Proceedings of the 3rd International Conference on Mechatronics and Smart Systems
DOI: 10.54254/2755-2721/2025.23355

Comparative Analysis of the Flow Field Characteristics of Axial Flow, Radial Flow, and Tangential Flow in Stirring Equipment

Zikun Xie

Engineering in Mechanical Technology, PSB Academy, Singapore
DEITY666@126.com

Abstract: Stirring equipment is widely used in industries such as chemical, pharmaceutical, and food processing, where its performance directly affects mixing, mass transfer, and heat transfer efficiency. The main flow modes in stirring processes include axial flow, radial flow, and tangential flow, whose interactions determine the overall stirring performance. This study integrates fluid mechanics principles to analyze the characteristics of these different flow patterns and explores how optimizing the design of stirring equipment can enhance efficiency. Furthermore, this paper utilizes computational fluid dynamics (CFD) simulations and experimental research to examine the influencing factors of different flow modes and proposes optimization strategies. The study further investigates how specific impeller configurations and tank geometries influence the development and dominance of certain flow patterns within the vessel. Through combining theoretical models with practical experiments, the paper identifies key operational parameters that affect the transition and balance between axial, radial, and tangential flows. These insights contribute to improving mixing quality, reducing energy consumption, and guiding the scale-up process of industrial stirring systems. The findings are expected to offer valuable references for the design and optimization of stirring equipment used in complex multiphase flow environments.

Keywords: Axial flow, Radial flow, Tangential flow, CFD simulation, Industrial Mixing

1. Introduction

Stirring equipment is a necessary component in fluid processing industries, primarily used to enhance mass and heat transfer through mechanical means. The main objective of this process is to improve the efficiency of mass and heat transfer. During agitation, the liquid undergoes complex flow patterns induced by the impeller, mainly including axial, radial, and tangential flows. The distribution of these flow modes determines the uniformity of mixing, turbulence intensity, and shear forces, which in turn directly affect the overall efficiency of the process.

Previous studies have mainly focused on specific flow types or on optimizing individual impeller designs, with limited comparative analysis across different flow patterns under unified conditions. Much of the early research prioritized radial flow fields and their shear characteristics, particularly in gas-liquid mixing and high-viscosity systems. However, the integrated understanding of how axial, radial, and tangential flow modes interact within a stirred tank remains insufficiently explored.

This study aims to fill this gap by providing a comprehensive analysis of the fluid dynamic characteristics of these three flow patterns. It investigates their formation mechanisms, interactions, and governing parameters through a combination of computational fluid dynamics (CFD) simulations and experimental validation [1].

With the advancement of CFD technology, it is now possible to accurately model velocity fields, pressure distributions, and turbulence characteristics in stirred tanks, offering powerful tools for process engineers and designers [2]. This research not only contributes to a deeper theoretical understanding of stirred flow fields but also provides practical guidance for optimizing equipment structure and operating conditions.

By presenting comparative insights into flow behaviors, this study offers meaningful references for improving industrial mixing efficiency, supporting energy-saving initiatives, and guiding equipment scale-up. It thus holds both academic and practical value for future development in chemical, pharmaceutical, and food processing industries.

2. Characteristics of different flow patterns

2.1. Flow mode of stirring

2.1.1. Axial flow

Axial flow refers to fluid motion along the axis of the impeller and is commonly produced by propeller-shaped impellers, such as marine propellers or pitched-blade turbines. This type of flow creates a top-to-bottom circulation pattern that enhances the overall mixing process. Due to its high flow rate and relatively low shear characteristics, axial flow is especially suitable for large-volume, low-viscosity fluid systems. It is often used in solid-liquid suspensions to keep particles evenly dispersed and prevent sedimentation. For instance, in the fermentation industry, axial flow is favored for maintaining homogeneous distribution of microorganisms in bioreactors. According to Paul et al., axial impellers are ideal when macro-mixing and suspension are primary goals [3]. Their ability to promote vertical circulation reduces dead zones and improves bulk mixing efficiency.

2.1.2. Radial flow

Radial flow occurs when fluid is discharged perpendicularly from the impeller axis, moving outward and then deflecting upward or downward after striking the tank wall. This flow pattern is typically generated by flat-blade disc turbines, such as the Rushton turbine. Radial flow provides high shear forces and is ideal for processes requiring intense local mixing, such as gas-liquid dispersion and liquid-liquid emulsification. A common example is its application in the chemical industry for producing fine emulsions or in wastewater treatment for enhancing oxygen transfer rates. Ranade pointed out that radial impellers are particularly effective at dispersing gas into liquid phases due to their strong shear zones near the impeller blades [2]. However, their circulation pattern is more localized, making them less effective for large-scale bulk mixing unless combined with other flow types.

2.1.3. Tangential flow

Tangential flow is characterized by circular motion of the fluid around the tank's horizontal direction, primarily driven by the inertia of the rotating impeller. This flow often dominates in unbaffled tanks, where it causes vortex formation and the development of recirculation loops near the wall. As a result, mixing becomes inefficient due to poor axial or radial movement. Tangential flow is sometimes used intentionally, such as in crystallization processes where gentle circulation is

DOI: 10.54254/2755-2721/2025.23355

preferred to avoid crystal breakage. However, in most mixing systems, especially those requiring homogeneity, tangential flow must be suppressed. Baffles are commonly installed on the tank wall to interrupt circular flow and redirect energy into axial or radial directions, thereby enhancing turbulence and improving mixing effectiveness. As noted by Nagata, tangential flow without control elements can lead to dead zones and stratification in the fluid body [4].

2.2. Key influencing factors of flow patterns

The formation and distribution of flow patterns in stirred tanks are influenced by a combination of fluid properties, equipment geometry, operating conditions, and modeling approaches. These factors not only determine the efficiency of mixing but also influence whether axial, radial, or tangential flow becomes dominant.

2.2.1. Turbulence models

In computational simulations, turbulence models such as the standard k-ε model, RNG k-ε model, and Reynolds Stress Model (RSM) are commonly used to describe turbulent flow behavior. These models help predict flow characteristics, including velocity distribution, turbulent kinetic energy, and shear stress, which are essential for analyzing the mixing mechanism in stirred tanks [3].

2.2.2. Reynolds number (Re)

The Reynolds number is a dimensionless quantity defined as Re=ρND2μ, where ρ is the fluid density, N is the impeller rotational speed, D is the impeller diameter, and μ is the dynamic viscosity. It is used to characterize the flow regime: low Re values (typically <10) indicate laminar flow, intermediate values indicate transitional flow, and high Re values (>10,000) indicate fully developed turbulent flow [5]. The flow regime significantly affects the flow pattern—turbulent flow tends to promote better mixing, and the dominance of axial or radial flow depending on the impeller design.

2.2.3. Fluid viscosity and density

Viscosity affects the fluid's resistance to deformation and hence influences the development of shear layers and turbulence. High-viscosity fluids are more likely to exhibit laminar or transitional flow patterns and require greater energy input for effective mixing. Density differences, especially in multiphase systems, can lead to stratification or uneven circulation, which modifies the expected flow distribution [5].

2.2.4. Tank geometry

The configuration of the tank, including the tank-to-impeller diameter ratio and the presence or absence of baffles, plays a critical role. Baffles are used to suppress tangential flow and induce turbulence, which helps in transforming inefficient swirl-dominated flow into desirable axial or radial flow patterns. A poorly designed tank without baffles may result in vortex formation and significant dead zones.

2.2.5. Power input from the impeller

The mechanical power delivered by the impeller influences flow intensity, turbulence generation, and mixing time. Higher power input generally leads to stronger flow fields, improved circulation, and enhanced turbulence, but also increases energy consumption. The balance between energy

DOI: 10.54254/2755-2721/2025.23355

efficiency and flow effectiveness must be carefully considered during system design and scale-up [6].

2.3. Application of Computational Fluid Dynamics (CFD) in stirring research

CFD methods are frequently applied to the simulation of the fluid flow inside of a stirred tank, which gives information about the velocity fields, the shear stress distribution, and the characteristics of the turbulence under varying impeller and stirring conditions. The large eddy simulation and the k-degree model are typical turbulence models (LES).

3. Research and data analysis

3.1. Experiment setup and method

The stirring experiments were carried out in a cylindrical tank made of clear acrylic, with a diameter of 0.5 meters and a height of 0.8 meters. To investigate the influence of impeller design on fluid dynamics, three distinct configurations were employed: an axial flow impeller (propeller-type), a radial flow impeller (Rushton turbine), and a tangential flow setup without baffles. Particle Image Velocimetry (PIV) was utilized to capture the velocity distribution within the tank, providing detailed insights into the flow behavior under each configuration. Complementary Computational Fluid Dynamics (CFD) simulations were conducted to further analyze the velocity fields, shear stress distributions, and turbulence energy, offering a comprehensive understanding of the hydrodynamic characteristics associated with each impeller type.

3.2. Result and discussion of experimental data

Table 1: Results of experimental data

Flow Mode	Max Velocity (m/s)	Turbulence Energy (m^2/s^2)	Shear Stress (Pa)
Axial Flow	1.25	0.45	2.1
Radial Flow	1.10	0.62	3.4
Tangential Flow	0.90	0.30	1.8

The experimental data indicate the influence of different flow modes on stirring performance:

The results in the table provide a clear comparison of the three flow patterns in terms of flow velocity, turbulence energy, and shear stress. Axial flow demonstrates the highest maximum velocity (1.25 m/s), indicating its strong capacity to promote bulk fluid circulation and vertical mixing throughout the tank. This feature is particularly useful in solid-liquid systems where uniform suspension is required. In contrast, radial flow exhibits the highest turbulence energy (0.62 m^2/s^3) and shear stress (3.4 Pa), suggesting that it is the most effective for high-shear mixing applications, such as emulsification and gas dispersion. Tangential flow, while having the lowest values in all three parameters, tends to generate circular motion along the tank wall, which can lead to the formation of vortex zones and poor axial mixing if not controlled by baffles. Therefore, each flow pattern exhibits distinct fluid dynamic characteristics, and their suitability depends on the specific requirements of the mixing process.

DOI: 10.54254/2755-2721/2025.23355

4. Case study& optimization suggestion

4.1. Case study: optimization of pharmaceutical stirring process

When the present solution was dominated by the presence of tangential flow, which resulted in the non-uniformity, a pharmaceutical company was in a situation of problems of insufficient mixing in which the current system was dominated by a combination of the two was ineffective, and the mix was not in an uncoordinated state of affairs due to the fact that the existing system was in the state of being dominated by the two. The introduction of a dual-layer impeller design was done in which the axial and radial flow was combined in order to increase the uniformity of the mixture, which resulted in a 15% increase in manufacturing efficiency.

4.2. Proposed suggestions

To improve flow distribution and mixing efficiency in stirred tanks, several strategies can be applied. First, adding baffles helps to reduce tangential flow, which often causes inefficient circular motion, and instead enhances axial and radial flows that are more effective for bulk circulation and shear-driven mixing. Second, selecting an appropriate impeller configuration is crucial. A dual-layer impeller system, such as a combination of a pitched-blade turbine and a Rushton turbine, can effectively balance the advantages of both axial and radial flow, promoting uniform mixing and improved mass transfer [1,3]. Third, adjusting the impeller speed to correspond with an appropriate Reynolds number allows for optimal control of mixing uniformity while managing power consumption. A proper Reynolds number ensures the flow regime remains within the desired range, whether turbulent or transitional, depending on the mixing requirements.

5. Conclusion

To examine the characteristics of axial, radial, and tangential flow in the stirring equipment, this paper uses both CFD simulations and experimental data in addition to this. This study looked at the ways in which these flow modes interacted with one another and how they affected the mixture performance. This paper suggested using an experiment, which included a baffle, choosing the best impeller combinations, and setting the stirring speed in order to make the most of these optimization approaches in both case studies and experimental data. The use of smart control technologies for the additional optimization of stirring performance might be investigated in the future.

References

[1] *Zhang, J., Xu, J., & Xie, W. (2017). "CFD simulation and experimental study on flow pattern and mixing characteristics in stirred tanks with different impellers." Chinese Journal of Chemical Engineering, 25(9), 1237–1245.*

[2] *Ranade, V.V. (2002)." Computational Flow Modeling for Chemical Reactor Engineering." Academic Press.*

[3] *Paul, E.L., Atiemo-Obeng, V.A., Kresta, S.M. (2004)." Handbook of Industrial Mixing: Science and Practice." Wiley.*

[4] *Nagata, S. (1975)." Mixing: Principles and Applications." Wiley.*

[5] *Chhabra, R.P., & Richardson, J.F.(2011). Non-Newtonian Flow and Applied Rheology: Engineering Applications. Butterworth-Heinemann.*

[6] *Armenante, P.M., & Nagamine, E.U. (1998). "Effect of low off-bottom impeller clearance on the minimum agitation speed for complete suspension of solids in agitated tanks." Chemical Engineering Science, 53(9), 1757–1775.*