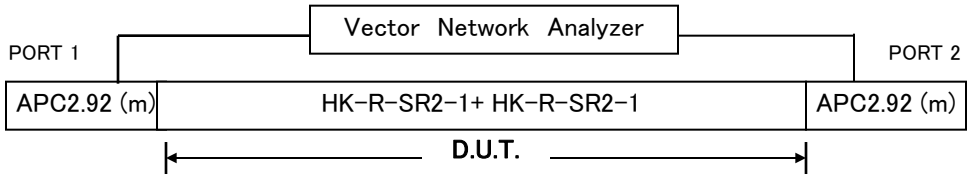
Frequency range : 25MHz – 50GHz / 2000 points

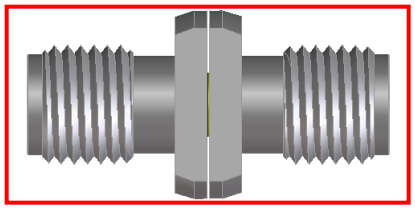
IF band : 300Hz

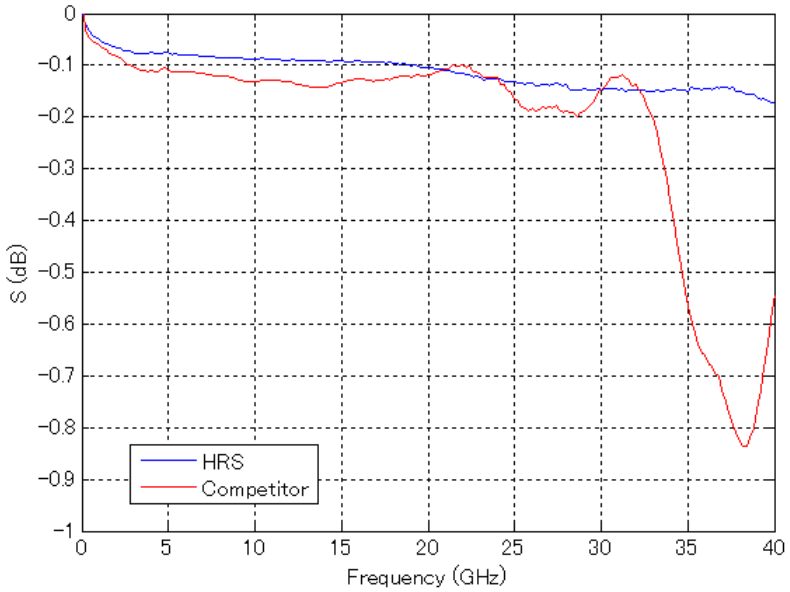


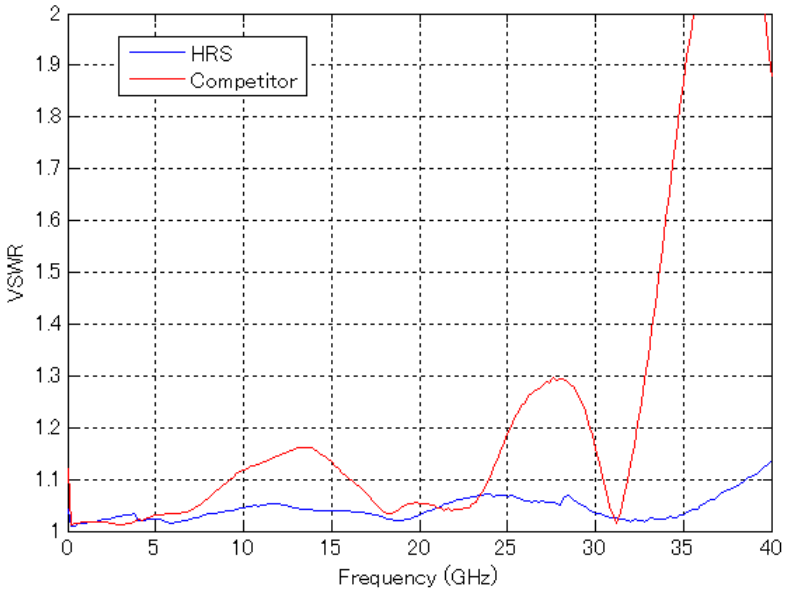
Electrical performance of back-to-back(HK)

◆ Measurement method of back-to-back



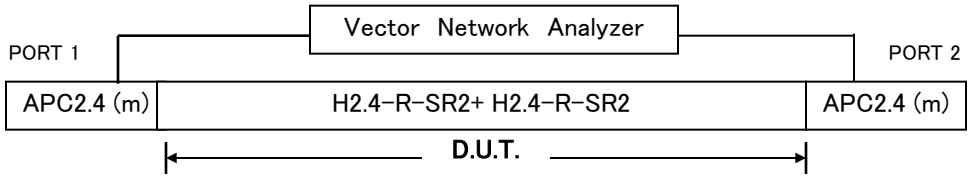


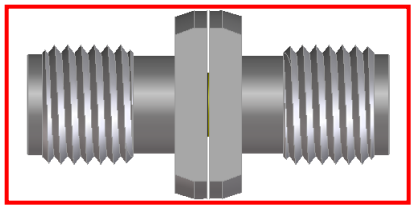


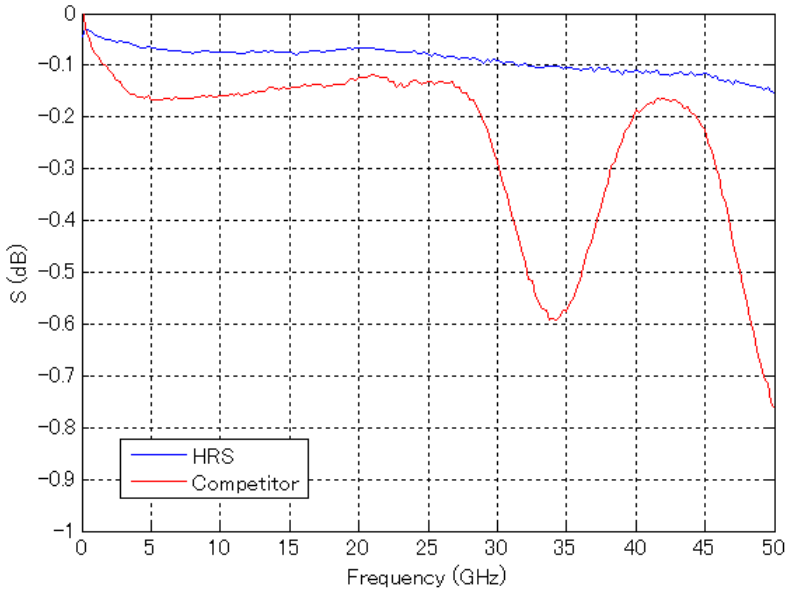


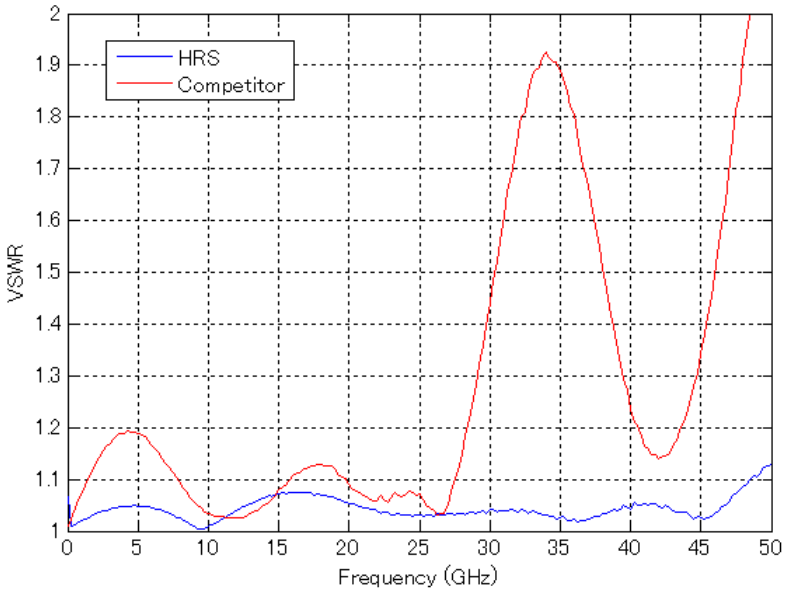
Electrical performance of back-to-back(H2.4)

◆ Measurement method by back-to-back

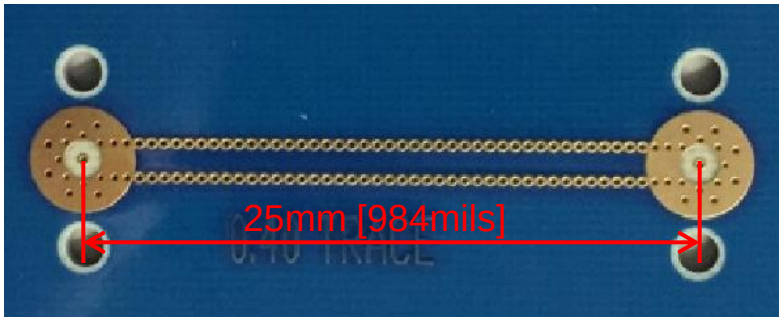




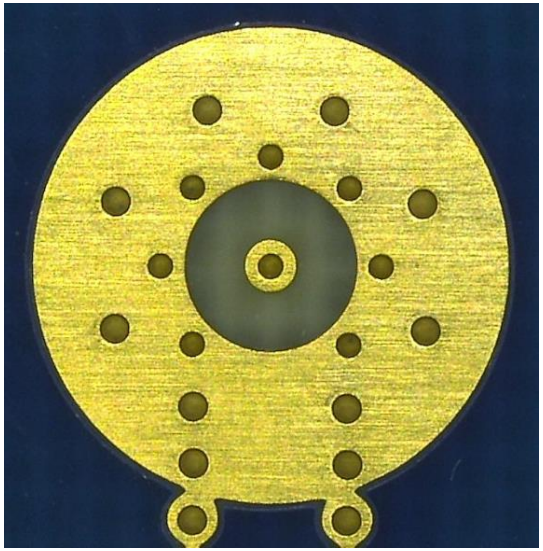




PCB stackup/foot print/routing layer



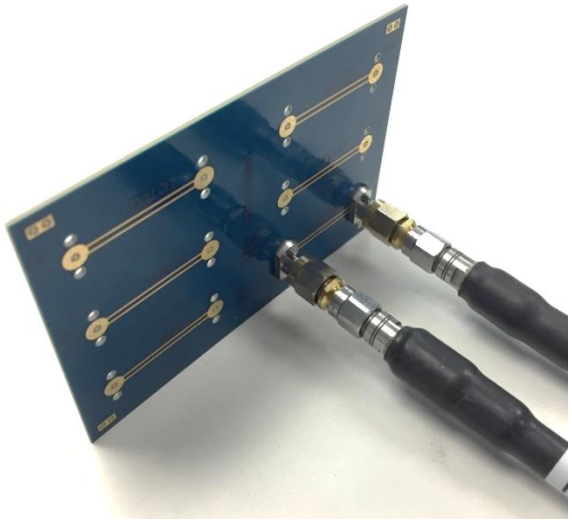
Evaluation PCB

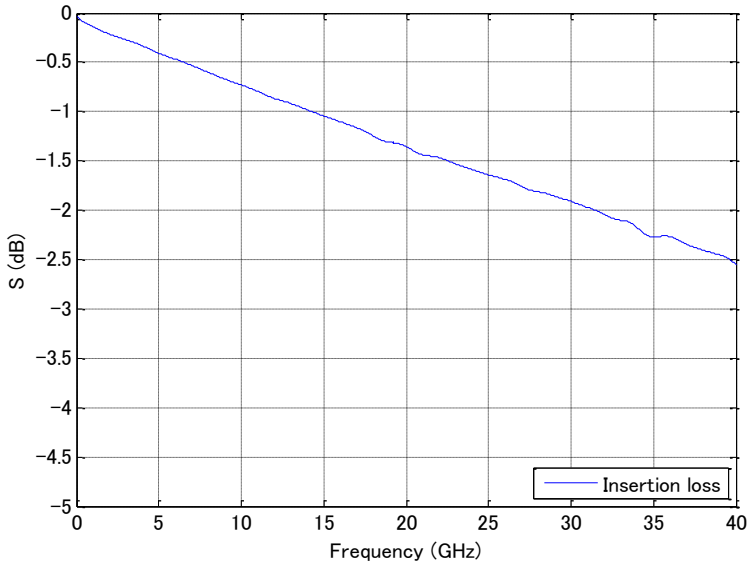


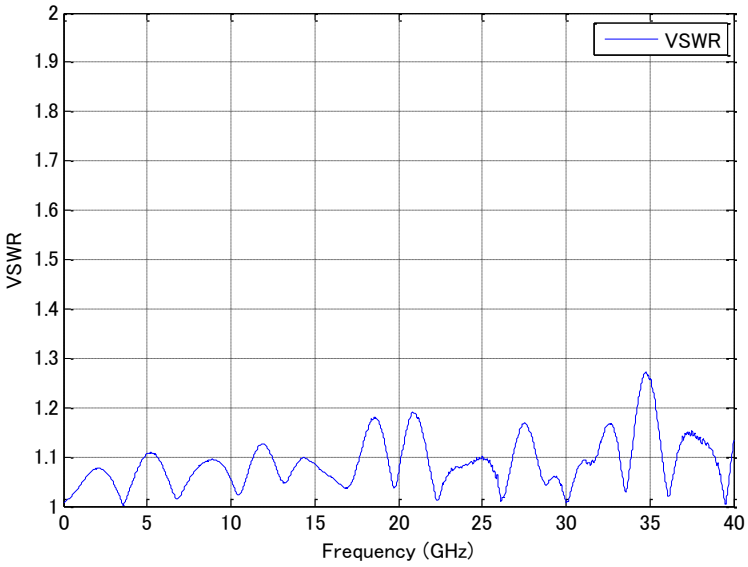
SMT land for coaxial connector

PCB stackup

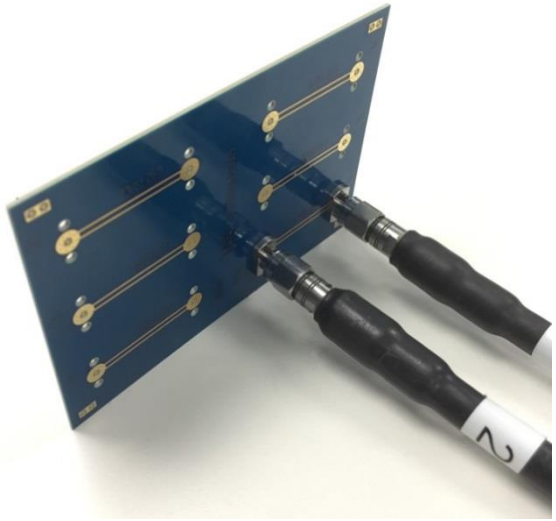
Performance (HK)

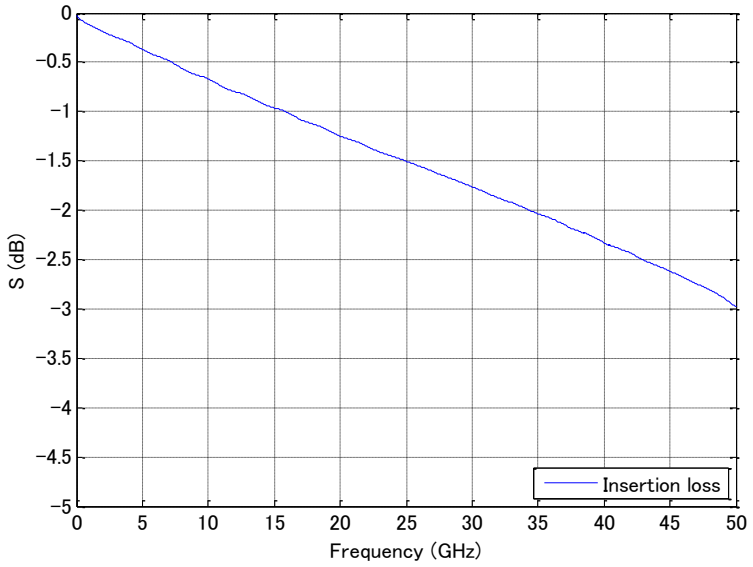


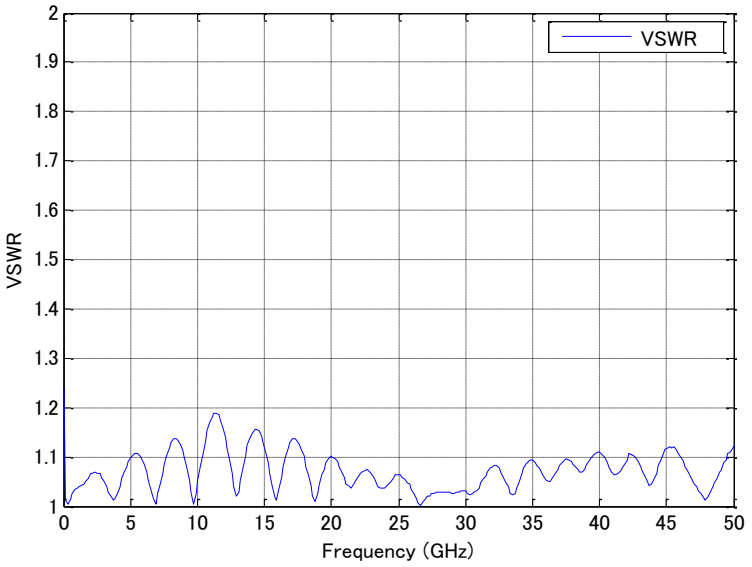




Performance (H2.4)







PCB design considerations

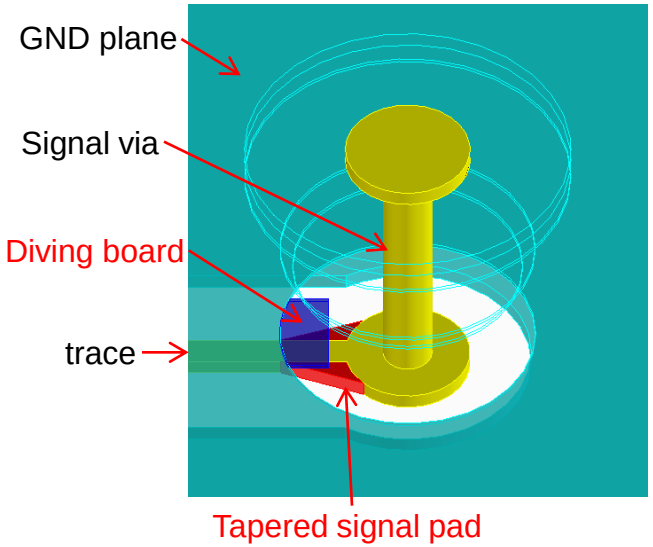
- *Optimized GND via locations for 50GHz*
- *Tapered trace and diving board design*
- *Via stub impact*
- *Consideration of copper wicking*
- **Stitching via space recommendation for CPWG**
- **Stitching via separation recommendation for CPWG**
- *Substrate thickness between 2 GND planes*
- *Antipad optimization(thick PCB)*
- *Antipad optimization(thin PCB)*
- *The effect of glass weave style*
- *Surface roughness of copper foil*
- *Measurement vs. simulation correlation*

Tapered trace and diving board design

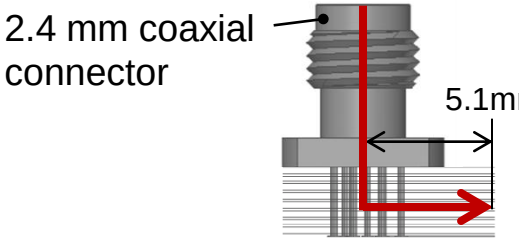
- Tapering signal pad to trace and pushing out GND, referred to as “diving board” design, achieves better impedance matching in the anti pad area and improves insertion loss and return loss beyond 35GHz.

Drill diameter

Signal: 0.25mm [10mils]
Ground: inner ring 0.25mm [10mils]
/outer ring 0.3mm [12mils]

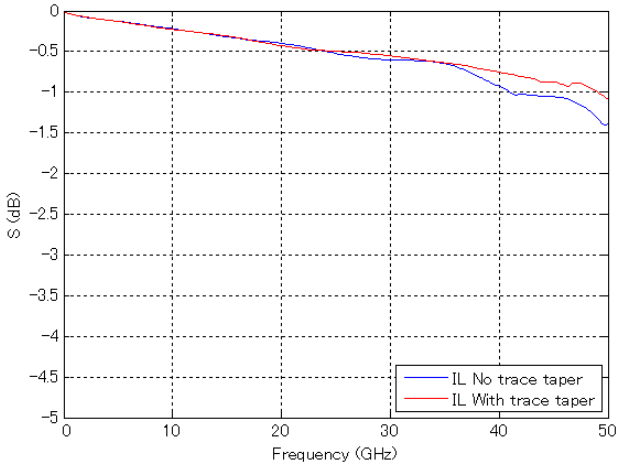


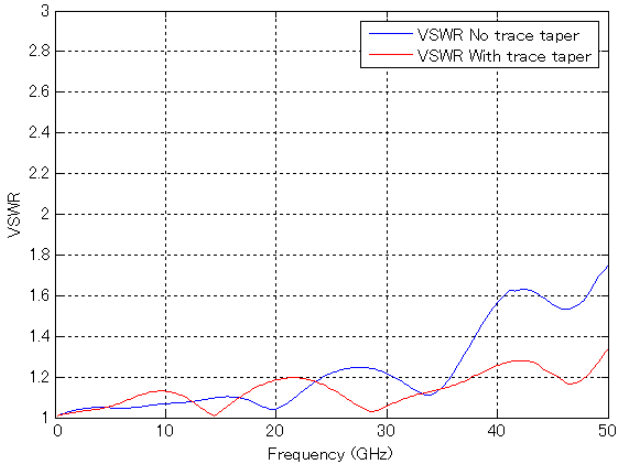
Tapered pad to trace transition
& Diving board structure



Megtron 6 (Dk : 3.6, Df : 0.004)

Simulation





Via stub impact

- For inner layer routing, via stub length shall be minimized to be less than 1.3mm [5 mils] for 50GHz. 0.5mm [20 mil] via stub will cause non-linear effects on insertion loss and return loss.

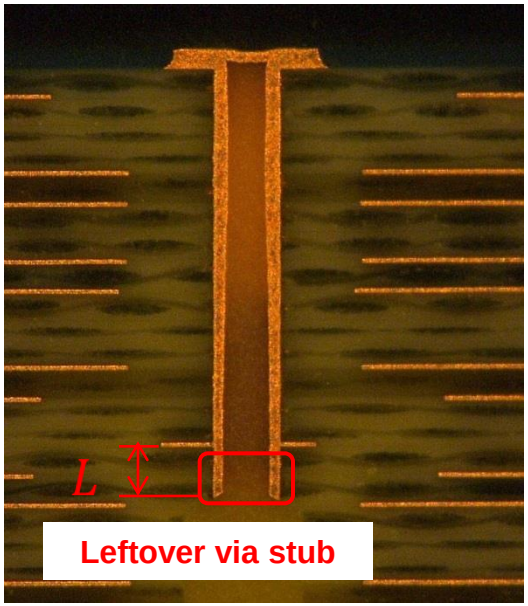
27.4mm [1080 mil] strip line

Drill diameter

Signal: 0.25mm [10mils]

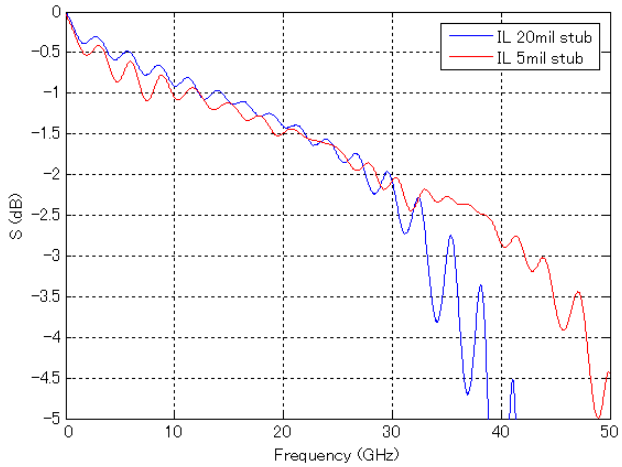
Ground: inner ring 0.25mm [10mils]

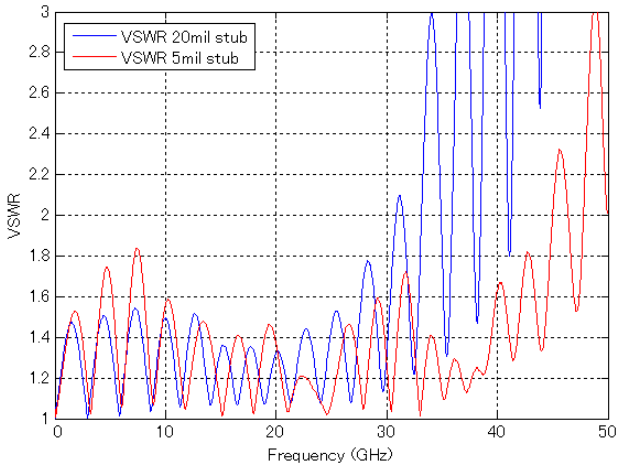
/outer ring 0.3mm [12mils]



Megtron 6 (Dk : 3.6, Df : 0.004)

Measurement





$$L < \frac{\lambda_0}{4\sqrt{\varepsilon_{eff}}}$$

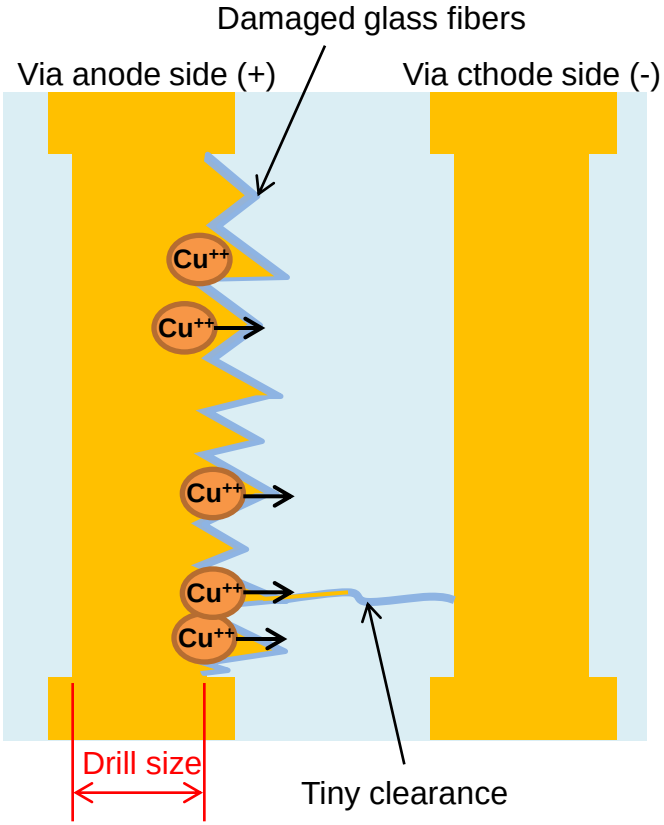
L : Via stub length

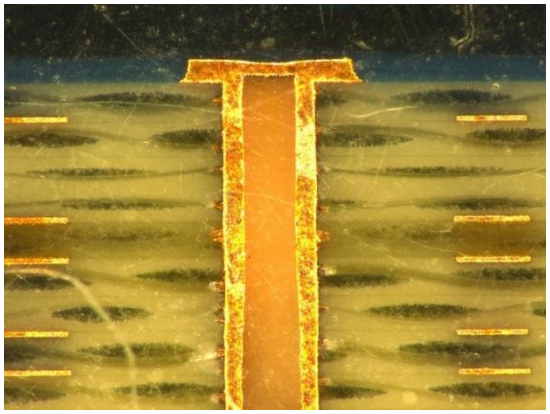
λ_0 : Wave length of electromagnetic wave in free space.

ε_{eff} : Effective dielectric constant.

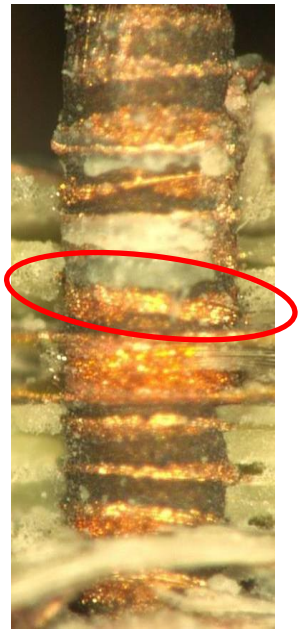
Consideration of copper wicking

- When substrate is drilled, damaged glass fibers and separation in the glass fibers to the resin leave holes. Because of migration of copper salt into the glass fibers of insulation material, these holes will be filled with copper during the electro-plating process. It will cause a discrepancy between design and actual PCB.
  To compensate for copper wicking, 5% to 10% of diameter is added to diameter of signal via model.





Cross section of via

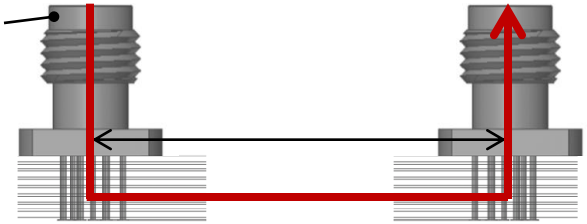


Wicking example

Stitching via space recommendation for CPWG

◆ For CPWG structures, stitching ground vias with 0.5mm [20 mil] space on both sides of the CPWG micro-strip trace are required to prevent the grounds on both sides from resonating.

2.4 mm coaxial connector



27.4mm [1080 mil] strip line

Drill diameter

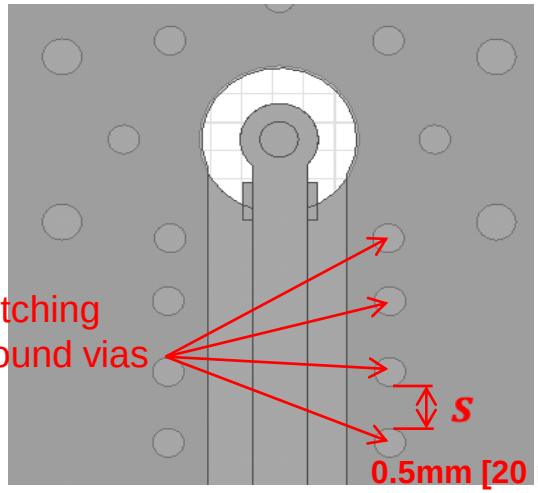
Signal: 0.25mm [10mils]

Ground: inner ring 0.25mm [10mils]

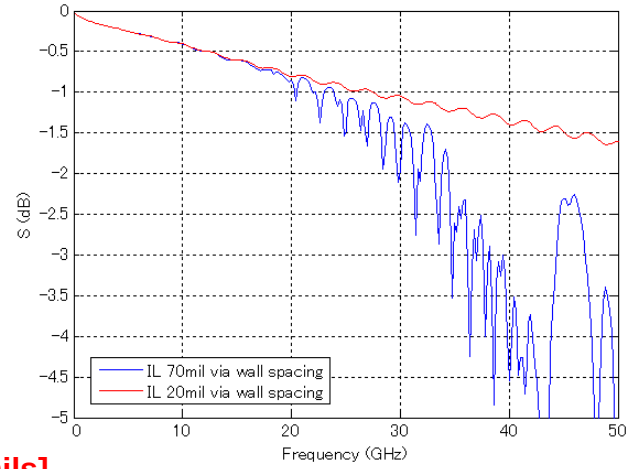
/outer ring 0.3mm [12mils]

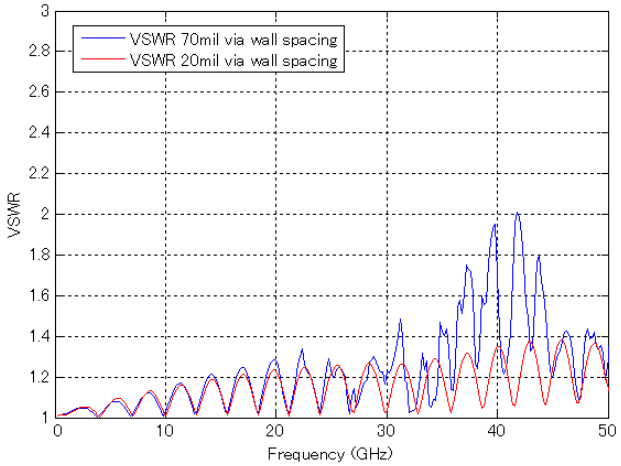
Megtron 6 (Dk : 3.6, Df : 0.004)

Measurement



0.5mm [20 mils] / 1.78mm [70 mils]





$$s < \frac{\lambda_0}{4\sqrt{\varepsilon_{eff}}}$$

s : Via space

λ_0 : Wave length of electromagnetic wave in free space.

ε_{eff} : Effective dielectric constant.

Stitching via separation recommendation for CPWG

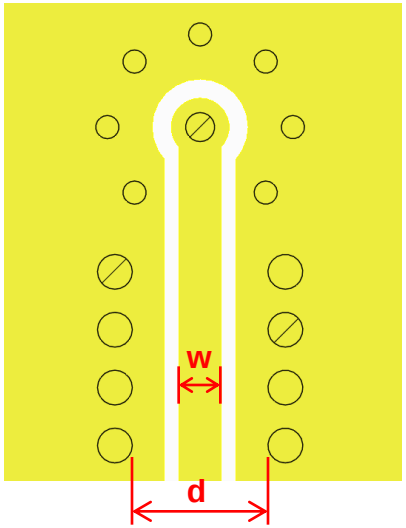
- The via separation should be a minimum of 3 strip widths. When vias are placed too close to edge of trace, they affect impedance. At the same time, if the via separation is too large, unwanted propagation modes can be excited, which affect performance.

Drill diameter

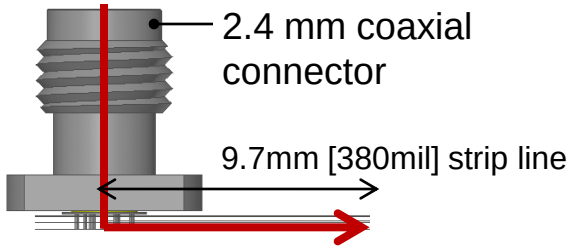
Signal: 0.25mm [10mils]

Ground: inner ring 0.25mm [10mils]

/outer ring 0.3mm [12mils]

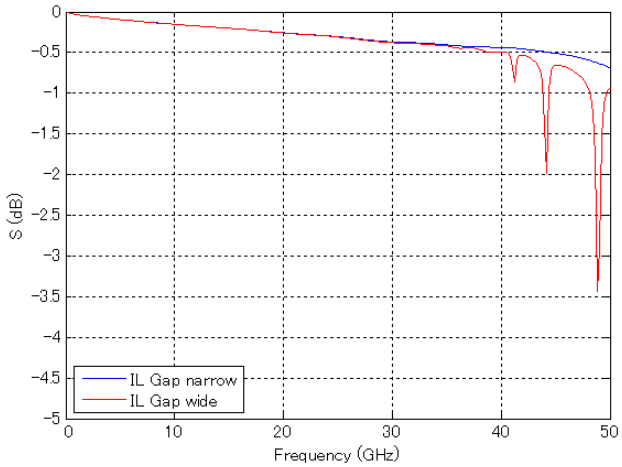


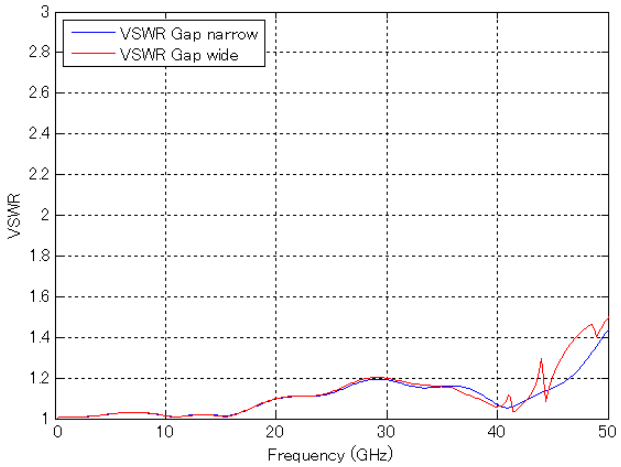
1.17mm [46 mils]
/ 2.17mm [85 mils]



RO4350B (Dk : 3.66, Df : 0.004)

Simulation





$$d > 3w$$

$$d < \frac{\lambda_0}{2\sqrt{\varepsilon_{eff}}}$$

d : Via separation

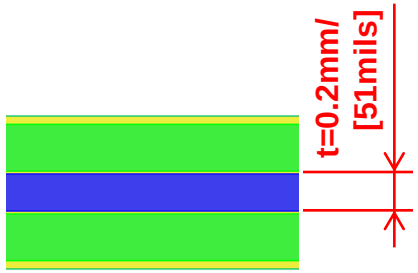
w : Trace width

λ_0 : Wave length of electromagnetic wave in free space.

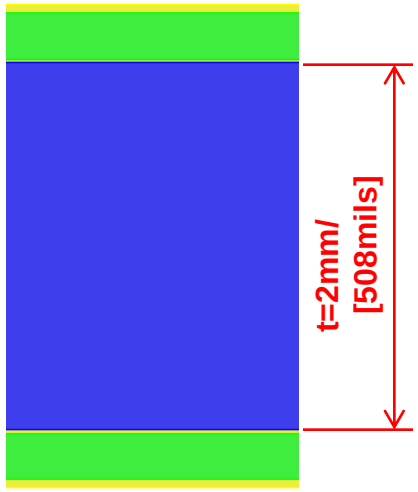
ε_{eff} : Effective dielectric constant.

Substrate thickness between 2 GND planes

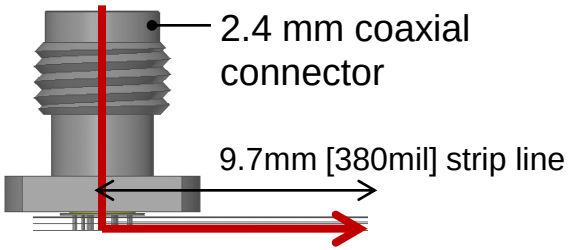
◆If substrate thickness is too large, unwanted spurious wave propagation can occur. It can interfere with the desired wave on the circuit.



Thin PCB substrate

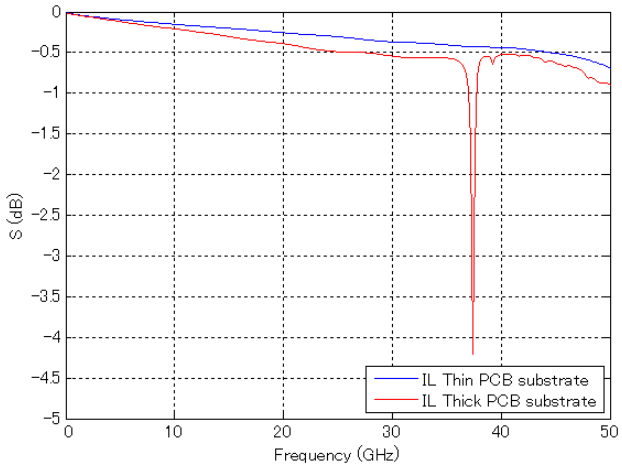


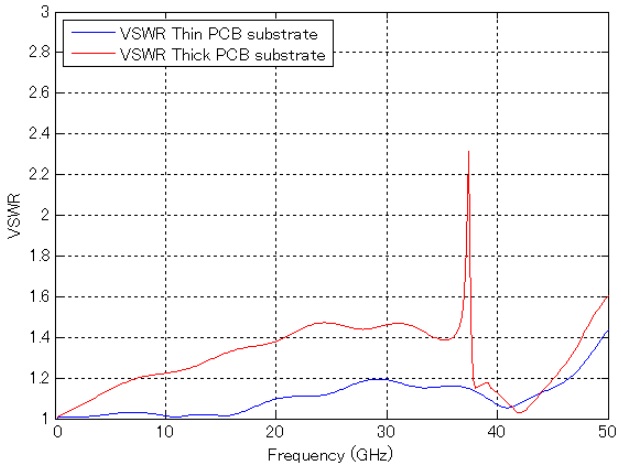
Thick PCB substrate



RO4350B (Dk : 3.66, Df : 0.004)

Simulation



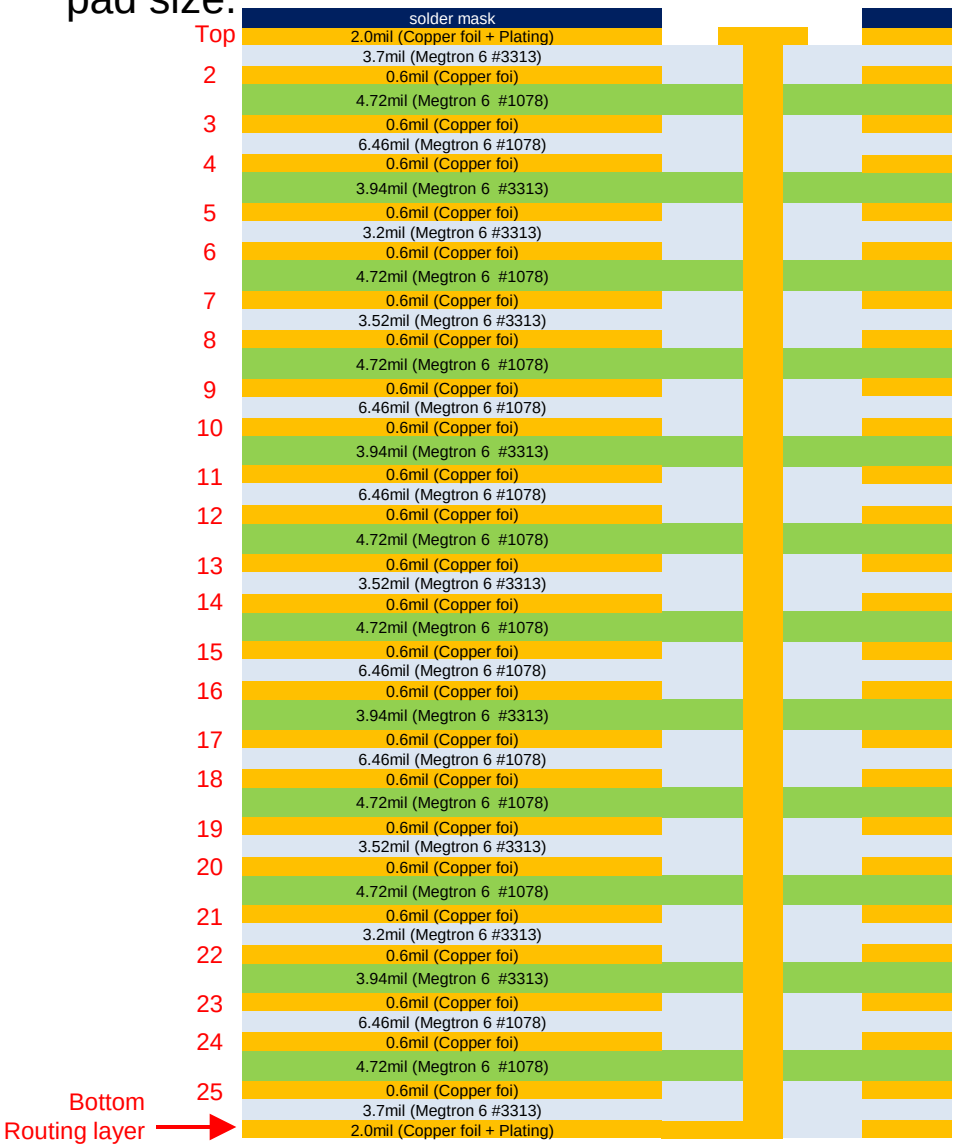


$$t < \frac{\lambda_0}{2\sqrt{\varepsilon_{eff}}}$$

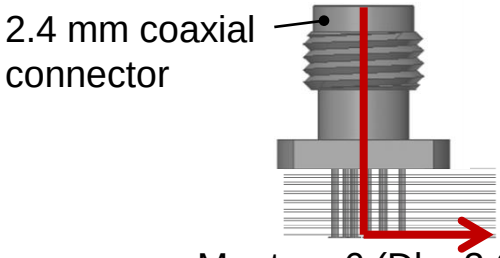
t : Substrate thickness
w : Trace width
λ_0 : Wave length of electromagnetic wave in free space.
ε_{eff} : Effective dielectric constant.

Anti-pad optimization (thick PCB)

- Antipad size is optimized at every layer by considering dielectric layer thickness and via pad size.



PCB stackup



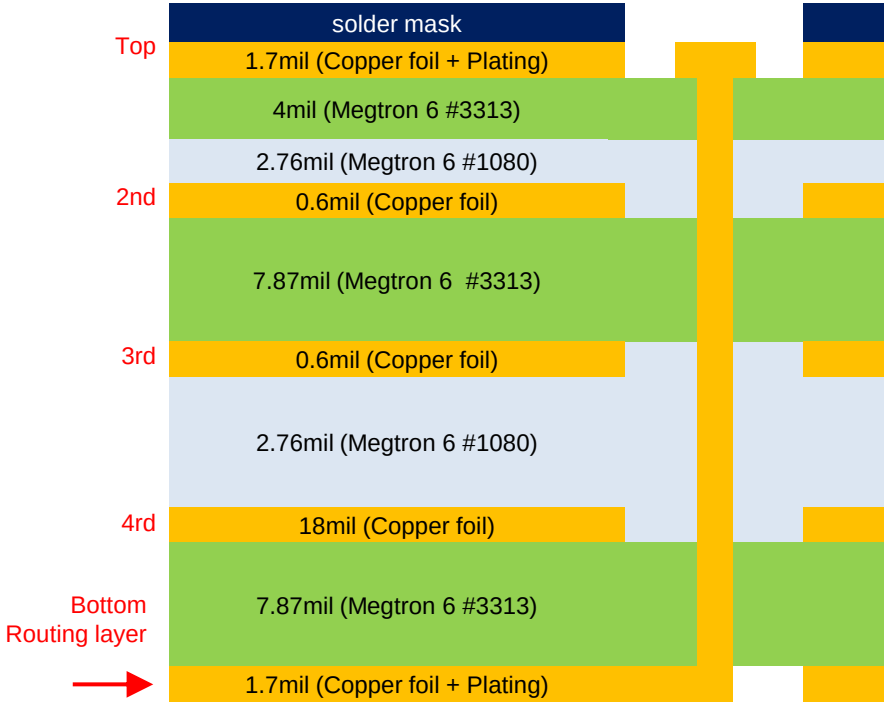
Megtron 6 (Dk : 3.6, Df : 0.004)

Layer	Anti-pad dia.
Top	1.5mm / [59 mils]
2	1.5mm / [59 mils]
Layer 3-24	0.94mm / [37 mils]
Layer 25	1mm / [40 mils]
Bottom	Microstrip line out

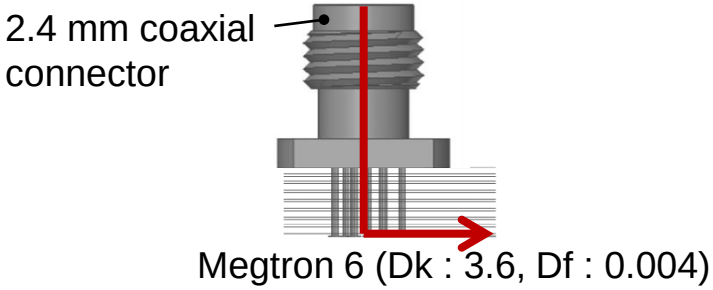
PTH anti-pad dimensions

Anti-pad optimization (thin PCB)

- Antipad size is optimized at every layer by considering dielectric layer thickness and via pad size.



PCB stackup

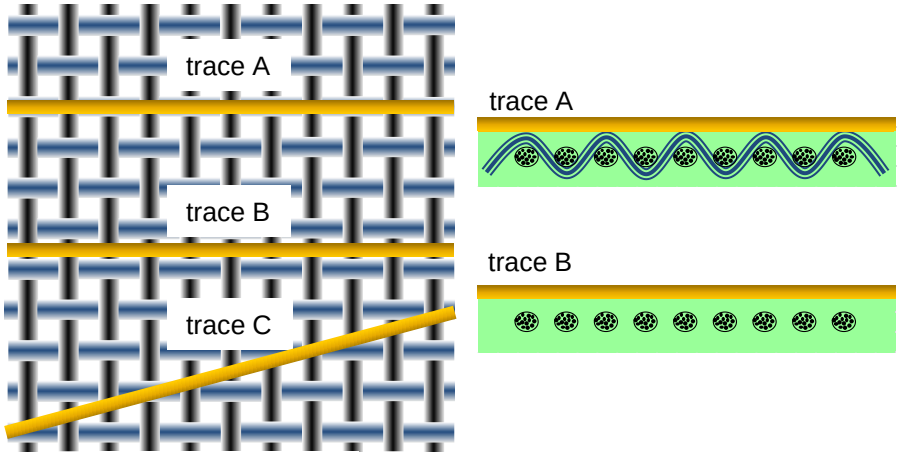


Layer	Anti-pad dia.
Top	1.54mm/ [60.6mils]
2	1.15mm/ [45.3mils]
3	0.97mm/ [38.2mils]
4	1.09mm/ [42.9mils]
Bottom	1.02mm/ [40.2mils]

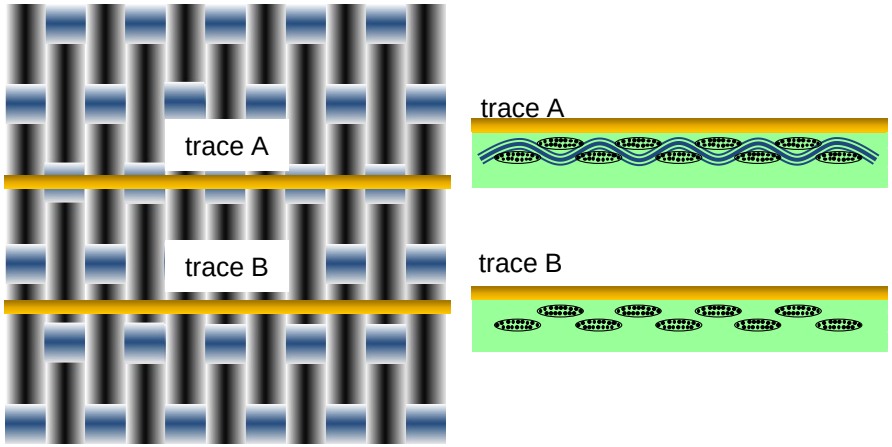
PTH anti-pad dimensions

The effect of Glass Weave style

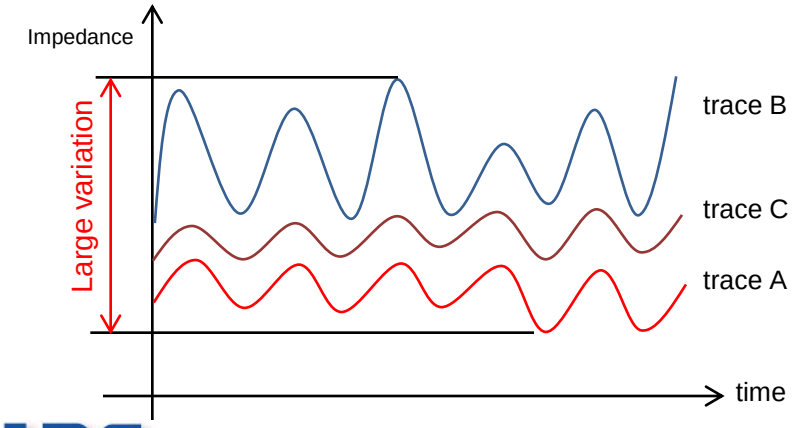
- Uneven glass cloth causes a large impedance variation and skew due to unevenness of dielectric constant.
  To use flat type glass cloth or to route trace tilting 10 to 20 degrees against glass weave orientation brings more stable performance.

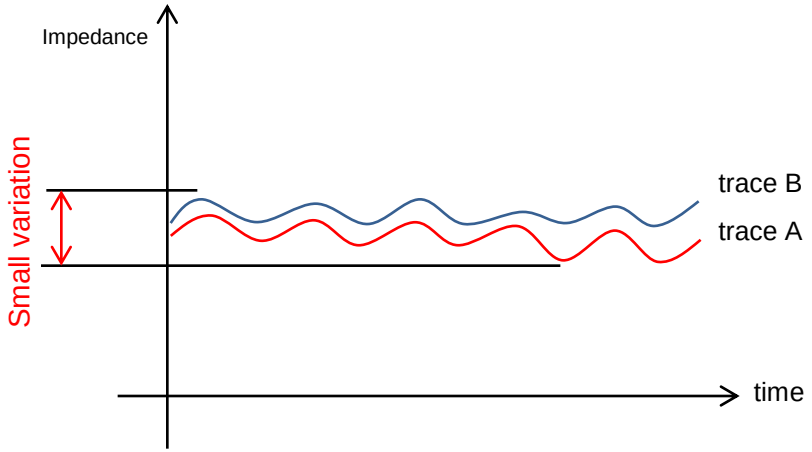


Normal glass cloth



Flat type glass cloth





Typical glass weave styles

Style	Fabric Count Warp x Fill (Per cm)	Yarn (SI)	Thickness (mm) (Reference Only)	Nominal Weight (g/m2)	Weight Tolerance (g/m2)
101	29.5 x 29.5	5 2.75 1x0 5 2.75 1x0	0.024	16.3	15.2 - 17.3 1
104	23.6 x 20.5	5 5.5 1x0 5 2.75 1x0	0.028	18.6	18.0 - 19.3 1
106	22.0 x 22.0	5 5.5 1x0 5 5.5 1x0	0.033	24.4	23.4 - 25.4 1
1078	21.3 x 21.3	5 11 1x0 5 11 1x0	0.043	47.8	46.8 - 49.2 1
1080	23.6 x 18.5	5 11 1x0 5 11 1x0	0.053	46.8	45.1 - 48.5 1
1081	27.6 x 23.6	5 11 1x0 5 11 1x0	0.06	58.3	56.4 - 60.6 1
1280	23.6 x 23.6	5 11 1x0 5 11 1x0	0.056	52.9	51.5 - 54.2 1
1500	19.3 x 16.5	7 45 1x0 7 45 1x0	0.149	164.1	157.7 - 170.5 1
1501	18.1 x 17.7	7 45 1x0 7 45 1x0	0.14	165	158.0 - 171.0 1
1504	23.6 x 19.7	6 33 1x0 6 33 1x0	0.125	148	142.8 - 153.2 1
1651	20.0 x 10.8	9 33 1x0 9 74 1x0	0.135	146.2	142.1 - 150.3 1
1652	20.5 x 20.5	9 34 1x0 9 34 1x0	0.114	138.3	133.6 - 143.1 1
1674	15.7 x 12.6	9 34 1x0 9 34 1x0	0.097	96.6	92.9 - 100.4 1
1675	15.7 x 12.6	6 33 1x0 6 33 1x0	0.101	96.3	92.6 - 100.0 1
1678	15.7 x 15.7	9 34 1x0 9 34 1x0	0.091	103.5	102.7 - 111.6 1
2113	23.6 x 22.0	7 22 1x0 5 11 1x0	0.079	78	75.6 - 80.4 1
2114	22.0 x 18.9	7 22 1x0 7 22 1x0	0.084	90.9	88.5 - 93.2 1
2116	23.6 x 22.8	7 22 1x0 7 22 1x0	0.094	103.8	100.7 - 106.8 1
2117	26.0 x 21.7	7 22 1x0 7 22 1x0	0.095	108	104.8 - 111.2 1
2125	15.7 x 15.4	7 22 1x0 9 34 1x0	0.091	87.5	82.7 - 90.9 1
2157	23.6 x 13.8	7 22 1x0 9 68 1X0	0.13	148	144.0 - 152.0 1
2165	23.6 x 20.5	7 22 1x0 9 34 1x0	0.101	122.4	116.3 - 126.1 1
2166	23.6 x 15.0	7 22 1x0 9 68 1X0	0.14	155	150.0 - 160.0 1
2313	23.6 x 25.2	7 22 1x0 5 11 1x0	0.084	81.4	79.0 - 83.7 1
3070	27.6 x 27.6	6 16.5 1x0 6 16.5 1x0	0.078	93.6	90.9 - 96.3 1
3080	20.0 x 12.0	6 16.5 1x0 6 16.5 1x0	0.059	53.4	51.5 - 55.3 1
3313	23.6 x 24.4	6 16.5 1x0 6 16.5 1x0	0.084	81.4	79.0 - 83.7 1
7628	17.3 x 12.2	9 68 1x0 9 68 1x0	0.173	203.4	198.0 - 208.9 1
7629	17.3 x 13.4	9 68 1x0 9 68 1x0	0.18	210	204.5 - 215.3 1
7635	17.3 x 11.4	9 68 1x0 9 102 1x0	0.201	232.3	226.5 - 238.0 1
7642	17.3 x 7.9 (texturized)	9 68 1x0 9 136 1x0	0.254	227.8	221.1 - 234.7 1

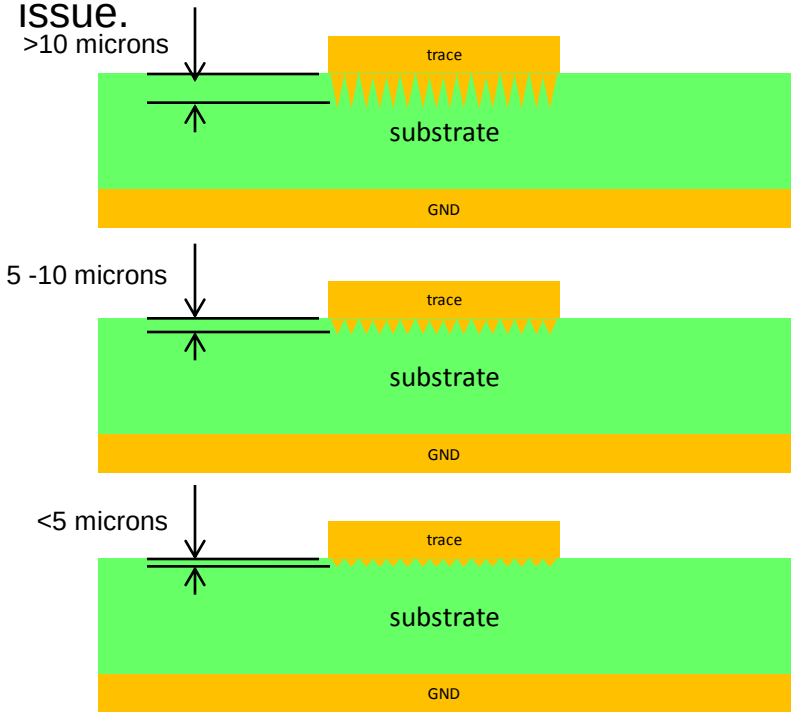
Good flat type is available

Flat type is available

※Quotation from IPC-4412

Surface roughness of copper foil

◆ At high frequencies, rough copper foil will cause large attenuation. Choosing smoother copper foil will provide improved quality on high frequency transmission line. However, smoother copper foil has less peel strength between substrate and copper. So, it is important to choose suitable copper foil by balancing mechanical issue and electrical issue.



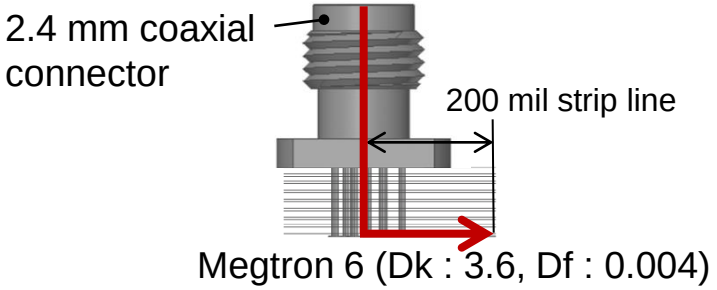
- Standard foil
  ED：Electrodeposited
  HTE：High Tensile Elongation
- Low profile
  RTF：Reverse Treated Foil
  VLP：Very Low Profile
- Very low profile
  e-VPL：Extra Very Low Profile
  H-VLP：Hyper Very Low Profile

Copper foil type	Peel strength	Stable impedance	Attenuation	Cost
Standard foil				
Low profile				
Very low profile				

Measurement vs. simulation correlation

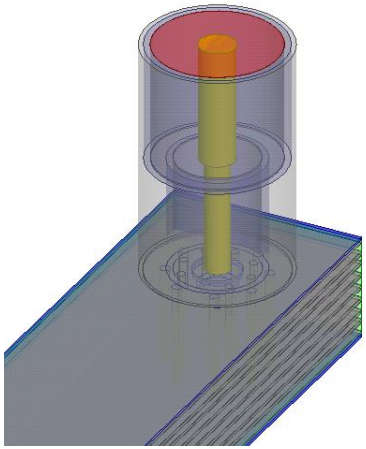
◆ A PTH to bottom micro-strip transition structure was examined.

Drill diameter
Signal: 10mils
Ground: 10mils
Signal pad diameter
Inner layer: 18mils
Outer layer: 20mils

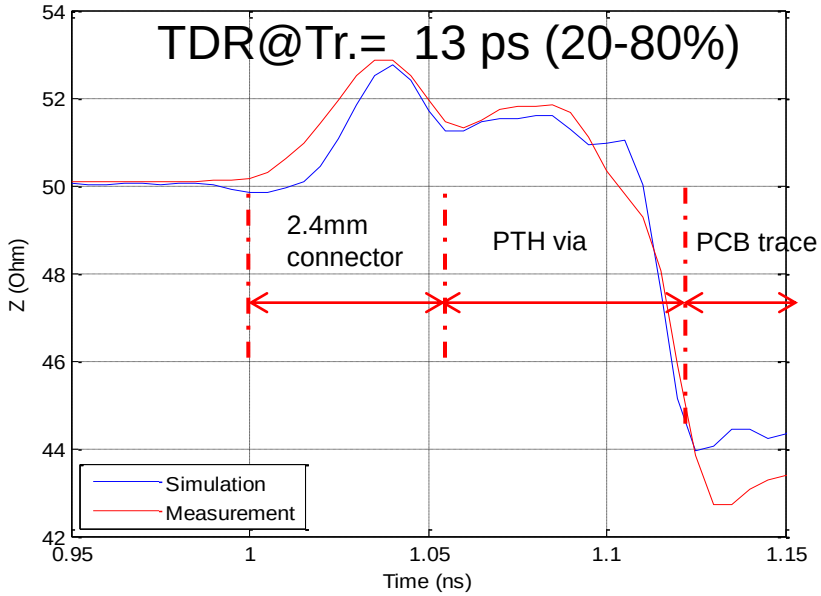


Megtron 6 (Dk : 3.6, Df : 0.004)

Test sample



Simulation model

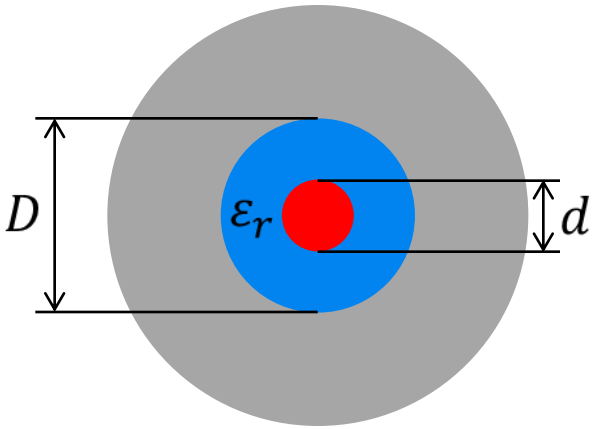


Appendix

- ***Cutoff frequency***
- ***2.4mm, 2.92mm comparison***
- ***Conversion of VSWR to Return loss***

Cutoff frequency

◆ Connector species have their own frequency limitation, which is determined by cutoff frequency. When frequency exceeds cutoff frequency, unwanted propagation mode (TE mode) will excite. This mode will degrade loss and VSWR.



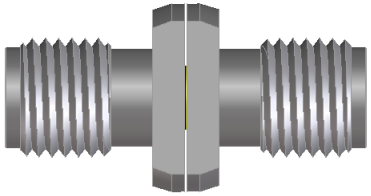
$$f_c = \frac{c}{\pi\sqrt{\varepsilon_r}\frac{D+d}{2}}$$

f_c:Cutoff frequency
ε_r:Dielectric constant
D:outer diameter
d:inner diameter
c:light speed

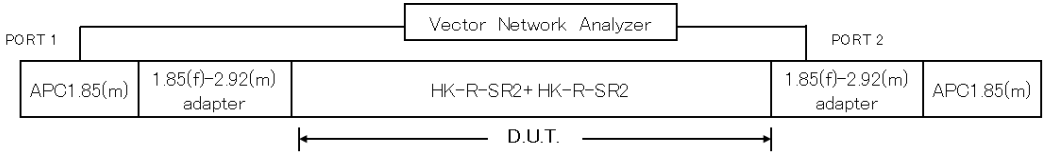
Connector species	Calculated cutoff frequency [GHz]	Frequency limit [GHz]
3.5mm	38.0	26.5
2.92mm	45.6	40
2.4mm	55.4	50
1.85mm	71.9	60
1mm	133	110

Representative connector species & frequency limit

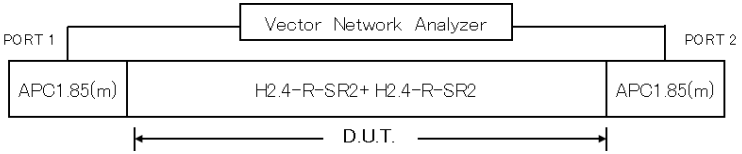
2.4mm,2.92mm comparison



Back to back measurement

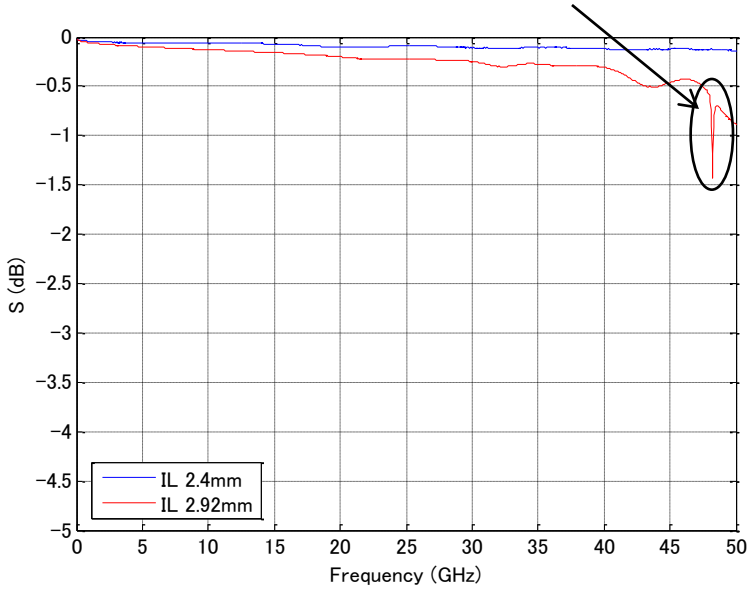


2.92mm measurement

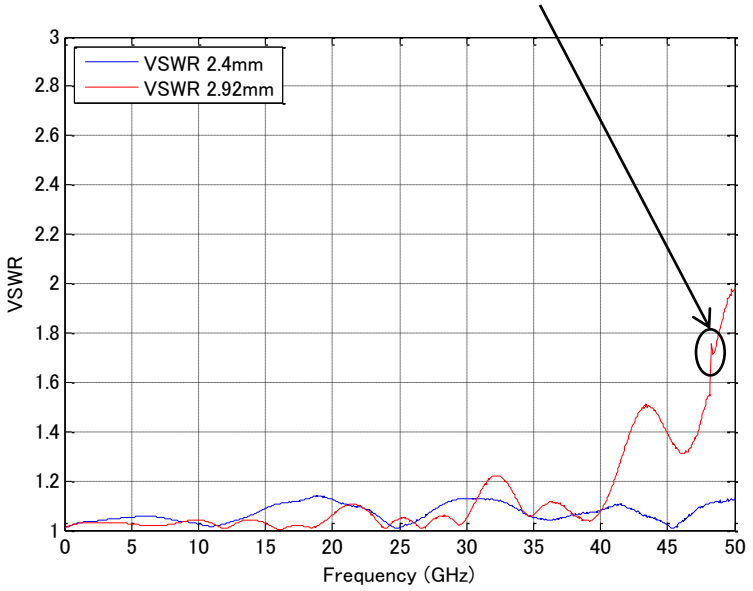


2.4mm measurement

Degradation due to unwanted mode



Degradation due to unwanted mode



Conversion of VSWR and Return loss

VSWR to Return loss

VSWR	Return loss [dB]
1	-
1.05	-32.26
1.1	-26.44
1.2	-20.83
1.3	-17.69
1.4	-15.56
1.5	-13.98
1.6	-12.74
1.7	-11.73
1.8	-10.88
1.9	-10.16
2	-9.54
2.5	-7.36
3	-6.02
3.5	-5.11
4	-4.44
4.5	-3.93
5	-3.52

Return loss to VSWR

Return loss [dB]	VSWR
-40	1.02
-30	1.07
-20	1.22
-15	1.43
-10	1.92
-5	3.57

- $\mathrm{VSWR} = \dfrac{10^{-\frac{\text{Return loss}}{20}} + 1}{10^{-\frac{\text{Return loss}}{20}} - 1}$

- $\text{Return loss} = -20 \log_{10}\left(\dfrac{VSWR+1}{VSWR-1}\right)$ [dB]