Numerical Analysis of a Novel Nuclear Upper Internal Structure Design for Flow Distribution Optimization to Mitigate Flow-Induced Vibration

Wenyu Mao[*,**], Jianbang Wu[*] and Wei Zhang[*]
Corresponding author: maowenyu@snerdi.com.cn
[*] Shanghai Nuclear Engineering Research and Design Institute, China.
[**] Shanghai Jiao Tong University, China.

Abstract: In pressurized water reactors, slender components such as control rod assemblies are prone to flow-induced vibration (FIV) caused by periodic hydraulic forces from the coolant. For the investigated 2×2 control rod guide tube array model in this research, periodic fluid excitation primarily arises from lateral flows in three regions: the upper nozzle region, the gap between the control rod guide tube flange and the core upper plate, and the outlet of the guide tube square slot. This numerical study conducts a comparative numerical study between a prototype and an optimized model, focusing on the changes in mass flow rate distributions across the core upper plate, as well as the lateral flow intensity in typical regions of the upper internal structure (UIS). The optimized model modifies the dimensions of four holes in the core upper plate and redesigns the support column base. The fluid dynamics simulation employs the Reynolds-Averaged Navier-Stokes (RANS) equations, with realizable k-epsilon model used for turbulence closure. In-house experiments conducted on a 1:4 scale 2×2 control rod guide tube array model demonstrate a significant improvement in flow distribution uniformity across the core upper plate, with the flow distribution ratios shifting from 1.11:0.82:0.96 in the prototype to 0.97:0.98:1.08 in the optimized model. The present numerical study corroborates these findings, exhibiting a consistent trend with the flow distribution ratios shifting from 1.17:0.75:0.90 to 0.98:0.96:1.07. Moreover, the experimental and numerical findings collectively confirm that the proposed design modifications effectively reduce lateral flow intensity in key regions of the upper internal structure, thereby suppressing FIV phenomenon.

Keywords: Flow-Induced Vibration (FIV), Control Rod Guide Tube Array, Reynolds-Averaged Navier–Stokes (RANS) Model, Computational Fluid Dynamics (CFD), Flow Distribution Optimization.

1. Introduction

Control rod assemblies function as one of the most critical components for nuclear reactor safety, regulating the fission chain reaction rate through neutron absorption to enable both power adjustment and serve as the primary method of emergency shutdown. However, high-speed coolant flows through the flow holes of the core upper plate into the guide tubes can induce fluid-elastic vibration in these slender structural components [1, 2, 3, 4, 5, 6, 7, 8]. This vibration may lead to excessive wear at the control rod-guide tube contacting interface, raising functional and safety issues. For instance, during an outage inspection at an operating nuclear power plant, abnormal wear was observed on several control rod bundles after one fuel cycle. The wear primarily occurred on fully withdrawn and stationary control rods, with one suspended rod even experiencing cladding perforation. Statistical analysis of the wear data indicates that the affected control rod assemblies are distributed across the entire radial region without a distinct distribution pattern. In the axial direction, however, the wear is predominantly concentrated in the upper nozzle region, the continuous guide section of the guide tube, and up to the area of the first guide plate. Another case of FIV-induced failure in slender nuclear components occurred at the No. 1 unit of the Taishan Nuclear Power Plant, which was required to shut down for maintenance after fuel rod failures resulted from abnormal vibration caused by FIV phenomenon [9, 10]. These observations indicate that FIV phenomenon can significantly affect the operational safety, service life, and structural integrity of slender nuclear components such as control rods and fuel rods. Specifically, the potential risks of control rod wear and fatigue cracks arising from these FIV issues may, under extreme conditions, delay the control rod drop time, thereby affecting nuclear safety. Therefore, during the initial design stage, it is essential to evaluate the impact of FIV phenomenon on reactor components within the complex, high-flow-velocity environment inside a pressurized water reactor (PWR) and to explore design improvements aimed at mitigating the effects of FIV on nuclear components.

Qiu et al. [7] performed transient fluid–structure interaction (FSI) simulations of a full-scale control rod guide tube (including 24 control rods, 6 guide cards, and a continuous guide section) using ANSYS (FLUENT for fluid, Mechanical for structure) with the k-omega SST turbulence model. Their results show that coolant discharge through guide tube square slots increases transverse velocity and induces vortices at the guide tube bottom, and that higher cross-flow velocity leads to larger control rod displacement due to FIV. This study considered only a single guide tube with a predefined coolant flow direction rather than flow conditions determined by actual in-core environments. Wu et al. [11] used a finite segment-based differential-algebraic equation (DAE) method to model control rod drop, capturing fluid flow distribution and rod motion simultaneously. Unlike traditional approaches limited to specific geometries, their method adapts to changes in structure and flow channels, improving design efficiency and reactor safety. However, the one-dimensional DAE approach relies on empirical pressure drop correlations and does not resolve the three-dimensional cross-flow, vortices, or lateral vibrations induced by flow holes. Kao et al. [12, 13] used STAR-CCM+ to perform CFD simulations of the core top and control rod guide tube regions, solving the RANS equations with a Realizable k-epsilon turbulence model. They found that small gaps and chamfers around the guide tube had a negligible impact on the predicted flow patterns and pressure drop (1% difference). Their results characterized the flow pattern and lateral flow in terms of the mass flow rates through various outlets, with approximately 81% of the total lateral flow exiting through the upper and lower windows. Xu et al. [14] developed a CFD model of the upper region of the AP1000 reactor vessel, including the core top, upper plenum, outlet nozzles, and part of the hot legs. Their results demonstrated that, regardless of the guide tube location, approximately two-thirds of the total flow exits through the upper and lower windows, highlighting the dominant role of these two openings. The study also showed that the lateral flow induced by the outlet nozzles significantly affects the flow distribution, with the flow rate through the upper window increasing when it is located closer to the outlet nozzle. While these studies have provided valuable insights into flow behavior and the effects of structural features, they did not propose specific strategies for optimizing the lateral flow within control rod guide tubes, nor did they fully account for three-dimensional cross-flow dynamics or realistic in-core boundary conditions. Addressing these limitations is essential for developing more effective flow optimization measures to mitigate flow-induced vibration wear.

In this research, a full scaled 2×2 control rod guide tube array model is employed to simulate the flow fields of the original and optimized configurations aimed at mitigating FIV. The optimization strategy involves modifying the dimensions of three types of flow holes on the upper core plate (the unobstructed flow hole, support column hole, and guide tube hole), as well as optimizing the design of the support column base. By integrating in-house experimental data with numerical simulations conducted using the commercial CFD software Fluent, this study validates the ability of the optimized design to improve flow distribution uniformity and suppress lateral flow in typical regions of the scaled model. Experimental results show that the optimized configuration significantly improves both the uniformity and intensity of flow distribution across the core upper plate. The measured flow distribution ratios for the three types of holes (the guide tube hole, the support column hole and the unobstructed flow hole) improved from 1.11:0.82:0.96 in the prototype to 0.97:0.98:1.08 in the optimized model. Numerical simulations further validate these findings, showing a consistent trend: the flow distribution ratios change from 1.17:0.75:0.90 to 0.98:0.96:1.07. The numerical results also demonstrate improvements in the flow patterns of key FIV excitation sources, lateral flow from the guide tube square groove outlet, the guide tube flange–core upper plate gap, and the upper nozzle region, before and after optimization. The mutual validation of experimental and simulation data provides consistent evidence that the optimized model reduces flow exchange among the three types of holes, decreasing lateral flow intensity at the upper nozzle region. It also suppresses both the mass flow entering the guide tube and the jet velocity from the gap between the guide tube flange and the core upper plate. These improvements in the flow field demonstrate that the optimized model can effectively decrease the lateral-flow caused excitation forces acting on the control rods, resulting in effective FIV suppression. This dual validation provides robust support for the new FIV-resistant 2×2 control rod guide tube array design.

2. Numerical Methodology

The present CFD study is based on the in-house experimental work of Mao et al. [8], who investigated the flow distribution and cross-flow characteristics in a 1:4 scaled 2×2 control rod guide tube array model. Figure 1 illustrates the key geometric features of the experimental model, including the core upper plate with three types of flow holes (two guide tube holes, one unobstructed flow hole, and one support column hole), the support column with an optimized base design, the control rod guide tubes, the upper nozzle, the control rods, and the three-section transparent casing (inlet, middle, and exit sections) with flange connections. The detailed dimensions of the flow holes and support column base are also indicated in the magnified view of the core upper plate.

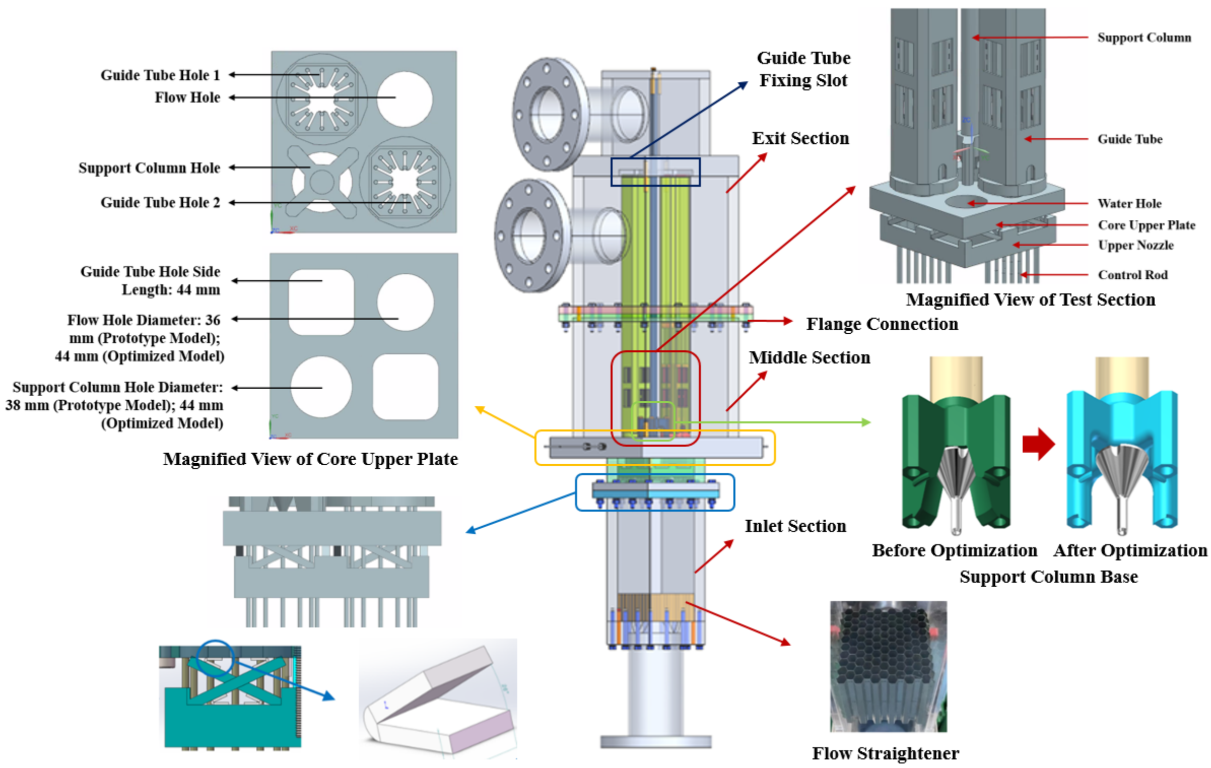

Figure 1: Schematic of Experimental Design

In the in-house experiments of Mao et al. [8], Pitot tube measurements and Particle Image Velocimetry (PIV) were combined to characterize the flow fields of both a prototype configuration and an optimized configuration. As shown in figure 2, the optimized configuration achieved a more uniform flow distribution across the core upper plate. The flow distribution ratios for the guide tube hole, the support column hole, and the unobstructed flow hole changed from 1.11, 0.82, and 0.96 in the prototype to 0.97, 0.98, and 1.08 in the optimized model, respectively. Additionally, the pressure drop of the optimized model was reduced to 0.83 times that of the prototype.

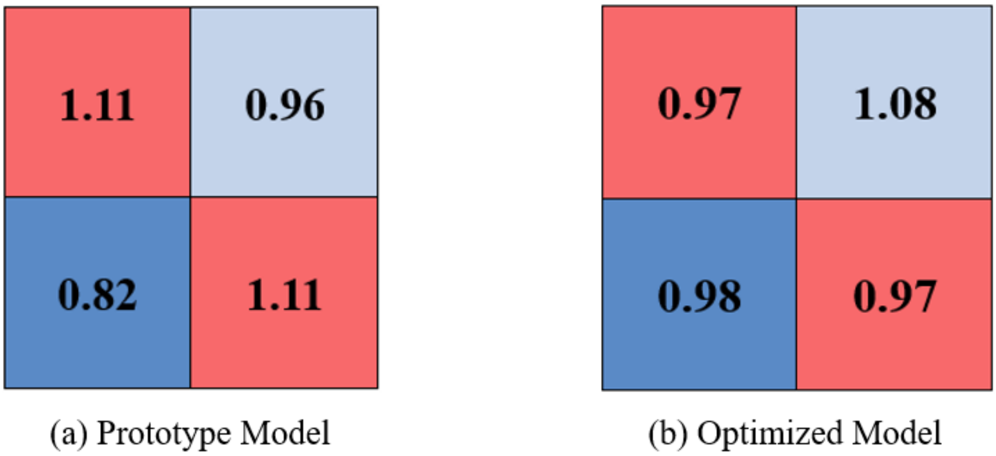

Figure 2: Flow Distribution Ratios on the Core Upper Plate in the Experiment: (a) Prototype Model, (b) Optimized Model

The numerical study employs the same 2×2 control rod guide tube array geometry as that used in the experiments of Mao et al. [8], at a 1:1 scale. The flow field is simulated using the commercial software FLUENT, where the Realizable k-epsilon model is employed for turbulence closure. The 2×2 control rod guide tube array model, as shown in Figure 3, consists of two guide tubes, two sets of control rods, one support column, the core support upper plate, and the upper nozzle. As observed from the top view of the upper plenum in Figure 4, this 2×2 control rod guide tube structural unit occupies approximately 80% of the core upper plate area (excluding the outermost flow hole region). Therefore, this 2×2 control rod guide tube array model is capable of effectively characterizing the typical flow field around the control rods in the upper plenum region. To improve the flow field and mitigate FIV phenomena within the control rod guide tube, optimizations are primarily applied to the core upper plate and the support column structure, as illustrated in Figures 5 and 6. Specifically, the flow hole in the core upper plate is enlarged from 144 mm to 176 mm, the support column hole is enlarged from 152 mm to 176 mm, and the support column base is redesigned as a streamlined structure.

Simulations are performed at a temperature of 321°C and a pressure of 15.5 MPa. Under these thermodynamic conditions, the coolant exhibits a density of 677.3 kg/m^3 and a dynamic viscosity of 8.0027×10^{-5} Pa·s. The numerical model of the fluid domain is presented in Figure 7. Based on the symmetry of the model, the numerical domain is taken as one-half of the 2×2 control rod guide tube array model, comprising one guide tube, one set of control rods, the support column, the flow hole, the core upper plate and the upper nozzle.

The boundary conditions of the computational model are illustrated in Figure 8, with a mass-flow inlet and a pressure outlet. Given that the computational domain constitutes one-half of the unit model, the mass flow rate at the inlet is correspondingly taken as half of that of the unit model, i.e., 200 kg/s. The outlet is set as a pressure outlet with a gauge pressure of 0 Pa.

A mesh independence study was conducted to determine the appropriate mesh resolution. The final mesh size selected for the prototype model is 82.4 million cells, while that for the optimized model is 82.31 million cells. The global and local meshes of the reference model are shown in Figure 9, and those of the optimized model are presented in Figure 10.

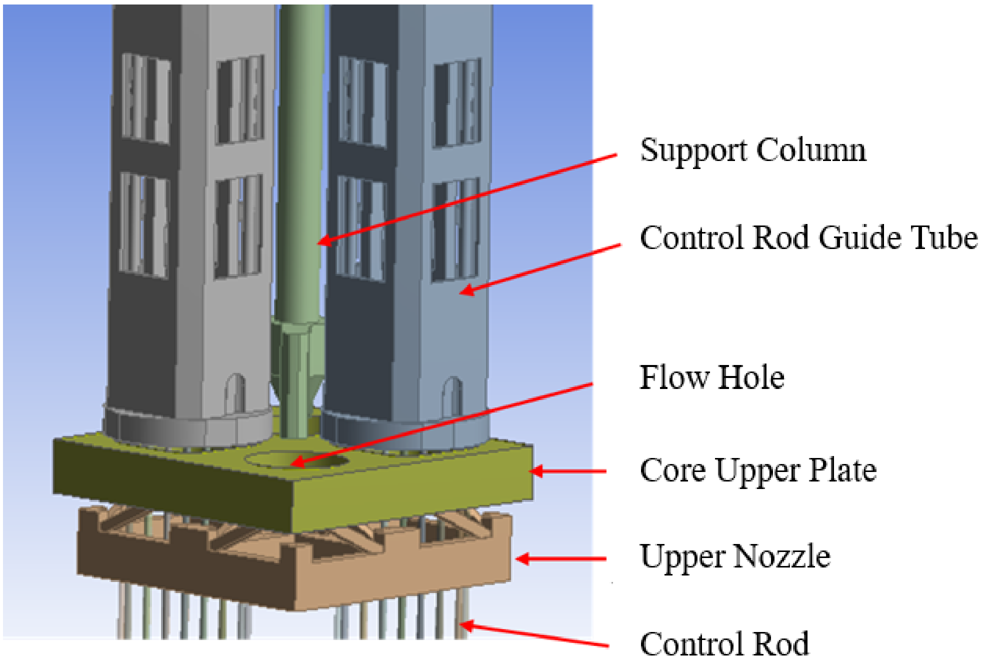

Figure 3: Schematic Diagram of the 2×2 Control Rod Guide Tube Array Model

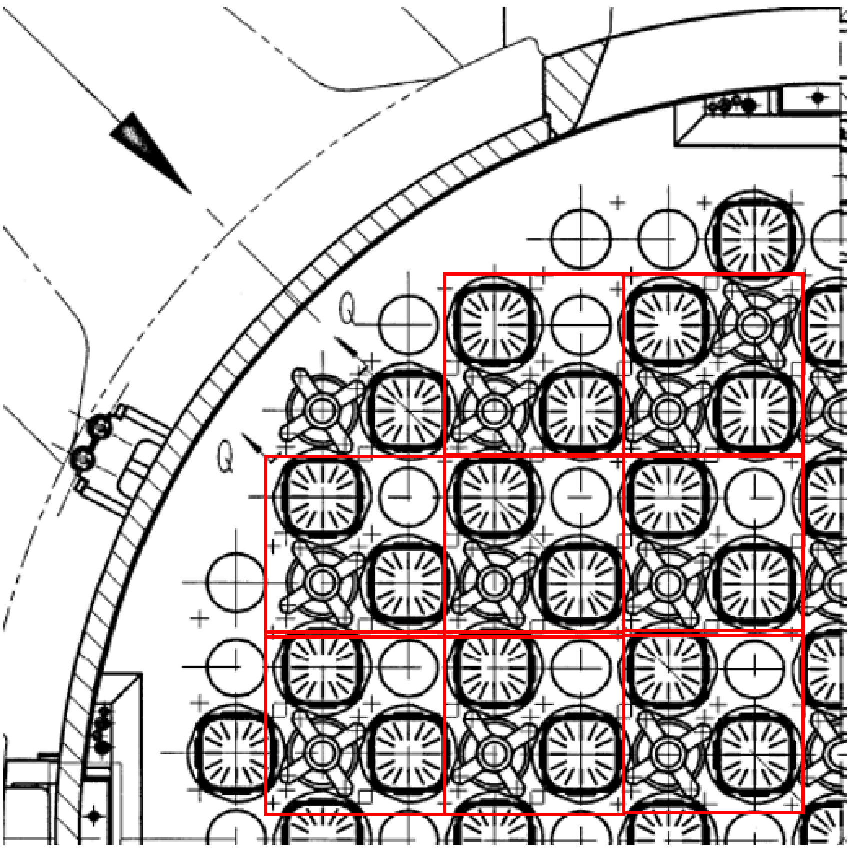

Figure 4: Top View Schematic of the Upper Plenum

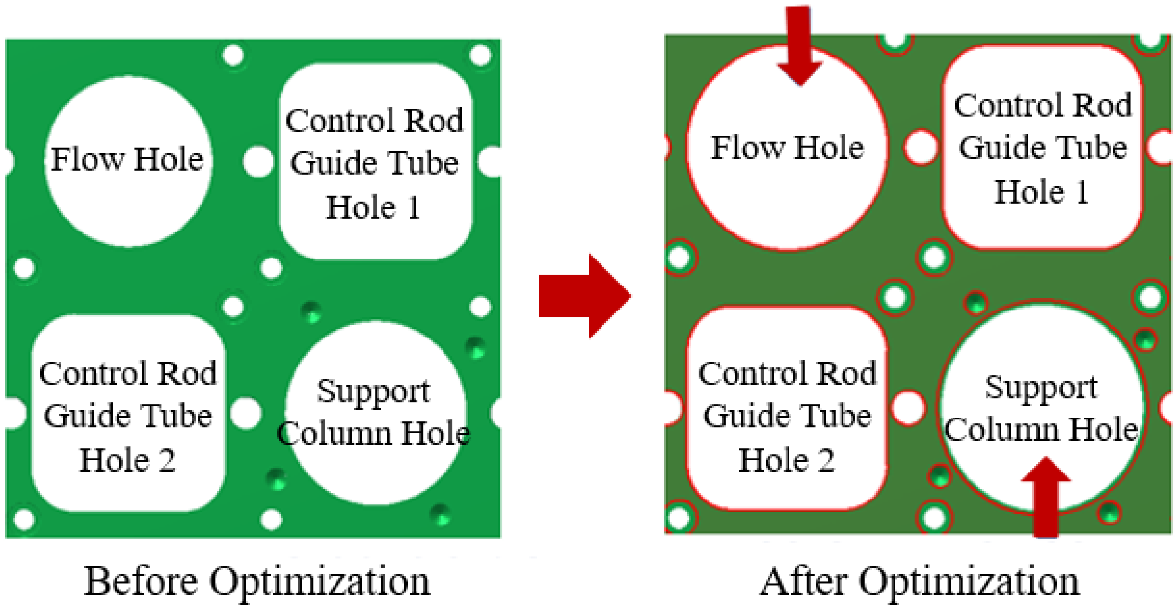

Figure 5: Schematic Diagram of the Core Upper Plate Optimization

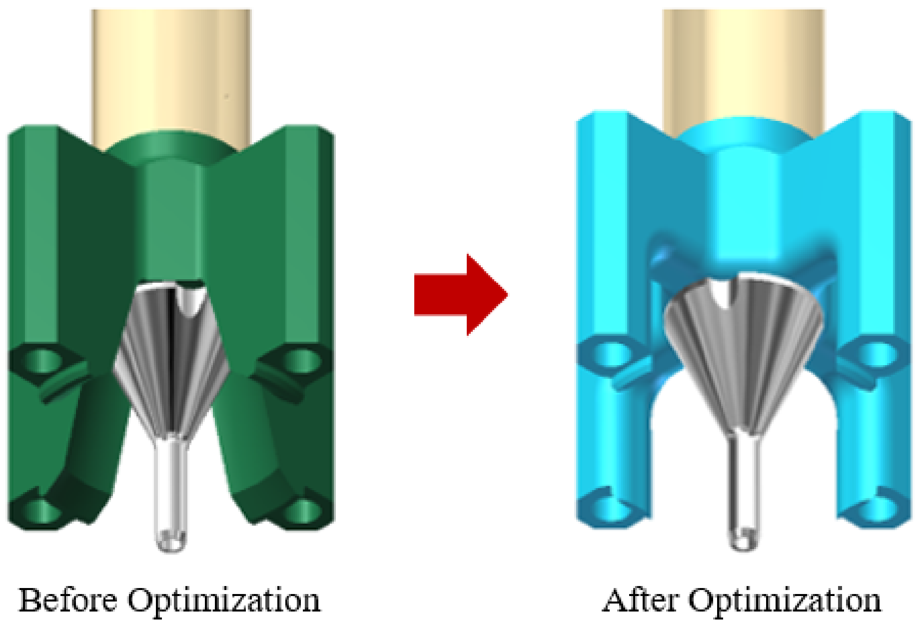

Figure 6: Schematic Diagram of the Support Column Base Optimization

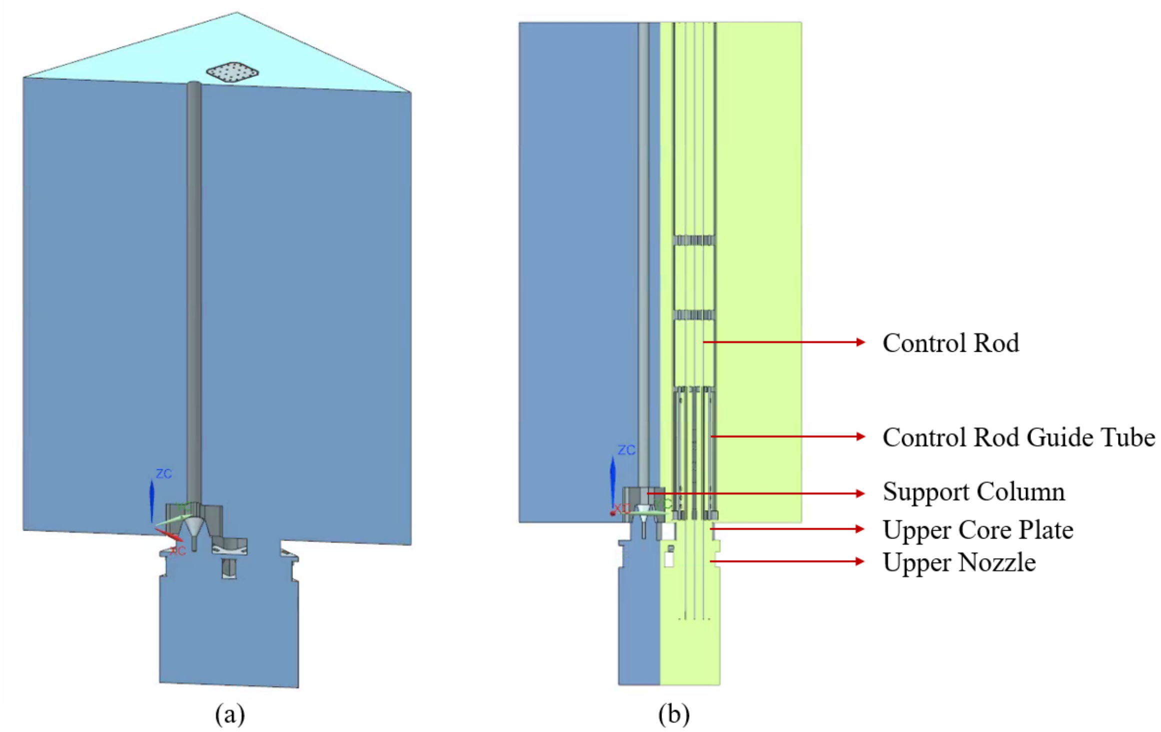

Figure 7: Schematic Diagram of the 2×2 Control Rod Guide Tube Array Numerical Model: (a) Overall View; (b) Sectional View

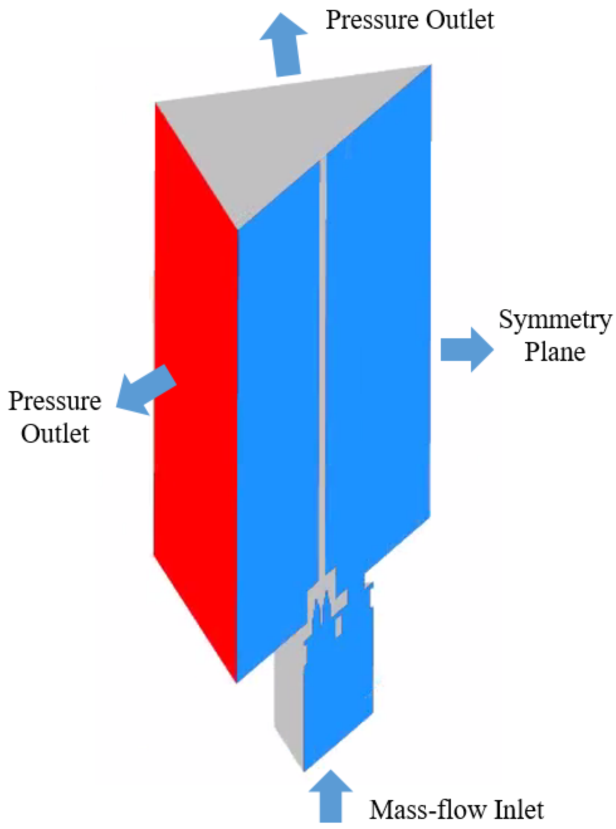

Figure 8: Boundary Condition Setup for the Numerical Model

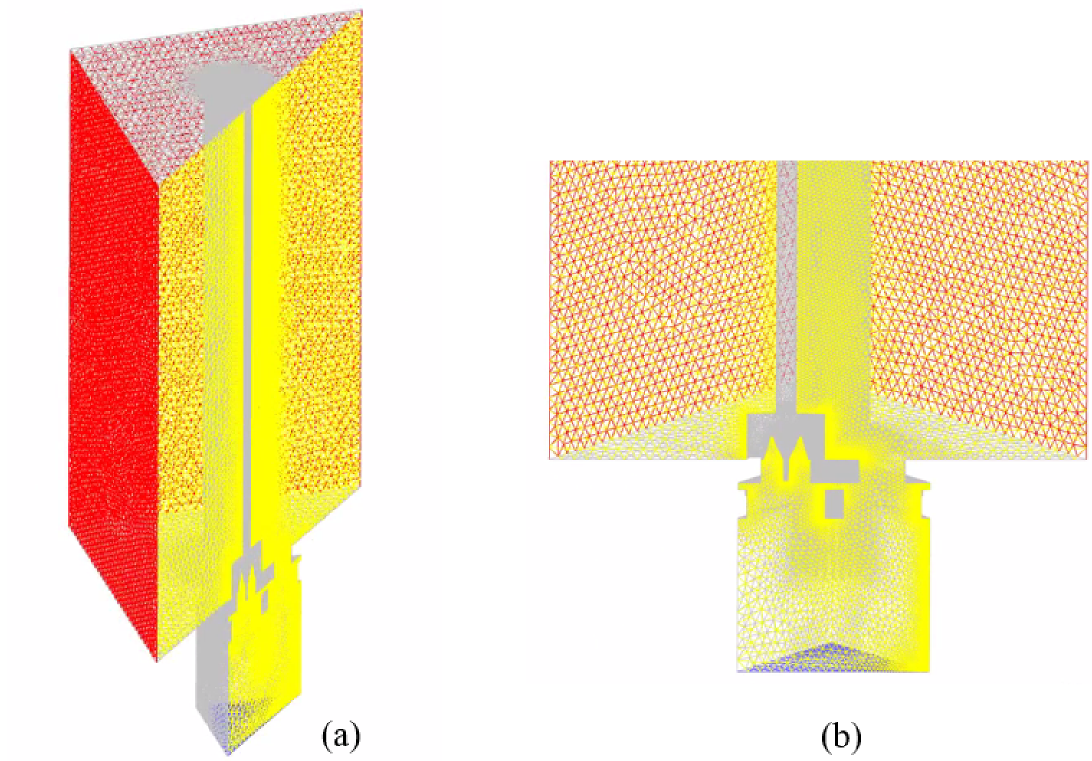

Figure 9: (a) Global Mesh of the Prototype Model; (b) Local Mesh of the Prototype Model

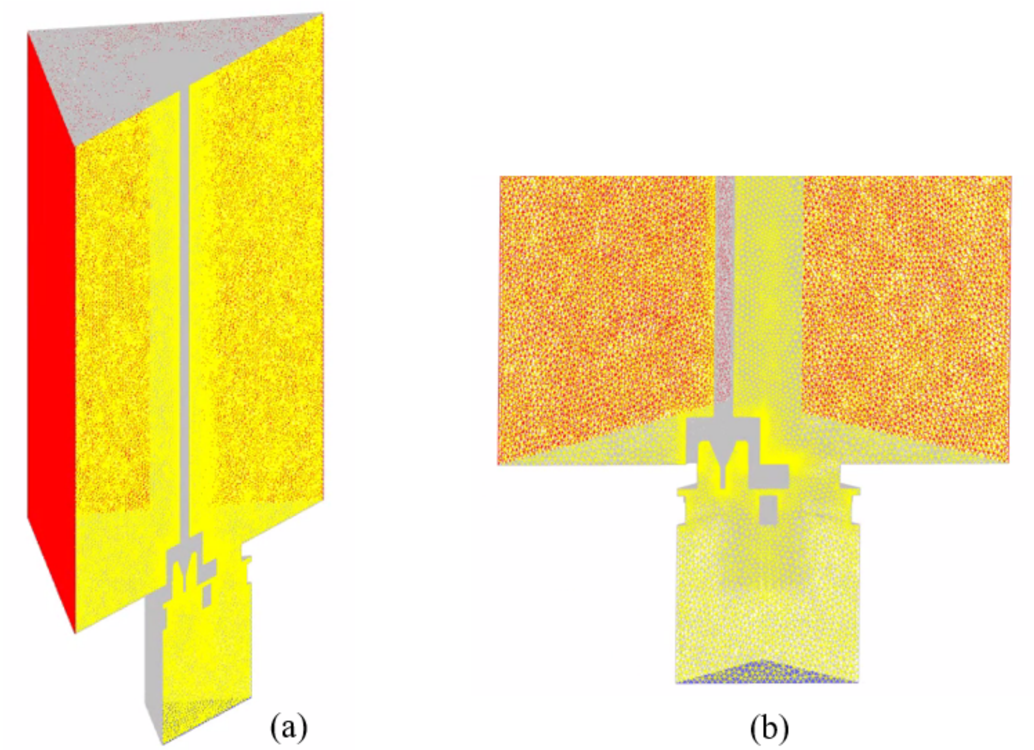

Figure 10: (a) Global Mesh of the Optimized Model; (b) Local Mesh of the Optimized Model

3. Results and Discussion

This study established a 2×2 control rod guide tube array model of the nuclear reactor to investigate the wear phenomenon caused by FIV phenomenon of control rods. The research focused on the lateral flow of coolant around the control rods in the upper plenum region, with particular emphasis on the lateral flow intensity at key locations including the guide tube upper square slot outlet, lower square slot outlet, guide tube flange-core upper plate gap, and upper nozzle. An optimized model of the nuclear upper internal structure was proposed, aiming to reduce the lateral flow velocity of the coolant around the control rods in the aforementioned regions of the upper plenum.

The contour of lateral flow velocity in the cross-section of the 2×2 control rod guide tube array model, obtained through numerical simulation, is shown in Figure 11. It can be observed that the lateral flow intensities at the upper square slot outlet, lower square slot outlet, guide tube flange-plate gap, and upper nozzle region are all reduced in the optimized model compared to the optimized model. Figure 12 presents the contour of lateral flow velocity in the guide tube flange-plate gap. Although the optimized model exhibits a localized increase in flow velocity in certain regions, the overall jet intensity, including jet distance and coverage, is significantly improved. Figure 13 indicates the specific elevations of the monitoring sections shown in Figures 14 through 16. As illustrated in Figures 14, 15, and 16, which respectively present the lateral flow velocity contour at the core upper plate section, the lateral flow velocity vector diagram in the upper nozzle region, and the lateral flow velocity vector diagram at the upper square slot outlet, the lateral flow velocities at these critical locations are significantly lower in the optimized model than in the optimized model.

Figure 17 illustrates the flow distribution on the core upper plate before and after the model optimization, which involved modifications to both the core upper plate and the support column base. The flow distribution ratio among the guide tube hole, the support column hole and the unobstructed flow hole changes from 1.17:0.75:0.90 to 0.98:0.96:1.07. This is consistent with the experimental results, where the ratio shifted from 1.11:0.82:0.96 in the prototype to 0.97:0.98:1.08 in the optimized model, demonstrating a consistent improvement in flow distribution. This improvement reduces the generation of cross-flow, thereby helping to mitigate FIV phenomenon.

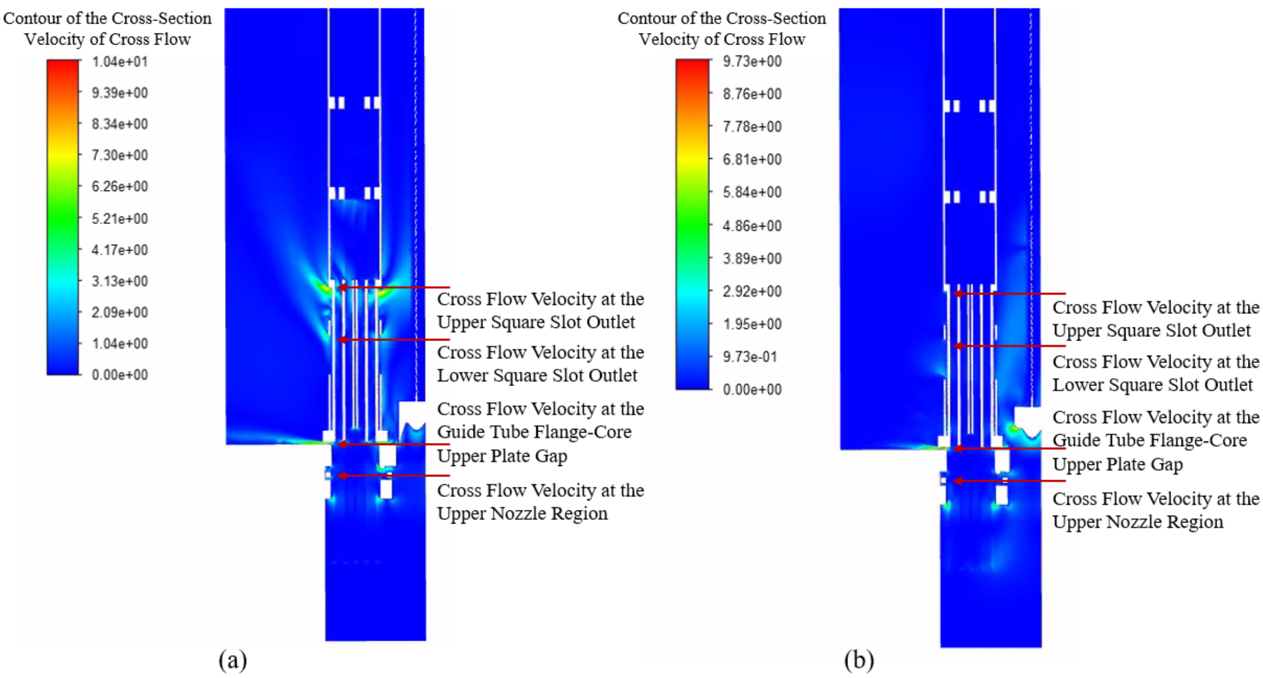

Figure 11: Cross Flow Velocity Contours in the Cross-Section of the Control Rod Guide Tube Array Model: (a) Prototype Model; (b) Optimized Model

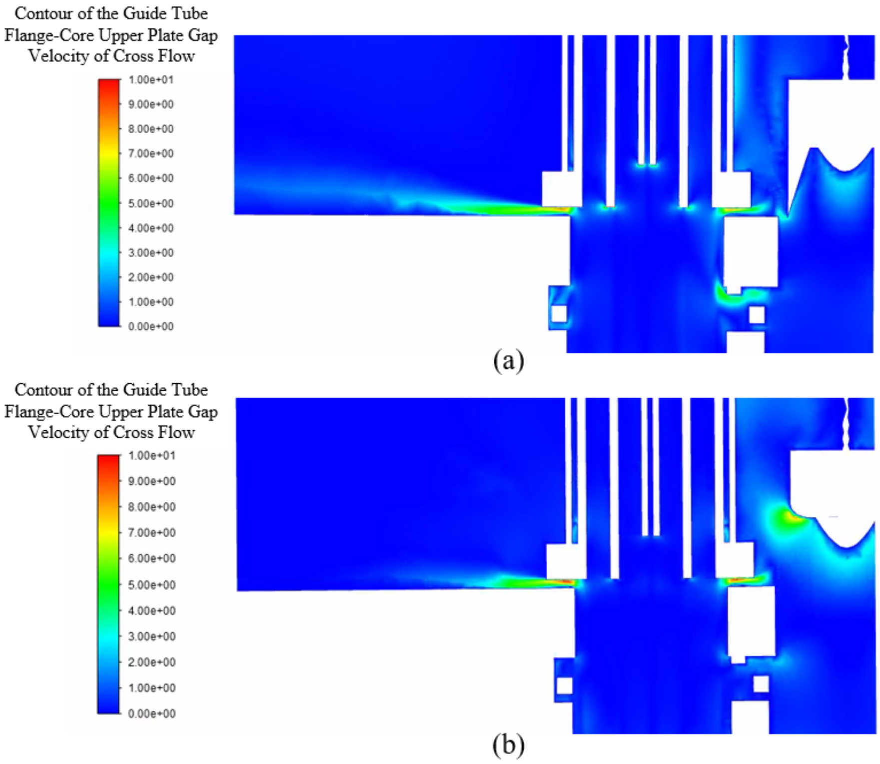

Figure 12: Cross Flow Velocity Contours at the Guide Tube Flange-Core Upper Plate Gap: (a) Prototype Model; (b) Optimized Model

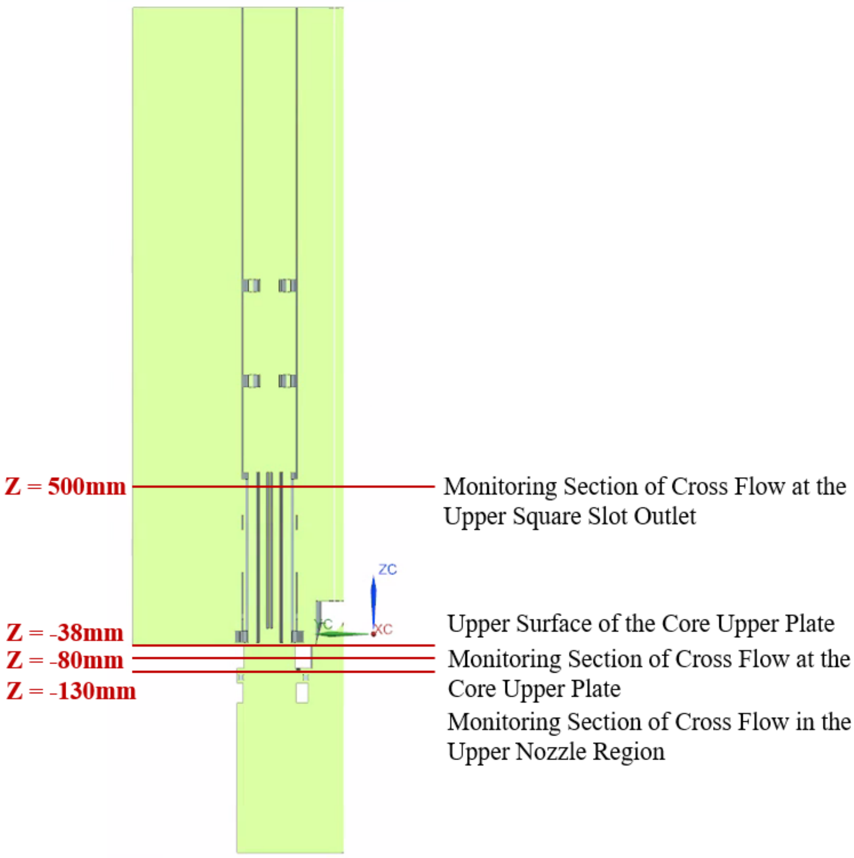

Figure 13: Schematic Diagram of Monitoring Section Elevations

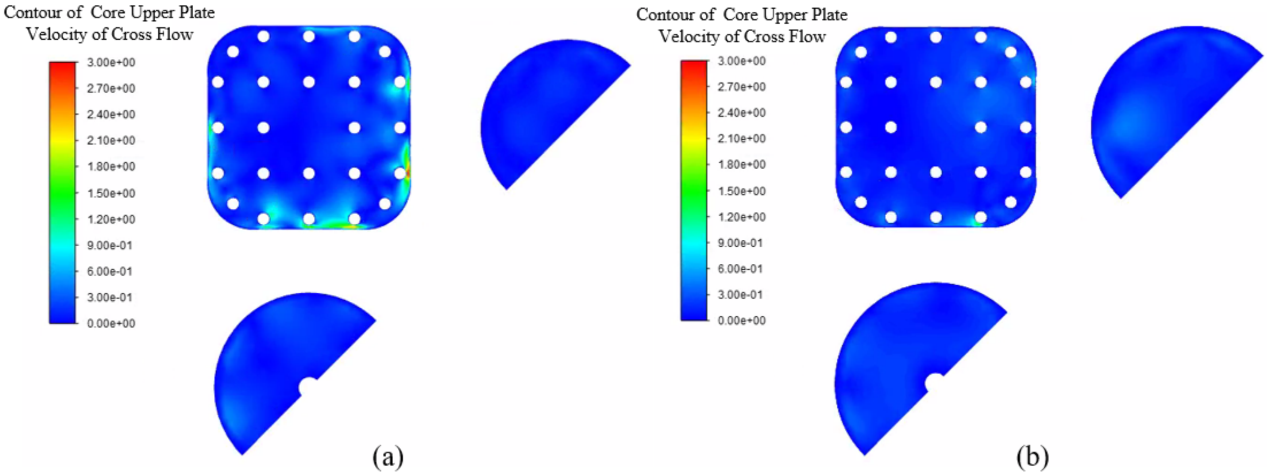

Figure 14: Cross Flow Velocity Contours at the Core Upper Plate: (a) Prototype Model; (b) Optimized Model

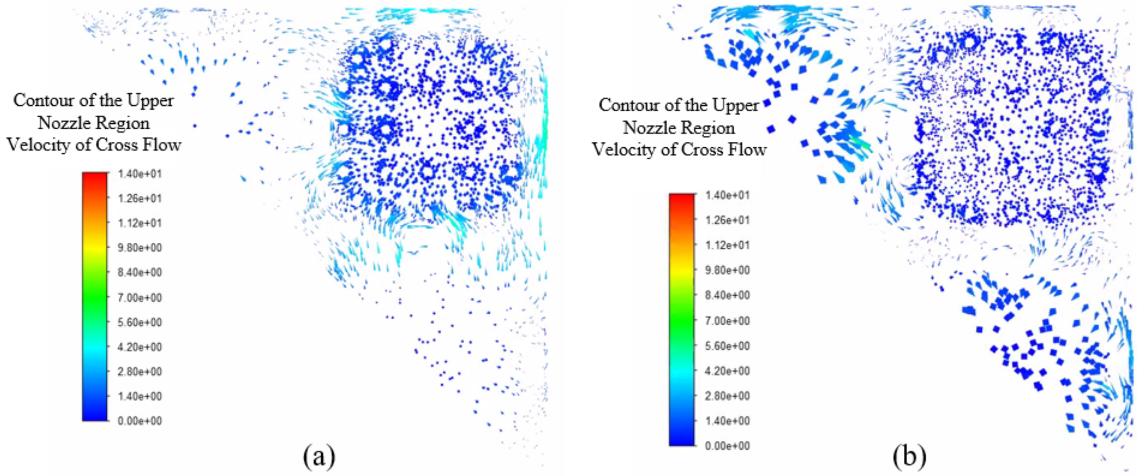

Figure 15: Cross Flow Velocity Contours at the Upper Nozzle Region: (a) Prototype Model; (b) Optimized Model

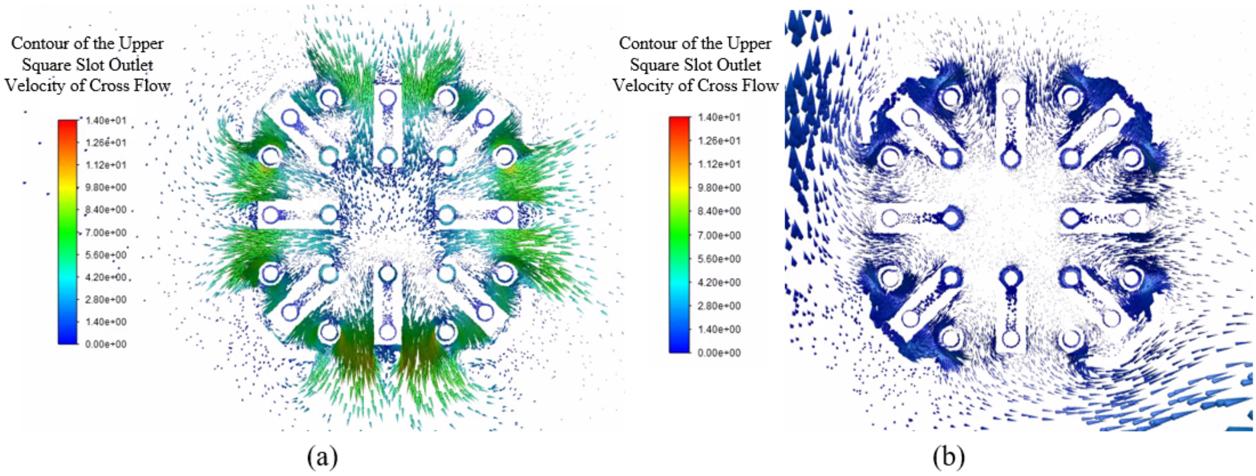

Figure 16: Cross Flow Velocity Contours at the Upper Square Slot Outlet: (a) Prototype Model; (b) Optimized Model

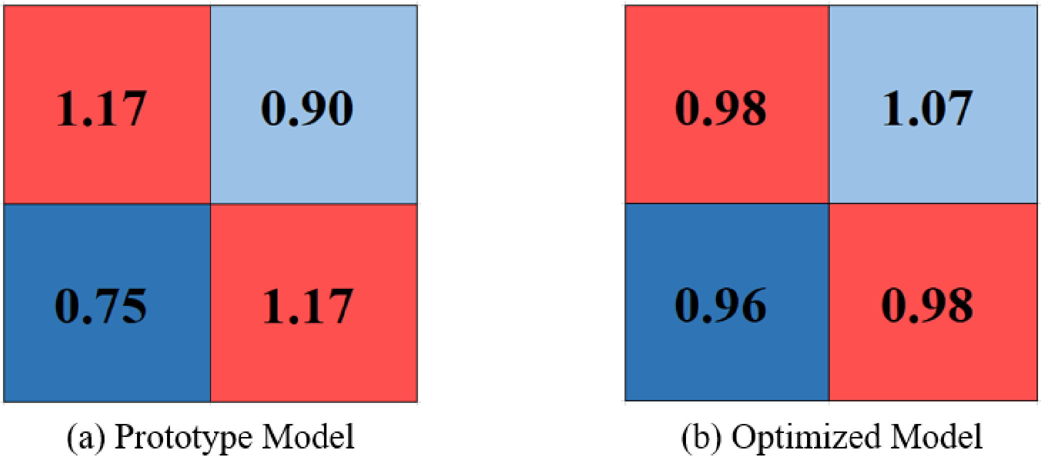

Figure 17: Flow Distribution Ratios on the Core Upper Plate in the Simulation: (a) Prototype Model, (b) Optimized Model

4. Conclusion

This study numerically investigates the prototype and an optimized 2×2 control rod guide tube model, sing the commercial CFD software FLUENT, focusing on the influence of the core upper plate configuration and the support column base design on flow distribution, particularly around the 2×2 control rod guide tube and adjacent nuclear components. The numerical results reveal that the optimized model significantly improves flow distribution uniformity across the core upper plate, with the flow distribution ratios changing from 1.17:0.75:0.90 in the prototype to 0.98:0.96:1.07 in the optimized model. This numerical trend is consistent with experimental observations (which showed a shift from 1.11:0.82:0.96 to 0.97:0.98:1.08), thereby validating the reliability of the present simulations. Meanwhile, compared with the prototype, the optimized model exhibits reduced lateral flow intensities at the upper square slot outlet, lower square slot outlet, guide tube flange–plate gap, and upper nozzle region. These improvements in the flow field within the upper internal structure (UIS) contribute to mitigating FIV of the control rods.

References

[1] D. De Santis, S. Kottapalli, and A. Shams. Numerical simulations of rod assembly vibration induced by turbulent axial flows. *Nuclear Engineering and Design*, 335:94–105, 2018.
[2] A. Cioncolini, J. Silva-Leon, D. Cooper, et al. Axial-flow-induced vibration experiments on cantilevered rods for nuclear reactor applications. *Nuclear Engineering and Design*, 338:102–118, 2018.
[3] A. Cioncolini, S. Zhang, R.A.M. Nabawy, et al. Experiments on axial-flow-induced vibration of a free-clamped/clamped-free rod for light-water nuclear reactor applications. *Annals of Nuclear Energy*, 190:109900, 2023.
[4] D. De Santis and A. Shams. Numerical study of flow-induced vibration of fuel rods. *Nuclear Engineering and Design*, 361:110547, 2020.
[5] H. Li, A. Cioncolini, S. Zhang, et al. Tip shape effects on the axial-flow-induced vibration of a cantilever rod. *Journal of Fluids and Structures*, 127:104132, 2024.
[6] W. Mao, H. Iacovides, A. Cioncolini, et al. A validated numerical methodology for flow-induced vibration of a semi-spherical end cantilever rod in axial flow. *Physics of Fluids*, 36:075155, 2024.
[7] H. Qiu, C. Zhang, M. Wang, et al. Analysis of the influence of bottom flow holes in control rod guide tubes on flow field and control rod displacement. *Annals of Nuclear Energy*, 211:111028, 2025.
[8] W. Mao, X. Sui, J. Wu, et al. Experimental study on flow distribution optimization for mitigating

flow-induced vibration in a 2 × 2 control rod guide tube array. *Annals of Nuclear Energy*, 236:112416, 2026.

[9] A. Muhamad Pauzi, W. Mao, A. Cioncolini, et al. Best practices for axial flow-induced vibration (FIV) simulation in nuclear applications. *Journal of Nuclear Engineering*, 7(3), 2025.

[10] S. Dai, X. Li, Y. Zhang, et al. Theoretical model of fluid-structure interaction and prediction of fluidelastic instability for wire-wrapped fuel rods in axial flow. *Nuclear Engineering and Design*, 449:114786, 2026.

[11] X. Wu, L. Zhao, F. Wu, et al. Dynamic modeling of the control rod drop problem considering changes of structure and flow channel. *Annals of Nuclear Energy*, 233:112273, 2026.

[12] M. Kao, C. Wu, C. Chieng, et al. CFD analysis of PWR reactor vessel upper plenum sections: flow simulation in control rod guide tubes. In *Proceedings of the 18th International Conference on Nuclear Engineering (ICONE18-29466)*, Xi'an, China, May 2010.

[13] M. Kao, C. Wu, C. Chieng, et al. CFD analysis of PWR core top and reactor vessel upper plenum internal subdomain models. *Nuclear Engineering and Design*, 241:4181–4193, 2011.

[14] Y. Xu, M. Conner, K. Yuan, et al. Study of impact of the AP1000® reactor vessel upper internals design on fuel performance. *Nuclear Engineering and Design*, 252:128–134, 2012.