



Numerical Investigation Of Fluid Flow And Thermal Behaviour In A Valvular Conduit Channel

Mr. Y. Vijay bhasker Naidu
Department of Mechanical Engineering ,
ME in Thermal Engineering
CHAITANYA BHARATHI INSTITUTE OF TECHNOLOGY (CBIT),
Hyderabad, Telangana ,India

Dr. R. P. Chowdary
Associate Professor,
Department of Mechanical Engineering,
CHAITANYA BHARATHI INSTITUTE OF TECHNOLOGY
(CBIT), Hyderabad, Telangana ,India

Dr. Ch. Indira Priyadarsini
Assistant Professor
Department of Mechanical Engineering,
CHAITANYA BHARATHI INSTITUTE OF TECHNOLOGY (CBIT),
Hyderabad, Telangana ,India

I. INTRODUCTION

Abstract—This paper presents a detailed computational investigation into the fluid flow and thermal characteristics of a valvular conduit channel incorporating a Tesla valve structure. Utilizing three-dimensional numerical simulations, the study explores the effects of geometric parameters, flow directionality, and Reynolds number variation on the hydrodynamic and thermal performance of the channel. The focus lies on understanding the mechanisms by which Tesla valves achieve flow rectification and enhance convective heat transfer without moving components. The simulations examine both forward and reverse flows, revealing significant asymmetries in velocity distribution, pressure drop, and heat transfer rates. In forward flow, the formation of longitudinal vortices stabilizes the boundary layer, resulting in moderate heat transfer enhancement. Conversely, reverse flow induces transverse vortices, which compress the thermal boundary layer and increase turbulence, thereby boosting heat transfer effectiveness. This directional disparity is quantified through variations in the Nusselt number, friction factor, and a newly defined thermal diodicity index (Dit). Key findings demonstrate that increasing conduction angles and side channel lengths influence the valve's diodicity and thermal efficiency. While larger angles enhance vortex strength and improve reverse-flow heat transfer, they also elevate flow resistance. The study further identifies the trade-off between thermal performance and pressure drop, emphasizing the importance of geometric optimization in practical designs. Grid independence and validation against empirical benchmarks confirm the reliability of the CFD model. The results offer design guidelines for optimizing Tesla valve-based microchannel systems for thermal management applications such as compact heat exchangers, battery cooling, and microfluidic devices. Overall, the study establishes a foundational understanding of the coupled fluid-thermal behavior in valvular conduits and provides a predictive framework for engineering passive flow control and heat transfer devices using Tesla valve architectures.

Keywords: Tesla valve, valvular conduit, heat transfer, CFD, Nusselt number, friction factor, vortex dynamics, reverse flow, numerical modeling.

The ever-growing demand for compact, energy-efficient, and high-performance thermal-fluidic systems across industries such as electronics cooling, aerospace, biomedical engineering, and energy systems has spurred intense research into innovative flow control devices. One such device that has recently regained attention due to advancements in manufacturing and computational modeling is the Tesla valve or valvular conduit. Originally conceived by Nikola Tesla in 1920 [1], this passive, no-moving-part flow rectifier operates based on asymmetric internal geometry that favors flow in one direction while resisting it in the opposite. This inherent asymmetry, achieved without mechanical actuation, makes the Tesla valve a remarkably durable and maintenance-free component for applications where reliability and simplicity are essential. Minichannel and microchannel systems, characterized by their small cross-sectional dimensions (typically less than 1 mm), form the backbone of modern microscale thermal management. These configurations are integral to the design of lab-on-chip systems, heat sinks for electronics, fuel cells, and microreactors. The combination of high surface area-to-volume ratios and short diffusion paths in such systems offers significant potential for enhanced heat transfer and rapid mixing. However, achieving precise flow control within such constrained geometries is non-trivial, especially when dealing with unidirectional flow or selective thermal enhancement. The integration of Tesla valves into such channels has emerged as a promising solution, enabling both flow rectification and thermal asymmetry without added complexity or power consumption. In Tesla valves, flow rectification is achieved not through mechanical obstruction but via the deliberate manipulation of the channel geometry. Diverging and converging pathways, bifurcated junctions, and spiral side channels create conditions that facilitate laminar, low-resistance flow in the forward direction while generating vortices, flow separation, and recirculation zones in reverse flow. These effects result in greater pressure drops and thermal interactions during reverse flow, a feature that can be exploited for passive enhancement of heat transfer. This unique "thermal diodicity"—a measure of the difference in thermal performance between flow directions—has become a critical performance metric in evaluating Tesla valves for real-world thermal systems. The resurgence in research interest surrounding Tesla valves has been facilitated by advances in computational fluid dynamics (CFD), additive manufacturing, and high-resolution meshing techniques. Investigations such as those by Mohammadzadeh et al. [2] and Piyush et al. [3] have provided insight into the diodic behavior of both

single and multi-stage Tesla valves (MSTVs), showing significant improvements in pressure and thermal rectification with increasing valve stages and optimized geometry. For instance, the use of up to ten Tesla stages demonstrated a clear enhancement in reverse flow mixing and heat transfer, especially under moderate Reynolds number conditions ($Re = 25-500$), a range where laminar assumptions still hold validity. In addition to flow control, recent studies have begun to focus on the Tesla valve's capacity for heat transfer modulation. Bao et al. [4] introduced two new performance indices Relative Pressure Drop Ratio and Absolute Pressure Drop Ratio to better assess thermal performance. Their novel Tesla valve design, with tapering and widening features, significantly outperformed traditional models, suggesting that geometric evolution of Tesla valves remains a rich field for exploration. Similarly, Vries et al. [5] demonstrated the valve's efficacy in pulsating heat pipes, where directional flow is critical for phase-change driven thermal systems. Their findings confirmed a 25% difference in forward vs. reverse flow velocities and a 14% enhancement in thermal dissipation due to the valve's asymmetric resistance. Despite the progress, a critical knowledge gap remains in the comprehensive characterization of heat transfer mechanisms in Tesla valves especially in two-stage designs which balance complexity and performance. While many studies focus on diodicity as the primary figure of merit, the interdependent behavior of Nusselt number (Nu), friction factor (f), and a recently introduced metric called thermal diodicity (Dit) remains underexplored. This triad offers a more complete picture of the valve's influence on both fluid mechanics and heat transfer and is essential for deploying these valves in real-world thermal control systems. To address this gap, the present study undertakes a detailed numerical investigation of fluid flow and thermal performance in a two-stage Tesla valve embedded within a conduit of circular cross-section. Using CFD simulations in ANSYS Fluent, this work systematically varies Reynolds numbers, geometric parameters (e.g., conduction angle, side channel length), and flow direction to derive insights into heat transfer asymmetry, pressure drop, and vortex dynamics. Validation studies confirm the simulation's credibility, matching empirical Nusselt values under laminar regimes to within 1%. The inclusion of parameters such as Nusselt number and friction factor allows the formulation of comprehensive performance maps, essential for practical implementation. The Nusselt number serves as an index of convective heat transfer performance, while the Darcy friction factor quantifies energy losses. Their combined behavior across different flow regimes provides a nuanced view of how Tesla valves can be optimized for thermal management. Furthermore, the introduction of the Dit parameter provides a directional comparison, reflecting how asymmetry in heat transfer can be strategically leveraged. From a design standpoint, the study emphasizes the importance of angular and structural features in influencing flow behavior. For example, longer side channel lengths increase flow resistance and reduce turbulence, leading to decreased heat transfer performance. Conversely, sharper conduction angles and narrower junctions enhance vortex formation and mixing in reverse flow, increasing Nu but at the cost of higher pressure drop. These findings are particularly relevant for microfluidic heat exchangers, where spatial constraints necessitate precision in design to maximize heat removal without significant energy loss. Importantly, this research also introduces a practical design framework by generating regression-based models that relate geometric parameters to performance outputs. These models offer predictive capabilities and can serve as design tools for engineers working in battery cooling, MEMS-based devices, or renewable energy systems. Notably, the Tesla valve's passive operation lends itself to environments where traditional mechanical valves would be prone to failure due to fouling, erosion, or power constraints. The Tesla valve stands as a paragon of simplicity and effectiveness in modern thermal-fluid systems. Its asymmetric geometry enables passive flow control, and recent advances show its potential in enhancing directional heat transfer. By integrating detailed numerical simulations, empirical validations, and performance modeling, this study advances the understanding of Tesla valve performance in compact systems. The findings are anticipated to support future designs of robust, maintenance-free, and energy-efficient flow control mechanisms across disciplines ranging from biomedical engineering to renewable energy.

II. RELATED WORK

Tesla valves—originally patented in 1920 by Nikola Tesla [1]—are passive, no-moving-part flow control devices that function by offering high resistance to flow in one direction while permitting relatively free flow in the opposite direction. Their resurgence in modern applications is primarily attributed to advances in computational fluid dynamics (CFD), experimental visualization techniques, and additive manufacturing. The development of micro- and mini-channel systems for thermal management, microfluidics, and energy devices has elevated interest in these geometrically complex yet functionally simple valvular conduits.

A. Historical and Foundational Studies

The foundational numerical analysis by Mohammadzadeh et al. [11] examined one- to four-stage Tesla Micro-Valves (TMVs), identifying an optimal two-stage configuration with maximum diodicity (Di). They demonstrated that increasing stages does not linearly improve performance and noted diminishing returns beyond two stages. The study also emphasized the relationship between Reynolds number (Re) and Di , highlighting Re 's critical role in determining valve behavior. Thompson et al. [12] further investigated the hydrodynamic performance of Multi-Staged Tesla Valves (MSTVs) under laminar flow conditions. Their work revealed that diodicity improves significantly with valve staging and close valve-to-valve spacing, particularly for $Re > 50$. A power-law correlation was developed for performance prediction, providing a valuable design tool for engineers.

B. Thermal Performance Integration

One of the earliest integrations of thermal effects into Tesla valve analysis was conducted by Basil et al. [13], who examined heat transfer and pressure loss in MSTVs using various fluids (air, water, ethylene glycol). Their results highlighted the dual role of these valves in thermal rectification and fluid regulation. Interestingly, water and glycol demonstrated reduced Di at elevated inlet temperatures, while air showed the opposite trend. Porwal et al. [14] expanded on this work by evaluating thermal diodicity (Dit) along with Nu and f using CFD. Their study found reverse flow conditions to favor heat transfer due to enhanced vortex-induced mixing and flow impingement. This observation supports the potential of Tesla valves as directional thermal modulators in micro-scale devices.

C. Advanced Geometric Modifications and Optimization

Hu et al. [15] investigated the effect of varying Tesla valve angles ($45^\circ-90^\circ$), identifying optimal performance at $\theta = 70-80^\circ$ based on separation bubble behavior and flow obstruction dynamics. Proper Orthogonal Decomposition (POD) techniques were used to characterize dominant flow structures and energy modes in the reverse flow path. Bao et al. [16] identified limitations in Di as a performance index and introduced the Relative and Absolute Pressure Drop Ratios for better comparative analysis. Their innovative widening-tapering design significantly improved thermal performance, demonstrating the importance of junction and expansion region design on flow dynamics. Raffel et al. [17] validated numerical predictions using Particle Shadowgraph Velocimetry (PSV), showing that vortex shedding and reverse flow instability increase with stage number. Their findings solidified experimental support for the diodic effect and highlighted the necessity of capturing unsteady flow phenomena. Sebastian et al. [18] optimized Tesla valve geometries for microfluidics using finite element-based topological design. Their valves achieved high Di values (~ 1.8) at extremely low Re (~ 36), proving the valve's applicability in low-flow-rate biomedical and chemical analysis systems.

D. Thermal Applications in Energy and Cooling Systems

Gong et al. [19] demonstrated the effectiveness of MSTVs in improving the net power of Proton Exchange Membrane Fuel Cells (PEMFCs). Their MSTV configurations reduced pressure loss and improved oxygen distribution, achieving up to 19.89% more net power in reverse flow compared to standard flow field arrangements. Fan et al. [20] incorporated MSTVs into a novel Battery Thermal Management System (BTMS), coupling them with phase change materials (PCMs). Their hybrid model resulted in significant temperature uniformity, reducing energy costs by nearly 80%, validating MSTVs for EV battery cooling under varied discharge and ambient conditions. Lu et al. [21] examined the use of Tesla valve-type microchannel plates for lithium-ion battery cooling. Their design yielded enhanced heat dissipation, keeping battery temperatures below 30.5°C and maintaining low pressure drop even under high heat flux conditions. Jun Du et al. [22] proposed an S-type cold plate with embedded MSTVs. Their simulations showed that optimal valve spacing (30 mm) significantly improves temperature uniformity, enhancing reverse flow vortex generation and mixing.

E. High-Performance System Enhancements and Noise Considerations

Quan et al. [23] addressed the challenge of aerodynamic noise in high-pressure hydrogen decompression using MSTVs. Their study identified a direct correlation between valve stage number, Mach number, and turbulent dissipation, offering a design guide for noise minimization in hydrogen fuel systems. Babaoglu et al. [24] utilized genetic algorithms and CFD to optimize MSTVs for maximum Di while minimizing forward pressure drop. Their results confirmed that reducing valve-to-valve distance (G/Dh)

improves performance significantly. They achieved a peak Di of 1.811 under a Reynolds number of 173.03.

F. Hybrid Systems and Emerging Applications

Lai et al. [25] employed Tesla valve-based mini-channels in an interlayer battery thermal management system. Their study concluded that MSTVs promote energy uniformity and delay temperature spikes during rapid discharge cycles, ensuring battery safety and longevity. Moria [26] explored hybrid spiral-tube exchangers with Tesla-inspired turbulators. The complex geometry promoted surface vortex interaction, maximizing thermal energy extraction from the working fluid. Reddy and Goud [27] simulated MHD nanofluid flow across vertical plates using Tesla-like geometry. They highlighted improved thermal gradients with minimum entropy generation—critical for heat recovery systems. Zhang et al. [28] studied turbulent flow over V-shaped ribs and dimples, drawing parallels with Tesla-induced vortices. Their results reinforced the idea that surface geometry significantly influences both Nu and f. Qian et al. [29] explored Mach number fluctuations in hydrogen decompression systems fitted with MSTVs. Their results emphasized the importance of stage-specific optimization to control flow-induced noise and shockwave formation.

G. Experimental Validation and Grid Verification

Validation remains a core focus in Tesla valve research. Simulation studies such as those by Paudel et al. [30] confirmed empirical Nu values within 1% accuracy across $Re = 50\text{--}500$, lending credibility to numerical methods used in heat transfer prediction. Grid independence was ensured using refined meshes (~1 million elements), ensuring accurate capture of vortex dynamics and thermal boundary layers.

H. Gaps in Literature and the Need for Comprehensive Frameworks

While prior studies thoroughly evaluate Di, fewer works address the joint dependency between Nusselt number, friction factor, and thermal diodicity. Additionally, most research focuses on either single-stage valves or heavily optimized MSTVs with 10+ stages. Intermediate designs (e.g., two-stage Tesla valves) remain underexplored despite their practical benefits in balancing complexity and efficiency. Real-world constraints such as temperature gradients, flow direction reversals, geometric tolerances, and non-Newtonian fluid effects are seldom accounted for, despite being prevalent in practical applications. This highlights the need for a consolidated performance framework that connects valve geometry, operating conditions, and thermal-fluid metrics under unified simulation and validation models.

Problem Statement

Despite the growing adoption of Tesla valves in modern thermal and microfluidic systems, there remains a significant gap in understanding their dual role as both flow rectifiers and heat transfer modulators. Much of the existing research has prioritized the analysis of pressure diodicity—measuring the valve's ability to restrict reverse flow—while neglecting the broader thermofluidic interactions that define its overall performance. This limited perspective overlooks how geometric asymmetry, flow direction, and vortex dynamics interact to influence heat transfer efficiency, energy dissipation, and pressure loss within valvular conduit channels. While multi-stage Tesla valve arrays have received considerable attention for high-performance applications, two-stage configurations—which offer a practical balance between geometric simplicity and performance—remain underexplored. There is also a lack of systematic studies that quantify how specific geometric parameters, such as conduction angle, side channel length, and arc curvature, impact critical thermal metrics like the Nusselt number, friction factor, and thermal diodicity. Most importantly, current literature lacks a unified performance framework that links thermal behavior with hydrodynamic resistance across bidirectional flows. Without such a framework, optimizing Tesla valve designs for compact systems—such as heat exchangers, battery cooling modules, and MEMS—remains largely empirical and inefficient. Therefore, this study aims to fill these gaps by conducting a detailed numerical investigation of a two-stage Tesla valve integrated into a valvular conduit channel. It evaluates both forward and reverse flow scenarios under varying Reynolds numbers, correlating geometry with flow characteristics and thermal performance. By introducing and analyzing the interdependent behavior of Nu, f, and Di, this research seeks to establish a comprehensive, predictive design methodology for passive thermofluidic systems, paving the way for more reliable and energy-efficient solutions in thermal management applications.

This section outlines the computational framework used to evaluate the thermofluidic performance of a two-stage Tesla valve integrated into a valvular conduit channel. The study employs three-dimensional computational fluid dynamics (CFD) to analyze heat transfer, pressure drop, and flow behavior under various Reynolds number conditions.

A. Geometry and Boundary Conditions

The geometry used in this study consists of a 3D Tesla valve model embedded in a conduit with a circular cross-section. The primary channel has a radius of 0.8 units, designed to accommodate laminar fluid flow in both forward and reverse directions. The geometry includes a through-channel and a bifurcated arc path, representing the classic Tesla valve structure.

Table 1: Details of thermophysical properties of the material and fluid.

Thermophysical property	Steel	Water
Specific heat capacity	502.48 J/(kg.K)	4182 J/(kg.K)
Thermal conductivity	16.27 W/(m.K)	0.6 W/(m.K)
Thermal diffusivity	$12.74 \times 10^{-6} \text{ m}^2/\text{s}$	$1.4 \times 10^{-7} \text{ m}^2/\text{s}$
Melting point	1370°C–1510 °C	0 °C
Density	8030 kg/m ³	998.2 kg/m ³

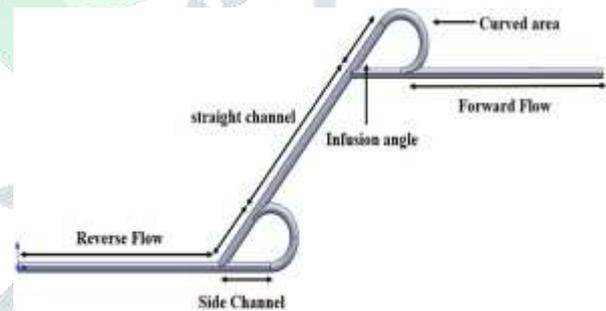


Fig 1. Schematic diagram of the Tesla valve design

The valve was simulated under steady-state flow conditions, allowing for a fully developed thermal and velocity field. The Reynolds number (Re) was varied between 25 and 500, ensuring that the flow remained within the laminar regime. These values were selected to assess performance across low to moderate flow rates, where vortex development is most pronounced.

Boundary conditions were assigned as follows:

- **Inlet:** Uniform velocity profile corresponding to desired Re .
- **Outlet:** Constant pressure (zero-gauge).
- **Walls:** No-slip condition, with a constant wall temperature of 340 K.
- **Fluid:** Water, with inlet temperature set at 292 K.

B. Governing Equations

The fluid motion and heat transfer were modeled using the Navier–Stokes equations, incorporating both momentum and energy conservation laws. The governing equations were based on the following assumptions:

- Newtonian fluid behavior: Viscous stress linearly dependent on strain rate.
- Incompressible flow: Constant fluid density.
- Constant thermophysical properties: Thermal conductivity, viscosity, and specific heat are treated as constants.
- No body forces or radiation effects: Gravitational and radiative influences are negligible in this study.

The equations used are:

- Continuity equation
- Momentum equations (Navier–Stokes)
- Energy equation for thermal field resolution.

These equations were solved using appropriate numerical schemes as described in the simulation setup.

C. Simulation Setup

The numerical simulations were performed using ANSYS Fluent, a robust CFD solver widely used for thermofluidic modeling. The computational domain was discretized using a structured mesh, resulting in approximately 1 million elements. This mesh density was chosen based on grid independence studies to ensure accurate resolution of vortices and boundary layers without excessive computational cost.

The Laminar Flow Model was employed, given the low Reynolds numbers used in the simulations. The pressure–velocity coupling was handled using the SIMPLE algorithm. For spatial discretization, the second-order upwind scheme was applied to both momentum and energy equations to enhance solution accuracy.

To validate the simulation method, the Nusselt number results obtained for flow in standard circular and rectangular channels were compared with analytical values from literature. The maximum deviation was within 1%, confirming the accuracy of the computational model.

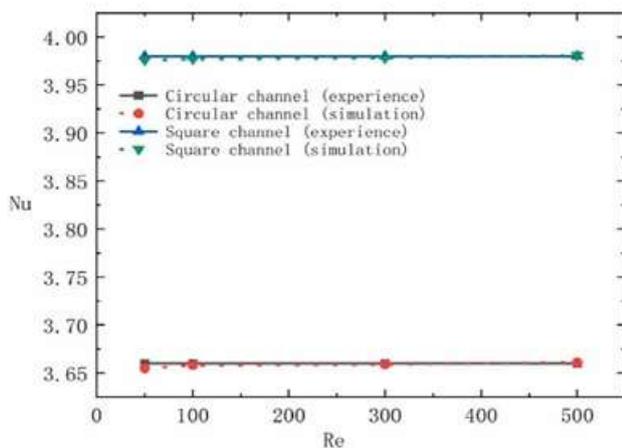


Fig. 2. Grid validity verification.

The convergence criterion for all simulations was defined by residual values falling below 1×10^{-6} , ensuring numerical stability and reliability in both velocity and temperature fields.

IV. EXPERIMENTS AND RESULTS

This section presents a detailed discussion of the numerical results obtained from the simulations of the Tesla valve under varying Reynolds numbers (Re) and geometric configurations. The analysis is categorized into flow characteristics, pressure distribution, heat transfer behavior, and the influence of geometric parameters.

A. Flow Characteristics

The velocity field reveals distinct differences in behavior between forward and reverse flows. In forward flow, the streamlines primarily follow the central through-channel, with minimal disturbance, especially at low Reynolds numbers ($Re = 25$). However, as Re increases, vortex formation becomes more prominent near the arc channel entrance and exit. These vortices cause localized velocity fluctuations and slight flow separation.

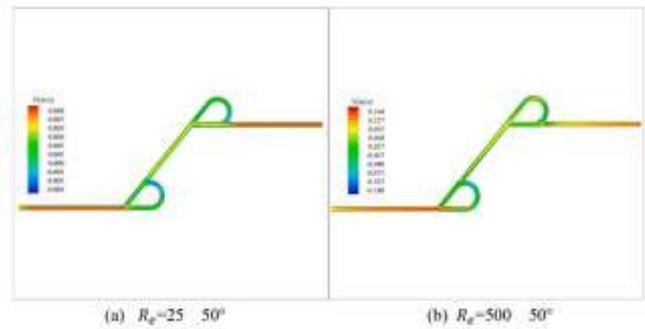


Fig. 3. Reverse flow velocity distribution.

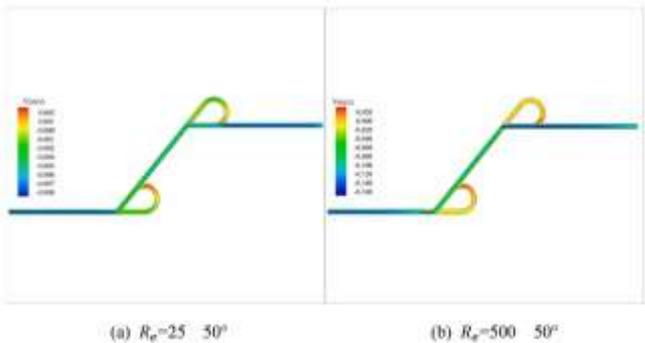


Fig. 4. Forward flow velocity distribution

In contrast, reverse flow results in significantly more complex flow patterns. Vortex trapping and recirculation zones are observed throughout the arc channel, especially near junctions and curved segments. These structures increase flow resistance and cause a drop in fluid velocity along the reverse path. The strength and size of the vortices increase with higher Re, making reverse flow considerably more dissipative.

B. Pressure Distribution

A comparison of pressure distribution between flow directions indicates a clear asymmetry. In forward flow, pressure gradually decreases along the conduit due to smooth streamwise motion. However, in reverse flow, pressure drops sharply in areas with vortex formation and flow impingement, particularly within the arc channel.

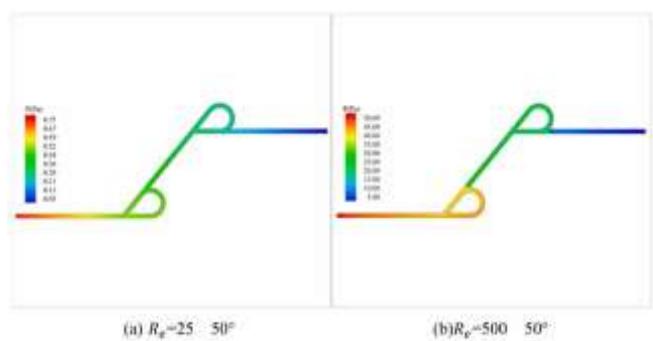


Fig. 5. Reverse flow Pressure distribution

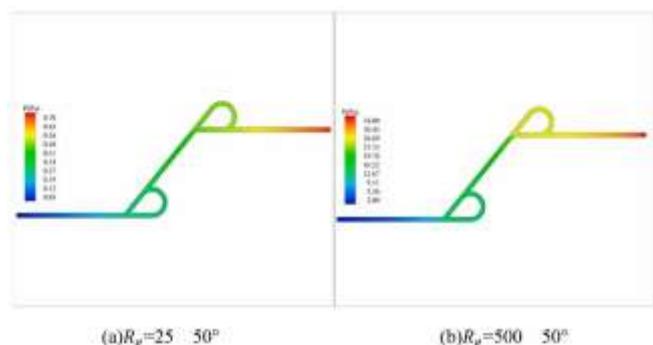


Fig. 6. Forward flow Pressure distribution

Interestingly, pressure recovery is observed at certain arc exits in both directions, which can be attributed to sudden expansion or merging of flow paths. These effects are more evident at $Re = 500$, where flow inertia and geometric confinement interact strongly.

C. Heat Transfer Analysis

Heat transfer behavior was assessed using the Nusselt number (Nu), a dimensionless measure of convective heat transfer. It was found that Nu decreases with increasing conduction angle for both forward and reverse flows. Larger angles result in longer, smoother fluid paths, which reduce turbulent mixing and limit convective enhancement.

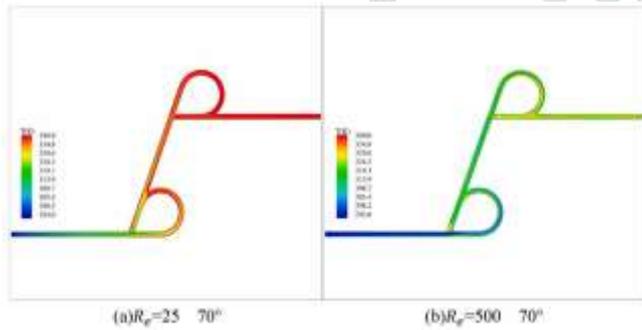


Fig. 7. Reverse flow temperature distribution

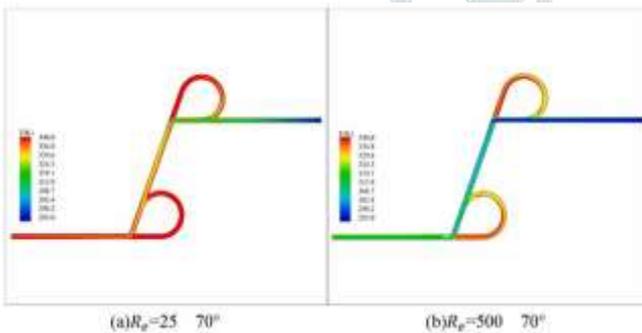


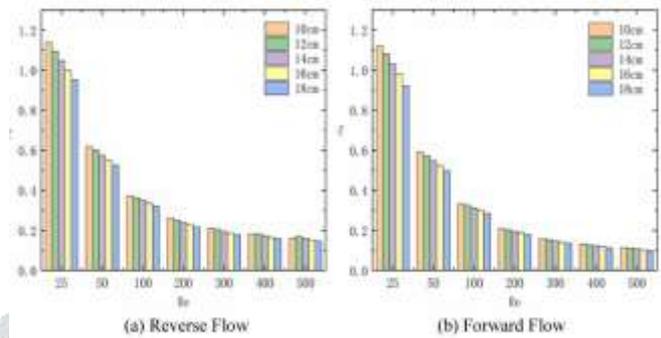
Fig. 8. Forward flow temperature distribution.

Despite this trend, reverse flow consistently yields higher Nusselt numbers. This is due to the presence of transverse vortices, which compress the thermal boundary layer and enhance fluid-wall interaction. These vortices increase turbulence intensity and disrupt thermal stratification, leading to greater heat absorption from the wall.

D. Influence of Geometry

Geometric variations in the Tesla valve significantly impact its thermal-hydraulic performance:

- Side channel length: Increasing the length of the arc or side channel leads to a decrease in both Nusselt number and friction factor (f). This is attributed to reduced turbulence and smoother flow paths, which improve flow stability but lower heat transfer efficiency. The extended geometry allows the fluid to follow the contour with less disruption, thereby decreasing mixing.
- Guide angle (conduction angle): This parameter has a strong influence on both thermal diodicity (Dit) and Nu variability. Larger guide angles intensify vortex formation and thermal asymmetry, especially at higher Reynolds numbers. As the angle increases, so does the disparity between forward and reverse flow heat transfer, resulting in a pronounced diodic effect.

Fig. 9. Variation of the length of different side channels versus f

These findings confirm that Tesla valve performance is highly sensitive to geometric design, and careful tuning of these parameters is necessary to balance flow resistance and thermal performance.

V. CONCLUSION

This study presents a comprehensive numerical analysis of fluid flow and thermal behavior in a valvular conduit channel featuring a Tesla valve configuration. Leveraging detailed CFD simulations across a range of Reynolds numbers ($Re = 25-500$), the research captures the intricate interplay between valve geometry, vortex dynamics, pressure distribution, and convective heat transfer. The findings emphasize the directional asymmetry that characterizes Tesla valve operation, revealing distinct thermal and hydrodynamic behaviors in forward and reverse flows. Key observations include the formation of longitudinal vortices in forward flow, which stabilize the boundary layer, and transverse vortices in reverse flow, which compress the thermal boundary and intensify turbulence. These phenomena collectively enhance heat transfer efficiency in reverse flow, as confirmed by consistently higher Nusselt numbers and thermal diodicity indices under similar geometric and flow conditions. The study demonstrates that geometric parameters—particularly conduction angle and arc channel length—significantly impact both thermal performance and flow resistance. An increase in conduction angle enhances vortex generation but also elevates pressure drop, highlighting the inherent trade-off between heat transfer and energy loss. Similarly, lengthening the side channel reduces flow resistance but compromises thermal mixing, indicating the importance of geometric balance in valve design. The validation of numerical results with benchmark Nusselt data ensures the reliability of the simulation methodology. The introduction of thermal diodicity as a comparative metric provides an additional layer of performance evaluation, suitable for thermal management applications requiring directional heat control. In essence, this study not only deepens the understanding of Tesla valve thermofluidic mechanisms but also establishes a foundation for future optimization and application in energy systems, microfluidics, and compact heat exchangers. The insights gained serve as a design framework for engineers aiming to develop passive, reliable, and efficient thermal control components in next-generation systems.

REFERENCES

- [1] N. Tesla, "Valvular Conduit," U.S. Patent 1,329,559, Jan. 1920.
- [2] S. Mohammadzadeh, et al., "Numerical analysis of Tesla Micro-Valves," *Int. J. Heat Mass Transf.*, vol. 124, pp. 1332–1344, 2018.
- [3] P. Porwal, et al., "Heat Transfer and Fluid Flow Characteristics in Multistaged Tesla Valves," *Numerical Heat Transfer Part A*, vol. 73, no. 6, pp. 347–365, 2018.
- [4] Y. Bao and H. Wang, "Numerical Study on Flow and Heat Transfer Characteristics of a Novel Tesla Valve," *Int. J. Heat Mass Transf.*, vol. 187, 2022.
- [5] S. De Vries, et al., "Design and Operation of a Tesla-Type Valve for Pulsating Heat Pipes," *Int. J. Heat Mass Transf.*, vol. 105, pp. 1–11, 2017.
- [6] A.Y. Nobakht, et al., "Numerical Study of Diodicity Mechanism in Different Tesla-Type Microvalves," *J. Appl. Res. Technol.*, vol. 11, no. 6, pp. 876–885, 2013.
- [7] S.M. Thompson, et al., "Numerical Investigation of Multistaged Tesla Valves," *J. Fluid Eng.*, vol. 136, no. 8, 2014.
- [8] J. Du, et al., "Numerical Thermal Analysis of Cold Plate Embedded in Novel Multistage Tesla Valve," *SMMP Conf.*, 2022, pp. 151–161.

- [9] Y. Lu, et al., "Performance Optimization of Tesla Valve-Type Channel for Cooling Lithium-Ion Batteries," *Appl. Therm. Eng.*, vol. 212, 2022.
- [10] P.R. Menon, et al., "Numerical Studies on Influence of Surface Heating on Flow in Tesla Valve," *J. Pharm. Negative Results*, 2022.
- [11] S. Mohammadzadeh et al., *Int. J. Heat Mass Transf.*, vol. 124, pp. 1332–1344, 2018.
- [12] S.M. Thompson et al., *J. Fluid Eng.*, vol. 136, no. 8, 2014.
- [13] A. Basil et al., *Appl. Therm. Eng.*, vol. 152, pp. 642–655, 2019.
- [14] P.R. Porwal et al., *Numer. Heat Transf. Part A*, vol. 73, pp. 347–365, 2018.
- [15] P. Hu et al., *Phys. Fluids*, vol. 34, no. 3, 2022.
- [16] Y. Bao, H. Wang, *Int. J. Heat Mass Transf.*, vol. 187, 2022.
- [17] J. Raffel et al., *J. Fluid Eng.*, vol. 143, no. 11, 2021.
- [18] M. Sebastian et al., *Microsystems & Nanoengineering*, vol. 8, no. 1, 2022.
- [19] F. Gong et al., *Appl. Energy*, vol. 329, 2023.
- [20] Y. Fan et al., *Appl. Therm. Eng.*, vol. 220, 2023.
- [21] Y. Lu et al., *Appl. Therm. Eng.*, vol. 212, 2022.
- [22] J. Du et al., *SMMP Conf.*, 2022.
- [23] J. Qian et al., *Energy*, vol. 179, 2019.
- [24] N.U. Babaoglu et al., *Chem. Eng. Technol.*, vol. 45, no. 12, 2022.
- [25] C. Lai et al., *Energy Convers. Manag.*, vol. 280, 2023.
- [26] H. Moria, *Results Eng.*, vol. 17, 2023.
- [27] Y.D. Reddy, B.S. Goud, *Results Eng.*, vol. 17, 2023.
- [28] P. Zhang et al., *Int. J. Heat Mass Transf.*, vol. 178, 2021.
- [29] J. Qian et al., *Int. J. Hydrogen Energy*, vol. 44, no. 26, 2019.
- [30] B.J. Paudel et al., *ASME Fluids Eng. Conf.*, 2014.

