

## Footprint Specification Sheet for PCB Mount RF Connectors

SV Microwave requests the information below in order to optimize our connector design for your specific application. The footprint will be designed and simulated per your input. Please fill out the below table and return to the address list above. Thank You.

SV Microwave Part Number <i>(if available)</i>	
Operating Frequency Range (GHz)	
Desired VSWR/Return Loss (dB)	
PCB Type (Stripline, Coplanar Waveguide, Microstrip)	
Connector Series (SMA, SMP, SMPM, etc.)	
Connector Type (Edge Launch, Surface Mount, Etc.)	
Signal Line Impedance (ohms)	
PCB Material Type (Rogers 4003, Megtron 6, Etc.)	
Ground Plane Layer 1 Thickness (T1)	
Substrate Layer 1 Dielectric Constant (Er1)	
Signal Line Width (W - see next page)	
Gap Width (G - see next page)	
Substrate Layer 1 Thickness (S1)	
Ground Plane Layer 2 Thickness (T2)	
Substrate Layer 2 Dielectric Constant (Er2)	
Substrate Layer 2 Thickness (S2)	
Ground Plane Layer 3 Thickness (T3)	
Additional Notes:	

If possible, please attach additional documentation detailing the design features of the PCB including relative layer dimensions and material composition

### Footprint Specification Sheet for PCB Mount RF Connectors

