1. SC14D Reference Design Antenna Product Sheet

Dialog Semiconductor BV has developed the SC14CVMDECT_AF01_SF01 design with the SC14CVMDECT_SFxx module. To supports FAD (e.g. for FP operation), this design includes a printed antenna pattern. This document describes this printed antenna design.

1.1   Quick Specification Sheet

Parameter	Value	Condition
Frequency of Operation (nominal)	1860MHz - 1940MHz	S11 < -8dB
Antenna Type	Printed monopole with integrated transformer	
#layers of antenna design	2	
Antenna Physical Area	4.25mm x 19.00mm	To groundplane edges
Maximum Antenna Gain	0.0 dBi	Carrier 00
	0.0 dBi	Carrier 02
	-0.1 dBi	Carrier 04

2. Description

The printed antenna that is part of the SC14CVMDECT_AF01_SF01 design can also be used directly, as part of the host PCB design.

Figure 1. SC14CVMDECT_AF01_SF01 with external printed antenna design

In that case, the host PCB design must adhere to the same PCB layer stackup (material, layer spacings) as mentioned in section 2.1.1. The antenna must be located in a corner and spaced from the SC14CVMDECT SFxx module in the same way as the SC14CVMDECT_AF01_SF01 design. Also the ground plane design (especially the spacing with respect to the antenna structure) should follow exactly the same outline as in the SC14CVMDECT_AF01_SF01 design. The printed antenna is not 50 Ohms by itself. It requires a printed transformer on the inner layer (indicated in red in the below picture) that is part of the SC14CVMDECT_AF01_SF01 design. The layout of this component is

critical to the antenna performance and should be copied accurately from the available gerber files, obtainable through your Dialog representative.
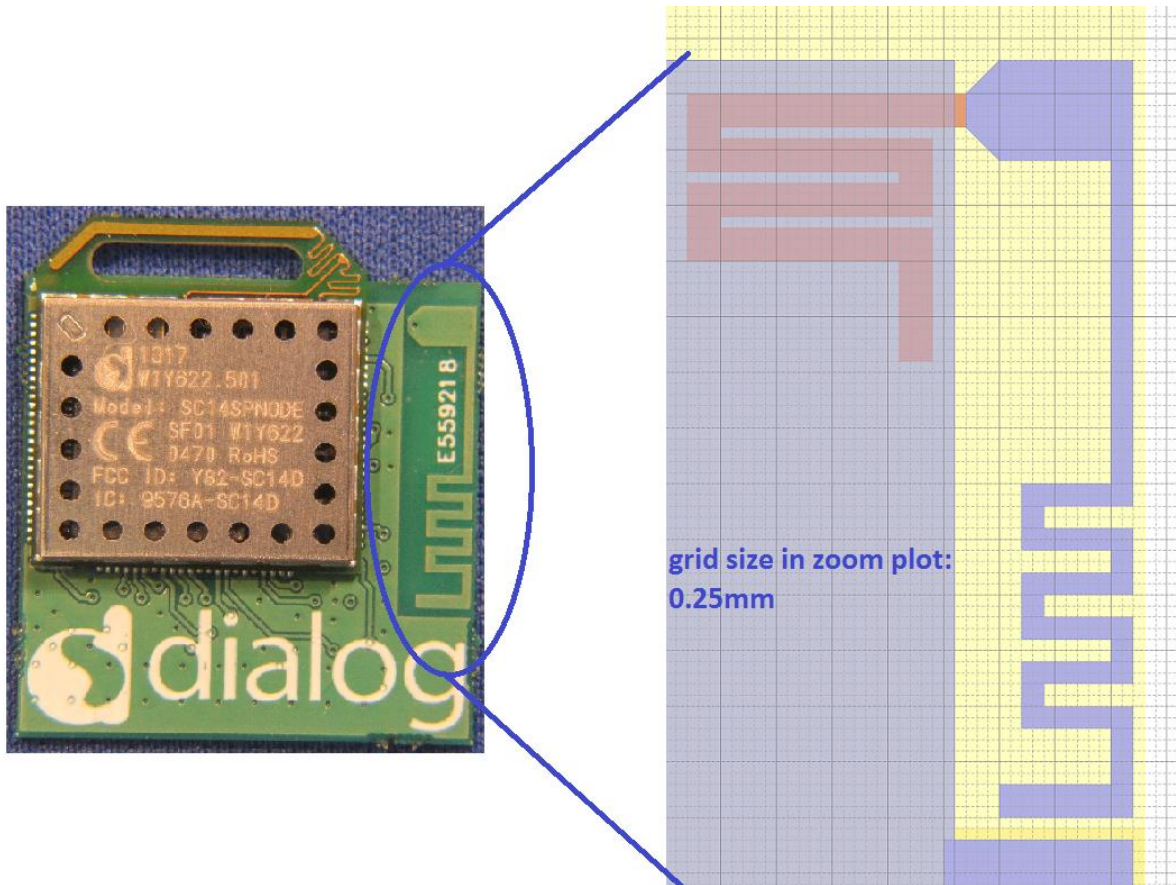

grid size in zoom plot: 0.25mm

Figure 2. Highlight of printed antenna pattern

The transformer should be placed on layer 3. Layer 4 should have continuous ground beneath the transformer pattern. No other signals/traces should be in the vicinity of the transformer structure.
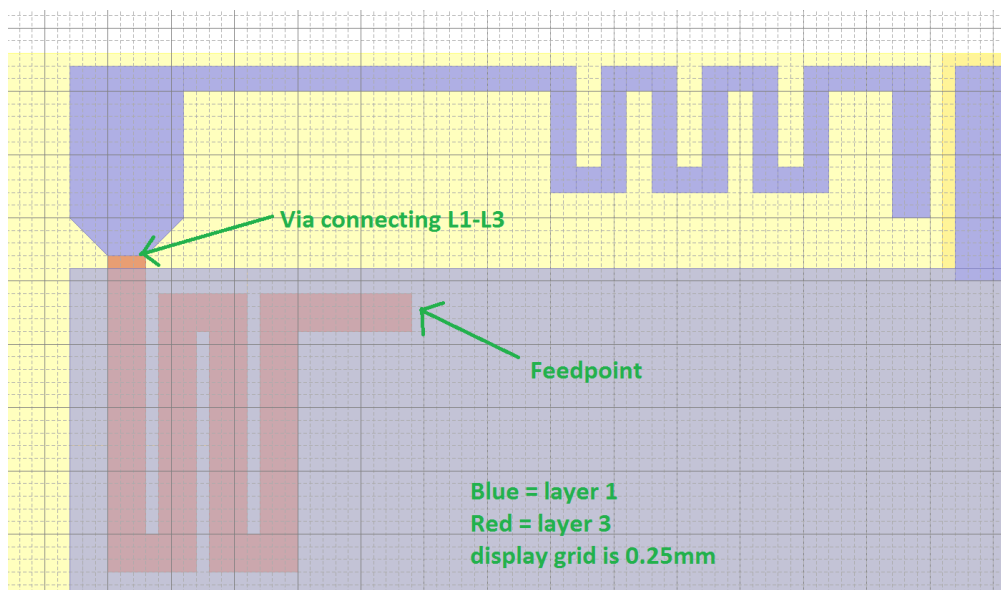

Via connecting L1-L3

Feedpoint

Blue = layer 1
Red = layer 3
display grid is 0.25mm

Figure 3. Details of antenna pattern

2.1.1 Design Parameters

In order to be allowed to leverage the CVMDECT Modular Approval for the SC14CVMDECT_AF01_SF01 design, the PCB should meet the following specifications:

• PCB material: FR4

• Dimensions: length x width x thickness = 25.9mm x 25.0mm x 0.8mm

• PCB layer stackup:

• L1 copper thickness: 18um

• L1-L2 prepreg thickness: 163um

• L2 copper thickness: 35um

• L2-L3 core thickness: 360um

• L3 copper thickness: 35um

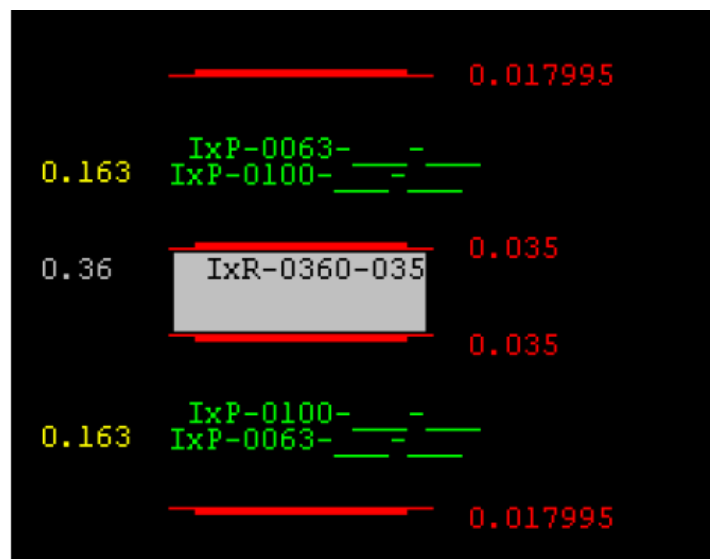• L3-L4 prepreg thickness 163um

• L4 copper thickness: 18um

Figure 4. Required layer stackup

The SC14CVMDECT_AF01_SF01 has been designed using 6mil lines/space and 0.25mm end-size drill holes rules.

When deviating in any way from these specifications, the Modular Approval will be invalidated and recertification on end-product/system level will be required.

2.2 Frequency Bandwidth

The operational frequency bandwidth (S11 < -8dB) has been measured using a calibrated Agilent N9923A and found to be 1860MHz – 1940MHz, as shown in the below measurement plot:
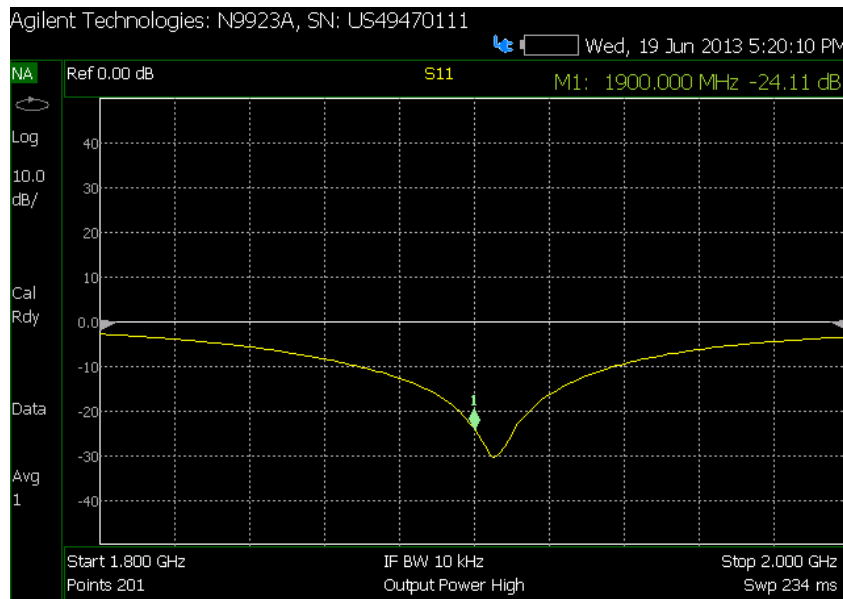

Figure 5. Measured S11 plot of printed monopole

2.3 Maximum Antenna Gain

The maximum antenna gain has been measured by Nemko AS. Next to this, measurement results have been correlated with the simulated peak antenna gain (taking into account additional connection losses) and found to correlate well.

The maximum antenna gain was found to be 0.0dBi nominally.

2.4   Radiation Pattern

An indicative plot of the antenna's radiation pattern is shown here below (from simulation):
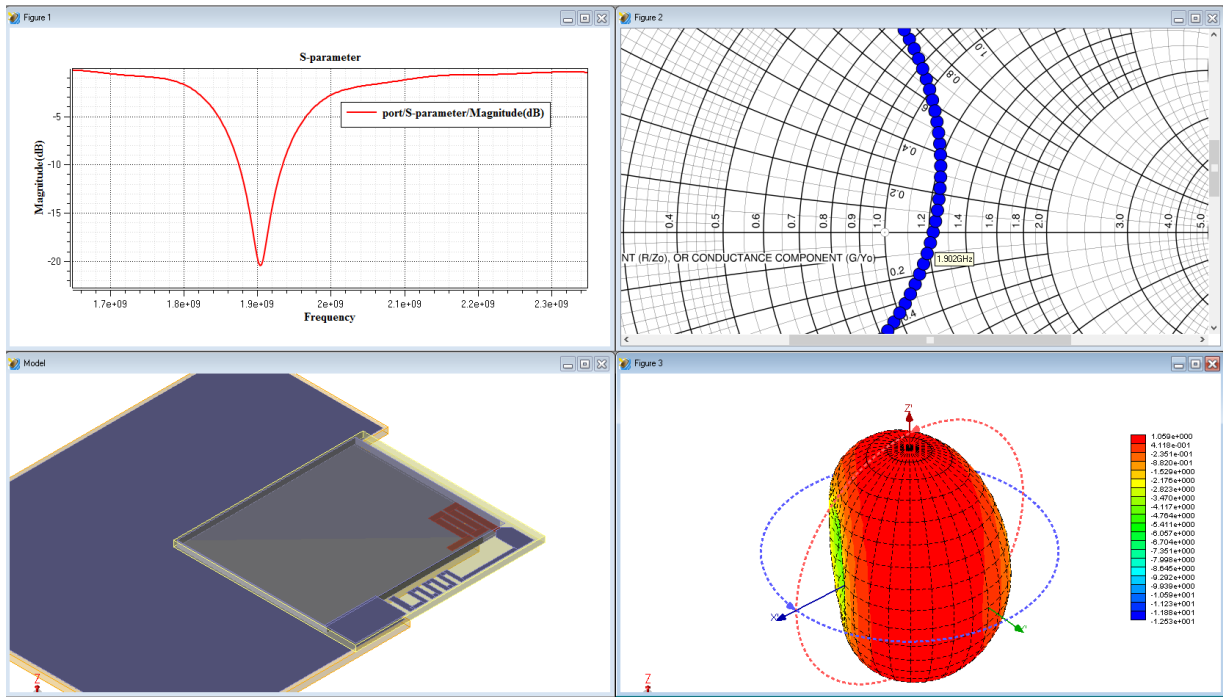

Figure 6. Simulation results of printed monopole

Note that the simulated antenna gain is approximately 1dB higher than measured. This is because the simulations didn't take into account structure losses.