

1. Features

- Antenna for 1164-1249 MHz, 1559–1609 MHz, GNSS for embedded applications
- GNSS bands covered are GPS L1, L2, L5; GLONASS L1, L2, L3; Galileo E1, E5a/b; BeiDou B1I, B2I, B3, B2a; QZSS L1, L1C, L1S, L2C, L5
- Solution for all global public constellations: GPS, GLONASS, Beidou and GALILEO.
- Maintains high performance on device: DFI (Designed For Integration)
- Designed for SMD mounting
- Supplied on Tape and Reel

2. Description

Raptor is intended for use with all positioning applications. The antenna has RHCP characteristics suitable for GNSS signals. A truly novel antenna approach with ultra-low profile, but with the high performance of a ceramic patch.

3. Applications

- Trackers
- Portable Devices
- Drones
- Wearable devices
- Positioning



4. Part Number

Raptor: SR4G053



5. General Data

Product name	Raptor
Part Number	SR4G053
Frequency	1164 – 1249MHz; 1559 – 1609 MHz
Polarization	Linear
Operating temperature	-40°C to140°C
Environmental condition test	ISO 16750-4 5.1.1.1/5.1.2.1 / 5.3.2
Impedance with matching	50 Ω
Weight	< 0.2 g
Antenna type	SMD
Dimensions	16.0 x 8.0 x 1.7 (mm)

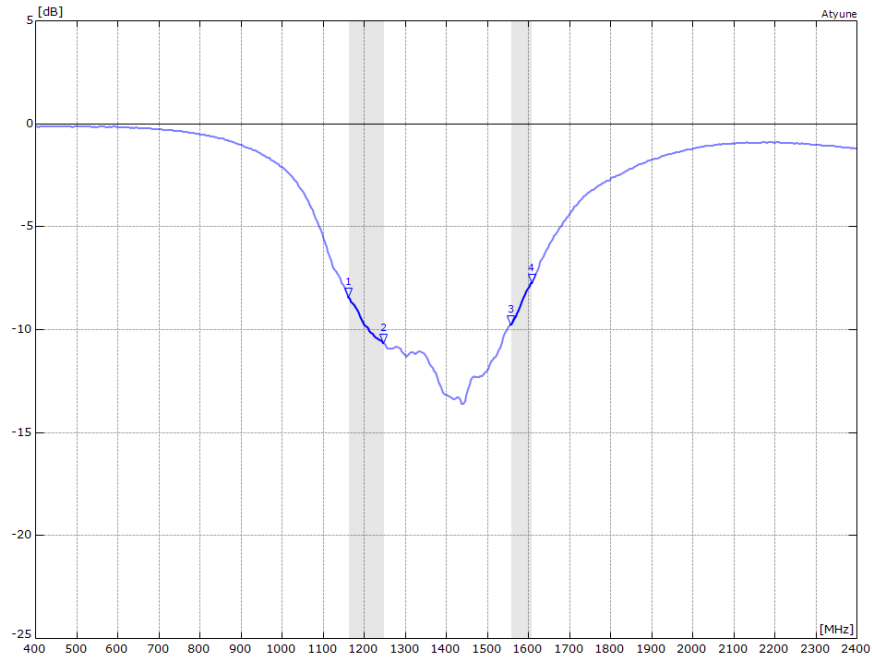
6. RF Characteristics

	1164 – 1249MHz	Conditions
Peak gain	2.3dBi	All data measured on Antenova's evaluation PCB Part No. SR4G053-EVB-1
Average gain (Linear)	-0.9dBi	
Average efficiency (Linear)	>75%	
Average efficiency (RHCP)	>38%	
Maximum return loss	<-8.5dB	
Maximum VSWR	2.3:1	

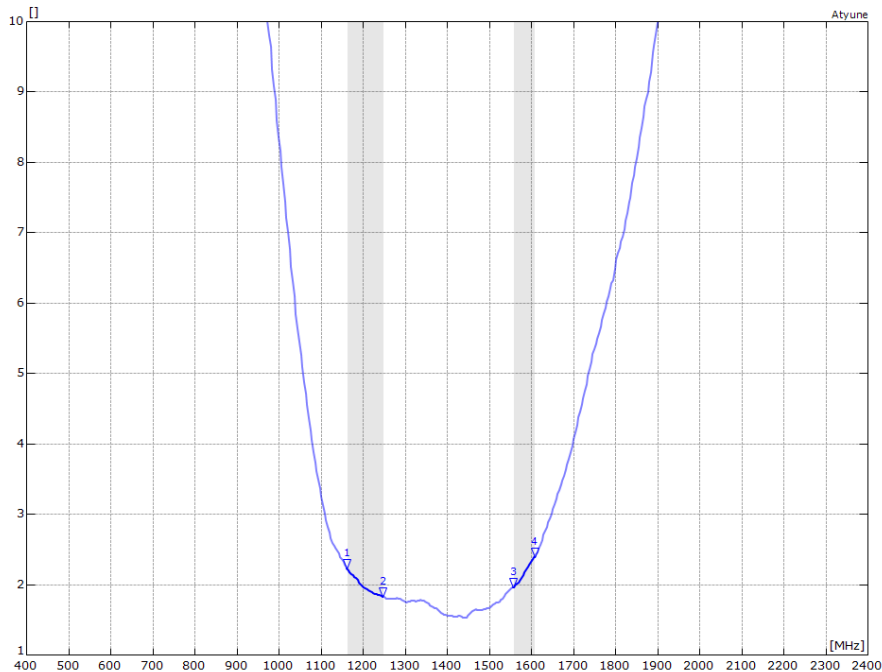
	1559 – 1609MHz	Conditions
Peak gain	2.2dBi	All data measured on Antenova's evaluation PCB Part No. SR4G008-EVB-1
Average gain (Linear)	-1.50dBi	
Average efficiency (Linear)	>65%	
Average efficiency (RHCP)	>33%	
Maximum return loss	<-7.5dB	
Maximum VSWR	2.4:1	

7. RF Performance

7.1 Return Loss

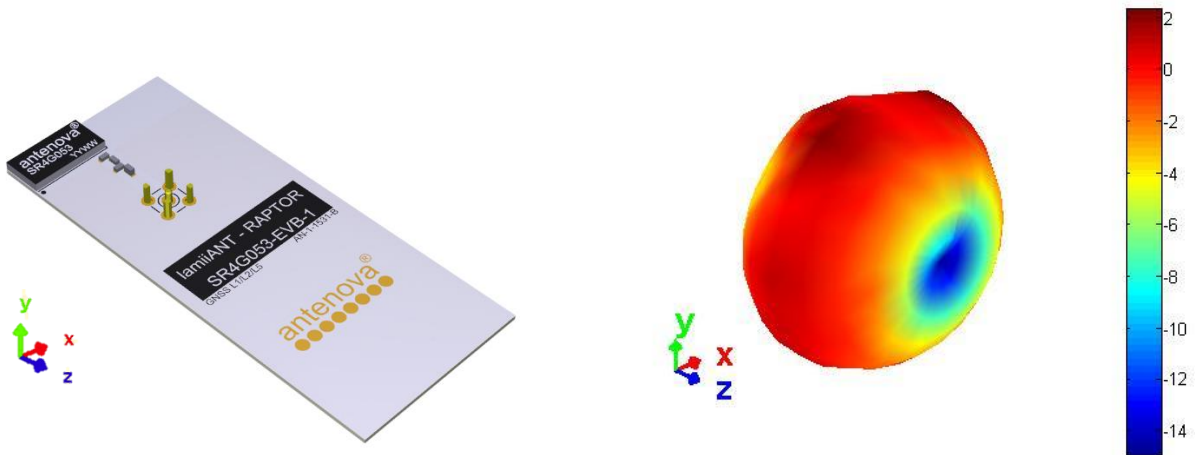


7.2 VSWR



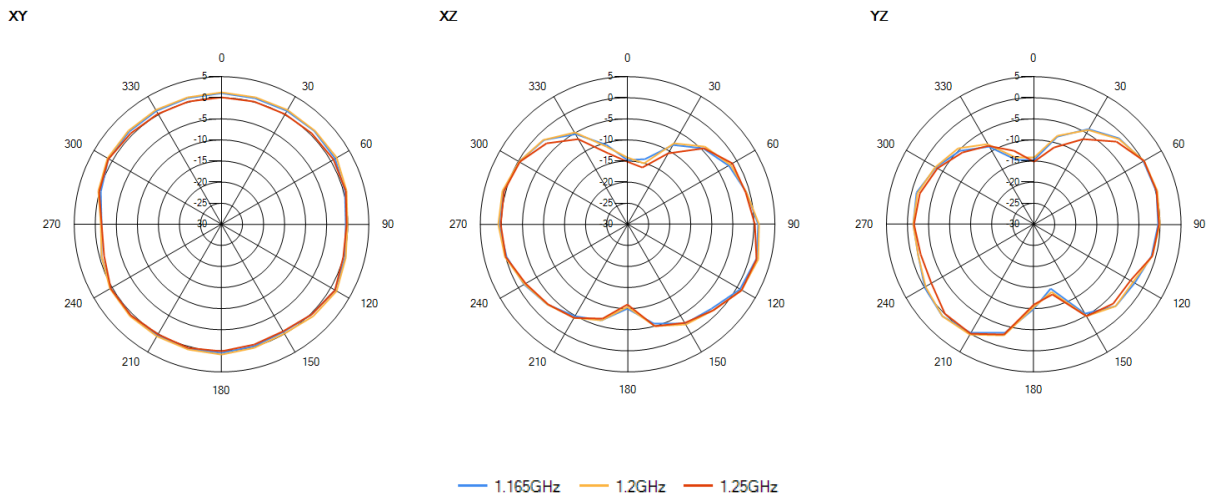
7.3 Antenna pattern

7.3.1 1164 MHz – 1249 MHz

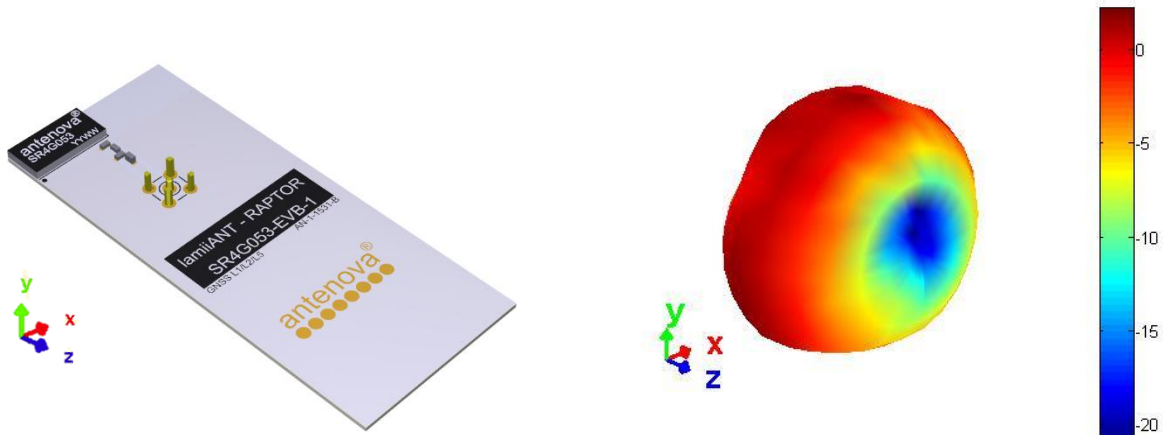


3D pattern at 1200 MHz

Drag to rotate pattern and PCB by using Adobe Reader

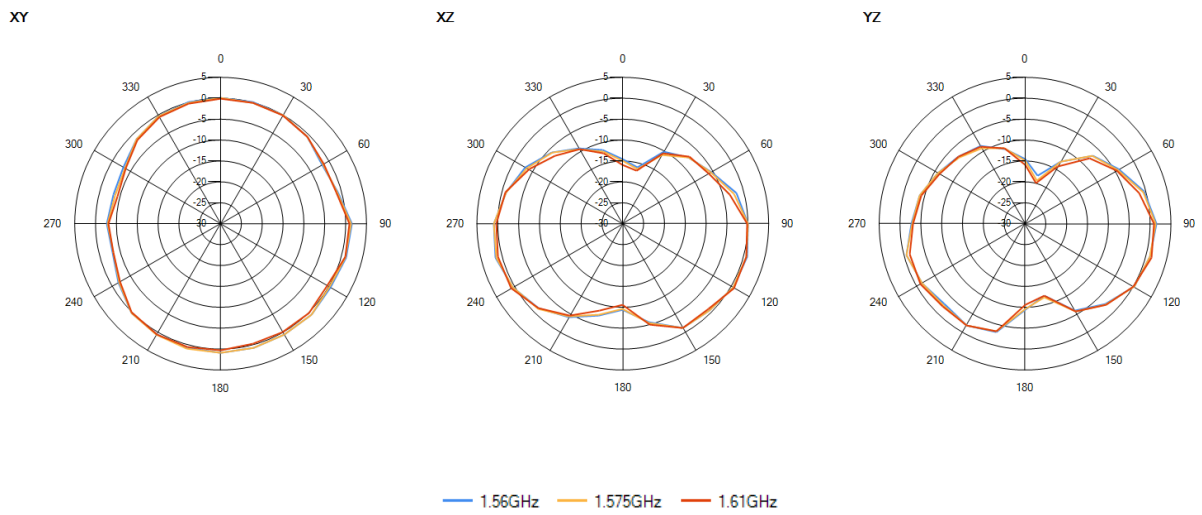


7.3.1 1559 MHz – 1609 MHz

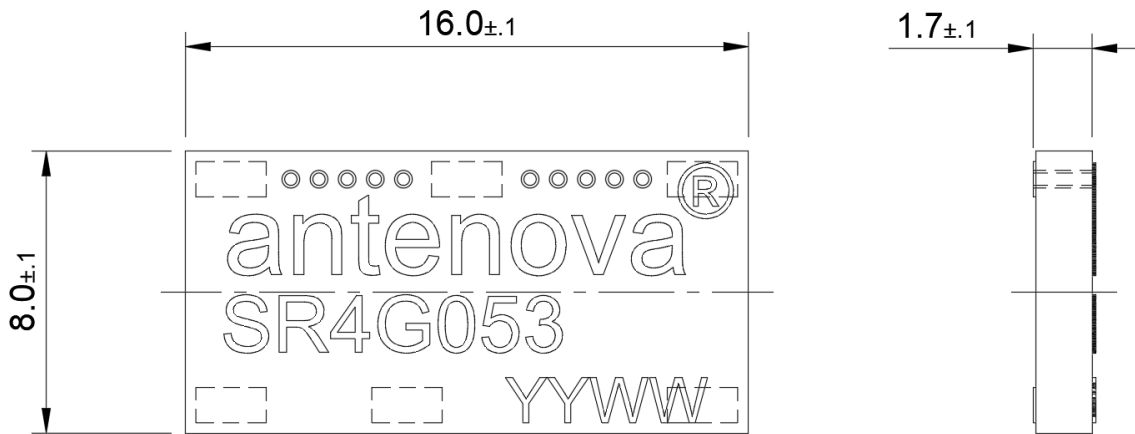


3D pattern at 1575 MHz

Drag to rotate pattern and PCB by using Adobe Reader



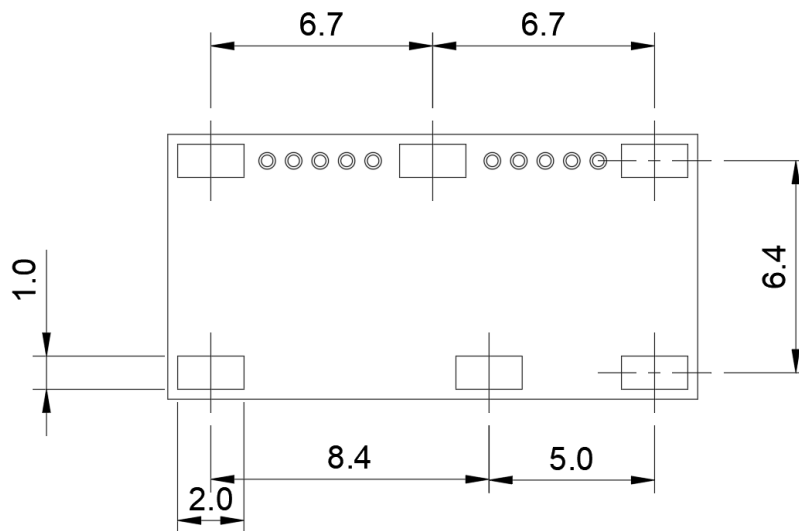
8. Antenna Dimensions



L	W	H
Length	Width	Height
16.0 ± 0.1	8.0 ± 0.1	1.7 ± 0.2

All dimensions in mm

Bottom side dimensions

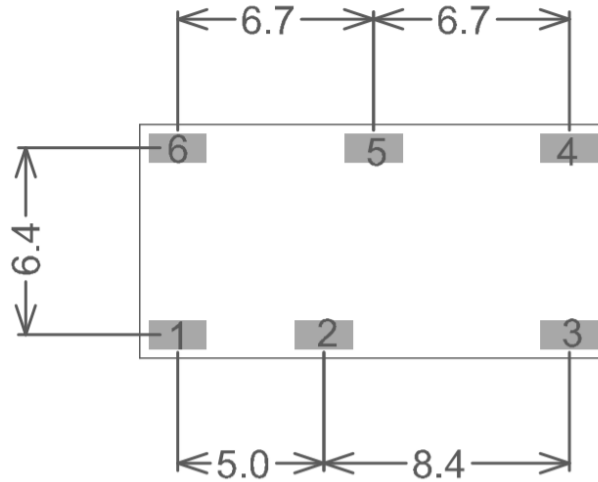


6 solder pads (2.0 x 1.0 mm)

Antennas for Wireless Applications

9. Antenna footprint

The recommended host PCB footprint is below. This is shown as it should be placed on the host PCB.

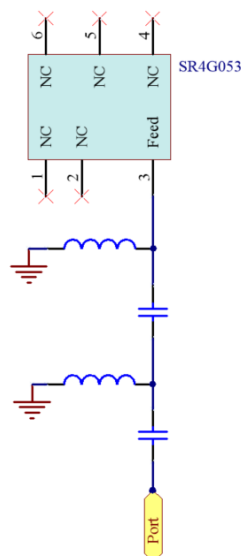


All Pads = 2.0 x 1.0 (mm)

*CAD files of the antenna footprint are available at www.antenova.com.

10. Schematic

The circuit for the antenna and the matching components is below. The RF feed connection and GND connections are critical to the function of the antenna, and must be followed as shown. This circuit can be used for the circuit capture of the host PCB.



11. Electrical Interface

11.1 Transmission Line

All transmission lines should be designed to have a characteristic impedance of 50Ω.

- The length of the transmission lines should be kept to a minimum
- Any other parts of the RF system like transceivers, power amplifiers, etc, should also be designed to have an impedance of 50 Ω

Once the material for the PCB has been chosen (PCB thickness and dielectric constant), a coplanar transmission line can easily be designed using any of the commercial software packages for transmission line design. For the chosen PCB thickness, copper thickness and substrate dielectric constant, the program will calculate the appropriate transmission line width and gaps on either side of the track so the characteristic impedance of the co-planar transmission line is 50 Ω.

11.2 Matching Circuit

The antenna requires a matching circuit that must be optimized for each product. The matching circuit will require up to four components and the following pad layout should be designed into the device so the correct circuit can be installed.

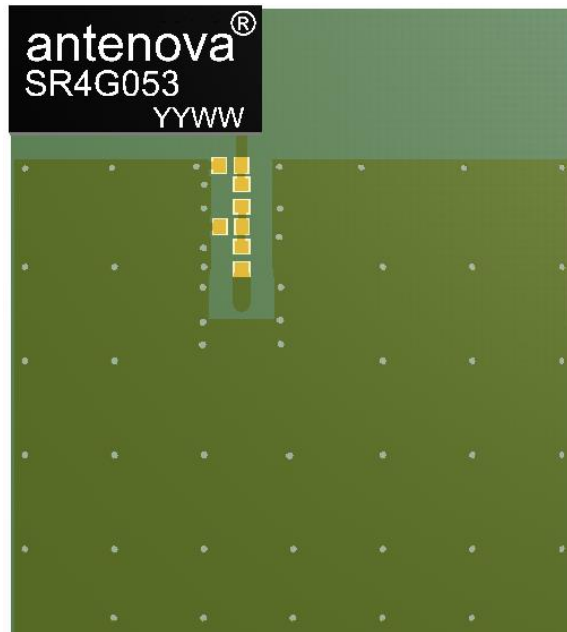
The matching network must be placed close to the antenna feed to ensure it is more effective in tuning the antenna.



12. Antenna Integration Guide

12.1 Antenna Placement

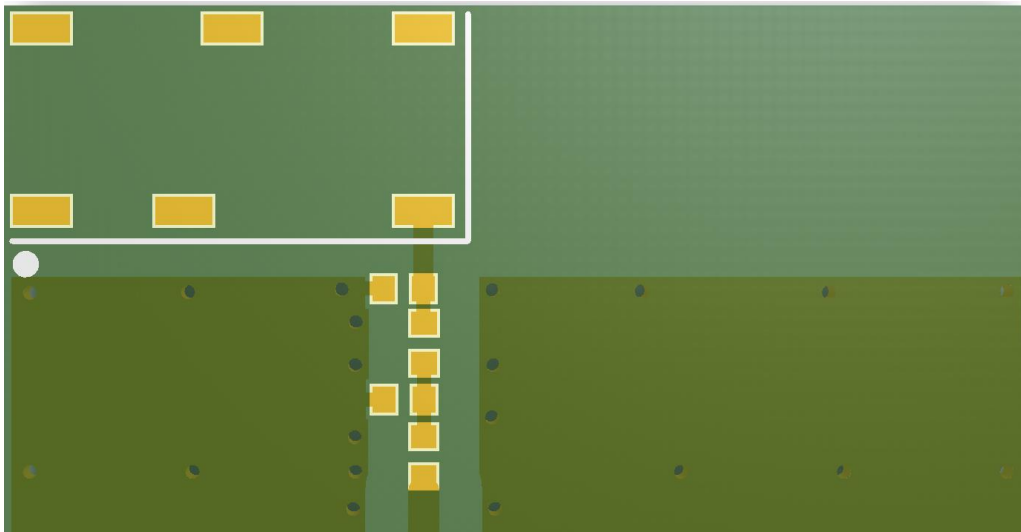
Whatever the size of the host PCB, the antenna should ideally be placed on the shortest side, at the left side edge.



12.2 Host PCB Layout

The footprint and clearance of the host PCB must meet the antenna specification. An example of the PCB layout below shows the antenna footprint with clearance. Pins 1, 2, 4, 5 and 6 are shown floating. The feed (Pin 3) connects to the matching circuit close to the antenna.

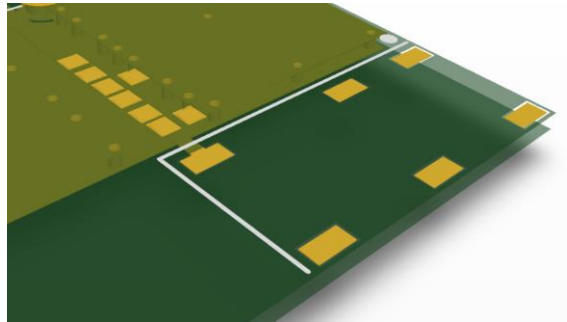
Example host layout



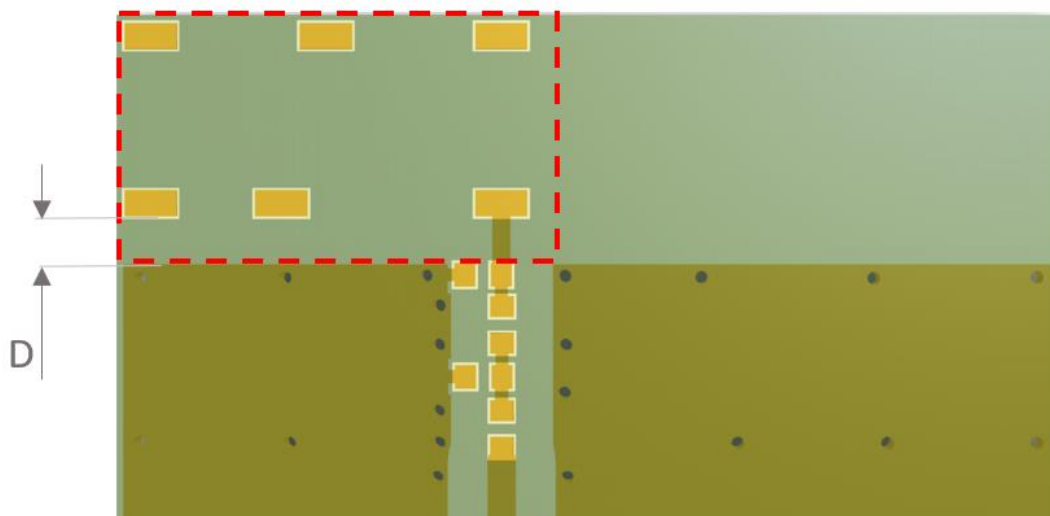
12.3 Host PCB Clearance

The diagram below shows the antenna's footprint and clearance through all of the layers on the PCB. Only the antenna pads and connections to the feed are present within this clearance area.

Example host layout



The ideal clearance area adjacent to the antenna is across the full width of the PCB. Ideally the antenna requires full clearance in the area adjacent to it for the entire board width on all layers. Implementing with the minimal GND clearance will result in a lower performance than stated.



D = 1.8mm distance from antenna pad

Red area represents minimum
clearance area of 17.0x 9.5(mm)

14. Reference Board

The reference board has been designed for the purpose of evaluating the SR4G053 antenna, and includes a SMA female connector.

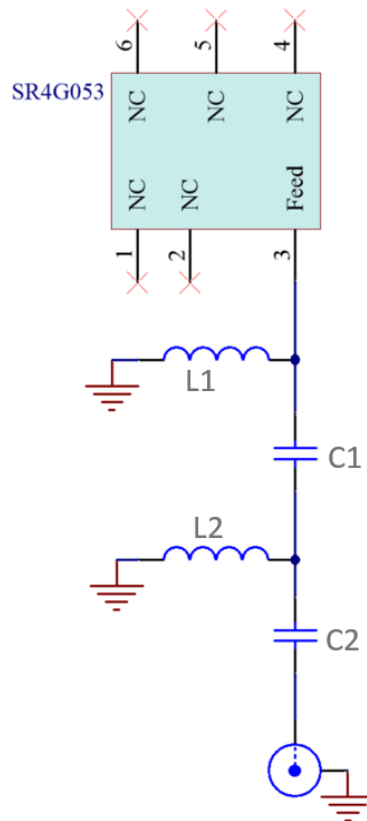
SR4G053 Evaluation Board



To order a reference board please see www.antenna.com

14.1 Reference Board Matching Circuit

The reference board has been designed for evaluation of the SR4G053 antenna, and is fitted with an SMA female connector.



Designator	Type	Value	Description
C1	Capacitor	1.8pF	Murata GJM15 series
L1	Inductor	5.6nH	Murata LQG15HN series
L2	Not fitted	Not fitted	Not fitted
C2	Resistor	0R	Non-specific

15. Soldering

This antenna is suitable for lead free soldering.

The reflow profile should be adjusted to suit the device, oven and solder paste, while observing the following conditions:

- The maximum temperature should not exceed 240°C
- However for lead free soldering, a maximum temperature of 255°C for no more than 20 seconds is permitted.
- The antenna should not be exposed to temperatures exceeding 120°C more than three times during the soldering process.

16. Hazardous Material Regulation Conformance

The antenna has been tested to conform to RoHS requirements. A certificate of conformance is available from Antenova's website.

17. Packaging

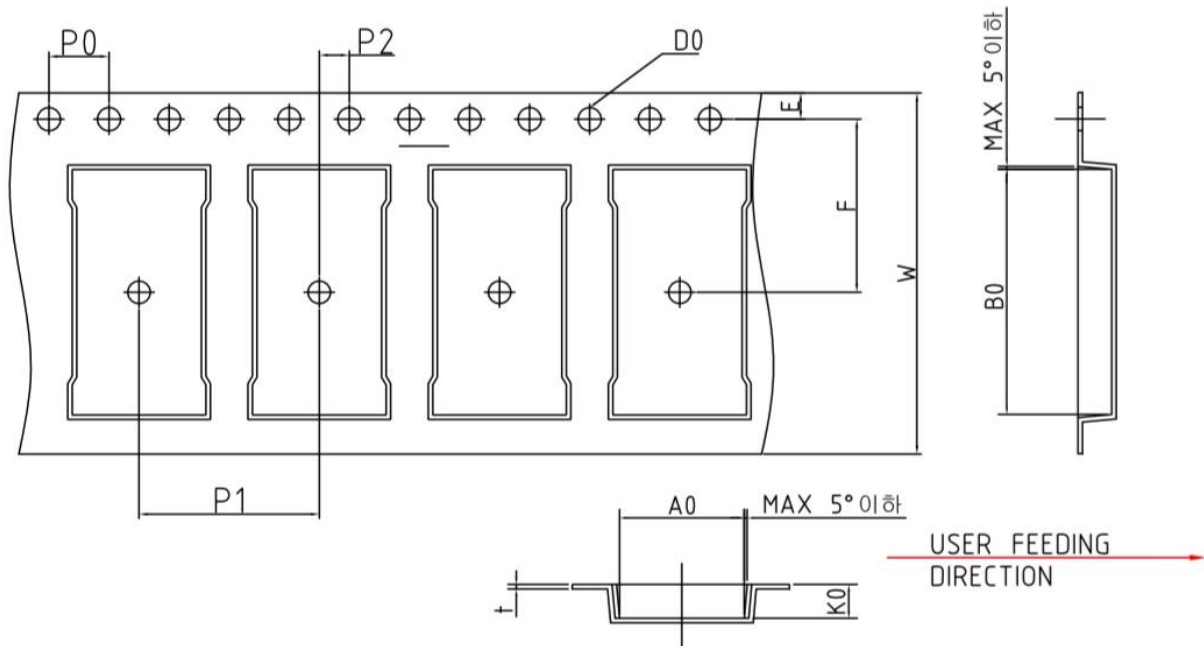
17.1 Optimal Storage Conditions

Temperature	-10°C to 40°C
Humidity	Less than 75% RH
Shelf life	24 Months
Storage place	Away from corrosive gas and direct sunlight
Packaging	Reels should be stored in unopened sealed manufacturer's plastic packaging.

Note: Storage of open reels of antennas is not recommended due to possible oxidization of pads on antennas. If short term storage is necessary, then it is highly recommended that the bag containing the antenna reel is re-sealed and stored in like storage conditions as in above table.

The shelf life of the antenna is two years, providing the factory seal on the package has not been broken.

17.2 Tape Characteristics



D0	A0	B0	P0	P1	P2
1.55 +0.1	8.30 ± 0.1	16.30 ± 0.1	4.00 ± 0.1	12.00 ± 0.1	2.00 ± 0.1

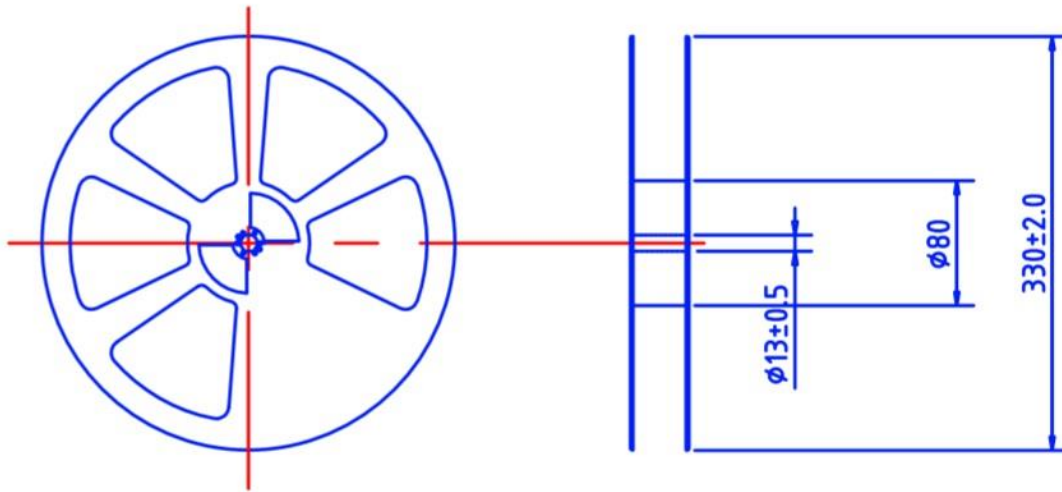
E	F	W	K0	T
1.75 ± 0.1	11.50 ± 0.1	24.00 ± 0.3	2.25 ± 0.1	0.3±0.05

Dimensions in mm

Notes:

- 10 Sprocket hole pitch cumulative tolerance = ± 0.2
- Camber not to exceed 1mm in 100mm
- A_0 and B_0 measured on a plane 0.1mm above the bottom of the pocket.
- K_0 measured from a plane on the inside bottom of the pocket to the top surface of the carrier.

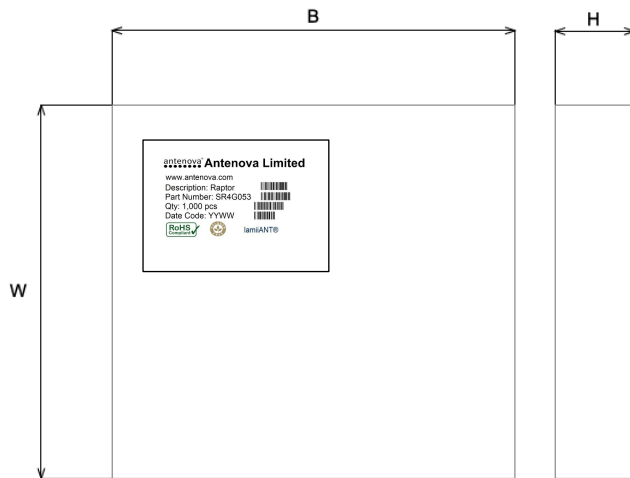
17.3 Reel Dimensions



All dimensions in mm

REEL DIMENSION	Type	Colour	Size	Hub
	PS	Black	$\phi 330$	$\phi 80$

17.4 Box Dimensions



Width (W)	Breadth (B)	Thickness (H)
340mm	360mm	45mm

17.5 Bag Properties

Reels are supplied in protective plastic packaging.

17.6 Reel Label Information

