SIM900-TE PCB Layout & Schematic for Reference

AN_ SIM900-TE PCB Layout & Schematic for Reference _V1.01

Document Title:	SIM900-TE PCB Layout for Reference
Version:	1.01
Date:	2010-3-30
Status:	Release
Document Control ID:	AN_SIM900-TE PCB Layout for Reference_V1.01

General Notes

SIMCOM offers this information as a service to its users, to support application and engineering efforts that use the products designed by SIMCOM. The information provided is based upon requirements specifically provided to SIMCOM by the users. SIMCOM has not undertaken any independent search for additional relevant information, including any information that may be in the user's possession. Furthermore, system validation of this product designed by SIMCOM within a larger electronic system remains the responsibility of the user or the user's system integrator. All specifications supplied herein are subject to change.

Copyright

Content

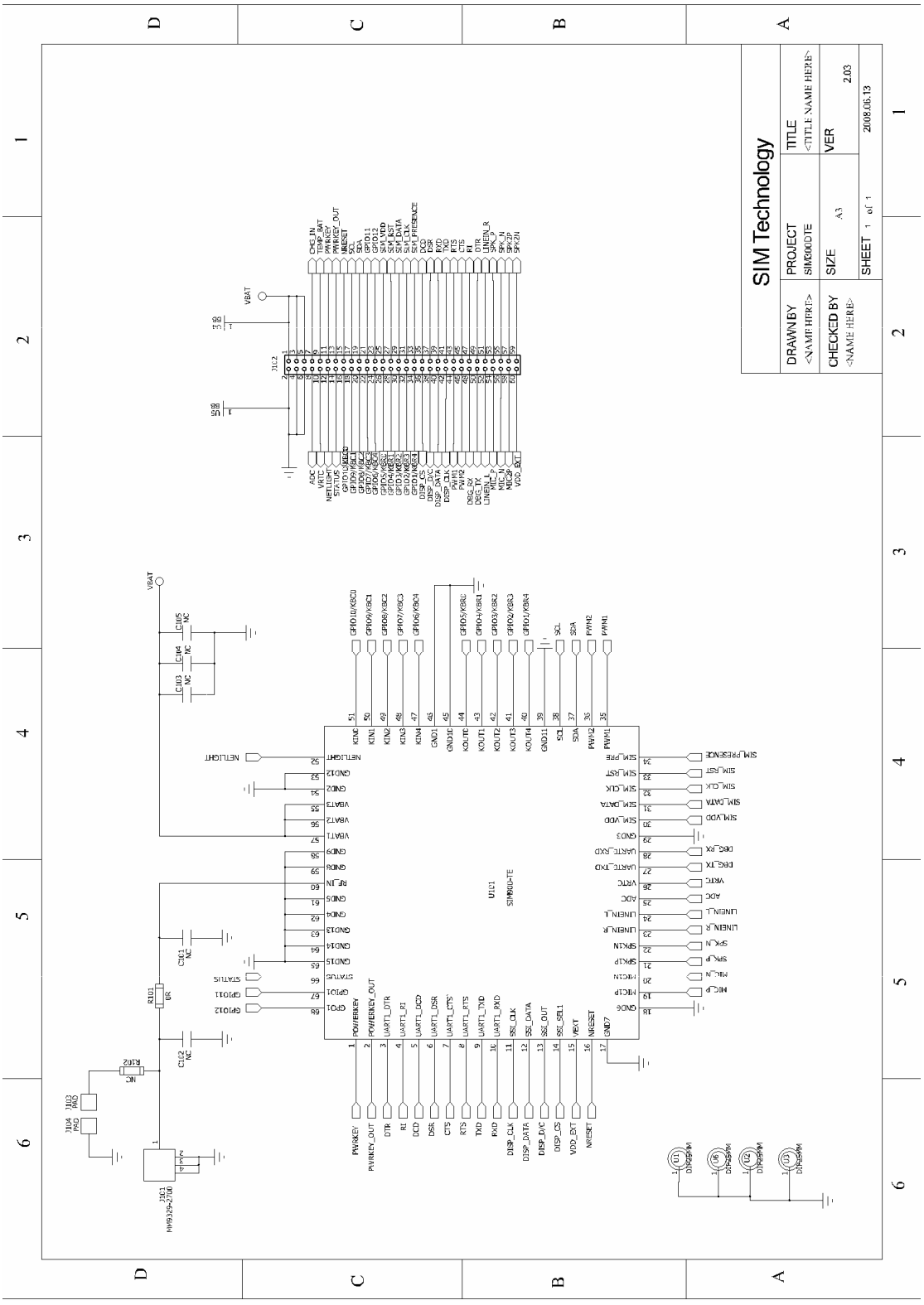
Version History

Data	Version	Description of change	Author
2010-3-30	01.01	Origin	Ye Haibing, Wang Guoqiang

1 Introduction

This document shows the detailed information about SIM900-TE PCB Layout and Schematic.

2 Schematic



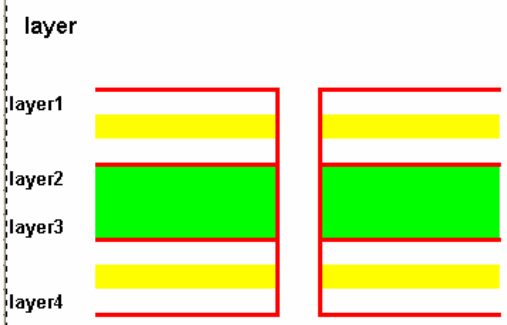
Note: In this schematic, the resistor R102 is a option for choosing either a GSC type coaxial RF cable（MXTK series, vended by Murata） or a soldered coaxial RF cable. The R102 is mounted with

a 0ohm resistor, a soldered coaxial RF cable can be chose for antenna connection, and if the R102 is not mounted, the antenna should be connected via a USC type coaxial RF cable.

3 PCB Layout

3.1 The SIM900-TE PCB's stack up

The board thinkness is 1.0mm.



number	Details for calculation	Total thick-ness average	After press nominal
	soldermask		10(min)
layer1	Copper	12+plating	25~48
	PP	76	70
layer2	copper	18+plating	25~48
	core0.665mm	665	665
layer3	copper	18+plating	25~48
	PP	76	70
layer4	Copper	12+plating	25~48
	soldermask		10(min)
TOTAL			1000

unit:um

finish thickness =1.0mm

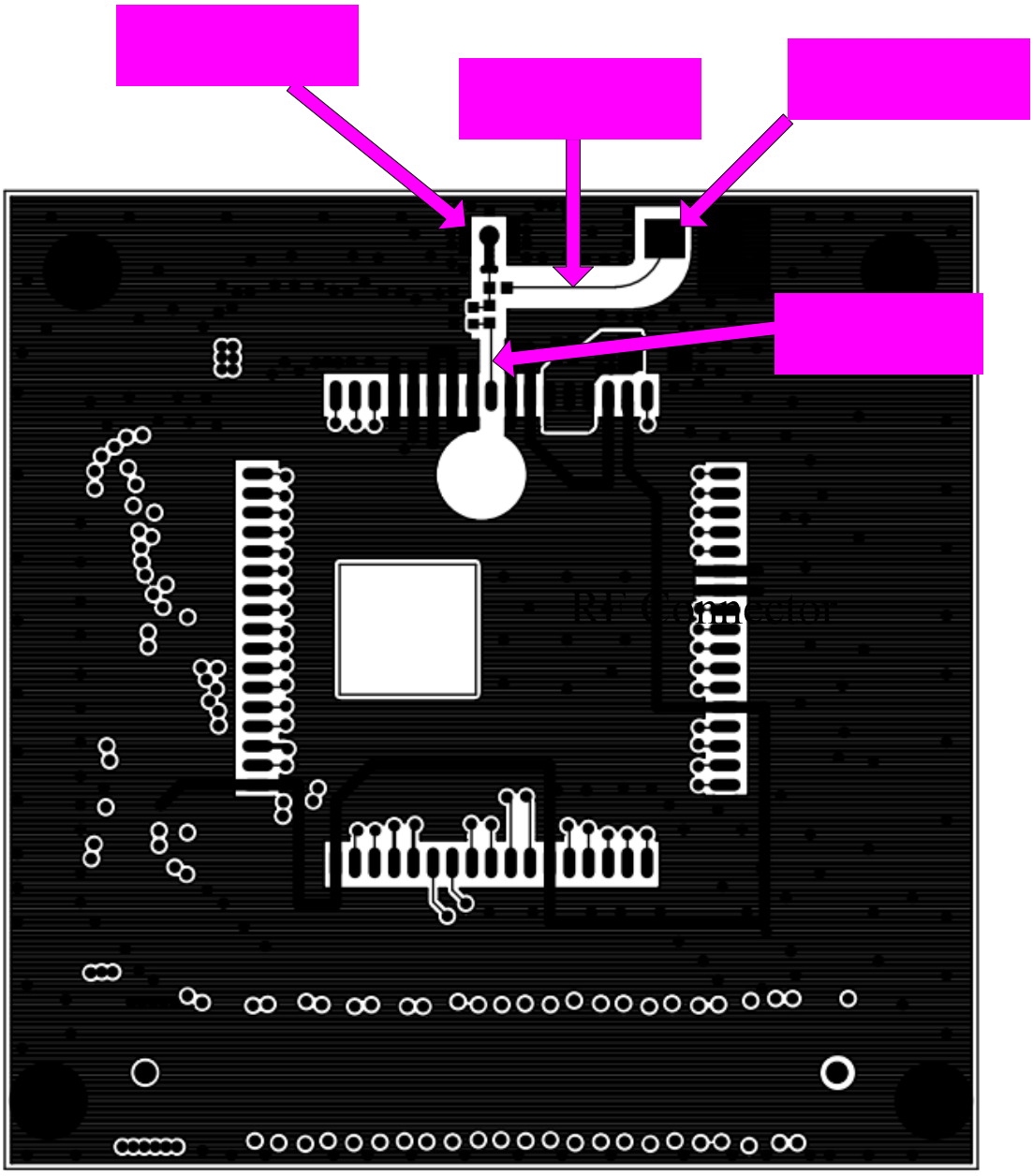
The SIM900-TE is a four layer PCB, the PCB's total thickness is 1.0mm, the clearance between the first layer and the second layer is 0.076mm, the clearance between the second layer and the third layer is 0.665mm.

The RF trace is routed on the top layer, and the second layer is the reference ground layer, for the clearance between the top layer and the second layer is only 0.076mm, so the RF trace's width is 0.11mm.

3.2 The SIM900-TE's PCB layout

The following picture are the detailed PCB layout of SIM900-TE.

Top Layer

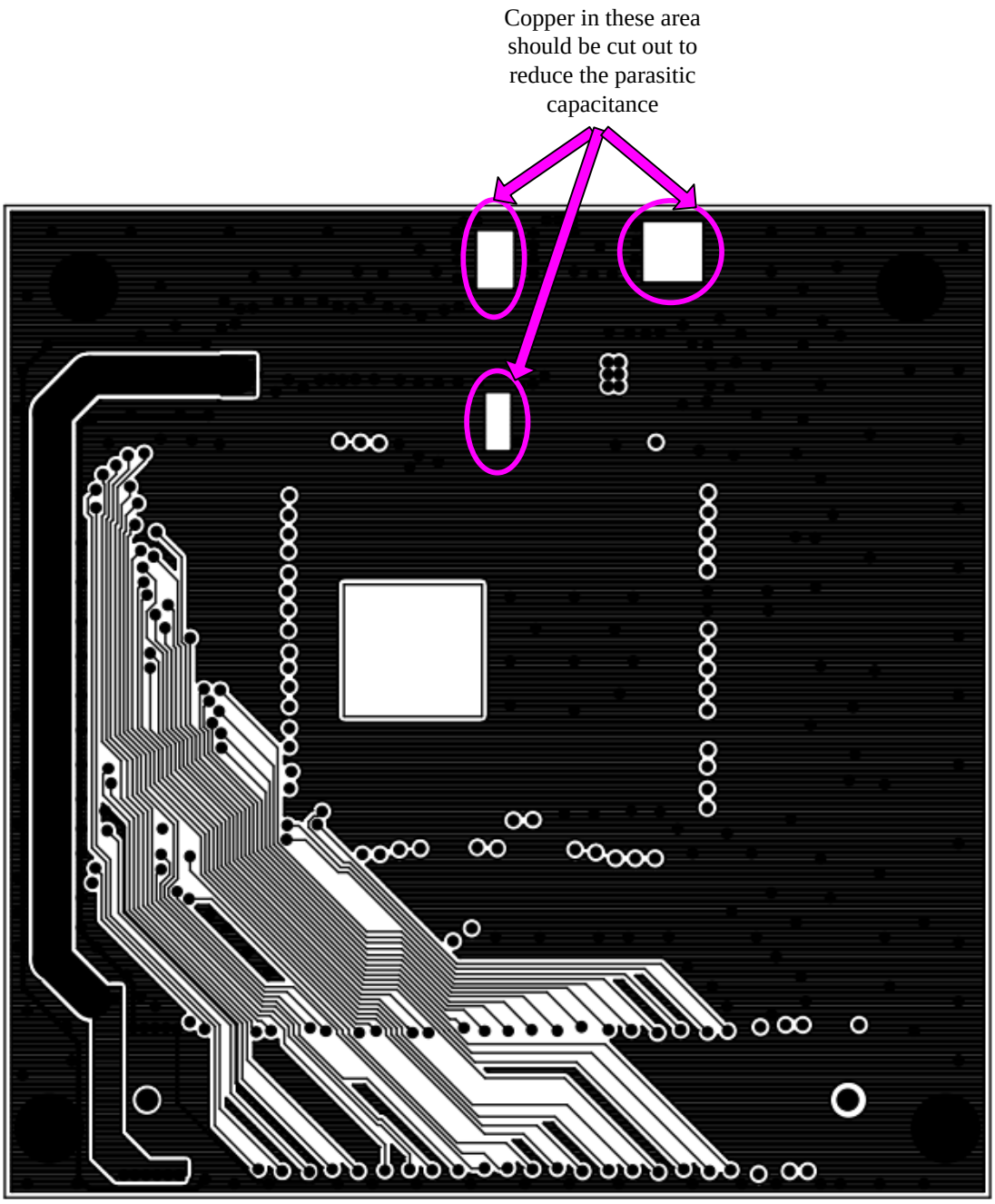


All the RF traces are 50ohm impedance controlled.

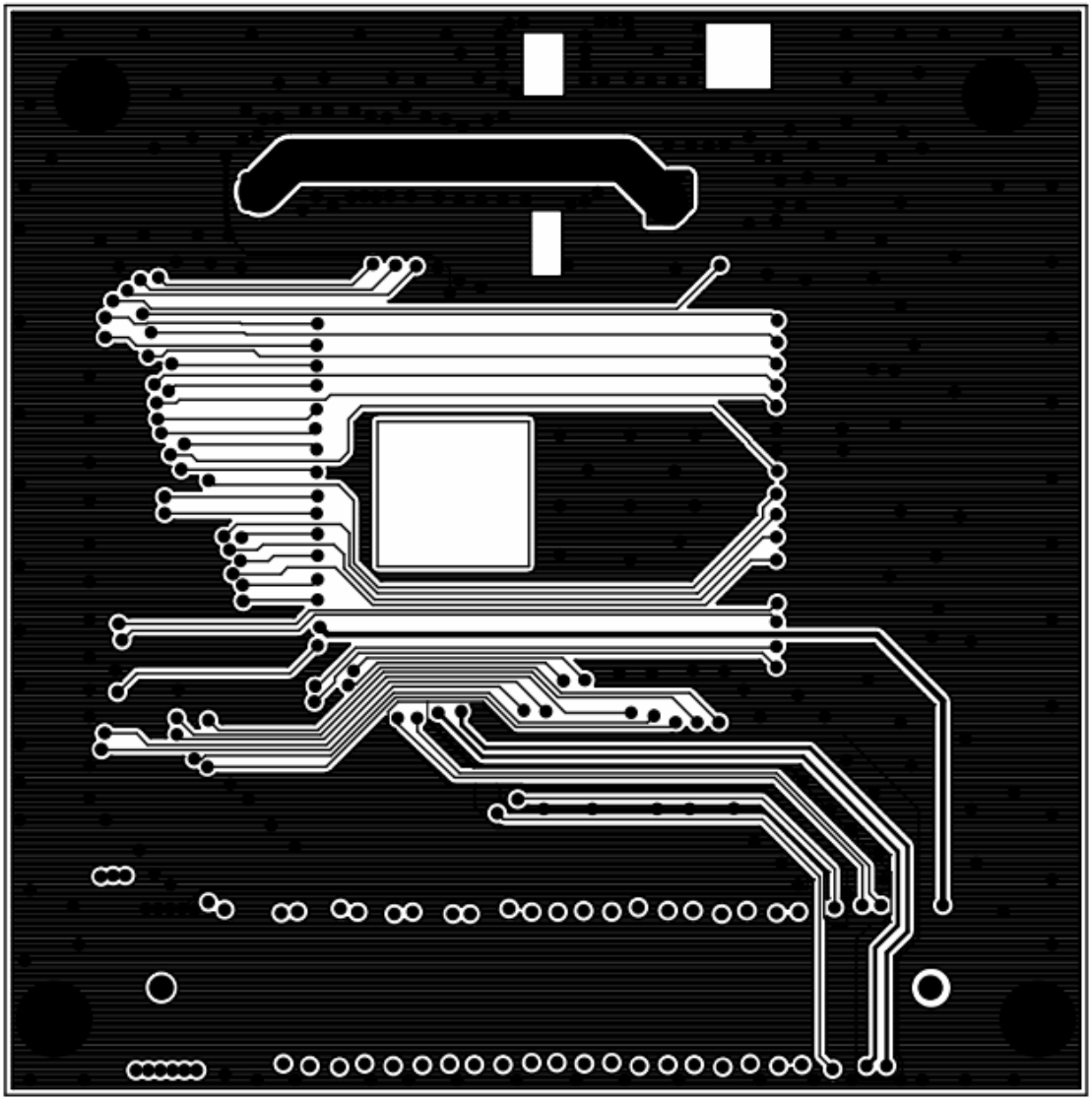
RF connector is used for connect with matched RF plug cable assembly, the RF cable should be 50ohm impedance controlled coaxial cable.

RF PAD is used for connect with solderable RF coaxial cable assembly, the RF cable is also should be 50ohm impedance controlled.

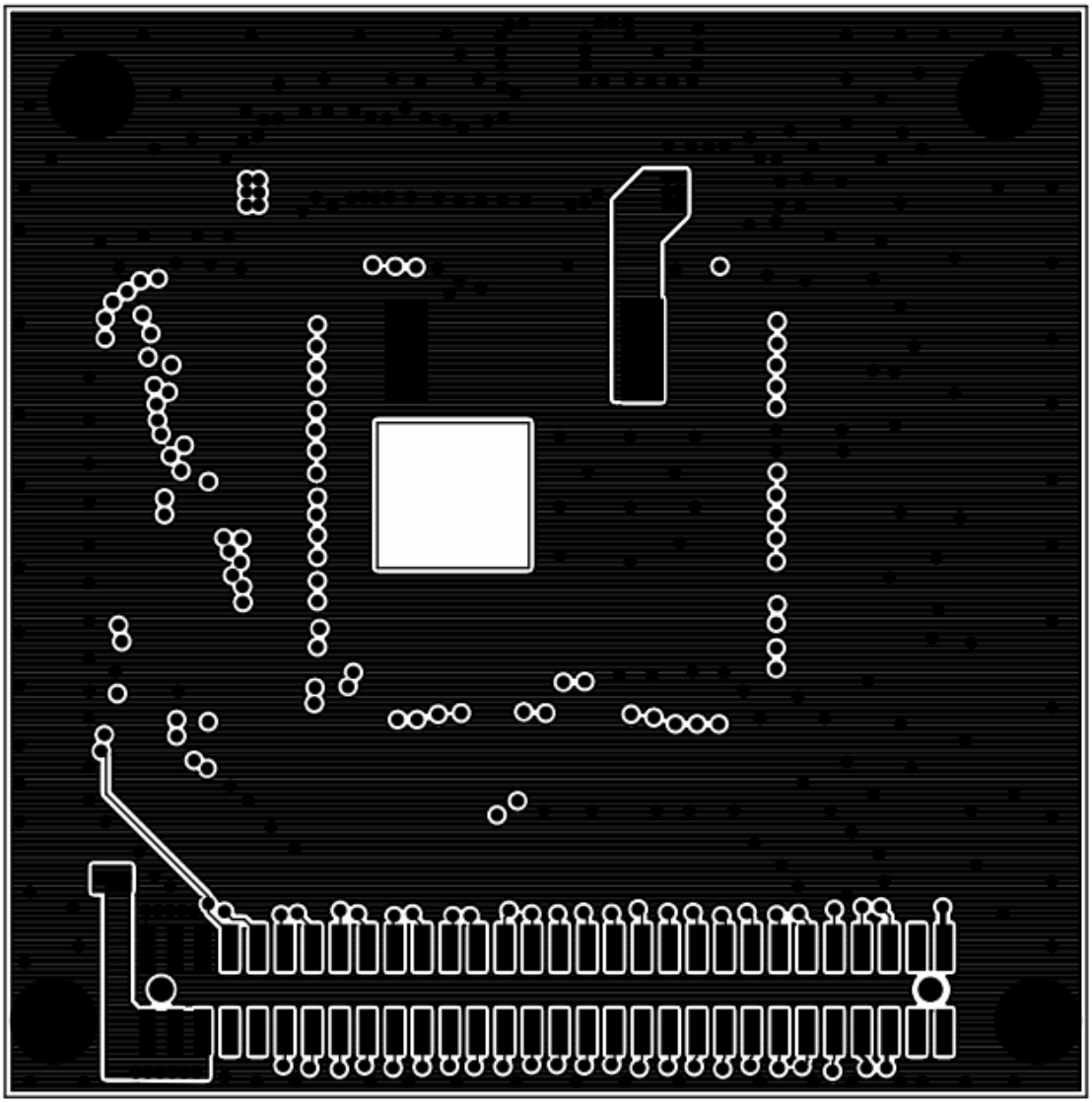
Layer 2


Copper in these area
should be cut out to
reduce the parasitic
capacitance

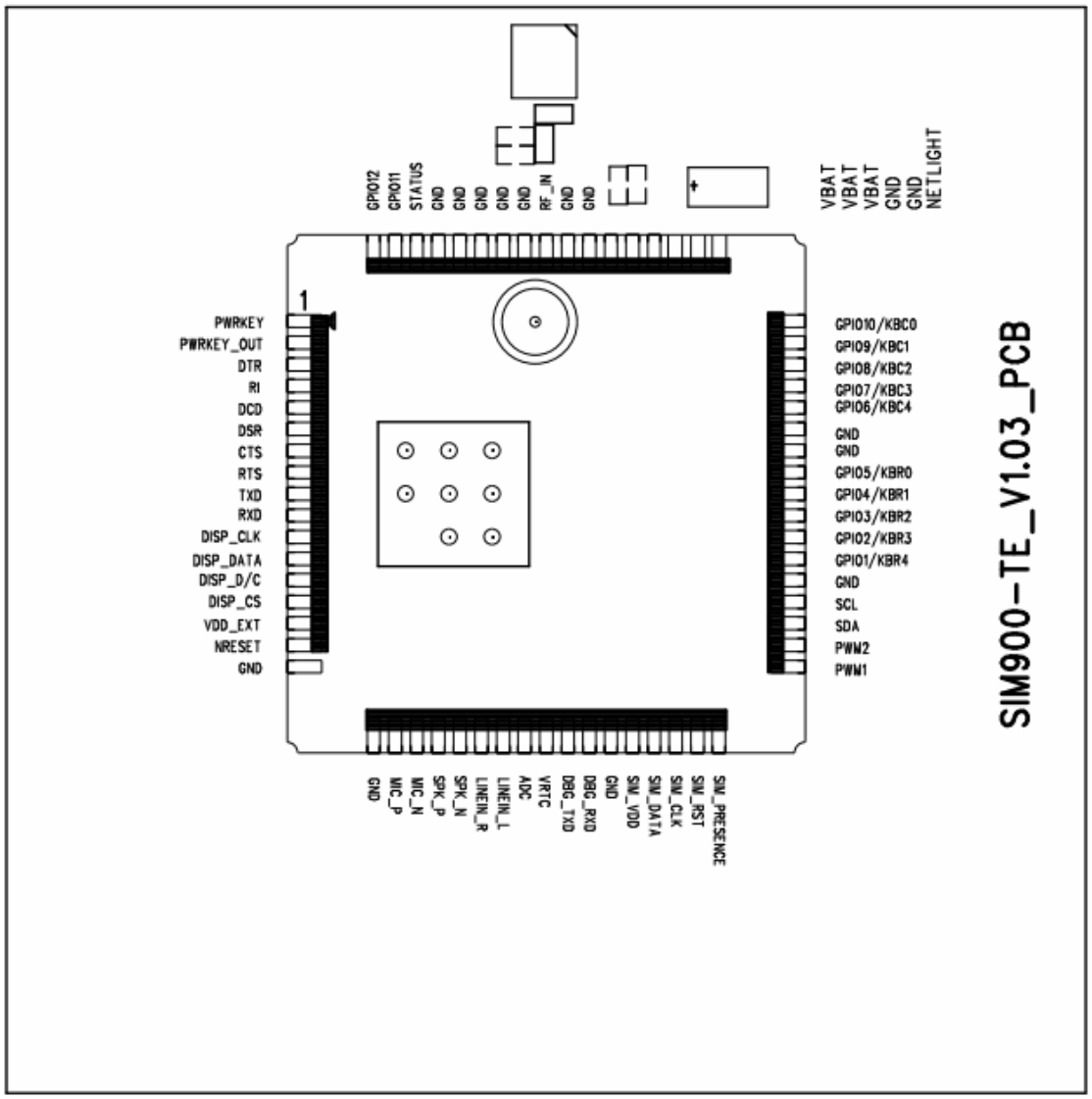
Layer 3



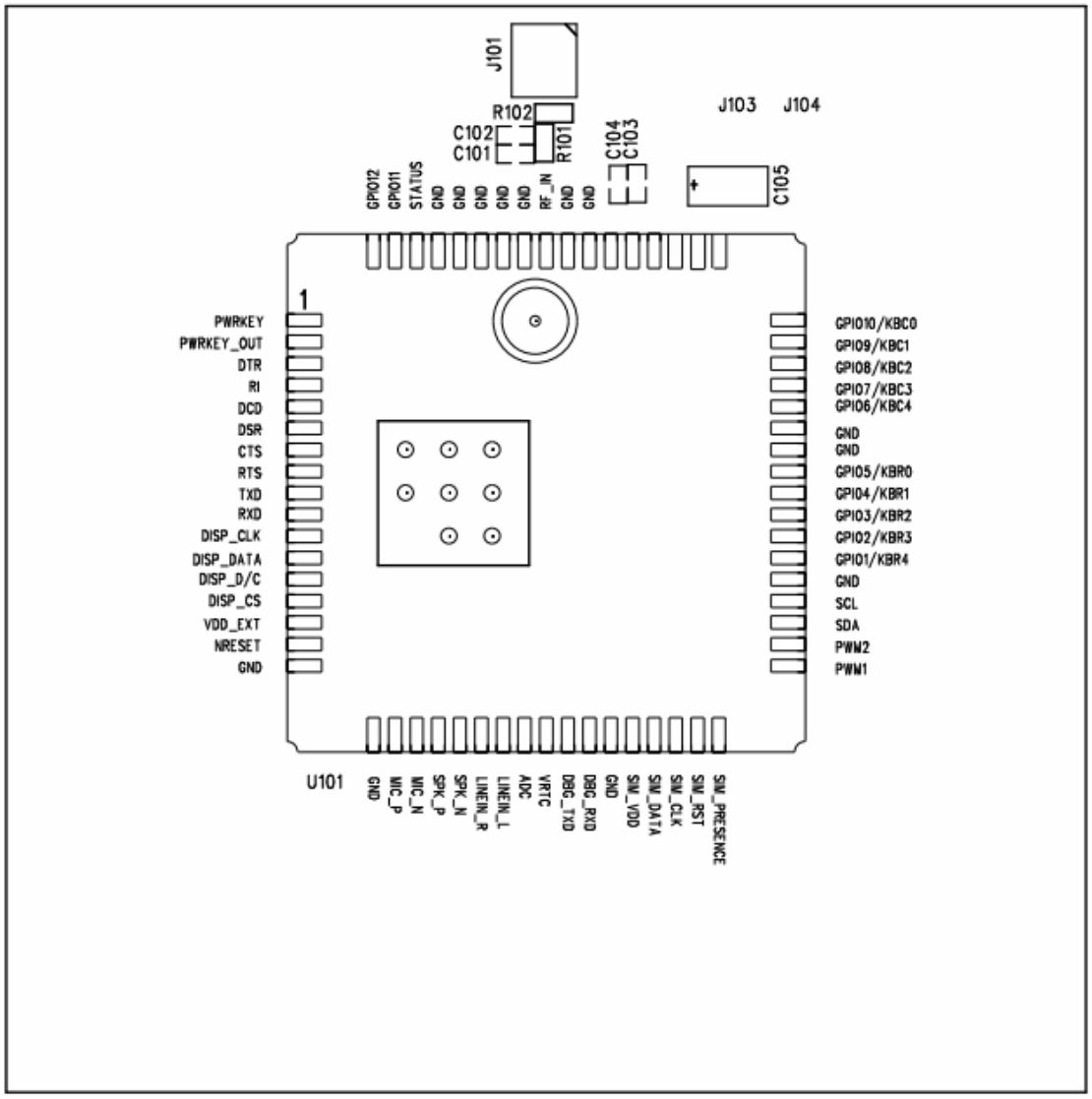
Bottom layer



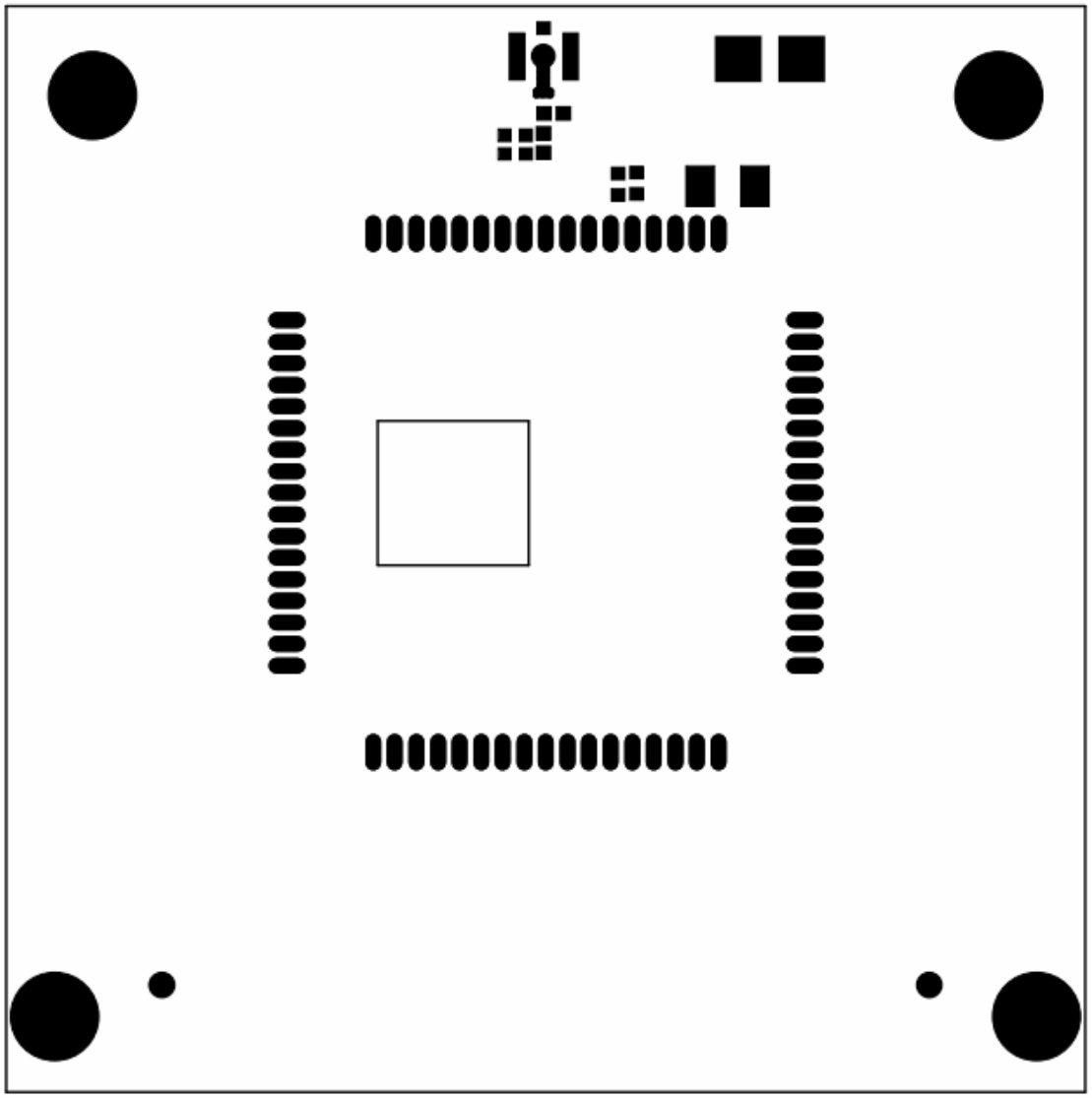
Silkscreen Top



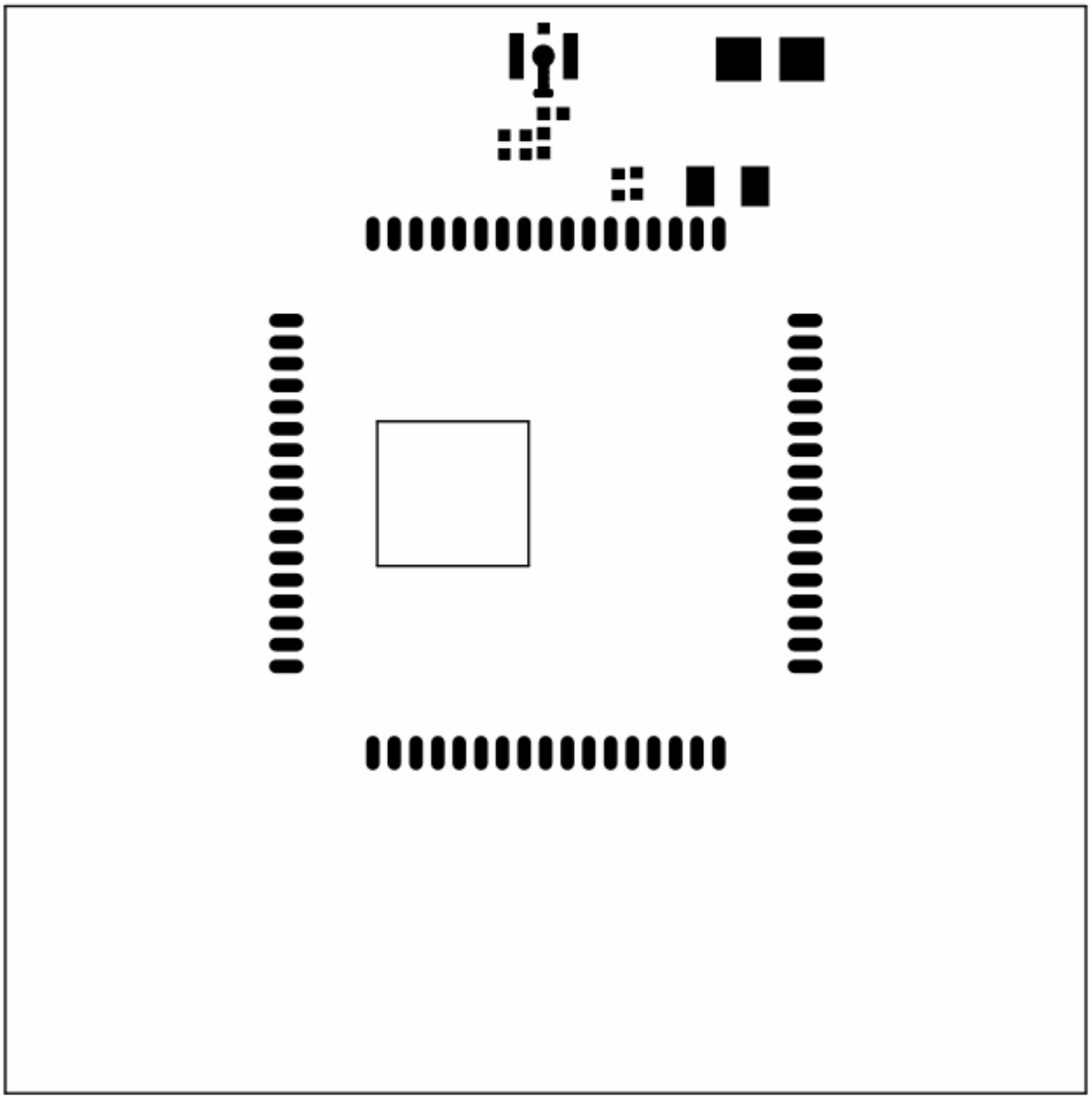
Reference designator Top



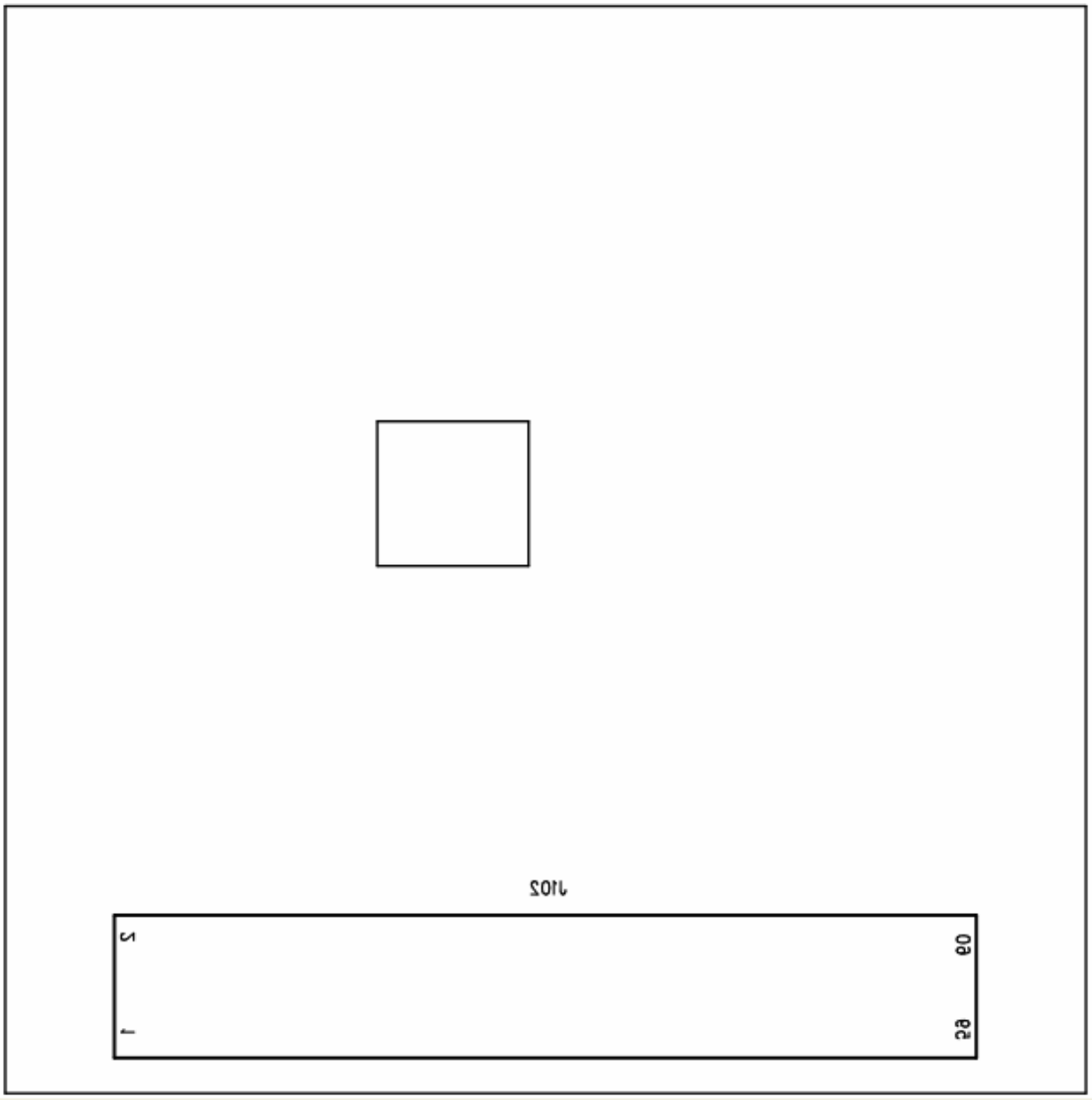
Solder mask Top



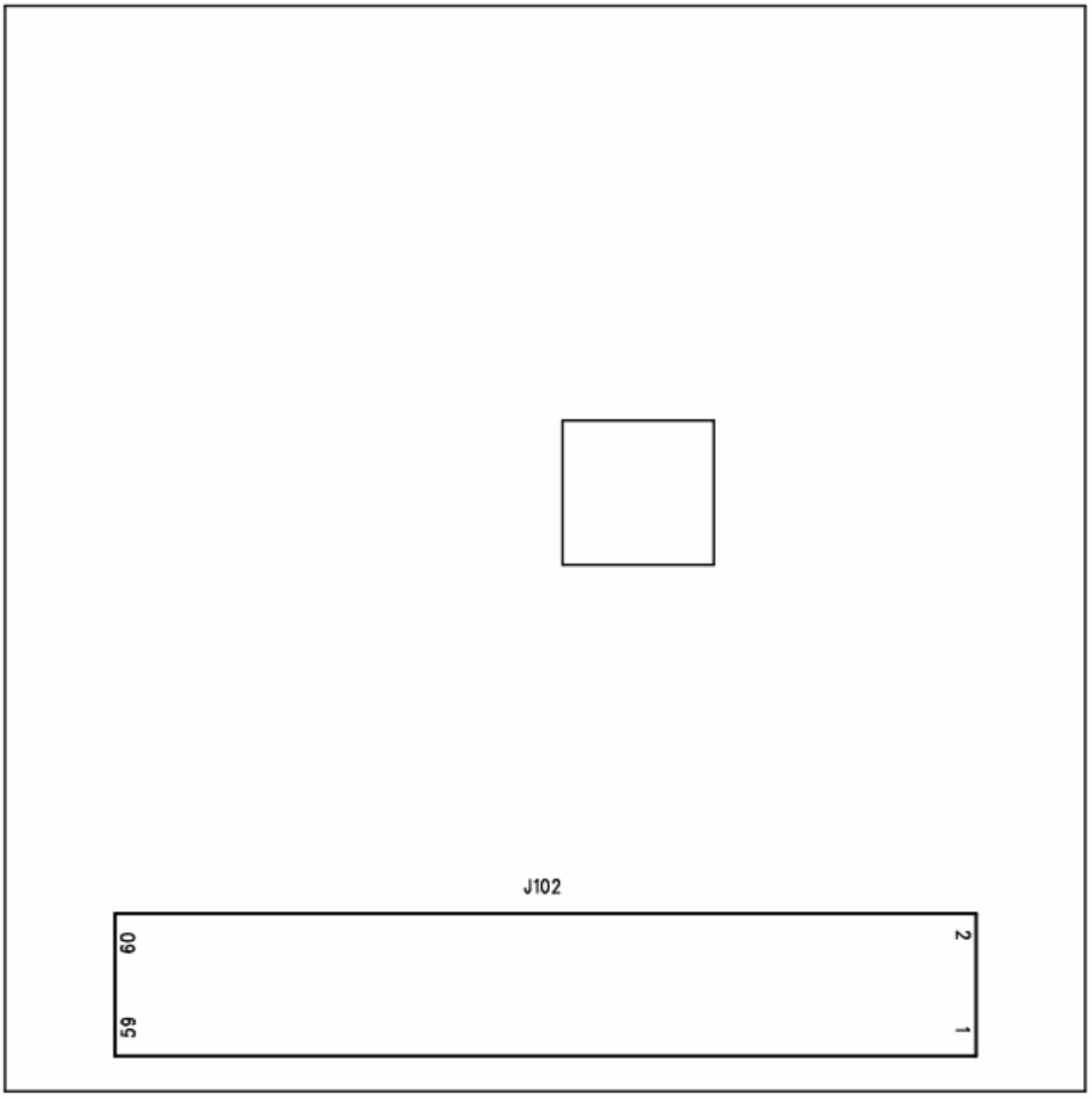
Paste mask Top



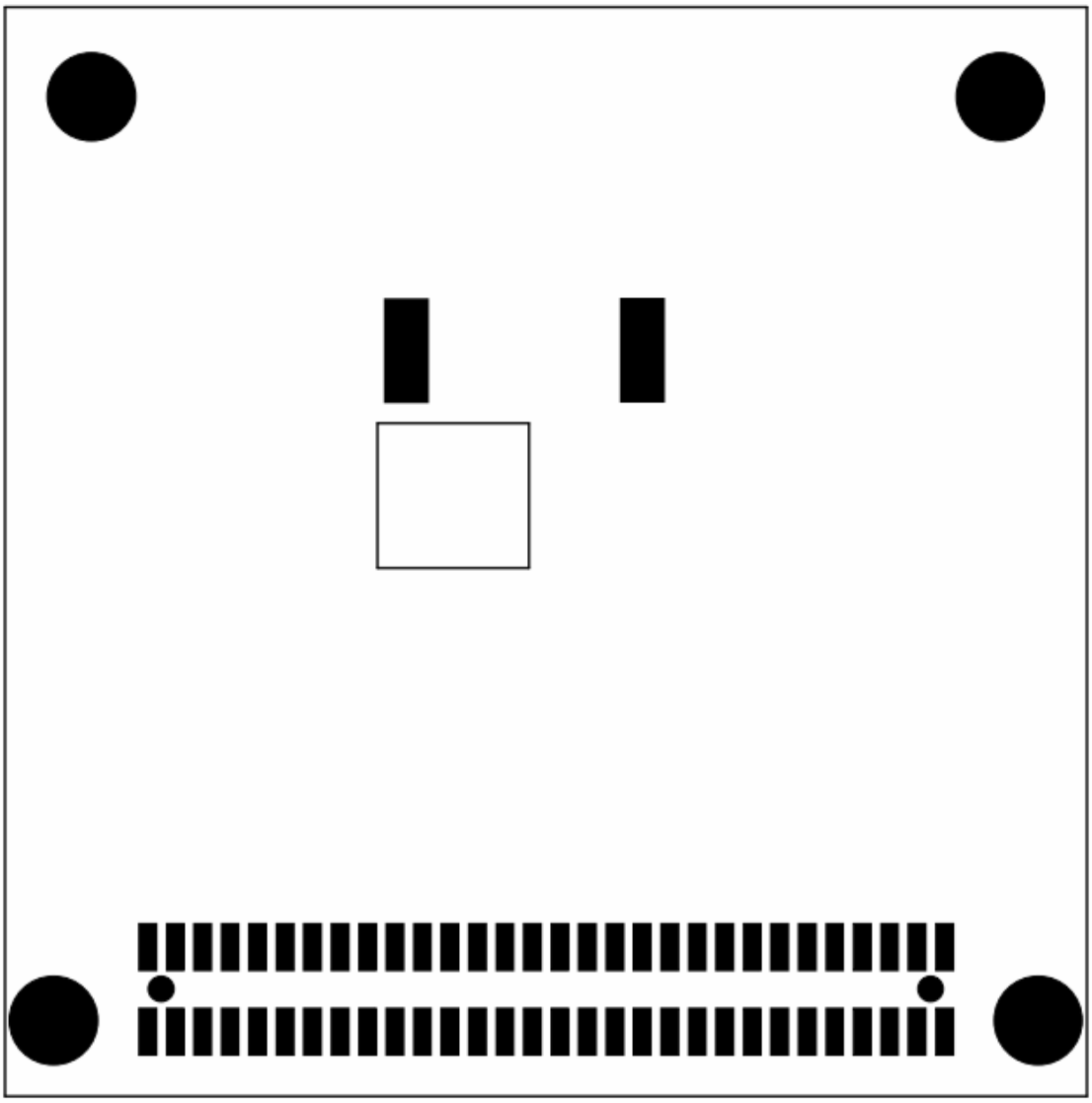
Silkscreen Bottom



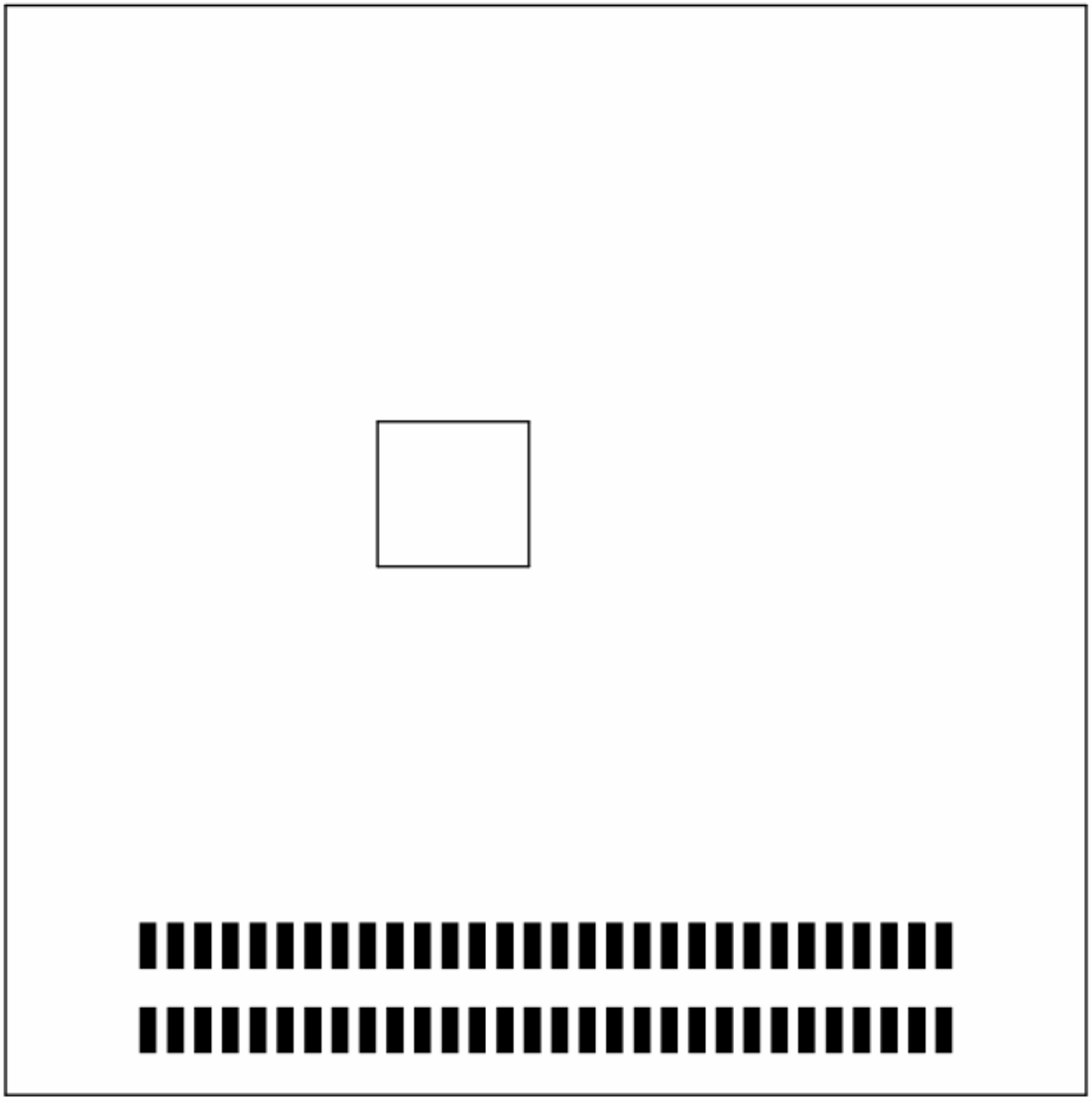
Reference designator Bottom



Solder mask Bottom



Paste mask Bottom



Contact us:

Shanghai SIMCOM Wireless Solutions Ltd
Add: SIM Technology Building, No. 633, JinZhong Road, Shanghai,PRChina 200335
Tel: +86 21 32523300
Fax: +86 21 32523200
URL:www.sim.com