International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

Thermal management of electric vehicle batteries: a parametric study and optimization of a micro channel heat sink

J. Praveen Kumar[1], J Praveen Kumar [2]
[1&2]Assistant Professor,Department of Electrical & Electronics Engineering, G. Pullaiah College of Engineering and Technology, Pasupula Village, Nandikotkur Rd, near Venkayapalle, Kurnool, Andhra Pradesh 518002.

ABSTRACT
The use of fin shapes and their arrangement inside micro channels has a significant impact on improving micro channel thermal performance. To improve thermal performance, this research employs microchannel heat sinks with three distinct fin configurations (circular, square, and rhombus). ANSYS Designmodeler created the microchannel heat sink. ANSYS Fluent does CFD simulations, whereas Design Expert performs parametric DOE. Twelve CFD simulations using RSM (Response Surface Methodology) are used to investigate the effect of factors (coolant velocity and fin shape) on the thermal performance of microchannel heat sinks. Temperature and pressure differences are evaluative measures. The rhombus-shaped fin outperformed other forms in terms of temperature difference and pressure drop.

Keywords: ***RSM, DOE, Microchannel, Heat sink, Fin geometry.***

I. INTRODUCTION

Due to increasing global pollution and diminishing fossil fuel supplies, the car industry is rapidly switching from internal combustion engines to electric engines. ADVANCED BATTERY TEMPERATURE CON Existing heat management systems include thermoelectric, forced air, and liquid [1]. Among the technologies, liquid cooling has the most potential. The liquid coolant is in indirect contact with the battery and removes heat produced during operation. The main difficulty with liquid cooling is designing a practical, small, and effective heat exchanger. A few researchers have studied the cross-sectional geometry of MCHSs (circular, rectangular, triangular, and trapezoidal) [1- 4]. Several writers [4-9] advocated increasing surface area and heat transfer coefficient by adding strips and flips to the channel. Some writers have focused on flow entry features in serpentine designs for microfluidic cooling, such as wavy [4], crisscross [5], and divergent-convergent [6].

Xia et al. [7] investigated the optimization of microchannel geometric parameters using offset fan-shaped reentrant grooves, triangular reentrant grooves, and modified fan-shaped reentrant grooves. In their review, Steinke and Kandlikar [8] summed up the different works made by scientists across the globe on single- phase heat transfer in the miniature or microchannel. They explored the influence of various parameters, like, thermal resistance, Nusselt number, Reynolds number, friction factor, with regards to fluid flow using experimental studies.

Transverse Micro Channels (TMC) for enhancing heat transfer rate in a heat sink were studied numerically by Esmail Ghasemisahebi et al. He employed 3D conjugate heat transfer for this. Modeling heat transport in both fluid and solids. He studied the impact of characteristics such transverse microchannel density, height, and Reynolds numbers on the decline inside the heat sink. The final findings showed that the temperature distribution and hotspot location were solely dependent on the number and size of transverse microchannels. Muhammad Noor Afiq Witri Muhammad et al (MCHS). Active and passive cooling methods were used to cool the microprocessor chip. A dynamic cooling system consumed energy, but passive cooling required no electricity. He evaluated the cooling systems to see which was more dependable. Finally, he discovered that passive cooling was not only cheaper but also more robust and dependable than active cooling with moving components. MCHS was one of the high-tech devices that used passive cooling to cool electronics. In addition, he examined the impacts of channel type, surface roughness, fluid additives, and Reynolds number in a microchannel heat sink.

Tingzhen Ming et al. [11] employed dimple features to increase the heat transfer rate of a microchannel heat sink using impinging jets (MIJ). His work investigated MIJs with and without dimples (mixed dimples, convex, concave) using numerical simulation. Among all tested cases, using MIJs with convex dimples outperformed MIJs without dimples, mixed dimples,

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

and concave dimples in terms of cooling performance, flow resistance, and overall performance. The author found the overall performance ratio between heat transfer and resistance. During the investigation, the convex dimple structure exhibited the maximum heat transmission performance at low pressure.

Sainan Lu et al. [12] compared two- and multi-layered microchannel heatsinks (MCHS) or thin films inside flexible complex seals. Here is a comparison of multi-layered MCHS outcomes with double-layered MCHS. Less thermal resistance and less pumping power are shown in DL-MCHS and ML-MCHS, which addressed SL-streamwise MCHS's temperature increase difficulties.

Increasing the thermal performance of microchannels is critical for improved battery system performance. Microchannel efficiency may be improved by optimising their shape or by improving heat transmission. This effort attempts to improve the thermal performance of microchannel heat sinks. This research uses 12 RSM CFD simulations to assess the impact of factors (coolant velocity and fin shape) on microchannel thermal performance.

II. DESIGN OF EXPERIMENTS

RSM creates the DOE utilising randomised optimum design. We examine one numerical (fluid velocity) and one categorical (fin form). The numerical factor uses discrete data, whereas the categorial factor uses nominal data (Level 1: Circle, Level 2: Square, Level 3: Rhombus). To analyse and improve performance factors such as temperature change and pressure drop, RSM is used.

The DOE of the study is represented in Table 1.

Table 1

Run	Factor 1 A: Fluid Velocity (m/s)	Factor 2 B: Fin Shape
1	0.3	Level 2 of B
2	1	Level 1 of B
3	0.3	Level 2 of B
4	0.3	Level 1 of B
5	0.3	Level 1 of B
6	0.5	Level 3 of B
7	0.1	Level 3 of B
8	1	Level 2 of B
9	0.5	Level 3 of B
10	1	Level 3 of B
11	1	Level 2 of B
12	0.5	Level 3 of B

3. MICROCHANNEL OPTIMIZATION

The microchannels are tuned for two responses, temperature change and pressure drop, using CFD analysis on DOE models in ANSYS Fluent. Responses ANOVA:

Response I: The first response is the temperature change of the fluid in the microchannel. Table 2 displays the ANOVA table for the first answer.

Table 2

Source	Sum	df	Mean Square	F-value	p-value
Model	166.26	3	55.42	27.87	0.0001 significant
A-Velocity	131.01	1	131.01	65.89	< 0.0001
B-circle - rhombus -square	36.58	2	18.29	9.20	0.0084
Residual	15.91	8	1.99		
Lack of Fit	15.91	3	5.30		
Pure Error	0.0000	5	0.0000		
Cor Total	182.17	11			

The model F-value of 27.87 indicates it is significant, and this might be related to noise. P-values 0.0001 suggest acceptable model terms. Table 3 shows the fit statistics.

Table 3

Std. Dev.	1.41	**R^2**	0.9127
Mean	11.33	**Adjusted R^2**	0.8799
C.V. %	12.44	**Predicted R^2**	0.7518
		Adeq Precision	16.5879

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
www.ijarst.in
IJARST
ISSN: 2457-0362

The Predicted R^2 of 0.7518 is in reasonable agreement with the Adjusted R^2 of 0.8799; i.e., the difference is less than 0.2. Adequate Precision of 16.588, which measures the signal-to-noise ratio, is desirable as a ratio of 16.588 indicates an adequate signal since it was greater than 4. The predicted vs. actual value of effectiveness is shown in Fig 1.

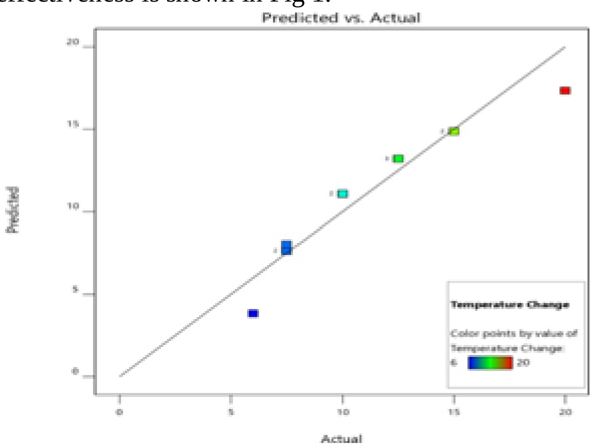

Fig. 1: Predicted vs. Actual for R-I

For the given levels of each factor, the regression equation used to make predictions about the 1st response is as follows: Temperature change for Level 1 of B = (+14.1829 + -10.343 * Velocity)
Temperature change for Level 2 of B = (17.9729 + -10.343 * Velocity) Temperature change for Level 3 of B = (18.3783 + -10.343 * Velocity.

A. Response II:

The pressure -drop in the microchannel is considered as the 2nd response. The ANOVA table for the 2ndt response isshown in Table 4.

Table 4

Source	Sum of Squares	df	Mean Square	F-value	p-value
Model	4.797E+08	1	4.797E+08	8.63	0.0148 significant
A-Velocity	4.797E+08	1	4.797E+08	8.63	0.0148
Residual	5.560E+08	10	5.560E+07		
Lack of Fit	5.560E+08	5	1.112E+08		
Pure Error	0.0000	5	0.0000		
Cor Total	1.036E+09	11			

The Model F-value of 8.63 implies the model is significant, and by 1.48% % chance, this could occur due to noise. P-values < 0.0500 indicate that the model terms are essential. The fit statistics are shown in Table 5

Table 5

Std. Dev.	7456.26	**R^2**	0.4632
Mean	8239.74	**Adjusted R^2**	0.4095
C.V. %	90.49	**Predicted R^2**	0.1039
		Adeq Precision	5.7540

The Predicted R2 of 0.1039 matches the Adjusted R2 of 0.4095. A signal-to-noise ratio of 5.754 is good, since a ratio of 5.754 shows an acceptable signal.The regression equation used to predict the 2nd response for given values of each element is as follows: -2788.42 + 19461.5 * Velocity.

B. Optimization:

The optimization is done based on the following constraints represented in Table 6. 15 solutions were found foroptimization. In the 3-solutions, the solution having the highest desirability is selected as an optimized solution.

Table 6

Name	Goal	Lower Limit	Upper Limit
Velocity	minimize	0.1	1
circle rhombus- square	is in range	Level 1 of B	Level 3 of B
Temperature Change	maximize	6	20
Pressure Drop	minimize	506.432	35785.8

From the RSM optimization, the selected parameters are:

1. Velocity: 0.1 m/s
2. Fin configuration: Rhombus
3. Temperature change: 17.344 k
4. Pressure-drop: 842.77 Pa.

III. COMPUTATIONAL MODELLING

A. Geometrical modeling:

The temperature change of the fluid in the

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

microchannel is considered the 1st response. The ANOVA table for the 1stresponse is shown in Table 2.

The microchannel heat sink is made up of copper, and water is taken as the working fluid. The geometric domain of the heat sink is shown in Fig. 3. Three different fin shapes are considered for this study: square, rhombus, and cylindrical. Five fins are present inside the microchannel, as shown in Fig. 3. A stabilization region is provided to get the fully developed flow. The different geometric dimensions used are detailed in Table 7.

Table 6

Parameter	Symbol	Value, mm
Channel length	L	55
Base width	W	1.25
Channel height	H	1.25
Stabilization length	L_s	12.5
Fins span	L_f	24
Fin diameter/length	D	0.5
Fin height	h	1
Distance between fins	d	6
Channel thickness	t	0.25

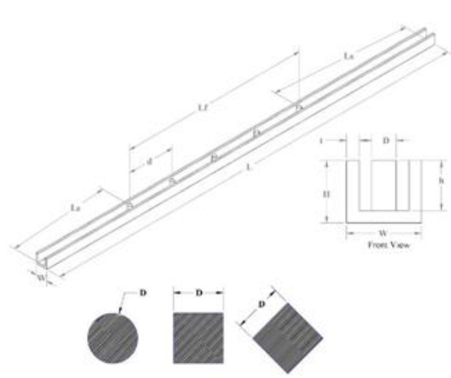

Fig. 2: Microchannel heat sink geometry

B. Mathematical modeling:

Following are the governing equations used are as follows:

Continuity equation:

Momentum equation:

$$\rho_l(\overrightarrow{u} \cdot \nabla \overrightarrow{u}) = -\nabla p + \mu_l \nabla^2 \overrightarrow{u}$$

The energy equation for solid:

$$\rho_l C_{pl}(\overrightarrow{u} \cdot \nabla T_l) = k_l \nabla^2 T_l$$

The energy equation for liquid:

$$\nabla(k_s \nabla T_s) = 0$$

C. Numerical Simulations:

The flow direction was represented by a two-dimensional model in numerical simulations. Fig. 3 shows the 2D model's geometry and boundary conditions. The intake and outflow accept water. The intake parameter is velocity, whereas the outflow is kept constant at 0 Pa. The fins and side walls are maintained at 4000 W/m2. A Reynolds number of 100 is used to simulate flow at 0.1, 0.3, 0.5, 0.75, and 1 m/s. The domain was meshed with triangular pieces, as seen in Fig. 5. For each mesh size, the grid independency test is run. The best grid size for the remainder of the runs is 48050.

The numerical simulations were run using Ansys Fluent 2021b on an Intel CoreTM i5 – 8400 CPU with 32704 MB RAM. The governing equations were two constant k turbulence models with energy equations. These were discretized using a second-order upwind method. Flow and energy coupling was used in pseudo transient analysis. Temperature, pressure, and velocity were computed for each simulation at various Reynolds numbers. The findings are listed below.

RESULTS

CFD investigation of microchannels with various fin shapes was performed at 100, 300, 500, 750, and 1000 Reynolds numbers. Each case's temperature and pressure differential were determined. Assume a temperature change and a pressure reduction of

$\Delta T = T_{w,o} - T_{w,i}$ (1)

$\Delta P = P_o - P_i$ (2)

CFD investigation of microchannels with various fin shapes was performed at 100, 300, 500, 750, and 1000 Reynolds numbers. Each case's temperature and pressure differential were determined. Assume a temperature change and a pressure reduction oftemperature, respectively. The subscripts w indicates the water, o indicates outlet, and I mean inlet.

As a representation, the pressure and velocity contours for different fin shapes for some cases at Re = 100 are shown in Fig. 6 and Fig. 7, respectively. The contours were length-wise clipped from 15 mm to 30 mm for better visibility of the zones.

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

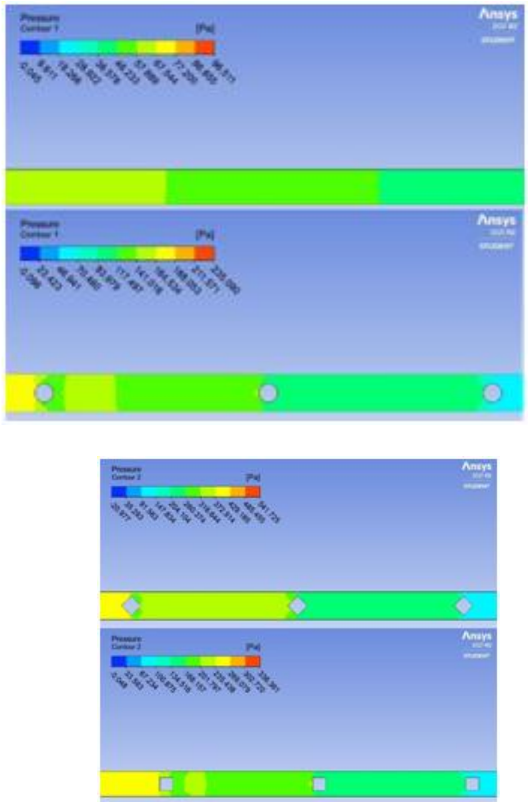

Fig4: Pressure contour at Re=100. (a) No fin, (b) circular, (c) rhombus, (d) square.

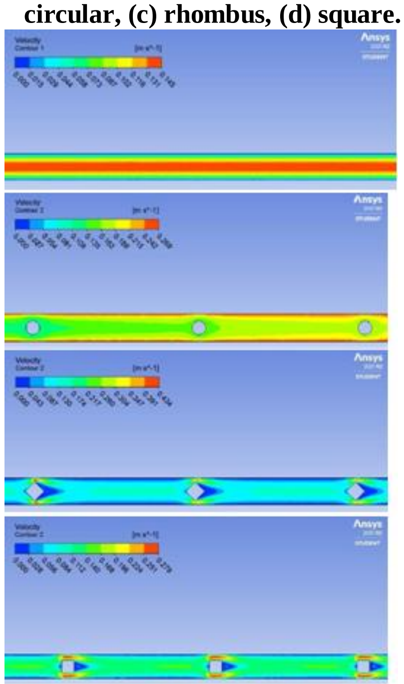

Fig. 5: Velocity contour at Re=100. (a) No fin, (b) circular, (c) rhombus, (d) square.

From Fig. 6, it can be observed that different pressure zones were created around the fins. Also, it can be observed that zones of turbulence were made at the back of fins concerning flow direction, shown in Fig. 7. Pressure drop and temperature differences were plotted at different Reynolds numbers for various fin shapes and are shown in Fig. 6 and Fig. 7, respectively.

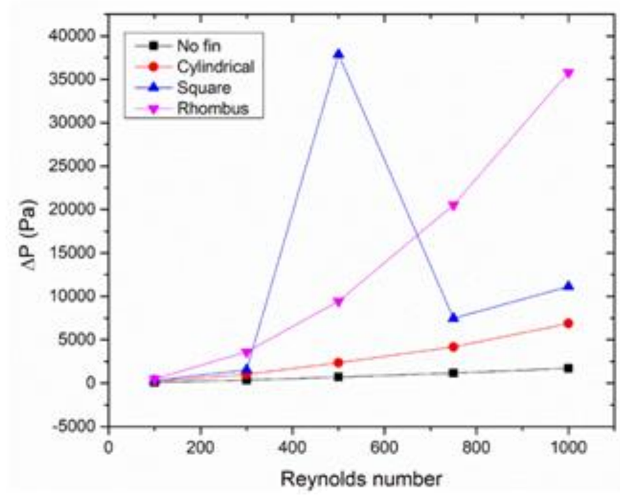

Fig. 6: Pressure drop vs Reynolds number

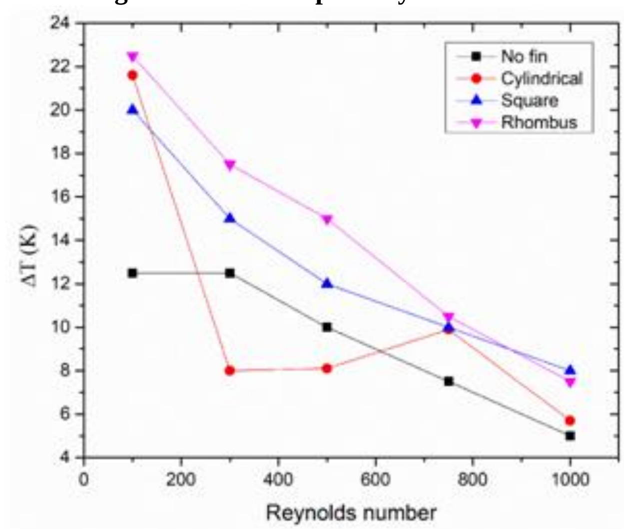

Fig. 7: Temperature difference vs. Reynolds number

As the flow rate increases, the pressure drop increases for all the cases, shown in Fig. 8. It can be observed that the highest-pressure drop is obtained for square fin at Re = 500. From Fig. 9, it can be observed that the highest temperature difference is observed for rhombus fin at Re = 100, while no fin configuration has the least for most of the cases

International Journal For Advanced Research In Science & Technology
A peer reviewed international journal
www.ijarst.in
ISSN: 2457-0362

CONCLUSIONS

Three different fin shapes, square, rhombus, and circular, inside a micro channel were proposed. Numerical simulations using coupled heat transfer and flow analysis were carried out at different flow rates to find the performance of the proposed fin shapes. The temperature difference and pressure drop are used to evaluate the performance of the micro channel for different fin shapes. Response surface methodology (RSM) is carried out to find the optimal working conditions.

REFERENCES

[1] J.P. McHale, S.V. Garimella, Heat transfer in trapezoidal microchannels of various aspect ratios, Int. J. Heat Mass Tran. (2010), https://doi.org/10.1016/j.ijheatmasstransfer.2009.09.020.

[2] Y. Chen, C. Zhang, M. Shi, J. Wu, Three-dimensional numerical simulation of heat and fluid flow in noncircular microchannel heat

[3] H.A. Mohammed, P. Gunnasegaran, N.H. Shuaib, The impact of various nanofluid types on triangular microchannels heat sink cooling performance, Int. Commun. Heat Mass Tran. (2011), https://doi.org/10.1016/j.icheatmasstransfer.2011.03.024.

[4] P. Gunnasegaran, H.A. Mohammed, N.H. Shuaib, R. Saidur, The effect of geometrical parameters on heat transfer characteristics of microchannels heat sink with different shapes, Int. Commun. Heat Mass Tran. (2010), https://doi.org/10.1016/j.icheatmasstransfer.2010.06.014.

[5] Y. Sui, C.J. Teo, P.S. Lee, Y.T. Chew, C. Shu, Fluid flow and heat transfer in wavy microchannels, Int. J. Heat Mass Tran. (2010), https://doi.org/10.1016/j.ijheatmasstransfer.2010.02.022.

[6] H.A. Mohammed, P. Gunnasegaran, N.H. Shuaib, Influence of channel shape on the thermal and hydraulic performance of microchannel heat sink, Int. Commun. Heat Mass Tran. (2011),https://doi.org/10.1016/j.icheatmasstransfer.2010.12.031.

[7] G. Xia, L. Chai, M. Zhou, H. Wang, Effects of structural parameters on fluid flow and heat transfer in a microchannel with aligned fan-shaped reentrant cavities, Int.

[8] M.E. Steinke, S.G. Kandlikar, Single-phase liquid friction factors in microchannels, ASME 3rd International Conference on Microchannels and Minichannels, vol. 1, American Society of Mechanical Engineers, 2005, pp. 291– 302.

[9] Soheil Soleimanikutanaei, Esmaiil Ghasemisahebi, Cheng-Xian Lin, “Numerical study of heat transfer enhancement using transverse microchannels in a heat sink” International Journal of Thermal Sciences 125 (2018) 89–100.

[10] Nor Azwadi Che Sidik, Muhammad Noor Afiq Witri Muhamad, Wan Mohd Arif Aziz Japar, Zainudin A. Rasid, “An overview of passive techniques for heat transfer augmentation in microchannel heat sink” International Communications in Heat and Mass Transfer 88 (2017) 74–83.

[11] Xiaoming Huang, Wei Yang, Tingzhen Ming, Wenqing Shen, Xiangfei Yu, “Heat transfer enhancement on a microchannel heat sink with impinging jets and dimples” International Journal of Heat and Mass Transfer 112 (2017) 113–124.

[12] Sajjad Baraty Beni, Alireza Bahrami, Mohammad Reza Salimpour, “Design of novel geometries for microchannel heat sinks used for cooling diode lasers” International Journal of Heat and Mass Transfer 112 (2017) 689–698.

Int. Commun. Heat Mass Tran. (2009

[13] Sainan Lu, Kambiz Vafai, "A comparative analysis of innovative microchannel heat sinks for electronics cooling International Communications in Heat and Mass Transfer 76 (2016) 271–284.

J. Therm. Sci. (2011),https://doi.org/10.1016/j.ijth