

1. Features

- RF Ceramic Bandpass Filter
- Usable bandwidth 210 MHz at Fc 2800 MHz
- RoHS Compliant

RoHS/RoHS2
(2015/863/EU) Compliant

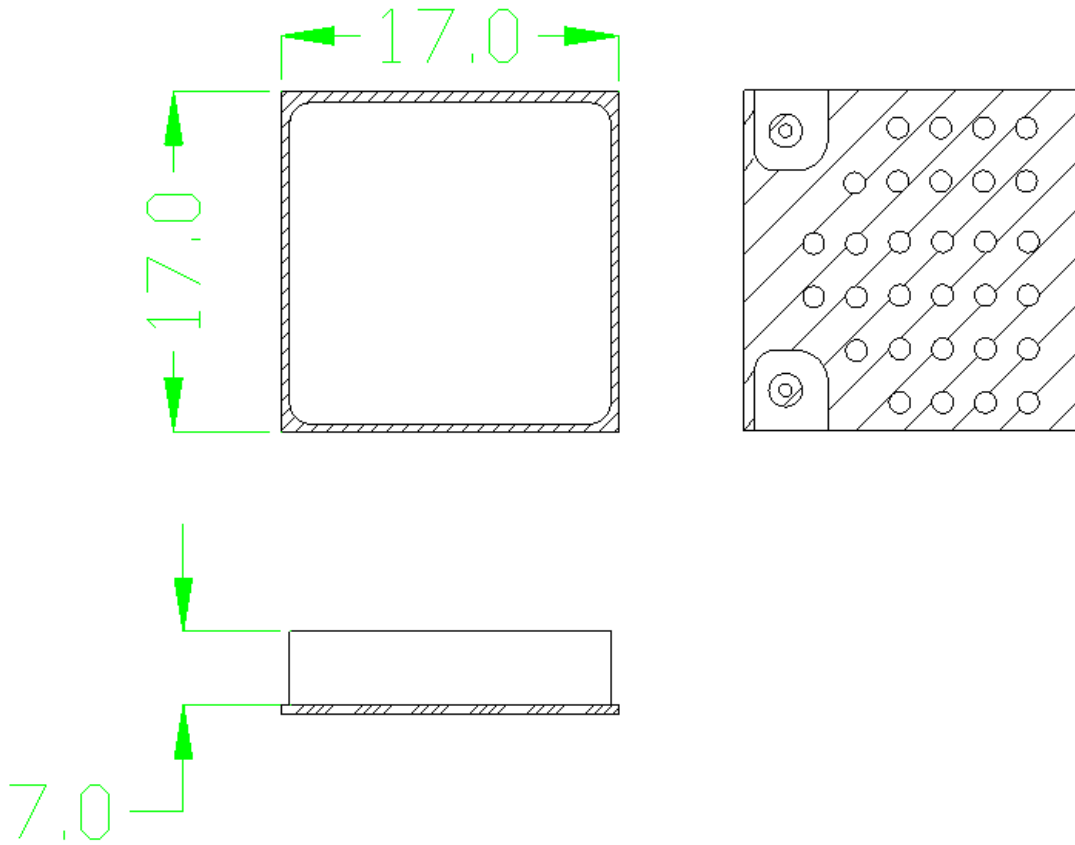
2. Preliminary Electrical Specifications

		Minimum	Typical	Maximum
Center Frequency (Fc)	MHz		2800	
Insertion Loss ¹⁾	dB	-	1.7	2.0
Ripple (2695 - 2905 MHz)	dB	-	0.4	0.5
Return Loss (2695 - 2905 MHz)	dB	15	16	-
Attenuation (DC - 1100 MHz)	dBc	70	75	-
Attenuation (1101 - 2150 MHz)	dBc	55	58	-
Attenuation (2151 - 2250 MHz)	dBc	45	50	-
Attenuation (2251 - 2500 MHz)	dBc	25	35	-
Attenuation (2551 - 2600 MHz)	dBc	10	20	-
Attenuation (3200 - 4000 MHz)	dBc	30	35	-
Attenuation (4001 - 5400 MHz)	dBc	40	35	-
Attenuation (5401 - 7000 MHz)	dBc	50	53	-
Input/Output Impedance	Ohm	-	50	-
Operating Temperature Range	°C	-40		+85
Power handling(CW)	dBm			30
Power handling(Instantaneous Peak)	dBm			33 (1% duty cycle, 1us pulsewidth)

1) Insertion Loss at Center frequency

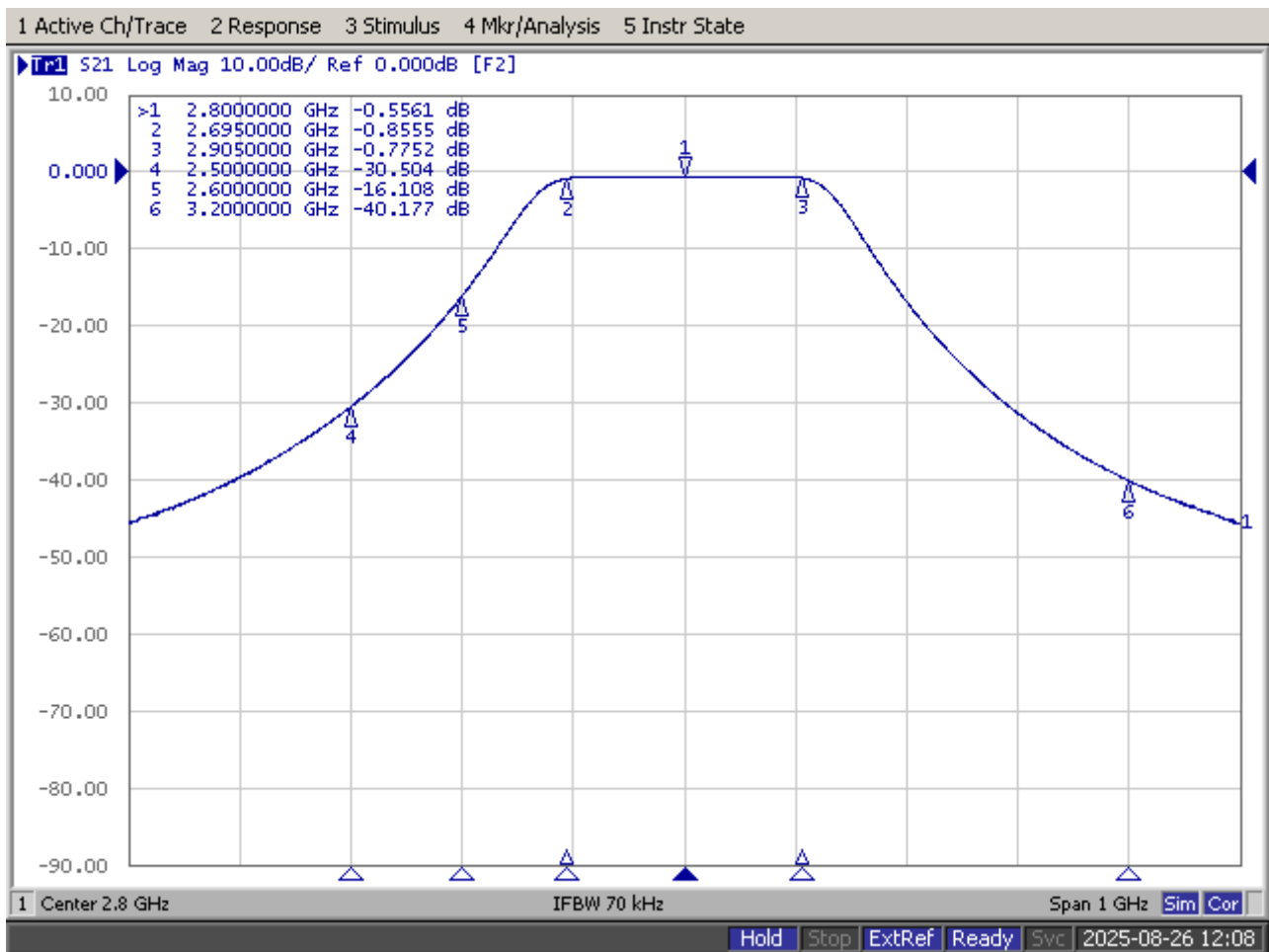
- * All specifications are based on simulation data and ITF reserves the right to amend the specification once sample filters are produced.
- * Actual electrical characteristics may vary due to user's PCB layout and parasitic.
- * Ref. Meas Data.

3. Dimension (17.0 × 17.0 × 7.0 mm)

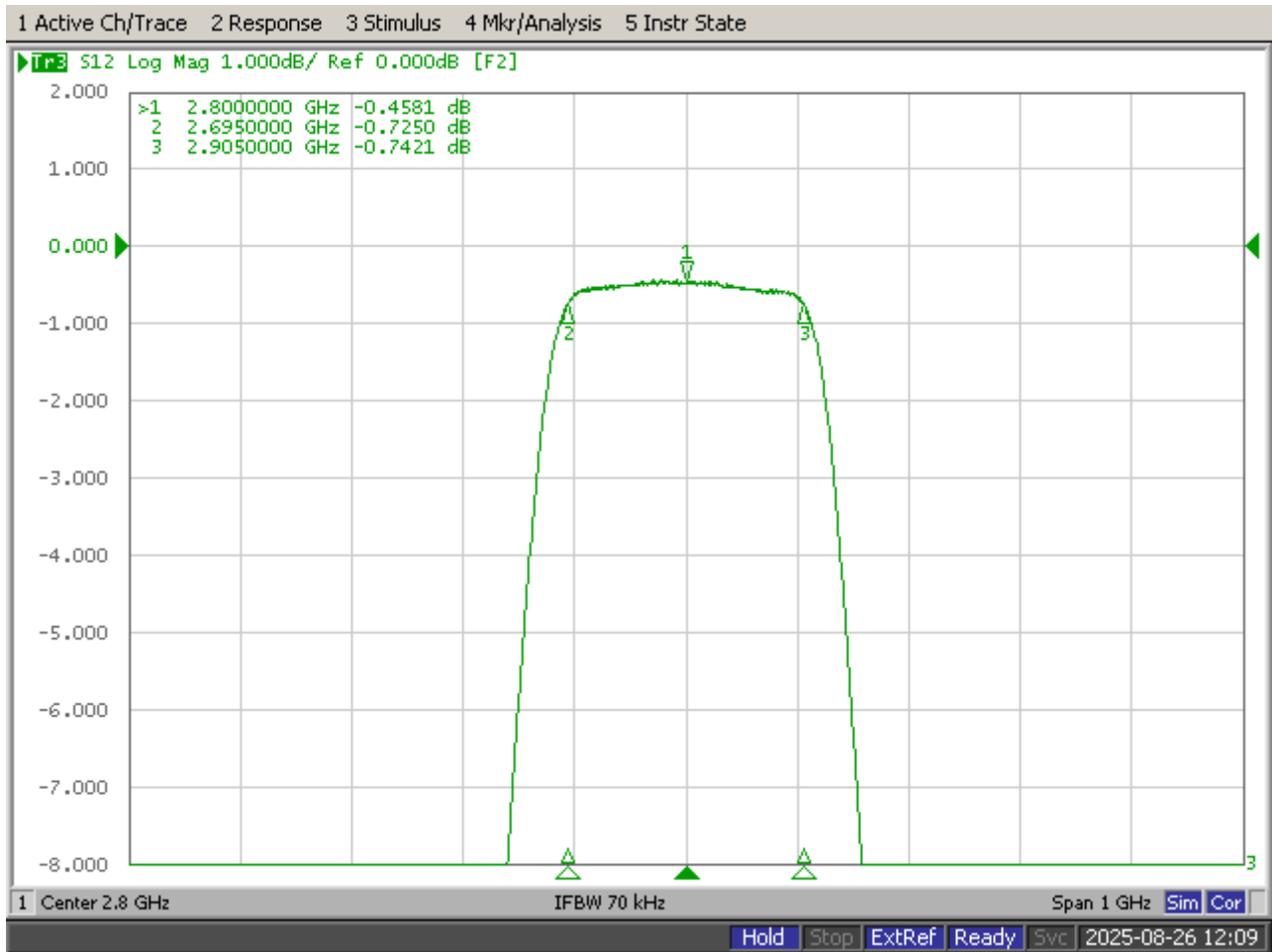


4. Frequency Response (Measurement Data)

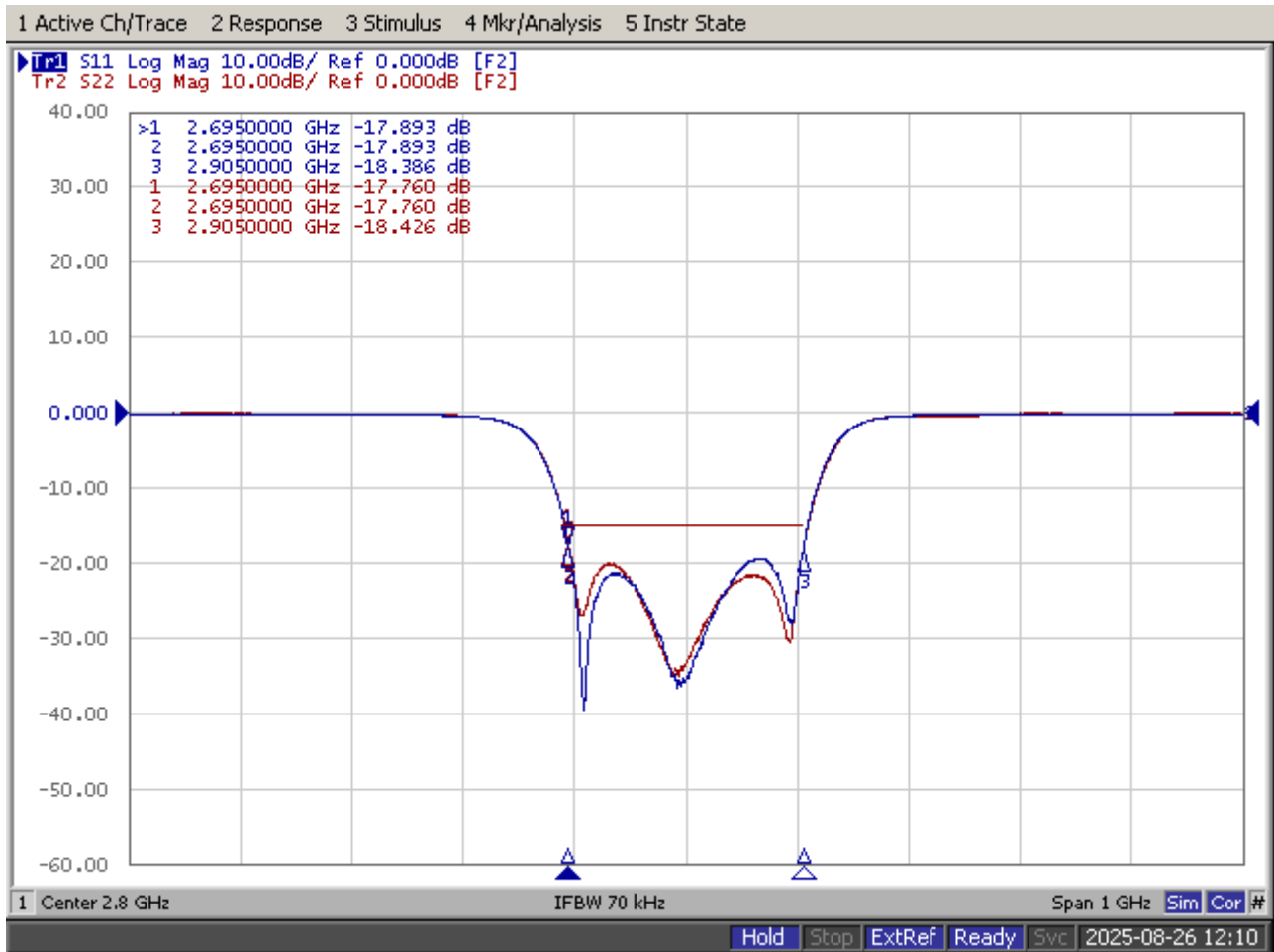
- Insertion loss(S21) & Attenuation



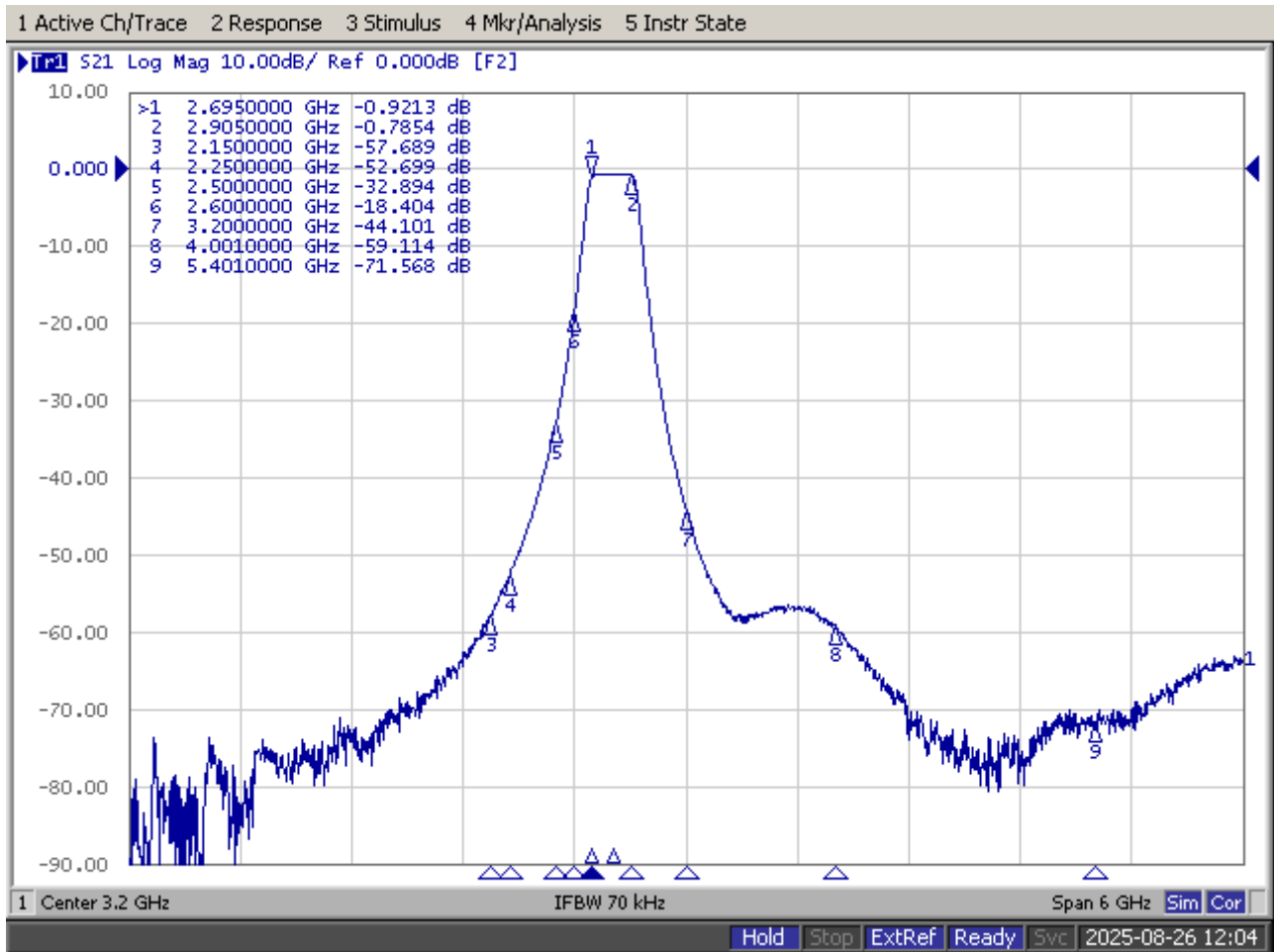
- Ripple



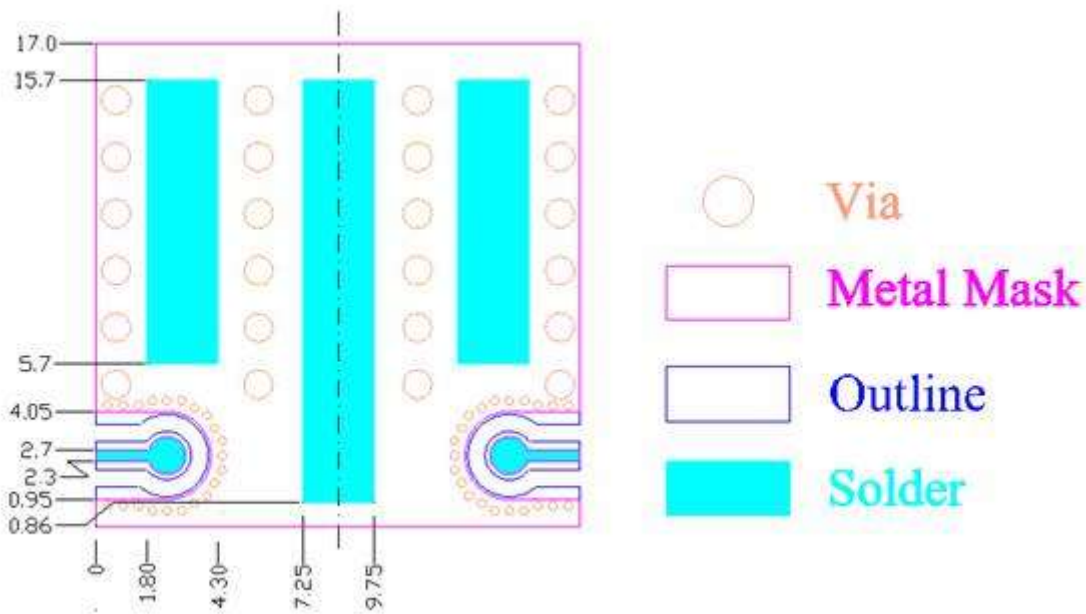
- Return Loss(S11 & S22)



- Wideband



5. Footprint



A. Ground Design

- **Maintain Ground Plane Continuity:**
Ensure continuous ground planes to minimize electrical noise and EMI.
 - **Ground Via Placement Around Critical Signal Paths:**
Provide a sufficient number of ground vias adjacent to critical signal paths to ensure stable grounding.
 - **Recommended Via Diameter:**
Use vias with a diameter of 0.3–0.5 mm, adjusted according to the substrate thickness.
 - **Exclusion of Ground Vias from Solder Mask Openings:**
Do not include ground vias within solder mask openings.
 - **Ground Pattern Dimensioning:**
Design the ground pattern to be 100-200 μm smaller than the bottom ground pattern of the DUT to prevent alignment issues or excessive contact.
-

B. Input and Output Port Design

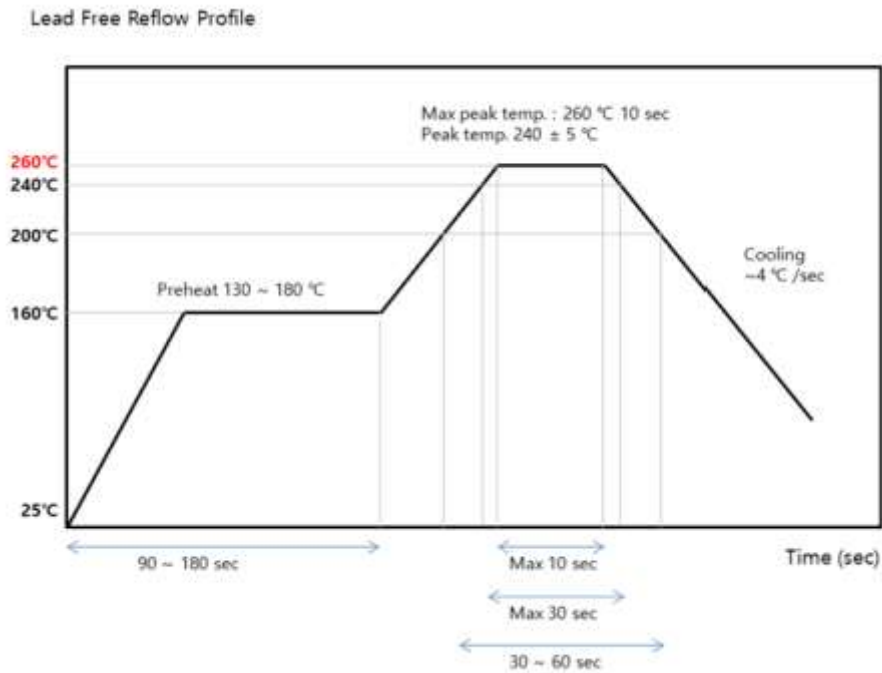
- **Optimize Signal Routing Around Ports:**
Design signal paths near the ports for proper impedance matching and to minimize transmission losses.
 - **Via and Solder Mask Treatment:**
Decisions on whether to remove or retain vias and solder masks near ports should be based on customer specifications.
 - **Adjustment of Trace Widths and Spacing:**
Optimize the trace width and spacing according to the design criteria and substrate characteristics.
 - **Transmission Line and Pad Dimensioning:**
When designing transmission lines or pads, implement them at 100-200 μm smaller than the bottom signal pattern of the DUT to compensate for potential misalignments.
-

Note:

These guidelines should be further refined according to customer specifications and ITF's design recommendations.

6. Reflow profile

- Pre- Heating $155 \pm 25^\circ\text{C}$ (90 ~120 sec.)
- Heating $200 \pm 5^\circ\text{C}$ (30~60sec.)
- Max Peak Temperature 260°C (10sec. Max)



7. Product Picture & EVB

