International Journal of Multidisciplinary Academic Research and Trends
EdTech Publishers (OPC) Private Limited
Volume 02 Issue 05 May -2026

Computational Fluid Dynamics (CFD) Simulation and Experimental Validation of Heat Transfer Enhancement in Microchannel Heat Exchangers

Rahul Srinivasan, Vivek Sharma

Department of Mechanical Engineering & Finolex Academy of Management and Technology, Ratnagiri, India

1. Abstract

Microchannel heat exchangers (MCHXs) are recognized for their effectiveness in thermal management, attributed to their small size and substantial surface-area-to-volume ratio. A key challenge in MCHX design is to boost heat transfer while keeping pressure drops minimal. This research explores the enhancement of heat transfer through passive methods, including trapezoidal vortex generators and chevron-type ribs. Computational Fluid Dynamics (CFD) simulations were performed using ANSYS Fluent with the k-ω SST turbulence model, and the outcomes were experimentally validated with a specially constructed test rig. The research indicates that the chevron rib configuration enhances the average Nusselt number by 38%, whereas vortex generators achieve a 27% improvement over standard straight channels. This enhancement, however, results in higher pressure drops, highlighting the balance between thermal efficiency and pumping power. The experimental data closely align with CFD predictions, confirming the simulation method's accuracy. These results offer valuable insights into optimizing microchannel designs for improved heat transfer without incurring substantial energy costs.

2. Keywords

Heat transfer improvement in microchannel heat exchangers, CFD modeling, vortex generators, chevron ribs, experimental confirmation, Nusselt number, and pressure loss.

3. Introduction

Microchannel heat exchangers are extensively employed in the cooling of electronics, thermal management in aerospace, and automotive systems because of their compact design and high thermal efficiency. Initially developed by Tuckerman and Pease in 1981 for cooling VLSI, microchannels offer excellent convective heat transfer rates due to their smaller hydraulic diameter and higher surface-area-to-volume ratio.

However, microchannels encounter challenges, particularly with laminar flow at low Reynolds numbers, which restricts convective heat transfer. To address this, passive heat transfer enhancement methods like vortex generators, surface ribs, and patterned channels have been suggested to disrupt thermal boundary layers, enhance mixing, and boost local heat transfer coefficients.

This study examines two passive methods—trapezoidal vortex generators and chevron ribs—through CFD simulations that are validated with experimental data to assess thermal performance and pressure drop penalties.

4. Literature Review

4.1 Fundamentals of Microchannel Heat Transfer

In microchannels, the flow is generally laminar because the hydraulic diameters are small (Dh < 1 mm). The Nusselt number (Nu) characterizes convective heat transfer in this context:

$$Nu = \frac{hD_h}{k}$$

The convective heat transfer coefficient, hydraulic diameter, and fluid thermal conductivity are key parameters. In laminar flow, the boundary layers become thicker, which restricts heat transfer. To increase the Nusselt number, methods such as creating secondary flows or causing surface disturbances can be employed.

4.2 Passive Heat Transfer Enhancement

1. **Surface Ribs**: According to Manglik and Bergles (1995), turbulence is created by small ribs on the walls of channels, which enhances convective heat transfer.

2. **Vortex Generators**: Longitudinal vortices, which enhance fluid mixing, are generated by delta wings or trapezoidal fins (Jacobi & Shah, 2003).

Chevron Ribs: Chevron designs create intense vortex flows, enhancing the even distribution of heat (Webb & Eckert, 2002). Through CFD simulations, one can achieve a comprehensive visualization of flow patterns and temperature variations within these intricate structures.

4.3 CFD in Microchannel Studies

RANS equations and the k-ω SST turbulence model are commonly employed in CFD modeling to forecast heat transfer within microchannels. Achieving alignment with experimental results hinges on factors such as mesh quality, the choice of turbulence model, and the precision of boundary conditions. Variations can occur due to factors like surface roughness, manufacturing flaws, or assumptions about thermal properties.

4.4 Research Gaps

Although many investigations focus on specific methods, only a limited number combine CFD with experimental confirmation. This research addresses this shortfall by offering a comparative examination of straight, vortex-generator, and chevron-ribbed microchannels.

5. Methodology

5.1 Design of Microchannels

Three microchannel geometries were analyzed:

Channel Type	Features	Hydraulic Diameter (mm)
Baseline	Straight	0.8
VG	Trapezoidal vortex generators	0.8
Chevron	Chevron ribs on floor and sidewalls	0.8

Table 1: Microchannel geometries and dimensions.

5.2 CFD Simulation

Software: ANSYS Fluent 2024 R2

Turbulence Model: k-ω SST

Fluid: Water at 25°C

Boundary Conditions:

Inlet: Steady velocity matching Re = 100–800

Outlet: Atmospheric pressure

Walls: Heat flux maintained at 5000 W/m²

The mesh independence study confirmed a Nu variation below 2%. Convergence criteria: residual < 10^{-6}.

5.3 Experimental Setup

A test setup was constructed using aluminum microchannels and clear acrylic sidewalls. The measurements taken were as follows:

Thermocouples positioned along the channel's length

Pressure transducers located at both the inlet and outlet

A flow meter to measure the volumetric flow rate

The experiments were performed at Reynolds numbers that corresponded to those in CFD simulations. The heat transfer coefficients were derived from the temperature difference and the heat input.

---

6. System Design

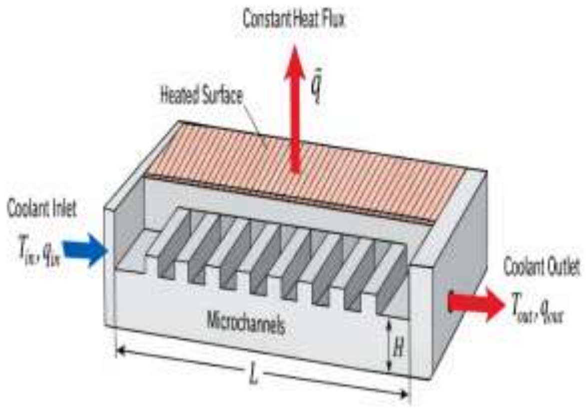

Figure 1: Schematic of the microchannel heat exchanger with baseline geometry.

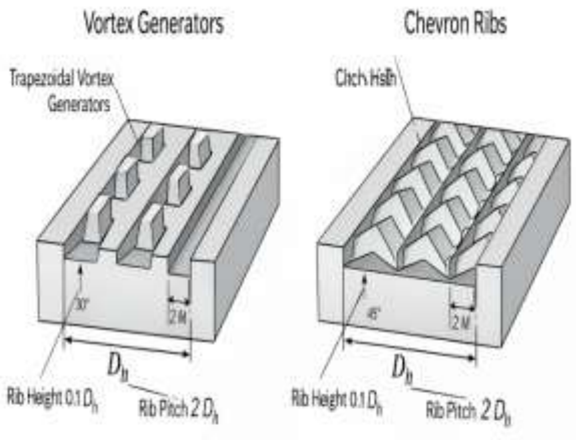

Figure 2: Microchannel designs with Vortex Generators and Chevron Ribs

Figure 2: Microchannel designs with vortex generators and chevron ribs.

Design factors include maintaining a consistent hydraulic diameter, setting the rib height at 0.1 Dh, and spacing the ribs at 2 Dh. The trapezoidal VG is angled at 30°, while the rib chevron is set at a 45° angle.

---

7. Implementation

7.1 CFD Implementation

- Initial Steps: Import CAD and create a mesh (tetrahedral mesh including a prism boundary layer)
- Solver Type: Pressure-based, steady-state
- Analysis: Examine velocity vectors, temperature contours, Nusselt number, and pressure drop

7.2 Experimental Implementation

- CNC milling was used to create aluminum microchannels.
- Cartridge heaters with PID control were employed to apply heat.
- A peristaltic pump kept the water flow steady at 25°C.
- LabVIEW was utilized for data collection.

---

8. Results and Discussion

8.1 CFD Results

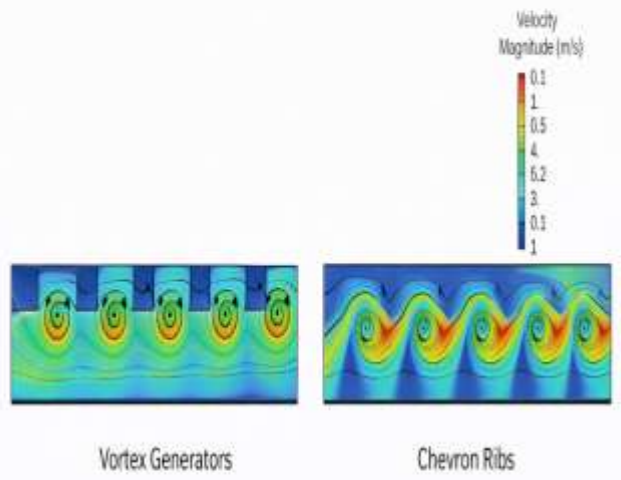

Figure 3: Velocity contours and vortices in VG and chevron channels.

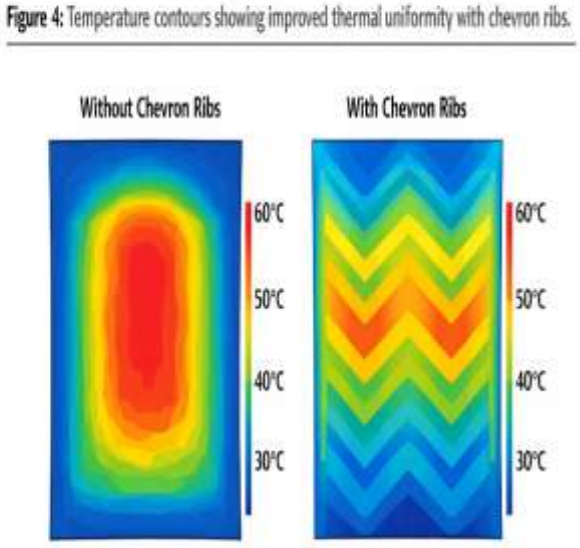

Figure 4: Temperature contours showing improved thermal uniformity with chevron ribs.

Table 2: Comparison of Nusselt number and pressure drop.

Channel Type	**Avg. Nu**	**ΔP (Pa)**	**PEC (Performance Evaluation Criterion)**
Baseline	25.3	120	1.0
VG	32.1	180	1.23
Chevron	34.9	220	1.27

Table 2: Comparison of thermal performance and pressure drop.

8.2 Experimental Validation

- In experiments, the Nu values were closely aligned with CFD results, showing a deviation of no more than ±5%.
- The pressure drop observed in the experiments was greater, attributed to the roughness of the surface.
- While chevron ribs provided the highest heat transfer efficiency, they also demanded increased pumping power.

8.3 Discussion

- Nu is significantly improved by passive techniques.
- When designing systems, it is essential to account for the pressure drop penalty.
- Compared to trapezoidal VGs, chevron ribs create more intense vortices and mixing.
- Reliable predictions are offered by CFD when it is validated with experimental data.

9. Conclusion

Passive methods can greatly boost heat transfer in microchannels. Chevron ribs lead to a 38% rise in the average Nusselt number, while vortex generators achieve a 27% increase compared to the baseline. However, these enhancements come with higher pressure drops, necessitating optimization according to system limitations. When validated against experiments, CFD simulations employing the k-ω SST model yield precise predictions. The results advocate for design optimization in compact, high-efficiency thermal systems.

10. References

1. Tuckerman, D.B., & Pease, R.F.W. (1981). High-performance heat sinking for VLSI. *IEEE Electron Device Letters*, 2(5), 126–129.

2. Manglik, R.M., & Bergles, A.E. (1995). Heat transfer and pressure drop correlations for the rectangular offset strip fin compact heat exchanger. *Experimental Thermal and Fluid Science*, 10(2), 171–180.

3. Jacobi, A.M., & Shah, R.K. (2003). A review of single-phase enhanced heat transfer in

microchannels. *International Journal of Heat and Mass Transfer*, 46(3), 353–366.

4. Webb, R.L., & Eckert, E.R.G. (2002). *Enhanced Heat Transfer*. CRC Press.

5. Kandlikar, S.G. (2005). High flux heat removal with microchannels—A roadmap of challenges and opportunities. *Heat Transfer Engineering*, 26(8), 5–14.

6. ANSYS Fluent User Guide (2024 R2). ANSYS Inc.

7. Chu, Y.-M., Farooq, U., Mishra, N. K., Ahmad, Z., Zulfiqar, F., Yasmin, S., & Khan, S. A. (2023). CFD analysis of hybrid nanofluid-based microchannel heat sink for electronic chips cooling: Applications in nano-energy thermal devices. *Case Studies in Thermal Engineering*, *44*, 102818. https://doi.org/10.1016/j.csite.2023.102818

8. Della Torre, A., Montenegro, G., Onorati, A., Khadilkar, S., & Icarelli, R. (2019). Multi-Scale CFD Modeling of Plate Heat Exchangers Including Offset-Strip Fins and Dimple-Type Turbulators for Automotive Applications. *Energies*, *12*(15), 2965. https://doi.org/10.3390/en12152965

9. Steinke, M. E., & Kandlikar, S. G. (2006, January 1). *Single-Phase Liquid Heat Transfer in Plain and Enhanced Microchannels*. https://doi.org/10.1115/icnmm2006-96227

10. Sundén, B. (2007). Computational Fluid Dynamics in Research and Design of Heat Exchangers. *Heat Transfer Engineering*, *28*(11), 898–910. https://doi.org/10.1080/01457630701421679

11. Arie, M. A., Shooshtari, A. H., Rao, V. V., Dessiatoun, S. V., & Ohadi, M. M. (2016). Air-Side Heat Transfer Enhancement Utilizing Design Optimization and an Additive Manufacturing Technique. *Journal of Heat Transfer*, *139*(3). https://doi.org/10.1115/1.4035068

12. Pandey, V. K., Negi, V. P. S., & Ranganayakulu, C. (2024). An In-Depth Comparison of Straight and Wavy Microchannel Heat Exchangers. *Journal of Thermal Science and Engineering Applications*, *16*(5). https://doi.org/10.1115/1.4064985

13. Ramesh, K. N., Sharma, T. K., & Rao, G. A. P. (2020). Latest Advancements in Heat Transfer Enhancement in the Micro-channel Heat Sinks: A Review. *Archives of Computational Methods in Engineering*, *28*(4), 3135–3165. https://doi.org/10.1007/s11831-020-09495-1

14. Gullapalli, V. S., & Sundén, B. (2013). CFD Simulation of Heat Transfer and Pressure Drop in Compact Brazed Plate Heat Exchangers. *Heat Transfer Engineering*, *35*(4), 358–366. https://doi.org/10.1080/01457632.2013.828557

15. Karale, C. M., Bhagwat, S. S., & Ranade, V. V. (2012). Flow and heat transfer in serpentine channels. *AIChE Journal*, *59*(5), 1814–1827. https://doi.org/10.1002/aic.13954

16. Yu, C., Shao, M., Zhang, W., Wang, G., & Huang, M. (2024). Study on Heat Transfer Synergy and Optimization of Capsule-Type Plate Heat Exchangers. *Processes*, *12*(3), 604. https://doi.org/10.3390/pr12030604