Understanding Precipitator Performance via CFD Modelling

Yuqing Feng [1], Lachlan Graham[2], Peter Witt[3], Bon Nguyen[4] and Yuanlong Huang[5]
1. Principal Research Scientist
2. Senior Research Engineer
3. Senior Principal Research Scientist
4. Senior Experimental Research Scientist
CSIRO Mineral Resources Research Unit - Processing Program, Clayton, Australia
5. Head of new type agitator division
Shenyang Aluminium and Magnesium Engineering & Research Institute (SAMI), Shenyang, China
Corresponding author: yuqing.Feng@csiro.au
https://doi.org/10.71659/icsoba2025-aa027

Abstract

CSIRO has a long-standing history of working with alumina refineries to investigate the hydrodynamics of precipitators. This includes the effects of draft tubes, multiple impeller systems, and SWIRLFLOW® impeller retrofits. While physical modelling has traditionally been the foundation of this research, the use of computational fluid dynamics (CFD) has always been a significant component of the work to support the optimisation of agitation systems that often goes unmentioned. Key areas of investigation include scale formation, particle suspension, heat transfer, and power consumption.

In this study, laboratory-scale analogue tanks were used to establish CFD calculated wall shear stress as a practical proxy for predicting scale formation. Measured scale thickness and total scale mass were compared with CFD-predicted wall shear stress. The results revealed that low-shear regions are related with heavy scale accumulation. The total scale mass was found to decrease with increasing shear stress, reinforcing the predictive capability of CFD.

The CFD model was extended to compare the hydrodynamic performance of industrial-scale draft tube and SWIRLFLOW® tanks. The analysis showed that the SWIRLFLOW® configuration generates a more uniform wall shear stress distribution and stronger upward flow, effectively reducing the potential for both scale formation and wall erosion. In contrast, the draft tube tank exhibited pronounced low-shear regions along the upper section of the tank wall and the outer surface of the draft tube – features consistent with severe scaling observed in practice and, in some cases, structural failure due to excessive scale buildup.

CFD also provided insights into the effect of agitation strategies on particle breakup in precipitators by comparing turbulent microscales for the draft tube and SWIRLFLOW® tanks.
These results confirm that CFD, when validated against experimental observations, provides a robust framework for understanding scale formation mechanisms and for optimising the design and operation of industrial precipitator tanks. CFD now plays a vital role in complementing physical experimentation, delivering insights into precipitator performance that are otherwise difficult or impractical to obtain through laboratory studies alone.

Keywords: Precipitation tank, Swirl flow, Scale, CFD, Draft tube.

1. Introduction

Scale formation continues to be a persistent and costly issue in stirred slurry tanks, particularly within hydrometallurgical, chemical, and mineral processing operations. It contributes to increased maintenance, reduced equipment longevity, and impaired process efficiency. Over several decades, CSIRO has partnered with the alumina industry to improve the understanding and control of such issues in precipitation tanks [1]. These efforts have explored the effects of agitation systems—including draft tubes, multiple impeller arrangements, and SWIRLFLOW® retrofits—using both physical experiments and computational fluid dynamics (CFD).

To support scale control strategies, CSIRO developed a novel analogue scale formation system using a controlled gypsum precipitation reaction within geometrically scaled 2 litre and 50 litre laboratory tanks. These experiments allowed systematic evaluation of scale morphology and distribution under different agitation conditions, generating data that closely resembled patterns observed in full-scale industrial operations. Based on these insights, wall shear stress was identified as a potential and practical CFD-accessible proxy for predicting scale formation.

Some hydrodynamic parameters in alumina precipitators are difficult or impractical to measure at laboratory or plant scale, making CFD a valuable complementary tool. In this work, CFD models were benchmarked against experimental results and then extended to assess performance across different precipitator configurations. Key analyses undertaken included:

- A correlation between experimentally measured scale thickness and CFD-predicted wall shear stress in lab-scale tanks, validating the use of shear stress as a proxy for scale propensity,
- A comparative CFD study of industrial-scale draft tube and SWIRLFLOW® designs, evaluating their impact on scale mitigation and flow distribution.

CFD was also used to provide evaluation of turbulence microscales to determine whether there was a significant difference between draft tube and SWIRLFLOW® agitation in terms of agglomeration or particle breakage.

2. CFD Model

2.1 Model Description

CFD simulations were performed by solving the Reynolds-averaged Navier–Stokes (RANS) equations, which govern the conservation of mass and momentum for incompressible turbulent flows. During the early stages of model development, several turbulence models were evaluated, including the standard k–ε, Shear Stress Transport (SST), and Reynolds Stress Model (RSM), to determine the most suitable approach for representing the highly swirling and recirculating flows typical of precipitator tanks. The RSM was ultimately selected due to its ability to resolve the transport of individual Reynolds stress components, offering improved accuracy in capturing anisotropic turbulence and swirling motion near impellers. This advanced turbulence modelling approach enables a more detailed representation of flow structure, particularly in regions critical for scale formation and solids suspension.

Various methods are available to model rotating impellers in CFD simulations, including the transient sliding mesh approach and steady-state techniques such as the frozen rotor. The sliding mesh approach captures full transient interactions between rotating and stationary zones, while Frozen rotor methods assume steady-state conditions, modelling the impeller region in a rotating reference frame and apply averaging across the interface between the rotating and stationary tank domains. These steady-state approaches are computationally efficient and well suited for predicting mean flow structures in mixing tanks. In this study, steady-state simulations using the frozen rotor method were initially performed to obtain a converged flow field, which then served as the initial condition for transient simulations using the sliding mesh approach. The transient simulations captured unsteady flow behaviour, and time-averaged results were extracted for

Further extraction of turbulent microscales from the CFD simulations have shown that microscales associated with turbulence are larger than typical hydrate particles over most of the tank volume for both Draft Tube and SWIRLFLOW® agitation. This result suggests that the particle attrition behaviour is likely similar between the two agitation types.

5. References

1. Julie Townsend et al., Reflections on 25 years of SWIRLFLOW® operation at QAL alumina refinery, *Proceedings of 42nd International ICSOBA Conference,* Lyon, France, 27 – 31 October 2024, *TRAVAUX* 53, 481-483. https://doi.org/10.71659/icsoba2024-aa021
2. Jie Wu et al., An analogous scale model for laboratory mixing studies, *Mineral Enginnering*, Vol 199, (2023) 1081111. https://doi.org/10.1016/j.mineng.2023.108111
3 Shen, L. et al., The effect of impeller speed on scale formation within a swirling flow tank, *24th Australasian Fluid Mechanics Conference (AFMC 2024)*, 1-5 December 2024, Canberra. Australia.
4 Jie Wu et al., Increased agitation reliability for slurry suspension in mineral processing, *Minerals Engineering,* Vol. 170, 2021, 107008. https://doi.org/10.1016/j.mineng.2021.107008
5. Rijkeboer, Ab, McFeaters, J. and Manché, D, Advances in process simulation of agglomeration in Bayer precipitation, *Proceedings of 40th International ICSOBA Conference,* Athens, Greece, 10 – 14 October 2022, *TRAVAUX* 51, 397-414.