



ARTICLE

Hydrodynamic Forces on Brush Wires in High-Speed Liquid-Cooled Slip Rings

Zhijun Guo, Jingwei Liu*, You Fu and Jiaxin Yang

AVIC Beijing Precision Engineering Institute for Aircraft Industry, Beijing, China

*Corresponding Author: Jingwei Liu. Email: liujw094@avic.com

Received: 28 June 2026; Accepted: 28 September 2026; Published: 19 October 2026

ABSTRACT: This study investigates the fluid forces acting on brush wires in high-speed liquid-cooled slip rings by combining computational fluid dynamics (CFD) simulations with theoretical modeling. The study first characterizes the two-dimensional (2D) and three-dimensional (3D) flow fields within a rotating annular domain representative of a liquid-cooled slip ring, and then uses a simplified single-wire model to quantify the hydrodynamic forces acting on a rigid brush wire over a range of rotational speeds. The results show that the normal pressure force becomes the dominant contribution to the total fluid force at high rotational speeds. Comparisons among different modeling approaches further demonstrate that 2D and axisymmetric models can overestimate the core plateau velocity by up to approximately 10% relative to finite-length 3D simulations, highlighting the importance of three-dimensional effects in predicting the flow field. Based on these findings, preliminary guidance is provided for selecting the appropriate brush preload under high-speed liquid-cooled operating conditions.

KEYWORDS: Fluid cooling; brush wire; flow characteristics; force coupling mechanism; CFD simulation

1 Introduction

The continuous accumulation of frictional heat generated during the high-speed operation of slip rings poses a critical bottleneck, severely challenging the reliability and operational lifespan of the entire system [1]. Wu et al. [2] provided a detailed discussion of the mechanisms by which environmental factors such as high temperature, humidity, corrosive gases, and vacuum influence the tribological properties of slip ring materials. Consequently, integrating an effective fluid cooling system has become an indispensable measure to maintain thermal stability and ensure consistent performance [3]. However, this necessary thermal management strategy fundamentally alters the operating environment. As explored by Zhang et al. [4] in the context of friction dynamics between brush wires and V-grooves, dry or conventional contact mechanisms are well-documented; yet, the introduction of a fluid coolant transforms the operating environment, introducing variable frictional interactions and precipitating several complex, poorly understood scenarios. Specifically, the intrusion of the fluid medium introduces additional hydrodynamic forces on the brush wire, which may alter the effective contact load at the brush-ring interface [5]. In this study, the fluid is treated as a single-phase incompressible Newtonian fluid, and the brush wire is assumed to be rigid; therefore, multiphase effects and elastic contact deformation are not considered.

The flow of a fluid in the annular gap between rotating and stationary concentric cylinders, commonly referred to as Taylor-Couette flow, has been extensively studied. At low rotational speeds, the flow remains laminar and is described by the classical analytical solution for Couette flow. As the rotational speed

increases, the flow undergoes a sequence of instabilities, beginning with the onset of Taylor vortices and eventually transitioning to fully turbulent flow at sufficiently high Reynolds numbers [6,7]. For liquid-cooled slip rings, the annular gap is typically narrow and operates at high rotational speeds, placing the flow well into the turbulent regime. Previous studies on high-speed rotating annuli have shown that end-wall effects and secondary flows significantly influence the velocity distribution and wall shear stress [8,9]. However, the specific interaction between such rotating annular flows and the brush wires used in slip rings has received limited attention.

Curvature-induced secondary flows in curved and serpentine channels have been shown to interact with swirl and significantly modify the velocity and temperature fields, while heat transfer enhancement in such configurations is governed by a trade-off between improved heat transport and increased irreversible losses [10]. Recent studies have also shown that calibrating the empirical coefficients of the SST (the shear stress transport) $k-\omega$ model via artificial neural network-particle swarm optimization (ANN-PSO) surrogate optimization can reduce deviations from LES to within 3% at moderate computational cost [11], which is relevant to the rotating annular flow considered here. Prior research has attempted to address thermal management through numerical approaches. For instance, Bin [12] utilized computational fluid dynamics (CFD) via ANSYS to simulate the temperature field of slip ring carbon brushes and ring structures, primarily focusing on macroscopic thermal performance and its impact on overall system efficiency. Taylor et al. [13] obtained the temperature field and leakage rates by using combined finite element analysis (FEA)-computational fluid dynamics (CFD) analysis and the anisotropic heat transfer theory. However, these studies predominantly treat the fluid as a passive heat sink, neglecting the intricate hydrodynamic interference it imposes on the microscopic contact pairs. To date, current research still exhibits a significant knowledge gap regarding the multiphysics coupling characteristics within rotating fluid domains. There is a particular lack of understanding concerning the influence patterns of diverse wall boundary conditions and working fluid thermophysical properties on localized flow behaviors.

To bridge this critical gap, this study proposes a comprehensive methodology that synergistically integrates CFD numerical simulation with theoretical modeling. Initially, through the analysis of typical benchmark cases, the research meticulously reveals characteristic phenomena such as vortex evolution and velocity distribution within 2D and 3D rotating fluid domains. This step rigorously verifies the applicability and accuracy of the theoretical model when applied to various working media under rotational effects. Furthermore, moving beyond macroscopic thermal analysis, a simplified single-wire model with a rigid brush wire is established to quantify the fluid forces acting on the wire under different rotational speeds. This model does not account for elastic deformation, friction-coefficient variation, or thermal effects. The results provide a quantitative basis for assessing the influence of hydrodynamic forces on the effective contact load, which is relevant for the preliminary selection of brush preload in liquid-cooled high-speed slip rings.

2 Flow Characteristic Analysis within the Fluid Domain

All simulations were performed using ANSYS Fluent 2023 R2 with the pressure-based solver and a steady-state formulation. For cases with $Re < 10^3$, the flow was treated as laminar; for $Re \geq 10^3$, the SST $k-\omega$ turbulence model was used. The inner cylinder wall was specified as a rotating no-slip wall via the moving reference frame (MRF) approach, while the outer wall and the brush wire surface were set as stationary no-slip walls.

Pressure-velocity coupling was handled using the Coupled scheme, with second-order upwind discretization for momentum and PRESTO! for pressure interpolation. Convergence was considered achieved

when scaled residuals dropped below 10^{-6} (baseline cases) or 10^{-4} (high-swirl cases), and when the monitored wall torque and fluid forces remained stable within 0.1% over 500 iterations.

The 2D domain was discretized using a structured O-grid mesh with approximately 52,000 quadrilateral cells, with near-wall refinement ensuring $y^+ < 1$. The 3D mesh was generated by extruding the 2D mesh over the 36 mm axial length with 120 grid points, resulting in approximately 2.4 million hexahedral cells. Mesh independence was assessed for both the 2D and 3D models. For the 2D domain, three mesh resolutions were tested: approximately 26,000, 52,000, and 104,000 quadrilateral cells. The resulting inner-wall torque at the benchmark condition differed by less than 1.5% between the medium and fine meshes, while the velocity profile at mid-gap changed by less than 0.5%. For the 3D domain, three grid levels were generated by extruding the corresponding 2D meshes with 60, 120, and 240 axial nodes (approximately 0.6, 2.4, and 9.6 million hexahedral cells, respectively). At a representative high-speed condition ($\omega = 1000$ rad/s), the difference in the predicted total fluid force on the brush wire between the medium and fine meshes was less than 2%, and the plateau velocity ratio $V/V_{\max}$ changed by less than 1%. Based on these results, the medium meshes (52,000 cells for 2D; 2.4 million cells for 3D) were selected for all subsequent simulations as a compromise between accuracy and computational cost.

Pressure and shear forces on the brush wire were obtained by direct integration of static pressure and wall shear stress over the wire surface using the built-in force report tools in Fluent. A summary of the boundary conditions used in the 2D baseline, 2D/3D variable-speed, and single-wire model cases is provided in Table 1.

Table 1: Summary of boundary conditions.

Boundary/Zone	2D Baseline	2D/3D Variable Speed	Single-Wire Model
Inner cylinder wall	Rotating no-slip, $\omega = 1$ rad/s	Rotating no-slip, $\omega = 1\text{--}4000$ rad/s	Rotating no-slip, $\omega = 1\text{--}20,000$ rad/s
Outer cylinder wall	Stationary no-slip	Stationary no-slip	Stationary no-slip
Brush wire surface	-	-	Stationary no-slip
Axial boundaries (2D)	Periodic	Periodic	-
Axial boundaries (3D)	-	Symmetry planes	Symmetry planes
Fluid	Gas, $\rho = 1$ kg/m ³ , $\mu = 2 \times 10^{-4}$ Pa·s	Insulating liquid, $\rho = 1200$ kg/m ³ , $\mu = 0.042$ Pa·s	Insulating liquid, $\rho = 1200$ kg/m ³ , $\mu = 0.042$ Pa·s

All simulations were performed using the steady-state RANS formulation, which is appropriate for statistically stationary rotating flows. Accordingly, no time-step dependence exists; for any future transient simulations (e.g., LES or URANS), a time-step independence study would be required.

The rotational speed range investigated here spans several flow regimes. At the lowest speed ($\omega = 1$ rad/s), the gap Reynolds number is approximately 1.45×10^2 , which already exceeds the critical value for Taylor vortex onset. As the speed increases, the flow rapidly transitions into fully turbulent conditions: at $\omega = 1000$ rad/s, $\text{Reg} \approx 1.45 \times 10^5$, and at the maximum speed of 20,000 rad/s, $\text{Reg} \approx 2.9 \times 10^6$. These estimates confirm that a laminar flow assumption is not appropriate for the majority of the cases and that turbulence modeling is essential for accurately predicting the fluid forces on the brush wire.

2.1 Governing Equations and Turbulence Modeling

The flow of the incompressible Newtonian coolant is governed by the continuity and Reynolds-averaged Navier–Stokes (RANS) equations. For a rotating frame with angular velocity ω , the steady-state equations are:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\rho(\mathbf{u} \cdot \nabla \mathbf{u} + 2\omega \times \mathbf{u} + \omega \times (\omega \times \mathbf{r})) = -\nabla p + \nabla \cdot [(\mu + \mu_t)(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] \quad (2)$$

where $\mathbf{u}$ is the velocity vector, p is the static pressure, ρ and μ are the fluid density and dynamic viscosity, and μ_t is the turbulent eddy viscosity. The second and third terms on the left-hand side of Eq. (2) represent the Coriolis and centrifugal forces arising from the moving reference frame (MRF) formulation. For turbulent cases ($\text{Re} \geq 10^3$), the SST k - ω model is employed. The transport equations for the turbulent kinetic energy k and the specific dissipation rate ω are:

$$\frac{\partial}{\partial x_j}(\rho k u_j) = \tilde{P}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] \quad (3)$$

$$\frac{\partial}{\partial x_j}(\rho \omega u_j) = \alpha \frac{\omega}{k} \tilde{P}_k - \beta \rho \omega^2 + \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \rho \sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (4)$$

where $\tilde{P}_k$ is the production term of turbulent kinetic energy, F_1 is the blending function, and the model constants are $\beta^* = 0.09$, $\alpha = 5/9$, $\beta = 3/40$, $\sigma_k = 0.5$, $\sigma_\omega = 0.5$, and $\sigma_{\omega 2} = 0.856$. The turbulent eddy viscosity is computed as

$$\mu_t = \frac{\rho a_1 k}{\max(a_1 \omega, S F_2)} \quad (5)$$

with $a_1 = 0.31$, S the invariant measure of the strain rate, and F_2 the second blending function.

2.2 Comparative Analysis of Typical Fluid Domain Characteristics

The typical rotating flow benchmark case VMFL001: Flow Between Rotating and Stationary Concentric Cylinders from the ANSYS Fluid Dynamics Verification Manual was selected. In this case, the inner wall radius is 17.8 mm, the outer wall radius is 46.28 mm, the inner cylinder boundary rotates at a constant angular velocity of 1 rad/s, and the outer cylinder boundary remains stationary (Fig. 1, schematic diagram).

For the rotating fluid domain, the momentum equation in the angular direction is [14,15]:

$$\frac{d^2 u_\theta}{dr^2} + \frac{d}{dr} \left(\frac{u_\theta}{r} \right) = 0 \quad (6)$$

The solution for the velocity in the angular direction has the form:

$$u_\theta = C_1 r + \frac{C_2}{r} \quad (7)$$

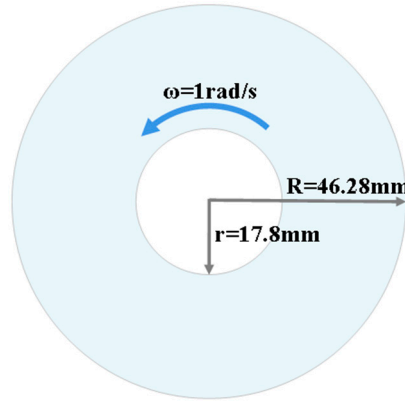


Figure 1: Schematic diagram of the benchmark case structure.

The boundary conditions are $u_\theta(r_0) = r_0\omega_0$ at the inner wall and $u_\theta(r_1) = r_1\omega_1$ at the outer wall. Applying the defined boundary conditions yields the analytical solution:

$$u_\theta = r_0\omega_0 \frac{\frac{r_1}{r} - \frac{r_1}{r_0}}{\frac{r_1}{r_0} - \frac{r_0}{r_1}} + r_1\omega_1 \frac{\frac{r}{r_0} - \frac{r}{r_1}}{\frac{r_1}{r_0} - \frac{r_0}{r_1}} \quad (8)$$

where, r_1 and ω_1 denote the radius and angular velocity of the outer cylinder boundary, respectively, r_0 and ω_0 denote those of the inner cylinder boundary. The wall shear stress τ is given by Newton's law of viscosity:

$$\tau_{r\theta} = \mu(\partial u_\theta / \partial r - u_\theta / r) \quad (9)$$

Differentiating the analytical expression for velocity gives:

$$\frac{\partial u_\theta}{\partial r} = -\frac{\omega r_i r_o}{r_o - r_i} \left(\frac{1}{r^2} + \frac{1}{r_o^2} \right) \quad (10)$$

Thus, at the inner wall $r = r_i$, the wall shear stress is:

$$\tau = \mu \left(-\frac{\omega r_i r_o}{r_o - r_i} \left(\frac{1}{r_i^2} + \frac{1}{r_o^2} \right) \right) \quad (11)$$

Integrating the shear stress over the wall surface yields the torque T :

$$T = \mu \left(-\frac{\omega r_i r_o}{r_o - r_i} \left(\frac{1}{r_i^2} + \frac{1}{r_o^2} \right) \right) \cdot A \cdot r_i = \frac{4\pi\mu\omega r_i^2}{(1 - (r_i/r_o)^2)} \quad (12)$$

Substituting the boundary conditions gives the analytical solution for the circumferential velocity distribution:

$$u_\theta = 0.008 \times \left(\frac{46.28}{1000 \times r} - \frac{1000 \times r}{46.28} \right) \quad (13)$$

The model was constructed in ICEM using the parameters from this case. The fluid material was set as “gas” with a density of 1 kg/m^3 and a dynamic viscosity of $0.0002 \text{ kg/(m}\cdot\text{s)}$.

The numerical simulation yielded results for the 2D fluid domain circumferential velocity distribution and inner wall torque. Fig. 2 compares the simulated circumferential velocity distribution with the analytical solution. The comparison shows that the numerical solution matches the analytical solution with a radial distribution conformity of 99.1%. The maximum relative error in circumferential velocity is less than 0.8%, primarily located near the wall region. The numerical solution for the inner wall torque is 8.97×10^{-7} N·m, with an error of 4.06% compared to the theoretical value of 9.35×10^{-7} N·m, meeting accuracy requirements.

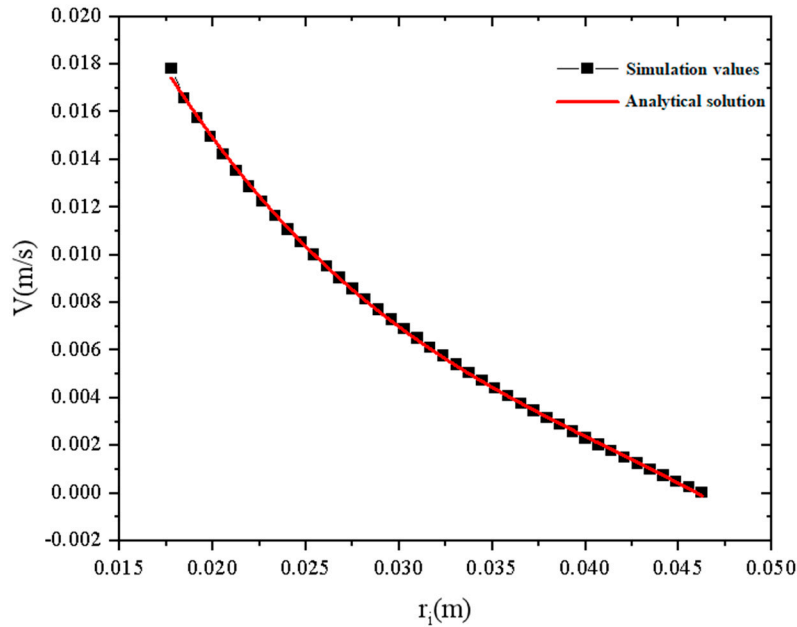


Figure 2: Comparison between the simulated circumferential velocity distribution and the analytical solution for the benchmark case.

To validate the numerical approach under high-speed turbulent conditions, the simulated torque was compared with the empirical correlation of Wendt [16] for turbulent Taylor-Couette flow between rotating and stationary concentric cylinders. For the present geometry and insulating liquid properties, the Wendt correlation gives the dimensionless torque coefficient as $C_T = T / (0.5 \rho \omega^2 r_i^4 L)$ a function of the gap Reynolds number. At $\omega = 1000$ rad/s ($Re \approx 1.45 \times 10^5$), the simulated torque differed from the Wendt prediction by approximately 8%, which is within the typical scatter of experimental data and confirms that the SST $k-\omega$ model captures the dominant turbulent momentum transport in the rotating annular flow. A similar comparison at $\omega = 500$ rad/s yielded a deviation of approximately 6%. These results provide confidence in the high-speed simulations beyond the laminar benchmark case.

2.3 Analysis of 2D Fluid Domain Flow Characteristics

Previous CFD studies of concentric annulus flows have examined flow development under various boundary conditions [17]. To evaluate the flow characteristics, a typical 2D fluid domain model was utilized with varying angular velocities applied to the inner cylinder boundary. FLUENT simulations were conducted using a periodic model, and the radial velocity profiles from the inner to the outer wall were recorded (Table 2). The corresponding velocity contour maps are presented in Fig. 3. At an angular velocity of 1 rad/s, the velocity distribution along the radial direction is relatively uniform. However, as the angular velocity increases, the high-velocity region progressively contracts toward the inner wall.

Table 2: 2D Fluid domain boundary angular velocity and corresponding linear velocity.

Angular Velocity (rad/s)	1	5	10	50	100	500	1000	2000	3000	4000
Linear Velocity (m/s)	0.0178	0.0890	0.178	0.890	1.78	8.9	17.8	35.6	53.4	71.2

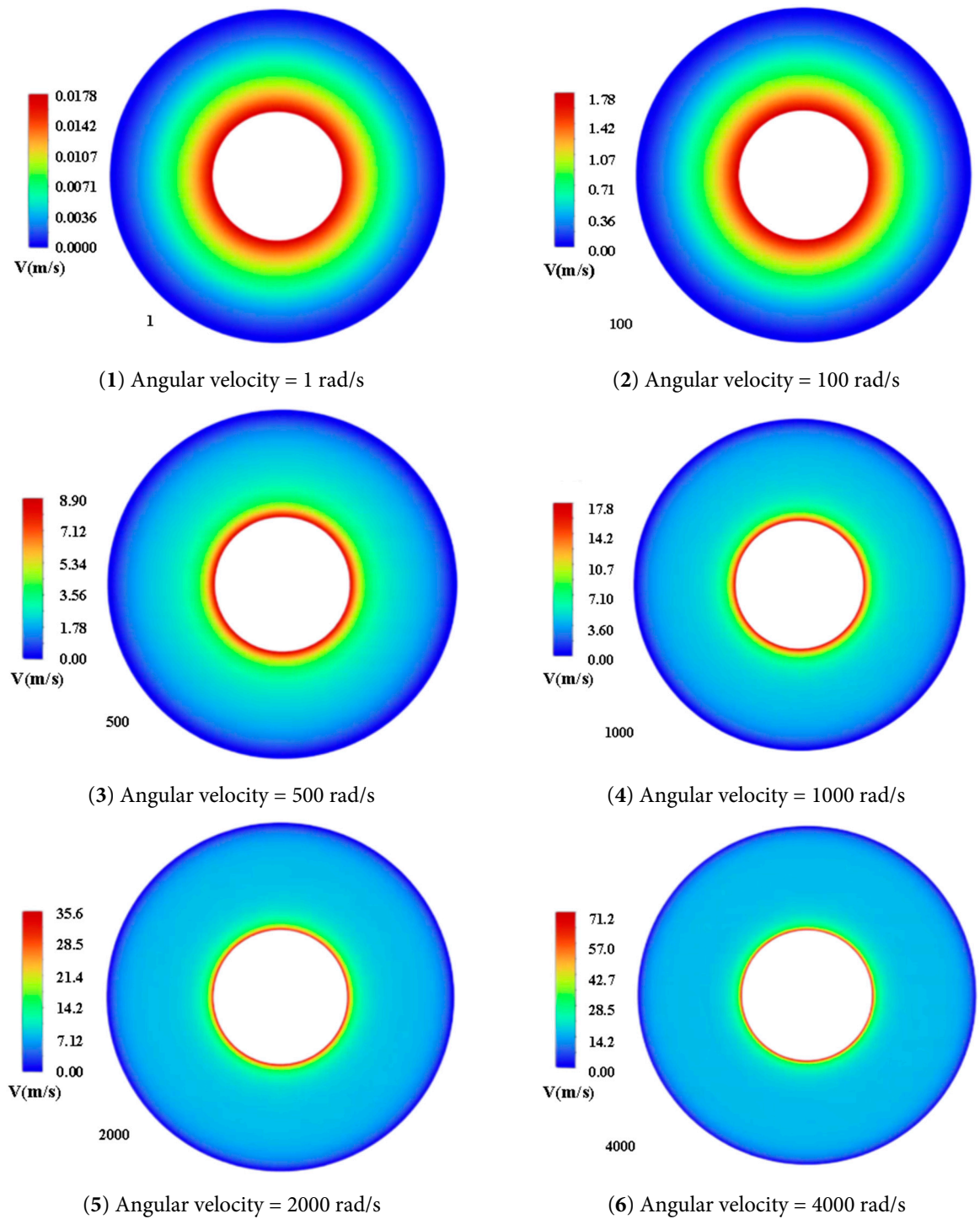


Figure 3: Velocity contour maps between the inner and outer cylinder boundaries at different angular speeds.

The velocity profiles were normalized using the maximum velocity at the inner cylinder boundary ($V/V_{\max}$), yielding the relative velocity distribution curves presented in Fig. 4. At angular velocities of 1 rad/s and 5 rad/s, the velocity decay along the radial direction is relatively gradual. However, when the rotational speed reaches 500 rad/s, the velocity drops sharply from the inner wall before plateauing into a localized constant-velocity zone, where the $V/V_{\max}$ ratio stabilizes between 19% and 21%. Furthermore, due to the stationary outer wall, a distinct inflection point emerges near $r/R_{\text{outer}} > 0.95$ for angular velocities ≥ 500 rad/s. This evolution signifies a fundamental transition in the flow regime: from viscous-force-dominated laminar flow at low speeds (e.g., 1 rad/s) to inertia-force-dominated turbulent flow at high speeds (e.g., 4000 rad/s), which consequently induces complex velocity gradients and localized constant-velocity regions.

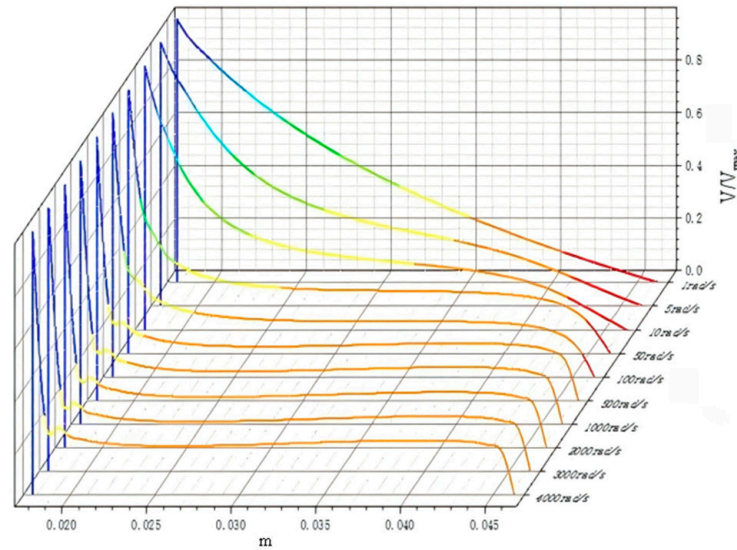
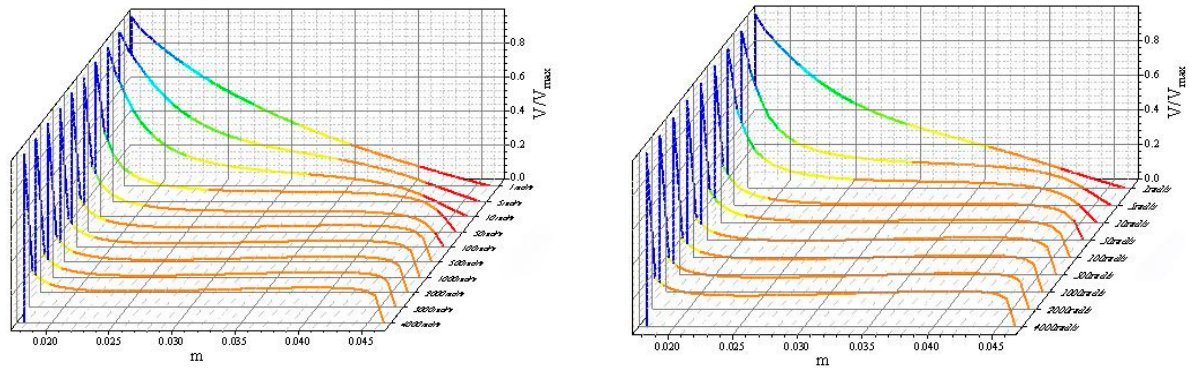


Figure 4: Variation of the normalized velocity ratio ($V/V_{\max}$) with the normalized radial coordinate (r/R_{outer}) for different angular velocities in the 2D fluid domain model.

In the thermal management of high-speed slip rings, the selection of the cooling medium plays a critical role, as the inherent thermophysical properties of different fluids uniquely dictate the flow and heat transfer characteristics. To investigate the sensitivity of the flow field to the working fluid, additional simulations were conducted by replacing the baseline fluid with water and a specialized insulating liquid (characterized by a mean density of 1.2 g/cm^3 and a kinematic viscosity of 10 cSt). For the three Newtonian fluids considered in this study (air, water, and a typical insulating liquid), the normalized velocity profiles (Fig. 5) reveal a notable consistency: the relative proportion of the constant-speed zone remains tightly bounded between 19% and 21%. This invariance suggests that, for these Newtonian fluids with constant, shear-independent viscosities, the formation and extent of this plateau region are predominantly governed by the system's angular velocity and geometric confinement, rather than by the specific fluid properties. Nevertheless, the distinct fluid properties do introduce localized variations in the velocity gradients, and the observed behavior should not be generalized to non-Newtonian coolants, such as nanofluids or shear-thinning liquids. Specifically, the higher dynamic viscosity and density of the insulating liquid alter the viscous diffusion and momentum transfer near the rotating wall. Consequently, compared to water, the insulating liquid triggers the onset of the constant-speed zone at a lower angular velocity, steepens the velocity decay rate in the immediate vicinity of the inner wall, and shifts the critical inflection point further outward, delaying its appearance until $r/R_{\text{outer}} > 0.93$.



(a) Fluid material: H₂O

(b) Fluid material: Insulating liquid with mean parameters

Figure 5: Variation of the normalized velocity ratio (V/V_{max}) with the normalized radial coordinate (r/R_{outer}) for different fluid media in the 2D fluid domain model.

2.4 Analysis of 3D Fluid Domain Flow Characteristics

To capture the spatial flow characteristics and account for potential end effects that a 2D analysis cannot resolve, the baseline 2D concentric cylinder model was axially extruded to a length of 36 mm, thereby constructing a 3D hollow cylinder fluid domain (Fig. 6). Mesh generation and domain definition for this 3D geometry were executed using ANSYS ICEM CFD. Within the computational setup, the fluid medium was explicitly defined as the insulating liquid. Based on the actual thermophysical properties of the coolant, the density was specified as 1200 kg/m^3 and the dynamic viscosity was set to $0.0042 \text{ kg/(m}\cdot\text{s)}$, ensuring an accurate representation of the hydrodynamic behavior within the complex rotating channel.

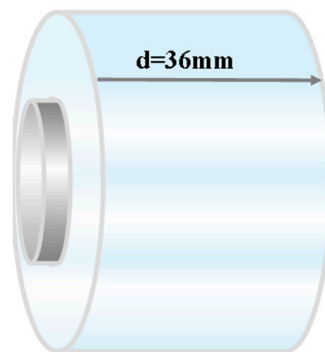


Figure 6: Schematic diagram of the 3D fluid domain structure.

To further investigate the flow dynamics, numerical simulations were conducted by applying varying angular velocities to the inner cylinder boundary. The derived linear velocities were recorded, utilizing the identical geometric configuration previously defined in Table 1. To accurately capture the internal flow structure without axial end-effects, the velocity contour maps were extracted from the mid-plane cross-section of the 3D model, as illustrated in Fig. 7.

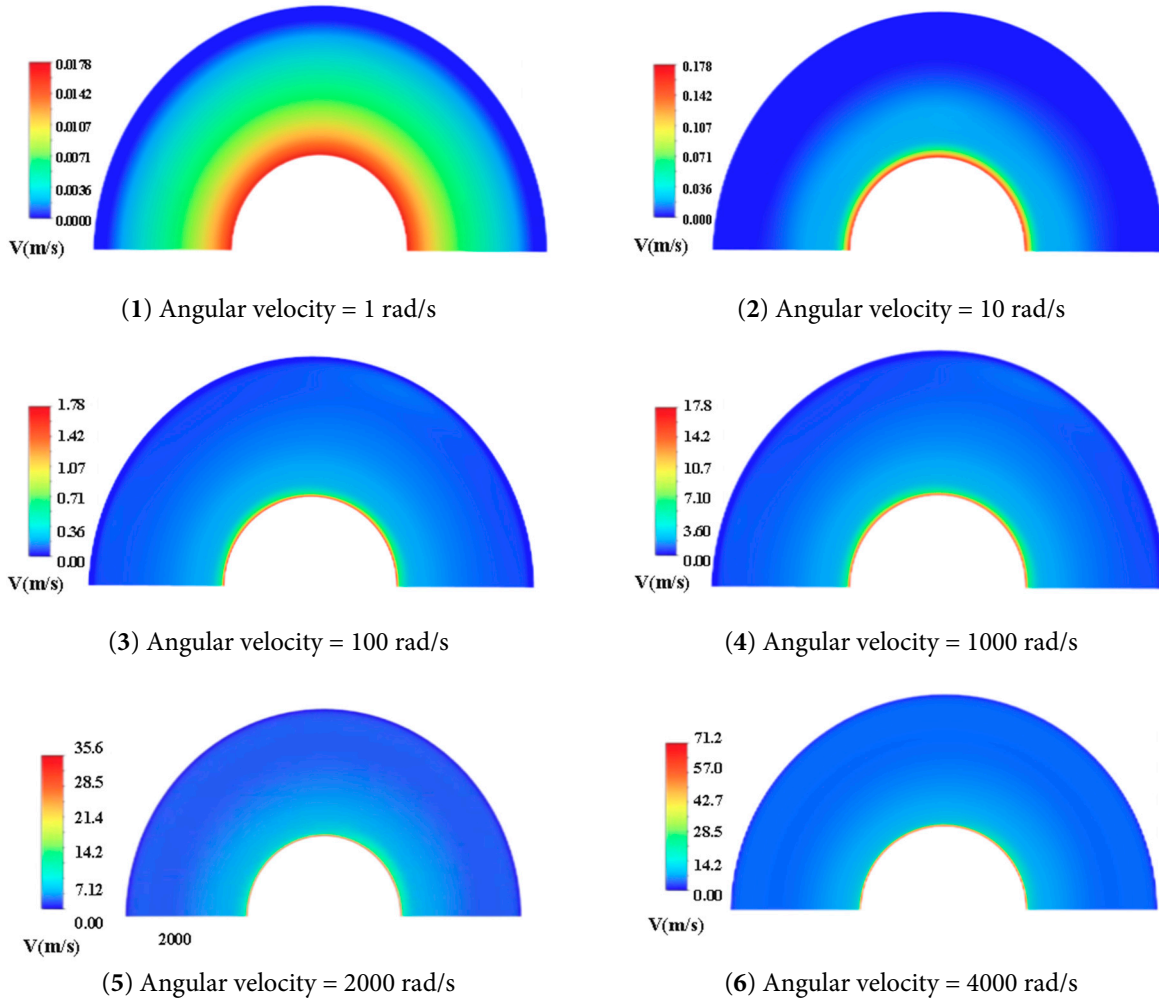


Figure 7: Velocity contour maps between the inner and outer cylinder boundaries at different angular velocities in the 3D model.

Compared to the 2D results, the 3D simulations exhibit consistent overall trends: a relatively uniform velocity distribution at low speeds, and accelerated decay near the inner wall at high speeds. However, at 10 rad/s, the 3D model demonstrates a more pronounced velocity decay, with the high-speed region near the inner wall diminishing more rapidly. This discrepancy is attributed to end-wall effects, which induce an axial velocity component (V_z). While the velocity distribution in the immediate vicinity of the inner wall ($R_{\text{outer}} < 0.3$) closely aligns with the 2D predictions, the decay rate in the mid-radius region ($0.4 < r/R_{\text{outer}} < 0.7$) increases by 5% to 8%. This accelerated dissipation is visually evident in the contour maps as expanded light-colored areas.

To systematically evaluate the spatial flow characteristics, the velocity data extracted from the 3D mid-plane cross-section were normalized by the maximum inner wall velocity, as illustrated in Fig. 8. At a low angular velocity of 1 rad/s, the radial velocity decay is highly uniform, a typical characteristic of viscous-dominated laminar flow where momentum diffuses smoothly outward. Starting from 5 rad/s, the flow regime begins to transition; the velocity decay accelerates sharply near the inner wall before stabilizing into a distinct plateau. Within this plateau region, the relative velocity maintains a higher constant proportion (20%–22%), and the inflection point is delayed outward to $r/R_{\text{outer}} > 0.9$. However, as

the angular velocity continues to increase, the inertial forces become more dominant, disrupting this stable plateau. The plateau proportion progressively decreases to a range of 15%–19%, indicating a strong negative correlation with the rotational speed. For instance, at a high rotational speed of $\omega = 1000 \text{ rad/s}$, the radial span of the high-speed region ($V/V_{\max} > 0.8$) in the 3D model is reduced to less than 50% of that observed in the 2D model. This substantial reduction demonstrates that 3D end-wall effects induce secondary axial flows (e.g., Taylor-Couette vortices), which significantly interfere with radial momentum transport, thereby reducing the overall momentum transfer efficiency in the core flow.

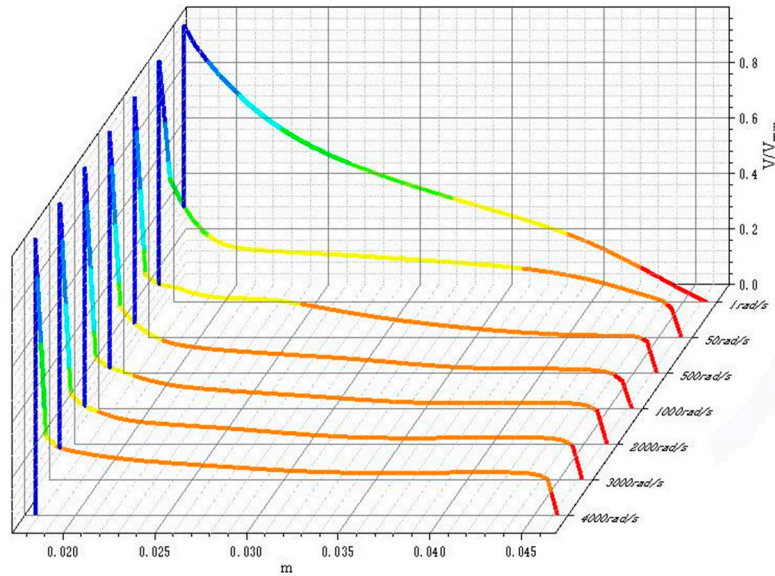


Figure 8: Variation of the normalized velocity ratio ($V/V_{\max}$) with the normalized radial coordinate (r/R_{outer}) for different angular velocities in the 3D fluid domain model.

A comparative analysis between the 2D and 3D results reveals that, within the 3D domain, axial pressure gradients likely induce secondary flows (e.g., Dean vortices). These secondary structures transport high-speed fluid toward the axial end-walls, thereby accelerating radial momentum dissipation. Furthermore, the inherent anisotropy of 3D turbulence facilitates the redistribution of kinetic energy from the primary tangential component into radial and axial fluctuations, which further disrupts the sustenance of a high-speed core flow. Consequently, the discrepancies between the 2D and 3D models remain marginal ($\Delta\eta < 3\%$) in the low-speed, viscous-dominated laminar regime. However, these differences become substantial in the high-speed, inertia-dominated turbulent regime, where the 3D model exhibits a 6%–10% increase in the velocity decay rate due to the additional kinetic energy dissipation associated with complex axial motions.

Because 2D or axisymmetric models systematically overestimate the core plateau velocity compared with finite-length 3D geometries, they correspondingly underestimate the local pressure and shear loads by up to approximately 10%. Consequently, gap dimensions and preload calculations based solely on 2D analyses should incorporate a corresponding safety margin, particularly near the axial ends of the cooling channel where end-wall-induced secondary flows are most pronounced.

2.5 Analysis of Flow Characteristics within the Constant-Speed Zone

At high rotational speeds, the flow enters a high-Reynolds-number regime dominated by inertial effects. The flow field may undergo transition from laminar to turbulent or develop complex coherent

structures (e.g., Taylor vortices), resulting in highly nonlinear velocity profiles [18]. Concurrently, the viscous boundary layer near the rotating inner cylinder thins significantly, driving a rapid initial velocity decay, whereas viscous effects remain pronounced near the stationary outer wall, enforcing a sharp deceleration. In the central region, where viscous damping is minimal and inertial forces dominate, an approximately constant-velocity zone emerges.

This central plateau is further stabilized by centrifugal forces, which push the high-momentum fluid radially outward against the opposing pressure gradient established by the outer wall. At rotational speeds such as 500 rad/s, turbulent transition enhances cross-stream momentum mixing, yielding a distinctly flatter velocity profile in the core. The stationary outer wall imposes a strict no-slip condition, forcing the fluid velocity to zero and generating a sharp near-wall decay gradient. The abrupt transition from the near-constant core velocity to this rapid outer decay gives rise to the distinct inflection point observed at $r/R_{\text{outer}} > 0.9$.

The formation of this constant-speed zone signifies a dynamic equilibrium in momentum transfer. In engineering applications such as centrifugal separators and rotating machinery, this equilibrium plays a critical role in enhancing operational stability and efficiency [19,20]. Specifically for high-speed slip rings, the uniform velocity within this zone, combined with empirical correlations derived from extensive parametric data, could enable the rapid and accurate estimation of flow-induced forces.

3 CFD-Based Fluid Force Calculation for a Simplified Single-Wire Model

As an independent check of the force calculation methodology, the fluid force on a smooth circular cylinder in cross-flow was first computed using the same numerical setup and compared with the well-established drag coefficient data. For a cylinder Reynolds number of $Re_d = 200$ (based on the wire diameter), the predicted drag coefficient differed from the empirical value by less than 5%, confirming the accuracy of the pressure and shear force integration procedure. This validation supports the reliability of the force results obtained for the brush wire configuration, even though direct experimental data for the specific slip ring geometry are not yet available.

For the single-wire conductive slip ring, a simplified CFD model is established to calculate the fluid forces acting on the brush wire, as shown in Fig. 9. For the single-wire conductive slip ring model, the tangential contact between the brush wire and the conductive ring groove is simplified, and the ring groove surface and the inner and outer cylindrical walls are treated as smooth. The present approach adopts a one-way coupling strategy, in which the fluid forces are extracted from the CFD solution and treated as external loads on the rigid brush wire; no elastic deformation or vibration of the wire is solved. The model is formulated based on the following assumptions:

- (a) The contact interface between the brush wire and the conductive ring groove is treated as a continuous elastic contact, thereby neglecting micro-scale surface roughness effects [21].
- (b) All solid materials are assumed to be isotropic, with their thermophysical properties varying linearly with temperature.
- (c) The cooling fluid is modeled as an incompressible Newtonian fluid. The flow is turbulent for the rotational speeds considered (up to 20,000 rad/s), and therefore the SST $k-\omega$ turbulence model is employed. The laminar assumption is retained only for the lowest speed case (1 rad/s) used in the analytical benchmark comparison.
- (d) The electromagnetic and thermal fields are quasi-statically coupled, explicitly ignoring high-frequency eddy current effects.

- (e) The brush wire is treated as a rigid body, and the fluid forces are extracted from the CFD solution without considering the elastic deformation or vibration of the wire (one-way coupling). Accordingly, the elastic modulus, wire stiffness, damping, natural frequencies, and displacement/stress fields of the brush wire are not defined or solved in the present model.

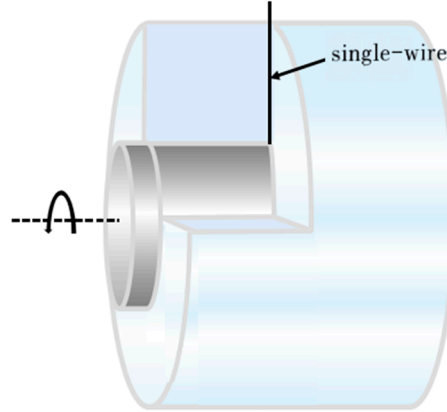


Figure 9: Schematic of the single-wire end-face contact slip-ring groove model.

The simplified single-wire model was simulated over an angular velocity range of 1–20,000 rad/s. The pressure, flow velocity near the wire, and resulting fluid forces (normal pressure force, tangential shear force, and total fluid force) were recorded (Table 3).

Table 3: Numerical values of pressure, shear force, and total fluid force for different inner wall angular velocities.

Angular Velocity (rad/s)	Corresponding Speed (rpm)	Reynolds Number Re (-)	Pressure Force (N)	Shear Force (N)	Total Force (N)
1	9.554140127	1.45E+02	1.66E−06	1.71E−06	3.37141E−06
10	95.54140127	1.45E+03	3.71E−05	3.36E−05	7.06329E−05
100	955.4140127	1.45E+04	0.000928297	0.000325058	0.001253355
1000	9554.140127	1.45E+05	0.07700677	0.007895174	0.084901944
2000	19,108.28025	2.90E+05	0.37455653	0.038319569	0.412876099
3000	28,662.42038	4.34E+05	0.82795073	0.084153548	0.912104278
4000	38,216.56051	5.79E+05	1.4120145	0.14268005	1.55469455
5000	47,770.70064	7.24E+05	2.0421167	0.19771859	2.23983529
6000	57,324.84076	8.69E+05	2.4943965	0.23708813	2.73148463
7000	66,878.98089	1.01E+06	3.6681198	0.32944642	3.99756622
8000	76,433.12102	1.16E+06	4.66309	0.41373995	5.07682995
9000	85,987.26115	1.30E+06	5.7831177	0.50267123	6.28578893
10,000	95,541.40127	1.45E+06	7.0279193	0.61010853	7.63802783
20,000	191,082.8025	2.90E+06	25.455347	1.9374368	27.392783

Quantitative analysis of the force distributions reveals that both pressure (normal dynamic pressure) and shear force (viscous shear) increase nonlinearly with rotational speed, though pressure exhibits a significantly steeper growth gradient. Over the range of $\omega = 1\text{--}20,000$ rad/s, the pressure contribution to the total fluid force increases from 50% to 92%, indicating the overwhelming dominance of dynamic pressure effects at high speeds. The fitted scaling laws yield $F_p \propto \omega^{1.8}$ for the normal pressure force and $F_\tau \propto \omega^{1.3}$ for the tangential shear force. These exponents were obtained by least-squares fitting of the data in Table 3 over the full speed range ($\omega = 1\text{--}20,000$ rad/s), with $R^2 > 0.99$ for both fits, and are intended only to illustrate the qualitative trend that the normal pressure force grows faster than the shear force.

In conventional brush wire assemblies, the initial preload is typically set at $F_{\text{preload}} = 6\text{--}8\text{ N}$, determined by the material yield strength and contact area [22]. However, when the system reaches a critical threshold of $\omega \approx 8000\text{ rad/s}$ ($v \approx 140\text{ m/s}$), the fluid pressure F_p rises to 70%–80% of the preload. At this juncture, the preload must be dynamically adjusted to $F_{\text{preload}} \geq 1.5F_p$ to counteract the hydrodynamic lift effect and prevent localized wire detachment, which would otherwise trigger severe contact resistance spikes. Conversely, under low-speed conditions ($\omega < 1000\text{ rad/s}$) where the total fluid force is negligible, the preload can be moderately reduced to 2–4 N. This minimizes frictional power loss from over-compression while maintaining contact stability.

Consequently, distinct operational strategies are recommended for varying speed regimes. For high-speed scenarios, a closed-loop preload control system is advised, dynamically regulating the preload based on real-time speed and fluid pressure monitoring to maintain a strict safety margin ($F_{\text{preload}}/F_p > 1.2$). For low-speed scenarios, surface micro-texturing of the brush wire (e.g., laser-processed micro-dimples) may reduce the effective viscosity by 20%–30%, thereby suppressing shear force magnitude and extending wear life. Ultimately, the relative contribution between pressure and shear forces dictates the preload design. Future development of a quantitative mapping model among rotational speed, fluid force, and preload could enable condition-adaptive control, ensuring electrical contact reliability while minimizing friction, wear, and energy consumption.

Fig. 10((1)–(6)) show mid-plane velocity contours at $\omega = 1, 10, 100, 1000, 10,000$, and $20,000\text{ rad/s}$, revealing flow separation and vortex evolution dynamics induced by the brush wire. At low speeds ($\omega \leq 10\text{ rad/s}$), velocity decays quasi-linearly radially ($V(r) = V_{\text{max}}(1 - r/R)$), closely matching the laminar solution of the brush-less model. Velocity differences between the wire's windward ($\theta = 0^\circ$) and leeward ($\theta = 180^\circ$) sides are small with no significant wake, indicating negligible wire disturbance under viscous dominance. At medium speeds ($10 < \omega \leq 1000\text{ rad/s}$), during transition, the velocity gradient increases steeply near the windward side ($\theta = -30^\circ\text{--}30^\circ$), while a low-velocity recirculation zone ($V/V_{\text{max}} < 0.2$) forms on the leeward side, creating an asymmetric profile and confirming flow instability onset. At high speeds ($\omega \geq 1000\text{ rad/s}$), dominated by inertia, a constant-speed zone ($V/V_{\text{max}} = 0.13 \pm 0.03$) forms in the outer region ($r < 0.9 R_{\text{outer}}$), occupying over 40% of the flow area. Furthermore, at high rotational speeds, a low-velocity wake region is observed downstream of the wire, indicating flow disturbance and localized velocity deficit. However, the steady RANS formulation used in this study cannot resolve periodic vortex shedding, and therefore no claim regarding vortex street frequency is made.

Fig. 11((1)–(6)) show 3D surface velocity contours at a fixed radial offset $\Delta r = 1\text{ mm}$. Comparative analysis reveals flow disturbance phenomena. At $\omega = 100\text{ rad/s}$, the axial velocity component V_z on the windward side shows a bimodal distribution (peak 1.2 m/s), while V_z on the leeward side approaches zero, forming an axial momentum transport barrier. At high speed ($\omega = 1000\text{ rad/s}$), three-dimensional flow disturbance structures appear in the surface plots, causing circumferential velocity fluctuations with an amplitude $\Delta V_\theta \approx 12\text{ m/s}$. At medium-to-high speeds ($\omega \geq 100\text{ rad/s}$), the velocity decay rate η along the fixed-radius surface remains at 60%–70%, indicating weak influence of rotational speed on boundary layer development. However, the phase of local velocity extremes (e.g., vortex core on the leeward side) shifts with increasing speed ($\Delta\theta \propto \log(\omega)$), reflecting vortex shedding. Quantitative analysis of brush-wire-induced flow separation shows significant windward/leeward velocity differences. At $\omega = 10,000\text{ rad/s}$, the corresponding pressure on the wire surface can reach 7 N. This non-uniform flow field pressure can induce lateral wire vibration, necessitating preload optimization to mitigate resonance risk.

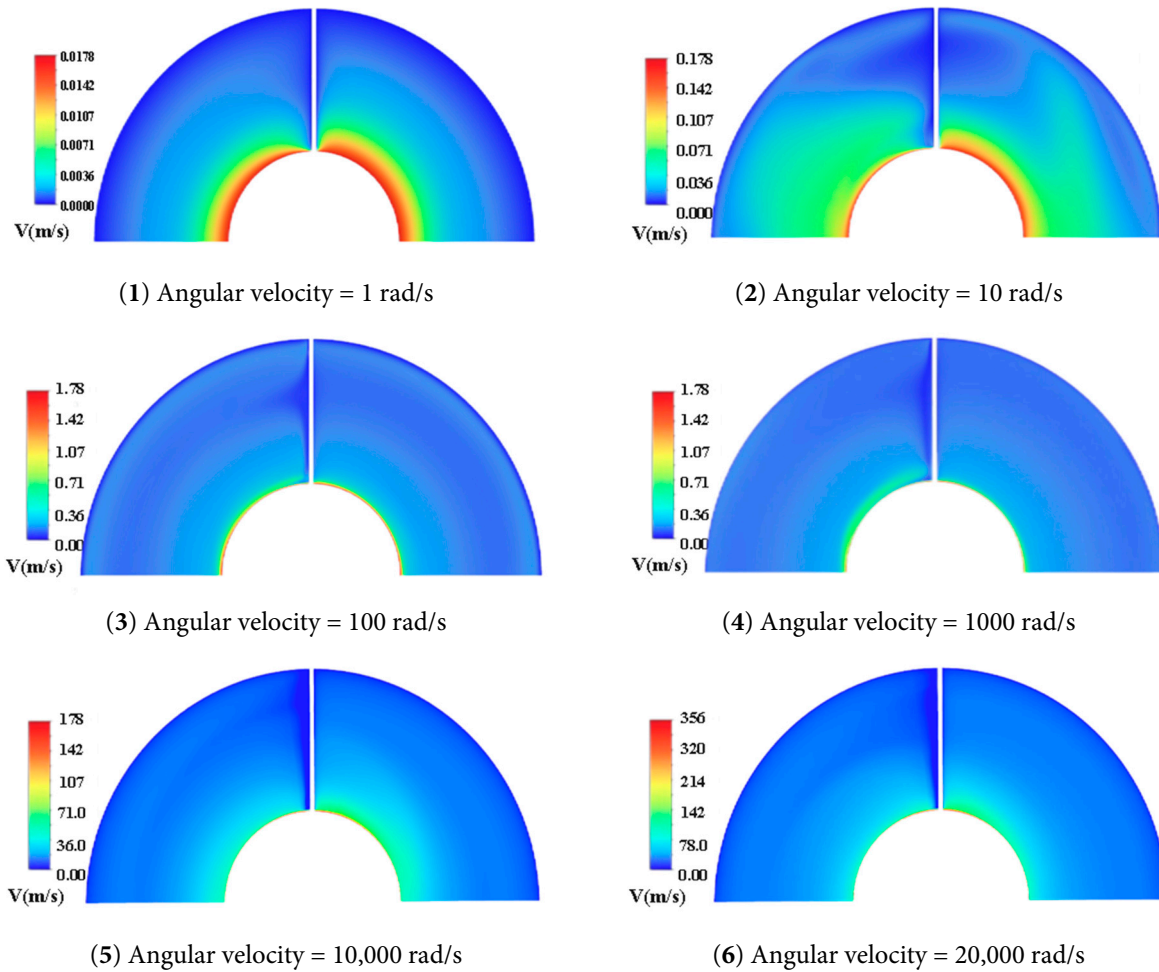


Figure 10: Velocity contour maps between the inner and outer cylinder boundaries at different angular velocities in the simplified single-wire conductive slip ring model.

The simplified single-wire model serves as an effective fundamental framework for demonstrating the inherent symmetry and baseline stability of the rotating annular flow field. However, the detailed flow visualizations and quantitative data presented herein highlight a critical phenomenon: as rotational speed increases, induced flow separation, periodic vortex shedding, and complex 3D disturbance effects emerge as the core instigators of dynamic instability at the brush-ring contact interface. Through comparative analysis across a comprehensive range of speed regimes and rigorous quantitative examination, this study uncovers the nonlinear regulatory mechanism by which brush wire geometry and rotational speed collectively govern flow structure evolution. Specifically, the transition from viscous-dominated laminar flow to inertia-driven turbulent flow triggers localized hydrodynamic fluctuations that directly challenge mechanical contact stability. Unveiling these intricate fluid-structure interaction dynamics provides a robust theoretical basis and clear optimization pathways, such as micro-structural surface modifications and condition-adaptive preload strategies, for the engineering design of high-reliability, high-speed conductive slip rings.

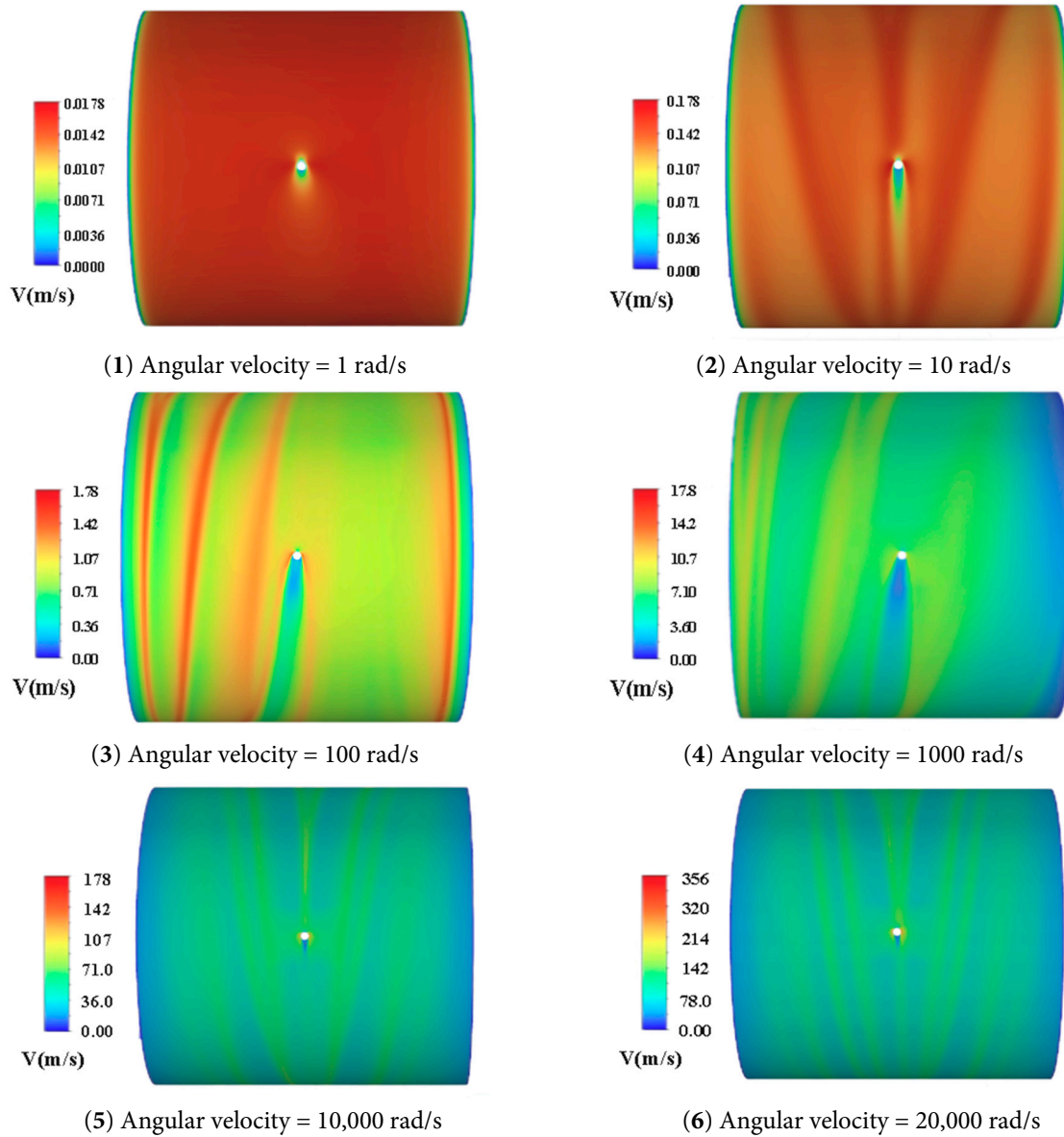


Figure 11: Surface velocity contour maps at a fixed radial distance (constant R) for different angular velocities in the simplified single-wire conductive slip ring model.

The present model treats the coolant as a single-phase incompressible Newtonian fluid, thereby neglecting multiphase effects such as bubble formation and cavitation. A first-order estimate at the maximum speed ($\omega = 20,000$ rad/s, $V = 356$ m/s) gives a characteristic dynamic pressure of $\frac{1}{2}\rho V^2 \approx 76$ MPa, while the vapor pressure of typical insulating liquids is on the order of 1 kPa or lower. The local pressure may therefore fall below the vapor pressure, indicating that cavitation cannot be excluded. Future work should incorporate multiphase modeling (e.g., volume of fluid (VOF) or mixture models) to account for potential bubble formation. In addition, brush wear and material degradation are not considered, and a coupled wear and structural analysis is recommended as a further extension. These simplifications do not affect the main conclusions regarding the velocity field characteristics and the dominant role of dynamic pressure at high speeds.

It should be noted that the present steady-state RANS simulations provide time-averaged flow fields and therefore cannot resolve unsteady phenomena such as periodic vortex shedding or transient vortex interactions. Accordingly, the descriptions of flow structures in this study are limited to those that can be inferred from time-averaged velocity and pressure fields; definitive identification of transient coherent structures would require time-resolved simulations (e.g., unsteady Reynolds-averaged Navier–Stokes (URANS), large eddy simulation (LES), or direct numerical simulation (DNS)) and is left for future work.

4 Conclusions

Based on a benchmark case of a typical rotating fluid domain, this study analyzes the 2D and 3D flow characteristics under high-speed conditions using identical structural parameters. Furthermore, a simplified single-wire conductive slip ring model was constructed to evaluate its mechanical behavior under variable friction conditions. The main conclusions are as follows:

- (a) Characterization of the constant-speed zone and inflection point phenomenon.

Simulation results indicate that under low-speed conditions, the flow exhibits laminar characteristics, and the radial decay of tangential velocity conforms to the theoretical linear solution ($V(r) \propto 1/r$), with over 95% agreement between the 2D and 3D velocity profiles. As the rotational speed increases, inertia-dominated secondary flows intensify significantly. In the 2D model, a constant-speed plateau ($V/V_{\max} = 19\%–21\%$) forms in the region of $r/R_{\text{outer}} = 0.3–0.8$. Conversely, in the 3D model, due to the interactions between end-wall effects and circumferential vortices, this plateau emerges earlier ($V/V_{\max} = 15\%–19\%$) with an expanded spatial range, indicating that 3D flows are more susceptible to forming inertia-dominated quasi-uniform structures. The velocity inflection point near the outer wall ($r/R_{\text{outer}} > 0.9$) is attributed to boundary layer separation, with its precise position regulated by the local ratio of viscous to inertial forces. Because the 2D model inherently neglects the axial velocity component, its predicted inflection point occurs approximately 5% farther outward compared to the 3D model.

- (b) Verification of the consistency of flow characteristics across the tested Newtonian fluids.

A multi-condition comparison within the 2D fluid domain was conducted using three typical Newtonian media (air, water, and a typical insulating liquid) at an identical inner-wall linear speed ($V_r = 10$ m/s). The results reveal highly similar dimensionless velocity decay curves ($V/V_{\max}$ vs. r/R_{outer}) with a maximum deviation of less than 4%, while the constant-speed proportion remains stable ($V/V_{\max} = 19\%–21\%$). Although variations in medium viscosity primarily affect the flow transition point, the velocity gradient within the plateau remains inertia-dominated. These findings indicate that, for the Newtonian fluids considered, the fundamental flow characteristics are governed by the relative weighting of viscous and inertial terms in the dimensionless Navier–Stokes equations, thereby providing a reasonable basis for model simplification across similar Newtonian working media. However, this conclusion should not be extended to non-Newtonian coolants, such as nanofluids or shear-thinning liquids, which require separate investigation.

- (c) Investigation of the proportional relationship between pressure and shear forces.

Based on the internal architecture of a single-wire slip ring and the simulation principles of rotating fluid domains, a simplified single-wire model was established. Comparative verification demonstrates that the simplified end-contact model yields highly accurate and comprehensive results. Using this model, the force characteristics under multi-speed conditions ($\omega = 1–20,000$ rad/s) were analyzed: the growth of the normal pressure component with speed aligns with fluid dynamic pressure theory, whereas the shear component growth approximates the viscous shear model. When $\omega > 1000$ rad/s, pressure contributes

over 90% of the total hydrodynamic force, signifying that inertia dominates energy transport. Concurrently, a stable “pressure shielding layer” forms on the brush wire surface, inducing a localized velocity plateau ($V/V_{\max} = 10\%–16\%$). This phenomenon is consistent with 3D fluid domain characteristics, though the dimensionless velocity constant is slightly lower due to the relative near-wall stability. Ultimately, flow field analysis based on cross-sectional and surface velocity contours confirms that flow separation, vortex shedding, and 3D disturbance effects act as the primary instigators of dynamic instability at the contact interface.

It should be noted that experimental validation under actual high-speed rotating and liquid-cooled conditions is planned for future work, as the present study focuses on theoretical and numerical analysis validated against analytical benchmarks. Furthermore, the present findings lay a foundation for future AI-driven preload control strategies that utilize real-time CFD surrogate models, enabling adaptive optimization for next-generation high-speed slip ring designs.

Acknowledgement: The authors would like to thank their colleagues for their technical assistance and helpful discussions during this study.

Funding Statement: The authors received no specific funding for this study.

Author Contributions: The authors confirm contribution to the paper as follows: study conception and design: Zhijun Guo, Jingwei Liu; data collection: You Fu; analysis and interpretation of results: Zhijun Guo, You Fu, Jiaxin Yang; draft manuscript preparation: Jingwei Liu, Jiaxin Yang. All authors reviewed and approved the final version of the manuscript.

Availability of Data and Materials: All data generated or analyzed during this study are included in this published article. The raw datasets are also available from the corresponding author upon reasonable request.

Ethics Approval: Not applicable.

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Li T, Wu J, Zhong J, Guo Z, Fu Y. Characterizing the contact force of a high-speed rotating fluid on brush wire. *J Phys Conf Ser.* 2024;2882(1):012095. [[CrossRef](#)].
2. Wu H, Zhao X, Li W, Li Y, Pan T, Yang W, et al. Research status and progress on the current-carrying friction and wear performance of conductive slip rings in harsh environments. *Coatings.* 2025;15(11):1347. [[CrossRef](#)].
3. Acquah M, Liu Z. Generative design of liquid-cooling channels for thermal management of 2.5D and 3D integrated advanced packaging. *arXiv:2608.22787.* 2026.
4. Zhang K, Liu K, Sun D, Gao T, Ye J. Friction variability in slip rings: Geometry and stiffness effects. *J Tribol.* 2024;146(10):101702. [[CrossRef](#)].
5. Grossman MI, Lopata AS, Timoshchuk VV, Guterman AB, Gubaliuk ZS. Possibility of compensating for hydrodynamic lift force in brush-collector contacts in liquid-filled electrical machines. *Elektromashinost I Elektrooborud.* 1977;25:107–10.
6. Meseguer A, Mellibovsky F, Avila M, Marques F. Instability mechanisms and transition scenarios of spiral turbulence in Taylor-Couette flow. *Phys Rev E Stat Nonlin Soft Matter Phys.* 2009;80(4 Pt 2):046315. [[CrossRef](#)].
7. Boulafentis T, Lacassagne T, Cagney N, Balabani S. Experimental insights into elasto-inertial transitions in Taylor–Couette flows. *Phil Trans R Soc A.* 2023;381(2243):20220131. [[CrossRef](#)].
8. Grossmann S, Lohse D, Sun C. High-Reynolds number Taylor-Couette turbulence. *Annu Rev Fluid Mech.* 2016;48:53–80. [[CrossRef](#)].
9. Sacco F, Verzicco R, Ostilla-Mónico R. Dynamics and evolution of turbulent Taylor rolls. *J Fluid Mech.* 2019;870:970–87. [[CrossRef](#)].

10. Chen C, Sun D, Chen Y, Xu J, Wang H. Thermo-hydraulic performance and entropy generation analysis of serpentine and straight tubes with twisted tape inserts using SST k - ω turbulence modeling. *Fluid Dyn Mater Process.* 2026;22(6):7. [[CrossRef](#)].
11. Li S, Tian Y, Deng G, Li W, Hu Y, Wen X. Optimization-based correction of turbulence models for flow prediction in control valves. *Fluid Dyn Mater Process.* 2025;21(8):1809–37. [[CrossRef](#)].
12. Bin L. Simulation analysis of local temperature field of electric propulsion slip ring. *Vibroeng Procedia.* 2024;54:1–7. [[CrossRef](#)].
13. Taylor E, Patel K, Ahmed M, Jay R, Ghosh S, Otto M, et al. Design and analysis of hexagonal honeycomb layers for thermal-structural performance in ammonia fuel tanks. In: *Proceedings of the ASME Turbo Expo 2026 Turbomachinery Technical Conference and Exposition*; 2026 Jun 15–19; Milan, Italy.
14. Greenspan HP, Howard LN. On a time-dependent motion of a rotating fluid. *J Fluid Mech.* 1963;17(3):385–404. [[CrossRef](#)].
15. Brewster DB, Nissan AH. The hydrodynamics of flow between horizontal concentric cylinders—I: Flow due to rotation of cylinder. *Chem Eng Sci.* 1958;7(4):215–21. [[CrossRef](#)].
16. Wendt F. Turbulent flows between two rotating coaxial cylinders. *Ing Arch.* 1933;4(6):577–95. (In German).
17. Jen LW, Alias MSM. Computational fluid dynamics (CFD) study of flow development in concentric annulus and eccentric annulus using taguchi method. *Int J Infrastruct Res Manag.* 2023;11(1):53–68. [[CrossRef](#)].
18. van der Veen RCA, Huisman SG, Merbold S, Harlander U, Egbers C, Lohse D, et al. Taylor–Couette turbulence at radius ratio: Scaling, flow structures and plumes. *J Fluid Mech.* 2016;799:334–51. [[CrossRef](#)].
19. Paoletti MS, Lathrop DP. Angular momentum transport in turbulent flow between independently rotating cylinders. *Phys Rev Lett.* 2011;106(2):024501. [[CrossRef](#)].
20. Lewis GS, Swinney HL. Velocity structure functions, scaling, and transitions in high-Reynolds-number Couette–Taylor flow. *Phys Rev E.* 1999;59(5):5457–67. [[CrossRef](#)].
21. Yasar I, Canakci A, Arslan F. The effect of brush spring pressure on the wear behaviour of copper–graphite brushes with electrical current. *Tribol Int.* 2007;40(9):1381–6. [[CrossRef](#)].
22. Guo Z, Zhong J, Dai C. Influence of brush wire pressure on instantaneous break of slip rings. *Aviat Precis Manuf Technol.* 2018;54(5):60–2. (In Chinese).