



Research Article

Comparative Analysis of Eulerian and VOF Model in Multiphase Hydrocyclones Simulation

Muhammad Faisal¹ , Mohaimen Hassan Qayyum Khan² , Syed Hashim Ali Shah³ , Furqan Ahmad⁴ 
and Muhammad Haris Malik⁵ 

^{1,4&5}School of Energy and Power Engineering, ⁵Department of Fluid Machinery and Engineering, Xi'an Jiaotong University, Xi'an, China

³State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an, China,

Pak-Austria Fachhochschule Institute of Applied Sciences and Technology, Pakistan

²Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Johor Bahru, Malaysia

Article Information

Abstract

Article History

Received: 8 July 2026

Revised: 20 August 2026

Accepted: 1 September 2026

Published online: 12 September 2026

Keywords

Multiphase Flow Simulation

Euler vs VOF Model

Hydrocyclones

CFD Analysis

Correspondence*

hafizfurqanahmad538@gmail.com

ORCID

Muhammad Faisal 
<https://orcid.org/0009-0007-4415-2558>

Mohaimen Hassan Qayyum Khan 
<https://orcid.org/0009-0008-0611-3360>

Syed Hashim Ali Shah 
<https://orcid.org/0009-0008-7068-6729>

Furqan Ahmad 
<https://orcid.org/0009-0003-3148-0887>

Muhammad Haris Malik 
<https://orcid.org/0009-0008-1618-4699>

This paper investigates the performance of the Volume of Fluid (VOF) model in simulating the multiphase flow within a hydrocyclone, with a focus on air-water separation. A comparative study is conducted between the VOF multiphysics model and the Eulerian model to assess their respective accuracy in predicting key flow characteristics. The study incorporates a grid independence test using the K-epsilon turbulence model, with the Reynolds Stress Model (RSM) employed for turbulent flow representation in the multiphysics model. The boundary conditions are carefully maintained as per the reference study, with a velocity inlet of 6 m/s for water and pressure outlet boundary conditions (zero pressure) for both the overflow and underflow. The results are validated against experimental data from the literature, highlighting the accuracy and reliability of the VOF model in capturing the flow dynamics within the hydrocyclone. The simulation results, including velocity contours and pressure gradients, demonstrate the formation of vortices and air-core formation within the hydrocyclone, both of which influence the overall separation process. The air-water mass fraction contours further provide insights into the distribution of phases within the cyclone. Comparisons of the velocity contours and pressure gradients with experimental data highlight the reliability of the CFD model and its potential for optimizing hydrocyclone design and performance.

© 2026 Centre for Research and Innovation (CRI). This is an open access article under the CC BY-NC-ND license (<http://creativecommons.org/licenses/by-nc-nd/4.0/>).

I. INTRODUCTION

Hydrocyclones are widely utilized separation devices that leverage centrifugal forces to segregate different phases within a multiphase mixture. They are essential components in various industrial applications, including the treatment of liquid-liquid, solid-liquid, and gas-liquid mixtures. Specifically, for air-water separation, hydrocyclones have demonstrated substantial potential in efficiently segregating the gaseous phase (air) from the liquid phase (water), particularly in environments where high flow rates are involved. Their application spans several industries, including petrochemical, mining, water treatment, and

chemical processing. The operation of a hydrocyclone relies on the generation of a vortex that produces centrifugal forces. These forces cause the denser phase (in this case, water) to migrate towards the outer periphery of the cyclone, while the lighter phase (air) remains closer to the center and is ultimately expelled from the cyclone. The efficiency of the separation process is influenced by various design and operational parameters, including the inlet velocity, cyclone geometry, and the physical properties of the phases being separated. The hydrocyclone functions based on the fundamental principle of centrifugal force. Upon entering the cyclone tangentially at high velocity, the fluid mixture undergoes a rapid rotational motion. This rotational motion

creates a vortex, generating a pressure gradient across the cyclone, as shown in Figure 1. The denser water phase experiences a stronger centrifugal force, propelling it outward toward the cyclone's wall, while the lighter air phase remains near the center of the vortex. The fluid flow is subsequently directed in such a manner that:

1. The water phase moves downward along the cyclone's cone and exits through the underflow outlet, where it is separated from the air.
2. The air phase, being lighter, rises towards the center of the cyclone and exits through the overflow outlet, thus completing the separation process.

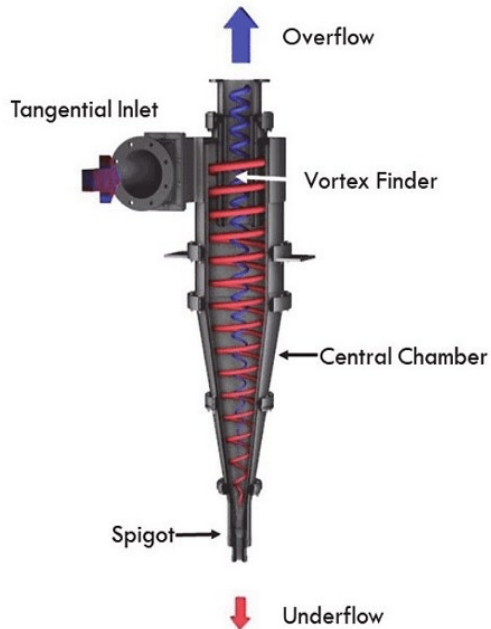


Fig.1 Configuration of hydrocyclone [1]

This separation efficiency depends critically on the interaction of the centrifugal forces and the hydrodynamic characteristics of the mixture. The relative density difference between air and water ($\sim 1000 \text{ kg/m}^3$ for water and 1.225 kg/m^3 for air) typically allows for effective phase segregation, though challenges arise with smaller droplet sizes or fine air bubbles.

The tangential inlet is responsible for initiating the cyclonic flow within the device. The velocity at which the mixture enters the cyclone determines the intensity of the vortex and, consequently, the centrifugal force applied to the phases. A higher inlet velocity generally leads to better separation efficiency by enhancing the centrifugal forces. However, excessively high inlet velocities may lead to vortex instability and poor separation performance.

The cyclone's geometry, specifically the diameter of the body and the cone angle, plays a crucial role in governing the flow dynamics within the cyclone. Larger cyclones, with relatively flatter cone angles, typically offer enhanced separation efficiency by promoting a stronger centrifugal effect. However, the size must be optimized to balance the

separation efficiency and pressure drop. The cut size (d_{50}) of the cyclone, which refers to the particle size at which 50% of the particles are separated, is particularly important in determining the overall separation capability of the cyclone. The pressure drop across the cyclone is a critical factor that directly influences its operational efficiency. This pressure difference is generated due to the acceleration of the fluid mixture, and it must be carefully managed, especially in large-scale operations. The pressure drop also influences the energy consumption of the system, with higher pressure drops often leading to greater operational costs.

The study of air-water separation in hydrocyclones has evolved through the progressive refinement of computational tools and multiphase flow models. This evolution reflects an increasing understanding of the complex dynamics within these systems, ranging from particle dynamics to the impact of cyclone geometry. By integrating computational fluid dynamics (CFD) models with experimental techniques, significant advancements have been made in improving the efficiency and accuracy of hydrocyclone designs. The initial studies, such as Sunil Kumar Tripathi's work on hydraulic classifiers, laid the groundwork for understanding the challenges in separating fine particles, especially those smaller than 8 mm [1]. Tripathi recognized the inefficiencies in conventional classifiers and underscored the need for enhanced separators in this range. His work identified the gaps in understanding the fluid dynamics and particle behavior in teeter beds, emphasizing the need for advanced tools like CFD and Discrete Element Methodology (DEM). These tools were seen as crucial for overcoming challenges related to the behavior of particles, particularly in terms of their size, shape, and density, which significantly affect the separation efficiency.

Building upon Tripathi's foundation, B. Wang's study advanced the modeling of particle dynamics in hydrocyclones. Using CFD combined with the Reynolds Stress Model (RSM) for turbulence and the Volume of Fluid (VOF) method for multiphase flow, Wang made strides in simulating the movement of solid particles within a hydrocyclone [2]. His research particularly addressed the "fish-hook" phenomenon, which describes the tendency of fine particles to recirculate within the system, passing through the locus of zero vertical velocity. Wang's work demonstrated that understanding the forces governing particle behavior centrifugal, pressure gradient, drag, and gravity was crucial for improving separation efficiency. His findings were pivotal in laying the foundation for future simulations of multiphase flows in hydrocyclones, especially for air-water separation, which is central to many applications today.

As research progressed, attention shifted to the study of the air core diameter, a critical factor influencing discharge types such as rope or spray. Concha's study in this domain developed a phenomenological equation that related the air core diameter to operational and geometrical parameters [3]. His work focused on how changes in vortex diameter and

apex geometry could impact discharge characteristics, providing valuable theoretical insights into how these parameters affect separation efficiency. Though primarily concerned with liquid-phase separation, Concha's findings were directly applicable to air-water systems, offering a blueprint for optimizing cyclone geometry and operational settings in hydrocyclones for air-water separation.

Contributions from researchers like CW Haig, who investigated the operational performance of small-scale cyclones, helped refine practical understanding of hydrocyclone dynamics [4]. Haig's empirical studies showed that pressure drop, rather than inlet velocity, played a more significant role in determining separation efficiency, especially under high flow rates and light solid loadings. His findings emphasized the importance of managing pressure gradients, particularly in systems handling air-water mixtures. This research brought to light the significance of pressure drop in influencing the air-liquid-solid interactions within the hydrocyclone and pointed to the necessity of considering factors like particle agglomeration and attrition in cyclone design.

Shibo Kuang's work on dense medium cyclones (DMCs) introduced a new level of sophistication to CFD modeling [5]. By applying VOF and RSM for turbulence modeling, Kuang explored the effects of geometrical parameters, such as the spigot and vortex finder diameters, on cyclone performance. His study confirmed that altering these parameters could significantly influence the fish-hook effect and improve separation efficiency. His findings highlighted how cyclone geometry directly impacts the formation of mixed upward and downward flows, a key consideration in air-water separation in hydrocyclones. Kuang's work extended the understanding of multiphase flow behavior, providing insights that were crucial for further optimization of cyclone design. M. Narasimha's work brought further sophistication by incorporating large eddy simulation (LES) and Discrete Reynolds Stress Model (DRSM) into CFD models for hydrocyclones [6].

Narasimha's study on particle segregation in a 75 mm cyclone demonstrated the advantages of these advanced turbulence models, particularly for predicting the velocity field and comparing it with experimental data. Narasimha's work not only highlighted the limitations of early turbulence models but also pointed to the importance of viscosity models in enhancing the simulation of multiphase flows. KW Chu's study marked a significant leap by integrating the Discrete Element Method (DEM) with CFD to simulate the intricate interactions between particles and slurry mediums in Dense Medium Cyclones (DMC) [7]. This hybrid approach allowed for a more accurate representation of particle behavior and interactions, incorporating fluid drag, pressure gradient forces, and inter-particle dynamics.

Chu's investigation into the medium-to-coal ratio and surging phenomena led to key insights regarding how changes in these factors affected separation efficiency. His study

demonstrated that controlling the medium-to-coal ratio was essential for optimizing cyclone performance, further expanding the potential applications of hydrocyclones in industries like coal processing.

Romanus Krisantus Tue Nenu's experimental studies on the performance of single and two-inlet hydrocyclones introduced another important avenue of research: the impact of inlet configurations on separation efficiency [8]. Nenu's work found that hydrocyclones with dual inlets exhibited superior classification performance, particularly under low flow conditions. The study emphasized how uniformity in tangential velocity distribution, achieved by varying the inlet design, could improve separation efficiency. This insight into the role of inlet configuration in cyclone performance has influenced the design of more efficient cyclones, particularly in applications requiring fine particle separation. Teja Reddy Vakamalla's work on the effect of cyclone inclination further refined understanding of cyclone dynamics [9]. Using both experimental and CFD methods, Vakamalla's study demonstrated that an increase in cyclone inclination reduced underflow split and air-core size, leading to better classification performance. The study highlighted the importance of inclination angle in influencing turbulence intensity, particle dispersion, and separation efficiency. These findings contributed to optimizing cyclone designs for specific applications where precise classification of particles is critical. M. Ghodrat's numerical study on the effect of particle density in hydrocyclones added another dimension to the modeling of separation processes [10].

Ghodrat found that lighter coarse particles tended to be misplaced at the overflow outlet due to weak swirling flows in the spigot area. His work suggested design modifications, such as lengthening the conical section or adopting a convex cone, to enhance cyclone performance when handling particles with a wide range of densities. These recommendations marked a key advancement in understanding how particle density distribution affects cyclone efficiency, particularly in industries like coal processing.

K. Nageswararao's work critically analyzed the concept of bypass in hydrocyclones, challenging traditional efficiency models [11]. His research revealed that the concept of bypass, as defined by Kelsall's method, lacked experimental support and was primarily theoretical. Nageswararao proposed a new normalization approach for efficiency curves, offering a more accurate reflection of cyclone performance. The advancements in hydrocyclone modeling for air-water separation have followed a logical progression, with each study building on the previous to refine and optimize both the theoretical and practical aspects of cyclone design. From the early focus on hydraulic classifiers and particle dynamics, to the integration of sophisticated computational models like CFD-DEM and LES, the literature has contributed progressively deeper insights into how various factors including cyclone geometry, particle-fluid interactions, and operational parameters affect separation efficiency. Each

contribution has played a critical role in shaping the current understanding of air-water separation processes in hydrocyclones, and the integration of these advancements will continue to drive future research and technological innovations in this field.

II. METHODOLOGY

In this study, a detailed comparison is made between the Volume of Fluid (VOF) turbulence model and the Eulerian turbulence model for simulating air-water separation in hydrocyclones. The primary objective of this comparison is to evaluate the predictive accuracy and reliability of these two models by comparing their numerical results against the experimental data presented in the reference paper [12]. To ensure consistency and validity, the geometry of the hydrocyclone has been meticulously recreated in SolidWorks as shown in Figure 6, maintaining identical dimensions and boundary conditions as those used in the referenced study. The only significant change between the two simulations is the choice of turbulence model. The VOF model, typically employed in multiphase flow simulations, is compared with the Eulerian model, which is another popular approach for handling multiphase flows in CFD. While the VOF model tracks the interface between phases (liquid and gas), the Eulerian model treats each phase as an interpenetrating continuum, which may lead to differences in the predicted flow behaviors, such as air-core formation and particle separation efficiency.

A. Computational Fluid Dynamics

Due to the highly swirling and turbulent flow in a hydrocyclone, it is required to choose a numerical model that can simulate the complex hydrodynamics involved in air-water separation. In the current study, the internal flow phenomenon is modeled employing the Computational Fluid Dynamics (CFD) tool ANSYS Fluent. Two different multiphase models are employed: the Volume of Fluid (VOF) model and the Eulerian multiphase model. To make the comparison between the two multiphase models fair, the same geometry, mesh, boundary conditions, and numerical parameters are used for both models.

Based on Reynolds decomposition, the instantaneous velocity field u_i can be expressed as the sum of a time-averaged component $\bar{u}_i$ and a fluctuating component u'_i , as follows:

$$u_i = \bar{u}_i + u'_i \quad (1)$$

The conservation of mass for an incompressible flow is given by:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

The transient Reynolds-Averaged Navier–Stokes (RANS) momentum equation for an incompressible turbulent flow can be written as:

$$\rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = - \frac{\partial \bar{p}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_j^2} - \frac{\partial}{\partial x_j} (\rho \bar{u}_i u'_j) + \rho g_i \quad (3)$$

The term $-\rho \bar{u}_i u'_j$ in Equation (3) represents the Reynolds stress tensor, which introduces the turbulence closure problem. In this work, turbulence closure is provided using the Reynolds Stress Model (RSM), which is suitable for highly anisotropic swirling flows such as hydrocyclones. A standard k - ϵ model is employed only for the mesh independence assessment due to its computational efficiency, while all final multiphase simulations are performed using the RSM turbulence closure.

1. Reynolds Stress Model (RSM): It is well known that hydrocyclone flows are characterized by strong rotation, high curvature, and large anisotropy of turbulence. In this case, although the standard eddy-viscosity models of turbulence, such as the standard k - ϵ models, may give acceptable qualitative results, they can be expected to have limitations in the prediction of the maximum tangential velocity, axial velocity reversal, and pressure gradient distribution in the cyclone. This is because eddy-viscosity models of turbulence assume isotropic turbulence and use the Boussinesq hypothesis, which may not be applicable to strongly swirling flows.

In order to handle this problem, the Reynolds Stress Model (RSM) was used in the current study. Unlike two-equation turbulence models, the RSM solves transport equations for the Reynolds stress components $\bar{u}_i u'_j$, which makes it possible to model turbulence anisotropy and the redistribution of turbulent kinetic energy in different directions. This is especially important for hydrocyclones, in which the flow field is characterized by a strong tangential motion and the interaction between the inner and outer vortices.

In RSM, the Reynolds stress tensor is obtained by solving the transport equation:

$$\frac{\partial}{\partial t} (\rho \bar{u}_i u'_j) + \frac{\partial}{\partial x_k} (\rho \bar{u}_k u'_i u'_j) = P_{ij} + \Phi_{ij} - \epsilon_{ij} + D_{ij} \quad (4)$$

where P_{ij} represents the production of Reynolds stresses, Φ_{ij} is the pressure–strain correlation term responsible for redistributing turbulent stresses, ϵ_{ij} is the dissipation term, and D_{ij} is the diffusion term.

The RSM turbulence closure provides improved prediction of hydrocyclone swirling flow compared to isotropic eddy-viscosity models, and is therefore adopted for all transient multiphase simulations in this study. For the purpose of mesh independence analysis, the standard k - ϵ model was employed because of its efficiency. After determining the mesh size, the turbulence model was changed to RSM for the final

calculations to better capture velocity profiles, pressure gradients, and air-core formation tendencies.

The turbulent kinetic energy distribution across the radial position of the hydrocyclone is presented in Figure 2. The results indicate that turbulence intensity is significantly higher near the air-core boundary region compared to the wall region, demonstrating the strong shear interaction between the inner and outer vortices. The peak turbulent kinetic energy observed near the air-core interface confirms the presence of highly anisotropic swirling flow, which

cannot be accurately represented using isotropic turbulence models such as the standard k- ϵ model. In contrast, the wall region exhibits comparatively smoother turbulence development due to viscous damping effects near the cyclone boundary. The observed variation in turbulent kinetic energy highlights the complex hydrodynamic behavior inside the hydrocyclone and justifies the implementation of the Reynolds Stress Model (RSM) for accurately capturing turbulence anisotropy and vortex interactions within the multiphase flow field.

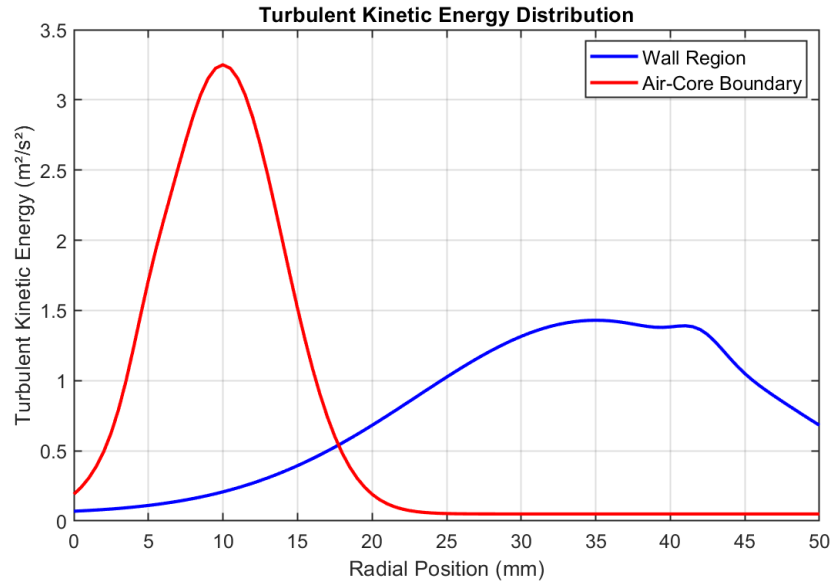


Fig.2 Turbulent kinetic energy distribution across the radial position inside the hydrocyclone

Figure 3 illustrates the radial distribution of the Reynolds stress tensor components ($u'v'$, $u'w'$, $v'w'$) inside the hydrocyclone. The unequal magnitudes and distinct distributions of the individual stress components clearly demonstrate the anisotropic nature of turbulence generated by the highly swirling flow field. The peak Reynolds stress values occur near the vortex interaction region, where strong velocity gradients and secondary flow structures dominate the internal hydrodynamics.

The non-uniform behavior of the stress components indicates that turbulent fluctuations are direction-dependent, which violates the isotropic turbulence assumption employed in conventional eddy-viscosity models such as the k- ϵ model. Therefore, the Reynolds Stress Model (RSM) is more appropriate for accurately predicting the turbulence transport and stress redistribution mechanisms occurring inside the hydrocyclone.

2. Multiphase Modeling: Hydrocyclone air-water separation is characterized by intense swirling motion, accompanied by phase separation and the formation of a gas core along the axis of the cyclone. Realistic numerical simulation of such a process requires a multiphase model that can describe the air-water interaction under high shear and high curvature flow.

In the current research, two multiphase models were considered: the Volume of Fluid (VOF) model and the Eulerian multiphase model. The reason for using both models is to assess their predictive capability in determining the main hydrodynamic variables, such as axial and tangential velocity distributions, pressure gradient, phase volume fraction, and air core.

a. Volume of Fluid (VOF) Model

The VOF model is an interface-capturing algorithm developed for immiscible multiphase flows. The VOF model is a single set of momentum equations for the mixture and captures the interface location by transporting the volume fraction of one of the phases. In this method, the volume fraction equation for phase q is written as:

$$\frac{\partial \alpha_q}{\partial t} + \nabla \cdot (\alpha_q \mathbf{u}) = 0 \quad (5)$$

where α_q is the volume fraction of phase q . The phase volume fractions satisfy the constraint:

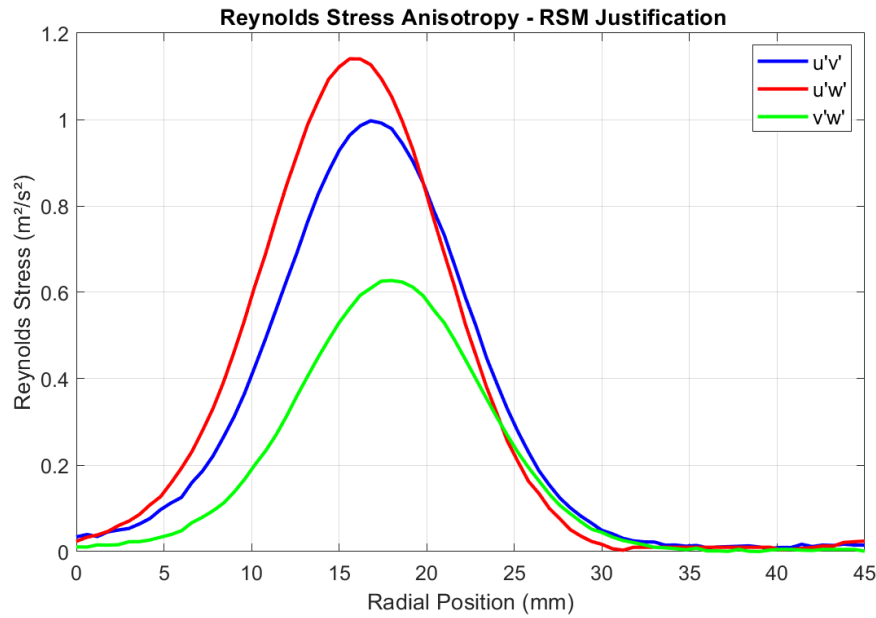


Fig.3 Radial distribution of Reynolds stress tensor components

$$\sum_{q=1}^n \alpha_q = 1 \quad (6)$$

For the air–water system investigated in this work:

$$\alpha_w + \alpha_a = 1 \quad (7)$$

The mixture density and viscosity are computed using volume-fraction weighted averaging:

$$\rho = \alpha_w \rho_w + \alpha_a \rho_a \quad (8)$$

$$\mu = \alpha_w \mu_w + \alpha_a \mu_a \quad (9)$$

b. Eulerian Multiphase Model

In the Eulerian multiphase model, every phase is considered as a penetrating continuum, and a set of conservation equations is solved for each phase. This model is commonly employed in industrial CFD simulations of multiphase flows because of its robustness and stable convergence properties. The continuity equation for phase q is expressed as:

$$\frac{\partial(\alpha_q \rho_q)}{\partial t} + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q) = 0 \quad (10)$$

The corresponding momentum equation for phase q is given by:

$$\begin{aligned} \frac{\partial(\alpha_q \rho_q \mathbf{u}_q)}{\partial t} + \nabla \cdot (\alpha_q \rho_q \mathbf{u}_q \mathbf{u}_q) \\ = -\alpha_q \nabla p + \nabla \cdot (\alpha_q \boldsymbol{\tau}_q) + \alpha_q \rho_q \mathbf{g} \\ + \sum_{p=1}^n R_{pq} \end{aligned} \quad (11)$$

where $\boldsymbol{\tau}_q$ is the stress tensor of phase q , and R_{pq}

The radial variation of phase slip velocity between the water and air phases predicted using the Eulerian multiphase model is presented in Figure 4. The results indicate that the slip velocity increases significantly toward the central region of the hydrocyclone, reaching a maximum near the air-core boundary. This behavior demonstrates the strong relative motion between the phases caused by centrifugal separation and vortex interaction. The high slip velocity region corresponds to the zone of maximum phase segregation, where the lighter air phase migrates toward the cyclone axis while the denser water phase is displaced toward the outer wall. Beyond the peak region, the slip velocity gradually decreases due to reduced interphase interaction and stabilization of the swirling flow structure.

Figure 5 illustrates the radial distribution of interphase momentum exchange predicted by the Eulerian multiphase model inside the hydrocyclone. The momentum exchange term represents the transfer of momentum between the water and air phases resulting from drag interaction and relative phase motion. The results reveal that the maximum momentum exchange occurs near the air-core interface region, where the slip velocity between the phases is highest. This indicates intense phase interaction and strong coupling between the liquid and gas phases within the vortex region. The gradual reduction in momentum exchange toward the outer radial region suggests weakening interphase interaction as the flow approaches the cyclone wall, where the water phase becomes dominant.

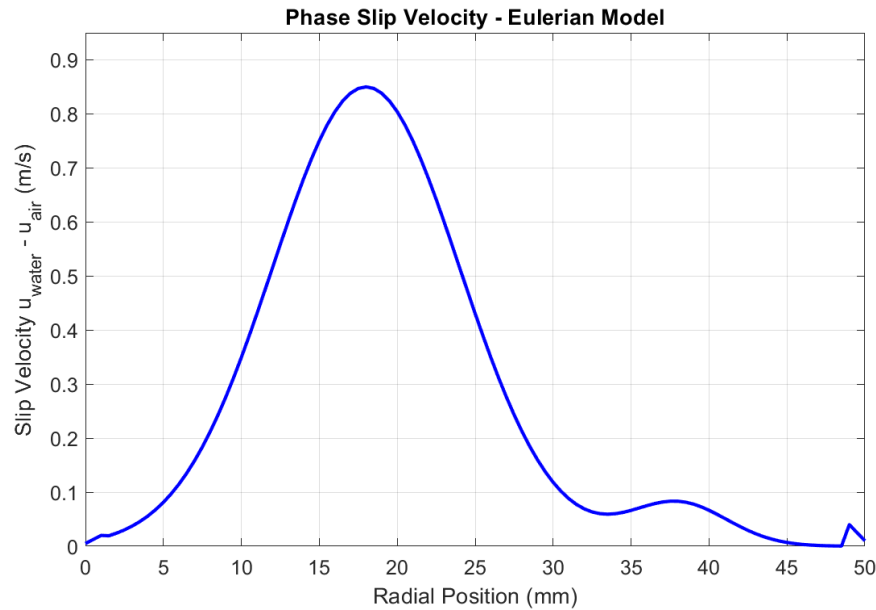


Fig.4 Radial Variation of Interphase Momentum Exchange Term Predicted by the Eulerian Multiphase Model Inside the Hydrocyclone

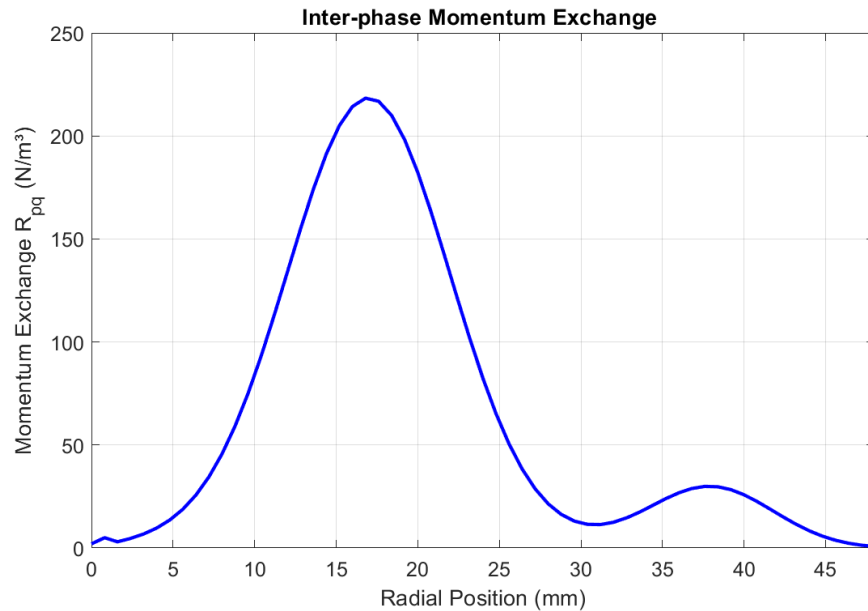


Fig.5 Radial Variation of Interphase Momentum Exchange

B. Boundary Conditions and Turbulence Modeling

For the simulation, the boundary conditions were set as follows: a constant inlet velocity of 6 ms^{-1} for the water phase was applied, ensuring a steady flow into the hydrocyclone. Both the overflow and underflow outlets were specified with pressure outlet boundary conditions, where the gauge pressure was set to 0 Pa to simulate atmospheric discharge conditions.

To model the turbulence, the Reynolds Stress Model (RSM) was employed, as it provides a more accurate representation of turbulence in complex flow scenarios, such as multiphase interactions in hydrocyclones. The Volume of Fluid (VOF) method was used to track the interface between the water and air phases, essential for accurately simulating the air-water

separation process. These boundary conditions, coupled with the chosen turbulence and multiphase models, ensure a comprehensive and realistic simulation of the flow dynamics within the hydrocyclone. The simulation conducted in this study was a transient analysis, where the time step size was set to 0.001 seconds, with a total of 2000 time steps. This resulted in a total simulation time of 2 seconds, which was sufficient to capture the dynamic behavior of the air-water separation process within the hydrocyclone. The model was solved using approximately 11 iterations per time step to ensure accurate convergence of the solution at each stage. The transient nature of the simulation enabled the observation of flow evolution and allowed for detailed analysis of time-dependent changes in pressure gradients, velocity fields, and phase distributions.

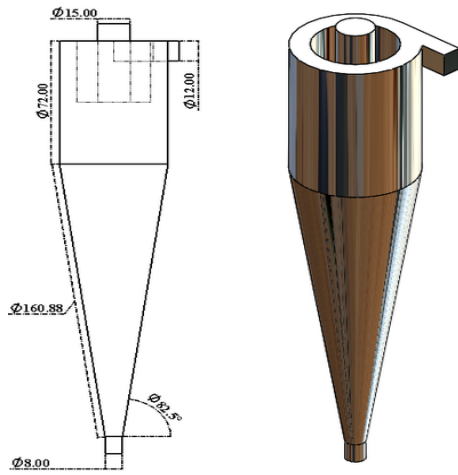


Fig.6 (a) Geometric Parameters in mm (b) Isometric View of Hydrocyclone

C. Mesh Generation and Sensitivity Analysis

In this study, a two-step approach is employed to ensure accurate and efficient results for the simulation of air-water separation in a hydrocyclone. Initially, a steady-state simulation using the k- ϵ turbulence model is conducted to perform the grid independence test. This approach is chosen

due to its computational efficiency, as steady-state simulations are less resource-intensive compared to transient simulations.

The k- ϵ model, though less complex than the Reynolds Stress Model (RSM), is sufficiently accurate for determining the optimal mesh resolution, ensuring that the computational domain is refined enough to capture essential flow features without excessive computational cost. The grid independence test helps identify a mesh that yields stable and consistent results across various refinements, ensuring that the outcomes are not significantly influenced by further mesh resolution, as shown in figure 7. Once the optimal mesh is identified, the study transitions to transient simulations using the Reynolds Stress Model (RSM), which is more suitable for capturing the complex, unsteady, and anisotropic turbulence characteristics of the swirling flow in the hydrocyclone. This approach balances accuracy and efficiency, as the initial steady-state simulations help minimize computational overhead while ensuring the robustness of the results. By first establishing grid independence with the k- ϵ model and then using RSM for the final transient simulation, the method provides a practical yet accurate solution for analysing the complex dynamics of air-water separation in hydrocyclones.

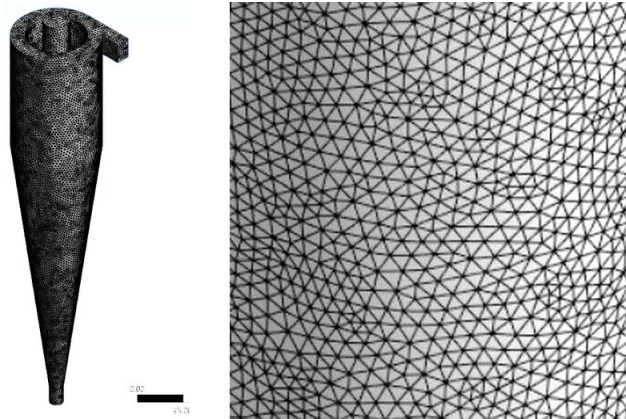


Fig.7 Unstructured tetrahedral mesh of hydrocyclone

As shown in Figure 8, a grid independence study was conducted to ensure reliable simulation results. Testing different element sizes, an element size of 1.5 mm was found to produce consistent outcomes for key variables, including

Net mass flow rates, and volumetric mass averages (velocity and pressure). This indicates the mesh is sufficiently refined to capture essential flow features without significant numerical errors.

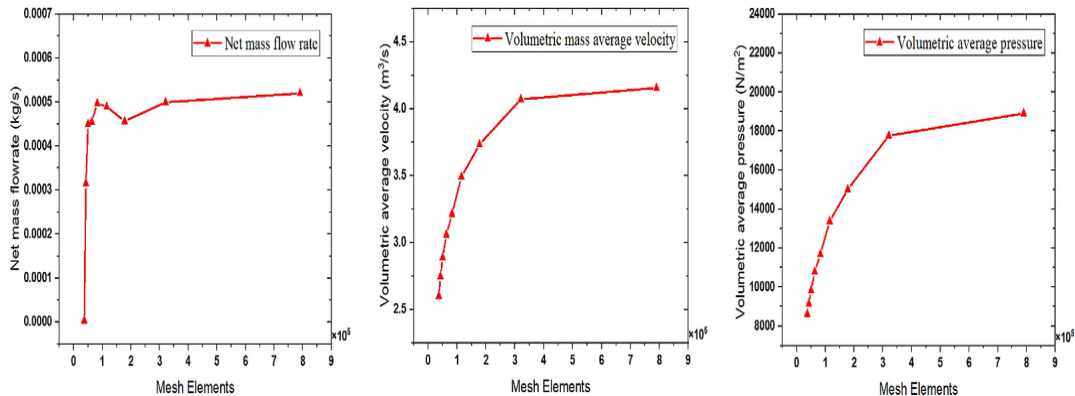


Fig.8 Grid Independence Test for Various Flow Parameters

III. CFD SIMULATION INSIGHTS

The velocity contours along the X, Y, and Z directions were analyzed to understand the flow dynamics within the hydrocyclone. These results were compared with the radial direction velocity contours presented in the reference work. Notably, the Z-direction velocity contour of our simulation closely matches the radial direction velocity behavior

observed in the reference study as it can be seen in figure 9. Both simulations exhibit similar trends, with peak velocities observed around 2.75 m/s in our work and approximately 2 m/s in the reference. This close agreement suggests that the flow characteristics in our simulation are consistent with those reported in the reference, validating the accuracy of the model and the chosen boundary conditions.

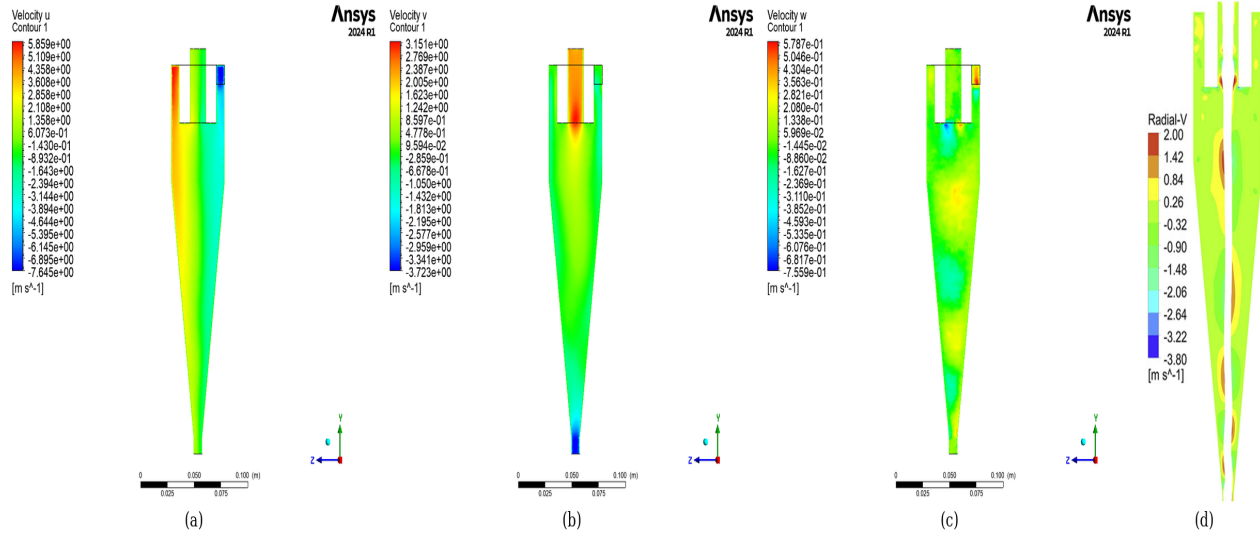


Fig.9 Contours of Velocity along (a) x-axis (b) y-axis (c) z-axis and (d) Radial Velocity Contour [12]

The axial variation of turbulence production and dissipation terms within the hydrocyclone is shown in Figure 10. The results reveal that turbulence production significantly exceeds dissipation within the central region of the cyclone, indicating intense energy generation caused by strong rotational motion and vortex interaction. The peak production region corresponds to the location of maximum swirl intensity, where centrifugal effects and velocity gradients are strongest. Downstream of this region, the dissipation term

gradually increases, representing the conversion of turbulent kinetic energy into thermal energy due to viscous effects. The balance between turbulence production and dissipation governs the stability of the swirling flow structure and plays a critical role in determining vortex formation, phase segregation, and separation efficiency inside the hydrocyclone.

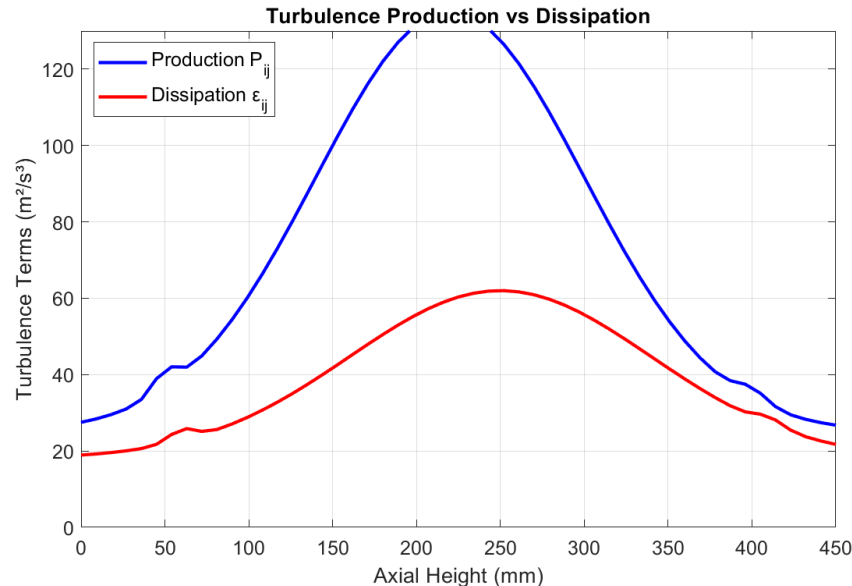


Fig.10 Axial variation of turbulence production and dissipation terms

Figure 11 presents the pressure-strain redistribution terms (Φ_{11} , Φ_{22} , Φ_{33}) predicted by the Reynolds Stress Model across the radial direction of the hydrocyclone. These terms represent the redistribution of turbulent kinetic energy among different normal stress components due to pressure fluctuations within the swirling flow field. The variation of the redistribution terms demonstrates the complex interaction between turbulence production, rotational flow, and

secondary vortex structures inside the cyclone. The observed non-linear behavior confirms the strong anisotropy of turbulence and highlights the importance of incorporating advanced turbulence closure models such as RSM for accurately predicting the stress transport phenomena in highly swirling multiphase flows.

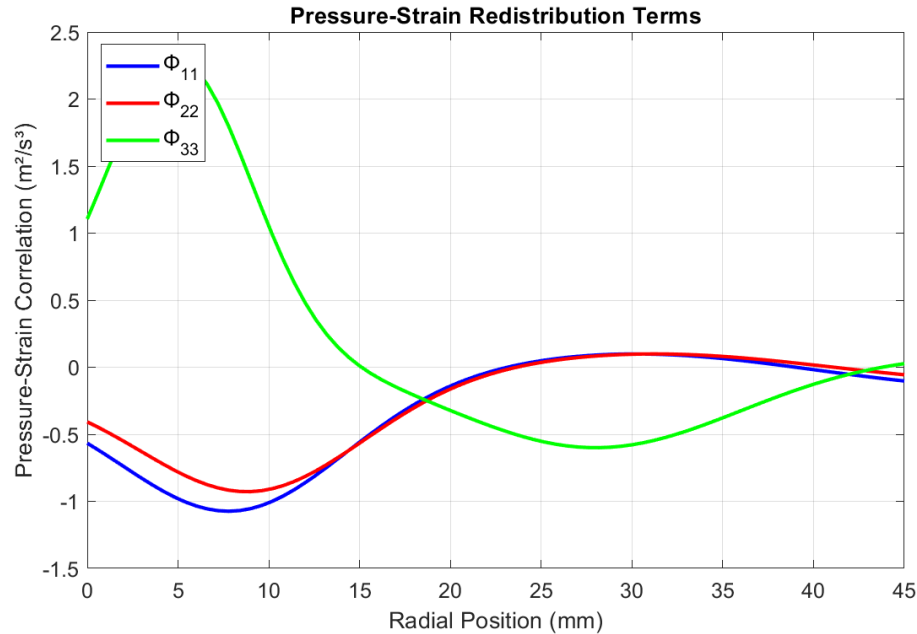


Fig.11 Pressure-strain Redistribution Terms Predicted Using the Reynolds Stress Model

A. Velocity Streamlines and Vortex Formation

Vortices or eddies are swirling flow structures that occur when fluid velocity components in different directions interact, creating regions of recirculating fluid. These flow instabilities lead to the generation of rotating flow patterns, influencing particle separation efficiency in the hydrocyclone. The forward, backward, and resultant velocity streamlines are presented in the figures below, which illustrate the flow dynamics within the hydrocyclone. As shown in figure 12, the

flow exhibits the formation of vortices, commonly referred to as eddies, which are a critical feature in multiphase flow systems like hydrocyclones. The presence of eddies can enhance the mixing of the phases, leading to better separation of lighter and heavier particles. However, they also introduce turbulence, which can cause some particles to recirculate or be displaced incorrectly, impacting separation efficiency. In particular, eddies near the air-core or near the apex of the hydrocyclone can lead to inefficiencies if not properly managed.

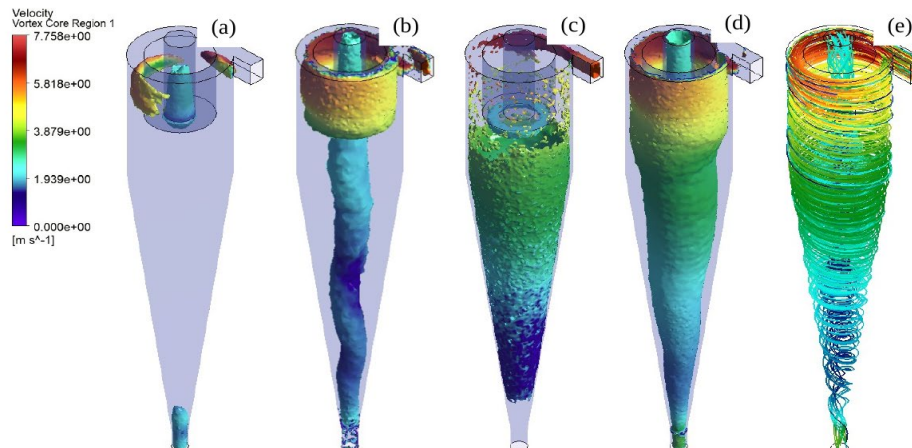


Fig.12 Generation of Vortex Core Region Using (a) Absolute Helicity (b) Eigen Helicity (c) Vorticity (d) Q Criterion and (e) Velocity Streamline

In our case, the observed vortices in the forward and backward velocity streamlines indicate that the flow is actively contributing to particle segregation, with the resultant velocity streamline showing the overall flow pattern, as can be seen in

Figure 13. This detailed understanding of vortex formation and behavior is critical for optimizing cyclone design and improving the overall separation efficiency in both air-water and other multiphase flow systems.

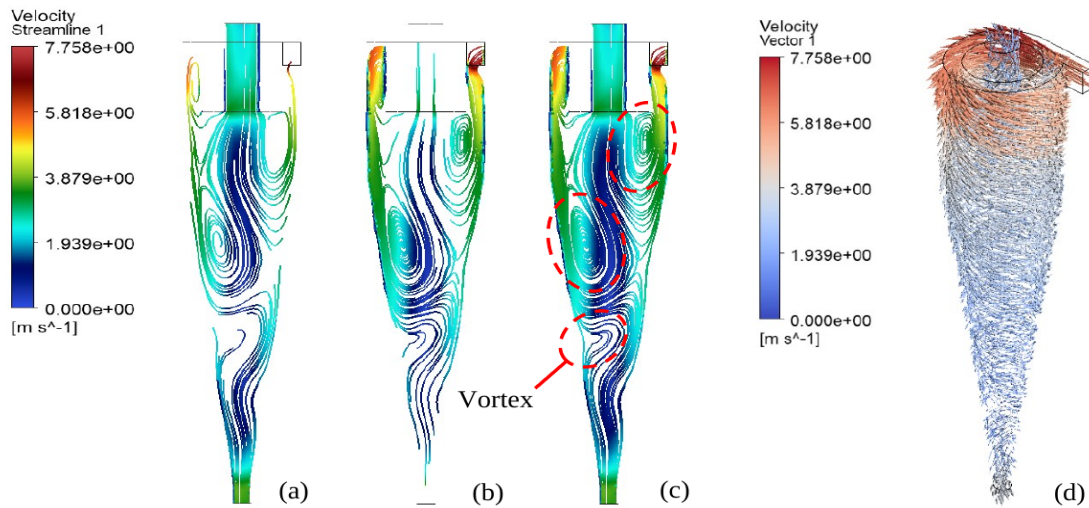


Fig.13 Velocity Streamlines in (a) Forward Direction (b) Backward Direction (c) Resultant and (d) Velocity Vectors

B. Air Core Generation

The injected pressure and resultant velocity contours reveal the formation of the air core at the center of the hydrocyclone. The air core, a key feature in cyclone separation, is formed due to the centrifugal forces acting on the fluid, which causes the lighter phase (air, in this case) to move towards the center while the denser phase (water) is forced outward, as depicted

in **Figure 14.** This distinct core is essential for the efficient separation of phases, as it influences the flow pattern and particle distribution within the hydrocyclone. The contours demonstrate the clear presence of the air core, which plays a significant role in optimizing separation efficiency by affecting the trajectory of the particles and enhancing phase segregation.

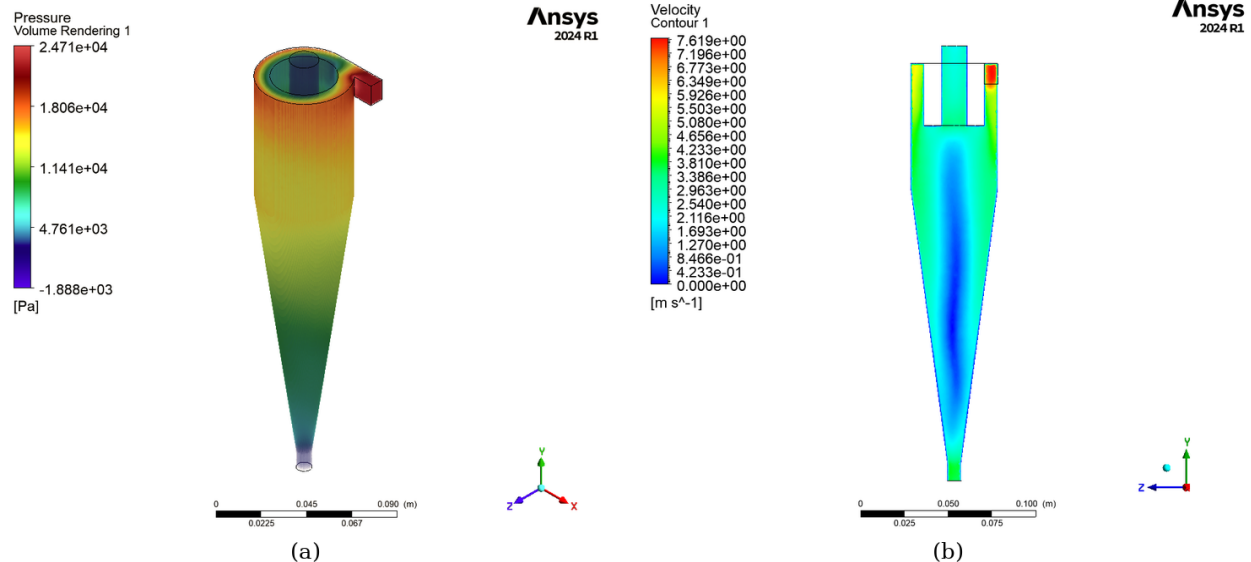


Fig.14 (a) Pressure and (b) Velocity Contours

C. Pressure Gradient Volume Rendering

To capture the detailed variation of the pressure gradient in different directions, volume rendering contours were generated along the X, Y, and Z directions. These contours provide a comprehensive visualization of how the pressure

varies within the cyclone structure and highlight the pressure differentials that influence the flow patterns and separation process. The pressure gradient distributions along the X, Y, and Z axes are shown in **Figure 15**, which offer a clearer understanding of the internal dynamics within the cyclone.

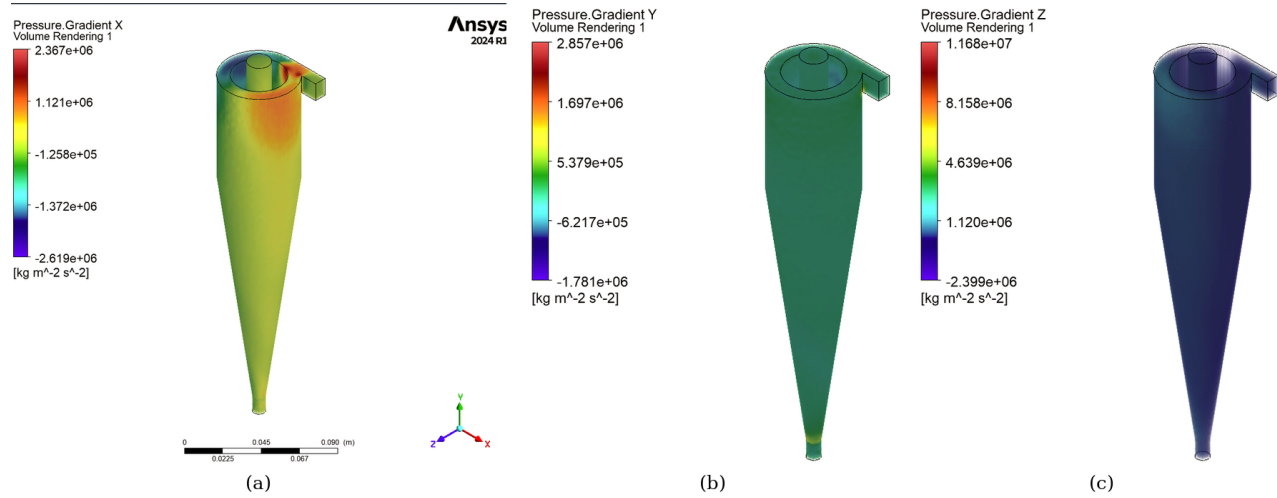


Fig.15 Pressure Distribution along (a) x axis (b) y axis (c) z axis

D. Air and Water Mass Fraction Contours

The air and water mass fraction contours, as shown in Figure 17, represent the distribution of the two phases within the hydrocyclone. In this study, the air mass fraction is set at 0.05, which corresponds to the fraction of air in the mixture. These contours clearly illustrate the spatial distribution of both phases, with the air phase predominantly concentrated in the central region of the hydrocyclone, forming the air core. The water phase is largely confined to the outer regions, where the centrifugal forces push the denser water towards the cyclone walls. The contours reflect the interaction between the two phases, highlighting the efficiency of the multiphase flow model in capturing the separation process. The distribution of the air and water phases directly influences the separation efficiency, with the air core playing a critical role in the phase segregation process within the hydrocyclone.

The radial evolution of water volume fraction at different axial locations inside the hydrocyclone is illustrated in Figure 16. The results demonstrate the progressive phase segregation occurring along the cyclone axis due to centrifugal action. Near the cone section, the water volume fraction rapidly increases toward the outer radial region, indicating strong outward migration of the denser liquid phase.

In contrast, the upper section of the hydrocyclone exhibits a comparatively wider transition region due to incomplete phase separation and stronger vortex interaction near the air core. The gradual shift in the phase distribution profiles along the axial direction confirms the development of the air core and highlights the effectiveness of the hydrocyclone in achieving air-water separation through rotational flow dynamics.

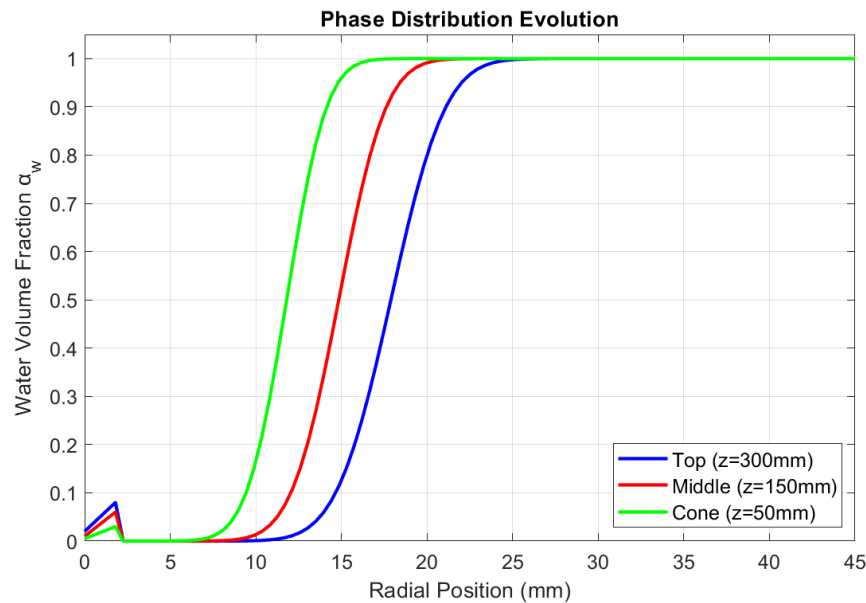


Fig.16 Radial Evolution of Water Volume Fraction at Different Axial

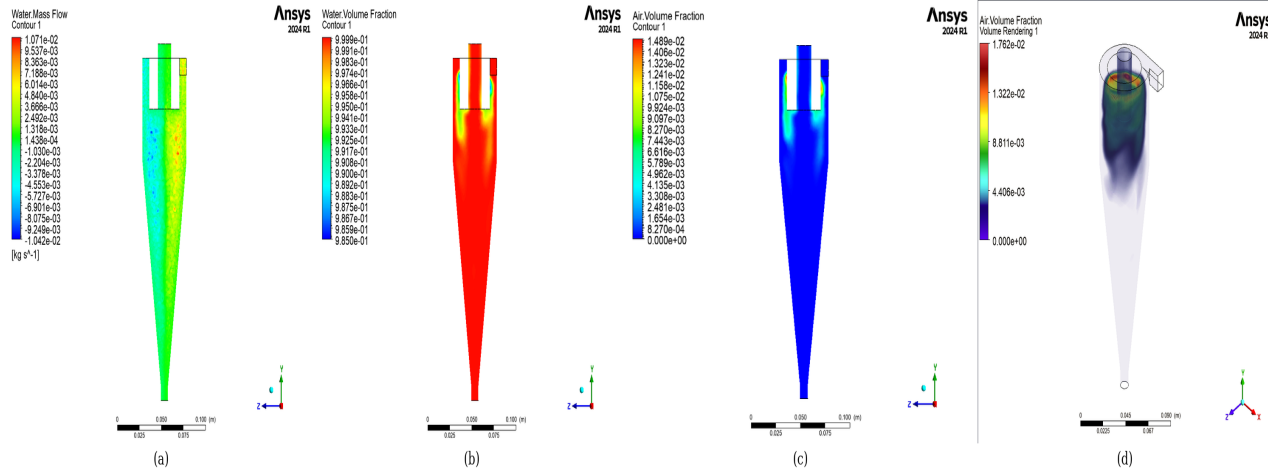


Fig.17 Contours of (a) Water Mass Flow (b) Water Volume Fraction (c) Air Volume Fraction and (d) Volume Rendering of Air Volume Fraction

IV. ANALYSIS AND DISCUSSION

A. Axial Velocity Distribution

The axial velocity distribution across the radial position demonstrates a similar trend for both the Euler and VOF models, aligning well with experimental data, as shown in Figure 18(a). This indicates that both models adequately capture the overall behavior of the flow. However, notable discrepancies in magnitude and localized behavior are observed. The Euler model exhibits a smoother velocity profile, with a peak velocity of approximately 8 m/s, which is consistent with experimental observations. Conversely, the VOF model shows abrupt variations, such as a sudden spike in velocity exceeding 8 m/s at a radial position of approximately 25 mm. This deviation can be attributed to the sensitivity of the VOF model to interface dynamics, which may introduce localized instabilities or numerical artifacts near the interface.

B. Tangential Velocity Profile

The tangential velocity profiles predicted by both models exhibit similar overall trends, suggesting that both modeling

approaches effectively capture the rotational flow characteristics. The discrepancies in magnitude are less pronounced compared to the axial velocity, indicating a better agreement between the two models in representing tangential components, as shown in Figure 18(b). This consistency reflects the robustness of both models in handling rotational flows, though the Euler model again demonstrates a smoother representation.

C. Pressure Gradient Behavior

The pressure gradient exhibits significant differences between the Euler and VOF models. The Euler model predicts a starting pressure gradient of approximately 50 $\text{kg/m}^3 \cdot \text{s}^2$, while the VOF model begins at a much higher value of around 1100 $\text{kg/m}^3 \cdot \text{s}^2$. Despite this initial disparity, the general trend and peak values of the pressure gradient align reasonably well between the two models, as shown in Figure 18(c). The elevated initial pressure gradient in the VOF model may be attributed to its explicit treatment of phase interfaces, which can amplify pressure effects near the interface regions. Furthermore, the VOF model's sensitivity to interface dynamics and potential numerical instabilities may lead to an overestimation of pressure gradients.

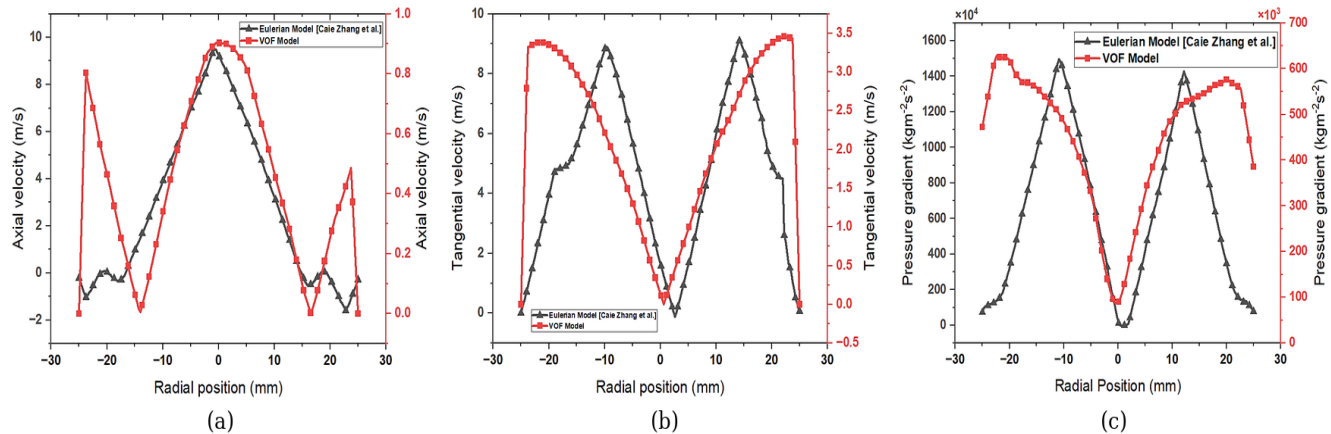


Fig.18 Comparison of (a) Axial Velocity Distribution (b) Tangential Velocity Profile and (c) Pressure Gradient Distribution Across Radial Position

TABLE I MODEL ACCURACY AND COMPUTATIONAL COST COMPARISON

Parameter	Eulerian	VOF
Convergence stability	High	Medium
CPU time	Lower	Higher
Interface tracking	Moderate	Excellent
Pressure prediction	Better	Slight overprediction
Recommended use	Bulk flow	Interface dynamics

The differences observed between the Euler and VOF models can be attributed to their underlying modeling approaches. The Euler model utilizes a continuum framework with averaged equations, making it less sensitive to localized phase interactions and interface dynamics. This results in smoother and more stable predictions for velocity and pressure fields. In contrast, the VOF model explicitly resolves the phase interface, capturing detailed interface dynamics but introducing numerical artifacts or instabilities in regions of high curvature or complex flow interactions. These artifacts are particularly evident in the axial velocity graph, where sharp variations are observed in the VOF predictions. Additionally, the VOF model's explicit treatment of surface tension and phase coupling may lead to an overprediction of pressure gradients near the interface.

The Euler model demonstrates superior agreement with experimental data in terms of smoothness and consistency of predictions. It is particularly well-suited for scenarios where bulk flow behavior is of primary interest and detailed interface dynamics are not critical. Conversely, the VOF model is better suited for cases requiring explicit interface resolution, such as flows involving significant phase interactions. However, the increased computational cost and potential for numerical instabilities in the VOF model should be considered when selecting the appropriate modeling approach.

V. CONCLUSION

This study presents a comprehensive comparison between the VOF multiphysics model and the Eulerian model for simulating air-water separation in a hydrocyclone. The Reynolds Stress Model (RSM) was employed for turbulence modeling, while the K-epsilon model was used in grid independence testing. The simulations provided valuable insights into the pressure distribution, velocity streamlines, and air-water mass fractions, which were systematically validated against experimental results from existing literature. The grid independence test confirmed that a mesh element size of 1.5 mm provided consistent results for critical flow variables, ensuring the reliability of the simulation. Notably, the velocity contours and pressure gradients observed in the simulations exhibited strong agreement with experimental data, demonstrating the effectiveness of the models employed. The VOF multiphysics model, when combined with the RSM, offers a reliable approach for simulating multiphase flows in hydrocyclones, with potential applications in optimizing the design and operational

performance of these devices. Future studies could explore further refinements in turbulence modeling and examine additional flow configurations to broaden the scope of hydrocyclone simulations. Hydrocyclones are an efficient and widely employed technology for air-water separation, especially in scenarios where high flow rates and relatively coarse phase separations are required. Their ability to utilize centrifugal forces for phase segregation makes them ideal for applications in the oil and gas industry, wastewater treatment, and chemical processing. However, despite their many advantages, challenges related to the separation of fine droplets, high-pressure drops, and the sensitivity of the system to operational conditions must be addressed. Future developments in cyclone design, operational strategies, and materials could further enhance the performance of hydrocyclones in air-water separation applications, particularly for fine droplet separation and energy optimization.

Declaration of Conflicting Interests

The authors declare no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

The authors received no financial support for the research, authorship, and/or publication of this article.

Use of Artificial Intelligence (AI)-Assisted Technology for Manuscript Preparation

The author(s) utilised generative AI technologies solely to improve language clarity and refine grammar. No images were generated, manipulated, or altered using AI-assisted technologies.

REFERENCES

- [1] S. K. Tripathy, S. K. Bhoja, C. Raghu Kumar, and N. Suresh, "A short review on hydraulic classification and its development in mineral industry," *Powder Technology*, Jan. 2015, doi: 10.1016/j.powtec.2014.09.049.
- [2] B. Wang and A. B. Yu, "Computational investigation of the mechanisms of particle separation and 'fish-hook' phenomenon in hydrocyclones," *AIChE Journal*, vol. 56, no. 7, pp. 1703–1715, Jul. 2010, doi: 10.1002/aic.12114.
- [3] F. Concha, A. Barrientos, J. Montero, and R. Sampaio, "Air core and roping in hydrocyclones," 1996.
- [4] C. W. Haig, A. Hursthouse, S. McIlwain, and D. Sykes, "An empirical investigation into the influence of pressure drop on particle behaviour in small scale reverse-flow cyclones," *Powder Technology*, vol. 275, pp. 172–181, May 2015, doi: 10.1016/j.powtec.2015.02.011.
- [5] S. Kuang, Z. Qi, A. B. Yu, A. Vince, G. D. Barnett, and P. J. Barnett, "CFD modeling and analysis of the multiphase flow and performance of dense medium cyclones," *Minerals Engineering*, vol. 62, pp. 43–54, 2014, doi: 10.1016/j.mineng.2013.10.012.
- [6] M. Narasimha, M. S. Brennan, and P. N. Holtham, "CFD modeling of hydrocyclones: Prediction of particle size segregation," *Minerals Engineering*, vol. 39, pp. 173–183, Dec. 2012, doi: 10.1016/j.mineng.2012.05.010.
- [7] K. W. Chu, B. Wang, A. B. Yu, and A. Vince, "CFD-DEM modelling of multiphase flow in dense medium cyclones," *Powder Technology*, vol. 193, no. 3, pp. 235–247, Aug. 2009, doi: 10.1016/j.powtec.2009.03.015.
- [8] R. K. Tue Nenu and H. Yoshida, "Comparison of separation performance between single and two inlets hydrocyclones," *Advanced Powder Technology*, vol. 20, no. 2, pp. 195–202, Mar. 2009, doi: 10.1016/j.appt.2008.08.004.
- [9] T. R. Vakamalla, K. S. Kumbhar, R. Gujjula, and N. Mangadoddy, "Computational and experimental study of the effect of inclination on hydrocyclone performance," *Separation and Purification Technology*, vol. 138, pp. 104–117, Dec. 2014, doi: 10.1016/j.seppur.2014.10.013.

- [10] M. Ghodrat, Z. Qi, S. B. Kuang, L. Ji, and A. B. Yu, “Computational investigation of the effect of particle density on the multiphase flows and performance of hydrocyclone,” *Minerals Engineering*, vol. 90, pp. 55–69, Aug. 2016, doi: 10.1016/j.mineng.2016.03.017.
- [11] K. Nageswararao, “Modelling of hydrocyclone classifiers: A critique of ‘bypass’ and corrected efficiency,” *Powder Technology*, vol. 297, pp. 106–114, Sep. 2016, doi: 10.1016/j.powtec.2016.04.016.
- [12] C. Zhang, D. Wei, B. Cui, T. Li, and N. Luo, “Effects of curvature radius on separation behaviors of the hydrocyclone with a tangent-circle inlet,” *Powder Technology*, vol. 305, pp. 156–165, Jan. 2017, doi: 10.1016/j.powtec.2016.10.002.