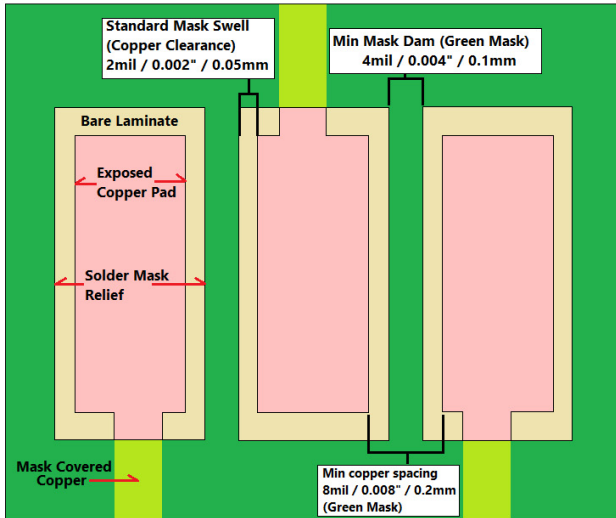


**Standard Solder Mask Clearance**    **Block (Gang) Mask Relief**  
**Minimum Solder Mask Dam**



If the design doesn't meet the min spacing requirement to print mask between pads, there are a few options.

1. Reduce the width of the copper pads just enough get the required minimum spacing to print mask dams.
2. Change the mask color to green if a different mask color was originally chosen.
3. Leave the copper pads at the original size, change the mask relief to a block relief and not print mask between them.

**Standard Solder Mask Clearance (Mask Swell)**

By default, mask apertures will be resized to allow ~2mil clearance between the edge of the copper pad and where the solder mask will be printed. There is a +/-2 mil (0.05mm) tolerance on the true placement in relation to copper.

If you require some solder mask features to be printed exactly as shown in your data, such as in cases where you have a [mask defined pad](#), please make a note of this in your fabrication notes.

**Minimum Solder Mask Dam Width**

Green is the best color for very fine pitch designs. It can be printed at a minimum width of 4mils (0.1mm) and still reliably adhere to the PCB laminate.

Taking into account the 2mil (0.05mm) min clearance between the mask and copper, this leaves us with 8mils (0.2mm) minimum copper to copper spacing and still be able to print a standard 4mil mask dam. There are some exceptions to this were we can go as low as 6-7mil gap but keep your copper spacing at 8mils for a more robust and manufacturable design.

**Minimum Mask Dam Widths By Color**

Green	4mil (0.10mm)
Blue	5mil (0.12mm)
Red	5mil (0.12mm)
Black	6mil (0.15mm)
White	6mil (0.15mm)

