BULETINUL INSTITUTULUI POLITEHNIC DIN IAŞI
Publicat de
Universitatea Tehnică „Gheorghe Asachi” din Iași
Volumul 71 (75), Numărul 2, 2025
Secţia
CONSTRUCŢII DE MAŞINI
DOI: 10.2478/bipcm-2025-0013

PCB COPPER TRACES STRESS DUE TO THERMAL EXPANSION CAUSED BY JOULE HEATING EFFECT

BY

GEORGE-GABRIEL CHIRIAC[1,3,*], GEORGIANA PRODAN-CHIRIAC[2], CĂTĂLIN GABRIEL DUMITRAŞ[1], DRAGOŞ-FLORIN CHITARIU[1,3], NECULAI-EDUARD BUMBU[1,3] and LUCIAN OANCEA[1,3]

[1]“Gheorghe Asachi” Technical University of Iași, Faculty of Machine Manufacturing and Industrial Management, Iași, Romania
[2]“Gheorghe Asachi” Technical University of Iași, Faculty of Electrical Engineering, Iași, Romania
[3]Academy of Romanian Scientists, Bucharest, Romania

Received: May 7, 2025
Accepted for publication: June 1, 2025

Abstract. This study investigates the stress induced by thermal expansion in Printed Circuit Board (PCB) components. A simplified PCB model was used for electrical simulation via Siemens Simcenter FLOEFD 2412, determining temperature distribution from Joule heating in copper layers and vias. These temperatures were input into an Ansys Workbench 2023 R1 structural Finite Element Analysis (FEA). Results show Joule heating causes thermal expansion, and differing material temperatures and expansion coefficients generate stress and strain. Equivalent stress and strain exceeded material yield values, indicating potential copper cracking under high temperatures or thermal cycling. The thermal and structural FEA method effectively predicts thermal stress and strain in PCBs.

*Corresponding author; *e-mail*: george-gabriel.chiriac@academic.tuiasi.ro

Keywords: PCB, Thermal Expansion, Joule Heating, FEA Simulation, Thermal Stress.

1. Introduction

More and more electronic components are used today in many industries, as automotive industry or machine manufacturing. This involves the use of Printed Circuit Boards (PCB) to conduct the electricity or signals and control the components (Mei *et al*., 2018).

Printed Circuit Boards are usually made from a few layers of copper connected between them with copper cylinders called vias. Between the copper layers and vias we find dielectric material, usually FR4, to isolate the layers between each other. When the electrical current comes, it passes the copper layers and thermal vias in order to reach the desired area, the copper heats up due to the Joule heating effect than can be found in any conductor (Zhang and Chen, 2024).

Since copper has a much higher thermal conductivity than the dielectric material, the heat spreads way faster in the copper material. Therefore, the temperature of the copper is increasing in the entire conductor, even if the joule heating effect is different in the connector volume, based on its shape (Bagnoli and Zhang, 2010). The temperature between copper and FR4 can be different on the entire volume. Additionally, the thermal expansion coefficient of the two materials is different, so a different thermal expansion is expected for the 2 materials (Shen *et al*., 2020).

The current investigation wants to evaluate the stress caused by the thermal expansion of the PCBs components. To do that, a very simple PCB model was made. The CAD model is used for the electrical simulation using Siemens Simcenter FLOEFD 2412. The result of the electrical simulation is the assembly temperature due to the dissipated power, when the DC current is passing through the copper layers and copper vias. The temperature results are exported and used as an input in the structural simulation. The structural FEA simulation is solved using Ansys Workbench.

2. Structural simulation

To solve the structural simulation, the Static Structural FEA simulation type from Ansys Workbench 2023 R1 is used. After the same CAD model is imported, the contacts are generated. All the contacts between the parts are considered bonded, so no movement is expected between the parts.

Due the very thin copper layers and thin vias cylinders, a small mesh element size is required to keep a good aspect ratio. Therefore, a high number of mesh elements and nodes is used. The mesh elements can be seen in Fig. 1.

Number of nodes: 4749928
Number of elemenets: 2699175

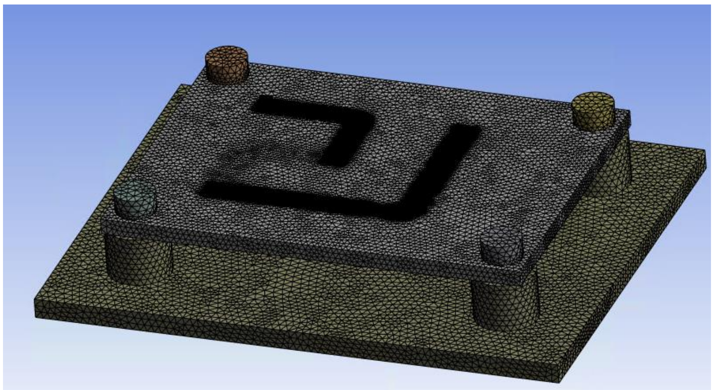

Fig. 1 – CAD geometry with mesh.

A detailed imagine of mesh elements on the copper layer and copper vias can be seen in Fig. 2. The mesh elements shape used is Tetrahedrons with Patch Conforming Method.

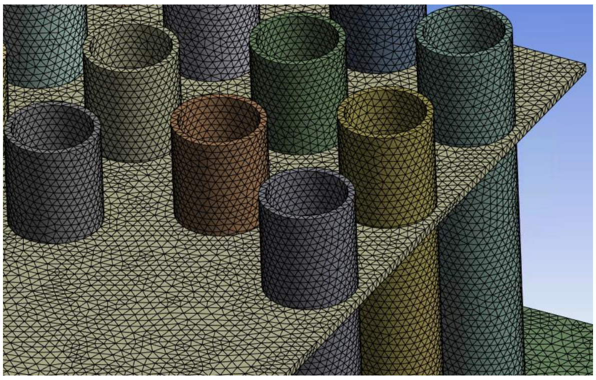

Fig. 2 – Mesh elements on vias.

Fixing the assembly in a thermal expansion simulation can be challenging. This is due to the fact that CAD model has to be kept in equilibrium, but in the same time it has to be free to expand, due to thermal expansion. Therefore, three frictionless supports were used on the bottom surface of the aluminum housing and two of the side faces. In this way, the aluminum housing can expand, if its temperature rises, but in the same time no abnormal movement will be present. This is the most time efficient setup if we want to simulate the assembly as it stands on a table.

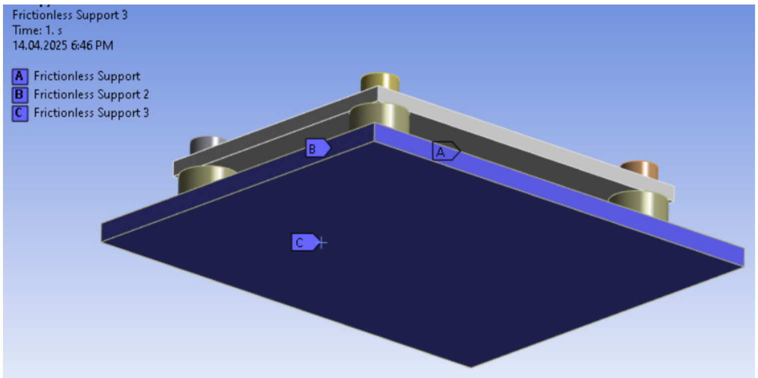

Fig. 3 – Frictionless supports for fixation.

To verify if the temperature imports has succeeded, the import body temperature can be seen as a body temperature plot on the CAD model. As it can be seen in Fig. 4, the temperature of the copper areas is higher than the rest of the components and the entire model has the same temperatures as it had in the electrical simulation.

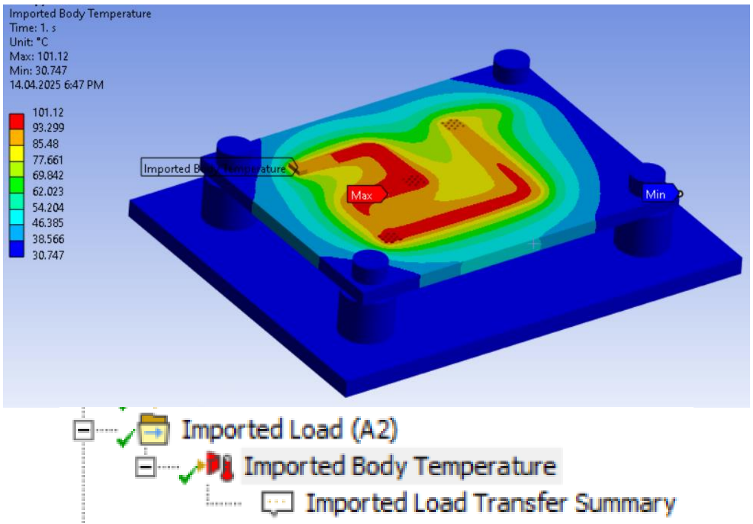

Fig. 4 – Imported body temperature.

In Table 1 the coefficient of thermal expansion for used materials can be seen. It can be observed that the coefficient of thermal expansion of FR4 is slightly smaller than the coefficient of thermal expansion of copper on X and Y axis but higher on Z axis. The Z axis is considered the axis perpendicular to the PCB face with the higher area.

Table 1
Coefficient of thermal expansion

		Coefficient of thermal expansion: [C^(-1)]		
Component	Material	X axis	Y axis	Z axis
Housing	Aluminum alloy	2.30E-05		
Traces and vias	Copper alloy	1.80E-05		
PCB dielectric	FR4	1.25E-05	1.14E-05	8.20E-05
Screws	Stainless steel	1.70E-05		

Because it's a structural simulation, other material properties are needed to define the materials inside the simulation. In Table 2 the Young Modulus, Poisson Ration, Yield Strength and Ultimate Strength values for all the material used can be seen.

Table 2
Material properties

	Young Modulus [MPa]	Poisson Ratio	Yield Strength [MPa]	Ultimate Strength [MPa]
Aluminum alloy	7.10E+04	0.33	280	310
Copper alloy	1.10E+05	0.34	280	430
FR4	2.04E04 on X axis 1.84E04 on Y axis 1.5E04 on Z axis	0.1	-	-
Stainless steel	1.93E+05	0.31	207	586

3. Simulation results

The maximum Equivalent Stress (von-Mises) on the copper parts in the simulation is approximately 574 MPa, but this is a singularity point and should be neglected. An overview and stress distribution can be seen in Fig 5.

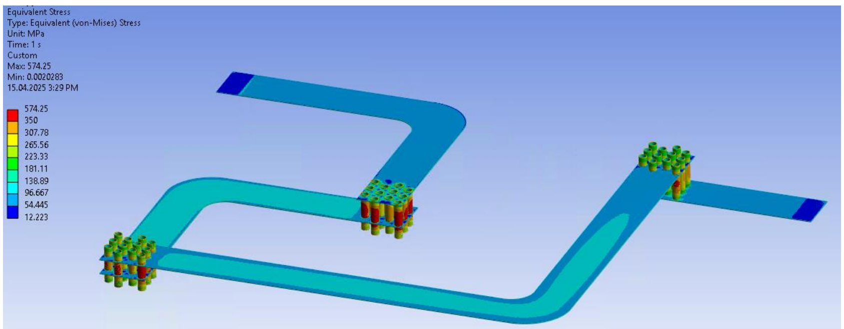

Fig. 5 – Equivalent stress.

The highest equivalent stress can be found on the copper vias area. After adjusting the stress plot, it can be seen that the highest stress is between 350 MPa and 400 MPa. A detail of the stress distribution on the vias can be seen in Fig. 6.

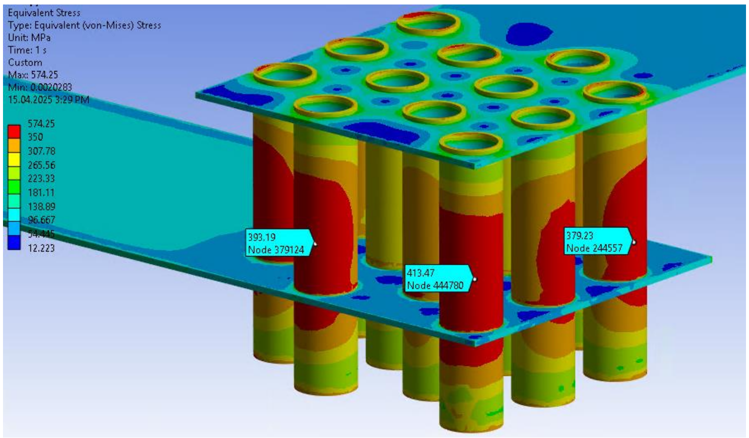

Fig. 6 – Equivalent stress on vias.

According to the material parameters used, the copper parts are reaching plastic deformation on some areas. Nevertheless, the ultimate strength stress values are not achieved in simulation, so chances of cracks in the copper conductors are less likely to happen.

The Equivalent Elastic Strain values show as well an uniform distribution overall on the copper layers. This shows us that a good thermal conductivity in the components helps also the strain and deformation on the components, as expected, because the difference between the maximum and minimum temperature on the copper layers is not very high. Therefore, the thermal expansion on different areas is not way different.

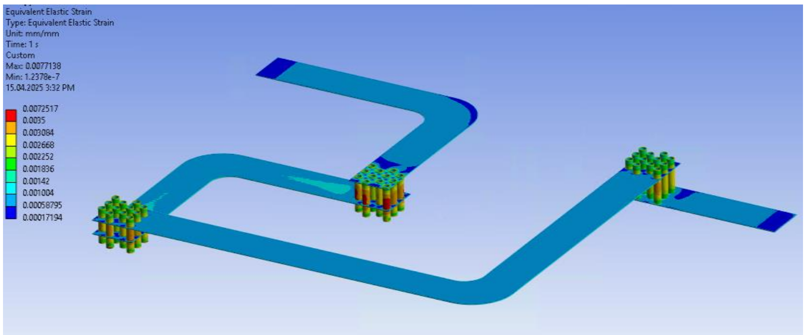

Fig. 7 – Equivalent strain overview.

Looking at the equivalent strain values in detail on the copper vias in Fig. 8, we can see that it exceeds the yield limit. The higher values of equivalent strain are between 0.35% and 0.35%.

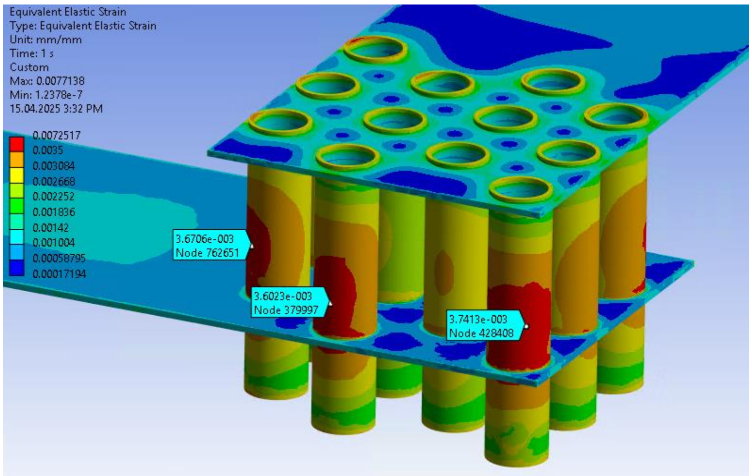

Fig. 8 – Equivalent strain on vias.

As before, when we evaluated the equivalent stress, the equivalent strain values are not exceeding the ultima strain limit for the copper material, so cracks in the material will most likely not appear.

4. Conclusions

The heat up generated by the Joule heating effect is the copper layers and copper vias of a PCB is causing the thermal expansion of the conductors. Due to different temperature and thermal expansion coefficients of the materials used inside the PCB, the thermal expansion is causing an additional stress and strain inside the materials.

For the given example, the equivalent stress and equivalent strain values are exceeding the yield values for the given materials. While this may not be a concern if the temperature is not increasing even more or the number of cooling and heating cycles is reduced, there still is a risk of cracks inside the copper areas if the stress or temperature is exceeding even more or often.

The thermal and structural FEA simulation is a very good method in predicting the areas or intensity of thermal stress and strain that may occur inside a PCB during functioning.

Acknowledgements. G.G.C., D.F.C., N.E.B., L.O. are grateful for the support received from the Academy of Romanian Scientists.

REFERENCES

Bagnoli P.E., Zhang Y., *Electro-thermal simulation of metal interconnections under high current flow*, Microelectronics Reliability 50 (2010) 1672-1677.

Mei J., Haug R., Lanier O., Grözinger T., Zimmermann A., *Effect of Joule heating on the reliability of solder joints under power cycling conditions*, Microelectronics Reliability 88-90 (2018) 684-690.

Shen Y., Wang H., Blaabjerg F., Zhao H., Long T., *Thermal Modeling and Design Optimization of PCB Vias and Pads,* IEEE Transactions on Power Electronics, Vol. 35, No. 1, (2020) 882-893.

Zhang Y., Chen L., *A modeling method for thermal steady-state simulation of the four-layer printed circuit board*, PLoS ONE 19(9): e0310237 (2024).

SOLICITAREA TRASEELOR DE CUPRU DIN PCB DATORITĂ DILATĂRII TERMICE CAUZATE DE EFECTUL JOULE

(Rezumat)

Acest studiu investighează solicitarea indusă de dilatarea termică în componentele plăcilor cu circuite integrate (PCB). Un model simplificat de PCB a fost utilizat pentru simularea electrică prin intermediul Siemens Simcenter FLOEFD 2412, determinând distribuția temperaturii rezultate din încălzirea Joule în straturile de cupru și via-uri. Aceste temperaturi au servit ca date de intrare pentru o analiză structurală cu Elemente Finite (FEA) realizată cu Ansys Workbench 2023 R1. Rezultatele arată că încălzirea Joule cauzează dilatare termică, iar temperaturile diferite ale materialelor și coeficienții de dilatare generează solicitări și deformații. Solicitarea și deformația echivalentă au depășit valorile limită de curgere ale materialului, indicând o potențială fisurare a cuprului la temperaturi ridicate sau în cicluri termice. Metoda FEA termică și structurală prezice în mod eficient solicitarea și deformația termică în PCB-uri.