ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

Investigating Non-Newtonian Fluid Behavior in Hydrocyclones Via Computational Fluid Dynamics

Raj Shekhar[1#]; Abir Chakravorty[1*#]
[1]Agricultural and Food Engineering Department, IIT Kharagpur
*Email: abir.28kh@agfe.iitkgp.ac.in, abir.cty2015@gmail.com
*OrciD: https://orcid.org/0000-0002-9253-2594
First authors of the manuscript

Abstract:- Expert researchers examine complex patterns of pressure, viscosity, and velocity in a CFD study of viscoelastic food inside hydrocyclones to obtain a detailed grasp of particle behavior and fluid dynamics. Velocity profiles show how fluids and particles flow through the hydrocyclone in complex ways, while pressure distributions show where high and low pressure is found, regions that are critical for maximizing separation efficiency. Furthermore, the analysis of viscosity fluctuations clarifies the intricate relationship between fluid rheology and flow dynamics, providing information on how food's viscoelastic characteristics affect particle trajectories and separation efficiency. Utilizing this comprehensive examination, scientists hope to optimize the design and functioning parameters of the hydrocyclones, which will in turn improve the efficacy and efficiency of particle separation procedures in viscoelastic food solutions. This will ultimately lead to improvements in food processing technology and product quality. Researchers look into the impact of geometric elements on flow patterns and separation efficiency in addition to these characteristics, such as hydrocyclone size and inlet configurations. Additionally, they investigate how different operating parameters, such rotational speed and flow rate, affect how well the hydrocyclone handles viscoelastic food items. Through the integration of these complex analyses, researchers hope to create all encompassing models that can precisely forecast and optimize the behavior of viscoelastic food flows inside hydrocyclones, opening the door to improved process control and food sector product quality.

Keywords:- *Particle Separation, Fluid Dynamics, Hydrocyclone Analysis, Viscoelastic Food, and Process Optimization.*

I. INTRODUCTION

Hydrocyclones are devices used to extract particles or liquids of varying densities from liquid streams. Hydrocyclones are used in many industrial applications, such as water treatment, food and mineral processing, petroleum extraction, and other sectors where it is essential to remove suspended particles from fluid streams. Dispersed particles can be extracted from fluid streams using centrifuges, hydrocyclones, and filters. Centrifuges consume a lot of energy due to their rotation. A hydrocyclone requires no rotating elements and has low operating costs. All that is required to overcome the pressure drop is energy. These are the explanations for why hydrocyclones are piqueing the interest of both business and academia, according to Li et al. (2023). Within a hydrocyclone, fluid velocity is both turbulent and whirling. Due to the centrifugal force produced by the rotational motion, which also causes differences in angular velocity inside the hydrocyclone, particles of different sizes, shapes, and densities separate. Typically, a cylindrical-conical hydrocyclone has one entrance and two exits, called the underflow and overflow. The axis of the hydrocyclone goes across both outputs. Underflow normally occurs at the cone's peak, while overflow typically comes from an inner tube called a vortex finder on top of the hydrocyclone. The density of the suspended particles is higher than the fluid medium's, according to Yucheng et al. (2015).

A. Geometry and Construction for Hydrocyclone

A normal hydrocyclone has two exits, referred to as underflow and overflow, and one entrance. A hydrocyclone lacks any spinning or moving parts. Tangentially, the inlet is introduced to the hydrocyclone's cylindrical portion. The hydrocyclone's axis is where both outputs are located. At the top of the cone, the underflow is where the majority of the heavy and coarse-sized particles are anticipated to exit. The overflow, located at the top of the vortex finder, is expected to discharge a fluid stream that contains smaller particles or is relatively clean. The liquid overflow pipe, which extends inside the hydrocyclone's cylindrical portion, is the vortex finder. To reduce the chance of the liquid feed leaving the hydrocyclone too soon, the vortex finder inside should extend below the feed outlet. Liquid slurry is pumped via the inlet and into the hydrocyclone body through the enclosed top portion of the cylinder. The tangentiality of the feed causes the fluid's linear velocity to change into whirling motion, which is necessary for particle separation. Typically, steel, ceramics, and polymers (polyurethane, polypropylene, etc.) are used to make hydrocyclones. Metals can be used as construction materials in hydrocyclones that need to withstand high temperatures and pressures. When abrasive materials such as sand are included in the feed, it is more convenient to use polyurethane instead of metal or ceramic. For greater durability and longevity, metal-lined polyurethane hydrocyclones can be used. Oliveira Diogo et al (2009). The figure 1 represents schematic diagram of the experimental setup for yeast separation by mini-hydrocyclones.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

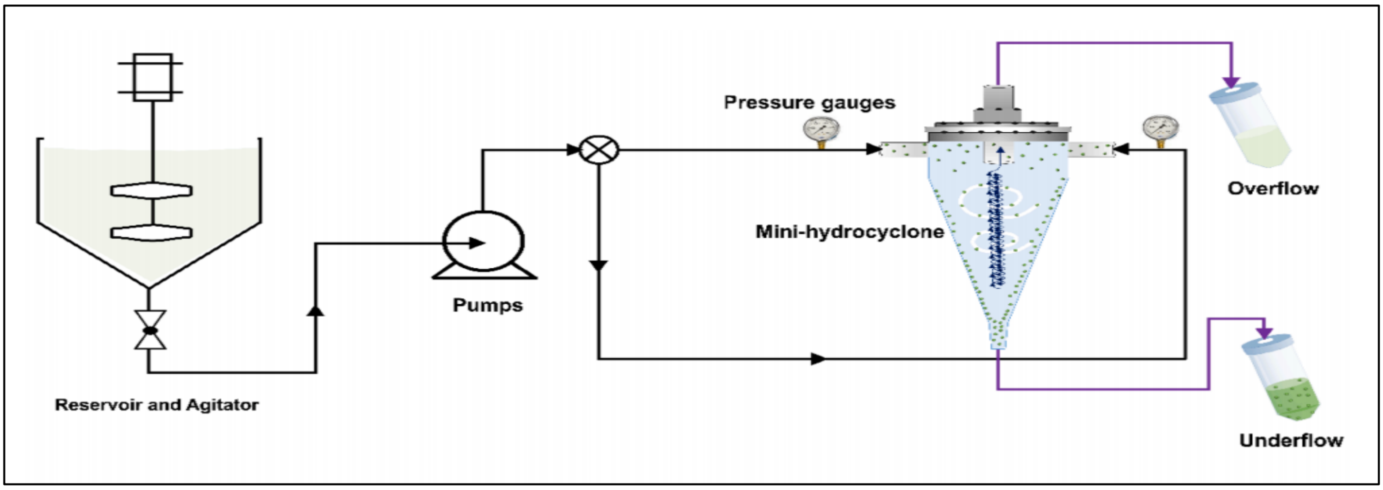


Fig 1: Schematic Diagram of the Experimental Setup for Yeast Separation by Mini-Hydrocyclones [Li et al 2023] [Figure Permission Obtained]

➢ *Definitions of Key Performance Parameters*

When it comes to benefits and drawbacks, hydrocyclones have many advantages in a range of industrial applications. First off, they are economically efficient due to their design, which eliminates any moving or rotating elements and lowers running expenses. Hydrocyclones also have a high volume capacity, which makes it easy to treat big amounts of material. Their compact design requires minimal installation area, improving space use, and their easy manufacture, installation, and operation further add to their cost-effectiveness. Hydrocyclones are very adaptable and can be used for a variety of purposes, which increases their usefulness. Hydrocyclones are more effective when managing clustered solids or thixotropic liquids because of their strong shear forces. Additionally, they outperform conventional classifiers in terms of efficiency, with a high classifying ratio of 80–85%. Hydrocyclones do, however, have a number of drawbacks that need to be taken into account. They may not be appropriate for many applications due to their restricted degree of separation, even though they are efficient. They are also prone to erosion, which over time may have an impact on their durability and effectiveness. Excessive flow rates can cause hydrocyclone fatigue and shorten their lifespan, which raises maintenance expenses. Hydrocyclones are also less effective in some situations due to their lower resolution and separation power when compared to other methods of separation. Hydrocyclones do not benefit from flocculation despite having beneficial shear forces, which may limit the amount of separation enhancement they may achieve. Last but not least, the connection of flow rate and separation effect limits the adaptability of hydrocyclones in certain applications and leads in rigid separation outcomes.

B. The Separation Efficiency or Partition number

Is defined as the volumetric ratio of the particles reported to the underflow to those fed into the hydrocyclone for the particular particle size. The slip ratio it is defined as the volume of suspension obtained in underflow to that collected from both overflow and underflow. The Solids recovery

The recovery of solids can be calculated as the ratio of the mass flow rate of solid particles in the underflow to that in the feed. The Concentration ratio it is defined as the ratio between the concentration of the particles in the underflow and the concentration of particles in the feed. It can also be denoted as the ratio of the solids recovery to the split ratio.

C. Particle Purity and Particle Yield

It is defined as the ratio of the partition number of a particular particle size to that of the others including the particle size considered, expressed in percentage. It is measured as the partition number of a specific particle size to the overflow.

D. Separation Mechanism

The separation mechanism of hydrocyclone depends upon three phenomena namely,(a) Primary Vortex, (b) Secondary Vortex, (c) Air Core

➢ *Primary Vortex*

The hydrocyclone's cylindrical section receives a tangential injection of pressurized slurry or fluid stream, which causes the fluid inside to whirl. The fluid's linear motion is transformed into whirling motion and continually fluctuating angular velocity by the tangential entry of the fluid. The formation of two spiral vortices inside a hydrocyclone that are vertically opposed is what separates the fluid streams and particles. Denser particles are carried by the outer and create underflow stream, which is driven by centrifugal force resulting from the swirling motion. Particles concentrate close to the hydrocyclone wall depending on the circuit layout and the ultimate cause. Recirculating and processing a coarse spiral concentration close to the cone's tip underflow requirements is possible.

➢ *Secondary Vortex*

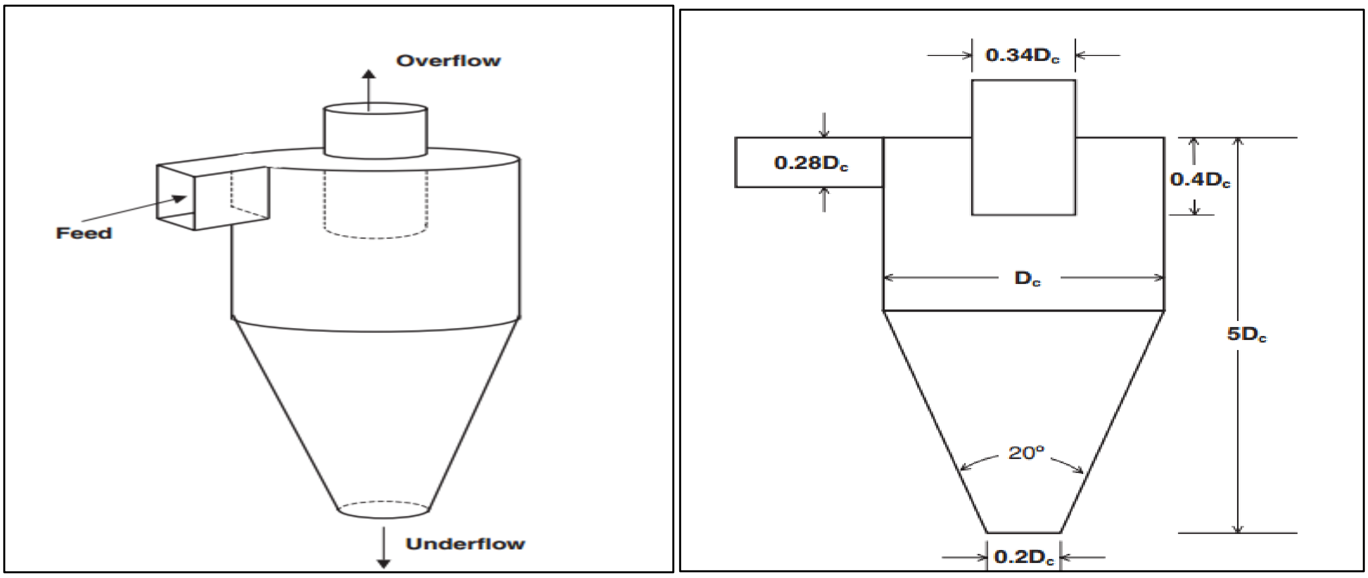
Via the vortex finder, secondary flow is composed of smaller particles with an inner spiral. The lighter particles form the inner spiral, and the separation is caused by the upward drag force acting on the particles within the inner spiral. This is smaller in radius than the primary vortex and is referred to as the secondary vortex. Because of the low-pressure zone, the fluid inside the hydrocyclone has a high axial velocity. The particles are subject to an upward drag force from this high axial velocity, which causes the particles to be carried away through the overflow.

Eddy flow and short circuit flow are also seen in hydrocyclones, which add to the overflow. A short circuit flow is a channel that runs from the hydrocyclone's input to the vortex finder on its roof. The restriction of tangential velocity close to the hydrocyclone roof causes short circuit flow. A portion of the liquid feed travels straight across the hydrocyclone roof and across the vortex finder wall to join the overflow stream inside the vortex finder due to the low-pressure area that is formed in the vicinity of the hydrocyclone wall, roof, and inner regions. It has been noted that roughly 15% of the feed liquid enters the overflow by way of a short circuit flow. Poor separation results from short circuit flow's greater feed volume. Since the fluid particles in a short circuit flow must travel a greater distance due to the vortex finder's longer length inside the hydrocyclone, the device's goal is to reduce the amount of short circuit flow. Less fluid in the short circuit flow as a result.

The vortex finder's inability to handle the full volume of overflow stream is what causes eddy flow. A portion of the overflow stream recirculates and rejoin the inlet stream because the vortex finder is unable to release the entire volume of stream at once. Recirculating eddies are created when a vertical stream forms between the intake and the outer wall of the vortex finder in an eddy flow. The regular overflow opening's incapacity to handle the vortex's inherent upward movement results in eddy flow Shingote et al. (2015).

- *Air Core Formation*

In hydrocyclones, the creation of air cores is an inevitable process. The hydrocyclone's air core is in action. Centrifugal force introduces coarse particles into the hydrocyclone and moves them away towards the wall along the central axis of the hydrocyclone. "In order to generate a sizable centrifugal force field inside the hydrocyclone, feed tangentially at a high speed. Coarse particles travel in the direction of the wall and exit with underflow after being swept down the wall to the apex. The figure 2 represents the schematic of the hydrocyclone and Rietema Standard Hydrocyclone diameter. The lighter-particle-carrying fluid stream approaches the cone's apex, reverses course in an axial direction, and spirals upward before exiting through the vortex finder. A region of low lying along the axis. Due to extremely high angular momentum, pressure is produced.



(a) (b)

Fig 2: (a) Schematic of a Hydrocyclone, (b) Dimensions of Rietema's standard Hydrocyclone [Rivas et al (2004)]

A spinning free liquid surface along the axis could result from this. An air core may occur if air is drawn via the outlets and drawn into the low-pressure zone due to vacuum pressure inside the zone if the outlets are open to the atmosphere. Depending on the operating conditions, the air core's geometry might take the shape of a linear or wavy cylinder. During operation, fluctuations within the flow field cause wavy cylinders to develop. Air core development has been identified in studies to be a sign of vortex stability. Hydrocyclone flow split is reliant on the air core's creation and stability. Air core is known to have a significant impact on hydrocyclone separation efficiency, while its precise effects on separation are debatable. While some researchers reported poor separation efficiency, others claimed that suppressing the air core produced superior results. Zhao and associates (2024).

E. Effect of Some Geometric Parameters on Separation

Studies were conducted to observe the effect of certain geometric parameters on hydrocyclone performance, yielding notable conclusions. Firstly, it was found that long, narrow rectangular openings offer slightly improved efficiency compared to circular openings of equivalent cross-sectional area. However, large openings tend to induce excessive turbulence, leading to higher pressure drops. Additionally, a reduction in vortex finder diameter, while maintaining other geometric dimensions constant, was observed to increase separation efficiency for lighter particles. For coarser particles, efficiency peaked and then declined with an increase in vortex finder diameter due to heightened short-circuit flow. Moreover, an increase in vortex finder depth was found to enhance separation efficiency for coarser particles by reducing flow short-circuiting, although efficiency decreased for lighter particles. Lastly, a reduction in underflow diameter, with other dimensions and feed pressure held constant, resulted in an overall decrease in separation efficiency.

F. Fluid Velocity Distribution

The velocity of fluid inside a hydrocyclone can be divided into three components, namely, axial, tangential, and radial velocity.

- *Axial Velocity*

The axial velocity serves as a measure of the strength of the two counteracting spirals and the distribution of fluids between the overflow and underflow streams. Within the hydrocyclone, a zone of zero axial velocity delineates the separation of the upward inner vortex from the outer downward vortex. Particles carried by the inner vortex towards the overflow are propelled by this axial velocity. Furthermore, axial velocity escalates with distance from the envelope, with the inner spiral exhibiting notably higher maximum velocities.

- *Radial Velocity*

The radial velocity component's magnitude increases radially outward, reaching its highest value close to the hydrocyclone wall. With the exception of eddy flow just outside the vortex finder, it is usually oriented radially inward. Particles must therefore encounter a centrifugal force in order to settle against it. The main cause of particle classification is centrifugal force-drag force balance, which causes larger particles to centrifugally drift towards the wall and smaller particles to be transported radially throughout the conical portion and into the center. Coarser particles discharge with the underflow as they swiftly travel towards the wall, where the absolute fluid velocity is lowest.

- *Tangential Velocity*

Tangential velocity is the key velocity component for the separation of particles in hydrocyclone. Tangential velocity increases in the radial direction toward the air core and it falls rapidly at the interface of the air core. The associated velocity gradients are steepest in the region below the vortex finder, The Tangential velocity profile has a complex vortex structure, known as the Rankine vortex. Tangential velocity varies proportionally with radial distance in the forced vortex region and inversely with radial direction in the free vortex region. A peak intermediate between the two vortices marks a gradual transition between the two distinct and uniquely defined vortices Fisher and Flack (2002).

G. Application in Food Industry

The hydrocyclone serves as a pivotal pre-treatment apparatus, frequently enlisted to enhance downstream equipment performance and cost-effectiveness across a spectrum of industries. Beyond its conventional roles in classification and thickening, novel applications and methodologies have emerged. Primarily deployed in sectors spanning:

- Particle Separation: Demonstrating efficacy in eliminating larger particles, including mammalian cells, hydrocyclones leverage centrifugal force to segregate particles by density, size, and morphology.
- Cell Separation: Playing a crucial role in tasks such as cell sorting, clarification of fermentation broths, and purification of biomolecules.
- Fruit and Vegetable Processing: Employed for solid-liquid separation, facilitating the production of clearer and superior-quality juices.
- Dairy and Beverage Production: Utilized for juice concentration and the clarification of diverse food products.
- Wine Processing: Engaged in the extraction of seeds, debris, and metallic impurities from crushed grape juice.
- Jams, Hot Sauces, and Purees: Employed to eliminate contaminants before the milling phase, safeguarding the milling machinery.
- Ketchup Production: Enlisted for contaminant removal preceding the cooking stage.

These multifaceted applications underscore the adaptability and efficiency of hydrocyclones within the food processing sector. Offering a swifter and more effective alternative to traditional methods such as centrifugation, filtration, or sedimentation.

II. LITERATURE REVIEW

Because of the hydrocyclone's benefits over alternative separation processes and low running costs, it has been widely used by various industries since its invention in 1900. A thorough understanding of the procedure is still required in order to examine how various parameters affect the efficiency of separation. To improve hydrocyclone functioning and design, a new strategy is required. A flexible method for making detailed flow field predictions across a broad range of design and operating state variations is computational fluid dynamics (CFD). Any CFD technique that uses numerical treatment of the Navier-Stokes equation yields the flow field. This was the outcome of both the quick advancement of computers and our growing comprehension of the numerical modeling of turbulence.

Bloor and Ingham et al. (2001) computed the flow field inside a hydrocyclone using the Navier-Stokes equation and came up with an analytical solution using reduced assumptions. Vortex conservation was used close to the axis under the assumptions of rotating and inviscid flow, producing the axial and radial components of velocity. In their 2007 study, Narasimha et al. used a single phase model

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

with water to investigate the impact of particle size on separation efficiency during the hydrocyclone's separation of solid particles from slurry. They found that increasing the inlet velocity increases both the continuous phase and centrifugal force. Additionally, they used multiphase models and CFD methodologies to visualize the air core, and they came to the conclusion that the air core lowers the hydrocyclone's separation efficiency. An experimental investigation was conducted by A. Goyal et al. (2010) to observe the impact of the air core on flow split. In order to compare the separation effectiveness of hydrocyclones, V. Kumar et al. (2007) used various turbulence models.

A. Hydrocyclone in Food Industry

Because there is a growing need for a variety of foods such as functional food, the food industry has experienced significant global growth. Particle separation is a critical component of food processing, and the capacity and efficiency of this industry have been severely strained by the increase in demand. Food processing uses a variety of particle separation technologies, each of which has a goal separation size, benefits, and drawbacks that are shown in the table. Li et al (2023).

Centrifugation is a commonly used technique in food processing to separate biological components such as proteins, vesicle subpopulations, cells, and cell detritus. By taking advantage of variations in densities and sizes, centrifugal force is used to isolate the particles. Another method of sorting particles according to size and molecular weight is membrane-based filtration, which uses membranes with predetermined cut-offs. These membranes are employed in the separation of a variety of particles, such as proteins, cells, and flocs or sludge. Furthermore, microfluidic platforms have become effective means of sorting particles; to increase efficiency, these platforms are frequently combined with more traditional methods like centrifugation and membrane filtration.

Although the separation technologies available today are useful, food processing faces difficulties due to their limited scalability and efficiency. Centrifugation, for instance, has a high particle separation efficiency but is difficult to scale due to the need for expensive, high-precision equipment, lengthy processing periods, and intricate workflows. However, despite being a cost-effective method for processing large numbers of samples, membrane filtration has drawbacks such pore blockage and fouling, which can affect the purity and yield of the final product.

Hydrocyclones have been successfully used in a variety of sectors for particle separation during the past few decades because of their low maintenance costs, continuous operation, and simple, variable geometries. They do have certain disadvantages, though, such as wall erosion, bypass effects, and particle misplacement. Because biological particles are delicate and sensitive to shear forces, hydrocyclones encounter difficulties, especially in the food processing industry where particles are smaller and frequently diverse.

Mini-hydrocyclones, also called micro-hydrocyclones, are used in food processing to overcome the difficulties caused by small particle sizes. This allows for better separation and smaller cut sizes than in mineral processing. Generally, these hydrocyclones are designed to maximize separation yield and efficiency at lower feed material concentrations. Their intricacy still presents a problem, though, with multiphase flow measurement being particularly tricky.

Notwithstanding obstacles, hydrocyclone technology has advanced quickly and been widely used over the past three decades, as seen by the rise in publications in this area. The rapid advancement of hydrocyclone research makes it difficult to present a comprehensive overview; nonetheless, reviews that do exist tend to concentrate on certain elements, such as hydrocyclone models, de-oiling applications, and geometrical and operational parameter optimization. This review explores the use of hydrocyclones in food processing, providing information on technological advancements and possible ways to increase particle separation in the food sector Li et al (2023).

B. Tailored Design for Food Processing

In the food processing industry, hydrocyclones provide an affordable way to separate, filter, and concentrate different types of particles. Conventional separation methods continue to be dominant because of the large gaps that exist between technological exploration and industrial implementation. To close this gap, hydrocyclone technology advancements are essential. It is critical to comprehend the complex multiphase flows that occur inside mini-hydrocyclones. To do this, experimental and numerical methods are required, which means that efficient techniques to reduce unfavorable wall effects like vibration and image distortion are required. Currently available mechanistic models based on Lagrangian particle tracking (LPT) and computational fluid dynamics (CFD) offer some insights, but they have drawbacks, particularly when it comes to the interactions between particles and fluid flow. Although a number of models have been put up to address these issues, food processing has not yet used them. Subsequent endeavors ought to concentrate on closing this information gap and simulating interaction-induced particle damages. Under some circumstances, mini-hydrocyclone design alterations and optimization can be guided by mechanistic models that are substantiated by physical experiments. Furthermore, data-driven artificial intelligence (AI) models exhibit potential for improving hydrocyclone performance and defect identification. Unique geometry combinations and feed property adjustments, among other innovations in mini-hydrocyclone design, have demonstrated substantial potential for efficient separation in the food business.

C. Modeling of Hydrocyclone

Hydrocyclone modeling is typically accomplished by creating empirical or semi-empirical correlations between operating factors and observed responses, or by using fluid flow and particle motion inside the cyclone as models. While rigorous in nature, fundamental models are attractive, they are not well suited to describe the interactions between

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

particles and fluid in hydrocyclones that operate at greater solid concentrations. Industrial hydrocyclone modeling and simulation mostly use semi-empirical and empirical models that relate partition curve parameters to cyclone design and operating variables. Many generic models have been created, especially for big diameter cyclones. The ambiguities in the flow pattern make these correlations imprecise, and each example requires the coefficients to be determined independently. Reddy et al. (2014) made a significant contribution to this discipline. A proposed empirical correlation for gas core prediction was made by Steffens et al. (1993).

➢ *Fundamentally Based Hydrocyclone Models*

Previous attempts to comprehend the physical laws regulating hydrocyclone size separation resulted in ideas based on crowding, residence time, and balance. Later, comprehensive simulations are created, in which the Navier-Stokes Equation solution is used to estimate the movements of particles and fluids.

➢ *Computational Fluid Dynamics (CFD) Models*

The solution is complex because the governing fluid flow equations are non-linear simultaneous partial differential equations. Complete flow modeling of the hydrocyclone, including flow pattern, pressure and velocity fields, slurry concentration profile, turbulent viscosities, and slip velocities of the particles concerning liquid phase for wide range size distributions, can be simulated using CFD modeling.

Empirical models relate the dimensions of the device and the properties of the slurry to a classification parameter, like cut size. Hwang et al (2012) Because they can only be applied to the extremes of the experimental data—from which the model parameters are derived—empirical and semi-empirical models have significant limitations. This makes mathematical models—which are derived from fundamental fluid flow equations—very attractive, and models of this kind invariably contain the Navier-Stokes equation. Certain parts of hydrocyclone vertical flow difficulties have been resolved using mathematical models of this type that make oversimplified assumptions. A. Rietema et al (1961).

With a modified Prandtl mixing length model and axisymmetric assumptions, M. Rhodes et al. (1987) reported a general-purpose simultaneous PDE solver to solve the flow field in a hydrocyclone. The author reported well-fitted velocity prediction of a 200mm hydrocyclone for a single set of experimental conditions. This model's sensitivity and stability for various flow rates and device sizes were not verified. Computational Fluid Dynamics (CFD) has revolutionized the practice of fluid dynamics and is a highly valuable tool for flow pattern prediction in various contexts. CFD provides a practical, efficient, and cost-effective method for simulating hydrocyclones by using physical insights into the underlying hydrodynamics of the device's flow.

The ability to mathematically model basic physical processes has been made possible in recent years by the rapid development of powerful digital computers. This modeling can be extended to the geometry of hydrocyclones. Certain limitations apply to CFD design and modification in typical industrial settings, such as physical allocable memories and the overall CPU time needed for simulation. The overall number of cells and modeling strategy are mostly determined by these limits, with the latter being mirrored in the selection of the turbulence and other physical models.

As previously mentioned, attempts were undertaken to enhance specific performance metrics through the incorporation of structural alterations to the traditional hydrocyclones. CFD gives you the flexibility to swiftly alter the shape in order to verify potential modifications to the flow pattern and separation efficiency. The right choice of mesh type and other physical models, including turbulent ones, determines how accurate the findings are projected to be. Hexahedral meshes are known to be more accurate than tetrahedral meshes, although fitting complex body-fitted geometries with hexahedral meshes is time-consuming and frequently not feasible.

Previous works demonstrated numerical prediction of hydrocyclone flow field by axisymmetric 2D geometric assumptions, but these investigations were limited by computer power and resources. 3D geometries are now being thought about for simulation. The majority of research employ time-averaged Reynolds Averaged Navier-Stokes (RANS) equations, which account for turbulence through Reynolds Stress Modeling.

Chu et al. (2012) conducted research on a numerical simulation of hydrocyclone interior turbulence. The k-e model Hseih and Rajamani (1991) and the Prandtl mixing length model were employed in earlier research, however they are inappropriate for simulating rapidly whirling flow, such as that found in hydrocyclones, because the turbulence in these cases is anisotropic. A comparative analysis of the hydrocyclone flow field employing the Partition mechanism using CFD modeling was presented by Cullivan et al. and Schuetz et al. (2004). While Slack et al. (2002) created a CFD tool for the automated creation of hydrocyclones, Delgadillo and Rajamani et al. (2005) published a comparative assessment of hydrocyclone flow field using three distinct turbulence models.

➢ *Modeling of Hydrocyclone Flow Field for Non-Newtonian Fluids*

The majority of researchers assumed that the continuous phase is Newtonian when modeling the hydrocyclone flow field. However, in practice, majority of industrial situations, a hydrocyclone is fed with dense slurry. Newtonian theology is not present in dense slurries of various minerals, effluent samples, or the fluids of food processing facilities. The fluid-particle and particle-particle interactions in non-Newtonian media differ from those in Newtonian fluid media.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

In order to investigate the impact of viscosity on flow fields, Hseih, Rajamani, and colleagues created a mathematical model for hydrocyclone flow fields. The projected velocity profiles and separation efficiency of this model were in good agreement with the experimental results acquired from LDV. In order to offer an empirical model of water split and semi-empirical models to predict corrected cut size and hydrocyclone capacity, Tavares et al. conducted tests with fluids of various theologies. Upadrashta et al. (2012) used the perturbation approach to determine the tangential velocity profile inside a hydrocyclone for pseudoplastic fluids with an analytical momentum equation solution. They measured the impact of the adjusted Reynolds number and flow behavior index on tangential velocity while assuming a laminar flow field. According to the study, the point value of tangential velocity should have increased as the flow behavior index decreased. The study also forecasted modifications to the fluid's behavior and viscosity inside the hydrocyclone. The impact of dense media slurry on the hydrocyclone flow field and cut size was documented by Kawatra et al. in 1996. According to a recent study by Lin Yang et al. (2015), non-Newtonian behavior has a significant impact on the tangential velocity and flow field of hydrocyclones. The study employed RSM for turbulence, the Mixture model as the multiphase model, and CFD tools.

According to Suasnabar and Fletcher et al. (1999), turbulence, not the medium's religion, determines segregation in hydrocyclones. The separation of several theological models and hydrocyclone flow field simulations were implemented.

In a recent work, Vakamalla and Mangadoddy et al. (2015) used multiphase simulations to adopt the Herschel-Bulkley model of viscosity and discovered that the projected data is close to the experimental data collected by gamma-ray tomography and adequate for computing viscosity. According to Yablonski et al. (2013), yield stress significantly affects the flow field during particle separation in hydrocyclone hydrodynamics studies for fluids that follow the Herschel-Bulkley model.

III. CFD ANALYSIS OF HYDROCYCLONE

In any CFD analysis, there are three steps to be followed, i.e. pre-processing, solver, and post-processing. Preprocessing includes defining the flow domain, meshing, and assigning boundaries. The Solver part includes choosing proper and suitable models and schemes and running the simulation. The post-processing part includes validation of the result obtained from simulation, generation of contours, plotting graphs and finally reporting the results. In this section, the steps are discussed in detail.

A. Pre-Processing

Pre-processing is the first step of CFD analysis. Using different Computer Aided Design (CAD) packages like AUTOCAD, GAMBIT, TGrid, CATIA, SolidWorks, ANSYS, COMSOL etc. geometry can be created and meshing can be done. Here COMSOL Multiphysics for the 3d model has been used.

➢ *Geometry*

In the present study, the hydrocyclone flow field was modeled using the commercial CFD software package Comsol Multiphysics software. The figure 3 represents the geometry of the Hydrocyclone The 3D base geometry was created using the same. Dimensions of the hydrocyclone are mentioned below.

The Cylindrical Part; Radius = 80 mm; Height = 200 mm. The Conical Part Radius of base = 80 mm, Radius of apex = 7 mm, Height= 180 mm. The Vortex Finder Radius = 30 mm Height = 50 mm Depth of vortex finder inside hydrocyclone body = 30 mm.

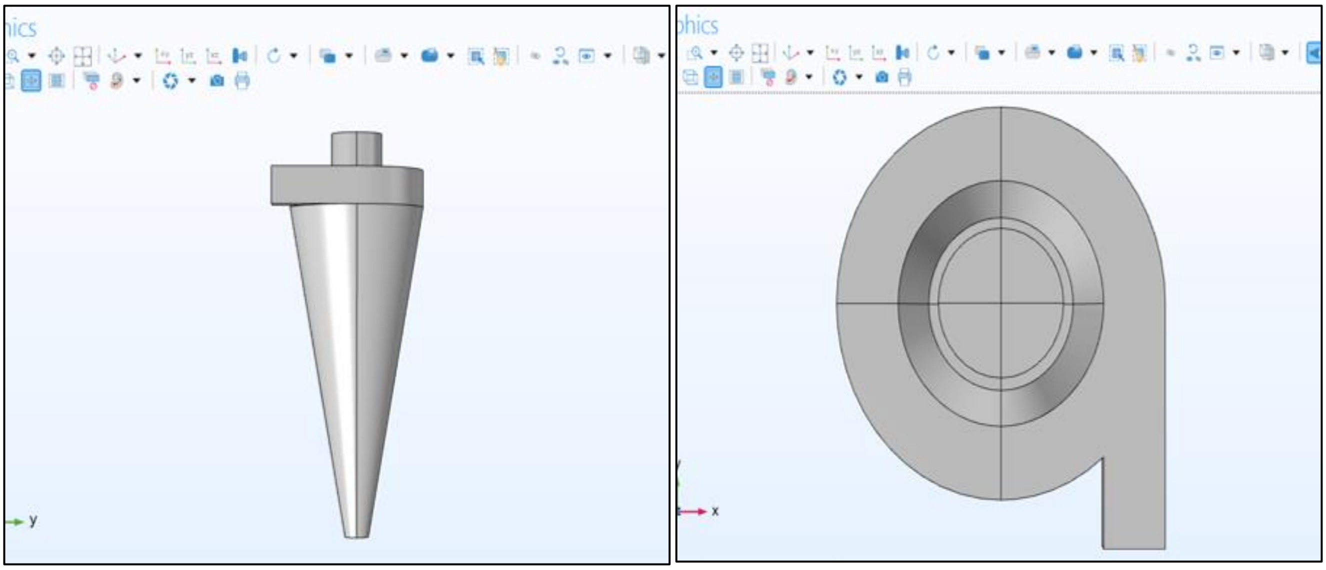


Fig 3: Geometry of the Hydrocyclone (Modeled in COMSOL Multiphysics)

➢ *Meshing*

Any CAD software program will offer a number of options for meshing a 3D geometry, including tetrahedral, hexahedral, hybrid, and others. Tetrahedral meshes are employed in the study to mesh the hydrocyclone geometry, and meshing is also carried out in COMSOL Multiphysics.

Hydrocyclone geometry is characterized by a lot of curved surfaces, acute angles, and other features that make it a difficult flow domain. In CFD research, this type of geometry is known as body-fitted geometry. Tetrahedral meshes are the best accessible meshing alternatives for body-fitting geometry. In situations where the inner portion is meshed using hexahedral meshes while the vicinity of the boundary is meshed with tetrahedral meshes, hybrid meshes can also be employed. Hexahedral meshes yield more accurate results and are more diffusive. However, they are unable to appropriately suit body-fitted geometry. Tetrahedral meshes are therefore recommended when dealing with body-fitted shapes. The figure 4 represents the meshed geometry.

The ability to compute the pressure field using the PRESTO, or pressure staggered, option is another benefit of utilizing tetrahedral meshes. Because the pressure field is kept on the staggered mesh separately from the velocity, PRESTO minimizes computational time by eliminating the need for interpolation when reconstructing the pressure field during iteration. When pressure and velocity fields are connected together, staggered meshes are created in PRESTO.

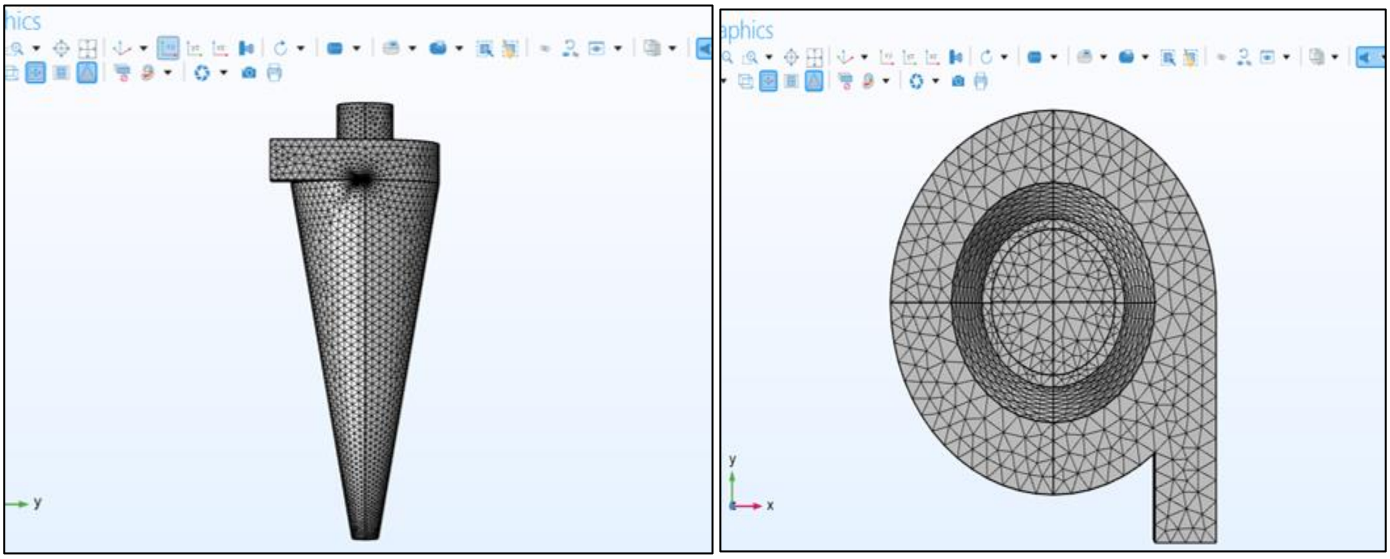


Fig 4: Meshed Geometry (Modeled in COMSOL Multiphysics)

➢ *Boundary Conditions*

To solve any problem using CFD techniques it is important to define the boundary conditions. It is the last step in pre-processing. There are three boundaries, i.e. one inlet and two outlets. The tangential inlet of hydrocyclone is marked as the "inlet" which is a velocity inlet, where feed velocity can be defined. Outlets are marked as "overflow" and "underflow" respectively. It is assumed that outlets are open to the atmosphere and hence they are pressure outlets, having zero gauge pressure and standard atmospheric conditions are applied at outlets. At both the outlets radial equilibrium pressure distribution is assumed.

B. *Mathematical Models*

Clarifying the direction of flow A number of mathematical models must be used for a thorough CFD analysis due to the intricate nature of the hydrocyclone's field. Isothermal conditions have been assumed in the research steady state, and Comsol's pressure-based solver was used to obtain the result. The following models of mathematics are used

- ➢ *Reynolds Stress Model (RSM)* - To compute the turbulence in the flow field.
- ➢ *Volume of Fluid (VOF)* — To track the liquid-air interface inside the hydrocyclone.
- ➢ *Modeling of Particle Motion* — To determine the trajectory of the dispersed particles in the flow field.
- ➢ *Power Law Model* — To compute the viscosity of the non-Newtonian fluid.
- ➢ *Herschel-Bulkley Model* — To compute the viscosity of non-linear viscoelastic fluid.

➢ *Reynolds Stress Model*

To compute turbulence in a flow field there are many models available, such as the Prandtl mixing length model, k-epsilon model, k-omega model, Reynolds Stress Model, Large Eddy Simulation (LES), Reynolds Averaged Navier-Stokes (RANS) equation and finally Direct Numerical Simulation (DNS).

Among all of these models, DNS provides the best result as it rigorously models the flow field and computes even the smallest scales of motion. However, DNS requires very high computational resources and takes a considerably long time for convergence.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

Simple models like Prandtl mixing length are hypothetical. Models like k-epsilon or k-omega are applicable for isotropic turbulence. But inside hydrocyclone the nature of turbulence is anisotropic and hence those models produce significant errors in computing turbulence.

Considering the computational resources RANS approach is used, where governing equations are averaged over some time, much larger than the period of turbulent fluctuations, and the resulting equations are integrated. Because of non-linearity of convective terms averaging of the governing equations produces an additional term called Reynolds Stress.

Comsol employs the RANS approach to avoid direct simulation of small-scale turbulent simulations. Employing this method, continuity of mass and momentum are expressed as follows.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \ldots\ldots\ldots\ldots (1)$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta\frac{\partial u_i}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-\underline{\rho u_i u_j}\right) \ldots\ldots (2)$$

In this case, the Reynolds stress term is denoted by — ρu_i u_j, where u_i is the velocity component term. By using the appropriate turbulence models, the unknowns in the aforementioned equation can be made closed. The intricacy of the flow, the amount of time needed for simulation, and accuracy all influence the choice of an appropriate model. RSM is a suitable alternative for simulating intricate and highly whirling flows, such as those found in hydrocyclones, since it accounts for the real behavior of turbulent viscosity and yields better results. Reynolds stresses are calculated by RSM using several transport equations, which are represented by

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta\frac{\partial u_i}{\partial x_j}\right)\right] + \frac{\partial}{\partial x_j}\left(-\underline{\rho u_i u_j}\right) \ldots\ldots (3)$$

The second term on the left term of the equation is referred to as the convection term, p_{ij} is stress production term, F_{ij} system rotation, G_{ij}, is for buoyancy. Turbulent and molecular diffusion are denoted by D_{ij}^T *and* D_{ij}^L respectively,, indicates pressure strain and, ε_{ij} indicates dissipation. A linear pressure strain model was employed to maintain solution stability.

- *Volume of Fluid Model*

Liquid-air interaction in hydrocyclone can be described as free surface flow. Hence in the modeling of hydrocyclone VOF is employed to track the interface of liquid-air, by solving the continuity equation for the volume fraction for each phase.

$$\frac{1}{\rho_q}\left[\frac{\partial(\alpha_q \rho_q)}{\partial t} + \Delta.(\alpha_q \rho_q v_q) = S_{\alpha q} + \sum_{p=1}^{n} \left(m_{pq}^* - m_{qp}^*\right)\right] \ldots\ldots (4)$$

where, α_q is the volume fraction of phase q and m_{pq} represents the mass transfer from phase q to phase P and $S_{\alpha q}$ represents source term,

The momentum equation is solved for the entire domain and the obtained velocity field is distributed among the phases. The momentum equation depends on the volume fractions of individual phases as density and viscosity vary for each phase,

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + \rho g_i \ldots\ldots (5)$$

Depending on the volume fraction, all other variables and attributes are computed at any particular cell. Based on the previously specified parameters, including the discretization model and order, the CFD solver performs the flow calculations and generates the output. We may alter the analysis at any point while it's being done because to the extensive interactivity of the Comsol CFD code.

- *Modeling of Particle Motion*

The motion of suspended particles in hydrocyclone is predicted with the particle trajectory method. The method provides a direct description of the particulate flow by tracking the trajectory of individual particles. The discrete Phase Model is used for tracking the trajectory of individual particles, which are described in detail by Leduc et al. (2006) trajectories of particles are computed in a Lagrangian frame and turbulent modeling can be done by either the particle cloud model or Stochastic tracking method. The Lagrangian multiphase model is used to describe the motion of a particle.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

The trajectory of a particle is obtained in DPM by integrating the force balance equation. Forces acting on a particle in a hydrocyclone are centrifugal force, drag force, buoyancy and gravity. The force balance equation is given as:

$$\frac{dv}{dt} = F_D(u_i - v_i) + \frac{(\rho_p - \rho)}{\rho} g_i \ldots\ldots\ldots\ldots\ldots (6)$$

Where $F_D(u_i - v_i)$ is the drag force per unit mass of particle, which is given by:

$$F_D = \frac{18\mu}{\rho_p d_p^2} C_D \frac{Re_p}{24} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (7)$$

Here, the vector u_i is particle velocity and v_i is fluid velocity, ρ_p is particle density, d_p is particle diameter, and C_D is drag coefficient, which depends on the particle Reynolds number Re_p, Which is defined as

$$Re_p = \frac{\rho d_p |v_i - u_i|}{\mu} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (8)$$

Every particle's trajectory is calculated over an extensive number of steps as it moves across the flow domain. DPM assumes that there are less than 10% of solids in the fluid phase. No interaction with the fluid phase, which is the continuous phase in this instance Particle-particle interaction, integration, and disintegration are taken into consideration because these scenarios demand additional models and are computationally costly. It is assumed that the particles are inert and that surface injections occur at the inlet normal to the inlet surface. A higher solid loading can be achieved by using DEM collisions.

The trajectory of the particles in the discrete random walk tracking model is obtained by using the instantaneous fluid velocity u+u' along their path. To find the turbulent dispersion of individual particles, these equations are integrated. The values of u' are measured under the assumption that they follow the Gaussian probability distribution and persist during the turbulent eddy's lifetime:

$$\mu' = \xi\sqrt{\overline{u'^2}} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (9)$$

Where, ξ is a normally distributed random number.

- *Power Law Model*

A hydrocyclone typically separates certain solids from liquids, such as water. The majority of scientists believe that the liquid follows Newtonian laws. Solids are introduced inside the hydrocyclone using DPM while "tracking the trajectory of particles" is being done. CFD analysis only computes the pressure and velocity field inside the hydrocyclone for the liquid phase. However, the majority of industrial effluents and mineral slurries are actually Non-Newtonian in nature and have complicated rheology.

As a result, it is hypothesized that the fluid inside the hydrocyclone obeys the power law model of viscosity, which describes

$$\tau = k\gamma^n \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (10)$$

Where, τ is shear stress tensor and S is the velocity gradient tensor. k and n are constants for a given fluid, k is called flow consistency index and n is called flow behavior index.

Viscosity μ is computed as:

$$\mu = k\gamma^{n-1} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (11)$$

The fluid is referred to as a pseudoplastic fluid if n<1, and its viscosity reduces as shear increases. We refer to this type of action as shear thinning. The fluid is referred to as dilatant if n>1, and its viscosity rises as shear increases. We refer to this type of activity as shear thickening. This involves running simulations for pseudoplastic fluids, or fluids with n less than 1.

- *Herschel-Bulkley Model*

Since yield stress is present in the majority of mineral slurries, their behavior is more akin to viscoelastic fluids than non-Newtonian fluids. In this investigation, the Harschel-Bulkley model has been utilized to calculate the viscosity of a viscoelastic fluid. The suspension of coal and water has been used as a working fluid. The fluid has yield stress rheology and shear thinning behavior. This is how the Herschel-Bulkley model is explained

$$\tau = k\gamma^n + \tau_0 \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (12)$$

Where τ is shear stress, τ_0 is yield stress, y is strain rate tensor. If $\tau < \tau_0$, the fluid behaves like solid and y=0.

If $\tau > \tau_0$, the flow occurs and y$\neq$ 0.

viscosity is computed as:

$$\mu = k\gamma^{n-1} + \frac{\tau_0}{y} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (13)$$

C. Solver Execution

A crucial component of numerical simulations is solver execution, which is essential for resolving intricate systems of equations controlling a wide range of physical events. The discretized equations are iteratively solved by the solver using complex techniques, such as finite element or finite volume methods, over the computational domain. The solver executes in multiple iterations, fine-tuning solution parameters until convergence requirements are satisfied. To maintain accuracy and stability, efficient solver execution necessitates careful consideration of elements such mesh quality, time step size, and solver settings. Furthermore, solution times can be accelerated by using parallel computing techniques, particularly for simulations that require a lot of work. By carefully controlling solver execution, COMSOL enables scientists and engineers to get trustworthy

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

understanding of various multiphysics phenomena, which helps them create and optimize novel engineering systems and procedures.

D. Post-Processing

This is the final step of CFD analysis and it involves the organization and interpretation of the predicted flow data and the production of contours, plots and animations. Comsol includes full Post-processing features. For plotting of the data Origin Pro 8.5 was also used. The figure 5 represents the velocity for Non-Newtonian flow inside the hydrocyclones.

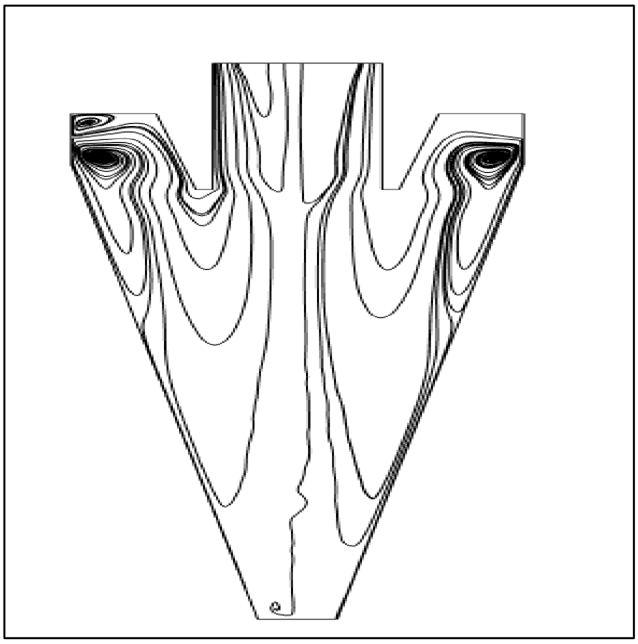


Fig 5: Velocity Profile for Non-Newtonian Fluid Inside the Hydrocyclone

E. Validation of Model

The data generated by CFD simulations are predictions, hence they must be verified by experimental or previously published data. The simulated data in the report is not verified through any experimental effort, which also stays as the study's future focus. A small body of published literature is used to validate the data. A mathematical model for predicting the axial and tangential velocity profile of non-Newtonian pseudoplastic fluids in a hydrocyclone was proposed by Hsiesh et al. (2002). The data from the CFD simulation also complies with Upadrasta et al.'s 1987 prediction that the intensity of tangential velocity will increase as the flow behavior index value decreases. The existence of an air core is not documented in the literature that was previously referenced, and the simulation results did not reveal the existence of a fully developed air core either. Thus, it can be said that the CFD model has been validated and the assumptions are accurate. The figure 6 represents the velocity and radius of curvature for Newtonian and Non Newtonian fluids, whereas the figure 7 represents the velocity vs radius curve nature for our pseudoplastic fluid.

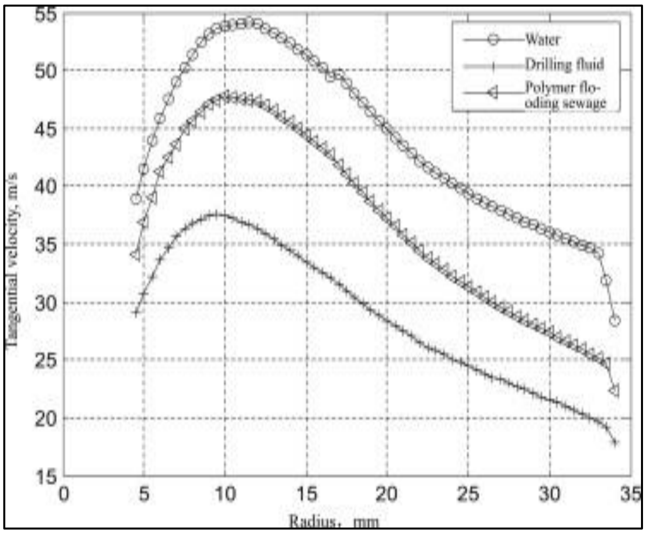


Fig 6: Velocity vs Radius Curve Nature for Newtonian and Non-Newtonian Fluids (L Yang et al. (2015))

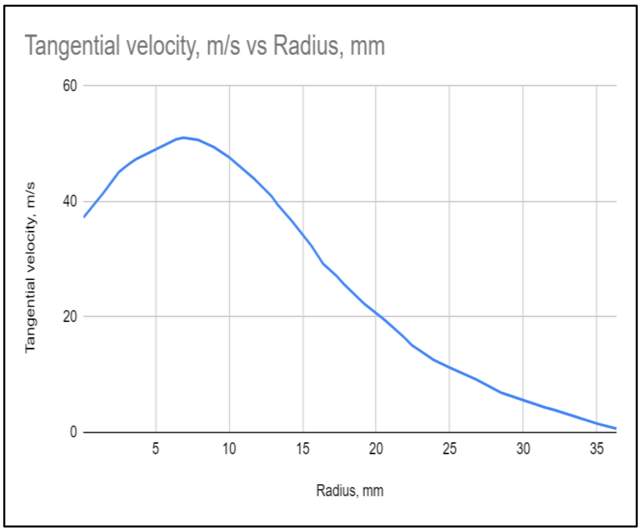


Fig 7: Velocity vs Radius Curve Nature for our Pseudoplastic Fluid

L.M. tavares et al.(2002) stated an equation relating shear rate to the velocity of incoming fluid at the cross-section of inlet of hydrocyclone as

$$\gamma = \frac{8n\lambda Q}{\pi D_i^2 D_c \; r_*^{n+1}} \ldots\ldots\ldots\ldots (14)$$

where λ is velocity reduction parameter

$$\lambda = 4.5(\frac{D_i}{D_c})^{1.13} \ldots\ldots\ldots\ldots (15)$$

Where Q is the flow rate at the inlet of hydrocyclone, D_i is the inlet dia of the hydrocyclone and D_c is the dia of the right cylindrical part of the hydrocyclone. Where r_* is the dimensionless radial distance in the hydrocyclone.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

$$r_* = \frac{2r}{D_c} \dots\dots\dots (16)$$

$$Q = \frac{\pi D_i^2}{4} u_{in} \dots\dots\dots (17)$$

Where u_{in} id the velocity at inlet.

After considering the equation discussed in 3.5 part, a simplify equation is formed as:

$$\gamma = \frac{2n(\frac{D_i}{D_c})^{1.13} u_{in}}{D_c} \dots\dots\dots (18)$$

From the above value of shear rate and power law we can plot the shear stress vs shear rate graph and check out whether the curve is following non-Newtonian nature. It is evident from the plot that it is showing pseudoplastic behavior as evident in the figure 8.

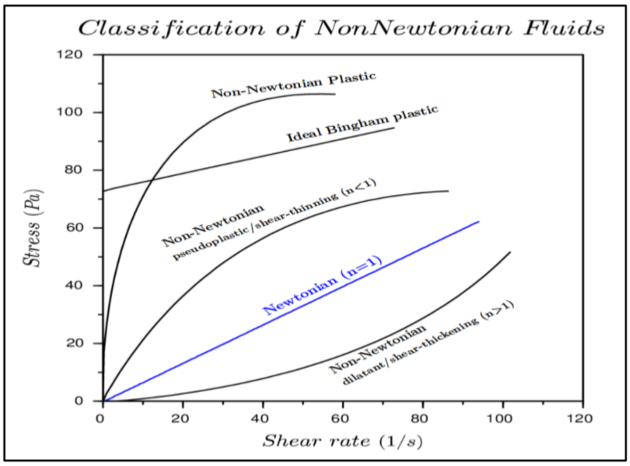


Fig 8: Shear Stress vs Shear Rate Curve for Different Fluids (Nguyen et al.(2012))

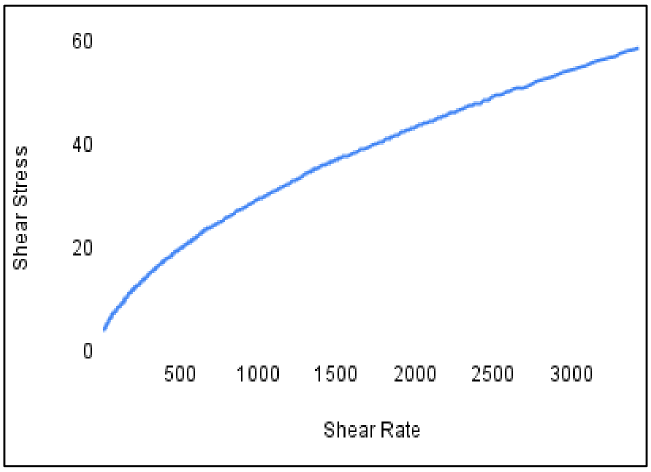


Fig 9: Shear Stress vs Shear Rate Curve Nature for our Pseudoplastic Fluid

The figure 9 represents the shear stress versus shear rate curve nature obtained in our simulation of pseudoplastic fluid.

F. Results and Discussion

The outcomes covered in this section are derived from simulations that were run utilizing the previously stated technique in standard hydrocyclone shape. They show broad patterns in the viscosity profile, pressure, and velocity fields. The three velocity components—tangential, axial, and radial velocity—are taken into consideration while making velocity field predictions.

- *Pressure Field Prediction*

The static pressure profile under multiphase flow conditions reveals that an area of low pressure forms around the hydrocyclone's axis due to the conservation of angular momentum. Nevertheless, the lack of a completely developed air core is indicated by the static pressure not being negative at every location along the axis. There is negative static pressure in the vortex finder, indicating that the air core is only partially developed and not entirely grown. The low-pressure area could resemble a cylinder with waves in it. Maximum pressure is recorded at the wall, and pressure rises radially from the axis to the wall. The figure 10 represents the enhanced static pressure near the overflow due to eddy flow circulation in the zones.

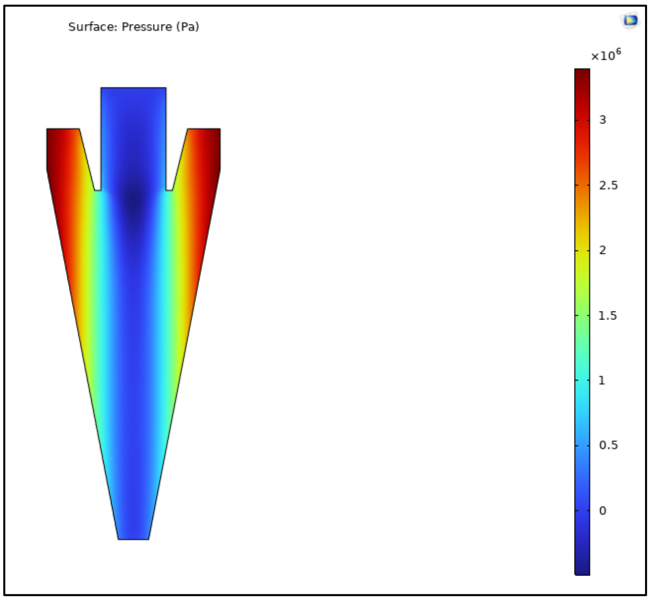


Fig 10: Static Pressure Contour for Feed Inlet Velocity 10m/s and Rheological Constant k=12.9 and n=0.41. Units are in Pascal

- *Velocity Field Prediction*

There is a noticeable increase in tangential velocity overall as it passes through the wall and toward the axis. Tangential velocity is lowest close to the wall because typical wall functions without slip are presumed. velocity increases from wall to maximum. Tangential velocity rapidly decreases with a very high velocity gradient at a point along the axis that interfaces with the low pressure zone. This trend is observable on the contours derived from multiphase simulation results and is suggestive of the dominant flow feature, the Rankine vortex. Reverse flow and stationary zones are seen in a relatively small number of locations close to the wall and vortex finder.

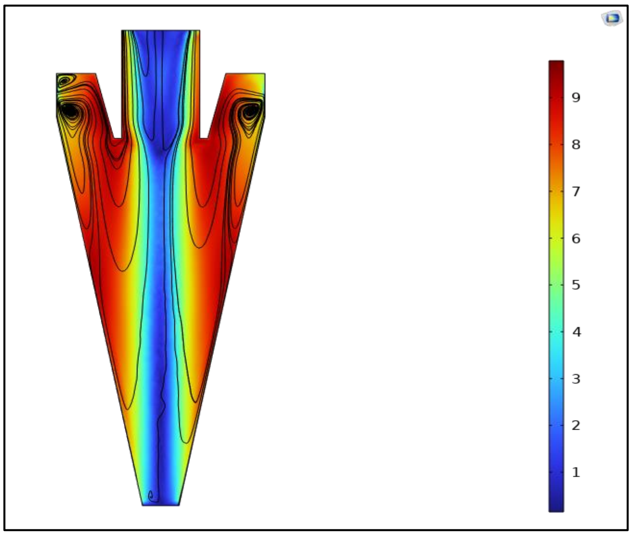


Fig 11: Contour of Tangential Velocity for Feed Inlet Velocity 5m/s and Rheological Constant k=12.9 and n=0.41.

The above figure 11 represents the tangential velocity for feed inlet showing higher velocities near the periphery.

➢ *Axial Velocity*

Contours of axial velocity reveal the phenomena of axial flow reversal inside hydrocyclone. Near the low pressure zone there is negative axial velocity, i.e. flow is in upward direction. Negative axial velocity is responsible for creating an upward direction drag force on particles in a low pressure zone. Near underflow axial value is positive, indicative of separation of particles through underflow. Flow reversal signifies the presence of the Locus of Zero Vertical Velocity (LZVV). LZVV is an envelope, which separates the upward inner vortex from downward outer Vortex. Outside LZVV axial velocity is always positive, i.e. particles outside LZVV will be rejected through underflow and inside LZVV axial velocity is negative and particles in this region will be rejected through overflow.

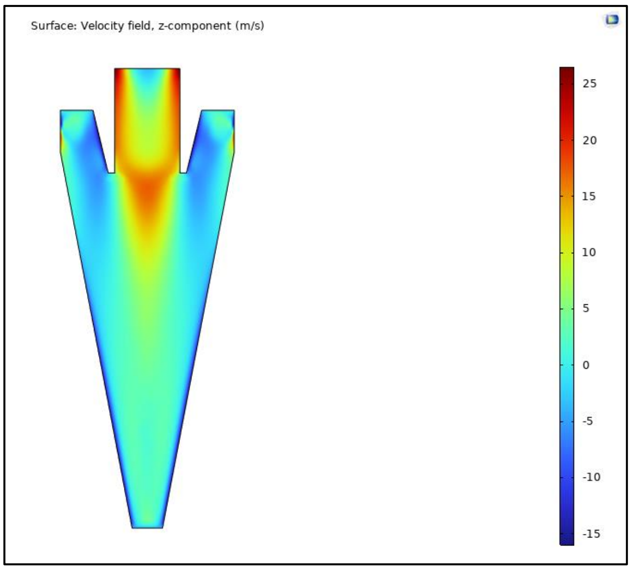


Fig 12: Contour of Axial Velocity for Feed Inlet Velocity 10m/s and Rheological Constant k=12.9 and n=0.41. Units are in m/s.

The figure 12 represents the axial velocity for feed inlet velocity 10 m/s and rheological constant depicts higher velocity near the overflow somehow to some extent throttling has taken place.

➢ *Viscosity Prediction*

The working fluid is assumed to be non-Newtonian. It is known that viscosity decreases with increasing shear stress for pseudoplastic fluids. From the data obtained from the simulation, a decrease in viscosity from the axis towards the wall is observed, and near-wall viscosity starts rising again. Near wall shear is maximum and viscosity is low because of pseudoplasticity. Although we assumed the feed liquid to be pseudoplastic, the rheology of slurries depends upon their physical properties and packing characteristics of suspended particles under shear.

Centrifugal force carries the heavier particles toward the wall and therefore the near-wall concentration of solids rises, which is the reason behind the increase in viscosity near the wall. It is interesting that at the low-pressure zone velocity gradient is very high as well as viscosity, which is contradictory to the rheology of pseudoplastic fluids.

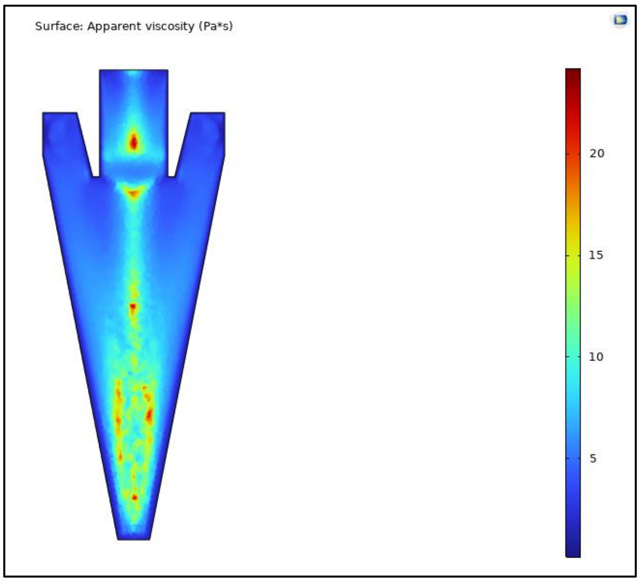


Fig 13: Contour of Effective Viscosity for Feed Inlet Velocity 5m/s and Rheological Constant k=12.9 and n=0.41, units are in m/s.

Figure 13 represents the contour of effective viscosity for feed inlet velocity 5 m/s and rheological constant k=12.9 and n=0.41, shows increased apparent viscosity near the overflow.

➢ *Pathline*

It is possible to mimic the detailed observation of particle trajectories and the analysis of each particle pathline's flow path based on particle ID. Inside the hydrocyclone, there is a definite sign that two distinct opposing vortices have formed. The two vortices are moving in opposition to one another. The overflow is notified by particles in the inner forced vortex, whereas particles in the outer free vortex report to the underflow. Comparing the

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380

simulation data to other literature, nearly same types of pathlines are obtained. Consequently, it can be said that the track of particles inside hydrocyclones is significantly influenced by rheology. However, since the residence times of particles in two distinct types of fluids differ, rheology has a major impact on particles.

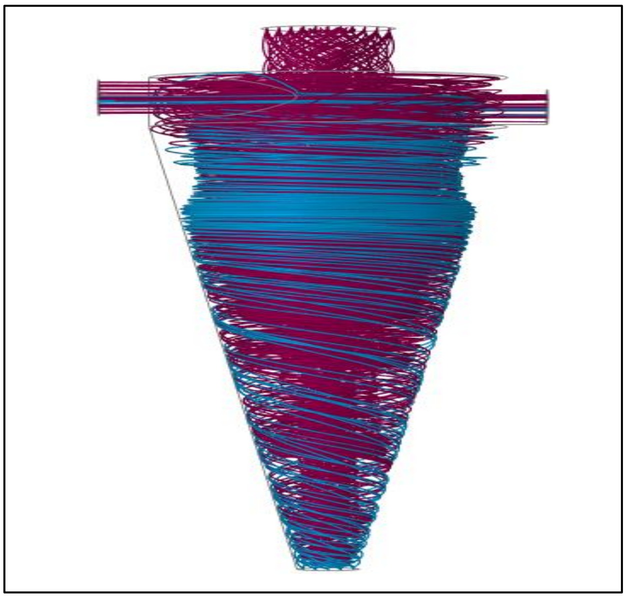


Fig 14: Pathline of Fluid for Feed Inlet Velocity 10m/s and Rheological Constant k=12.9 and n=0.41. Units are in m/s.

Figure 14 represents the pathline of the fluid for various feed inlet velocities depicting the swirling motion inside the hydrocylone.

G. Inference

It was determined which models—turbulence, multiphase, and others—were appropriate for multiphase CFD simulation using particular discretization techniques. Pressure, velocity, and viscosity profiles that are physically plausible and consistent with the literature were acquired.

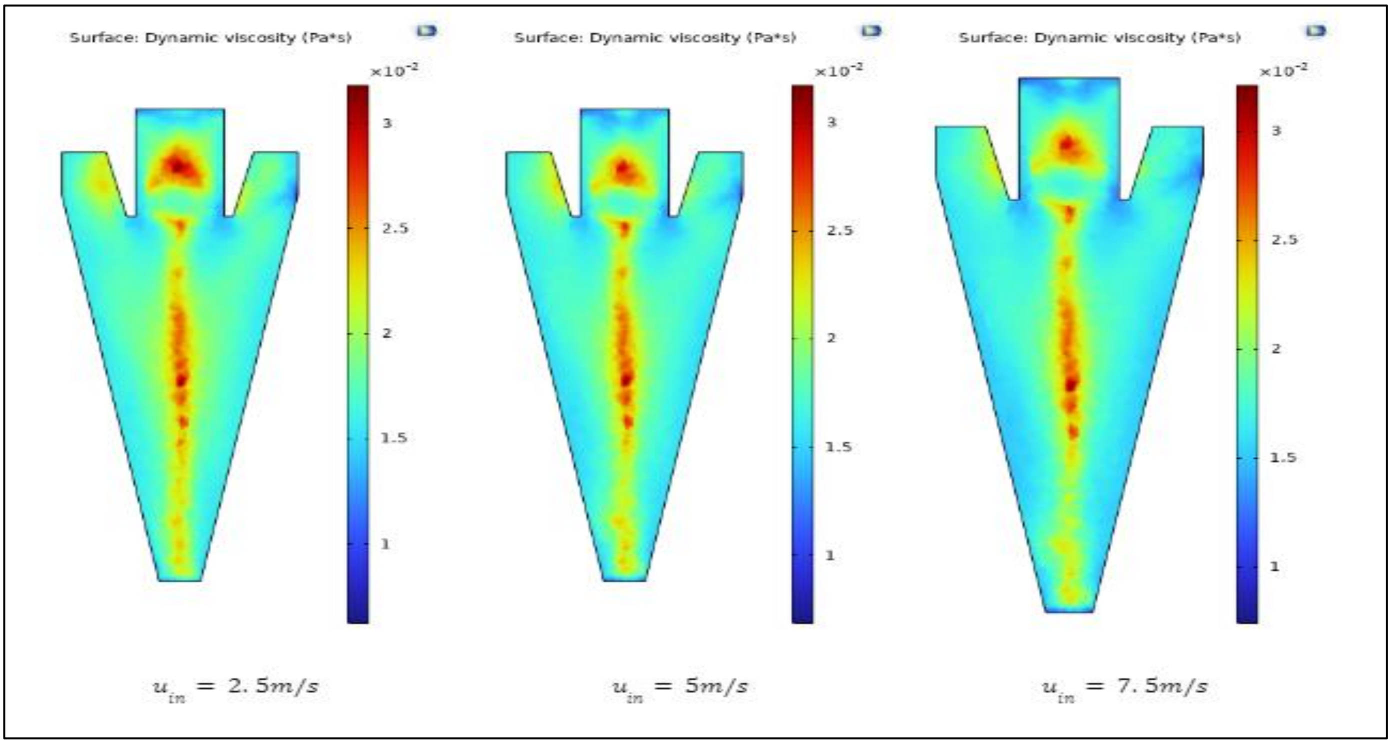
For multiphase simulations, it was discovered that the solver never converged to a stable state. Small-amplitude periodic oscillations were visible in the residuals. In line with the findings in the literature, it was not able to achieve a constant steady state since the pressure and velocity fields changed occasionally Schuetz et al (2004).

Particle paths match the results of earlier experiments and published research. The result's consistency is contingent upon the pressure and velocity fields being pre-established. Therefore, better results are obtained by using improved meshes and selecting appropriate discretization algorithms. As a result, CFD offers promise as a tool for investigating and forecasting the impact of modifications to operational factors. In the following sections of the study, the performance of a hydrocyclone with non-Newtonian fluids under various feed flow rates and variations in the flow field for various non-Newtonian fluids are examined.

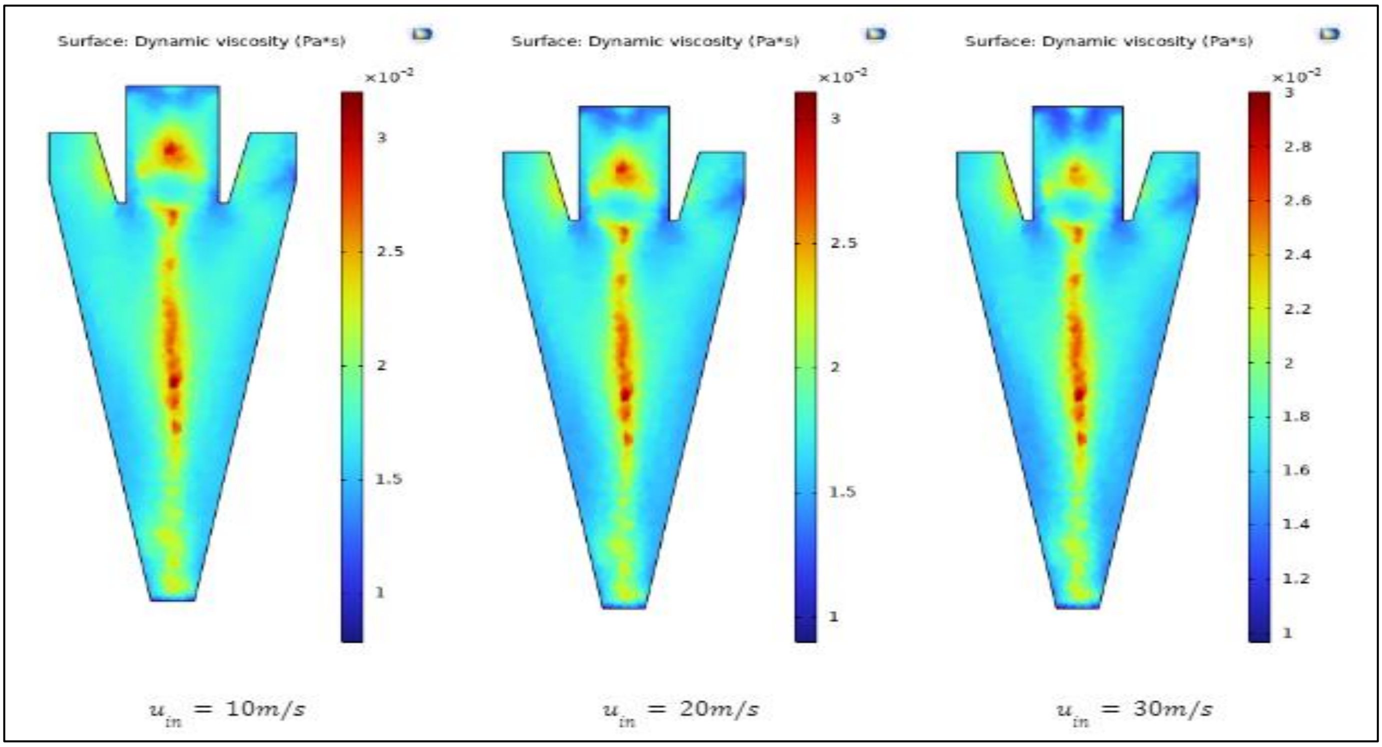
IV. RESULTS AND DISCUSSION

A. Static Pressure

In the static pressure graphs for the inlet velocities 2.5 m/s, 5 m/s, and 7.5 m/s the central core has rotational velocities In the central region of Figure 16 swirling flow and rotational velocities are there giving rise to increased dynamic viscosities.



(a)



(b)

Fig 15: Contour of Static Pressure for Different Feed Inlet Velocities

The figure 15 represents the contour of static pressure for different feed inlet velocities depicting the contpurs of pressure building up on the central core region.

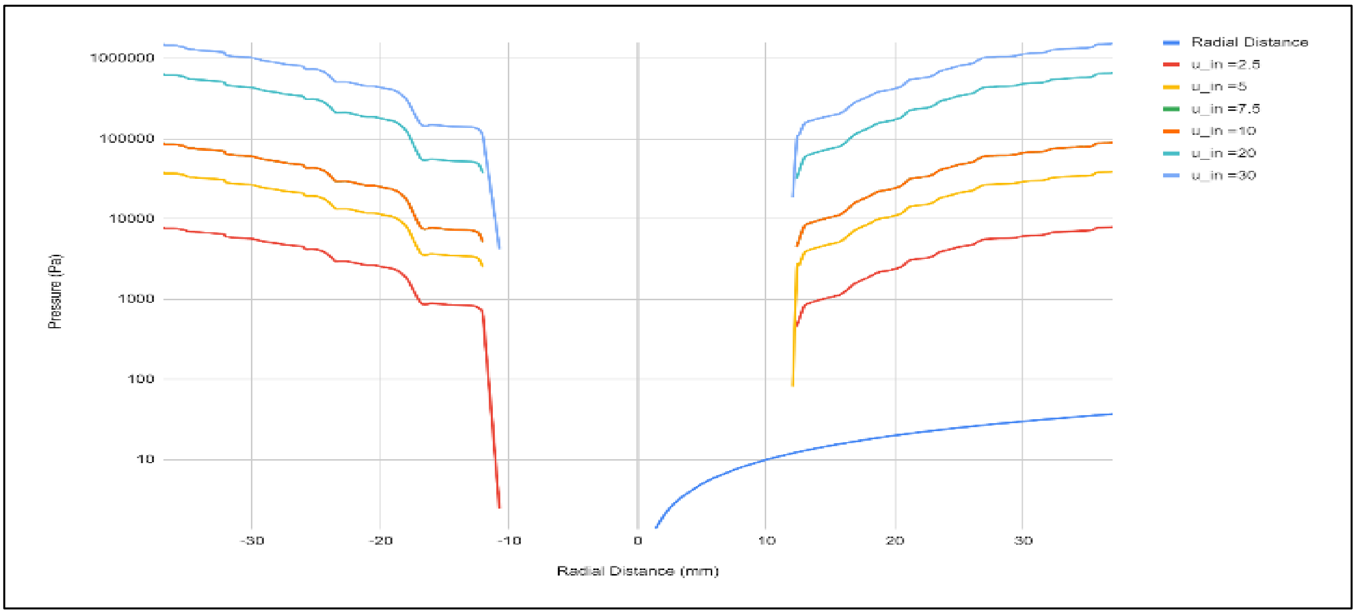


Fig 16: Comparison of Static Pressure Profile for Different Inlet Velocities

The comparison of the mean radial fluctuating pressure is given in the left section of the figure 16. At axial location 60 mm from the top of the cyclone, we observe a very good comparison of the dynamic k-equation subgrid model predicted axial fluctuation levels with the experimentally reported values in both the near wall and free vortex regions. Slight over-prediction in the forced vortex region was observed. The WALE sub-grid model show under-predicted velocity fluctuation levels in all the regions.

B. Velocity Field

As discussed, earlier velocity inside hydrocyclone has three components, of which two are responsible for separation process, i.e. axial and tangential velocity. In this section, Axial and tangential velocity fields are discussed.

➢ *Tangential Velocity*

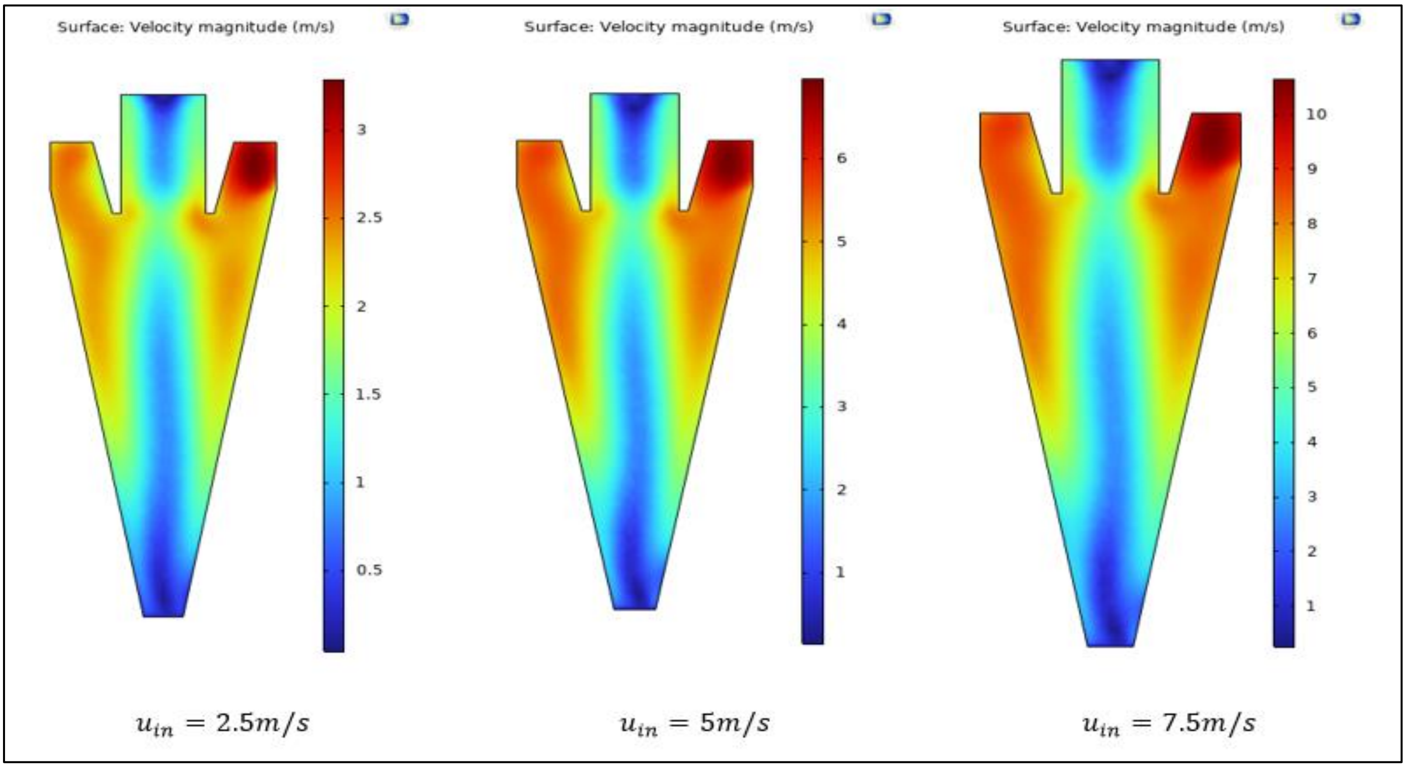


Fig 17: Contours of Tangential Velocities for Feed Inlet Velocity

In figure 17 the velocity magnitude represents the eddy circulation. that the particles of smallest size mostly get segregated at the bottom of the separator in the conical region noticed from the relatively high concentrations near the spigot region. The particles of intermediate size and coarser size particles, generally get segregated at the top region of the cyclone near feed inlet and the bottom tip of the vortex finder, through which they finally get discharged as overflow. However, no tendencies of these two size-class particles were observed while moving to the lower part of the cyclone (or through the spigot). The figure 18 represents the comparison of the tangential velocity profiles for different feed inlet velocities.

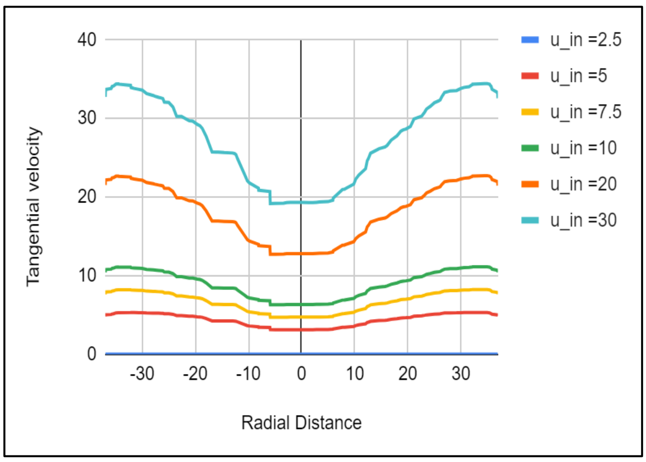
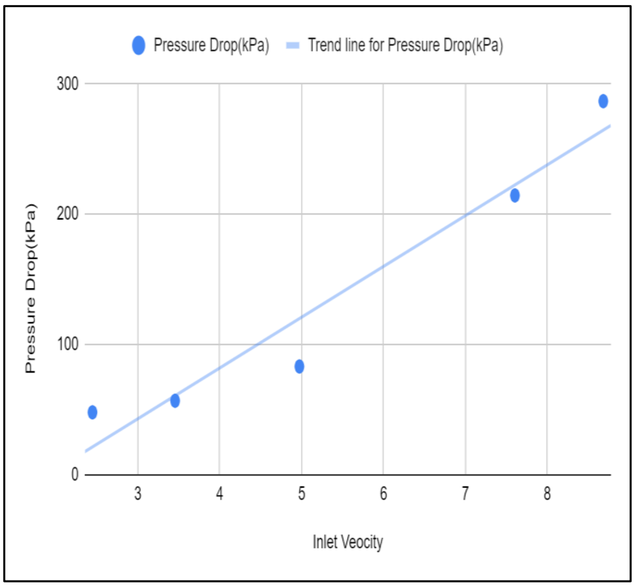


Fig 18: Comparison of Tangential Velocity Profiles for Different Feed Inlet Velocities

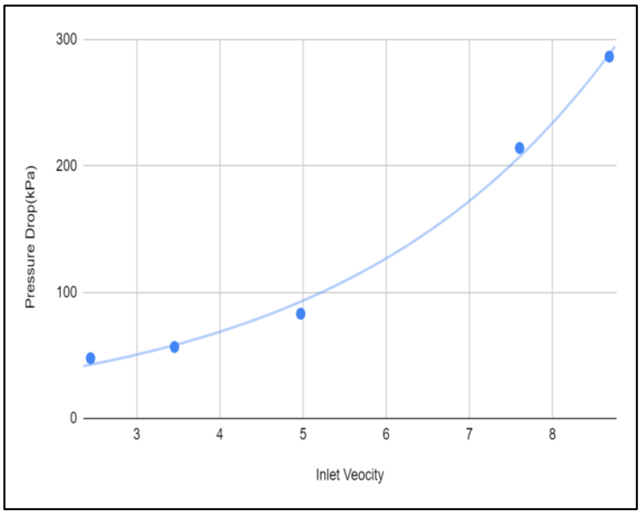
C. *Pressure Drop*

Total pressure drop inside hydrocyclone can be computed numerically using CFD software. Total pressure drop has been calculated using Mass- Average volume integral of all computational cells. It is known that with increasing velocity, turbulence inside hydrocyclone will increase and pressure drop will also increase. Pressure drop is plotted against inlet velocity and fitted in both linear and parabolic regression. Parabolic fit gives the best result.



(a) Linear Fit

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/IJISRT24APR2380



(b) Parabolic Fit
Fig 19: Characteristics of Pressure Drop vs. Feed Inlet Velocity

D. Viscosity Profile

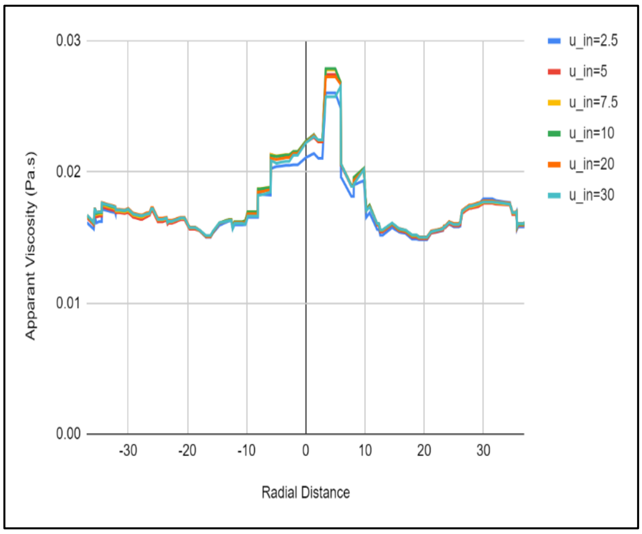


Fig 20: Viscosity Profile for Different Feed Inlet

low inlet velocities, velocity gradient at low pressure zone is very less but in the figure magnitude of viscosity at low pressure zone is very high. The figure 19 and 20 represents the characteristics of Pressure Drop versus Feed inlet velocities and the viscosity profile for different feed inlet. This is contradictory to previous discussion about viscosity profile. The reason is that for low inlet velocities, a highly turbulent field does not develop and velocity gradient is low therefore. Velocity gradient does not exceed the threshold limit of pseudoplasticity. Because of poor turbulence, at the central region shear stress is very low and fluid offers high resistance to flow because of its rheological property.

In case of high flow rate, turbulent field is well developed and viscosity profile has already been discussed in earlier section. The near wall viscosity increases with increasing inlet velocity, which is a fascinating phenomenon. However, as was previously said, because there would be less applied shear close to the wall, the minimum viscosity value should increase with decreasing flow rate. As was previously said, as inflow velocity increases, turbulence within the hydrocyclone rises and the degree of separation increases correspondingly. Coarser particles accumulate close to the hydrocyclone wall, which causes the viscosity to rise. The viscosity profile does not accurately follow the trend at an intake velocity of 2.5 m/s. It is outside the purview of this study's explanation. Although the likelihood of a change in flow behavior is uncertain, it is still within the purview of future research.

E. Inference

It is clear from the quick analysis of the hydrocyclone flow field that the hydrocyclone's performance increases with inlet velocity. It is recommended to maintain a high feed inlet velocity in a hydrocyclone since turbulence plays a crucial role in the separation of particles. A higher pressure drop results from increased turbulence caused by increasing feed velocity. Additionally, a parabolic increase in pressure drop with feed input velocity is demonstrated. For optimal performance, flow rate optimization should be carried out. Since the viscosity change is still not fully understood, further research on this topic is still needed.

V. CONCLUSIONS

CFD is a versatile technique for forecasting flow patterns in any fluid mechanics problem, and it may be used to meet a broad range of operating and design circumstances. Advances in computational methods and computer technology have rendered CFD a useful tool for studying the flow dynamics of a variety of physical problems.

In the current work, three-dimensional CFD simulations were performed using RSM. The same set of solver parameters were used in every simulation to facilitate comparison. Tangential velocity increases as one advances away from the Rankine vortex, reaches a maximum, and then lowers again because of typical wall functions and the boundary layer. Two opposing vortices rotate inside the hydrocyclone to form a constant pressure field. The wall of the hydrocyclone has the maximum pressure, while the central axis has the lowest. As the rotating radius reduces, pressure decreases as a result of an increase in tangential velocity.

To examine the effect of a non-Newtonian fluid on the hydrocyclone flow field, the entrance velocity is varied. Axial, tangential, and static pressure observations are made in order to predict the separation properties. Viscosity is also mentioned. Axial and tangential velocities are measured while different rheological fluids are provided as feed in order to understand the effect of non-Newtonian rheology. Pathlines are simulated to confirm the flow of particles inside the hydrocyclone. Pathli ensures that the particles within a hydrocyclone produce both forced and free vortices, as well as helical patterns of flow.

The flow field flow rate of both viscoelastic fluids was changed in order to comprehend the impact of non-Newtonian fluids on hydrocyclones. It was discovered that improved separation occurs at increasing inlet velocities. The influence of the turbulent field is minimal and the separation is quite poor at low input velocities. But high pressure results from rising inlet velocity. Therefore, in order to achieve the best performance, optimization is required.

For pseudoplastic fluids that obeyed the power law model, the flow behavior index was modified in order to examine the impact of rheology on the flow field. The findings show that rheology has no discernible impact on axial velocity or pressure field. The fluid rheology is found to have a substantial effect only on tangential velocity. It may be inferred that when the fluid's pseudo-plasticity grows, tangential velocity also increases, indicating a corresponding increase in separation sharpness.

➢ *Future Research Scope*

- The models were validated using a few findings from past studies. Experimental confirmation of the work is still included in the future scope of the project.
- Other rheological models for viscoelastic fluids, like the Cross model, can be used to study the effect on the hydrocyclone flow field.
- Every simulation employed the same geometry for hydrocyclones. Changes to the hydrocyclone design can be made to examine how geometric parameters affect the non-Newtonian flow field inside the hydrocyclone and its separation qualities.

REFERENCES

[1]. Bhaskar, K. U., Murthy, Y. R., Ramakrishnan, N., Srivastava, J. K., Sarkar, S., & Kumar, V. (2007). CFD validation for fly ash particle classification in hydrocyclones. Minerals Engineering, 20(3), 290-302.
[2]. Goyal, A., Roy, P., & Banerjee, P. K. (2010). Effect of air core on flow rate and split in a hydrocyclone. In Proceedings of the XI International Seminar on Mineral Processing Technology (MPT-2010) (Vol. 1, No. Section 2, pp. 124-131). National Metallurgical Laboratory.
[3]. Ji, L., Paul, P., Shanbhag, B. K., Dixon, I., Kuang, S., & He, L. (2023). Emerging application of hydrocyclone in biotechnology and food processing. Separation and Purification Technology, 309, 122992.
[4]. Kuo-Jen Hwang, Ya-Wen Hwang, Hideto Yoshida, Kazuha Shigemori, &quot (2012); Improvement of particle separation efficiency by installing conical top-plate in hydrocyclone " Powder Technology 232 41-48.
[5]. K. Rietema, &quot (1961); Performance and design of hydrocyclones-I: General considerations ", Chemical Engineering Science 15 (1961) 298-302.
[6]. Liu, Y., Cheng, Q., Zhang, B., & Tian, F. (2015). Three-phase hydrocyclone separator–A review. Chemical engineering research and design, 100, 554-560.
[7]. M. Rhodes, D. Geldart, &quot (1987); A model for the circulating fluidized bed ", Powder Technology 53 155-162.
[8]. Narasimha, M., Brennan, M., & Holtham, P. N. (2007). A review of CFD modeling for performance predictions of hydrocyclone. Engineering Applications of Computational Fluid Mechanics, 1(2), 109-125.
[9]. N. Yoshioka, Y. Hotta, &quot (1955) ;Liquid cyclone as a hydraulic classifier ", Journal of Chemical Engineering of Japan 19 632-640
[10]. Nguyen, Hung & Nguyen Ngoc, Diep. (2012). Incompressible Non-Newtonian Fluid Flows. 10.5772/26091.
[11]. Oliveira, D. C., Almeida, C. A., Vieira, L. G., Damasceno, J. J., & Barrozo, M. A. (2009). Influence of geometric dimensions on the performance of a filtering hydrocyclone: an experimental and CFD study. Brazilian Journal of Chemical Engineering, 26, 575-582.
[12]. Ortega-Rivas, E. (2004). Applications of the liquid cyclone in biological separations. Engineering in life sciences, 4(2), 119-123.
[13]. P. Fahlstrom, &quot (1963); Studies of the hydrocyclone as a classifier " Proc 6th International Mineral Processing Congress, Cannes Pergamon, London 87-112.
[14]. Shingote, C. (2018). Experimental and CFD Investigations of the Characteristics of Fluid Flow and Air Core Inside a Hydrocyclone Separator (Master's simulation, Case Western Reserve University).
[15]. Svarovsky, L. (2001). Hydrocyclones. In Solid-Liquid Separation (pp. 191-245). Butterworth-Heinemann.
[16]. Vakamalla, T. R., Kumbhar, K. S., Gujjula, R., & Mangadoddy, N. (2014). Computational and experimental study of the effect of inclination on hydrocyclone performance. Separation and purification technology, 138, 104-117.
[17]. Yang, L., Tian, J. L., Yang, Z., Li, Y., Fu, C. H., Zhu, Y. H., & Pang, X. L. (2015). Numerical analysis of non-Newtonian rheology effect on hydrocyclone flow field. Petroleum, 1(1), 68-74.
[18]. Zhao, Q., Cui, B., Ji, A., Song, T., & Shen, Y. (2024). Experimental and numerical study of the effect of particle size distribution on hydrocyclone classification. Advanced Powder Technology, 35(4), 104398.
[19]. Ji, L., Paul, P., Shanbhag, B. K., Dixon, I., Kuang, S., & He, L. (2023). Emerging application of hydrocyclone in biotechnology and food processing. Separation and Purification Technology, 309, 122992.