

# Application Note 190

## Surface Mounting GRF172 Series Relays on RF Test Boards

APPLICATION NOTE 190 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' GRF172 surface mountable, ultraminiature, Centigrad® relays.

This page and the next give the following information for laying out the printed circuit board (dimensions shown are in inches):

GRF172 PCB Layout:

- Circuit side
- Ground plane side

GRF172 Stencil Layout and Aperture Size

GRF172 Artwork:

- Circuit side trace
- Solder mask
- Relay outline
- Ground plane side

Printed Circuit Board

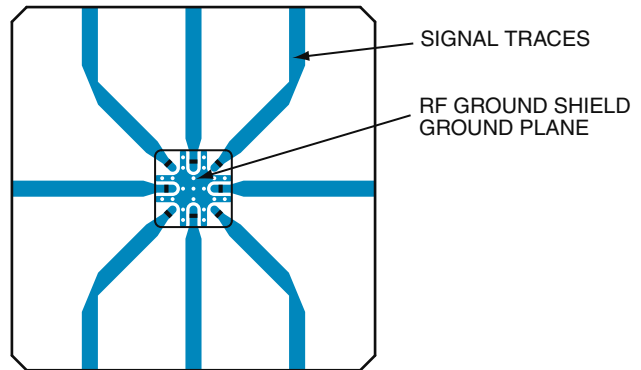
- a. Material: RT/duroid®6002 [RT/duroid®6002 is a registered trademark of Rogers Corporation]
- b. Dielectric thickness: 0.030"
- c. Copper foil thickness: 0.00134"
- d. All hole dimensions are after plate

- Suggested solder stencil thickness: 0.008"
- Trace configurations, board material, outline, size, etc. may require changes per user's application requirements.
- GRF172 relays may be subjected to solder reflow peak temperatures of 270°C maximum, for 1 minute, 3 passes.
- Check with solder supplier for recommended solder reflow temperature profile for selected solder paste and specific application requirements.
- Solder bottom of RF ground shield to PCB component side RF ground plane for optimal RF performance.

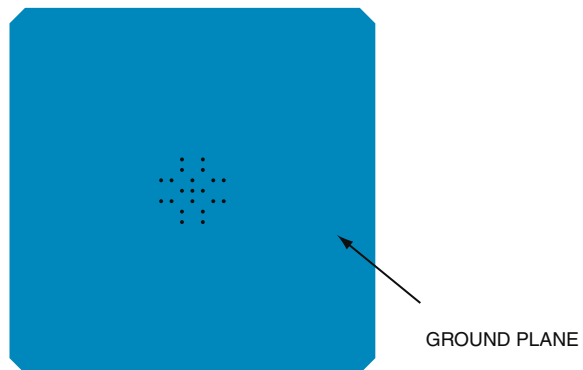
For more information, call Teledyne Relays at (800) 284-7007.

### GRF172 PCB Layout

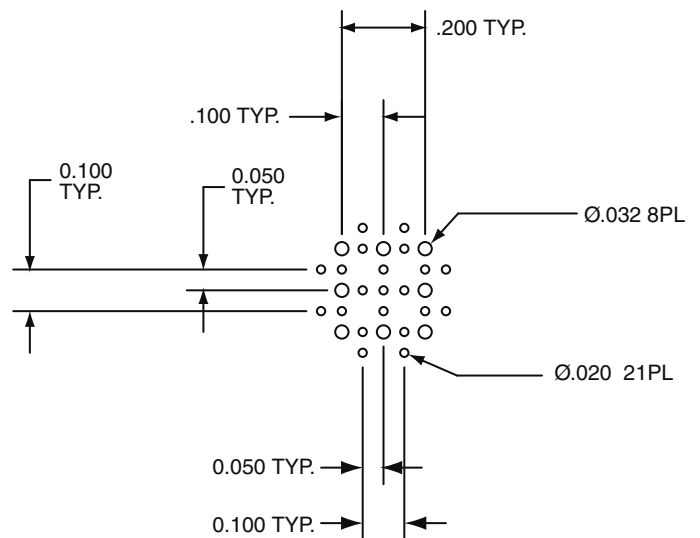
#### CIRCUIT SIDE



#### GROUND PLANE SIDE

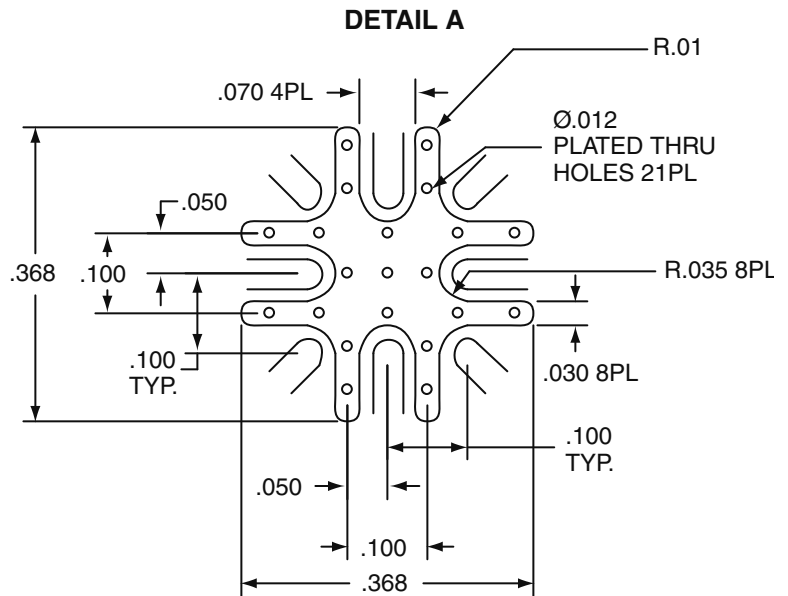
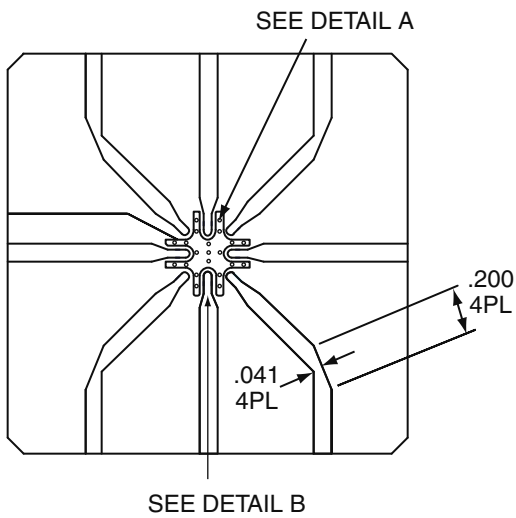


### GRF172 Stencil Layout and Aperture Size

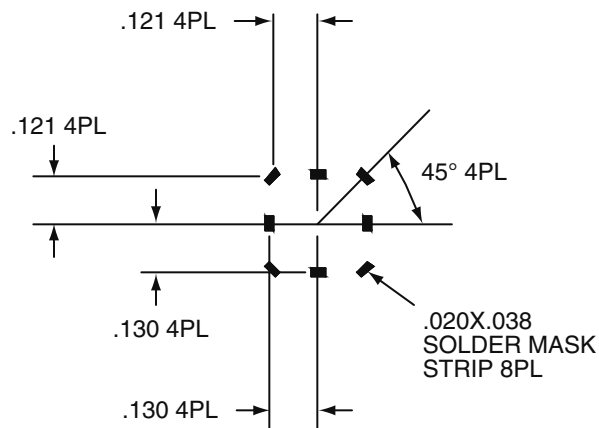


**GRF172 Artwork**

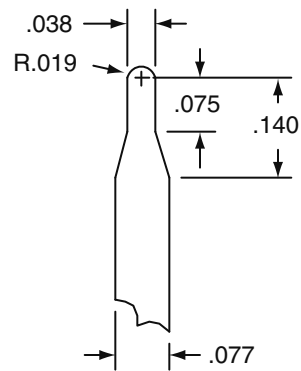
**CIRCUIT SIDE, TRACES**



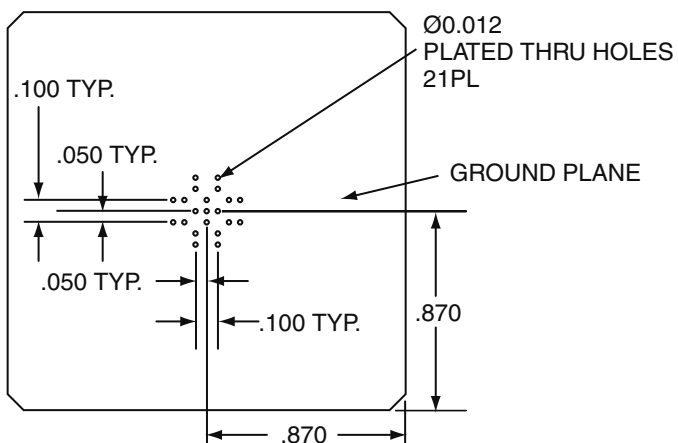
**SOLDER MASK**



**DETAIL B**



**GROUND PLANE SIDE**



**RELAY OUTLINE**

