

BULETINUL INSTITUTULUI POLITEHNIC DIN IAȘI

Publicat de

Universitatea Tehnică „Gheorghe Asachi” din Iași

Volumul 69 (73), Numărul 1, 2023

Secția

CONSTRUCȚII DE MAȘINI

DOI:10.2478/bipcm-2023-0007



A STUDY ON THE INFLUENCE OF THERMAL INTERFACE MATERIAL IN PASSIVELY COOLED ELECTRICAL COMPONENTS WITH ATTACHED HEATSINK

BY

**GEORGE-GABRIEL CHIRIAC^{1,*}, CĂTĂLIN GABRIEL DUMITRAȘ¹ and
IONUȚ MERTIC²**

¹“Gheorghe Asachi” Technical University of Iași, Romania,
Department of Machine Tools

²S.C. Preh Romania S.R.L.

Received: April 2, 2023

Accepted for publication: April 20, 2023

Abstract. This paper studies the effect of the presence of a thermal interface material between a heat source and a heat sink. A heat sink is used as a passive cooling device for a thick film heating element used as a heat source. The heating element is fixed using 2 screws. The temperatures are measured in 2 scenarios: first, where the heating element is placed directly on the heat sink, and second, where a thermal pad is placed between the heat source and heat sink. Using a similar CAD model as the one tested and the same boundary conditions, finite element analysis were solved. The goal is to see if the finite element method delivers similar results as the measured ones and if the method is reliable in predicting the thermal behaviour of the electronic components. Temperature measurements show an improvement in heat source cooling when the thermal interface material is placed between the heat source and heating element. Also, the temperatures from the finite element analysis are similar to the measured ones, making the method reliable for similar applications.

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

© 2023 George-Gabriel Chiriac et al.

This is an open access article licensed under the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International License (CC BY-NC-ND 4.0).

Keywords: FEA, heat sink, thermal, passive cooling, thermal pad, thermal interface material.

1. Introduction

Electronic components cooling is and will always be a challenge. The electronic components used in the automotive industry are more efficient but dissipate more power in the same or a smaller volume than before. This causes the need for improving the cooling of the heat sources and predicting the thermal behaviour of the assemblies (Moore and Shi, 2014).

Heat transfer is done through three transfer methods: conduction, convection and radiation. When cooling an electronic component, the greatest amount of heat can be transferred away from the heat source by conduction.

Heat conduction can be found inside solid bodies, but also between 2 or more solid bodies that are in contact with each other. Since geometric tolerances and roughness prevent us from having perfect contact between bodies, air gaps are always present between them (Rajput, 2012).

Air gaps between solid bodies decrease the thermal transfer efficiency by increasing the thermal resistance from the heat source to the cooler or to the ambient (Rajput, 2007).

Using thermal interface materials such as thermal pads or thermal paste between two or more solid bodies, we can remove the air gaps between them and try to improve the cooling by reducing the thermal resistance between them.

To find the best product design and to predict the thermal behaviour of the product, finite element analysis is often used in the automotive design industry (Toprak *et al.*, 2022).

The goal of this paper is to observe if the cooling can be improved by adding a thermal pad between a heat source and a heatsink in the passive cooled system. An additional goal is to simulate the same geometry using the same boundary conditions and to conclude if finite element analysis can be a reliable method in predicting the thermal behaviour of an assembly or product.

Simcenter Siemens Flotherm XT is used to solve the finite element simulations. Krane *et al.* (2022) used Flotherm to predict the thermal behaviour of electronic components with good results.

2. Experimental study and Finite Element Analysis results

To investigate the influence of thermal interface material in heat conduction inside a system, a test assembly was created. The same geometry was created in a 3D virtual model used for the finite element analysis. The software used for the finite element simulations is Siemens Simcenter Flotherm XT 2021.2.

To evaluate the heat transfer difference that the thermal interface material is generating, a heat source is needed. For this, Telpod HTS-16-230-300-3/6.3 thick film heating element was used to generate heat. A power supply was used to supply 56.7 V and 0.3 A to the heating element to gain a total power dissipated of 17 W. The same total power is used inside the simulation.

The thick film heating element has 75.6 mm length, 40 mm width and 1 mm thickness. Two 4.2 mm diameter holes are present at 28 mm distance between them, making the heating element suitable to be mounted using screws.

In all the scenarios, the thick film heating element was fixed on a heatsink using 2 M4 screws. The contact pressure between the thick film heating element and heatsink is unknown and not measured. Since the assembly will replicate a passive cooling scenario, an aluminum finned heatsink Relpol RH17A is used.

Seven RS Pro Type K thermocouples are used to measure the temperatures. One is reserved for measuring the ambient temperature, while the other six were placed on the heat source and heatsink and their placement is presented in Fig. 1 and Table 1.

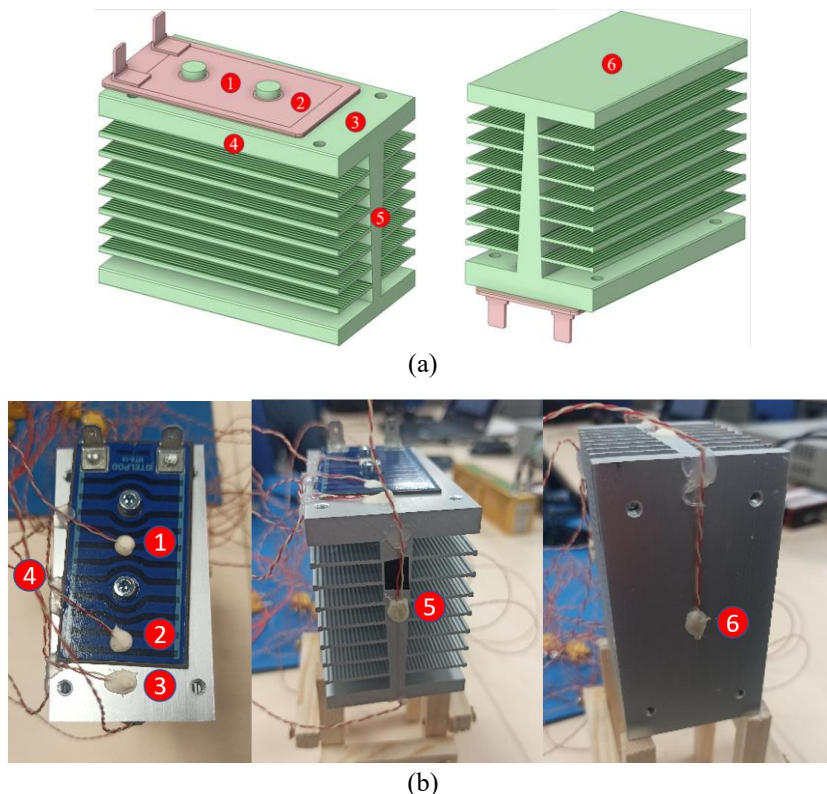


Fig. 1 – (a) Thermocouple placement in CAD model; (b) Thermocouple placement in test scenario.

Table 1
Thermocouple placement

Thermocouple number	Thermocouple placement part	Details
1	Heat source	Center of the heat source
2	Heat source	Side of the heat source
3	Heat sink	Side area, next to the heat source
4	Heat sink	Side area of the heat sink, next to the heat source middle
5	Heat sink	Top of the heat sink; central column;
6	Heat sink	Opposite side of the heat source, center area
7	Ambient	

The heat sink fins are perpendicular to the ground and the gravity vector is parallel to them. This scenario will allow the natural air flow generated by the heated air to pass between the heatsink fins. This scenario is further called Setup 1 and can be seen in Fig. 2 as a virtual model.

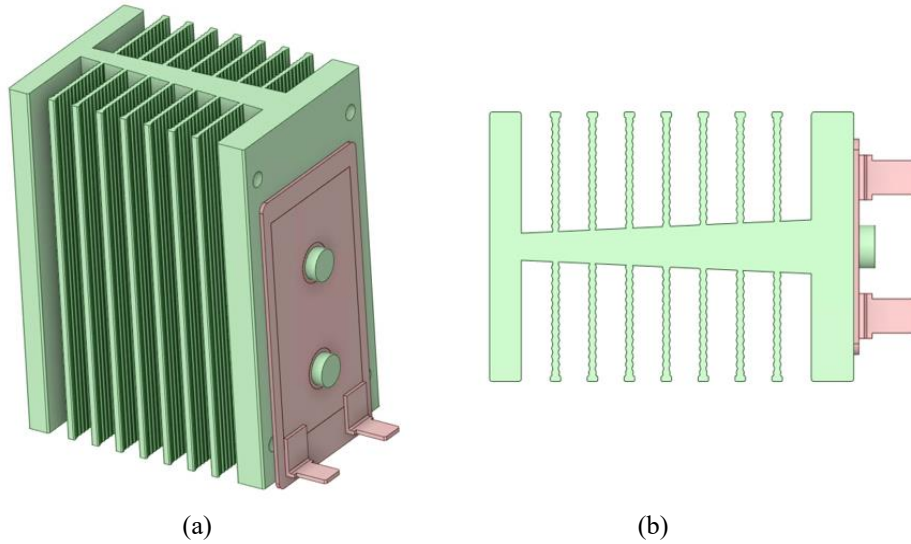


Fig. 2 – (a) Isometric view of Setup 1; (b) Top view of Setup 1.

Gravity vector orientation and the experimental assembly can be observed in Fig. 3.

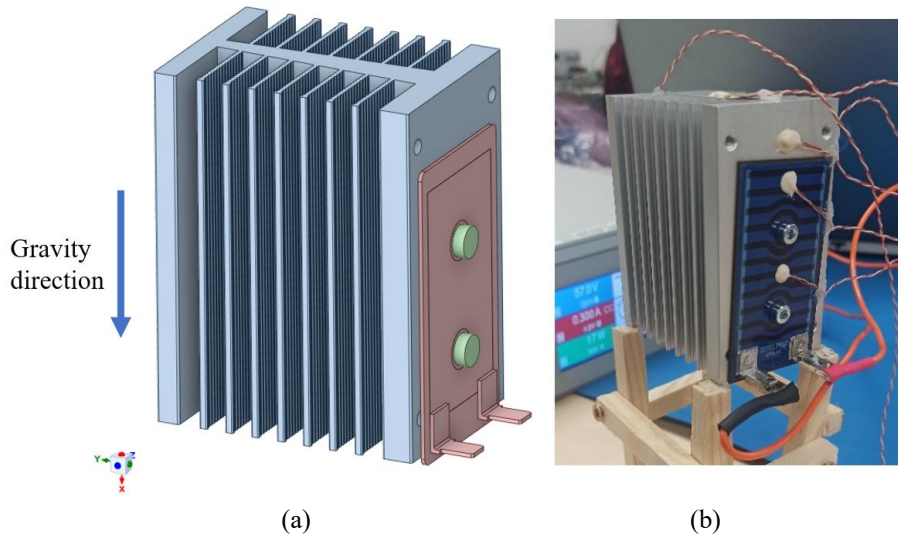


Fig. 3 – (a) Gravity orientation; (b) Setup 1 temperature measurement.

To create the Setup 2, a thermal gap pad was used as a thermal interface material between the thick film heating element and heatsink. The virtual model of Setup 2 can be observed in Fig. 4.

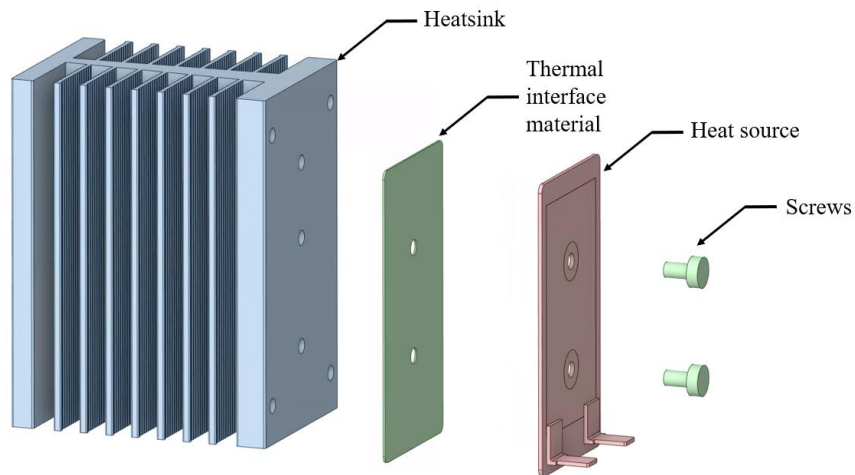


Fig. 4 – Exploded view of Setup 2 virtual model.

The thermal gap pad used as a thermal interface material is Bergquist Gap Pad 5000S35 with a thermal conductivity of 5 W/(mK) and an initial thickness of 1 mm . After being pressed between the heat source and heatsink, to remove all the air gaps and create a proper contact between the components, the thickness of the gap pad was reduced, therefore in the simulation CAD model, a

0.5 mm thickness is used for this component. Gravity orientation of Setup 2 and the experimental assembly can be observed in Fig. 5.

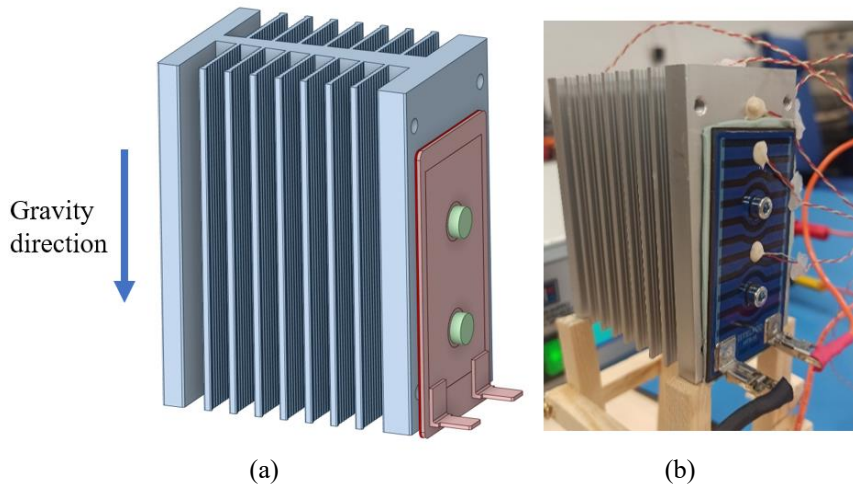


Fig. 5 – (a) Gravity orientation of Setup 2; (b) Setup 2 temperature measurement.

To simulate the absence of thermal interface material, a thermal resistance was defined between the heat sink and heat source. To do this, a thermal insulance of $0.003 \text{ (Km}^2\text{)/W}$ is set in the simulation setup. When a thermal interface material is added, the thermal insulance is removed from the boundary conditions. After both scenarios were tested and temperature measurement were done, FEA simulations were solved for both setups in order to see the reliability of the FEA simulation and to compare the measured and simulated temperature results. The results obtained from the measurements can be seen in the Table 2.

Table 2
Measured temperature results

	Setup 1	Setup 2
thermocouple number	Measurement [$^{\circ}\text{C}$]	Measurement [$^{\circ}\text{C}$]
1	72.5	65.6
2	74.2	63.6
3	57.4	60.6
4	58.0	61.5
5	56.0	59.5
6	54.4	57.7
7	27.3	28.7

Because the ambient temperature was not controlled and was changing, the measurements were performed at different ambient temperatures. In order to reduce the difference caused by the ambient influence, all the measured temperatures from Setup 2 will be adjusted. To adjust the temperature values, 1.4°C will be subtracted from all the values. This value resulted as a difference between the ambient temperature of the 2 setups. The resulted values can be seen in Table 3.

Table 3
Adjusted measured temperature results

	Setup 1	Setup 2
thermocouple number	Measurement [$^{\circ}\text{C}$]	Measurement [$^{\circ}\text{C}$]
1	72.5	64.2
2	74.2	62.2
3	57.4	59.2
4	58.0	60.1
5	56.0	58.0
6	54.4	56.2
7	27.3	27.3

By comparing the results of Setup 1 and Setup 2, we can clearly see that the thermal interface material improves the cooling of the heat source. The temperature measured by the first 2 thermocouples have a difference of 8.3°C and 11.2°C . We can conclude that the thermal interface material has an important role in cooling the heat source.

The thermocouples placed on the heat sink saw an increase in the measured temperature. This is because the thermal interface material improved the heat transfer between the heat source and heat sink. Since the heat sink is heated more in the Setup 2 because of the improved transfer, it was expected that the measured temperatures may increase.

In Table 4 the simulation results can be seen for both scenarios. The FEA simulations tried to replicate the same scenarios. Since not all the boundary conditions were exactly known and since the measuring devices also have a measurement tolerance, differences between the simulation and measurement were expected and are normal. RS PRO Class 1 Type k thermocouples were used to measure the temperature. This model of thermocouples have an accuracy of $\pm 1.5^{\circ}\text{C}$. The thermocouples were connected to a Ipetronik M-Thermo 16 receiver with a typical accuracy of $\pm 0.025\%$ at an ambient temperature of 25°C .

Table 4
Adjusted measured temperature results

thermocouple number	Setup 1		Setup 2	
	Measurement [°C]	Simulation [°C]	Measurement [°C]	Simulation [°C]
1	72.5	76.0	64.2	63.5
2	74.2	75.6	62.2	63.0
3	57.4	60.8	59.2	61.1
4	58.0	61.8	60.1	62.3
5	56.0	58.9	58.1	59.3
6	54.4	57.0	56.3	57.2
7	27.3	27.3	27.3	27.3

A mesh independent investigation was performed before solving all the simulations, to make sure that the results is not influenced by the mesh size or the total number of elements used in solving the FEA simulation.

In Table 5 an overview of the measured and simulated temperatures can be seen. The difference between the measured temperatures and be seen in the second to last collumn. In the last collumng, the difference between the simulated temperatures can be observed.

Table 5
Difference between Setup 1 and Setup 2 temperatures

thermocouple number	Measurement [°C]	Simulation [°C]	Measurement [°C]	Simulation [°C]	Delta T measurement	Delta T simulation
	Setup 1		Setup 2			
1	72.5	76.0	64.2	63.5	8.3	12.6
2	74.2	75.6	62.2	63.0	11.9	12.6
3	57.4	60.8	59.2	61.1	-1.9	-0.3
4	58.0	61.8	60.1	62.3	-2.1	-0.5
5	56.0	58.9	58.1	59.3	-2.1	-0.4
6	54.4	57.0	56.3	57.2	-1.8	-0.3
7	27.3	27.3	27.3	27.3	0	0

We can see that the same behaviour is observed in the FEA simulation, as in the measurements. The temperatures measured on the heat source are smaller in the second setup. Also, the temperatures on the heatsink are not

significantly reduced, so we can say that the simulation is able to replicate the same real-life behaviour.

3. Conclusions

Even if the assembly of the thick film heating element was made using 2 screws that generated a contact pressure between the parts, an air gap that generated a thermal resistance was still present.

Thermal interface material fills the air gaps between the heat source and heat sink, improving in this way the cooling. Thermal measurements show smaller temperatures on the heat source when thermal interface material is used.

Even if the thermal interface material adds an additional layer or material between the heat source and heat sink, it has a higher thermal conductivity than the air and the cooling is improved using this method.

Finite element analysis using Siemens Simcenter Flotherm XT shows similar results and similar behaviour as in the real life testing. Using a geometry close to reality, with minimal simplifications or assumptions, the simulation results are similar with the measured ones, for the same areas of the assembly.

After comparing the results of the measurements and simulations, we can conclude that finite element analysis it's an useful and fast method to predict the thermal behaviour of an assembly.

Acknowledgements. This paper is the result of a study that will be included in the doctoral work developed at the "Gheorghe Asachi" Technical University of Iași.

REFERENCES

- Krane P., Gonzalez C.D., Lozano F., Paniagua G., Marconnet A., *Sensitivity coefficient-based inverse heat conduction method for identifying hot spots in electronics packages: A comparison of grid-refinement methods*, J. Electron. Packag. Trans. ASME, 2022, **144**, 011008.
- Moore A.L., Shi L., *Emerging challenges and materials for thermal management of electronics*, Mater. Today, 2014, **17**, 163-174.
- Rajput R.K., *Engineering Thermodynamics*, Third Edition, New Delhi: Laxmi Publications Ltd., 2007.
- Rajput R.K., *A Textbook of Heat and Mass Transfer*. New Delhi: S. Chand & Company Ltd., 2012.
- Toprak B.İ., Oskouei S.B., Bayer Ö., Solmaz İ., *Experimental and numerical investigation of a novel pipe-network mini channel heatsink*, Int. Commun. Heat Mass Transf, 2022, **136**, 106212.

STUDIU ASUPRA INFLUENȚEI MATERIALULUI DE
TRANSFER TERMIC AL COMPONENTELOR ELECTRONICE RĂCITE
PASIV CU AJUTORUL UNUI RADIATOR

(Rezumat)

Această lucrare studiază efectul generat de prezența unui material de transfer termic între o sursă de căldură și un radiator pasiv. Radiatorul este utilizat în acest caz pentru a îmbunătăți răcirea unui element rezistiv utilizat ca și sursă de căldură. Sursa de căldură este asamblată utilizând două șuruburi. Temperatura componentelor a fost măsurată în două scenarii, primul în care sursa de căldură a fost fixată în contact direct cu radiatorul și un al doilea scenariu, în care un material de transfer termic a fost utilizat între sursa de căldură și radiator.

Utilizând un model virtual similar cu cel testat și aceleași condiții la limită, au fost rezolvate analize cu element finit. Scopul este de a observa dacă în cadrul analizei cu element finit rezultatele obținute sunt similare cu cele măsurate experimental și dacă astfel, metoda elementului finit este utilă în observarea comportamentului termic al componentelor electronice.

Temperaturile măsurate au arătat că utilizarea unui material de transfer termic între sursa de căldură și radiator îmbunătățește răcirea sursei de căldură. De asemenea, temperaturile obținute în cadrul analizei cu element finit sunt similare cu cele măsurate, această metodă fiind una utilă pentru aplicații similare.