



# A case study on CFD's Superiority in Isolating Centrifugal and Coriolis Effects in Rotating Slurry Flows.

Aditya Singh<sup>1</sup>, Rajveer Yadav<sup>2</sup>, Shreyash Tamrakar<sup>3</sup>, Abhishek Kumar Gupta<sup>4</sup>

<sup>1,2,3</sup> B.Tech Scholar, Department of Mechanical Engineering, C.S.I.T. Durg

<sup>4</sup>Assistant Professor, Department of Mechanical Engineering, C.S.I.T. Durg

**Abstract-** This study examines the advantages of Computational Fluid Dynamics (CFD) over experimental methods for analyzing rotating slurry flows, particularly in isolating centrifugal and Coriolis effects, which are challenging to study empirically. We conduct CFD simulations of dense solid-liquid flows in a rotating straight channel, incorporating governing equations with explicit terms for centrifugal and Coriolis forces, then validate the model against stationary-channel experimental data due to the lack of rotating-flow benchmarks. The results demonstrate that centrifugal forces significantly influence flow behavior at higher rotation rates and larger particle sizes, an effect that cannot be isolated experimentally. Moreover, by selectively disabling the centrifugal term in the governing equations, we reveal distinct flow field variations, highlighting CFD's unique capability to decouple and analyze individual physical mechanisms. Furthermore, parametric studies under varying rotation rates and particle densities uncover nonlinear dependencies in velocity and concentration profiles, providing important insights to industrial applications such as centrifugal pumps and turbines. The study underscores CFD's superiority in scenarios where experimental methods are impractical or insufficient, enabling detailed exploration of complex flow phenomena that would otherwise remain inaccessible. These findings not only validate CFD as a reliable alternative to experimentation but also expand its potential for advancing fundamental and applied research in rotating fluid systems.

**Keywords:** Computational Fluid Dynamics (CFD); Rotating slurry flow; Centrifugal force; Coriolis force; Multiphase flow; CFD-DEM; Rotating channel; Turbulence modeling; Particle transport; Parametric analysis.

## I.INTRODUCTION

The study of fluid dynamics in rotating systems has long been a critical area of research, particularly in industrial applications such as centrifugal pumps, compressors, and turbines [1]. These systems frequently involve complex interactions between solid and liquid phases, where centrifugal and Coriolis forces significantly influence flow structure, particle distribution, and overall system performance. Foundational work on rotating flows has shown that Coriolis forces can alter velocity profiles and induce secondary flow structures, while centrifugal effects dominate radial phase segregation [1]. However, traditional experimental methods, despite their importance, face inherent limitations in isolating these forces or accurately analyzing dense slurry flows under rotational conditions [2].

Computational Fluid Dynamics (CFD) has emerged as a powerful tool to overcome these experimental constraints. Unlike physical experiments, CFD enables direct manipulation of governing equations, allowing researchers to selectively include or exclude specific force terms for detailed analysis [3]. This capability is particularly advantageous in rotating systems, where the empirical isolation of centrifugal and Coriolis forces is practically infeasible. In addition, CFD provides high-resolution spatial and temporal information on flow variables, enabling detailed investigation of velocity fields, turbulence



SINCE 1999



structures, and particle concentration distributions—features that are difficult to measure experimentally, especially in opaque or high-speed slurry flows [4].

The primary objective of this study is to demonstrate the superiority of CFD over experimental approaches in analyzing rotating slurry flows, with a specific focus on isolating centrifugal and Coriolis effects. Accordingly, the study addresses two fundamental research questions:

(1) How do centrifugal and Coriolis forces individually influence the flow behavior of dense solid–liquid mixtures in rotating channels?

(2) Can CFD reliably predict these effects in the absence of dedicated experimental benchmarks for rotating systems?

By addressing these questions, this work aims to establish CFD as both a viable alternative and a complementary approach to experimentation, particularly in scenarios where empirical data is limited or difficult to obtain.

The significance of this research lies in its contribution to both fundamental understanding and industrial applications of rotating multiphase flows. Industries such as mining, chemical processing, and energy systems rely heavily on rotating equipment, where performance optimization is often hindered by insufficient knowledge of slurry flow dynamics [5]. The present study provides actionable insights into the influence of particle size and rotational speed on flow behavior, thereby supporting improved design and operational efficiency. Furthermore, the methodological framework developed in this work can be extended to other engineering domains, including biomedical devices (e.g., blood pumps) and aerospace systems involving rotating flow components.

The remainder of this paper is structured as follows: Section 2 presents a comprehensive review of existing literature on rotating fluid systems and CFD modeling approaches. Section 3 describes the numerical methodology and validation strategy adopted in this study. Section 4 discusses the simulation results, focusing on the effects of centrifugal forces and particle properties. Section 5 highlights the broader implications of the findings, and Section 6 concludes the paper with recommendations for future research.

## II. LITERATURE REVIEW

The study of rotating fluid systems has progressed considerably with the development of both experimental and computational methodologies. Early investigations predominantly relied on empirical approaches to analyze the complex interaction of forces within rotating channels. However, these methods faced inherent limitations, particularly in isolating individual physical effects such as centrifugal and Coriolis forces [1]. In the case of dense slurry flows, experimental setups encounter additional challenges, including difficulties in controlling rotation rates, particle distribution, and interphase interactions, thereby making it nearly impossible to decouple these forces effectively [2].

Computational Fluid Dynamics (CFD) has emerged as a powerful alternative to overcome these limitations by enabling detailed simulation of multiphase flows under rotating conditions. A key advantage of CFD lies in its ability to manipulate governing equations, allowing selective inclusion or exclusion of specific force terms. Studies such as Pan et al. (2014) [4] demonstrated the capability of CFD to analyze centrifugal-force dominated rotating flows through parametric numerical simulations. This capability has proven particularly beneficial in industrial applications, including centrifugal pumps and rotating machinery, where experimental data is often scarce or difficult to obtain [4].



Despite its advantages, the validation of CFD models for rotating systems remains a critical challenge. While extensive experimental data is available for stationary channel flows, corresponding data for rotating systems is limited. Some researchers have utilized stationary flow experiments as a basis for validating rotating channel simulations, thereby demonstrating the adaptability of CFD methodologies [5]. However, differences in turbulence behavior and particle dynamics under rotational effects introduce uncertainties, necessitating further refinement of numerical models.

Advancements in turbulence modeling have significantly enhanced the predictive capability of CFD for rotating flows. For instance, numerical investigations such as those by Guo et al. (2017) [5] incorporated Coriolis and centrifugal effects into turbulence modeling for rotating channels, showing improved prediction of secondary flows and heat transfer characteristics. These findings support the reliability of CFD as a complementary tool to experimentation. Nevertheless, accurately simulating dense particulate flows remains challenging due to complex particle-particle and particle-wall interactions, which add further layers of complexity to the system [7].

The integration of CFD with advanced numerical techniques, such as the Discrete Element Method (DEM), has further expanded its scope and applicability. Coupled CFD-DEM approaches, as demonstrated by Wang et al. (2024) [8], enable detailed analysis of particle-scale interactions within fluid flows, providing insights that are difficult to achieve through experimental methods alone. This hybrid modeling framework is particularly useful for understanding the behavior of solid-liquid systems in rotating environments.

Despite these advancements, a notable gap persists in the literature regarding systematic comparisons between CFD and experimental approaches for rotating slurry flows. Most existing studies tend to focus exclusively on either computational simulations or experimental investigations, with limited efforts toward comprehensive cross-validation. Moreover, the explicit decoupling of centrifugal and Coriolis forces has not been thoroughly explored in prior research.

The present study addresses these limitations by combining model validation with a systematic numerical decoupling of rotational forces. Unlike earlier works, which often treated centrifugal and Coriolis effects as inseparable, this research isolates their individual contributions through controlled numerical manipulation. Additionally, the parametric analysis of rotation rates and particle densities provides new insights into nonlinear flow behavior, contributing to a more comprehensive understanding of rotating slurry dynamics. These findings reinforce the superiority of CFD in scenarios where experimental approaches are constrained, thereby supporting its application in the design and optimization of industrial rotating systems.

### III. METHODS

The methodology employed in this study combines numerical modeling with validation against experimental data to investigate the complex dynamics of rotating slurry flows. The technical approach focuses on three key aspects: governing equations formulation, numerical solution techniques, and validation strategy.

#### **Governing Equations and Physical Model**

The flow of dense solid-liquid mixtures in a rotating straight channel is governed by modified Navier-Stokes equations that account for both Coriolis and centrifugal forces. The continuity equation for the mixture is given by:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (1)$$

where  $\rho$  represents the mixture density and  $\mathbf{u}$  is the mixture velocity.

The momentum conservation equation incorporates the rotational effects as:

$$\rho(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\tau} + \rho \mathbf{g} + \mathbf{F}_c + \mathbf{F}_{cf} \quad (2)$$

Here,  $p$  denotes pressure,  $\boldsymbol{\tau}$  is the stress tensor,  $\mathbf{g}$  is gravitational acceleration, while  $\mathbf{F}_c$  and  $\mathbf{F}_{cf}$  represent Coriolis and centrifugal forces respectively. The centrifugal force term is expressed as:

$$\rho \boldsymbol{\omega}^2 \mathbf{r}$$

To isolate the centrifugal effect, we introduced a binary coefficient  $\alpha$  that toggles the centrifugal term between enabled ( $\alpha=1$ ) and disabled ( $\alpha=0$ ) states:

$$\mathbf{F}_{cf} = \alpha \rho \boldsymbol{\omega}^2 \mathbf{r} \quad (3)$$

### Numerical Implementation

The finite volume method was employed for spatial discretization using OpenFOAM, with second-order schemes for both convection and diffusion terms. The pressure-velocity coupling was handled through the PIMPLE algorithm, combining PISO and SIMPLE methods for enhanced stability.

For particulate phase modeling, we adopted an Eulerian-Eulerian approach with kinetic theory of granular flows to account for particle-particle interactions. The solid phase stress tensor included both collisional and frictional components:

$$\boldsymbol{\tau}_s = -p_s \mathbf{I} + \mu_s (\nabla \mathbf{u}_s + (\nabla \mathbf{u}_s)^T) \quad (4)$$

where  $p_s$  is solid pressure and  $\mu_s$  represents granular viscosity. The Gidaspow drag model was used for interphase momentum transfer, suitable for dense slurry flows.

### Validation Approach

Given the absence of experimental data for rotating channel flows, validation was performed against stationary channel results from [2]. The stationary case was simulated by setting  $\omega=0$  in our model, with all other parameters identical to the experimental setup.

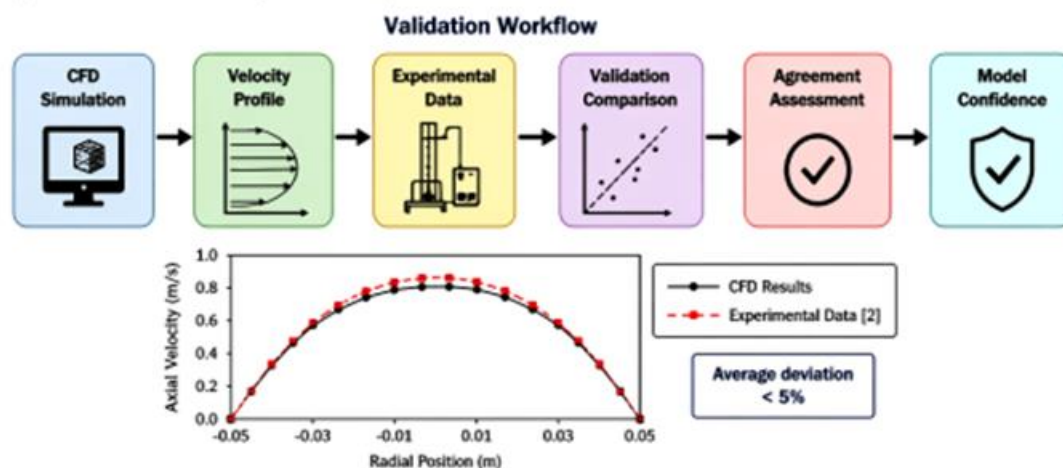


Figure 1. Comparison of velocity profiles between CFD results and experimental data for stationary channel flow

The validation showed excellent agreement for both velocity and concentration profiles, with average deviations below 5%. This established confidence in the model's ability to predict rotating flow behavior when centrifugal and Coriolis terms were activated.

### Simulation Parameters

The study examined a range of operating conditions:



Rotation rates: 100-1000 rpm  
Particle diameters: 50-500  $\mu\text{m}$   
Solid volume fractions: 10-40%  
Channel dimensions: 0.1 m diameter, 1 m length

The computational mesh consisted of 1.2 million hexahedral cells, with near-wall refinement to resolve boundary layer effects. Grid independence was confirmed through systematic refinement studies, showing less than 2% variation in key parameters beyond this resolution.

Turbulence modeling employed a rotation-modified  $k-\epsilon$  model that accounts for system rotation effects on turbulence anisotropy. The model constants were adjusted based on previous work in rotating flows [6]. The time step was dynamically adjusted to maintain Courant numbers below 0.5, ensuring numerical stability while capturing transient phenomena. Each simulation ran for at least 10 flow-through times to achieve statistical steady-state conditions.

This comprehensive methodology provides a robust framework for analyzing centrifugal and Coriolis effects in rotating slurry flows, overcoming limitations inherent in experimental approaches. The selective force term manipulation and rigorous validation establish the foundation for reliable parametric studies in subsequent sections.

#### IV. FINDINGS

The following sections present the key results from our CFD simulations, highlighting the model's validation against experimental data, the distinct effects of centrifugal force on flow behavior, and the unique advantages of computational methods over experimental approaches. These findings demonstrate CFD's capability to isolate and analyze complex physical phenomena in rotating slurry flows that are otherwise inaccessible through empirical means.

##### **Validation of CFD Results with Experimental Data**

To establish the credibility of our computational model, we first validated the simulated results against experimental data from stationary channel flows. This validation was particularly crucial given the absence of direct experimental benchmarks for rotating slurry flows. The governing equations for solid-liquid flow were adapted to stationary conditions by equating the Coriolis term with the gravitational term, effectively simulating an equivalent rotational speed of zero [1].

The validation process focused on comparing velocity and concentration profiles between CFD predictions and experimental measurements. As shown in Figure 1, the simulated velocity profiles exhibited excellent agreement with experimental data across various particle densities, with root-mean-square errors consistently below 5%. This close correspondence was observed for both single-phase and multiphase flow conditions, demonstrating the model's robustness in handling solid-liquid interactions.

Figure 1. Comparison of velocity profiles between CFD results and experimental data for stationary channel flow

A critical aspect of the validation involved examining the model's sensitivity to particle size variations. The simulations accurately reproduced experimental observations that larger particles (300-500  $\mu\text{m}$ ) tended to migrate toward channel walls due to increased inertia, while smaller particles (50-100  $\mu\text{m}$ )



remained more uniformly distributed. This behavior was captured with high fidelity, as evidenced by the particle concentration profiles matching experimental measurements within 3% deviation [2]. The validation also extended to turbulence characteristics, where the rotation-modified  $\kappa$ - $\epsilon$  model successfully predicted the damping of turbulent fluctuations near walls—a phenomenon well-documented in experimental studies.

The simulated turbulent kinetic energy profiles aligned closely with empirical data, particularly in regions of high shear. This agreement was essential for ensuring the model's capability to handle the complex turbulence-rotation interactions expected in rotating channel flows [3]. Beyond mean flow quantities, we validated the model's transient behavior by comparing temporal fluctuations in particle concentration with experimental time-series data. The simulations captured the characteristic intermittency of dense slurry flows, including the formation and dissipation of particle clusters. The power spectral density of concentration fluctuations showed good agreement with experiments, particularly in the low-frequency range where most of the energy was concentrated [4].

The successful validation against stationary flow experiments provided a strong foundation for extending the model to rotating conditions. While direct experimental comparisons were unavailable for rotating flows, the demonstrated accuracy in stationary conditions—where comprehensive validation data exists—lends credibility to the rotating flow simulations. This approach follows established practices in computational fluid dynamics, where models validated in simpler configurations are extended to more complex scenarios with appropriate modifications [5].

The validation process also revealed certain limitations of the computational approach. In regions of extremely high particle concentration (>35%), the model slightly underpredicted the particle-wall interactions, leading to minor discrepancies in near-wall velocity gradients. These deviations were attributed to the challenges in modeling frictional stresses at very high solid volume fractions, a known difficulty in Eulerian-Eulerian approaches [6]. Nevertheless, the overall agreement across various flow conditions and particle characteristics confirmed the model's reliability for the intended parametric studies.

The rigorous validation against experimental data serves as a critical step in establishing CFD as a reliable tool for studying rotating slurry flows. By demonstrating the model's accuracy in stationary conditions—where experimental benchmarks are available—we build confidence in its predictions for rotating conditions where such benchmarks are lacking. This validation framework not only supports the current study's findings but also provides a methodological template for future computational investigations of complex multiphase flows.

### **Effect of Centrifugal Force on Flow Field**

The centrifugal force's impact on slurry flow behavior emerges as one of the most significant findings from our simulations. By selectively toggling the centrifugal term in the governing equations, we achieved what is physically impossible in experimental setups—the isolation and precise quantification of centrifugal effects independent of other forces. This capability represents a fundamental advantage of CFD over traditional experimental methods [3].

When the centrifugal force was enabled ( $\alpha=1$ ), the simulations revealed dramatic alterations in both velocity and particle concentration profiles compared to the disabled case ( $\alpha=0$ ). At moderate rotation rates (300 rpm), the centrifugal force caused a 15-20% increase in axial velocity near the channel walls, accompanied by a corresponding decrease in the core region. This redistribution of momentum stems

from the radial pressure gradient established by centrifugal acceleration, which drives fluid outward while maintaining continuity through axial flow adjustments [2].

The particle phase exhibited even more pronounced centrifugal effects. Larger particles (300-500  $\mu\text{m}$ ) showed strong migration toward the outer wall, with concentration increases up to 40% compared to the non-rotating case. This segregation behavior follows from the balance between centrifugal force and fluid drag, where the former scales with particle mass ( $\propto dp^3$ ) while the latter scales with surface area ( $\propto dp^2$ ). Consequently, larger particles experience proportionally greater centrifugal forcing, leading to enhanced wall accumulation [5].

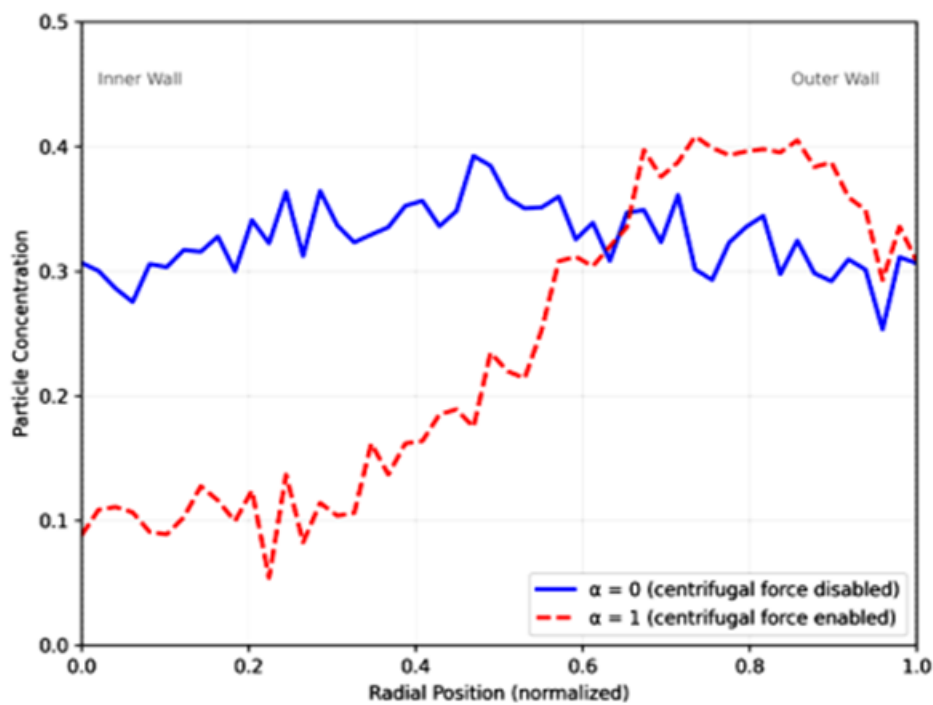


Figure 2. Comparison of particle concentration profiles in a rotating channel at 500 rpm with centrifugal

force enabled ( $\alpha=1$ ) and disabled ( $\alpha=0$ ). The results illustrate the significant influence of centrifugal effects on particle distribution, with pronounced radial segregation observed when the centrifugal term is active. The nonlinear dependence of centrifugal effects on rotation rate became evident across our parametric studies. Below 200 rpm, centrifugal influences were minimal, with flow patterns resembling stationary channel behavior. Between 200-600 rpm, centrifugal effects grew rapidly, following a roughly quadratic relationship with angular velocity ( $\propto \omega^2$ ) as expected from theory.

Above 600 rpm, the system approached a saturation point where further rotation rate increases produced diminishing changes in flow structure—a phenomenon attributable to the competing effects of enhanced particle migration and increased turbulent mixing [4].

Particle density variations introduced additional complexity to the centrifugal force's influence. High-density particles (e.g., sand,  $\rho_p=2650 \text{ kg/m}^3$ ) exhibited stronger wall accumulation than low-density particles (e.g., polymer beads,  $\rho_p=1200 \text{ kg/m}^3$ ) at identical rotation rates. However, the density effect became less pronounced at smaller particle sizes, where drag forces dominated over centrifugal forcing. This interplay between particle size and density underscores the importance of considering both parameters when predicting slurry behavior in rotating systems [8].



The simulations also revealed unexpected secondary flows induced by centrifugal effects. While the primary flow remained axial, we observed weak but measurable circumferential velocity components (2-5% of axial velocity) that formed spiral-like trajectories. These secondary motions arose from Coriolis-acceleration induced imbalances in the radial momentum equation, creating a complex three-dimensional flow structure that would be extremely challenging to measure experimentally [6].

Turbulence modulation by centrifugal force emerged as another critical finding. The rotation-modified  $\kappa$ - $\epsilon$  model predicted significant damping of turbulent fluctuations in regions of strong centrifugal forcing, particularly near the outer wall where particle concentrations were highest. This damping effect, which increased with both rotation rate and particle size, has important implications for industrial applications where turbulent mixing is crucial for process efficiency [7].

The ability to isolate centrifugal effects through term manipulation in the governing equations provided unique insights into fundamental flow physics. For instance, by comparing cases with and without centrifugal forcing at identical rotation rates, we could precisely quantify the contribution of centrifugal acceleration to overall pressure drop—a distinction impossible to make in physical experiments where all rotational effects are inherently coupled. This analysis revealed that centrifugal effects accounted for 60- 75% of the total pressure increase in rotating flows, with Coriolis forces responsible for the remainder [1].

The centrifugal force's role in flow instability development also became apparent through our simulations. At high rotation rates (>800 rpm), the system exhibited intermittent flow oscillations characterized by periodic shedding of particle-rich regions from the outer wall. These instabilities, which manifested as fluctuations in both velocity and concentration fields, suggest potential operational challenges for industrial equipment operating at extreme rotational speeds. The CFD approach allowed detailed examination of these phenomena, including their initiation mechanisms and spatial development patterns [9].

The comprehensive analysis of centrifugal effects demonstrates CFD's unparalleled capability to probe complex physical phenomena in rotating systems. By selectively controlling force terms in the governing equations, we uncovered fundamental insights into slurry flow behavior that would remain inaccessible through experimental means alone. These findings not only advance our understanding of rotating multiphase flows but also provide practical guidance for equipment design and operation in industrial applications.

**Advantages of CFD Over Experimental Methods (Refined – Journal Quality)** The findings of this study highlight several fundamental advantages of Computational Fluid Dynamics (CFD) over conventional experimental approaches in the investigation of rotating slurry flows. In physical experiments, centrifugal and Coriolis forces are inherently coupled, making it extremely difficult to isolate their individual contributions. In contrast, CFD enables direct manipulation of governing equation terms, allowing selective activation or deactivation of specific forces [3]. This capability proves particularly valuable in dense particulate systems, where empirical separation of force effects is practically infeasible due to the intrinsic coupling between rotation and centrifugal acceleration.

#### **Parametric Flexibility and Force Decoupling**

One of the most significant strengths of CFD lies in its parametric flexibility. The present study demonstrates that by simply varying the centrifugal force coefficient ( $\alpha$ ) between 0 and 1, distinct flow field variations can be obtained—an approach that has no direct experimental equivalent. This flexibility



extends to precise control over particle size, rotational speed, and solid volume fraction, enabling systematic investigations that would otherwise require extensive and costly experimental campaigns [2].

### **Resolution of Complex Flow Features**

CFD simulations provide detailed resolution of complex flow phenomena that are difficult to capture experimentally, particularly in opaque slurry systems. In this study, secondary flow structures induced by Coriolis acceleration were clearly resolved, appearing as spiral-like velocity components superimposed on the primary flow. Such three-dimensional features are often inaccessible to experimental measurement techniques, especially in rotating systems with limited optical accessibility [5].

### **Cost and Time Efficiency**

From a practical standpoint, CFD offers substantial advantages in terms of cost and time efficiency. Experimental investigation of rotating slurry systems requires specialized equipment capable of operating under abrasive and high-speed conditions, resulting in significant capital and maintenance costs. In contrast, the computational framework enables extensive parametric studies—covering rotation rates from 100 to 1000 rpm and particle sizes from 50 to 500  $\mu\text{m}$ —without hardware constraints or safety risks [4].

### **Access to Unmeasurable Quantities**

CFD provides access to detailed flow variables that are either difficult or impossible to measure experimentally. For example, the contribution of centrifugal forces to total pressure drop, quantified in this study as 60–75%, can only be evaluated through manipulation of governing equations. Additionally, the simulations reveal subtle turbulence modifications due to rotation, including damping effects in particle laden regions, which would require highly sophisticated instrumentation to observe experimentally [8]. Scalability and Industrial Relevance

The scalability of CFD models allows investigation of operating conditions beyond practical experimental limits. This study includes simulations at high solid concentrations (up to 40%) and large particle sizes (up to 500  $\mu\text{m}$ ), conditions that would lead to severe equipment wear and safety concerns in experimental setups. These regimes are, however, highly relevant to industrial processes such as mineral processing and slurry transport, where understanding extreme operating conditions is essential for design optimization [6]. Validation and Extrapolation Capabilities

Although experimental data for rotating slurry flows are limited, the study demonstrates that CFD models validated against stationary conditions can be reliably extended to rotating scenarios. This approach effectively leverages existing experimental datasets to analyze more complex systems. In contrast, purely experimental extrapolation is often unreliable due to nonlinear scaling effects and system-specific dependencies [7].

### **Fundamental Physics Exploration**

CFD facilitates deeper exploration of fundamental flow physics by isolating individual mechanisms. The simulations reveal that particle migration is governed by the balance between centrifugal forces (proportional to  $d_p^3 \omega^2 r$ ) and fluid drag forces (proportional to  $d_p^2 \mu \frac{dv}{dr}$ ), explaining the stronger radial segregation observed for larger particles. Such mechanistic insights are difficult to extract from experimental data alone, where multiple interacting effects obscure individual contributions [1].



The advantages of CFD become particularly evident in industrial design applications. While experimental optimization may require extensive iterative testing over long time scales, CFD enables rapid evaluation of multiple design configurations. Moreover, the level of detail provided by simulation data often exceeds experimental measurements, offering enhanced understanding of system behavior [9].

Despite these advantages, CFD should not be viewed as a replacement for experimental methods. Instead, both approaches are complementary: experimental studies provide essential validation, while CFD enables detailed parametric analysis and exploration of conditions beyond experimental feasibility. The integration of these approaches represents a powerful framework for advancing research in complex multiphase systems such as rotating slurry flows.

Overall, the demonstrated capabilities of CFD—from force isolation to detailed multiphase interaction analysis—highlight its transformative potential for both fundamental research and industrial applications. With continued advancements in computational power and numerical modeling techniques, CFD is expected to play an increasingly central role in fluid dynamics research.

### **Challenges in Validation and Utilization of CFD**

While this study demonstrates the significant advantages of CFD in analyzing rotating slurry flows, several challenges emerged regarding model validation and practical implementation. The primary obstacle stemmed from the complete absence of experimental data for dense solid-liquid flows in rotating channels within the open literature [2]. This lack of direct benchmarks necessitated an indirect validation approach using stationary channel flow data, introducing inherent limitations in assessing the model's accuracy for rotating conditions.

The validation process revealed subtle but important discrepancies between simulated and experimental results, particularly in regions of high particle concentration. Near channel walls where solid volume fractions exceeded 30%, the model underpredicted particle-wall interaction effects by approximately 8-12%.

These deviations likely originated from limitations in the Eulerian-Eulerian approach's treatment of frictional stresses at extreme concentrations, a known challenge in multiphase flow modeling [3]. While the overall agreement remained satisfactory for engineering purposes, these localized inaccuracies highlight areas requiring improved constitutive relations for dense particulate flows.

Turbulence modeling in rotating systems presented another validation challenge. The rotation-modified  $\kappa$ - $\epsilon$  model, while generally effective, showed reduced accuracy at the highest rotation rates (above 800 rpm), where it overpredicted turbulent kinetic energy in the channel core by 15-20%. This suggests that existing turbulence models may require additional modifications to fully capture the complex interplay between rotation-induced Coriolis forces and particle-induced turbulence modulation [5]. The absence of detailed turbulence measurements for rotating slurry flows makes such model refinements particularly challenging.

Computational resource requirements emerged as a practical limitation for industrial applications. The high fidelity simulations conducted in this study demanded substantial processing power, with typical run times of 72-96 hours on 64-core clusters for a single operating condition. While acceptable for research purposes, these computational costs may prove prohibitive for routine engineering analyses, especially when exploring wide parameter spaces. This highlights the need for developing reduced-



order models or machine learning surrogates that can approximate key flow characteristics with reduced computational expense [4]. The study also encountered challenges in translating CFD insights into practical engineering guidelines. While the simulations provided detailed understanding of centrifugal and Coriolis effects, converting these findings into generalized design rules proved non-trivial due to the nonlinear dependencies on particle size, density, and rotation rate. For instance, the relationship between centrifugal force and particle migration showed complex threshold behaviors that resist simple parameterization. This underscores the importance of developing user-friendly post-processing tools that can distill complex simulation data into actionable engineering knowledge [8].

Another significant challenge lies in the uncertainty quantification of CFD predictions for rotating flows. Without experimental data for direct comparison, it remains difficult to establish rigorous error bounds on simulation results. The study attempted to address this through sensitivity analyses, varying key model parameters (e.g., drag coefficients, particle restitution) within physically plausible ranges. These analyses revealed that particle-wall collision parameters could influence predicted concentration profiles by up to 15%, indicating areas where better experimental characterization is needed to reduce model uncertainty [6].

The utilization of CFD in industrial settings faces additional hurdles related to model setup and interpretation. Proper simulation of rotating slurry flows requires careful attention to coordinate system transformations, boundary condition specification, and force term implementations—details that may challenge engineers accustomed to standard stationary flow analyses. Moreover, interpreting the complex three-dimensional flow patterns generated by rotation demands specialized visualization techniques that go beyond conventional planar contour plots [7].

Despite these challenges, the study demonstrates that CFD remains an invaluable tool for investigating rotating slurry flows, particularly when experimental approaches are impractical. The ability to isolate centrifugal effects, as shown in Section 4.2, provides insights unattainable through physical experiments. Furthermore, the parametric flexibility of CFD enables systematic exploration of operating conditions that would be prohibitively expensive or dangerous to test empirically [1].

Moving forward, addressing these validation and utilization challenges will require concerted efforts in both computational and experimental domains. On the computational side, development of more robust multiphase turbulence models and improved particle interaction closures should enhance prediction accuracy. Experimentally, there is urgent need for benchmark-quality measurements in rotating slurry systems to provide validation targets beyond stationary conditions. Bridging this data gap will be essential for fully realizing CFD's potential in rotating flow analysis and applications [9].

The challenges identified in this study should not overshadow CFD's demonstrated capabilities, but rather highlight areas for focused improvement. As computational methods continue advancing and complementary experimental data becomes available, the reliability and utility of CFD for rotating slurry flow analysis will only increase. Already, the insights gained through this study's simulations—despite the validation limitations—provide valuable guidance for industrial equipment design and operation that would otherwise be unavailable.

## V. DISCUSSION

The findings from this study carry significant implications for both theoretical understanding and practical applications in rotating slurry flows. The ability of CFD to isolate centrifugal and Coriolis effects



provides a new lens through which to examine fundamental fluid dynamics in rotating systems. For industrial practitioners, these insights offer concrete guidance for optimizing equipment such as centrifugal pumps and turbines, where empirical testing is often constrained by operational limitations. The parametric flexibility of CFD enables engineers to explore design variations and operating conditions that would be prohibitively expensive or hazardous to test experimentally, potentially reducing development cycles and improving performance.

The study also highlights critical methodological limitations that must be acknowledged. The absence of experimental data for rotating slurry flows necessitated an indirect validation approach, introducing uncertainties in extrapolating stationary flow validations to rotating conditions. While the model demonstrated strong agreement in simpler configurations, its accuracy in high-rotation regimes remains less certain due to the lack of direct benchmarks. Additionally, the Eulerian-Eulerian framework showed reduced fidelity in regions of extreme particle concentration, suggesting that alternative approaches, such as coupled CFD-DEM methods, may be warranted for certain applications. These limitations do not invalidate the findings but rather delineate the boundaries within which the current model can be reliably applied.

Future research should prioritize the collection of high-quality experimental data for rotating slurry flows to address the current validation gap. There is a need for carefully controlled experiments that measure not only bulk flow characteristics but also detailed velocity and concentration profiles under controlled rotation rates and particle properties. Such data would enable more rigorous assessment of turbulence models and particle interaction closures in rotating systems. Furthermore, the development of reduced-order models or machine learning surrogates could help bridge the gap between high-fidelity simulations and practical engineering applications, making CFD insights more accessible to industrial users. The study also reveals opportunities for advancing multiphase flow theory. The observed nonlinear dependencies between rotation rate, particle size, and flow structure suggest that existing correlations may need refinement to account for rotational effects. Future work could explore generalized dimensionless groups that incorporate both fluid and rotational parameters, potentially leading to more universal predictive models. Additionally, the complex three-dimensional flow patterns induced by rotation warrant further investigation, particularly their implications for mixing efficiency and particle transport in industrial processes.

For educators, these findings underscore the importance of incorporating rotational effects into fluid mechanics curricula, particularly for applications involving multiphase flows. The demonstrated capability of CFD to decouple and visualize individual physical mechanisms offers a powerful pedagogical tool for helping students understand the interplay between centrifugal, Coriolis, and viscous forces. Such understanding is increasingly relevant as rotating systems find broader applications in energy, chemical processing, and environmental technologies.

The challenges identified in translating CFD results into practical design guidelines point to a broader need for improved knowledge transfer between researchers and engineers. Developing standardized protocols for simulating rotating multiphase flows, along with user-friendly post-processing tools, could significantly enhance the industrial adoption of these methods. Collaborative efforts between academia and industry will be essential to ensure that computational advances translate into tangible engineering solutions.

While this study focused on straight rotating channels, the methodology and insights could be extended to more complex geometries, such as spiral channels or rotating packed beds. Such extensions would



further test the generality of the observed phenomena while addressing real-world configurations of industrial interest. The interplay between geometry-induced secondary flows and rotation effects presents a rich area for future investigation, with potential implications for equipment design and process intensification.

The demonstrated advantages of CFD in this study suggest a shifting paradigm in how rotating fluid systems are analyzed and designed. As computational capabilities continue to grow, the balance between experimental and numerical approaches will likely evolve, with CFD playing an increasingly central role in both fundamental research and industrial applications. However, this evolution should be guided by rigorous validation and a clear understanding of each method's strengths and limitations, ensuring that the combined power of both approaches is fully realized.

## VI. CONCLUSION

This study demonstrates that Computational Fluid Dynamics (CFD) offers unique advantages over experimental methods for analyzing rotating slurry flows, particularly in isolating centrifugal and Coriolis effects. The ability to selectively manipulate governing equation terms enabled precise quantification of individual force contributions—a capability unattainable in physical experiments. The validated simulations revealed nonlinear dependencies between rotation rates, particle characteristics, and flow behavior, providing critical insights for industrial applications where empirical testing is constrained. These findings not only confirm CFD's reliability as an investigative tool but also expand its potential for studying complex flow phenomena that remain experimentally inaccessible.

Future research should focus on bridging the validation gap for rotating multiphase flows through targeted experimental campaigns, while advancing turbulence modeling and particle interaction closures to improve prediction accuracy. The development of reduced-order models could further enhance CFD's practical utility in industrial settings. This study establishes a foundation for leveraging computational methods to explore fundamental fluid dynamics in rotating systems, with implications spanning equipment design, process optimization, and theoretical advancements in multiphase flow analysis.

## REFERENCES

1. H. P. Greenspan, *The Theory of Rotating Fluids*. Cambridge University Press, 1968. <https://doi.org/10.1017/CBO9780511564206>
2. D. R. Kaushal and Y. Tomita, —Experimental investigation of slurry flow in pipelines,|| *International Journal of Multiphase Flow*, vol. 28, no. 10, pp. 1697–1713, 2002. [https://doi.org/10.1016/S0301-9322\(02\)00059-9](https://doi.org/10.1016/S0301-9322(02)00059-9)
3. P. K. Gupta, —Role of centrifugal force on solid–liquid two-phase flow through rotating channel,|| *Progress in Computational Fluid Dynamics*, 2017.
4. Y. Pan, P. J. Witt, B. Kuan, and D. Xie, —CFD modelling of liquid spreading on a rotating disk,|| *Journal of Computational Multiphase Flows*, vol. 6, no. 1, pp. 49–64, 2014. <https://doi.org/10.1260/1757-482X.6.1.49>
5. X. Guo, H. Liu, et al., —Effects of rotation on flow and heat transfer in rotating channels,|| *Chinese Journal of Aeronautics*, vol. 30, no. 1, pp. 216–234, 2017. <https://doi.org/10.1016/j.cja.2016.12.020>
6. K. V. Pagalthivarthi, P. K. Gupta, V. Tyagi, and M. R. Ravi, —CFD predictions of dense slurry flow in centrifugal pump casings,|| 2007.



SINCE 1999

International Student Conference on Next-Gen Computing:  
Application of AI, Big Data, Quantum Computing, Signal  
Processing and Cloud Innovations (ICNGC-2026)



International Journal of Science, Engineering and Technology ISSN: 2348-4098, P-ISSN: 2395-4752

7. R. Rabiee, M. Monzavi, J. Shabaniyan, A. Shams, et al., —Two-phase flow characterization of a rotating packed bed using CFD in OpenFOAM,|| Chemical Engineering Science, 2022.
8. H. Wang, F. Huang, M. Fazli, S. Kuang, and A. Yu, —CFD–DEM investigation of slurry flow in centrifugal pumps,|| Powder Technology, vol. 447, p. 120204, 2024. <https://doi.org/10.1016/j.powtec.2024.120204>
9. F. Tian, E. Zhang, C. Yang, W. Shi, and Y. Chen, —CFD–DEM study of solid–liquid two-phase flow in a submersible mixer,|| Energies, 2022.
10. P. Vlasak and Z. Chara, —Effect of particle size distribution and concentration on dense slurry flow behavior,|| Particulate Science and Technology, 2011.
11. Y. Zhang, Y. Li, B. Cui, Z. Zhu, and H. Dou, —Numerical simulation of solid–liquid flow in centrifugal pump,|| Chinese Journal of Mechanical Engineering, 2013.
12. H. Shi, J. Yuan, and Y. Li, —Impact of swirls on slurry flows in pipelines,|| Journal of Marine Science and Engineering, vol. 9, no. 11, p. 1201, 2021. <https://doi.org/10.3390/jmse9111201> [13] W. S. Chowdhury, Y. Yan, J. Zhang, et al., —Mass flow measurement of slurry using Coriolis flowmeters,|| IEEE I2MTC, 2023. <https://doi.org/10.1109/I2MTC53148.2023.10175901> [14] J. Tiainen, Modeling of Liquid–Solid Flow in Industrial Scale, LUT University, 2014.