iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 10 **Issue:** IV **Month of publication:** April 2022

DOI: https://doi.org/10.22214/ijraset.2022.41740

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IV Apr 2022- Available at www.ijraset.com

Multi-objective Heat Sink Optimization by Using Taguchi Method

Anil Shinde[1], Ajit Mane[2], Adinath Palake[3], Balkrushan Mohite[4], Akash Khot[5], Tirupati Vhatkar[6], Ankur Kharavate[7]
[1, 2, 3, 4, 5, 6, 7]*Annasaheb Dange College of Engineering and Technology, Ashta*

Abstract: Nowadays, with the rapid growth in electronic devices, compact and efficient heat sinks are required in day-to-day life. That's why it's need to be use optimized heat sinks. The excess heat generated causes damage in different parts also generates major problems in the system. So, in this work three-dimensional square channel of copper material is used to study heat transfer coefficient and pressure drop. The comparative numerical analysis of thermal management system different parameter like coolant fluid, inlet velocity of fluid, shape of obstacles, aspect ratio of obstacles and number of obstacles is presented in this paper. Analysis and optimization of the heat sink are done by three different methods i.e., experimental, analytical, and simulation. In this work analytical and simulation method is used for the optimization of a heat sink. There are many software's are available for analysis and optimization, by using the "COMSOL Multi-physics software" we optimize multi objectives heat sink. After analytical and numerical analysis, to optimize best result Taguchi method is preferred, for that MINITAB software is used.
Keywords: Heat transfer coefficient, pressure drop, micro heat sink, Taguchi method, Analysis, optimization, COMSOL Multi-physics software, MINITAB software.

I. INTRODUCTION

As improvements in technologies electronics devices become more compact so, the design of a heat sink is more important for dissipating heat from the device to the atmosphere. Heat transfer from the device impacts the performance of the device. To better performance, maximum amount of excess heat should dissipate properly and maintain temperature under working conditions. So, to cool devices, there is a need to increase heat transfer surface area and heat transfer coefficient. Also, the pressure drop is affected by heat transfer. To increase heat transfer surface area adds obstacles in the path of fluid for those different shapes and sizes of dimples are placed inside the channel. There are different methods to design a heat sink i.e., analytical, experimental, and simulation. Here heat sink design by using simulation method for that COMSOL Multiphysics software is used. The micro-channel heat sink of copper material with a specific size and dimension is validated with existing research work. The same size of micro-channel heat sink geometry is used for further work. To the better performance of heat transfer, the heat sink design was checked on different parameters. Different types of Coolant fluid, no. of an obstacle, aspect ratio of obstacles, the shape of obstacles, and spacing between obstacles this different five parameters used to design heat sink. Heat sink performance measures in terms of heat transfer coefficient and pressure drop. Optimization of heat sink done by using Taguchi method on MINITAB software. All results from COMSOL software analysis are put on Minitab and using that optimization is carried out.

II. LITERATURE REVIEW

Copper microchannel is observed to be having the best performance. Jadhav et al. [1]. In the presence of rectangular shape obstacles in micro-channel, turbulence is created and heat transfer rate is increased as compared with micro-channel with normal shape. Wadekar et al. [2]. The numerical analysis and the enhancement in the heat transfer performance with dimple. Jadhav et al. [3]. The Fin base is the most significant variable for improving design of heat sink. Deshmukh et al. [4]. The minimum thermal resistance is always attained at the largest allowable aspect ratio. In practical designs, the aspect ratio would be determined by the limits on the micro channel depth and the substrate thickness. Liu et al. [5]. Extended Surface (Fin) is used to improve the heat transfer from surfaces in a wide variety of applications. The fin content is usually high in thermal conductivity. Bhalerao et al. [6]. The thermal resistance of a heat sink can be obtained to illustrate the cooling performance under various design conditions. Nafar et al. [7]. The Optimization of multiple performance characteristics of the heat sink such as the average heat transfer coefficient, pressure drop, thermal resistance, mass and the radiations emitted has been carried out using Taguchi-based GRA technique. Manivannan et al. [8]. The traditional rectangular plate fin heat sinks widely used by thermal engineers for electronic cooling was set as the base of performance. Amoako et al. [9]. The work carried out on Micro-channel heat sinks with different sizing, geometries, and cross-sections, using turbulators, single and double layers, and topology optimized heat sink. Hussein et al. [10].

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IV Apr 2022- Available at www.ijraset.com

Thermal Design and Analysis of Heat Sink Optimization and its Comparison with Commercially Available Heat Sink. Prabhakar et al. [11]. The multi-objective optimization of LED Heatsink with the help of MOGA evolutionary approach and all the design parameters have certain effect on the thermal performance of the LED Heatsink. Suresh et al. [12]

For deciding the performance of heat sink it is necessary to examine which parameters are affect on heat transfer coefficient. The physical properties are affected on heat transfer coefficient in the term of size and shape. The surface area of contact with fluid, affects on heat transfer coefficient so, it is requirement to analysis heat transfer coefficient by using different geometrical (square, circle, elliptical) obstacles. Mostly water is used as cooling material but in modern engineering it is demand to find alternative fluid for water instead of air.

III. MODEL SPECIFICATIONS

In this paper, the analysis of the heat sink is carried out by using the software COMSOL Multiphysics. It is appropriate to compare the performance of heat sinks by using different coolant fluids and analysis of different parameter's effects on heat transfer coefficient and pressure drop. Instead, analysis of the complete heat sink geometry, one of the parts, is to be taken to the simulation and analysis to prevent more compute time and analysis cost. It is necessary to give proper boundary conditions because they become important for the correct result and behaviour of the complete process. In this work, a heat sink has several microchannels from which a single channel is chosen and no. of trials are carried out on it. for the validation of basic model, the simulation results are compared with the result from Yadav et al. [2021]. The material used for the heat sink channel is copper, because copper has high thermal conductivity and a high heat transfer coefficient [1]. In this work, mainly three coolants are used, i.e., water, air, and ethylene glycol. Also, while simultaneously sweeping other parameters like shape of obstacles, Velocity, aspect ratio of obstacles, and number of obstacles, the schematic of the microchannel model employed in Figure 1. user-defined meshing applied to both solid and liquid parts of the model as show in the Figure 2. Mesh plays an important role in heat transfer, but as the quality of mesh increases, the time to compute increases. To maintain a balance between quality and time, normal mesh is suitable for this model. At the bottom surface of the channel, a uniform heat flux is applied. Heat is transferred into fluid through the base and vertical walls.

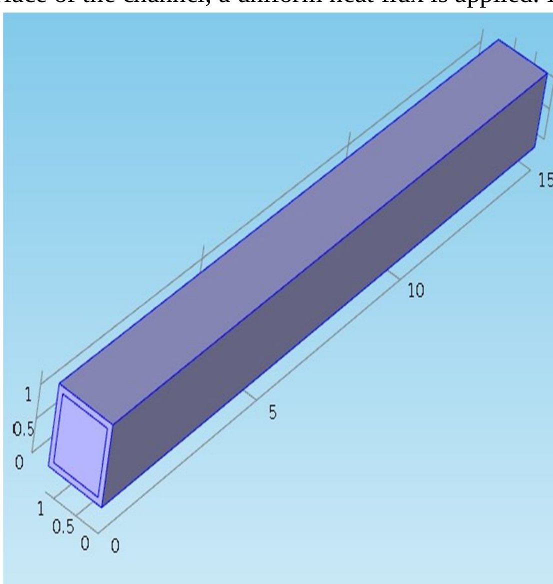


Figure 1. The base model used in the analysis

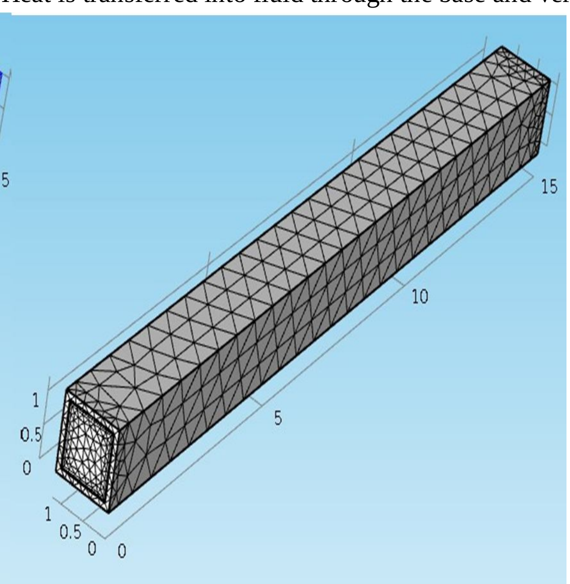


Figure 2. Meshed model analysis

Some assumptions are made for simplification. The fluid is incompressible and steady. The fluid flow is laminar with constant velocity at the entrance of the channel. There is no heat transfer by radiation mode and all the physics of fluid and solid are temperature and time independent.

The fluid with laminar flow and constant velocity enters the microchannel from its front end. The temperature of the fluid at the inlet is 293.15K. The velocity of fluid at the inlet is allocated to be 0.2 m/s to 1m/s. At the outlet boundary of the microchannel, atmospheric pressure is allocated. All the necessary equations are solved by software after applying all the boundary conditions mentioned in Table 1. After solving all the equations, the software calculates temperature, average velocity, absolute pressure, and average heat transfer coefficient at different points.

Table1. Boundary conditions employed for the model.

Boundary	Fluid Boundary condition	Thermal Boundary Conditions
Front fluid inlet	Inlet	Adiabatic
Front fluid outlet	Outlet	Adiabatic
Channel left surface	Symmetry	Symmetry
Channel right surface	Symmetry	Symmetry
Channel bottom surface	Wall	Uniform heat flux
Channel top surface	Wall	Adiabatic
Channel front solid	Wall	Adiabatic
Channel back solid	Wall	Adiabatic

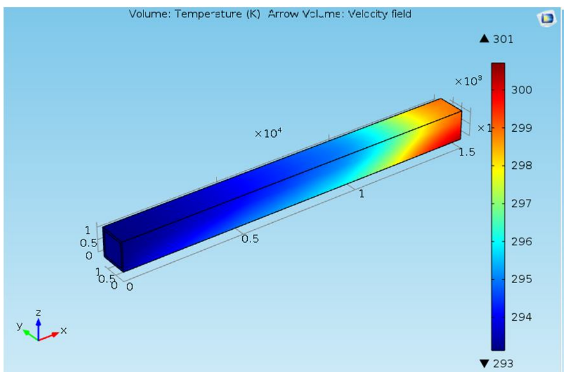


Figure 3. Base Model For validate Temperature

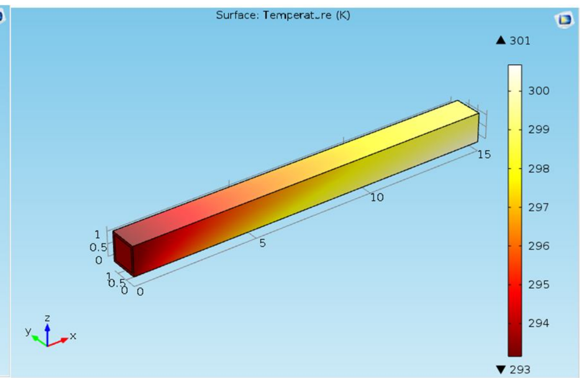


Figure 4. Validated model for temperature

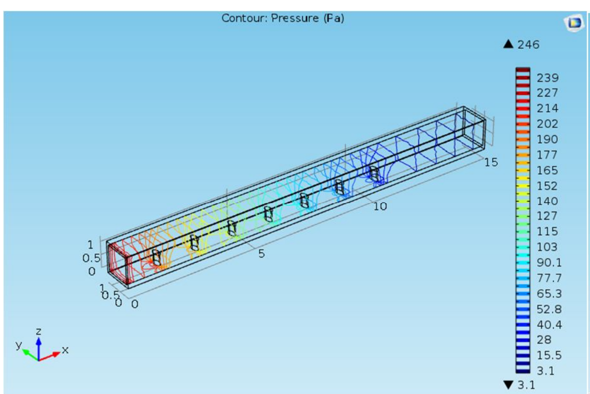


Figure 5. Base Model For validate Pressure

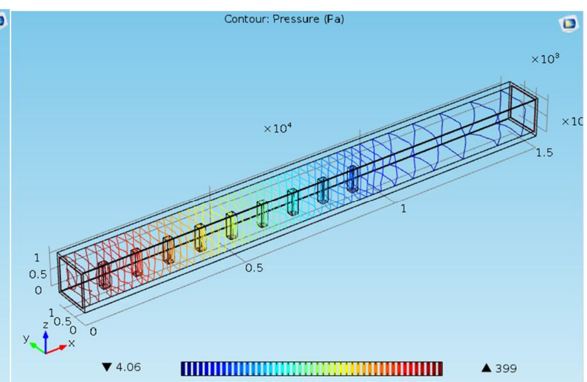


Figure 6. Validated model for Pressure

IV. COMPUTATIONAL ANALYSIS

The parameters mentioned in Table 2. Are used for analysis, these parameters are constant in overall analysis process. But some parameters like thermal conductivity of fluid, density of fluid are changes with respect to change in fluid used. There are no any extra efforts require because software provide all those parameters related to fluid properties. Just only we have to choose “From material” option where is it available.

Table2. Set of parameters used for analysis.

Parameter	Baseline value
Channel wall thickness	0.1mm
Channel length	15mm
Channel fluid flow area	1mm × 1mm
Heat flux	20w/m^2 K

The following governing equations are used for describing the heat transfer and fluid flow

$\rho C_p u \cdot \nabla T = \nabla \cdot (k \nabla T) + Q$

……. Heat transfer in solid

$\rho C_p u \cdot \nabla T = \nabla \cdot (k \nabla T) + Q + Q_{vd} + Q_p$

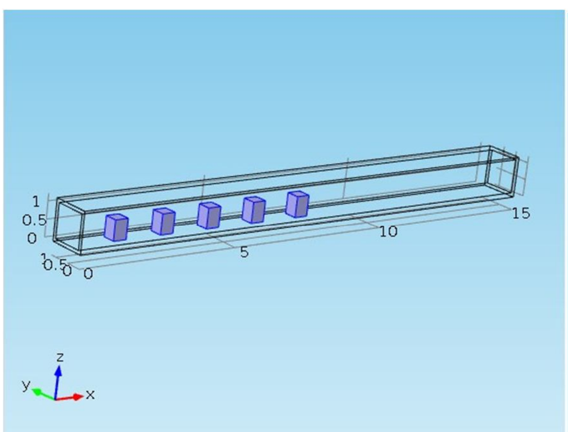
……. Heat transfer in fluid

$\nabla U = 0$ ……. Continuity equation

All the above equations are studied at stationary mode there is no any one of the quantities is varied with respect to time. In the Table 3. Describe the variation in parameter and its range.

Table 3. Parameter sweeping

Parameter	Type/ Range
Shape of Obstacles	Square, Circle, Elliptical
No. of obstacles	3, 5, 7
Velocity of fluid at inlet	0.2m/s, 0.6m/s, 1.0m/s
Aspect ratio of obstacles	0.5, 1.0, 1.5
Type of Fluid	Water, Ethylene glycol, Air



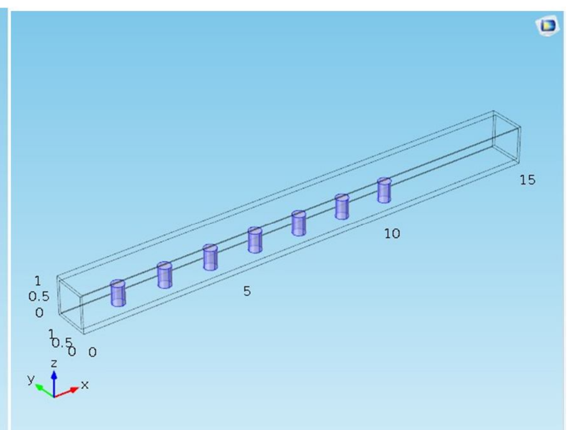


Figure 7. Model with five square shape obstacles Figure 8. Model with seven circular shape obstacles

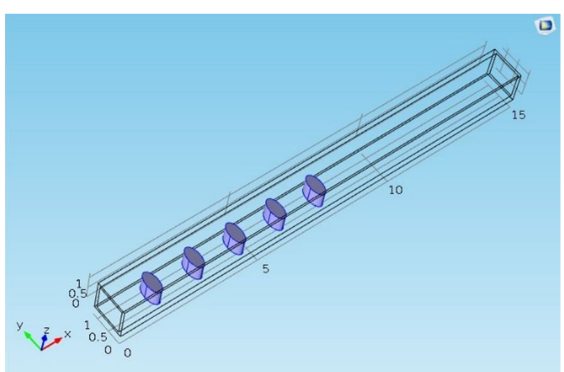


Figure 9. Model with five elliptical shape obstacles

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 10 Issue IV Apr 2022- Available at www.ijraset.com

Table 4. Dimensions of obstacles

Shape of obstacles	Dimension of Obstacles
Square	0.4mm×0.4mm
Circular	0.4mm radius
Elliptical	0.2mm minor and 0.4 major diameter

V. CALCULATION

For the calculation of Heat transfer coefficient and pressure drop some required data necessary data is taken from simulated model. The values of temperature and pressure is necessary to further calculation. Here heat is dissipated in mode of forced convection so, Newton's law of cooling is used for calculate heat transfer coefficient.

$$Q = h \times A \times (T1- T2) \quad \text{………. Equation of newtons law of cooling}$$

Q = Rate of heat flux. (W/m^2)
h = Heat transfer coefficient. (W/m^2.K)
A = Heat transfer surface area. (m^2)
T1 = Temperature of the object's surface. (K)
T2 = Temperature of the fluid at inlet. (K)
Pressure drop is simply calculated by subtracting pressure at outlet from pressure at inlet.

A. Taguchi

Taguchi is a design of experiment method, which gives a best result in less no. of trials, for that MINITAB software is used. With the use of all data as mention in table 3, a basic structure is created as show in table 5. This structure helps for making models on COMSOL. According to that total 27 models are created with different combination and derived results from these models. By using MINITAB software, different graphs and tables where created, with the help of these graphs, it is easy to study effect of different parameters on heat transfer coefficient and pressure drop.

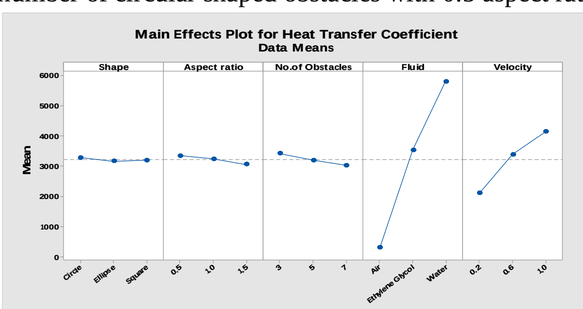
Table 5. Value of Heat transfer coefficient and pressure drops after analysis

Sr no.	Shape	No. of Obstacles	Aspect ratio	Fluid	Velocity (m/s)	Heat transfer Coefficient(W/m^2K)	Pressure Drop (Pa)
1	Square	3	0.5	Water	0.2	4235.16	176.85
2	Square	3	0.5	Water	0.6	6309.3	751.30
3	Square	3	0.5	Water	1	7799.19	1583.71
4	Square	5	1	Air	0.2	305.12	2.8165
5	Square	5	1	Air	0.6	309.92	9.3048
6	Square	5	1	Air	1	315.04	17.020
7	Square	7	1.5	Ethylene Glycol	0.2	2033.65	5299.05
8	Square	7	1.5	Ethylene Glycol	0.6	3345.53	17653.8
9	Square	7	1.5	Ethylene Glycol	1	4144.15	32643.5
10	Circle	3	1	Ethylene Glycol	0.2	2450.04	2838.9
11	Circle	3	1	Ethylene Glycol	0.6	4031.44	9109.92
12	Circle	3	1	Ethylene Glycol	1	5003.12	16273.9
13	Circle	5	1.5	Water	0.2	3617.07	356.93
14	Circle	5	1.5	Water	0.6	6013.03	1728.02
15	Circle	5	1.5	Water	1	4960.15	3774.81
16	Circle	7	0.5	Air	0.2	314.96	2.2919
17	Circle	7	0.5	Air	0.6	319.98	7.3687

18	Circle	7	0.5	Air	1	325.29	13.1529
19	Ellipse	3	1.5	Air	0.2	308.57	2.6551
20	Ellipse	3	1.5	Air	0.6	317.54	8.7652
21	Ellipse	3	1.5	Air	1	326.78	15.9716
22	Ellipse	5	0.5	Ethylene Glycol	0.2	2311.17	2423.75
23	Ellipse	5	0.5	Ethylene Glycol	0.6	3795.241	7569.17
24	Ellipse	5	0.5	Ethylene Glycol	1	4700.8	13163.8
25	Ellipse	7	1	Water	0.2	3480.92	248.91
26	Ellipse	7	1	Water	0.6	6100.87	623.98
27	Ellipse	7	1	Water	1	7198.99	2786

VI. RESULT AND DISCUSSION

This work is performed to analyze effect of different parameters on the performance of microchannel heat sink. For that the basic model of heat sink remain same and made changes in different parameters to see changes in heat transfer coefficient and pressure drop. In fig 10. There is a plot for heat transfer coefficient with respect to different five parameters. The heat transfer coefficient is high when water is used as coolant fluid. The heat transfer coefficient when water coolants is used it is near about double than when Ethelene glycol is used. The thermal conductivity of air is less therefor heat transfer coefficient is also low. The heat transfer coefficient is increases when velocity get increases, more velocity there is more turbulence so heat dissipated in large amount. The effect of remaining three parameters on heat transfer coefficient is negligible. The heat transfer coefficient is slightly more when minimum number of circular shaped obstacles with 0.5 aspect ratio are used. The surface temperature of body is portrayed in fig 11.



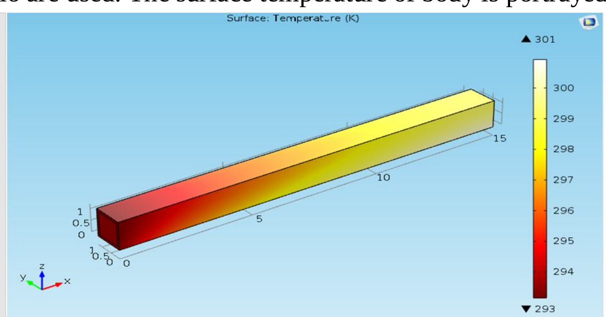
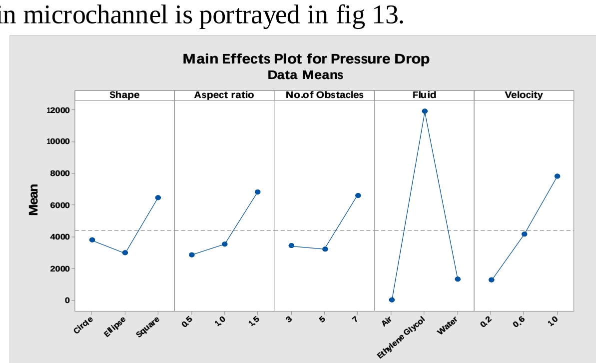


Figure 10. Main effect plot for heat transfer coefficient Figure 11. Surface temperature in microchannel

The plot for pressure drop with respect to different parameters is poytared in fig 12. The pressure drop is directly proportional to inlet velocity of fluid. The value of pressure drop is high when fluid entres in microchannel with high velocity. Pressure drop occure low when air is used as coolant fluid. And pressure drop is high when ethlene glycol is used as coolant fluid.Pressure drop in water as coolant fluid is more than air and less than Ethelene glycol.Here other three parameters play an important role in pressure drop. Elliptical shape obstacles having ability of less pressure drop, and square shape obstacles tend to more pressure drop. Pressure drop is less when aspect ratio is less but as increase in number of obstacles pressure drop firstly decresses then increases. The pressure countour in microchannel is portrayed in fig 13.



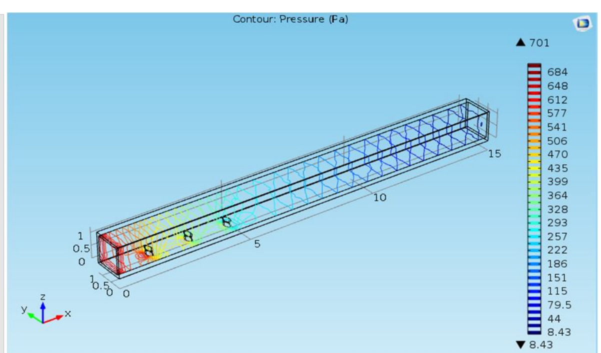


Figure 12. Main effect plot for pressure drop Figure 13. Pressure conture in microchannel

VII. CONCLUSION

Three dimensional square microchannel of copper material is compared to observe the effect of change in different parameters on heat transfer coefficient and pressure drop. Firstly, basic model is validated with using the result of Jadhav et. al. Then made changes in various parameter like coolant fluid, inlet velocity of fluid, shape of obstacles, number of obstacles and aspect ratio of obstacles.

A. It is observed that model with water fluid having range of heat transfer coefficient value is 3480.92W/m^2K to 7799.19W/m^2K. It is more than the values of model with air and ethylene glycol. Heat transfer coefficient and pressure drop is directly proportional to velocity of fluid.

B. The impact of aspect ratio of obstacles, no. of obstacles and shape of obstacles on result of heat transfer coefficient is no more. But pressure drop is affected in large amount by parameters aspect ratio, no. of obstacles and shape of obstacles.

C. It is observed that, if elliptical shape obstacle is used then pressure drop is less than circular shape and square shape obstacles, the mean value of pressure drop for elliptical shape obstacles is 3000Pa. As similar when three obstacles with 0.5 aspect ratio is taken then mean pressure drop is 3100Pa.

D. The maximum heat transfer coefficient i.e., 7799.19 W/m^2K is obtained when three square shaped obstacles of aspect ratio 0.5 and water fluid at velocity 1m/s is used.

E. After the analysis of results graphs and figure is observed that water coolant with moderate velocity, circular obstacles with 0.5 aspect ratio is the best combination to get more heat transfer coefficient at low pressure drop.

REFERENCES

[1] S V Jadhav1, P M Pawar2, S S Wangikar3, N N Bhostekar4 and S T Pawar, “Thermal Management Materials for Advanced Heat Sinks used in Modern Microelectronics.” Published by IOP Conference Series: Materials Science and Engineering, S V Jadhav et al 2020 IOP Conf. Ser.: Mater. Sci. Eng. 814 012044

[2] P M Wadekar1, A B Shinde2, V B Patil3, P D. Kulkarni4, P V Kengar5 and S.V. Jadhav, “Computational Investigation of Micro-channel Heat Sink with Rectangular Shape Obstacles.” Published by Journal of Physics: Conference Series, P M Wadekar et al 2021 J. Phys.: Conf. Ser. 2070 012181

[3] S. V. Jadhav, P. M. Pawar, “NUMERICAL ANALYSIS OF A HEAT SINK WITH DIMPLES.” Published by INTERNATIONAL RESEARCH JOURNAL OF MULTIDISCIPLINARY STUDIES & SPPP's, Karmayogi Engineering College, Pandharpur Organize National Conference Special Issue March 2016, Vol. 2, Special Issue 1, March, 2016 ISSN (Online): 2454-8499 Impact Factor: 1.3599(GIF), 0.679(IIFS).

[4] Somesh Deshmukh, Chandrashekhar Nagendra (March 2021) “Thermal Analysis and design optimization of heat sink using Taguchi method”. JETIR March 2021, Volume 8, Issue.

[5] Dong Liu, Suresh v. Garimella. “Analysis and optimization of the thermal performance of microchannel heat sinks.” (2005). CTRC research publication. Paper 59

[6] Sayali R. Bhalerao. “Optimization of Pin Fin Heat Sink Using Taguchi Method”. Journal of Thermal and fluid science, August 2020, Volume 1, Issue 1.

[7] Mehdi Nafar Mohammad Tavassoli. “An Analysis for Optimization of Heat Transfer for Various Heat Sink Cross-section and Length”. Australian Journal of Basic and Applied Science,2011, Volume5, Issue 12.

[8] S. Manivannan, S. Prasanna Devi, R. Arumugam and N. M. Sudharshan. (2010) “Multiobjective optimization of flat plate heat sink using Taguchi-based Grey relational analysis”. Springer – Verlag London Limited 2010.

[9] Archibald Allswell Amoako. “Optimization of Heat Sinks in a Range of Configurations”. Open PRAIRIE,2018. Electronic Theses and Dissertations.2425.

[10] Hussein T. Dhaiban, Maha A. Hussein. “The Optimal Design of Heat Sinks: A Review”. J. Appl. Comput. Mech, 2020.

[11] Pawar Shreekant Prabhakar, Prof. Ghuge N. C. “Thermal Design and Analysis of Heat Sink Optimization and its Comparison with Commercially Available Heat Sink”. International Journal of Engineering Research and Technology, December 2015, Volume 4, Issue

[12] Suresh Babu, Manjunatha A, Dr. M Sihivashankar and Vijay Kumar Mishra. (2018) “multiobjective optimization of a led heatsink using evolutionary algorithm with the help of CFD simulation”. SSRG International Journal of electronics and communication Engineering 2018