PCB Design for AXICOM High Frequency Relays (RF Switches)

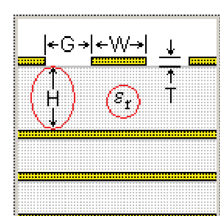
A. Introduction

1. As widely known for microwave PCB-design it is essential to obey the electromagnetic laws. RF-impedance matching therefore is a must. For the following steps one of the following tools (or similar) are very helpful.
2. a) Freeware-tool "txline": http://web.appwave.com/Products/Microwave_Office/Feature_Guide.php
   b) Freeware-tool "AppCAD": http://www.hp.woodshot.com/
   c) More tool-links: http://www.circuitsage.com/tline.html
3. Details on microwave PCB-materials like {ε_r} etc. can be found in the Internet with Google for example: "microwave laminates comparison".
4. The given footprint in the HF3-datasheet serves as general recommendation and proposal to get started with. In order to meet the nominal impedance minor modifications of the given footprint particularly trace-widths {W} may be necessary.

   These are related to the selected PCB-material {ε_r}, the ground-layer spacing {H} and gap-size {G}.
5. Since the HF3-Relay is designed in accordance to the CPW-pattern (CPW=Coplanar- Wave-Guide) the HF3 PCB-Design-Tips mainly refer to this type of design.
6. It is recommended to start with the PCB-design from the Relay-pads. Thus the following remarks are focused primarily on the **pad design**.

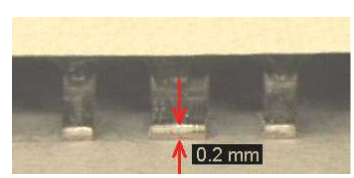
   With the given footprint two interconnected factors can be calculated now:
   a) The ground-layer-distance {H}
   b) The dielectric constant {ε_r}

   If the resulting impedance does not match to the rated impedance (50 / 75) either {H} and/or {ε_r} is incorrect. Of course if the PCB material {ε_r} is already given, {H} can be determined only. The same vice versa can also be calculated. If at last the desired nominal impedance cannot be achieved then modifications of {W} and {G} can also be taken into consideration.



   Important: The terminals after being soldered i.e. measure {T} should also to be taken into consideration. The height of the terminals is 0.2 mm.

   Therefore {T} in the terminal-area will be ≈ 0.3 mm.



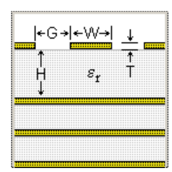
7. As a rule of thumb calculated impedance may deviate from rated impedance as follows:
   a) ±3..5% max. in 50 - applications
   b) ±4..8% max. in 75 - applications
8. Use of CPWG Layout is recommended. Microstrip with ground for connecting-traces may also be used.
9. PCB-material with a low dielectric constant ε_r is preferable. Standard FR4-PCB-material due to several reasons is normally not recommended for microwave applications.
10. If there are open terminals (14 or 20) they must be terminated with 50 or 75 in order to match the corresponding impedance.
11. All ground-terminals should be connected by the shortest way directly to the ground layer. This is accomplished by using vias with dual-layer boards and blind-vias with multi-layer boards.
12. Further RF-Design-Tips can be found here:
    http://www.jlab.org/accel/eecad/pdf/050rfdesign.pdf or
    http://www.rfcafe.com/references/app_note_links.htm

B. Insertion-loss

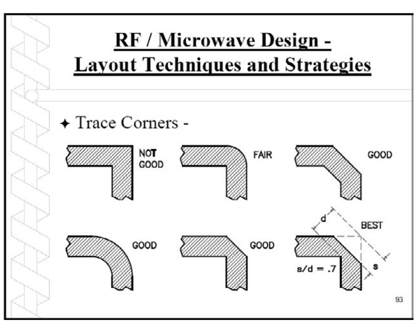
1. Low loss-tangent and of course short PCB-traces help to keep insertion-loss low. Materials with lower {ε_r} in general have also lower loss-tangents. Materials with loss-tangent values ≤ 0.005 for good insertion-loss and ≤ 0.0015 for excellent insertion-loss results can be found on the PCB market.
2. Trace-widths {W} for good insertion-loss results:

50 -relays	75 -relays
* ≥0.8 mm	* ≥0.5 mm

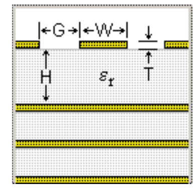
*Recommended trace-widths are only values for orientation and may differ due to several reasons. Certainly if trace-widths are increased a reduction of {ε_r} or an increase of {G} might be needed to keep up proper impedance match.



3. In order to keep insertion-loss low (coplanar-waveguide design) **extremely narrow gap-widths** {G} should be **avoided**. With this the normal PCB-manufacturingdeviation would also have an excessive impact on impedance-deviation.
4. Trace corners for better reflection results:



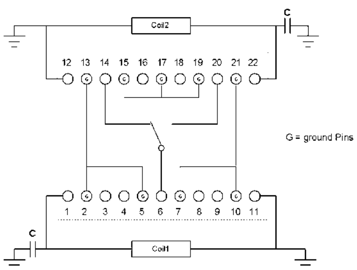
C. Isolation



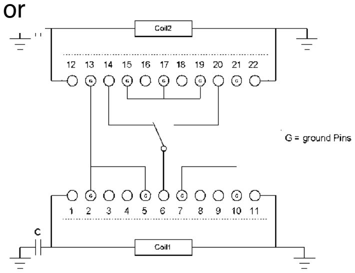
1. To improve isolation basically means to reduce reflections by avoiding unwanted radiation. Pure micro-strip will radiate a small amount of the signal into the air. Therefore if excellent isolation-results are needed the coplanar-wave-guide pattern might be taken into consideration. On the other hand insertion-loss will suffer in particular if the gap-size {G} is extremely small.

2. In applications with high isolation requirements and frequencies below ≈1.5 GHz coil terminal(s) should be grounded on one side. On the other side an RF-capable 10 pf capacitor to ground should be applied. Unused coil-terminals should be grounded as well.

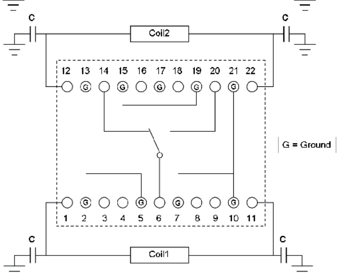
   If due to electrical reasons grounding of coil-terminal(s) is not possible alternatively 10 pf capacitors on both sides of the coil(s) will also help. The reason for this effect is that the coil takes the function of a shield in some way and part of the RF-radiation which normally creates leakage is absorbed therein.



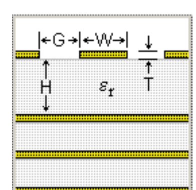
or



or



   The capacitor(s) should be placed as close as possible nearby the coil-terminal(s). Good results have been obtained with RF-capable 10 pf -thin-film 603 smd-capacitors.

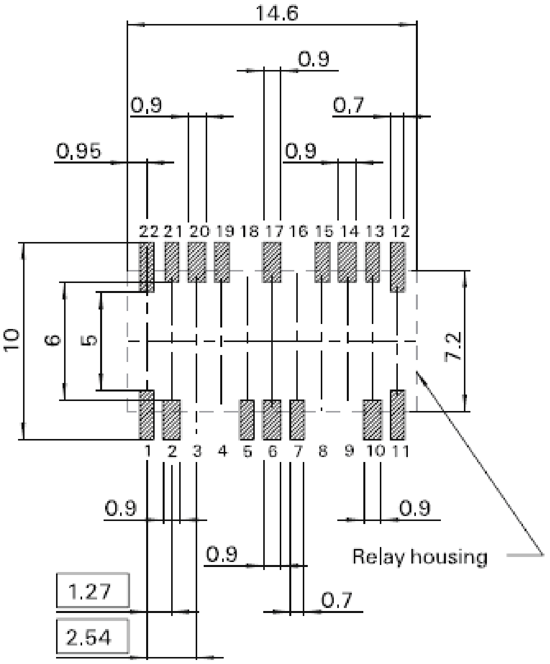


3. Ground-connections (vias) should be placed as close as possible to the ground-terminals of the relay. If possible the diameter of through-platings should be smaller than the thickness {H} of the PCB.

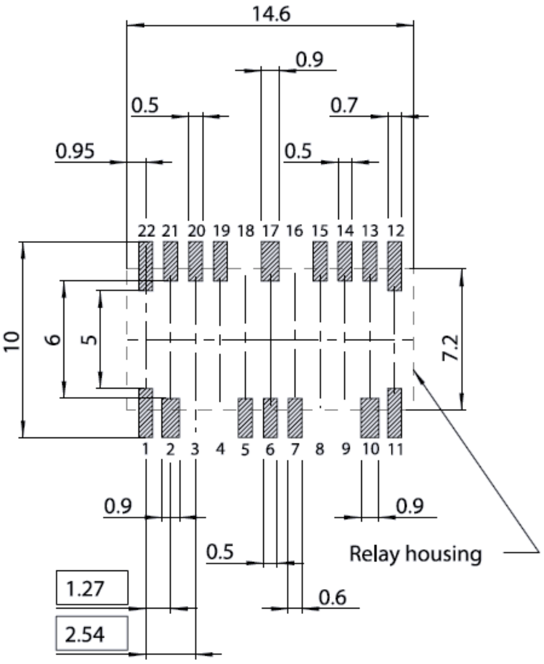
4. RF-trace-corners or parallel microstrip-traces close nearby the relay may deteriorate isolation results. Therefore RF-scattering-fields in close environment of the HF3-relay should be minimized. In special cases shielding may help to improve isolation-results.

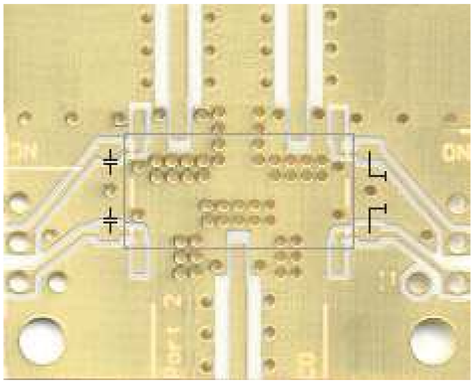
D. Footprints and layout-examples

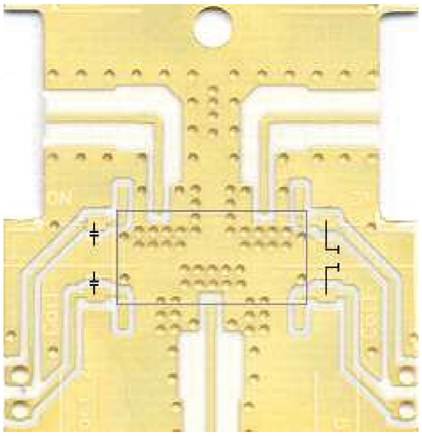
HF3-50 Ω - Relay



HF3-75 Ω - Relay







The boards here are solely to give an idea to get started with.
Soldering issues etc. have not yet been taken into consideration.

te.com

1-1773931-4 07/2017 JN

USA: +1 (800) 522-6752
Canada: +1 (905) 475-6222
Mexico: +52 (0) 55-1106-0800
Latin/S. America: +54 (0) 11-4733-2200
Germany: +49 (0) 6251-133-1999
UK: +44 (0) 800-267666
France: +33 (0) 1-3420-8686
The Netherlands: +31 (0) 73-6246-999
China: +86 (0) 400-820-6015