Featured image:

https://www.sfcircuits.com/userfiles/image/qfn-package-on-board.jpg

QFN Packages: Design, Assembly, and Soldering Considerations

Quad flat no-lead (QFN) packages are common in electronic designs where board space, electrical performance, and heat dissipation matter. Their compact, low-profile construction eliminates projecting leads, allowing for smaller PCB footprints and shorter package-to-board connections.

However, QFN packages also present specific design and assembly challenges. Because most solder joints are located underneath the component, they cannot be inspected visually. The exposed pad, thermal vias, solder-paste deposition, and reflow conditions all play an important role in achieving reliable solder joints.

So, what should designers and manufacturers consider when working with QFNs? San Francisco Circuits takes a closer look at QFN package types, PCB footprint design, solder-paste and stencil requirements, assembly processes, common soldering challenges, and inspection methods.

Read more about QFN packages, their design requirements, and assembly considerations here.