



Figure 12: Reference circuit with 8-pin SIM card holder

2.7 ESD protection

It is suggested that it's better to do some ESD protection in your application to improve the ESD character of the module, especially for the signal connecting to external interface, for example, MIC, SPK, PWRKEY, VBAT, DBG, SIMCARD.

2.8 Consideration in PCB layout

In product's PCB design, a good PCB layout will help the improvement of the whole product performance, including reliability, EMC performance, etc. The following are some consideration for referenced:

- 1) The power trace should be short and wide, recommended above 80mil.
- 2) The layout of GROUND is very important. You should keep a full ground on top layer, and most of traces should be layout on the bottom layer, especially the audio traces, keep a whole GROUND under the module shield.
- 3) The audio traces (MIC & SPK) are better to be protected with ground.
- 4) The ground layer (top and bottom) must be connected with many vias, especially under the module's shield case.
- 5) The width of MIC traces should be 8-10mil. The width of SPK traces should be 12-14mil. The audio traces had better use difference connection and keep parallel.
- 6) The Layer1.Layer2 under SMT module test port should be copper keep out, layer3 should be GND;
- 7) The Layer2 under SMT module RF pad should be copper keep out, layer3 should be GND;
- 8) The Layer1.Layer2 under RF test connector should be copper keep out, layer3 should be GND