TAOGLAS

Datasheet

169 MHz VHF Chip Monopole Antenna

Part No:
CA.69

Description:

25.2*5.1*0.8mm CA.69 169MHz 7dBi Chip Antenna

Features:

169 MHz VHF Chip Monopole Antenna
Peak Gain – approx. -7 dBi (on evaluation board)
Efficiency 10~15% (on evaluation board)
Low Profile
Dimensions: 25.2*5.1*0.8mm
RoHS & REACH Compliant

1. Introduction

The CA.69 Chip Antenna from Taoglas, 169 MHz is specifically designed for VHF 169MHz band applications. It is a high efficiency miniature SMD edge mounted antenna with small footprint requirement. This chip antenna uses the main PCB as its ground plane, thereby increasing antenna efficiency. It is tuned for different PCB sizes by simply changing the value of the matching circuit. CA.69 antenna electrical properties are symmetrical therefore the antenna can be soldered to the board from either side.

This antenna is delivered on tape and reel. Small low frequency antennas such as CA.69 need to be carefully tuned and integrated into devices to perform optimally given the narrow band tuning required, so contact your regional Taoglas sales office for support on gerber review of your layout, advice on ground-plane layout and transmission line design. Taoglas also recommends we test your final device prototype with CA.69 on board and provide final matching values.

Taoglas has tested the CA.69 mounted in realistic conditions in metal or semi metal meter housings with the latest high power modules from Telit and achieved read ranges of more than one hundred metres.

Applications:

- VHF Band Applications

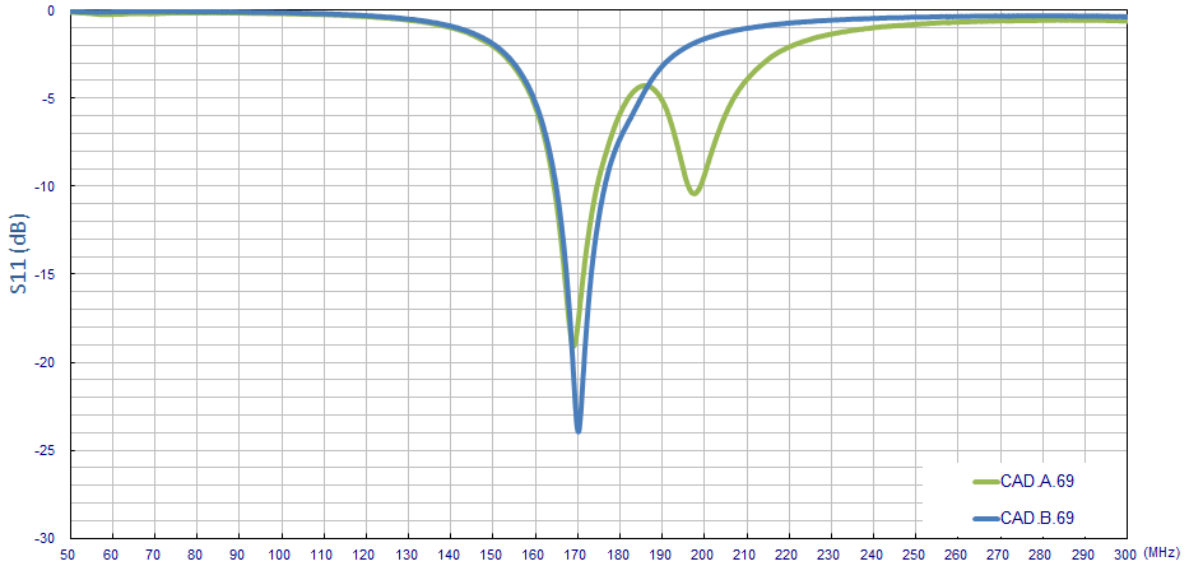
For further optimization to customer-specific device environments and for support to integrate and test this antennas performance in your device, contact your regional Taoglas Customer Services Team.

2. Specifications

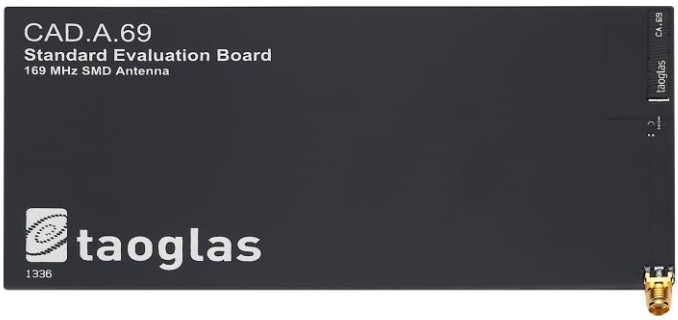
Antenna	
Frequency (MHz)	169 MHz
Bandwidth (MHz)	8 (under -10dB Return Loss)
VSWR	2 max.
Impedance (Ω)	50Ω
Polarization	Linear
Radiation Pattern	Omni
Mechanical	
Dimensions (mm)	25.2 x 5.1 x 0.8
Ground plane (mm)	110 x 55 (Recommended)
Material	FR4
Mechanical	
Temperature Range	-40°C to 85°C
Humidity	20% to 70% RH

3. Antenna Characteristics

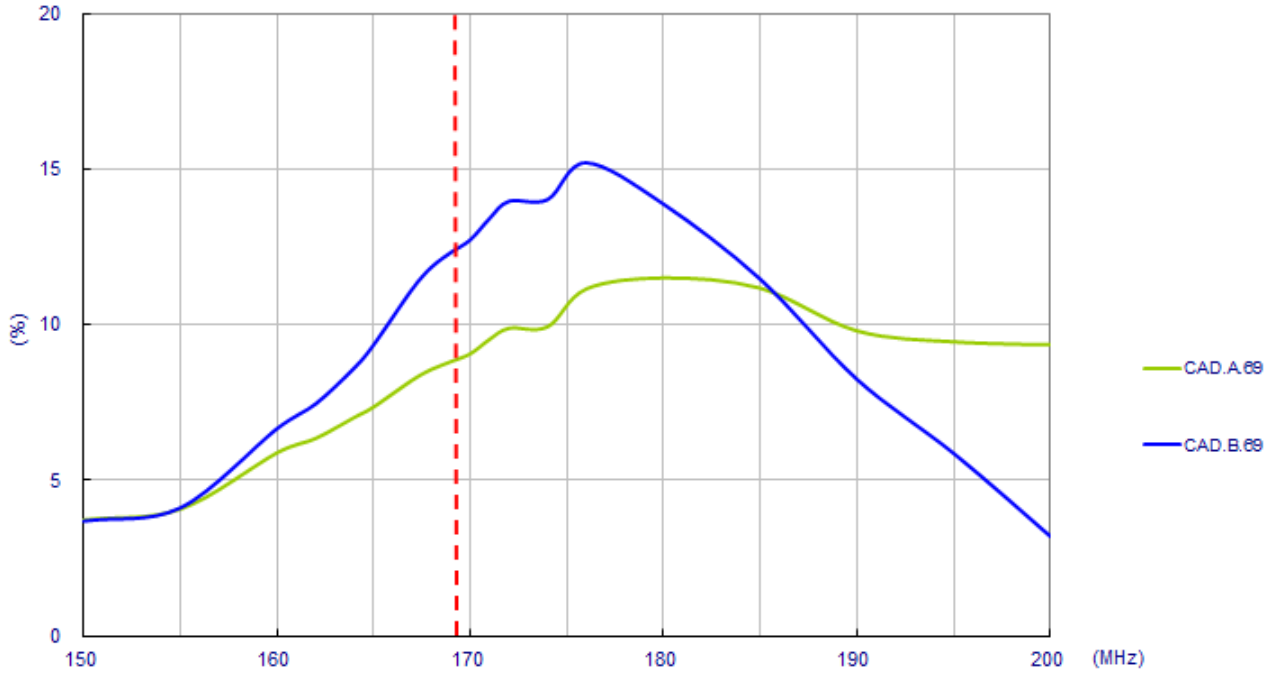
3.1 Return Loss



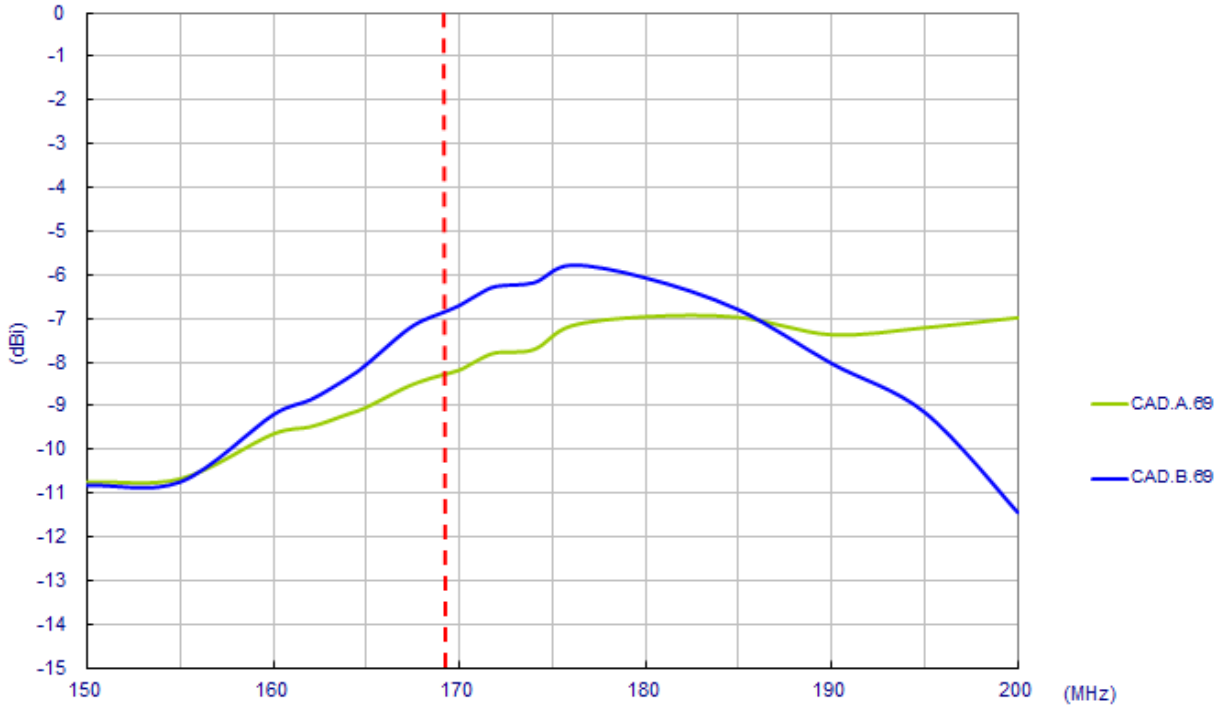
*The antenna tuning depends on different antenna ground plane application. Taoglas provides CAD.A.69 and CAD.B.69 evaluation boards to show performance when antenna is parallel mounted to the ground plane or when it is orthogonally mounted to the ground-plane.



3.2 Efficiency



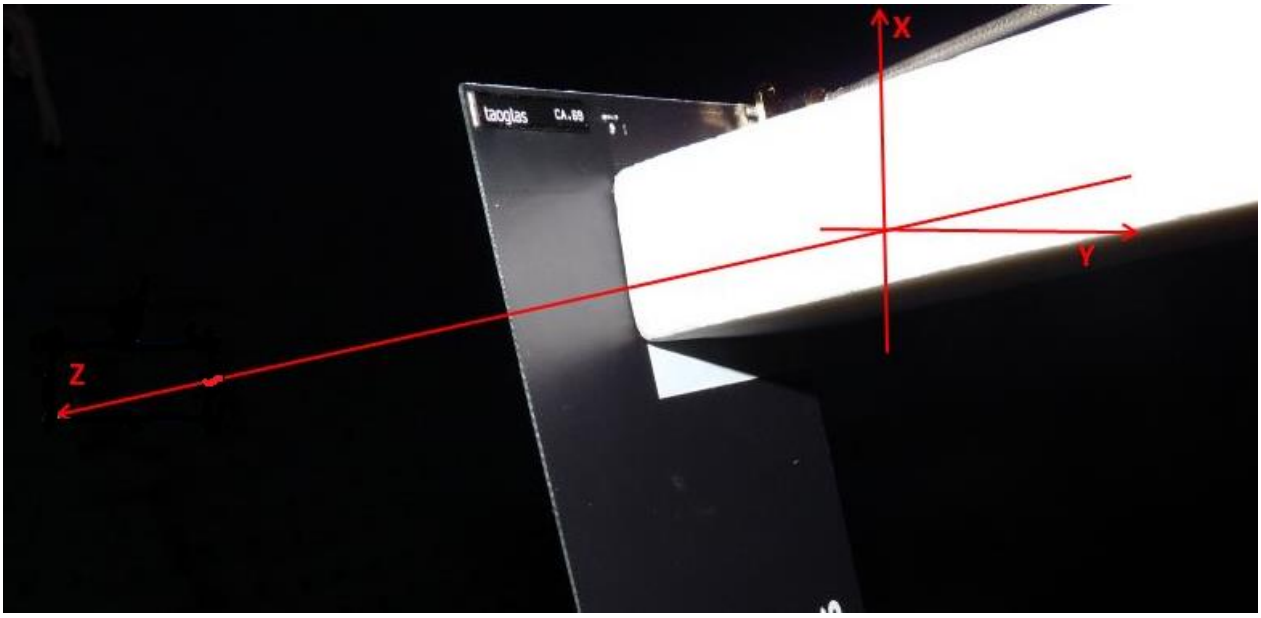
3.3 Peak Gain



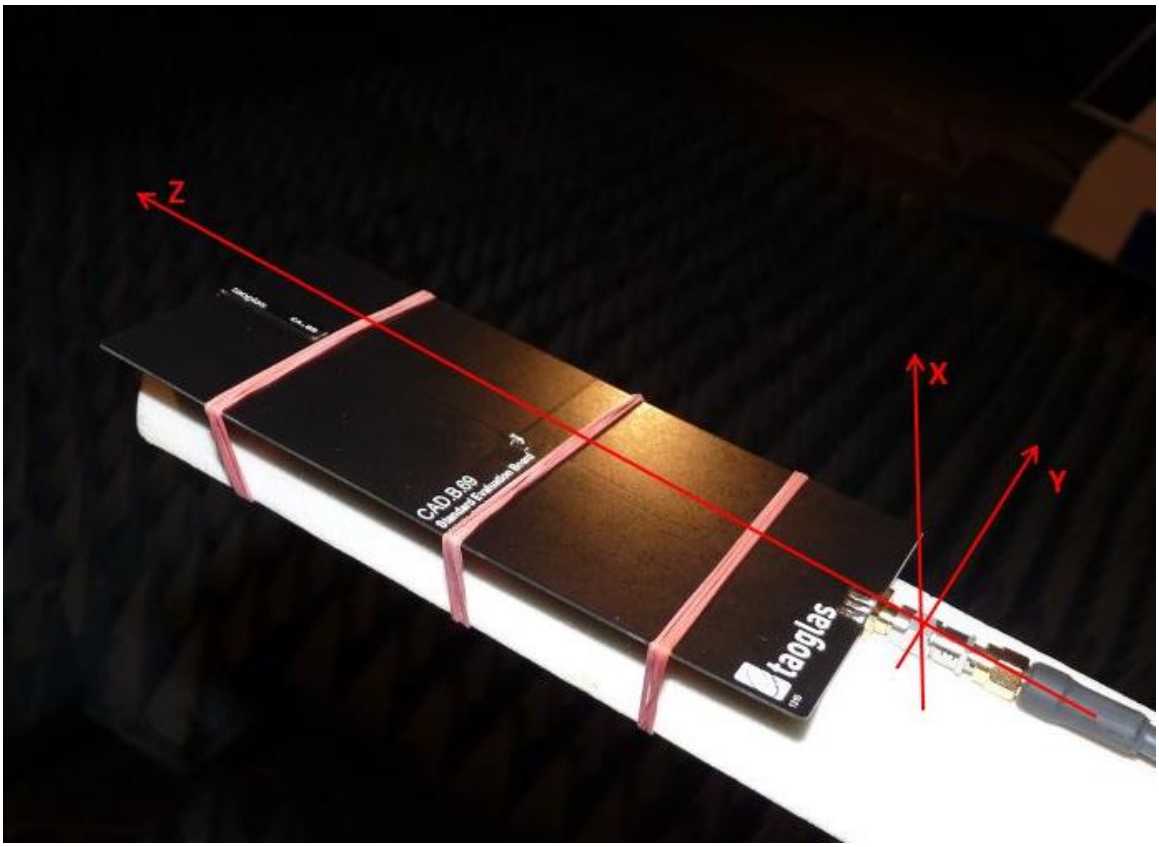
4. Radiation Patterns

4.1 Test Setup – Antenna on Evaluation Board

CAD.A.69

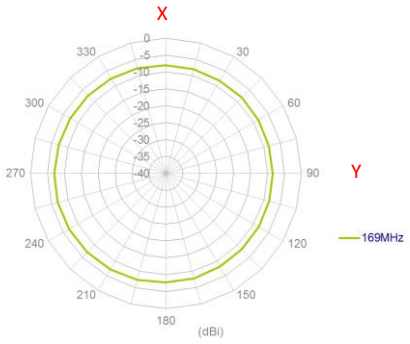


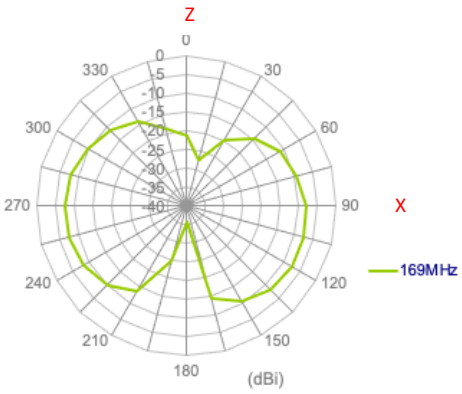
CAD.B.69

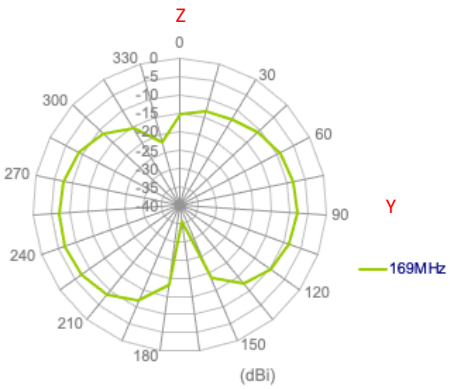


4.2 169 MHz Radiation Patterns

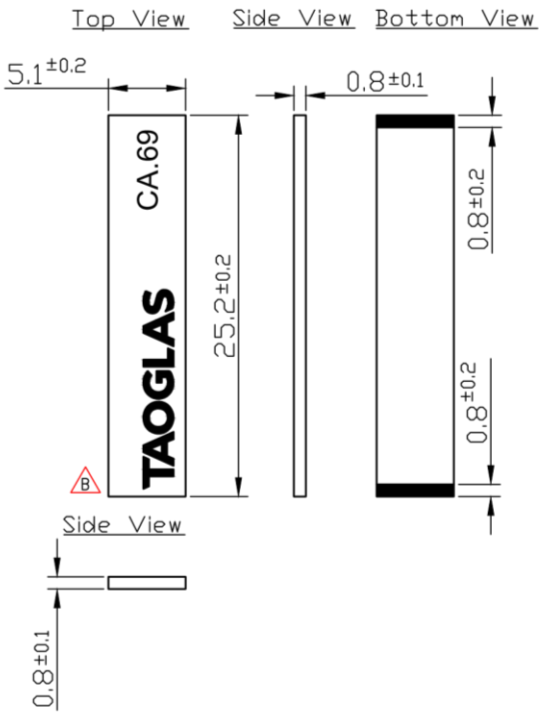
XY Plane	XY Plane Flipped	XZ Plane

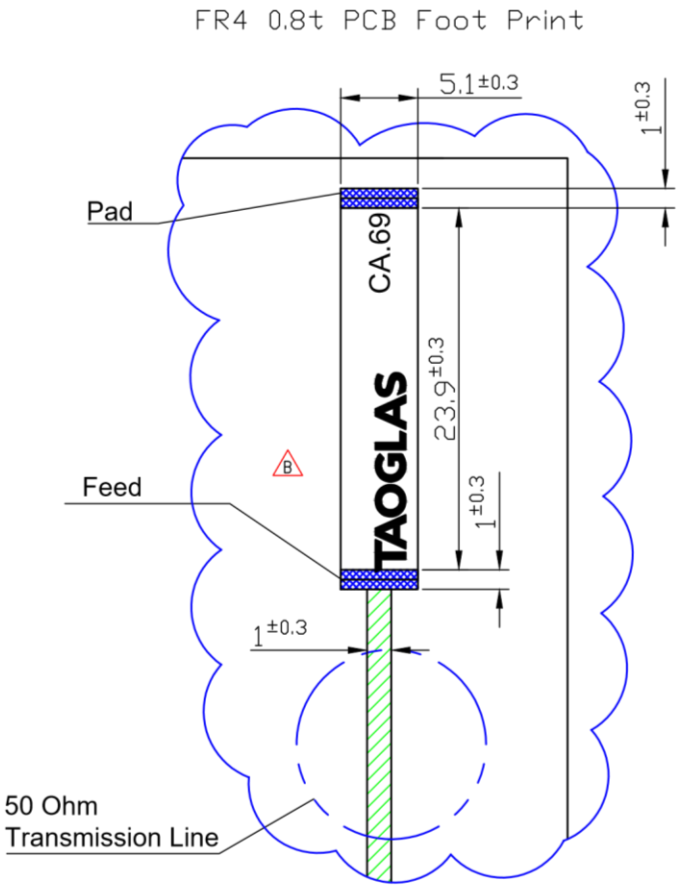






5. Mechanical Drawing



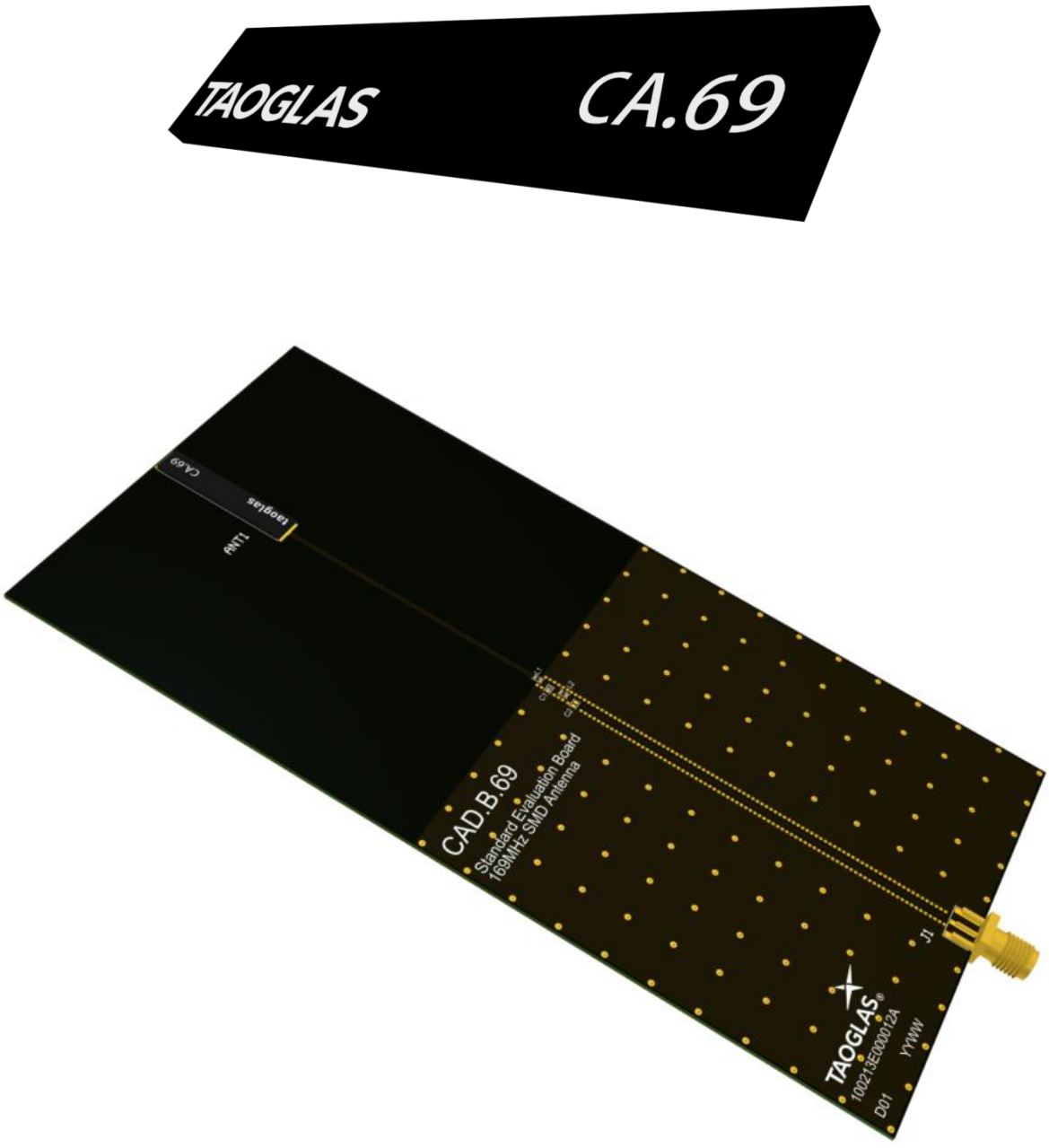


Note:
1.Ni Plated Area
2.Logo & Text Ink Printing : White
3.Clearance Area
4.Copper Area
5.Solder Area

6. Antenna Integration Guide

6.1 Evaluation Board size 160.1*80.6m – CAD.A.69

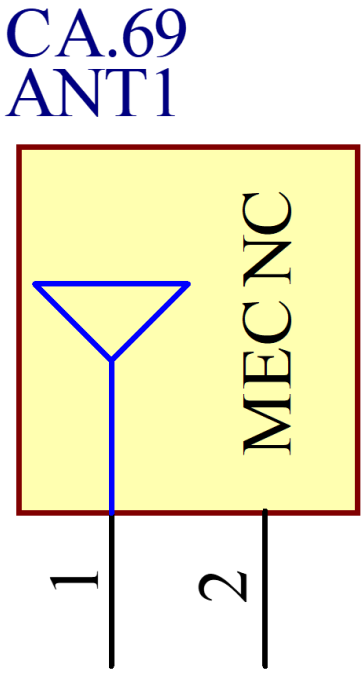
6.1.2 Evaluation Board size 80.1*159.1mm– CAD.B.69



6.2 Schematic Symbol and Pin Definition - CAD.A.69 and CAD.B.69

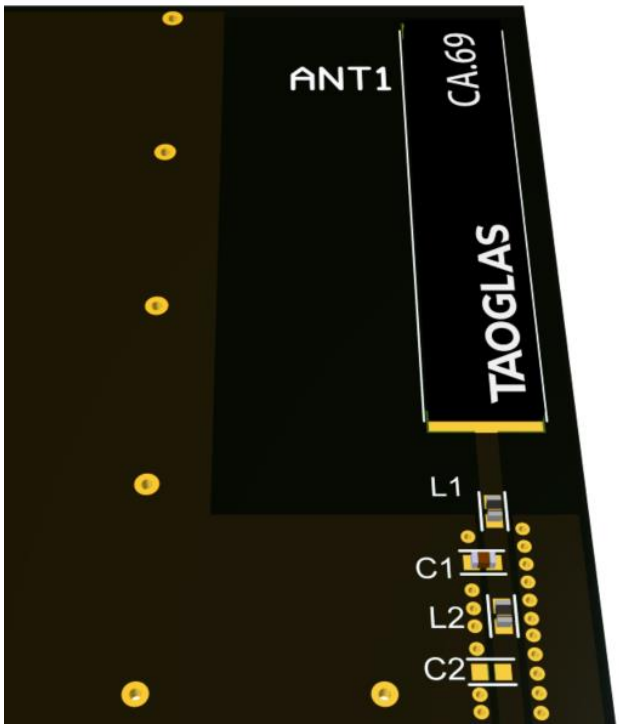
The circuit symbol for the antenna is shown below. The antenna has 2 pins with only one pin (Pin 1) as functional. Pin 2 is for mechanical strength.

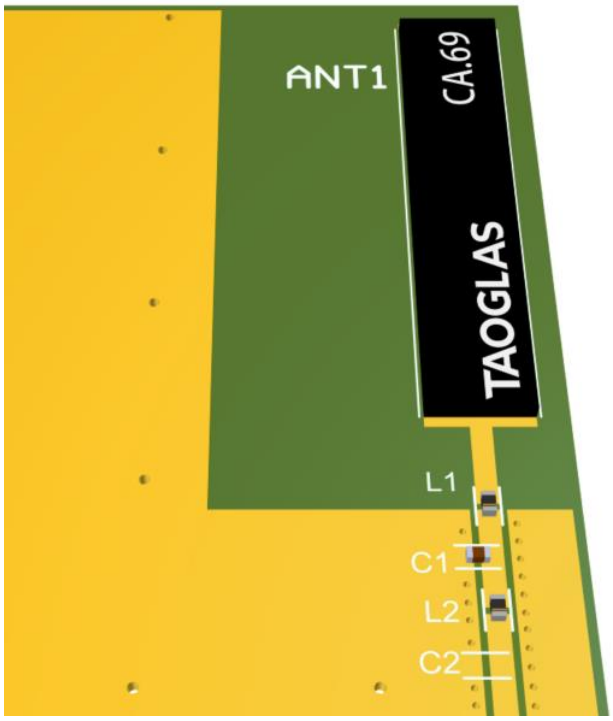
Pin	Description
1	RF Feed
2	Mechanical, Not Connected



6.3.1 Antenna Integration - CAD.A.69

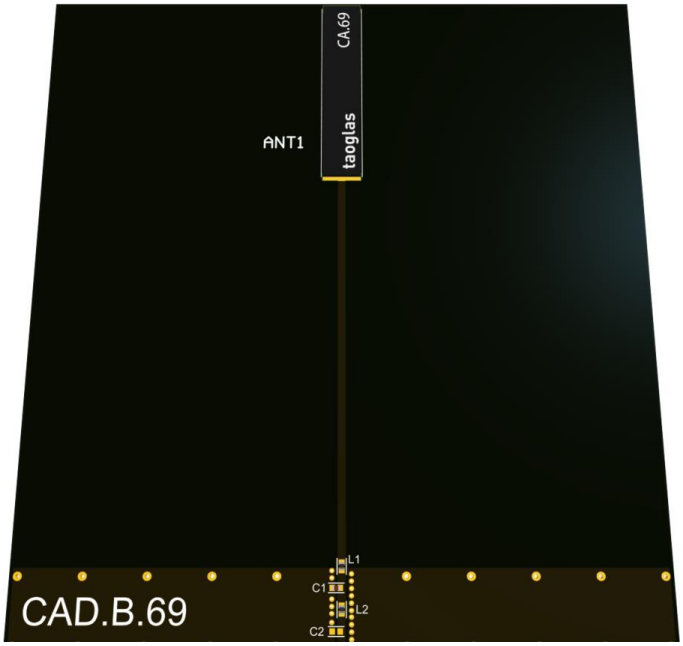
For any given PCB size, the antenna should ideally be placed in the corner of the PCB's shortest side, to take advantage of the ground plane. Optimized matching components can be placed as shown.

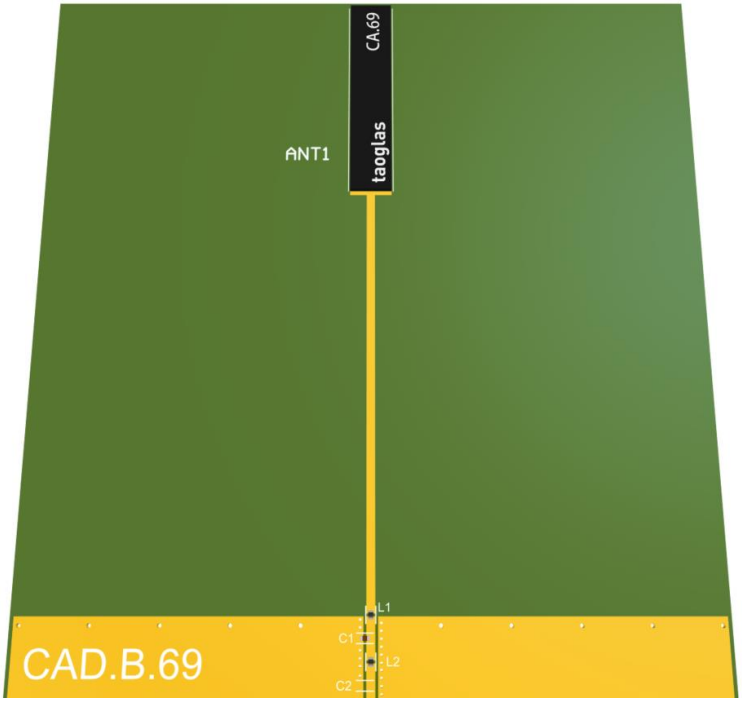




6.3.2 Antenna Integration - CAD.B.69

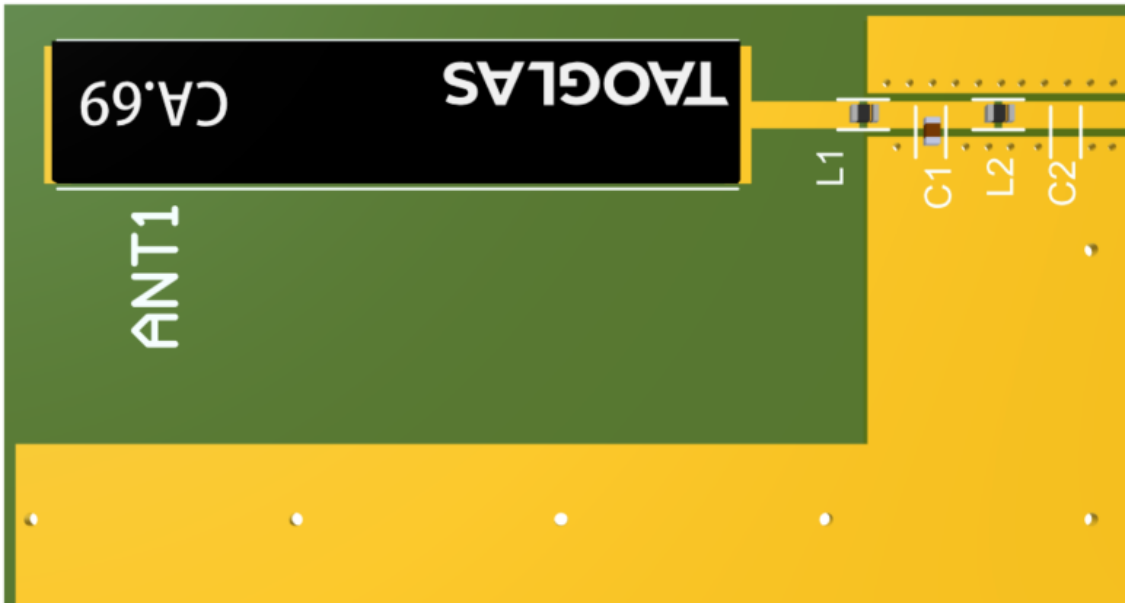
For any given PCB size, the antenna should ideally be placed in the corner of the PCB's shortest side, to take advantage of the ground plane. Optimized matching components can be placed as shown.





6.4.1 PCB Layout - CAD.A.69

The footprint and clearance on the PCB must meet the layout drawing in (Footprint Drawing). Note the placement of the optimized components. L1 is placed as close as possible to the RF feed (pad 1) but still within the transmission line. C1 is then placed tightly in parallel after that, followed by L2 in series. C2 is an optional component but the footprint is recommended in case it is needed.

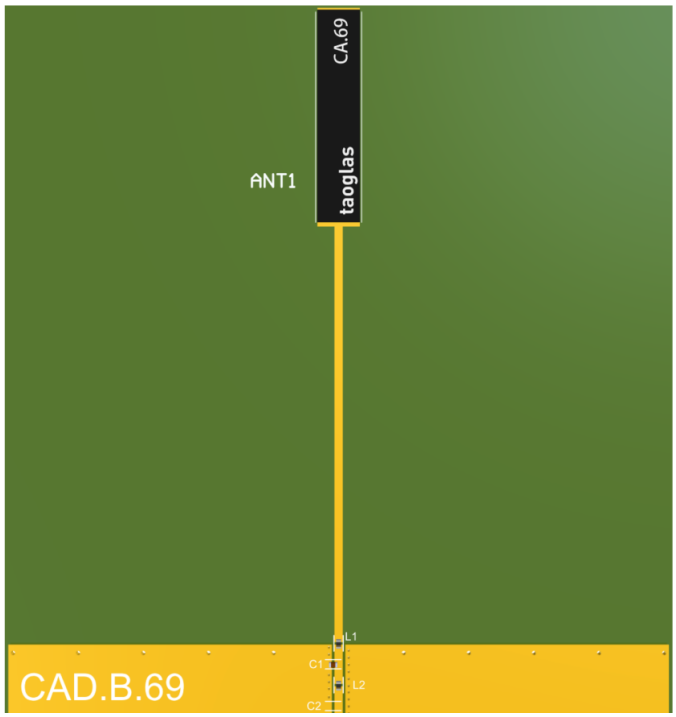


Topside

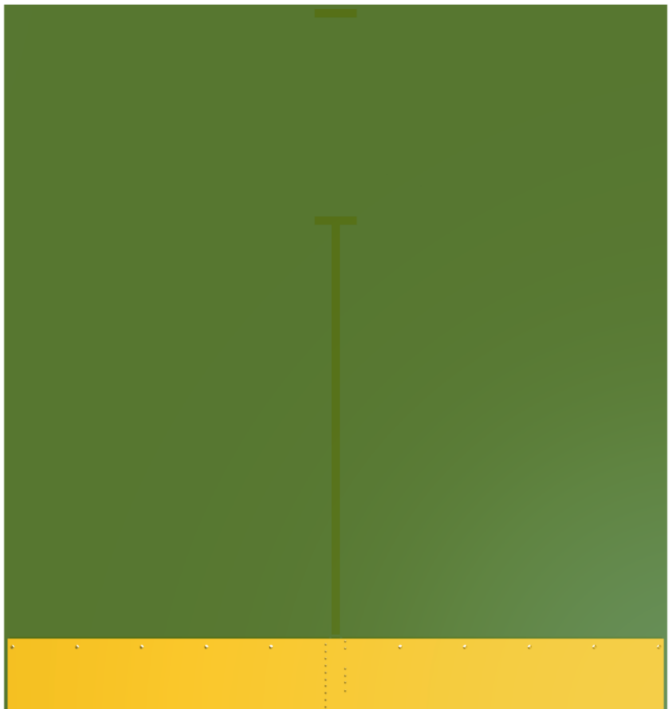
Bottom Side

6.4.2 PCB Layout - CAD.B.69

The footprint and clearance on the PCB must meet the layout drawing in (Footprint Drawing). Note the placement of the optimized components. L1 is placed as close as possible to the RF feed (pad 1) but still within the transmission line. C1 is then placed tightly in parallel after that followed by L2 in series. C2 is an optional component but the footprint is recommended in case it is needed.



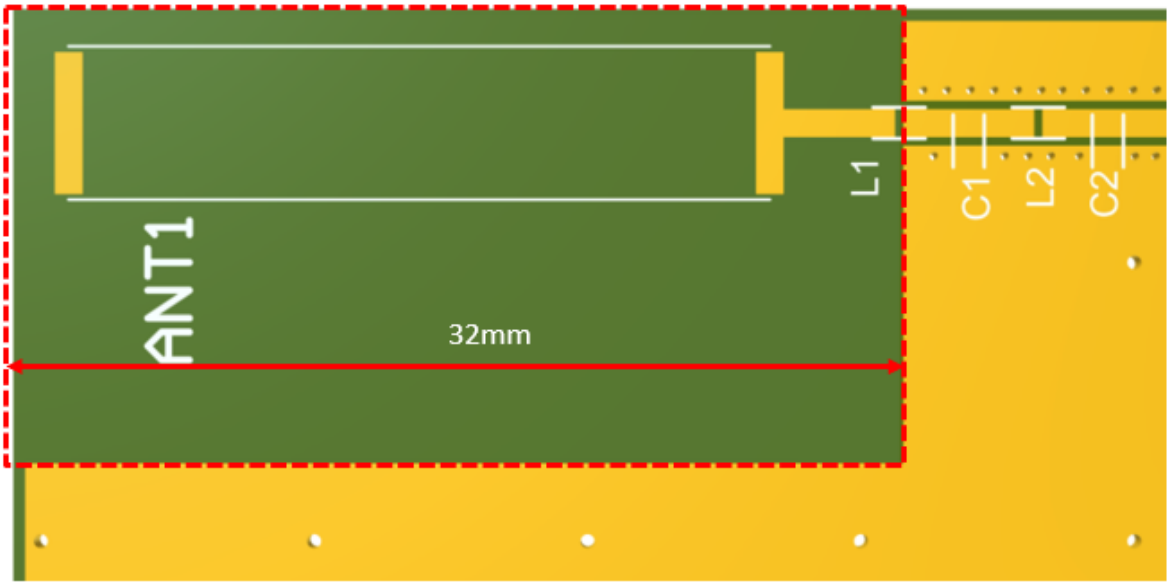
Topside



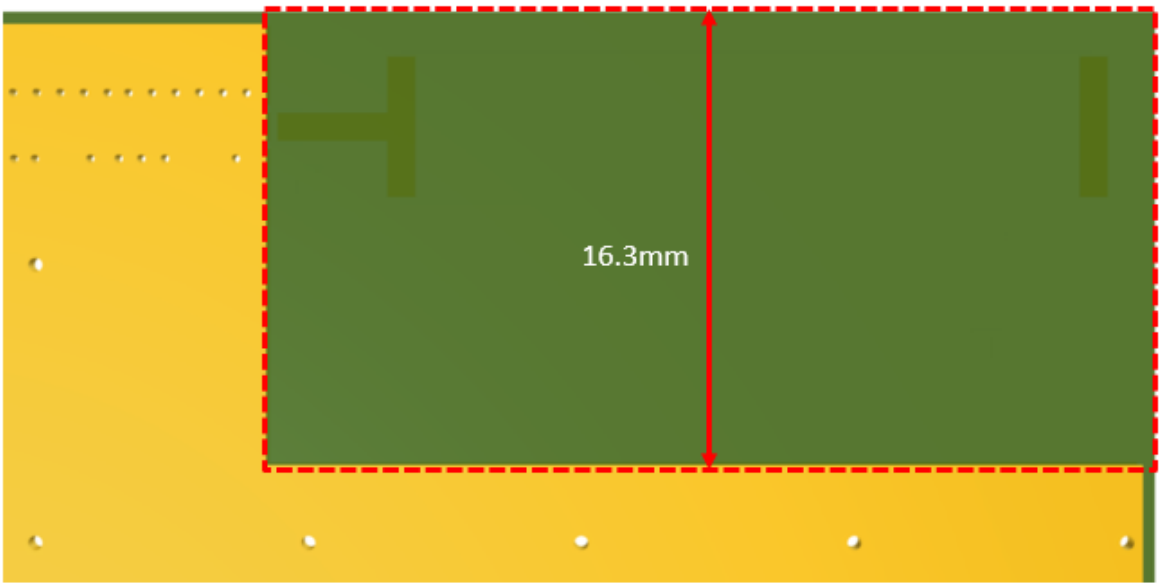
Bottom Side

6.5.1 PCB Clearance - CAD.A.69

Below shows the antenna footprint and clearance through ALL layers on the PCB. Only the antenna pads and connections to feed and GND are present within this clearance area (marked RED). The clearance area extends to 16.3mm in width and 32mm in length from the corner of the PCB. This clearance area includes the bottom side and ALL internal layers on the PCB.



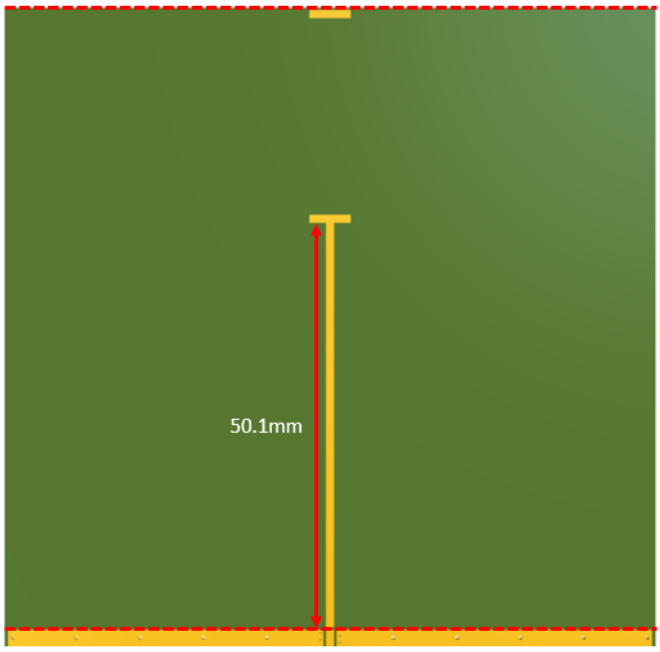
Topside



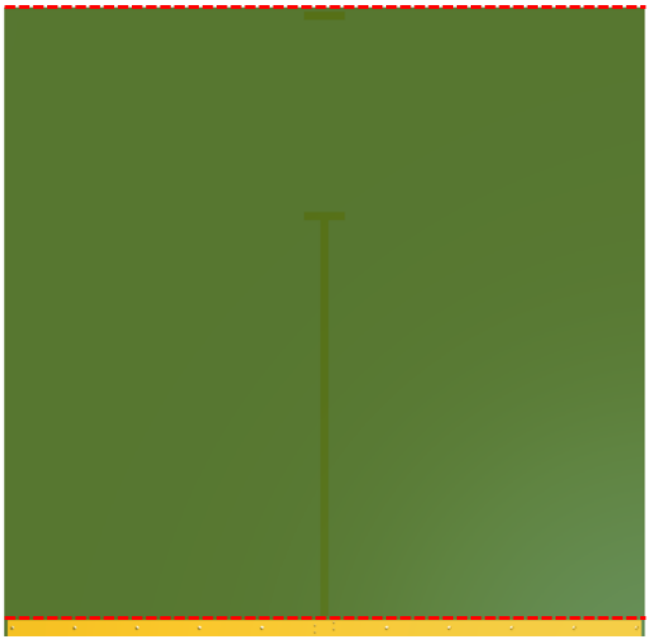
Bottom Side

6.5.2 PCB Clearance - CAD.B.69

Below shows the antenna footprint and clearance through ALL layers on the PCB. Only the antenna pads and connections to feed are present within this clearance area (marked RED). The clearance area extends to 50.1mm from the antenna mechanical pads to the ground area. This clearance area includes the bottom side and ALL internal layers on the PCB.

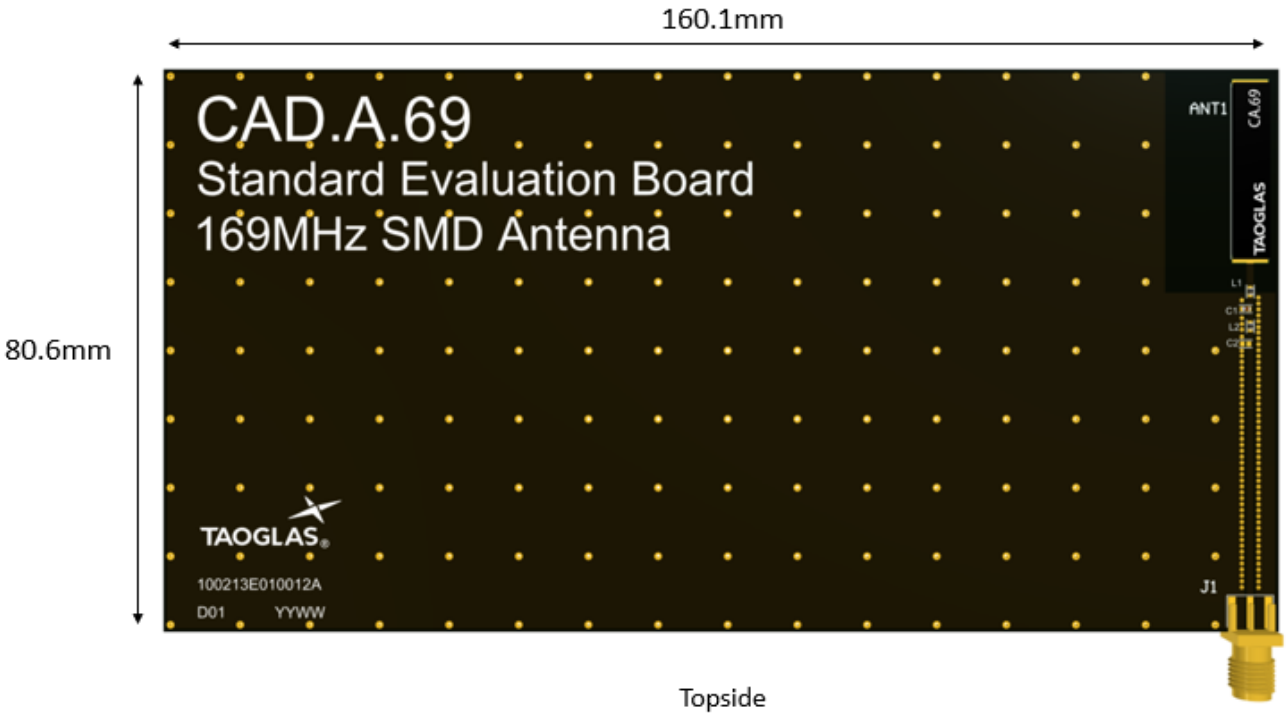


Topside

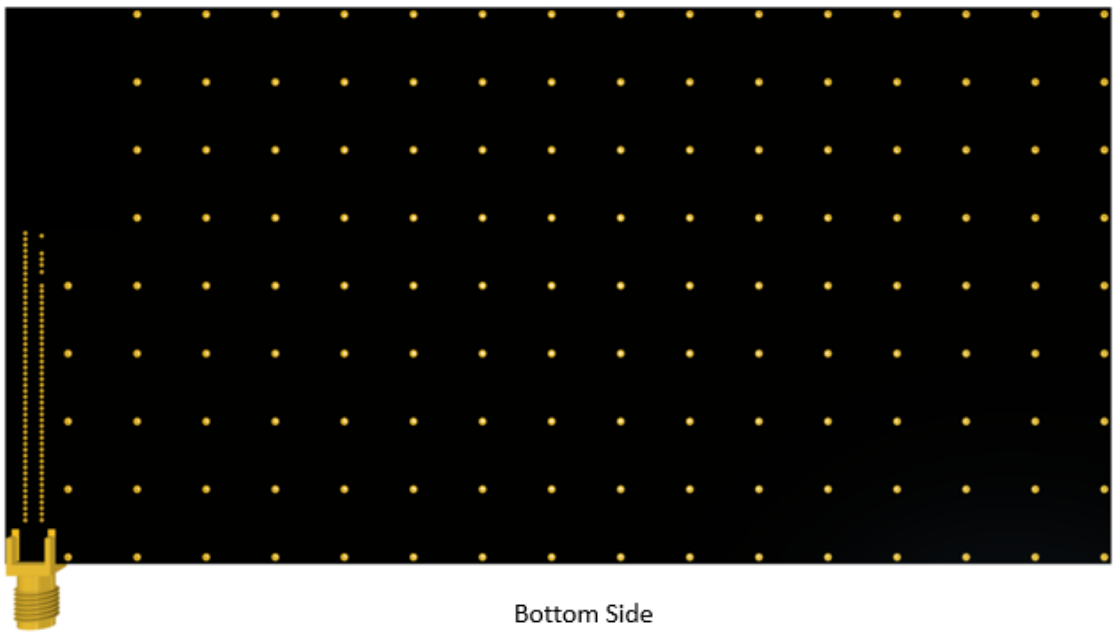


Bottom Side

6.6.1 Evaluation Board - CAD.A.69

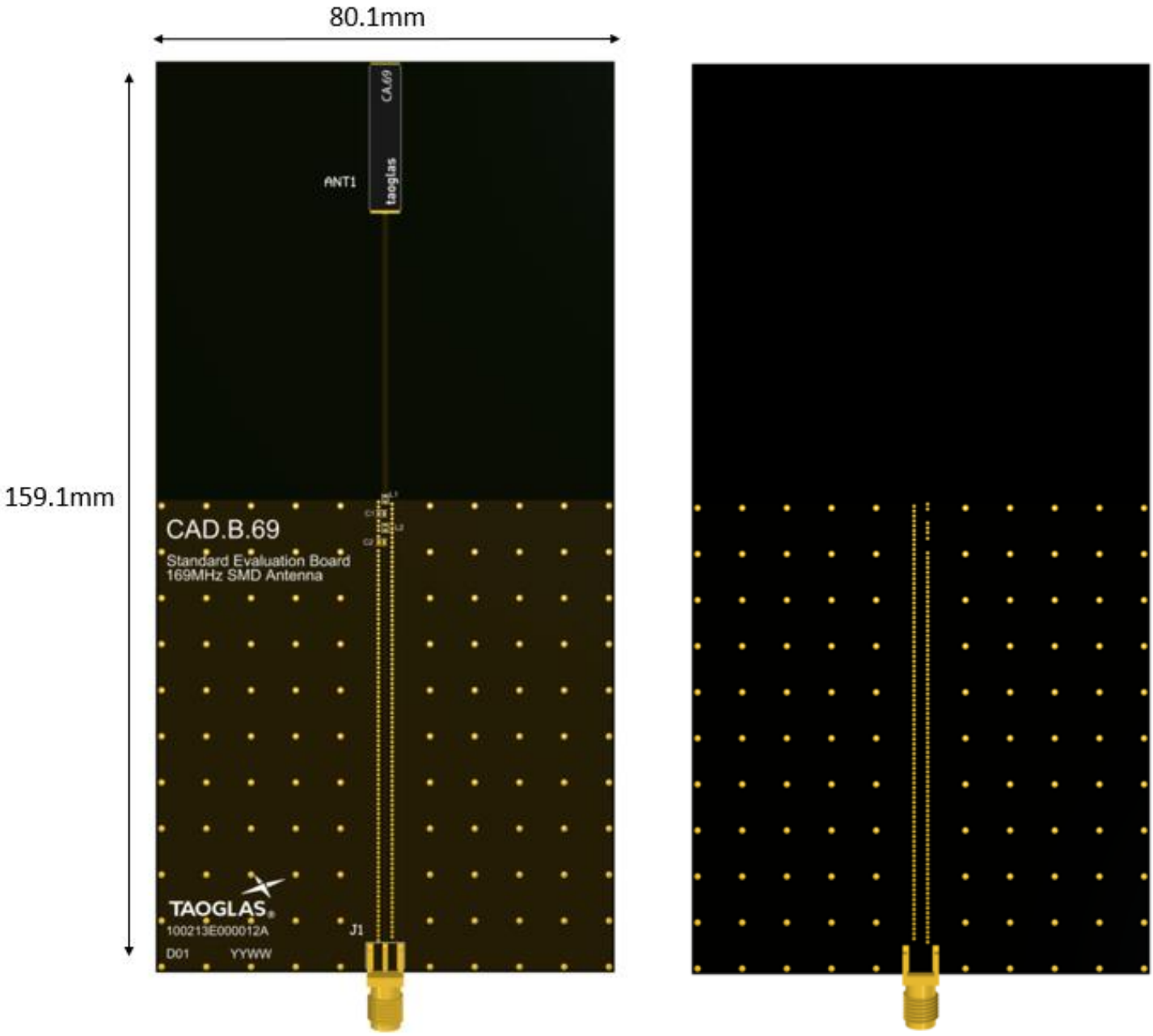


Topside



Bottom Side

6.6.2 Evaluation Board - CAD.B.69



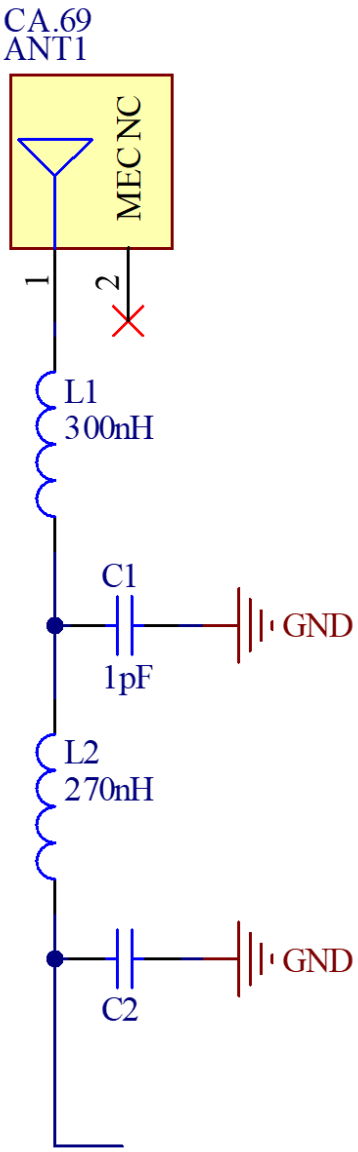
Topside

Bottom Side

6.7.1 Evaluation Board Matching Circuit - CAD.A.69

A matching component (L1) in series with the CA.69 is required for the antenna to have optimal performance on the evaluation board, located outside of the ground plane in the space specified in the above images. Additional matching components may be necessary for your device, so we recommend incorporating extra component footprints, forming a “pi” network, between the cellular module and the edge of the ground plane.

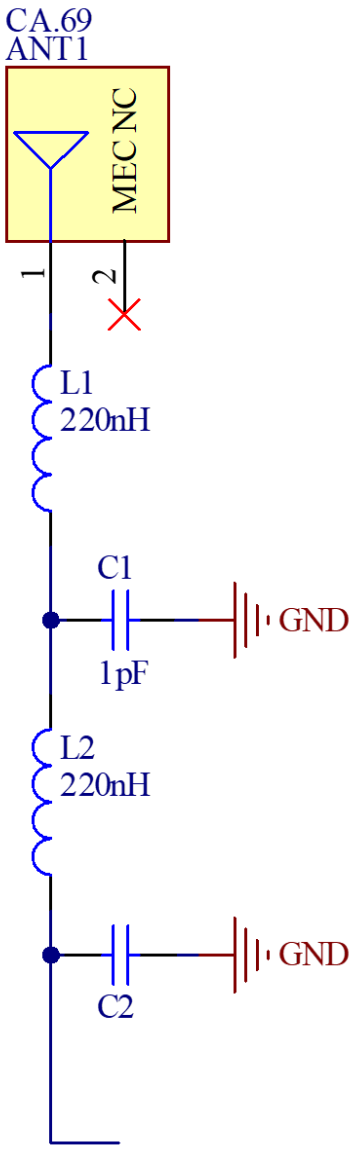
Designator	Type	Value	Manufacturer	Manufacturer Part Number
L1	Inductor	300nH	TDK	MLG1005SR30JT000
L2	Inductor	270nH	TDK	MLG1005SR27JT000
C1	Capacitor	1pF	Murata	GRM1555C1H1R0CA01D
C2	Capacitor	Not Fitted	-	-



6.7.2 Evaluation Board Matching Circuit - CAD.B.69

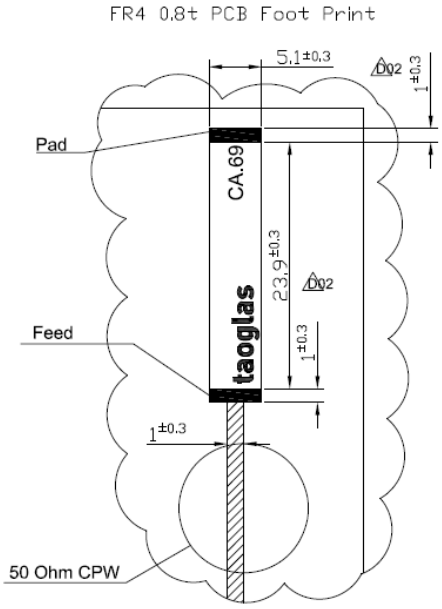
A matching component (L1) in series with the CA.69 is required for the antenna to have optimal performance on the evaluation board, located outside of the ground plane in the space specified in the above images. Additional matching components may be necessary for your device, so we recommend incorporating extra component footprints, forming a “pi” network, between the cellular module and the edge of the ground plane.

Designator	Type	Value	Manufacturer	Manufacturer Part Number
L1	Inductor	220nH	TDK	MLG1005SR22JT000
L2	Inductor	270nH	TDK	MLG1005SR22JT000
C1	Capacitor	1pF	Murata	GRM1555C1H1R0CA01D
C2	Capacitor	Not Fitted	-	-

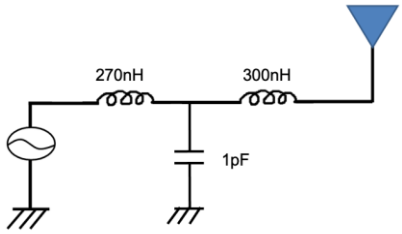


7. Layout Guide

6.1 Solder Land Pattern



6.2 Matching circuit CAD.A.69

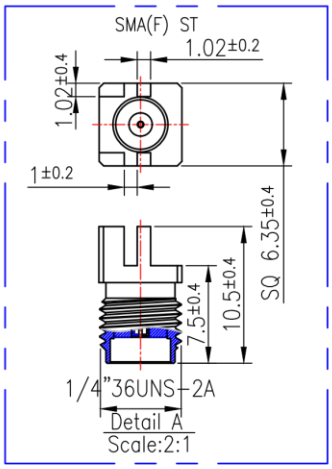


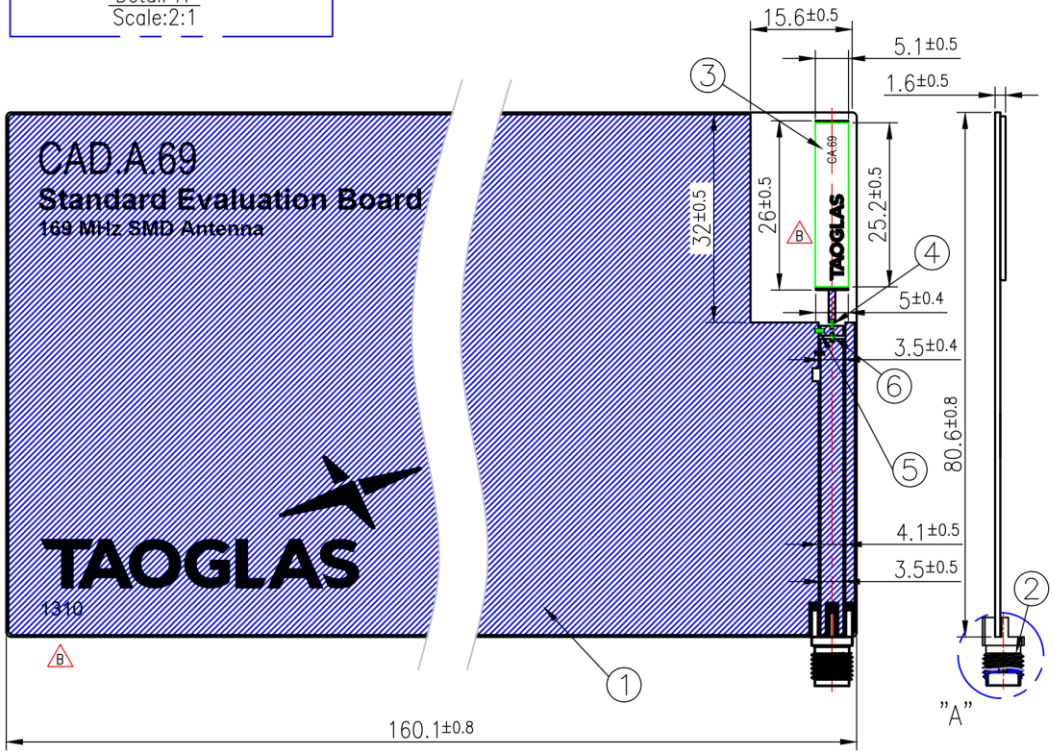
8. Evaluation Board Mechanical Drawing - CAD.A.69

ISO NO: EDW-18-8-0676

<Release>

REV	ZONE	DESCRIPTION	ENG	APPROVED	ISSUED DATE
A	ALL	Initial Design	Eva	Paul	2018/08/29
B	ALL	Replace the new LOGO <ECR-18-8-259>	Karry	Aaron	2023/01/31





Note:
1.Week Batch Code
Example: 2010 Week 1=01.10
2.Slodered area
3.Copper area
4.Logo & Text Ink Printing : White
5.Ground Clearance Area
6.All Material Must Be RoHS Compliant.

	Name	P/N	Material	Finish	QTY
1	CAD.A.69 EVB PCB	100213E010012A	Composite 0.8t	Black	1
2	SMA(F) ST	200413H000002A	Brass	Au Plated	1
3	CA.69 Antenna	001513E020012A	Composite 0.8t	Black	1
4	Inductor (L=300nH) 0402	001513G020055A	Ceramic	N/A	1
5	Capacitor (C=1pF) 0402	001513G010055A	Ceramic	N/A	1
6	Inductor (L=270nH) 0402	001513J000055A	Ceramic	N/A	1

UNLESS OTHERWISE SPECFIED TOLERANCES ON:
.X± 0.2
XX.± 0.5 .XX± 0.1
X.± 0.3 .XXX± 0.05

DATE: 2013/08/29 | MAT'L:
UNIT: mm | FINISH:
THIRD ANGLE PROJECTION | SCALE: 1/1

APPROVED BY: Wayne | CHECKED BY: Aine | DRAWN BY: Jason | CUSTOMERS SIGNATURE / DATE

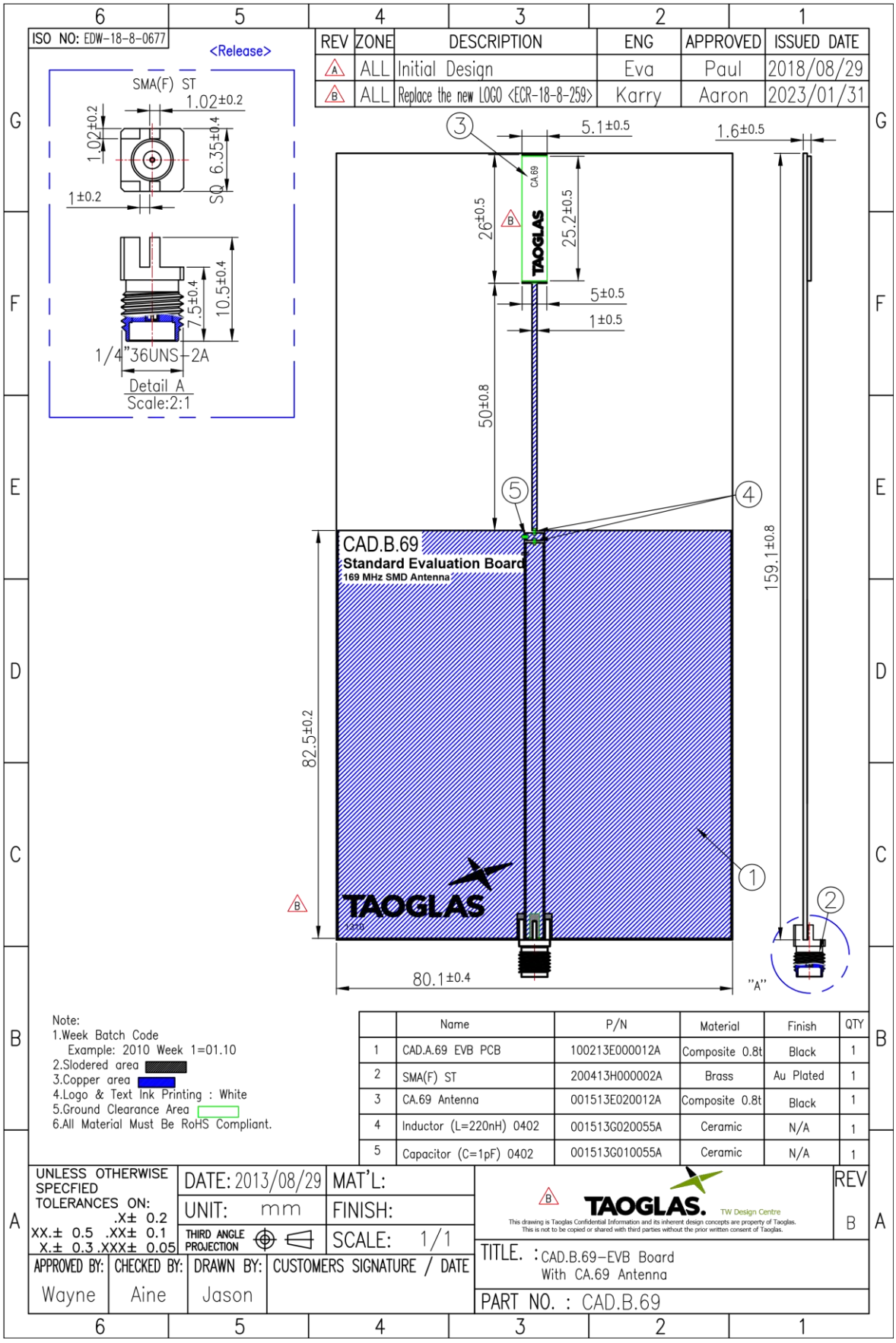
TAOGLAS. TW Design Centre

REV B

TITLE. : CAD.A.69-EVB Board With CA.69 Antenna

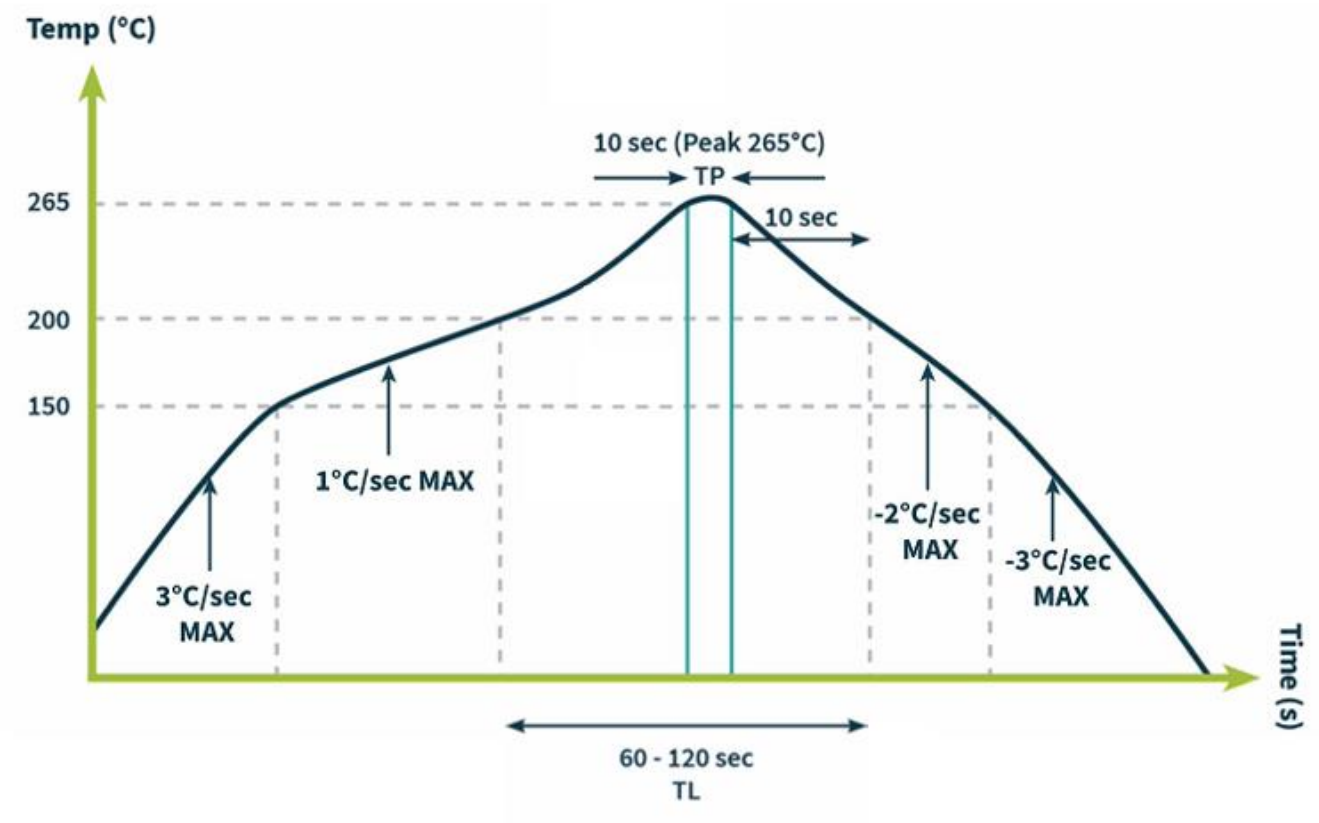
PART NO. : CAD.A.69

9. Evaluation Board Mechanical Drawing - CAD.B.69



10. Soldering Conditions

The CA.69 can be assembled by following the recommended soldering temperatures are as follows:



*Temperatures listed within a tolerance of +/- 10º C

Smaller components are typically mounted on the first pass, however, we do advise mounting the CA.69 when placing larger components on the board during subsequent reflows.

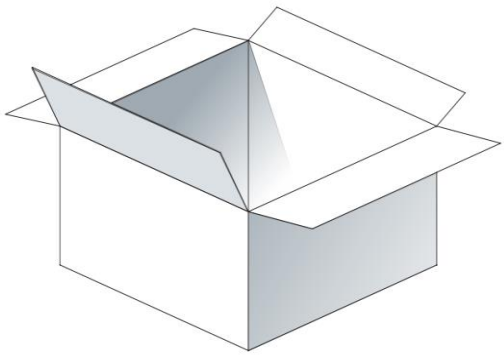
Note: Soldering flux classified ROL0 under IPC J-STD-004 is recommended.

11. Packaging

CA.69
1000 pcs / Reel

1pcs/ Vacuum bag

3000 pcs / Carton
Carton(mm): 350x340x200



Changelog for the datasheet

SPE-13-8-077– CA.69

Revision: D (Current Version)	
Date:	2023-11-23
Changes:	Updated specifications
Changes Made by:	Cesar Sousa

Previous Revisions

Revision: C	
Date:	2022-02-23
Changes:	Integration Guide added
Changes Made by:	Cesar Sousa

Revision: B	
Date:	2021-10-05
Changes:	Format Change, MSL
Changes Made by:	Erik Landi

Revision: A (Original First Release)	
Date:	2017-10-11
Notes:	Initial Release
Author:	STAFF