

# Central Frequency 1800 MHz Spring Antenna

## FEATURES & BENEFITS

- Light Weight
- Easy Installation
- Good compatibility, Sensitive signal reception, Excellent stability
- Reduced Cost and Time-to-Market

## APPLICATIONS

- Automated Meter Reading
- Wireless Sensor Networks
- Home and Building Automation
- Wireless Alarm and Security Systems
- Industrial Monitoring and Control
- Wireless M-BUS
- Internet of Things (IoT) Devices
- Smart Agriculture



## RODUCTS

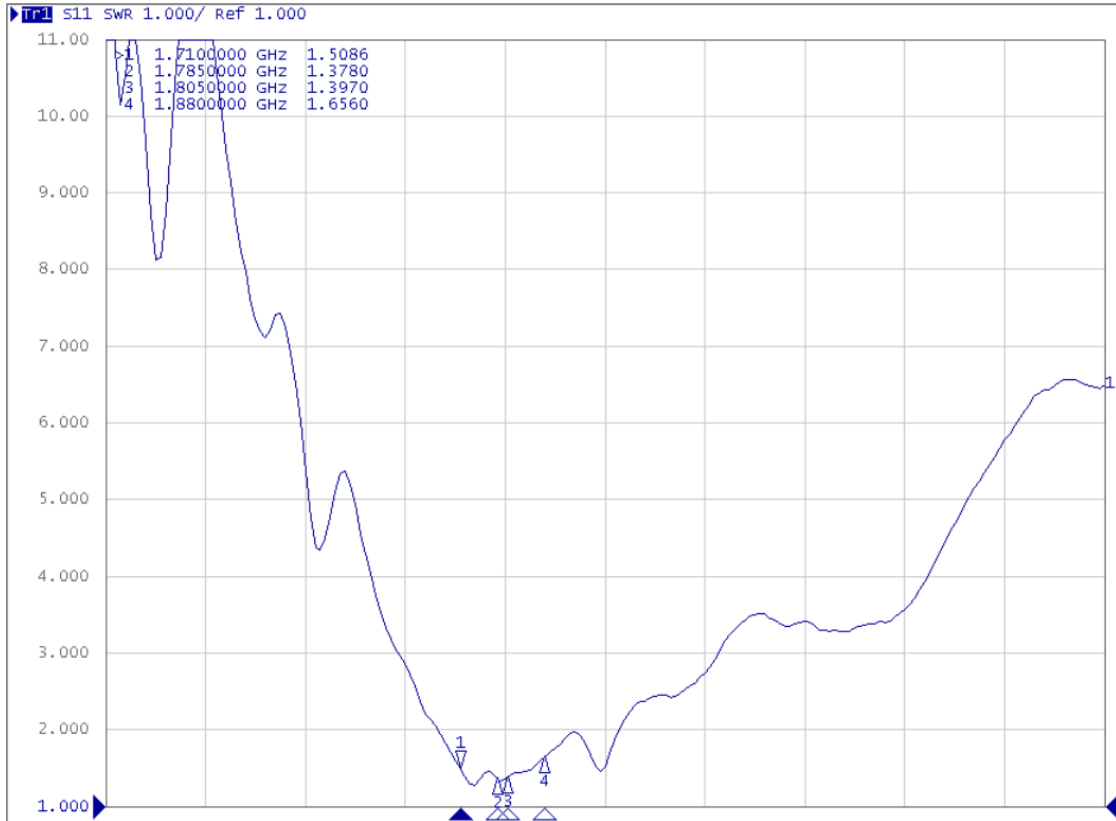
Model	Part No.	Weight	Dimensions (L x W)	Color
AICM002	M04-0101950R0A	0.12g	Φ 5.6*30.29	Silvery

## SPECIFICATIONS

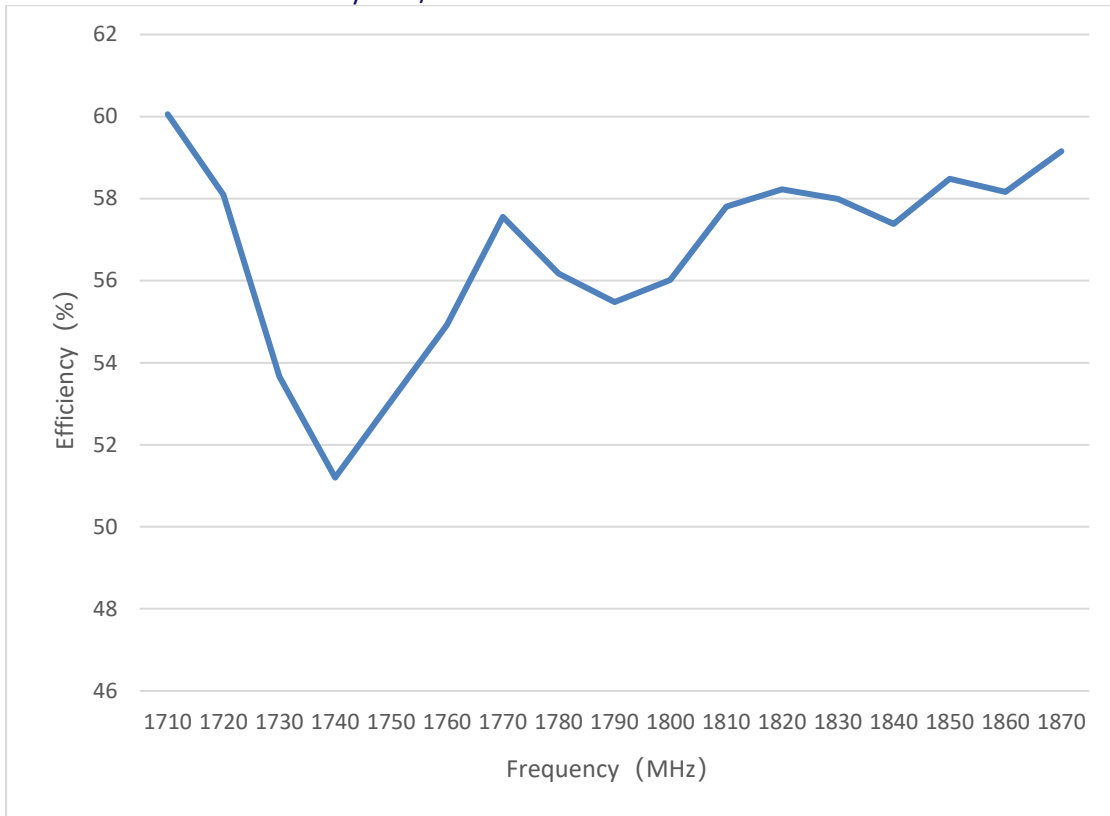
PARAMETER	SPECIFICATION
Frequency Bands, MHz	1710-1880
VSWR (Max)	2.0:1(100*28mm PCB board)
Peak Gain, dBi (Typ)	Up to 1.4(100*28mm PCB board)
Nominal Impedance	50 Ω
Max Power (ambient temp of 25°C)	10 Watts
Azimuth Beam Width (deg)	Omnidirectional
Polarization	Linear, Omnidirectional
Color	Silvery
Storage Temperature Range (°C)	-40° C to +85° C
Operational Temperature Range (°C)	-40° C to +85° C
Material Substance Compliance	REACH/RoHS Compliant

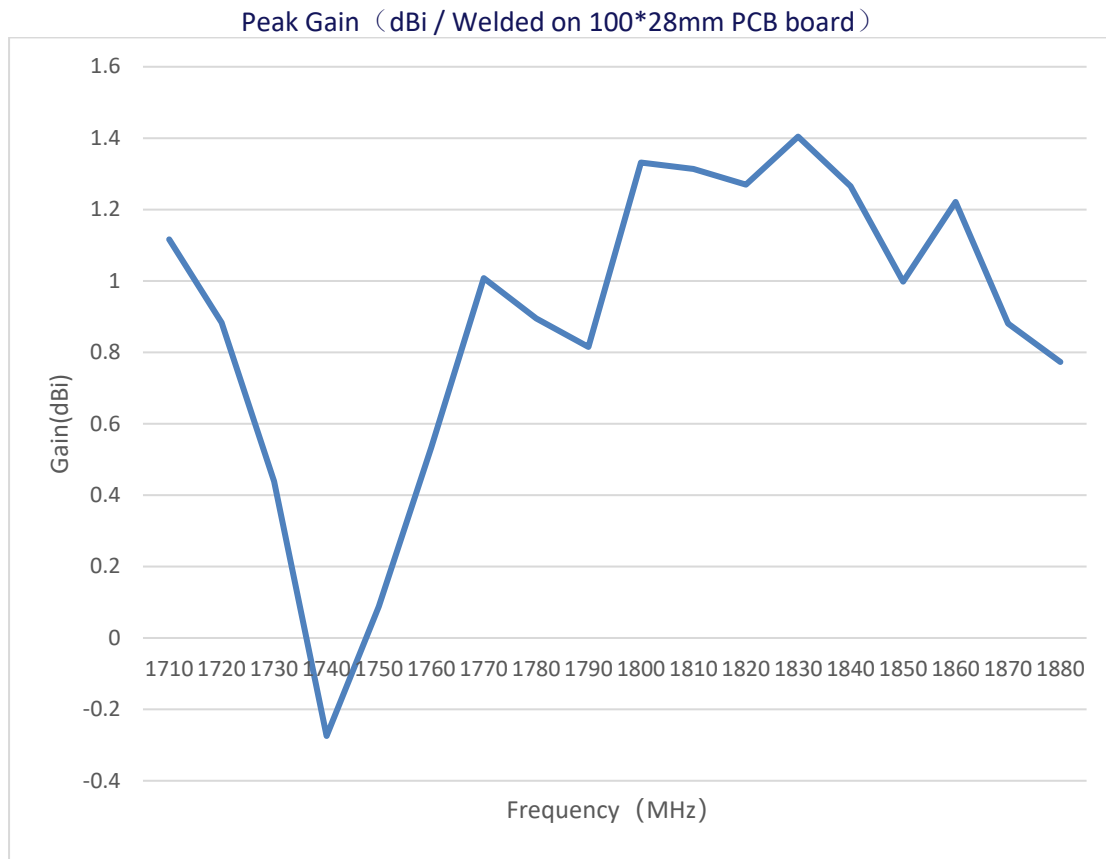
**ELECTRICAL DATA**

V.S.W.R (Welded on 100\*28mm PCB board)



Efficiency (% / Welded on 100\*28mm PCB board)





### ANTENNA APPLICATION DESIGN GUIDE

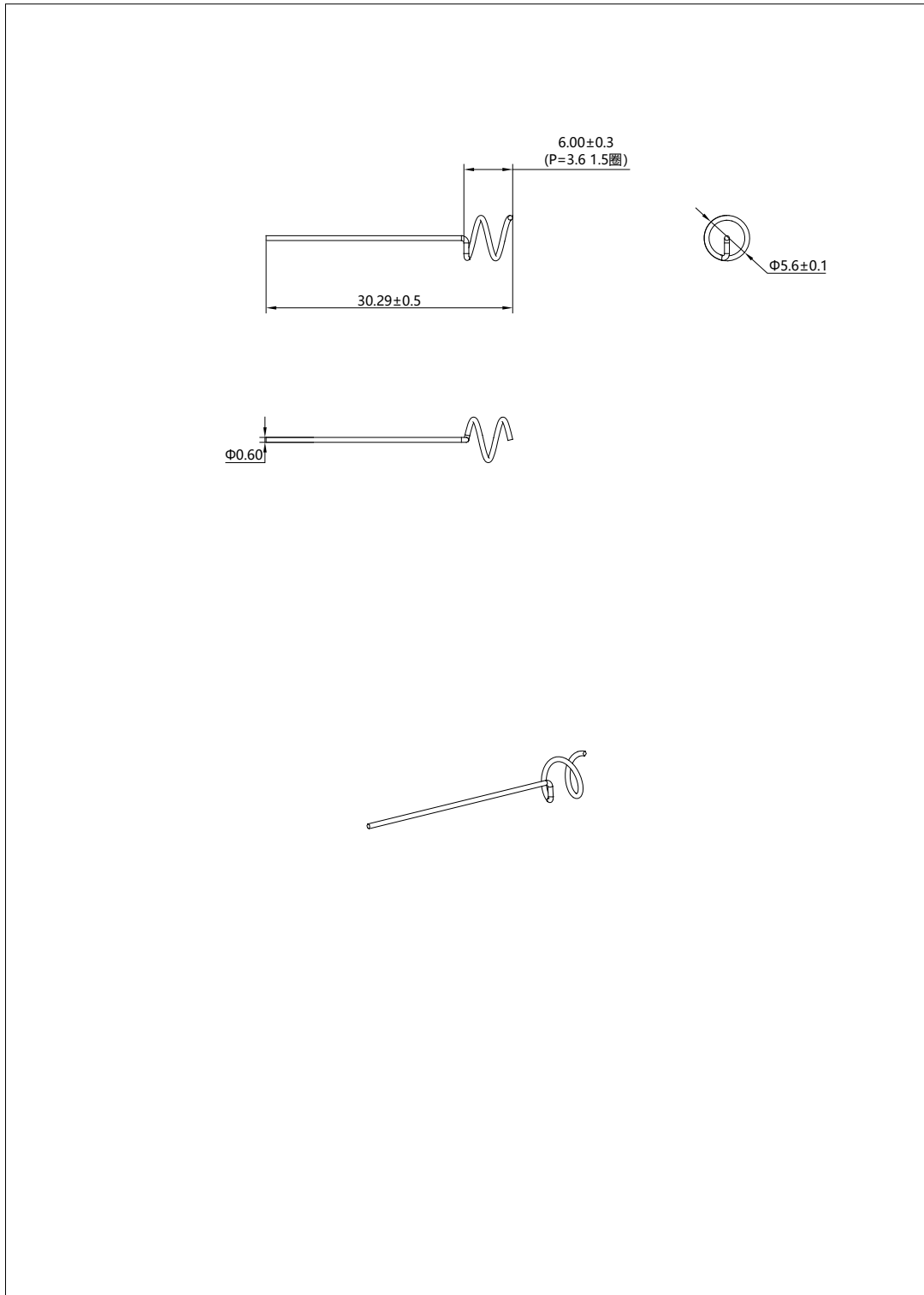
**Note:** In the antenna design process, the placement, angle, ground clearance, and height above the PCB substrate of the spring antenna should be determined by considering the product's structural design, the location of RF module signal input/output interfaces, and internal interference sources.

A  $\pi$ -type matching network should be reserved to facilitate antenna tuning. Ensure the complete product enclosure and internal PCBA functional boards are provided when debugging the antenna. External interference sources and parasitic capacitance should be accounted for in the matching process to achieve optimal antenna performance and efficiency.

The left diagram shows a side view, while the right diagram shows a top view. A recommended trace width of 0.5mm should be used for the PCB traces of the matching network. The ground clearance on both network sides should be 0.35mm to maintain good impedance characteristics.

If you have any questions, please send PCB documents to this e-mail: [support@aboosty.com](mailto:support@aboosty.com)

HOUSING CONFIGURATIONS



**Aboosty™ is owned by Shenzhen MyAntenna RF Technology Co., Ltd. (often abbreviated as MyAntenna).**