Modelling & Analysis of Cooling of Electronic Packaging Using Synthetic Jet Impingement

Satyajit.S.Dhore[1], Kanchan.D.Rajmane[2], Rahul.R.Gaherwar[3], Somnath.P.Ghavate[4], Suraj.D.Bhete[5], Abhishek.A.Shinde[6]

[1]Asst. Prof., Dept of Mechanical Engineering
[2, 3, 4, 5, 6]Dept of Mechanical Engineering
[1, 2, 3, 4, 5, 6] Dr. D. Y. Patil College of Engineering & Innovation, Varale, Pune, INDIA

Abstract- *Recently, the cooling process for electronics components has attracted many researchers and several techniques for improving the cooling efficiency and heat transfer rate have been demonstrated. One of the best efficient techniques is the introduction of a synthetic jet. The synthetic jet releases air by collecting from surrounding. It has a hollow cylindrical body with single port as orifice which works as an input and output also. Opposite end to the orifice actuator has fitted. Actuator fluctuate its piston or diaphragm and this action cause air jet. When actuators move towards orifice it imparts air towards specimen and actuators move far from the orifice it produces vacuum and surrounding air is fill inside due to pressure goes decreasing below atmospheric pressure. In the present study, the mathematical calculation for cooling effect is done and practically readings are taken for cooling effect. The mathematical calculation is then experimentally validated by taking the readings. Further the experimental values then validated by using Computational fluid dynamics in Ansys software. In CFD velocity and distance between surface to be cooled and orifice are change according to practical readings and the results are validated.*

Keywords- Synthetic Jet, CFD, Cooling effect etc.

I. INTRODUCTION

Synthetic jet releases air by collecting from surrounding. It has a hollow cylindrical body with single port as orifice which works as an input and output also. Opposite end to the orifice actuator has fitted. Actuator fluctuate its piston or diaphragm and this action cause air jet. When actuators move towards orifice it imparts air towards specimen and actuators move far from the orifice it produces vacuum and surrounding air is fill inside due to pressure goes decreasing below atmospheric pressure. A novel multiple orifice synthetic jet cavity is designed and its performance characterized. The utility of synthetic jet for practical electronics cooling is demonstrated by quantifying the behaviour of the thermal resistance of a heat sink employed along with an impinging synthetic jet, and by measuring the average heat transfer coefficient in a narrow confined enclosure in the form factor of a flow duct that is typical of widely used electronic system. It is shows that synthetic jet can be used for localized cooling of electronic components in the duct. In the study, a synthetic jet is created for a single and multi-nozzle orifice with the help of vibrations of sound system. The heat transfer characteristics of a synthetic jet are studied in this work. The maximum heat transfer coefficient with the synthetic jet is found to be 9.6 times more than that of natural convection and 3 times more than that of cooling fan used in electronic device.

II. PROBLEM STATEMENT

While designing the synthetic jet impingement machine the various design consideration are taken in account such as actuator media, distance between orifice and heating plate, orifice angle with respect to hot plate and orifice shapes are taken into consideration. The practical readings are taken by doing various experiments but the readings are not validated in computational fluid dynamics. Hence to validate the practical physical model. The model is then design in software and validated in Ansys.

III. OBJECTIVE

The objective of the research are as follows:

- The objective of jet air cooling is to do CFD analysis in Ansys fluent workbench.
- To change the air velocity and distance in iterations.
- To validate the mathematical calculation with practical readings.
- To validate it practical readings by using computational fluid dynamics.

IV. SCOPE

The scope of project is to validate the practical readings with computational fluid dynamics simulation. The project is designed mathematically first and practical readings are taken, then the modelling of project is to be done on

Solidworks and simulation is done on Ansys. The reading will be taken by changing the distance between office and heating plate and also angle of inclination is changes. The number of holes and shapes of orifice will also change.

V. METHODOLOGY

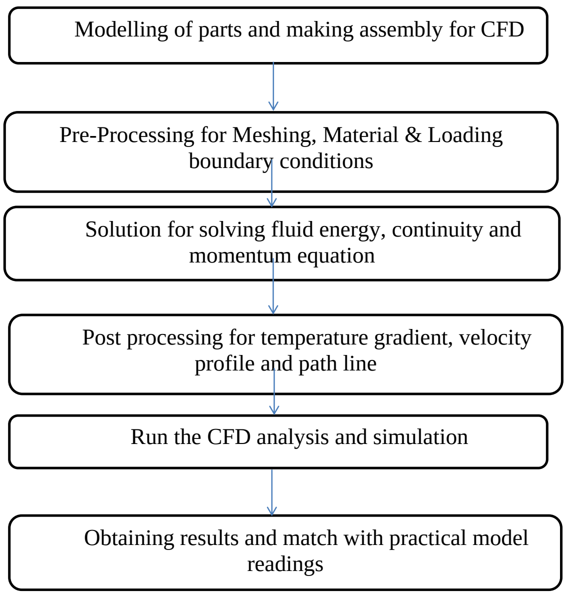

Figure 1 Methodology

VI. ANSYS SIMULATIONT

CAD modelling was done in Solidworks 2018 and fluid model was created in Ansys Design Modeller is used for creating cad models & Ansys CFX 19.2 is used to calculate temperature drop due to synthetic jet by performing CFD analysis.

A. Geometry Detail:

Solid works is use to create CAD geometry and Ansys Design Modeller is for de-featuring of geometry. Geometry prepared in solid works is as below.

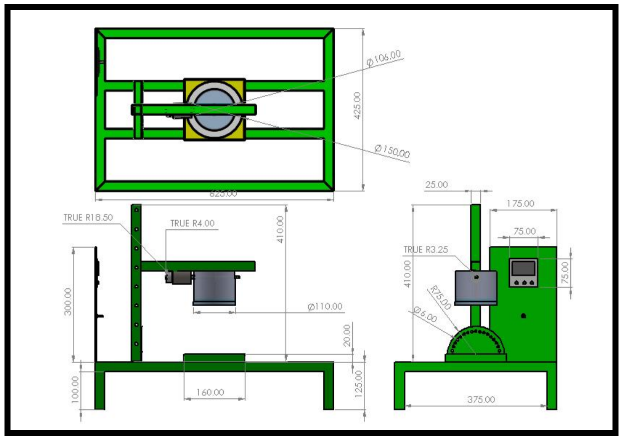

Figure 2 Dimensional drawing of synthetic jet machine

After doing de-featuring in ANSYS DM, below is the final fluid domain considered for CFD analysis as below,

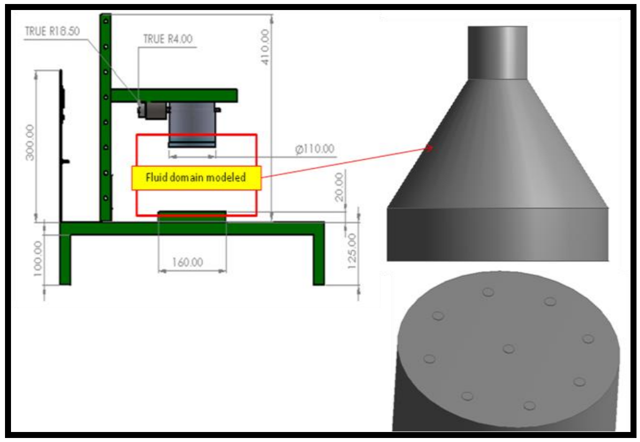

Figure 3 Fluid geometry

Analysis is performed for below cases:

1) Distance of 133mm and air velocity at 6.2 m/s
2) Distance of 100mm and air velocity at 6.2 m/s
3) Distance of 100mm and air velocity at 2 m/s
4) Distance of 100mm and air velocity at 2 m/s

1) CFD Results Case1:

Temperature plot: The temperature distribution of the plate surface shown as below and observed temperature drop from 70Deg C to 64DegC.

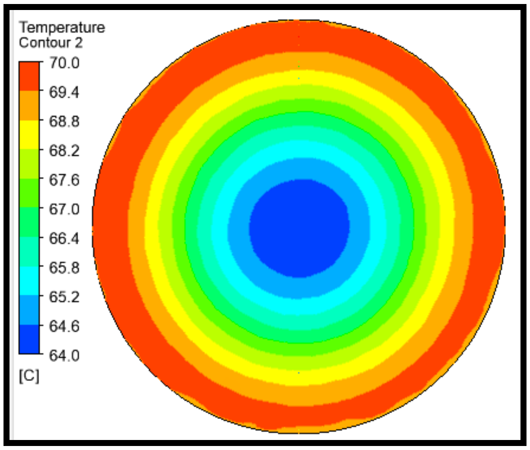

Figure 4 Temp plot of Plate for case1

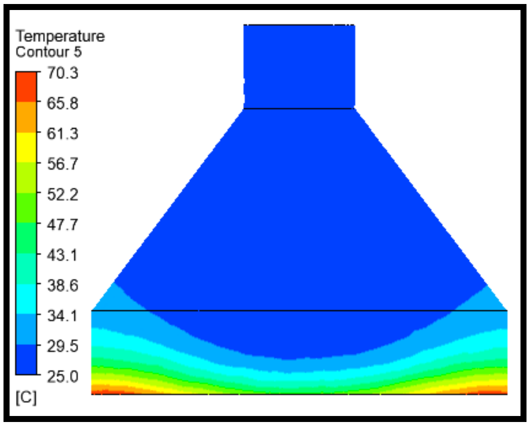

Figure 5 Temp plot of air for case1

The velocity plot: Shows that the flows at a high speed at inlet supporting the turbulent flow assumption, also the plot are helpful to understand the development of flow during jet cooling.

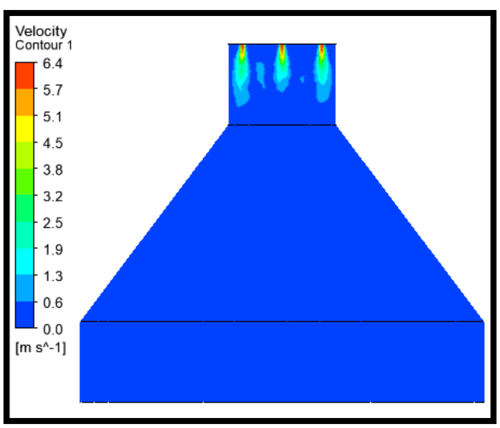

Figure 6 Velocity plot of air for case1

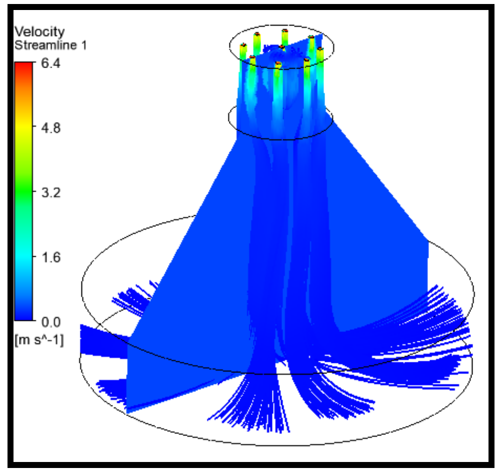

Figure 7 path lines of air for case1

2) CFD Results Case2:

Temperature plot: The temperature distribution of the plate surface shown as below and observed temperature drop from 70Deg C to 55DegC.

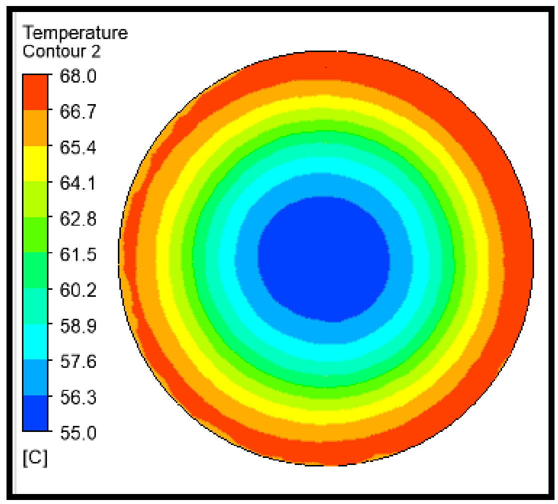

Figure 8 Temp plot of Plate for case2

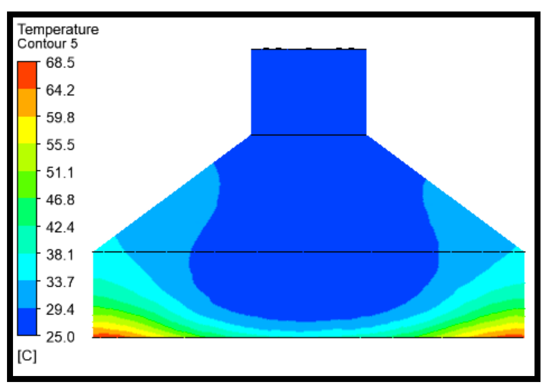

Figure 9 Temp plot of air for case2

The velocity plot: Shows that the flows at a high speed at inlet supporting the turbulent flow assumption, also the plot are helpful to understand the development of flow during jet cooling.

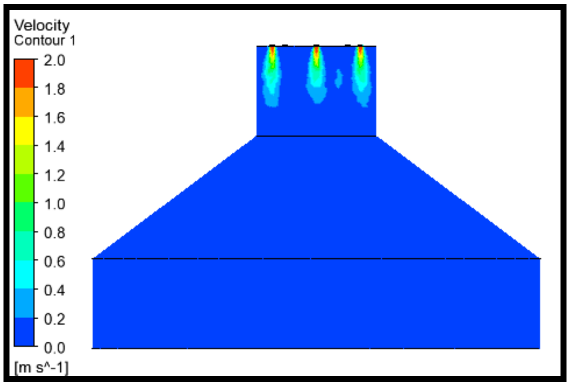

Figure 10 Velocity plot of air for case2

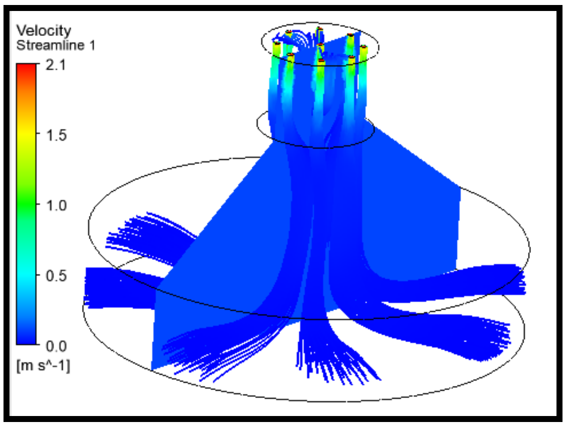

Figure 11 path lines of air for case2

3) CFD Results Case3:

Temperature plot: The temperature distribution of the plate surface shown as below and observed temperature drop from 70Deg C to 66DegC.

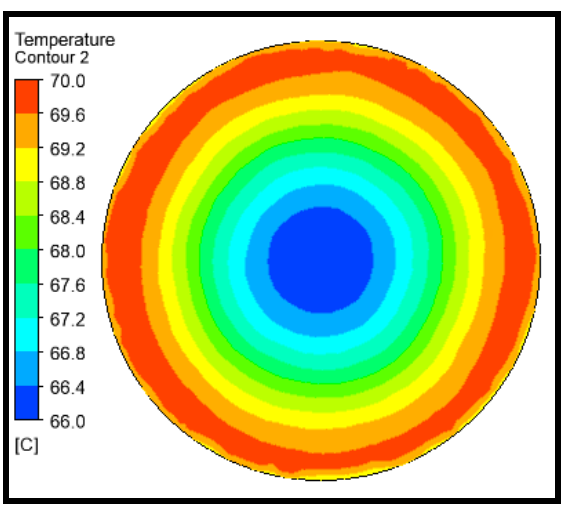

Figure 12 Temp plot of plate for case3

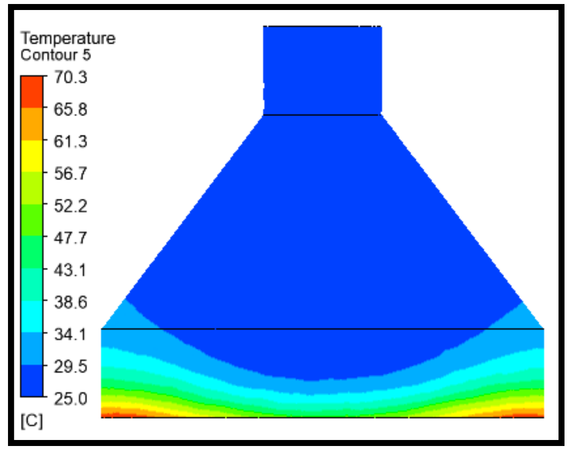

Figure 13 Temp plot of air for case3

The velocity plot: Shows that the flows at a high speed at inlet supporting the turbulent flow assumption, also the plot are helpful to understand the development of flow during jet cooling.

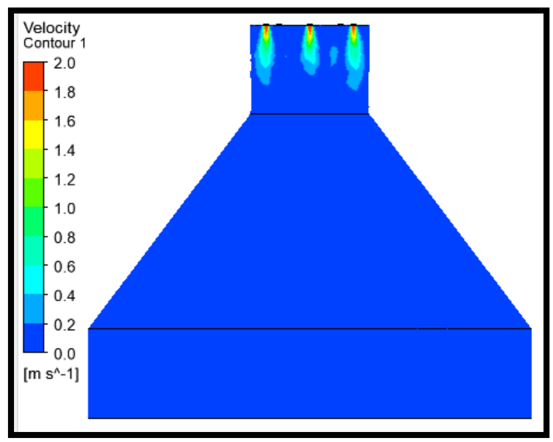

Figure 14 Velocity plot of air for case3

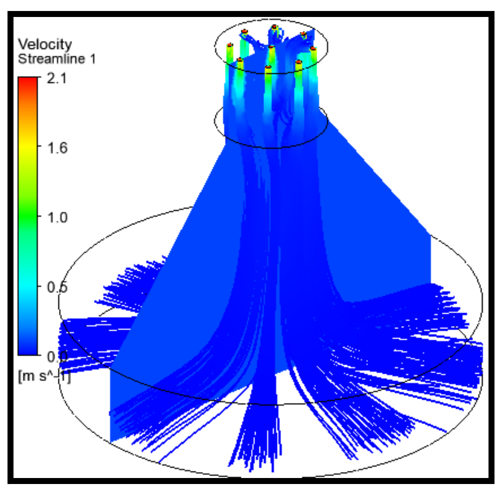

Figure 15 path lines of air for case3

4) CFD Results Case4:

Temperature plot: The temperature distribution of the plate surface shown as below and observed temperature drop from 70Deg C to 60DegC.

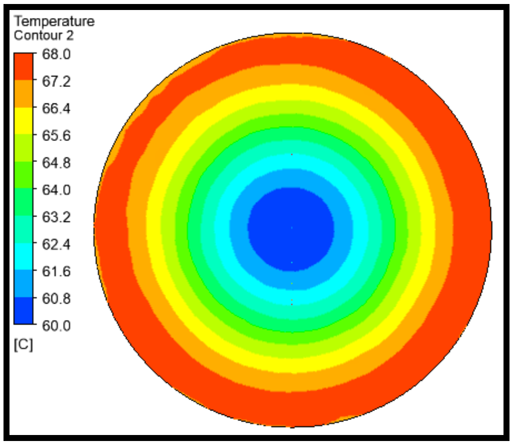

Figure 16 Temp plot of plate for case4

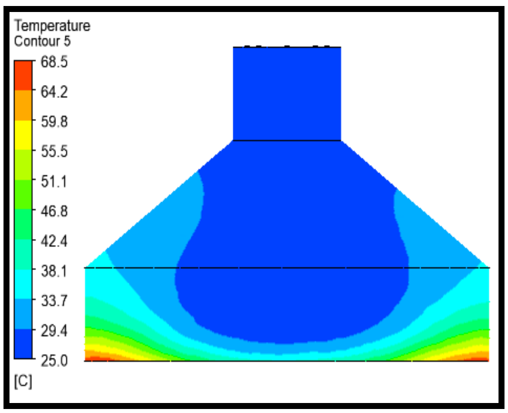

Figure 17 Temp plot of air for case4

The velocity plot: Shows that the flows at a high speed at inlet supporting the turbulent flow assumption, also the plot are helpful to understand the development of flow during jet cooling.

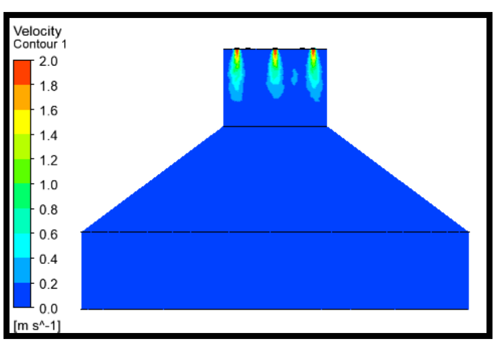

Figure 18 Velocity plot of air for case4

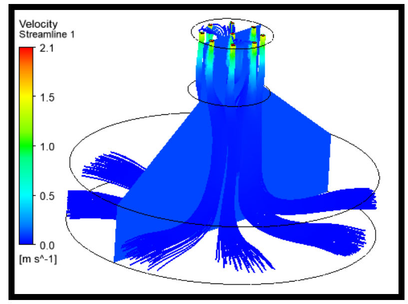

Figure 19 path lines of air for case4

VII. EXPERIMENTAL VALIDATION

The Jet Air cooling CFD analysis in ANSY fluent was successfully completed for below cases:

- Air velocity at 6.2 m/s and ambient temperature of 25 Deg C at inlet location and distance of 133 mm
- Air velocity at 6.2 m/s and ambient temperature of 25 Deg C at inlet location and distance of 100 mm
- Air velocity at 2 m/s and ambient temperature of 25 Deg C at inlet location and distance of 133 mm
- Air velocity at 2 m/s and ambient temperature of 25 Deg C at inlet location and distance of 100 mm

TABLE I

Case	**Final results**			
	Distance (mm)	*Air velocity (m/s)*	*Initial temp (degC)*	*Temperature drop (degC)*
1	133	6.2	70	64
2	100	6.2	70	55
3	133	2	70	66
4	100	2	70	60

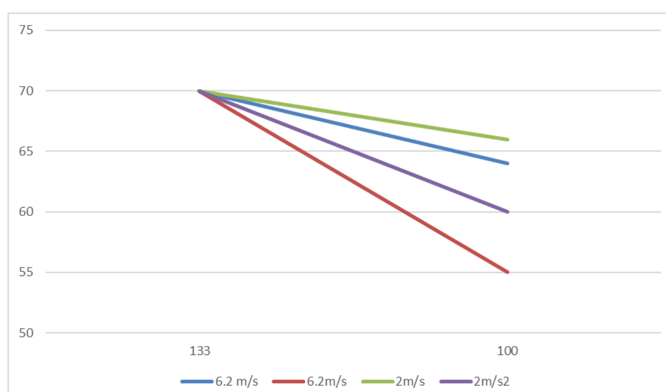

Figure 20 Generated graph for results

VIII. CONCLUSION

In this project the design of Synthetic Jet Impingement system is successfully done and CAD model is properly designed with exact dimensions. The model is then converted into Ansys file for simulation work and CFD. In simulation the properties of material used for designing this system is given as an input value and properties is provided in proper tabular form. The CFD simulation is done on Ansys workbench and result is concluded. The temperature drop is obtained of 55 deg C for 6.2m/s air velocity. From the above reading it is concluded that Synthetic Jet Impingement system is efficiently working and it consumes zero electricity. The Synthetic Jet Impingement system CFD analysis is done and temperature drop at different velocity is obtained. Temperature and Distance graph for heating plate is also plotted. We can also come to a conclusion for higher heat transfer performance due to change in the parameters and numerical analysis.

IX. FUTURE SCOPE

This study will be carried out using air as the coolant medium that impinge with fluctuating flow region from the orifices to the heat source. This study will start with different pattern shape obtain by the orifice. The heat source is also at constant state and has limit of heat dissipation also dimensions of hot region are kept constant. So beyond this we can go and bring out more conclusions and also we add more parameters to study and further analysis can be done. This study and setup will hold general cooling system constraints and they will have global status of usages which make this experiment study more efficient and useful. Changing some parameters and keeping other known parameters will also support the effective way of study and analysis.

REFERENCES

[1] Vassighi, M. Sachdev, Thermal and Power Management of Integrated Cir- cuits, Springer, 2006, doi: 10.1007/0-387- 29749- 9 .

[2] W. Gao, W. Wang, D. Dimitrov, Y. Wang, Nano properties analysis via fourth multiplicative ABC indicator calculating, J. Arab. J. C. 11 (2018) 793–801https: //doi.org/10.1016/j.arabjc.2017.12.024 .

[3] M. Bahiraei, S. Heshmatian, M. Keshavarzi, Multi-attribute optimization of a novel micro liquid block working with green graphene nanofluid regarding preferences of decision maker, J. Appl. Therm. Eng. 143 (2018) 11–21 https: //doi.org/10.1016/j.applthermaleng.2018.07.074 .

[4] M. Bahiraei, S. Heshmatian, Thermal performance and second law character- istics of two new microchannel heat sinks operated with hybrid nanofluid containing graphene–silver nanoparticles, J. En. Con. Man. 168 (2018) 357–370 https://doi.org/10.1016/j.enconman.2018.05.020 .

[5] M. Bahiraei, S. Heshmatian, M. Keshavarzi, A decision-making based method to optimize energy efficiency of ecofriendly nanofluid flow inside a new heat sink enhanced with flow distributor, J. Pow. Tec. 342 (2019) 85–98 https://doi. org/10.1016/j.powtec.2018.10.007 .

[6] M. Bahiraei, S. Heshmatian, Electronics cooling with nanofluids: A critical review, J. En. Con. Man. 172 (15) (2018) 438–456 https://doi.org/10.1016/j. enconman.2018.07.047 .

[7] M. Goodarzi, I. Tlili, Z. Tian, M. Safaei, Efficiency assessment of using graphene nanoplatelets-silver/water nanofluids in microchannel heat sinks with different cross-sections for electronics cooling, Int. J. Num. Meth. Heat Fluid Flow, Vol. ahead-of-print No. ahead-of-print. https://doi.org/10.1108/ HFF- 12- 2018- 0730

[8] M.R. Safaei, M. Gooarzi, O.A. Akbari, M.S. Shadloo, M. Dahari, Performance evaluation of nanofluids in an inclined ribbed microchannel for electronic cooling applications, electronics cooling, S M Sohel Murshed, IntechOpen, doi: 10.5772/62898

[9] R. Dadsetani, M. Salimpour, M. Tavakoli, M. Goodarzi, E. Pedone Bandarra Filho, Thermal and mechanical design of reverting microchannels for cool- ing disk-shaped electronic parts using constructal theory, Int. J. Num. Meth. Heat Fluid Flow, Vol. ahead-of-print No. ahead-of-print. https://doi.org/10. 1108/HFF- 06- 2019- 0453.

[10] M. Bahiraei, S. Heshmatian, M. Goodarzi, H. Moayedi, CFD analysis of employ- ing a novel ecofriendly nanofluid in a miniature pin fin heat sink for cooling of electronic components: Effect of different configurations, J. Pow. Tec. 30 (2019) 2503–2516 https://doi.org/10.1016/j.apt.2019.07.029.