

# Application Note

---

## **SURFACE MOUNTING GRF180 SERIES RELAYS**

Application Note T0120 provides suggested surface mounting information and printed circuit board [PCB] artwork for circuit traces and RF ground plane configurations. Suggested information provided is intended for use with Teledyne Relays' GRF180 surface mountable, magnetic latching, ultraminiature, Centigrad<sup>®</sup> relays.

The following pages give information for a printed circuit board layout and assembly process that will allow the the GRF180 to be tested in a manner similar to tests performed at Teledyne Relays and give a base line for production lay out.



# Application Note

## Introduction

The GRF180 is a surface mount, RF, magnetic latching relay that employs a butt lead attachment scheme. Normally this form of attachment method does not allow for easy grounding of the relay case which is very important as frequencies increase. To allow for surface mount and proper relay grounding, Teledyne Relays has developed the ground shield. The ground shield not only allows the relay to be easily grounded, it protects against inadvertent misalignment of the leads as well as isolates the leads from one another after mounting them to the board. To optimize the ground shield connection the following is presented to facilitate testing and layout for use in production:

Page 3 GRF180 PCB Layout for test:

- Circuit side
- Ground plane side

Page 4 GRF180 PCB General Layout:

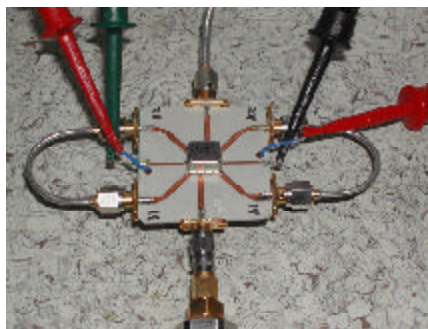
- Circuit side trace
- Lead trace
- Coil minus (-) connection

Page 5 GRF180 Artwork:

- Soldermask
- Stencil and Aperture Size
- Relay outline

Page 6 GRF180 General Notes

RF characterization data presented in the GRF180 data sheet is de-embedded from the RF circuit test board.

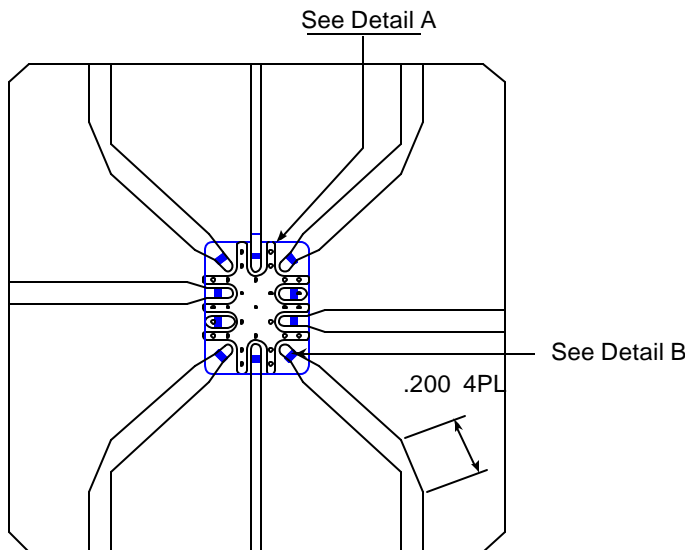


# Application Note

## RF Test Boards

RF performance characteristics of the GRF180 was obtained by mounting them on test boards developed by Teledyne Relays. To the test boards 6 female, 2 hole panel mount, solder cup contact SMA connectors were carefully soldered to the signal traces and the bodies soldered to the back side ground plane.

## GRF180 ARTWORK CIRCUIT SIDE



**Figure 1**

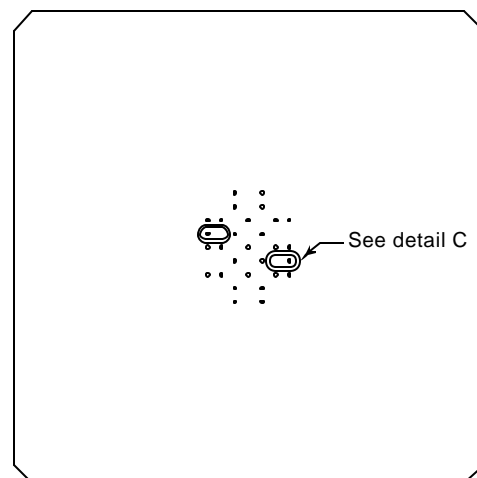
Figure 2 shows the ground plane side of the test board. There are 27 plated through vias connecting the ground trace on the component side of the board to the ground plane.

There are two additional connections specifically for the coil. This is detailed on the following page.

Figure 1 shows the component side of the RF test board.

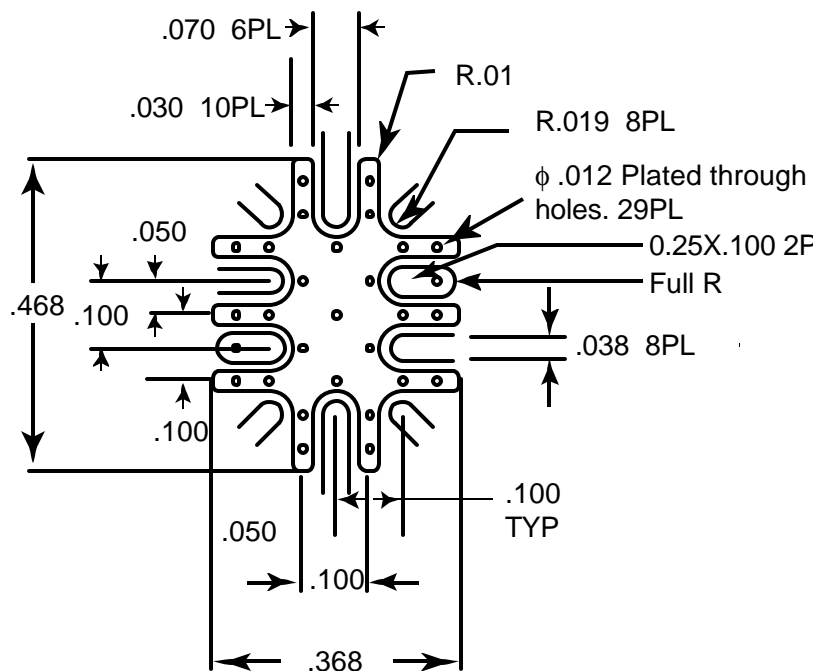
Layout details for the ground attachment pad and the signal traces are on the following pages.

## GRF180 ARTWORK GROUND PLANE SIDE



**Figure 2**

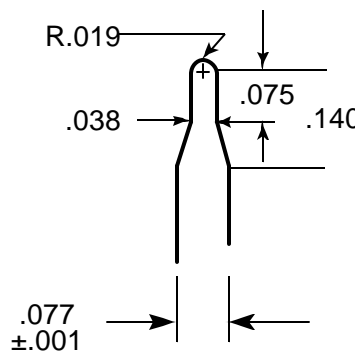
# Application Note



**Detail A**

## **Detail A**

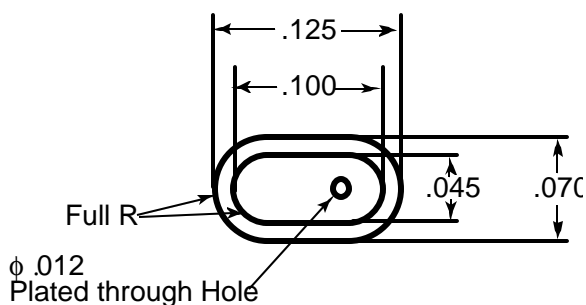
Detail A defines the ground pad layout that will interface with the GRF180 ground shield. Also defined are the GRF180 relay pin connections as well as the relationship of the vias connecting the ground pad to the ground plane on the opposite side of the board. Via details are given in Figure 4.



**Detail B**

## **Detail B**

Detail B defines the connection for the GRF180 signal pins. Connections for the plus (+) coil connection are given in Detail A and Detail C

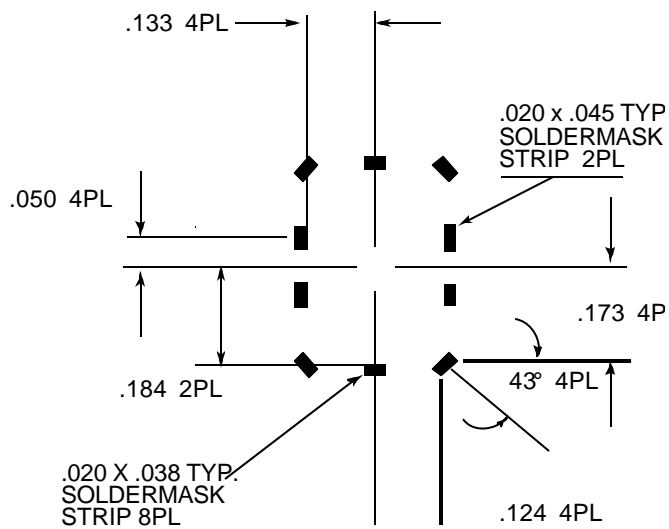


**Detail C**

## **Detail C**

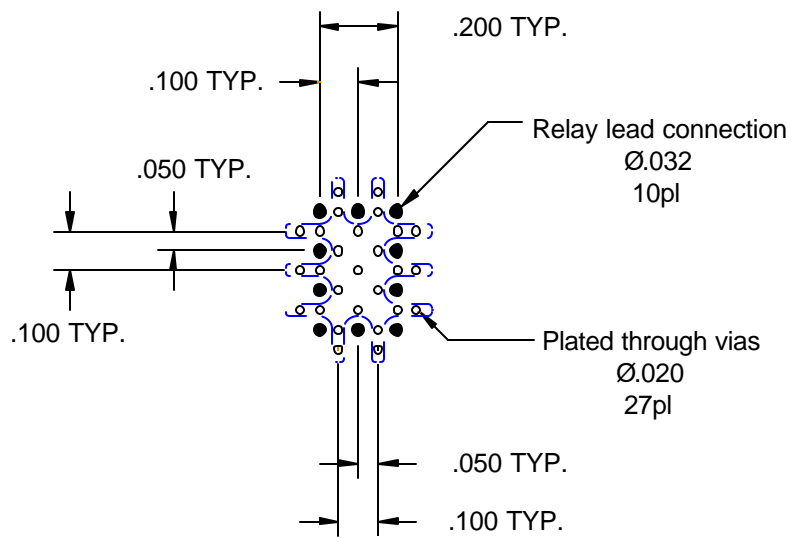
Detail C defines the plus (+) coil connection on the ground plane side of the test board

# Application Note



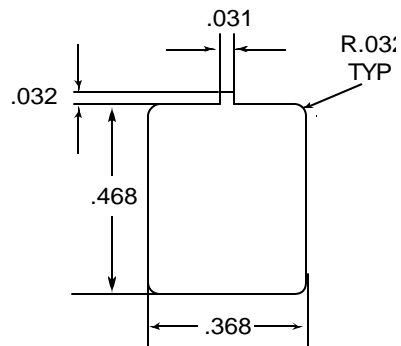
**Solder Mask**

**Figure 3**



**Stencil and Aperture Size**

**Figure 4**



**Relay Outline**

**Figure 5**

# Application Note

## General Notes

### GRF180 Printed Circuit Test Board Details

- a. Material: RT/duroid®6002 [RT/duroid®6002 is a registered trademark of Rogers Corporation]
- b. Thickness: 0.030"
- c. Copper foil thickness: 0.00134"
- d. All hole dimensions are after plate

- Suggested solder stencil thickness : 0.008"

- Trace configurations, board material, outline, size, etc. may require changes per user's application requirements.

- GRF180 relays may be subjected to solder reflow peak temperatures of 260°C maximum, for 1 minute, 3 times.

- Solder bottom of RF ground shield to PCB component side RF ground plane for best RF performance.

- Check with solder supplier for recommended solder reflow temperature profile for selected solderpaste and specific application requirements.

### Reference data:

The following web site may be usefull to those unfamiliar with SMT technology:

<http://www.pegasustech.com/gems/sm.htm>

[http://www.national.com/ms/MO/MOUNTING\\_OF\\_SURFACE\\_MOUNT\\_COMPONENTS-MISC.pdf](http://www.national.com/ms/MO/MOUNTING_OF_SURFACE_MOUNT_COMPONENTS-MISC.pdf)