

Passive Directional Airflow Control in Building Façades Using Tesla Valve Geometry: CFD Simulation and Experimental Validation

Yuto Morishita ^{*1}, Jiaye Li ¹, Haozheng Wang ¹ and Jihun Kim ¹

¹Weitzman School of Design, University of Pennsylvania, USA

*myuto@alumni.upenn.edu

Abstract

This study investigates the integration of Tesla valve geometries into double-skin façades (DSFs) as a passive strategy for enhancing natural ventilation and mitigating flow instability. The system leverages the Tesla valve's asymmetric, fixed-geometry design to promote unidirectional airflow and restrict reverse flow without mechanical actuation. A hybrid methodology was employed, combining computational fluid dynamics (CFD) simulations and full-scale physical experiments to evaluate the aerodynamic and thermodynamic behavior of the system under wind-dominated and buoyancy-driven conditions. Results demonstrate that under wind-dominated scenarios (3.0 m/s inlet), the valve geometry significantly accelerates upward flow, achieving top-vent velocities of approximately 4.8 m/s—a 1.6-fold amplification—while maintaining negligible flow at the bottom vent (<0.5 m/s). Under buoyancy-driven conditions, the system effectively sustained passive airflow via the thermal stack effect, with peak velocities reaching 0.26 m/s. Experimental validation using a full-scale prototype confirmed these directional trends and evaluated ventilation performance through CO₂ decay monitoring. The integration of a slit-partition recorded approximately 60% reduction in pollutant concentration within 100 seconds under wind-driven conditions, consistent with displacement ventilation effects. Quantitative validation using Root Mean Square Error (RMSE) further confirmed the model's reliability in predicting momentum and pollutant flushing dynamics, despite observed thermal discrepancies under buoyancy. These findings suggest that Tesla valve integration provides a robust, low-maintenance mechanism for passive airflow regulation in high-performance building envelopes, offering a resilient alternative to complex automated façade systems. This research establishes a foundational framework for utilizing fluidic diodes in climate-responsive architectural design.

Keywords: Computational Fluid Dynamics (CFD), Natural Ventilation, Passive Airflow Control, Tesla Valve

Introduction

The Tesla valve, also known as a valvular conduit, was first patented by Nikola Tesla in 1920 as a passive fluid control device designed to favor unidirectional flow without requiring any moving parts (Tesla, 1920). The valve's geometry — typically consisting of curved channels and bifurcations — introduces high flow resistance in the reverse direction while maintaining low resistance in the forward direction (Figure 1).



Figure 1. CFD Visualization of Directional Flow and Velocity Distribution through a Tesla Valve Geometry

These asymmetric flow properties have been recognized as valuable in fields including microfluidics, chemical engineering, and thermal management (Nalawade et al., 2024; Purwidyantri & Prabowo, 2023; Zhang et al., 2025). Recent advancements in Tesla valve design explore geometric modifications that allow enhanced diodicity or even reversible flow under external stress, further expanding their range of applicability (Al Balushi & Dahi Taleghani, 2024; Zeidan et al., 2024). These developments reinforce the value of Tesla valves as compact, low-maintenance components with high durability due to the absence of mechanical elements. While Tesla valves are well-documented in these traditional engineering systems, their application in architectural airflow systems remains largely unexplored. In building envelope research, passive flow control within Double-Skin Façade (DSF) cavities has been identified as a persistent challenge—particularly the risk of flow reversal, recirculation, and pressure instability under varying wind conditions (Dama et al., 2017; Xu et al., 2024). Existing DSF solutions typically incorporate operable shading devices

and façade elements, including vents and motorized components, with increasing use of automated control strategies to manage thermal and environmental performance (Shameri et al., 2011; Zelenay et al., 2011). These active systems introduce additional complexity, maintenance requirements, and potential operational failure modes. According to the current state of knowledge, despite the established use of Tesla valves in engineering disciplines, their potential to resolve these specific pressure and flow instabilities in DSFs has not yet been systematically investigated.

This study addresses this gap by adapting the Tesla valve's asymmetric channel geometry to the vertical orientation of building façades. The primary objective is to evaluate the valve's effectiveness as a fixed-geometry diode for passive airflow control and natural ventilation. By stabilizing and directing internal cavity flow without relying on mechanical actuation, this geometric intervention provides a robust, low-maintenance alternative to traditional dynamic façades.

To demonstrate that these capabilities are not merely theoretical, this research employs a hybrid methodology combining computational fluid dynamics (CFD) simulations and full-scale physical experiments to:

1. Investigate the feasibility of Tesla valve integration to mitigate reverse flows and enhance directional airflow within the envelope.
2. Evaluate the system's measurable contribution to natural ventilation and pollutant removal, specifically quantified through CO₂ concentration changes.

Initially, the Tesla valve geometry was modified for the vertical orientation of building façades, resulting in two variations (Figure 2). While Option (a) retains the complex curves of the original patent, this study focuses on the validation of Option (b) for its simplified profile, which prioritizes cost-effectiveness and architectural constructability. Option (c) illustrates the modular sample assembly diagram of the Option (b) geometry within a building envelope system.

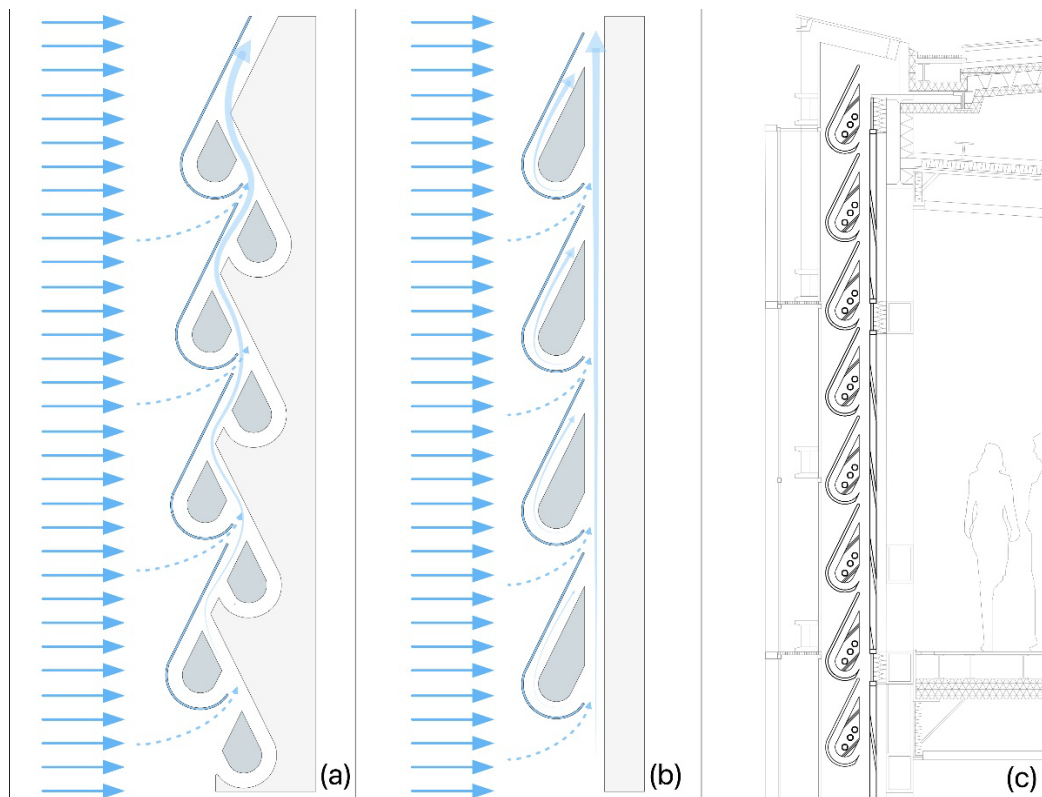


Figure 2. Tesla Valve-integrated DSF Design: Original Curvature Adaptation (a); Proposed Simplified Geometry for Validation (b); Sample Modular Assembly (c)

Methodology

CFD Simulation Setup

This research employed a hybrid methodology comprising computational fluid dynamics (CFD) analysis and full-scale physical validation experiments to evaluate the thermo-aerodynamic behavior of the proposed Tesla valve-integrated DSF system. To analyze airflow characteristics, CFD simulations were conducted using RhinoCFD (powered by the established PHOENICS solver). The Chen-Kim modified k-epsilon (KECHEN) model was selected as the turbulence model. This variant was chosen for its computational superiority and accuracy in resolving complex flow behaviors, such as flow separation, recirculation, and swirling vortices, which are critically anticipated within the asymmetric geometry of the Tesla valve. The reliability of the Chen-Kim model has been extensively validated against standard k-epsilon models in foundational aerodynamic studies (Kim & Marshall, 1987; Weber et al., 1990). Furthermore, its application is well-documented and validated specifically within built-environment research, including full-scale confined space measurements and scaled wind-tunnel experiments for architectural spaces (Nguyen & Reiter, 2011). To capture the unsteady aerodynamic and thermal behaviors within the cavity, a transient simulation scheme was adopted. Thermal simulations utilized active radiation equations to account for heat transfer across the envelope. Global convergence criteria were established at a residual of 0.01, executing 50 sweeps per time step, with manual relaxation factors finely tuned to ensure numerical stability.

The CFD study evaluated two primary environmental scenarios:

- **Wind-Dominated Conditions:** A constant inlet velocity of 3.0 m/s was applied for 1,800 seconds with radiation disabled, isolating the pure aerodynamic effects of the valve geometry.
- **Buoyancy-Driven Conditions:** To simulate thermally induced flow without external wind, initial temperature boundary conditions were assigned across the glazing and concrete layers. The simulation spanned a 10,800-second (3-hour) heating phase, followed by a 32,400-second (9-hour) cooling phase.

In both cases, airflow velocity, temperature, and pollutant (CO₂) concentration level were monitored at virtual sensor locations within the cavity. For ventilation evaluation, additional 5-minute pollutant stabilization was followed by CO₂ decay monitoring (Figure 4).

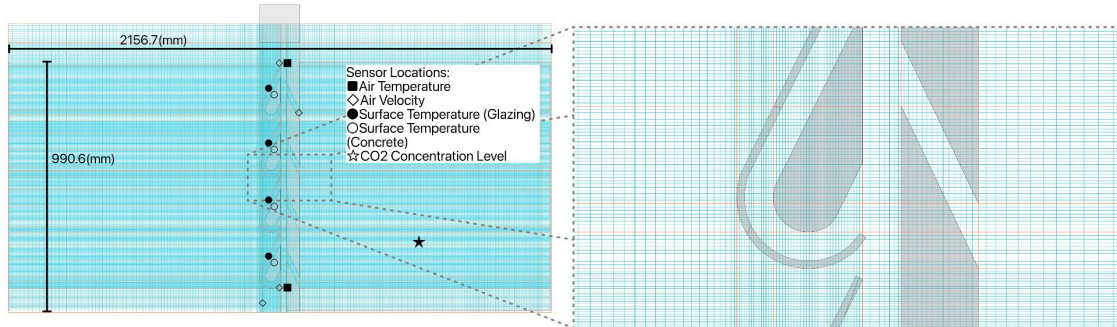


Figure 4. Sensor Locations and Domain Discretization

Boundary and Initial Conditions

The ambient air temperature was maintained at 20°C. Initial surface temperatures for the glazing and concrete components were defined based on empirical measurements (Table 1) to accurately simulate thermal gradients during the heating and cooling phases. To assess pollutant dispersion, the initial cavity CO₂ concentration was set to 1,700 ppm at the source, anticipating dispersion to target a monitored baseline concentration of approximately 1,500 ppm at the sensor locations.

Table 1: The List of Initial Temperature Assigned to Geometries (Blue: Glazing; Gray: Concrete; #1: Top, #4: Bottom) under Buoyancy-driven Condition (°C)

Start Time	End Time	#1	#2	#3	#4	#1	#2	#3	#4
0	900	22.79	22.84	22.86	22.7	22.91	23.06	23.19	22.98
901	1800	28.52	27.44	25.38	23.96	26.82	25.99	24.89	23.99
1801	2700	30.11	28.36	25.76	24.09	28.63	26.84	25.56	24.35
2701	3600	30.75	28.78	25.98	24.33	30.02	27.39	26.03	24.65
3601	4500	31.25	29.04	26.26	24.55	31.06	27.82	26.41	24.93
4501	5400	31.45	29.26	26.49	24.67	31.77	28.16	26.75	25.16
5401	6300	31.5	29.4	26.61	24.73	32.41	28.46	27.0	25.34
6301	7200	31.91	29.41	26.62	24.76	32.98	28.63	27.24	25.47

7201	8100	32.05	29.59	26.7	24.76	33.42	28.95	27.44	25.64
8101	9000	32.14	29.66	26.69	24.92	33.76	29.06	27.59	25.76
9001	9900	32.06	29.56	26.76	24.7	33.97	29.06	27.72	25.81
9901	10800	32.16	29.67	26.75	24.91	34.23	29.29	27.84	25.89

Geometry and Model Configuration

The two-dimensional simulation domain was constructed in Rhino3D to replicate the geometric profile of the physical prototype. The computational model comprised the teardrop-shaped exterior glazing, the concrete core, and the internal polyurethane wall. Material thermodynamic properties were assigned using RhinoCFD's BLOCKAGE attribute panel, defined as follows:

1. Glazing: emissivity 0.1 (exterior), 0.9 (interior); solar absorption 0.1 (exterior), 0.2 (interior)
2. Concrete: emissivity 0.9; solar absorption 0.9
3. Polyurethane insulation: emissivity 0.9, solar absorption 0.2

Meshing Strategy

A non-uniform structured mesh was generated to capture critical temperature and velocity gradients, using a maximum growth rate of 1.2 near boundary zones (Figure 4). The grid consisted of 250 divisions in the x-direction, 1 in the y-direction, and 476 in the z-direction, totaling 119,000 cells in the 2D simulation domain. Finer resolution was applied around Tesla valve geometries and cavity interfaces to resolve flow separation, recirculation, and shear layers, while coarser spacing was used in regions with minimal gradient variation to optimize computational efficiency. Grid sensitivity checks confirmed mesh independence and numerical stability. The CFD model outputs—velocity vectors, temperature profiles, and pollutant decay—provided the basis for performance evaluation and were later validated through full-scale physical experiments, as described in the following section.

Physical Experiment Setup

To validate the computational findings, full-scale physical experiments were conducted using a physical prototype of the Tesla valve-integrated DSF system. The physical model was constructed at a 1:1 scale relative to the computational domain to enable a direct, empirical comparison between the simulated and measured results.

Prototype Construction

The full-scale prototype featured a vertically oriented, teardrop-shaped thermal mass cast from concrete, paired with exterior glazing constructed from heat-formed acrylic sheets. To rigorously guide the internal airflow, structural sidewalls and internal partitions—both with and without designated boundary slits—were CNC-machined from high-density urethane (HDU). This fabrication method ensured strict geometric precision and structural stability. The final assembled prototype faithfully replicated the spatial configuration and cross-sectional geometry of the 2D computational domain (Figure 5).



Figure 5. Full-scale Physical Prototype Assembly

Measurement Equipment and Data Acquisition

Table 2 summarizes the technical specifications of the instruments utilized to monitor air velocity, surface and air temperatures, CO₂ concentrations, and thermal distribution. This instrumentation network enabled synchronized data acquisition under both wind-dominated and buoyancy-driven conditions, facilitating a direct comparison with the computational outputs. Physical sensor placements corresponded precisely to the virtual monitoring points established in the CFD domain (Figure 4).

Table 2: Technical Specifications and Accuracy of Sensor Equipment Used in Physical Experiment

Equipment	Purpose	Quantity	Accuracy
Self-Describing Air/Water/Soil Temperature Sensor	Air temperature	2	-40° to 100°C: +/- 0.15°C
Type K 6 ft Beaded Thermocouple Sensor	Surface temperature	8	0 to 285°C: +/- 2.2°C
Air Velocity & Temperature Remote Head Sensor	Air velocity	4	0.15 to 1.0 m/s: +/- 0.05 m/s; 0.5 to 10 m/s: +/- 0.10 m/s
K30 FR Fast Response 10,000 ppm CO ₂ Sensor (030-8-0010)	CO ₂ concentration level	1	0 to 10000ppm: +/- 30 ppm +/- 3 % of reading
Flir C2 IR Camera	Thermal imaging	1	+/- 2°C or 2%

Experimental Conditions

To simulate controlled environmental boundaries, a custom wind tunnel (1,000 mm × 1,000 mm × 310 mm) was constructed from transparent acrylic panels. The tunnel featured an array of 100 embedded PC fans capable of delivering a uniform, variable-speed airflow up to a maximum velocity of 3.0 m/s. The full-scale prototype was centrally positioned within the tunnel, dividing the space into windward (front) and leeward (rear) chambers for precise observation of the cavity's aerodynamic behavior. (Figure 6).

Two environmental scenarios were physically replicated across both partition configurations (solid and slit-integrated) to directly validate the CFD boundary conditions. To ensure consistent assessment across both scenarios, the solid partition was utilized to evaluate airflow acceleration, whereas the slit-integrated HDU board was employed specifically for pollutant flushing. Prior to all ventilation tests, the rear chamber was pre-filled with smoke to establish a target baseline CO₂ concentration of 1500 ppm. The specific environmental inputs were applied as follows:

1. Wind-Dominated Conditions: Airflow at 3.0 m/s was supplied continuously for 30 minutes without heat input to isolate aerodynamic effects (Figure 6 (a)).
2. Buoyancy-Driven Conditions: A 300 W infrared heat lamp mounted above the cavity provided thermal input during a 3-hour heating phase, followed by a 9-hour passive cooling period to replicate the thermal decay modeled in the CFD simulation (Figure 6 (b)).

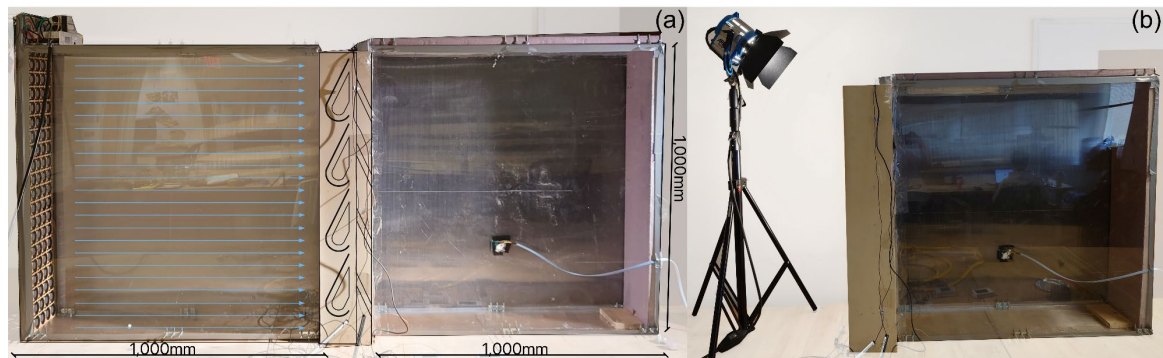


Figure 6. Physical Experimental Setup: Wind-dominated Conditions (a), Buoyancy-driven Conditions (b)

Results and Discussion

Wind-Dominated Conditions

Airflow Acceleration (Solid Partition)

Figure 7 illustrates the temporal velocity profiles at selected sensor locations under wind-dominated conditions using the solid partition. The CFD-predicted airflow at the top vent reached a stable peak of

approximately 4.8 m/s. This represents a 1.6-fold amplification of the 3.0 m/s inlet velocity, demonstrating the system's capacity to enhance airflow via geometric directional acceleration. Conversely, air velocity at the bottom vent remained below 0.5 m/s in both the simulation and the physical experiment, confirming strong flow asymmetry and the effective restriction of reverse flow. Overall, the experimental measurements show close agreement with the CFD predictions, validating the model's ability to capture the unidirectional acceleration induced by the Tesla valve geometry.

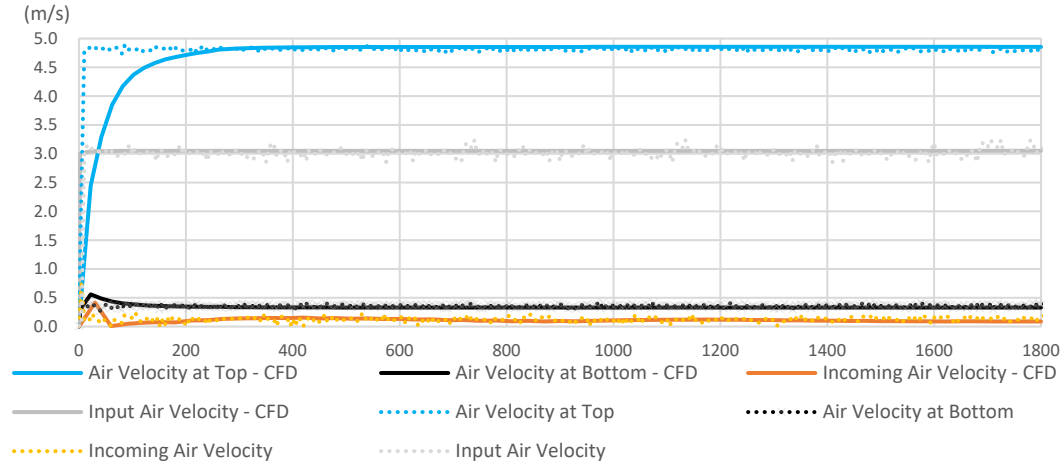


Figure 7. Air Velocity Profiles (Solid Partition)

Airflow and Pollutant Flushing (Slit-Integrated Partition)

Figure 8 (Left) presents the temporal velocity profiles at the top vent and highest slit under wind-dominated conditions using the slit-integrated partition. Compared to the solid configuration, the top vent's peak velocity reduced to approximately 4.2 m/s, indicating that the slits moderately dissipate directional momentum to drive interior air exchange. Velocity at the slit stabilized near 0.25 m/s, with CFD and experimental trends aligning closely after initial computational overshoots. As visualized in the CFD velocity field (Figure 9(a)) and corresponding airflow diagram (Figure 9(b)), this dynamic induces a displacement ventilation effect—drawing lower-level air inward while driving interior circulation and upward exhaust through the slit and top vent.

This displacement mechanism effectively flushes pollutants from the rear chamber, successfully simulating indoor air renewal. As shown in Figure 8 (Right), physical experiments recorded a rapid CO₂ concentration decay from approximately 1,440 ppm to 597 ppm within the first 100 seconds — representing a reduction of approximately 60%. While CFD predictions indicated a slightly more accelerated decay toward zero, both datasets conclusively demonstrate the system's high capacity for passive ventilation. Visual smoke tracking (Figure 10) corroborated these findings, confirming the progressive upward discharge of CO₂-laden air from the simulated indoor environment.

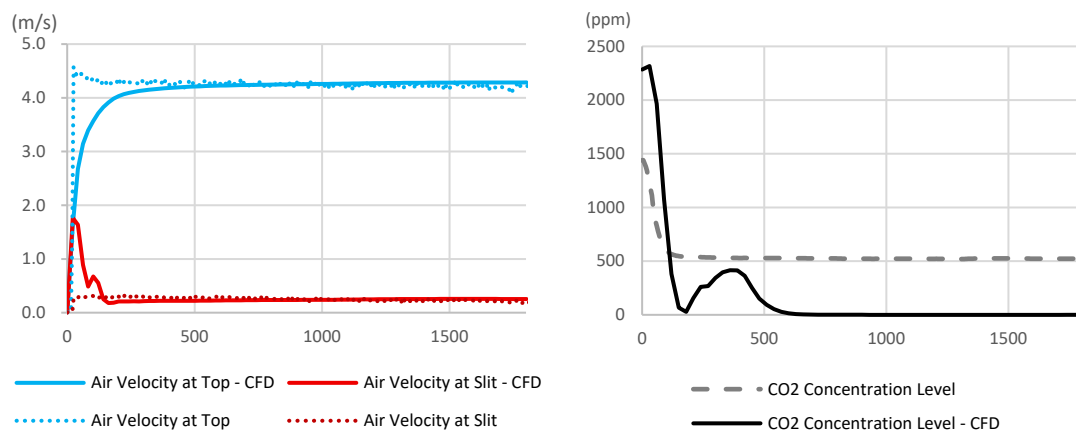


Figure 8. Air Velocity Profiles of (Left) and CO₂ Concentration Decay (Right) for the slit-integrated partition.

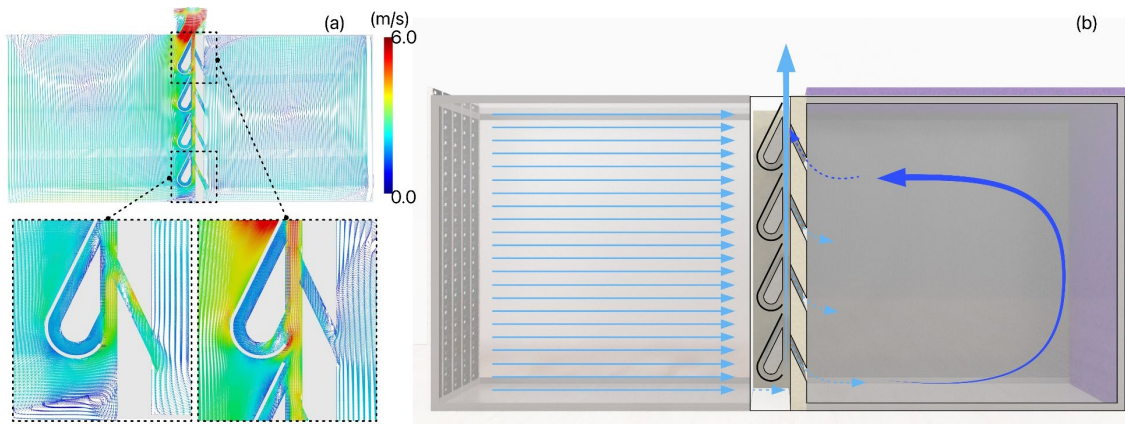


Figure 9. CFD Velocity Field (a) and Induced Airflow Loop Diagram (b)

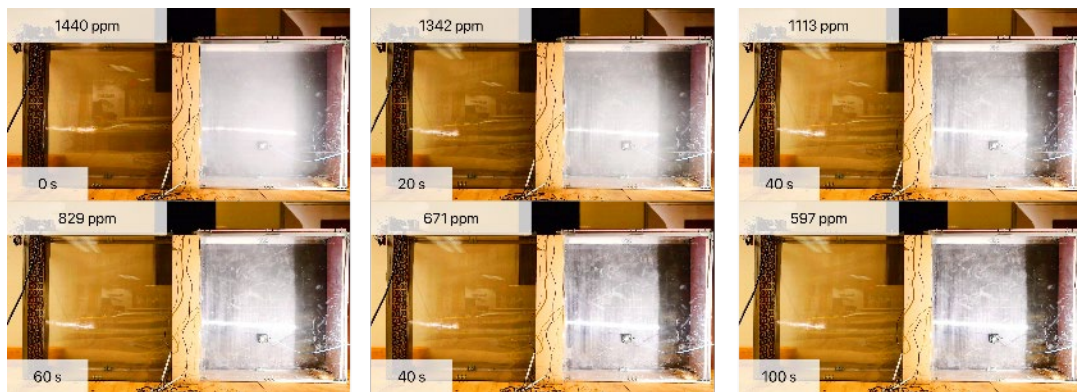


Figure 10. Visual Tracking of CO₂ Concentration Decay during Physical Experiment

Buoyancy-Driven Conditions

Thermally Induced Airflow (Solid Partition)

Figure 11 presents the temporal evolution of air velocity under buoyancy-driven conditions using the solid partition. During the initial 3-hour (10,800 s) heating phase, the thermal stack effect drove a steady increase in upward airflow. CFD simulations predicted a peak top-vent velocity of approximately 0.24 m/s, while physical experiments recorded a closely aligned peak of 0.26 m/s.

Throughout the subsequent 9-hour passive cooling phase, both datasets exhibited a gradual velocity decline. While CFD data indicated a smooth decay, physical sensor readings at the top vent fell below detection thresholds after approximately 6 hours of cooling. Crucially, air velocity at the bottom vent remained negligible (~ 0.01 m/s) throughout the entire 12-hour cycle in both models, validating the dominance of unidirectional natural convection. These results confirm the system's ability to passively generate and maintain low-rate upward ventilation solely through thermal buoyancy, while restricting reverse flow.

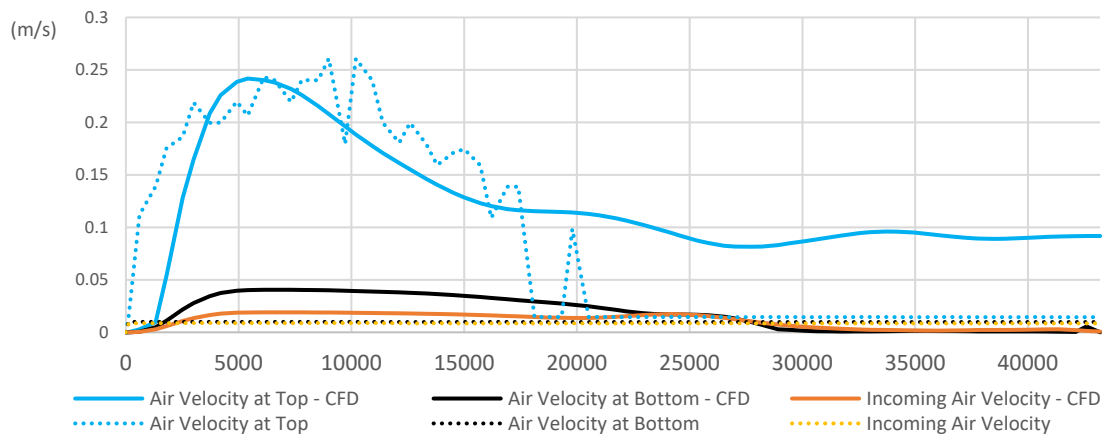


Figure 11. Air Velocity Profiles during Heating and Cooling Phases (Solid Partition)

Airflow and Pollutant Flushing (Slit-Integrated Partition)

Figure 12 (Left) illustrates the temporal velocity profiles under buoyancy-driven conditions using the slit-integrated partition. The highest slit recorded a peak velocity of approximately 0.05 m/s, indicating localized exhaust flow from the simulated interior into the cavity. Compared to the solid configuration, the top vent exhibited attenuated peak velocities of ~ 0.16 m/s (CFD) and ~ 0.18 m/s (Experimental). This reduction indicates that the slits geometrically alter the pressure distribution, diverting a portion of the upward flow and slightly diminishing the primary vertical stack effect.

Supporting these thermo-aerodynamic shifts, Figure 12 (Right) displays the corresponding two-phase CO₂ concentration decay. During the 3-hour active heating phase and an ensuing 2 hours of strong residual uplift, experimental CO₂ levels dropped by over 50% (from $\sim 1,500$ ppm to ~ 750 ppm). While the CFD simulation initialized at a higher concentration, it exhibited a parallel decay slope, reaching ~ 700 ppm over the same 5-hour period. A secondary, slower decay phase persisted for the remaining 6 hours, driven solely by residual thermal gradients. The strong agreement in decay rates between the physical and computational datasets validates the model and confirms that the system can sustain low-rate, passive indoor air renewal under thermal uplift, even with slit-induced airflow redistribution.

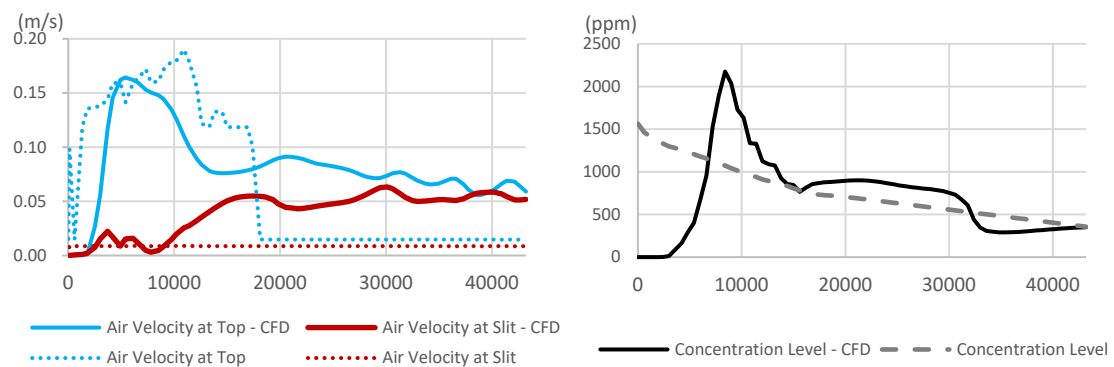


Figure 12. Air Velocity Profiles (Left) and CO₂ Concentration Decay (Right) during Heating and Cooling Phases (Slit-Integrated Partition)

Quantitative Model Validation (RMSE)

To quantitatively evaluate the predictive accuracy of the CFD model, Root Mean Square Error (RMSE) was used. To quantitatively evaluate the CFD model's predictive accuracy, the Root Mean Square Error (RMSE) was calculated across key parameters for both environmental scenarios (Figure 13).

Under wind-dominated conditions, temperature predictions showed strong agreement, with RMSEs of 0.28 °C and 0.08 °C at the bottom and top vents, respectively. Velocity RMSE averaged ~ 0.22 m/s across all sensor locations—representing only $\sim 5\%$ of the peak 4.8 m/s airflow—indicating robust momentum prediction. While the absolute CO₂ concentration RMSE was high (645 ppm) due to challenges in replicating the exact initial physical smoke dispersion, the corresponding CO₂ decay rate RMSE (234 ppm/s) confirms the model successfully captured the dynamic pollutant removal trends.

Under buoyancy-driven conditions, thermal prediction proved more challenging, yielding higher RMSEs (3.35 °C at the bottom, 7.10 °C at the top). These deviations highlight the inherent limitations of computationally modeling complex radiative-convective heat transfer from a localized physical heat lamp. Despite these thermal discrepancies, velocity RMSEs remained exceptionally low (0.01–0.02 m/s), validating the model's precision in capturing low-magnitude buoyant flow fields. Furthermore, CO₂ tracking yielded an RMSE of 226 ppm and a decay rate RMSE of 161 ppm/s, reinforcing the model's credibility in predicting two-phase passive ventilation.

Overall, the RMSE analysis supports the CFD model's validity for representing directional airflow and ventilation behavior. While precise thermal gradients under buoyancy remain harder to replicate, the model effectively predicts airflow magnitudes and pollutant flushing trends, demonstrating its utility in evaluating Tesla valve-integrated DSF ventilation strategies.

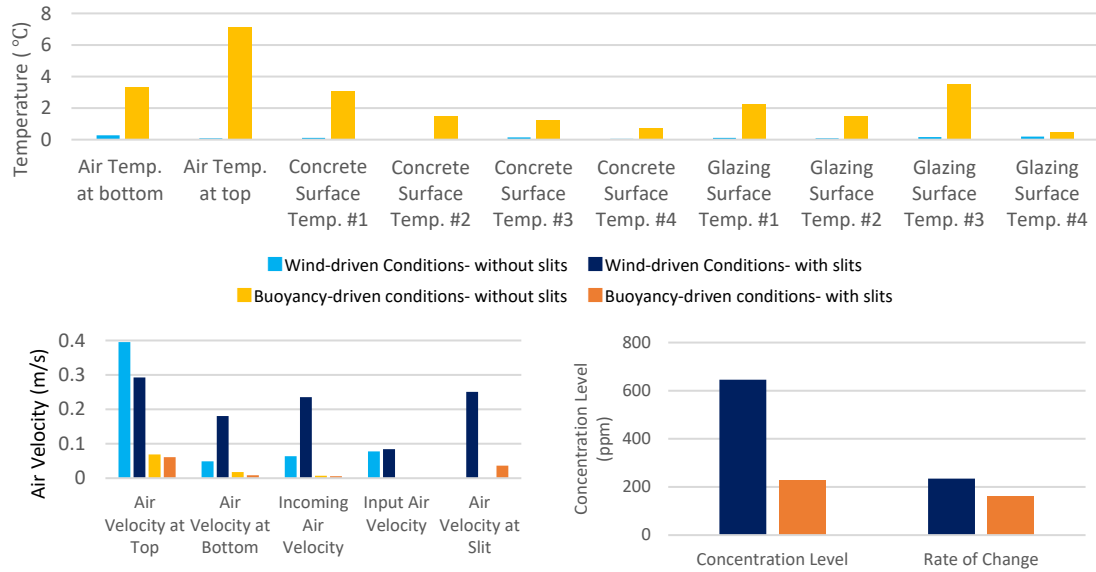


Figure 13. RMSE Comparison between CFD and Experimental Results for Temperature (Top), Air Velocity (Bottom Left), and CO₂ Concentration (Bottom Right)

Conclusion

This study investigated the integration of Tesla valve geometries into double-skin façade (DSF) systems as a passive, fixed-geometry mechanism for environmental control. Utilizing a hybrid methodology of CFD simulations and full-scale physical experiments, this research successfully addressed its two primary objectives:

1. **Mitigating Reverse Flows and Enhancing Directional Airflow:** The system demonstrated robust directional airflow control across both environmental scenarios. Under wind-dominated conditions, the solid partition configuration achieved a 1.6-fold amplification of upward airflow velocity compared to the inlet, while maintaining negligible velocity (<0.5 m/s) at the bottom vent, effectively restricting flow reversal and recirculation. Under buoyancy-driven conditions, the geometry successfully sustained continuous, unidirectional low-rate air exchange driven strictly by thermal uplift.
2. **Evaluating Contributions to Natural Ventilation and Pollutant Removal:** The system proved highly effective at passive indoor air renewal. Using the slit-integrated partition, physical and computational models recorded an approximate 60% reduction in CO₂ concentration within the first 100 seconds under wind-dominated conditions, and a sustained two-phase pollutant flushing under buoyancy-driven conditions.

Quantitative RMSE validation confirmed that the CFD framework reliably predicted these momentum and pollutant decay trends. However, discrepancies in absolute values—particularly regarding temperature under buoyancy—highlight the inherent limitations of replicating complex physical radiative-convective heat transfer in simulations.

Future research should focus on refining these computational thermal models, exploring in-situ building deployment across diverse climate zones, and conducting lifecycle cost assessments. Ultimately, by demonstrating robust passive ventilation and reverse-flow mitigation through static geometric manipulation, this research contributes a novel, low-maintenance pathway toward resilient, low-energy envelope design in climate-responsive architecture.

Acknowledgements

The authors gratefully acknowledge Max Hakkarainen for his invaluable support in the construction of the wind tunnel. The authors also extend their appreciation to the Department of Architecture at the Weitzman School of Design, University of Pennsylvania, for providing the facilities and resources necessary to carry out this study.

References

- Al Balushi, F., & Dahi Taleghani, A. (2024). A reversible miniaturized Tesla valve. *ASME Open Journal of Engineering*, 3, 031013. <https://doi.org/10.1115/1.4065510>
- Dama, A., Angeli, D., & Larsen, O. K. (2017). Naturally ventilated double-skin façade in modeling and experiments. *Energy and Buildings*, 144, 17–29. <https://doi.org/10.1016/j.enbuild.2017.03.038>
- Kim, S., & Marshall, G. (1987). *Computation of turbulent flows using an extended k-epsilon turbulence closure model* (NASA Contractor Report No. CR-179204). NASA.
<https://www.semanticscholar.org/paper/NASA-CONTRACTOR-REPORT-NASA-CR-179204-COMPUTATION-Kim-Marshall/d863d9ec5dd33a05a80f105e6dc09d01aa66c5dc>
- Nalawade, M., Wali, K., Kamble, H., Khandagale, S., & Khairnar, Y. (2024). Design and analysis of Tesla valve by comparing it with current valves. *International Journal of Research Publication and Reviews*, 5(11), 6695–6699.
- Nguyen, A. T., & Reiter, S. (2011). The effect of ceiling configurations on indoor air motion and ventilation flow rates. *Building and Environment*, 46(5), 1211–1222. <https://doi.org/10.1016/j.buildenv.2010.12.016>
- Purwidyantri, A., & Prabowo, B. A. (2023). Tesla valve microfluidics: The rise of forgotten technology. *Chemosensors*, 11(4), 256. <https://doi.org/10.3390/chemosensors11040256>
- Shameri, M. A., Alghoul, M. A., Sopian, K., Zain, M. F. M., & Elayeb, O. (2011). Perspectives of double skin façade systems in buildings and energy saving. *Renewable and Sustainable Energy Reviews*, 15(3), 1468–1475. <https://doi.org/10.1016/j.rser.2010.10.016>
- Tesla, N. (1920). *Valvular conduit* (U.S. Patent No. 1,329,559). U.S. Patent and Trademark Office.
- Weber, R., Visser, B. M., & Boysan, F. (1990). Assessment of turbulence modeling for engineering prediction of swirling vortices in the near burner zone. *International Journal of Heat and Fluid Flow*, 11(3), 225–235. [https://doi.org/10.1016/0142-727X\(90\)90041-9](https://doi.org/10.1016/0142-727X(90)90041-9)
- Xu, H., Yu, S., & Chen, Y. (2024). Experimental and theoretical modelling of fluctuating internal wind pressures on a ventilated double skin facade of a building. *Energy and Buildings*, 323, 114795. <https://doi.org/10.1016/j.enbuild.2024.114795>
- Zeidan, M., Németh, M., Abhijith, G. R., Wéber, R., & Ostfeld, A. (2024). Transient flow dynamics in Tesla valve configurations: Insights from computational fluid dynamics simulations. *Water*, 16(23), 3492. <https://doi.org/10.3390/w16233492>
- Zelenay, K., Perepelitza, M., & Lehrer, D. (2011). *High-performance facades: Design strategies and applications in North America and Northern Europe*. Center for the Built Environment, University of California, Berkeley. <https://escholarship.org/uc/item/4vq936rc>
- Zhang, Y., Ning, Y., & Liu, C. (2025). Research progress and application review of Tesla valve. *Academic Journal of Science and Technology*, 14(1), 262.