1140 NW 3rd Avenue
Canby, Oregon 97013
866-784-5887
www.screamingcircuits.com

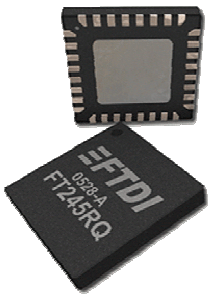
QFN Layout guidelines

While QFN (quad flatpack, no leads) and DFN (dual flatpack, no leads) packaged parts are becoming more and more common in new component releases, they aren't getting much easier to use. The advantages of the form factor are pretty clear. It allows smaller geometries, better grounding and improved thermal properties over other types of surface mount packages.

Most QFNs have a center metal pad on the underside of the part, typically for grounding or heat conduction. It's this center metal pad that makes this form factor so difficult to use. DFN packages are identical, for the purposes of assembly, except that they have pin rows on only two sides of the part.

QFN Float

At a recent trade show, I received a sample part in a 3 x 3 mm QFN package. While I haven't tried it yet, I'm pretty sure that, like a water bug, the part is light enough to float on a water surface because it doesn't have enough weight to break the surface tension. But, that's not what I'm talking about.



The middle of the part has a metal contact pad – like most QFN packaged parts. It may be there for grounding or heat conduction, depending on the specific part. The float that I'm talking about happens when we lay too much solder paste on the PCB for that center pad.

To a small extent, the height of the solder paste deposit is proportional to the aperture in the solder stencil opening (bigger opening = taller deposit). With most parts, that isn't a problem because either all of the pads are big enough so that that ratio doesn't have a first order impact, or because all of the pads are the same size and will be equally impacted.

Since the QFN center pad is a much larger opening in the stencil than the signal pad openings, and the signal pad openings are in the 10 - 20 mil or less range, this deposit height to width ratio will have a first order impact.

QFN mounted with too much solder in center pad area

When the opening for the center pad on the QFN is too large, the solder paste deposit in the center will be taller than the deposits on the small signal pin pads. The part high-centers and never gets the opportunity to contact the signal pads. In some cases, the part will tilt a little sideways and contact some of the signal pads but not all.

QFN mounted with reduced solder paste in center pad area

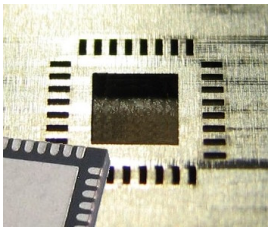
Solder Paste Stencil

Typically, the signal pads should have a standoff height of 2 - 3 mils after assembly. If too much solder is deposited in the center, the part can very easily float up beyond that height and prevent the signal contacts from connecting. To help prevent this, the solder stencil opening should be broken into a series of smaller openings and should cover between 50 and 75% of the pad area.

This means that when you lay out your PCB, you need to look carefully at the solder paste layer for your QFN components. If the solder paste layer in the CAD package part library just follows the copper pad

pattern or the solder mask opening, you may need to customize the CAD package part library to avoid leading yourself into trouble.

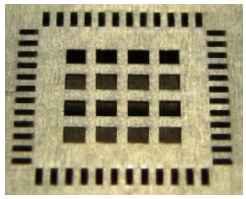
To better illustrate the proper way to make your solder paste stencil for QFN parts, I went to our back room and took a couple of photos of good and bad solder paste stencil practices.



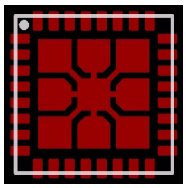
This is what a just about worst-case stencil would look like. Actual size for this part is 7 x 7 mm. Note how much surface area that the center pad has compared to the row of side cutouts. With most SMT components, it is standard procedure to reduce the size of the paste cutout area in the stencil. In a case like this, it is difficult to reduce it enough and still get even paste distribution. The proper option is to segment to solder stencil area.

If you just reduce the paste opening aperture, providing one smaller opening, but don't segment it, you may end up with a part that is still too high in the middle to assure good contact on the signal pads and is also unstable and will likely tilt to one side. With leaded solder, a single 50% sized opening may work because of the wicking properties of lead-based solder. Since lead-free solder does not wick as well, it is very unlikely to work in a RoHS process. In both cases, the most consistently reliable method is to segment the stencil pattern.

This is an example of recommended practice. The basic idea is that you distribute a lower quantity of solder over a broader area. You reduce your chances of high-centering and other problems associated with large paste areas, such as out gassing and spattering. This will give good solder distribution with little chance of high-centering or outgassing problems.



Specialized Copper Pad



Some parts, especially high-frequency parts, require a segmented copper pad under the QFN. If this is the case, it is important to segment the solder paste stencil to match the custom pad. It is fairly common practice to use a standard full-size square opening and hope that surface tension will end up distributing the solder in the right places. While that may happen, the chances of it not happening are equally great or greater. For best reliability and buildability, make sure that the openings match your copper layer underneath the stencil openings. Be sure that your stencil openings only fall above the copper and not over any solder-mask or bare-board sections.

Larger opposites

With larger QFN parts, the opposite problem can occur in the center pad area. When the square opening for the solder paste stencil is fully open on a larger part – say 10 x 10 mm or larger – the paste squeegee may deform and actually scoop too much of the paste out of the opening. This can lead to uneven paste and solder voids. Both are potential reliability problems. The solution is the same. Segment the stencil opening to create an even paste distribution.

QFN with paste void in center

QFN mounted with reduced solder paste in center pad area

Summary

The QFN form factor delivers a number of advantages over other SMT package form factors. It is generally a smaller part and, with the center pad, can have better grounding and thermal properties. These advantages are partially offset by layout and assembly difficulties. But by following a few simple guidelines, you can use the parts with good confidence. Check the layout guidelines in the component applications notes. Segment your solder stencil opening for the center pad. Make a custom component library for your CAD package if you need to. Then Design away.