Bojay Electronics Company

Finite Element Analysis Report

Superfuzz_AC 28-102746-01/73-102898-01 - Agilent 3070 ICT Fixture

Analysed by DanHui Hu

Report date: 24/3/2022

Summary

- A linear static finite element analysis (FEA) was performed for a printed circuit board when subjected to typical forces imparted during test. The analysis was performed for the fixture in the actuated position. In this position, all probes and board supports contact the board under test.

- The FEA calculates the following maximum strains and deflection:

FEA Result	ε_{xx}	ε_{yy}	deflection
Peak Values	298µm/m	292µm/m	0.036mm

- Board-level strain was examined to assess the potential effect on components mounted on the board. Board-level strains result from bending that is induced by the probe forces acting on the board. In general, strains greater than 400 µm/m can cause failure in some surface mounted components.

- For this board, the peak calculated strains are below the desired limit of 400 µm/m. Therefore, the fixture can provide adequate support to the board under test.

Board and Fixture Details

- Fixture Details: Vacuum Box
- Board Details: 2.4-mm thick
- Probe Count:
    - Bottom:
        - 687 100 mil probes
        - 172 75 mil probes
        - 1405 50 mil probes
        - 0 39 mil probes
    - Top:
        - 81 100 mil probes
        - 31 75 mil probes
        - 94 50 mil probes
        - 0 39 mil probes
- Board Supports:
    - Bottom:
        - Zero flex milling on support plate
    - Top:
        - Push blocks
        - Push fingers

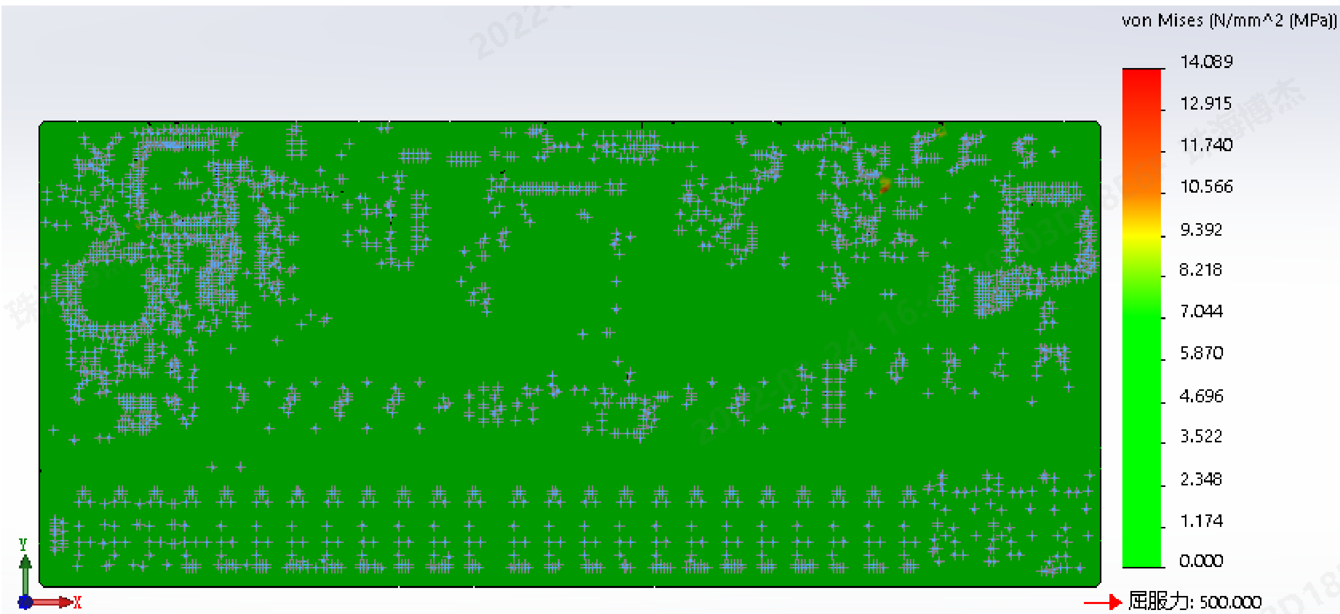
Modeling Assumptions

- The board under test is modeled with shell elements with elastic isotropic material properties. Probe forces are connected at the mid plane of the shells.
- Probes are modeled as static forces. Full probe force will be applied regardless of board deflection.
- Board Under Test Physical Properties:
    - Young's Modulus: 24000N/mm^2
    - Poisson's Ratio: 0.3

FEA Results

Stress

- The peak board level stress was calculated to be 14.1 N/mm^2, which is below the elastic limit of FR4/G10 base material, while the elastic limit of FR4/G10 base material is 500 N/mm^2. Therefore, board-level material failure is not a concern



FEA Results(cont.)

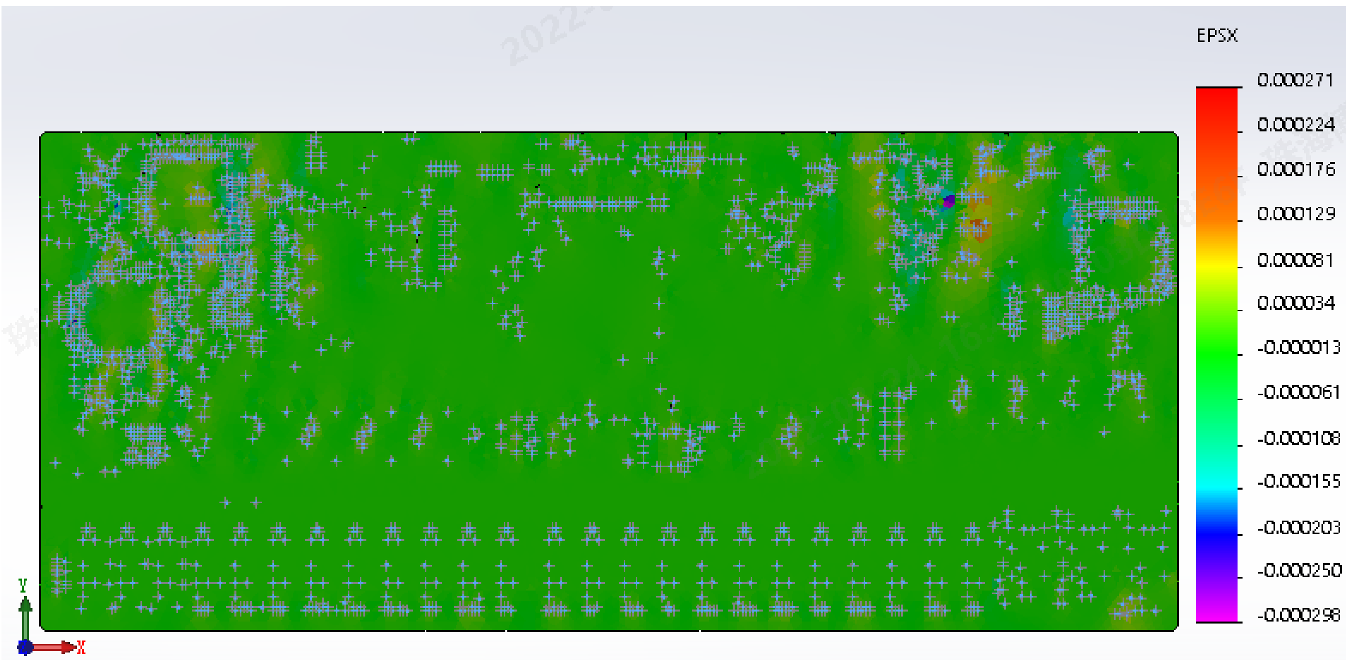
Strain,Y-direction

- The following plot shows strain distribution on the top of the board. The strains on the bottom of the board are equal magnitude but opposite sign (compressive strains on the top are tensile on the bottom).
- The peak strain in the y-direction was calculated to be 292 µm/m. The peak strain locations are shaded in magenta in the plot below. These strains are below the desired limit of 400 µm/m.

FEA Results (cont.)

Strain,X-direction

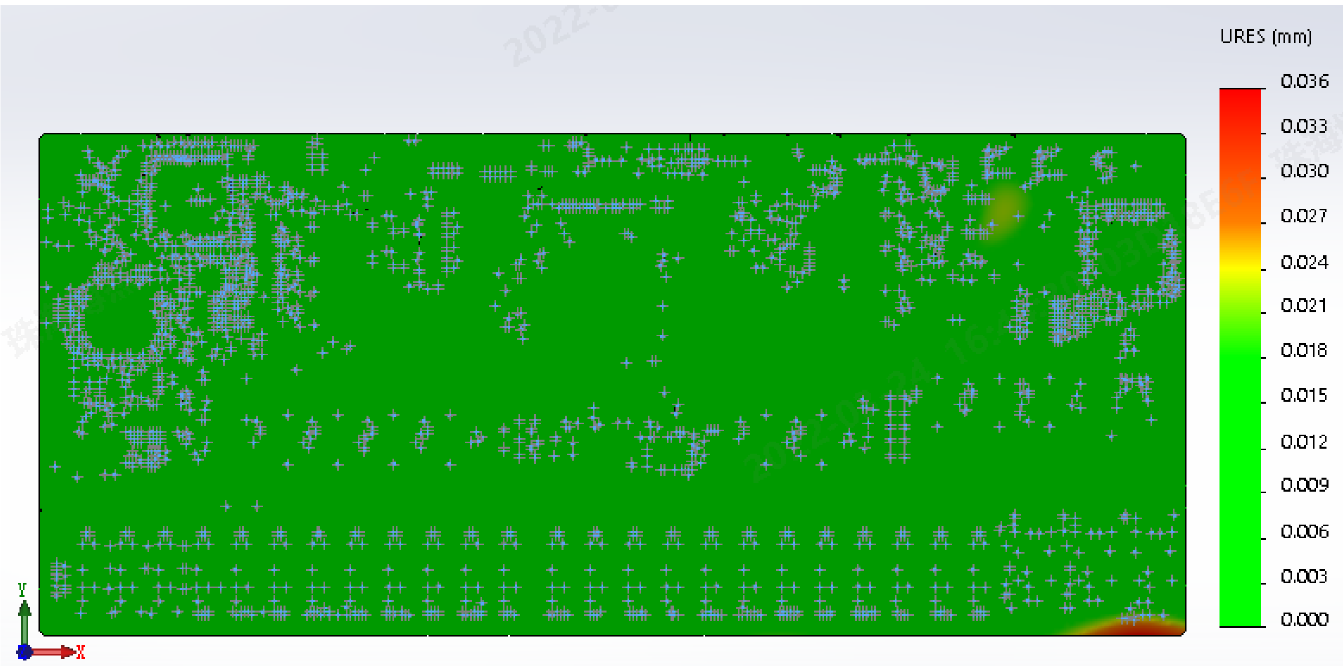
- The following plot shows strain distribution on the top of the board. The strains on the bottom of the board are equal magnitude but opposite sign (compressive strains on the top are tensile on the bottom).
- The peak strain in the x-direction was calculated to be 298 μm /m. The peak strain locations are shaded in magenta in the plot below. These strains are below the desired limit of 400 μm/m.



FEA Results (cont.)

Displacements

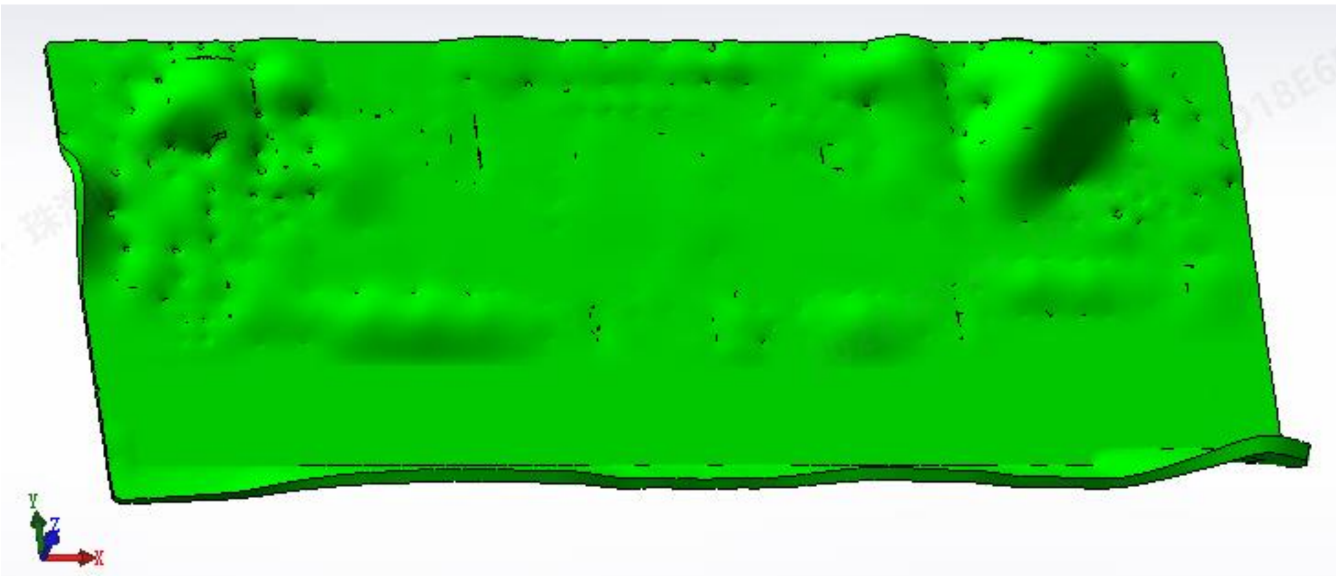
- The overall maximum deflection predicted is 0.036 and it is passing displacement criteria Guidelines. The peak strain locations are shaded in magenta in the displacement plot below.



FEA Results (cont.)

Deformed Shape

- A deformed shape plot is provided to show an exaggerated view of the board reaction to the forces imparted during test. The topography identifies the areas of maximum displacement on each image.



Conclusions

- Board-level strain was examined to assess the potential effect on components mounted on the board. Board-level strains result from bending that is induced by the probe forces acting on the board. In general, strains greater than 400 µm/m can cause failure in some surface mounted components.

- For this board, the peak calculated strains are below the desired limit of 400 µm/m. Therefore, the fixture can provide adequate support to the board under test.

- Finite Element Analysis only simulates analysis of stress, to get the more real strain result to avoid the risks, you should use the sensor to verify the strain and stress, and do the red ink experiment.