3. RF

3.1. RF Routing

Both the transceiver connection to the antenna (XCVR_RF) and the GNSS passthrough connection to the GNSS receiver (GNSS_RF_OUT) need to be routed on the host PCB as an impedance controlled $50\ \Omega$ track. For the XCVR_RF signal it is recommended to use either a microstrip or a coplanar waveguide track topology on the host PCB routing to an antenna connector, following the design guidelines and layout requirements in the following sections.

Consult Iridium before adopting a different track topology from microstrip or coplanar waveguide. This will confirm current design guidelines, layout requirements and regulatory advice.

The reference plane for the RF routing on the host PCB should connect to the GND signal on the Iridium 9704 transceiver using the LGA pads adjacent to the RF pad. The RF signals from the transceiver are referenced to this common GND ground plane within the transceiver.

To achieve a $50\ \Omega$ microstrip or coplanar track the copper and substrate dimensions shown in Figure 1 should be considered, along with the dielectric constant, ϵ_r, of the substrate used. An online calculator tool can be used to find the exact dimensions appropriate for the host board's stackup, with Table 1 giving some example values. The impedance control requirements should also be explicitly stated in the design documents provided to the bare PCB fabricator to ensure the dimensions are adjusted to align with the final fabrication processes.

Table 1. Example RF track dimensions for host PCB antenna traces

Track Impedance [Ω]	T	W [mm]	H [mm]	S [mm]	ε_r	Topology
50	1.0 oz/sq-ft Foil + Plating	0.11	0.076	≥ 0.22	4.2	Microstrip
		0.34	0.2	≥ 0.68		
		0.5	0.3	≥ 1.0		
		0.9	0.5	≥ 1.8		
		2.0	1.0	≥ 4.0		
		0.25	≥ 0.5	0.11		Coplanar Waveguide
		0.5	≥ 0.5	0.14		
		1.0	≥ 0.75	0.23		
		1.5	≥ 1.5	0.28		
		2.0	≥ 1.5	0.35		

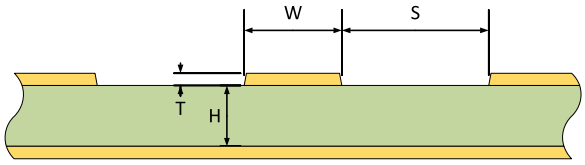

Figure 1. RF Track Dimensions

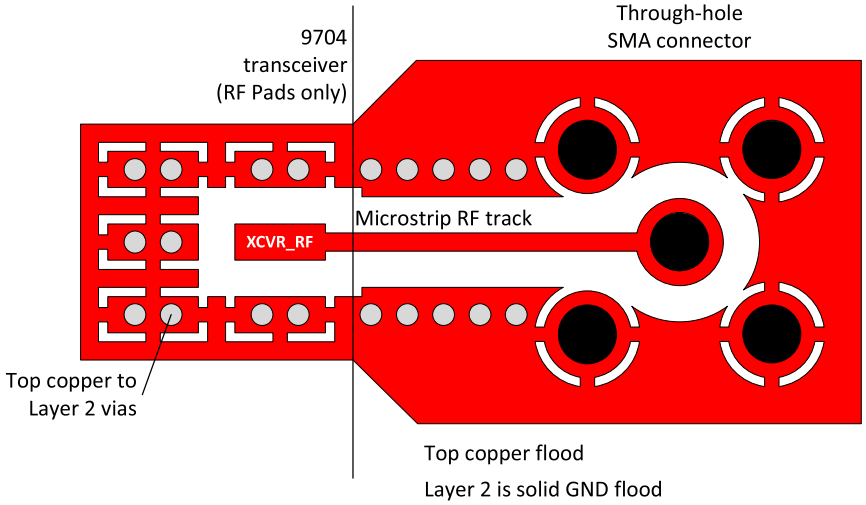

Figure 2. Example Layout For RF Routing Showing Microstrip From XCVR_RF LGA Pad to Antenna Connector

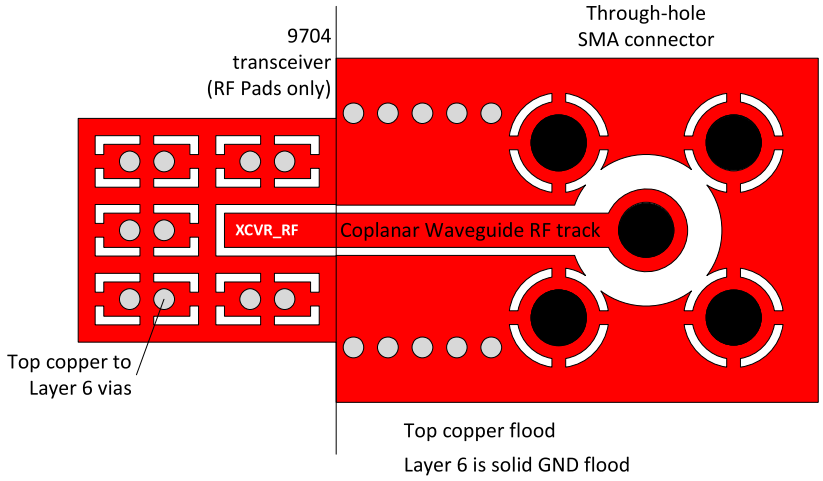

Figure 3. Example Layout For RF Routing Showing Coplanar Waveguide From XCVR_RF LGA Pad to Antenna Connector

The example coplanar waveguide design shown in Figure 3 uses a track width of $1\ mm$, with $0.23\ mm$ spacing to the coplanar ground plane on top copper. Layers 2 through 5 are clear of any vias, tracks or planes under the RF track, to a distance greater than the recommended $0.75\ mm$ separation. Layer 6 is then provided as a complete ground plane, this is more than $0.75\ mm$ below top copper, also following the guidance in Table 1. Whilst a ground flood on layer 6 and the associated via stitching between multiple ground planes is not a mandatory design rule, it is best practice to maintain low impedance in the ground return path.

The microstrip examples given show a simplified PCB with only two copper layers. If a PCB with a higher layer count is to be used, either:

- The reference plane for the RF should be on the adjacent layer to the signal. With H being the substrate thickness between the two layers.
- If the reference plane is not on the adjacent layer, then all the copper on the intermediary layers must be removed (including all tracks, vias and planes) out to beyond the separation distance S. The distance H is then the separation distance between the bottom of the copper track and the top of the reference plane, accounting for all substrate and copper thickness on the intermediary layers.

For coplanar waveguide designs on higher layer count PCBs, all copper on intermediary layers with a spacing to top copper less than indicated in Table 1 must be removed (including all tracks, vias and planes).

As well as the track dimensional requirements, the following should also be met. These are applicable when using either microstrip or coplanar:

- The RF track should be routed directly from the 9704 transceiver to the antenna connection
- The RF track length should be kept as short as possible to minimize transmission line loss
- No discrete components should be placed in the RF route (series or shunt)
- Avoid the use of any right-angle bends in the RF track. If a large direction change is required, this should be achieved using multiple smaller-angled transitions.