Determining the Mean Residence Time Distribution in Flocculation Zone of Jet Clarifier by Computational Fluid dynamics (CFD) Analysis

Ploypailin Romphophak[1,2,*]; Arnaud Cockx[2]; Alain Liné[2]; Carole Coufort-Saudejaud[3]; Pisut Painmanakul[1,4,5]

[1] Chulalongkorn University, Department of Environmental Engineering, Faculty of Engineering, 254 Phayathai Road, 10330 Bangkok, Thailand.
[2] LISBP, Université de Toulouse, CNRS, INRA, INSA; LISBP-INSA, 135 Avenue de Rangueil, 31077 Toulouse, France
[3] Laboratoire de Génie Chimique (LGC), Université de Toulouse, CNRS, INPT, UPS, Toulouse, France.
[4] Research Unit on Technology for Oil Spill and Contamination Management, Chulalongkorn University; Bangkok 10330, Thailand.
[5] Research Program on Development of Technology and Management Guideline for Green Community, Center of Excellence on Hazardous Substance Management (HSM), Bangkok 10330, Thailand.

Corresponding Author(s): Ploypailin.R@Chula.ac.th

Among the various existing technologies for water treatment to remove suspended solids, the jet clarifier is considered as an effective and compact system as it couples flocculation and clarification in a single unit. Generally, Camp and Stein *Gt* parameter is recommended to evaluate clarifier efficiency, where *G* stands for a global velocity gradient and *t* a characteristic time scale (contact time) since they are essential parameters to control an agglomeration of flocs. Hence, this work aims to investigate the local mean resident time and velocity gradient of the jet clarifier to better standing how it works. This work occupied computational fluid dynamics (CFD) techniques to model the detailed information prevailing in the reactors within different liquid flow rates that affect the hydrodynamic behavior directly. Indeed, the experiments to investigate fluid dynamics of the reactor are still necessary required in order to decide the suitable CFD model and validate simulations. Thus, the mean residence time distribution (RTD) analysis, which is a common tool for detecting fluid dynamic problems in continuously operating system such as dead zones or short-circuiting flow, was become a general technique to decide to validate some of the model predictions of the pilot plants under various operating conditions (3 liquid flow rates) along with the Laminar, Standard k-ε, and Detached Eddy Simulation (DES) models. Determination of the mean velocity field is performed by RTD. The RTD data are processed to detect the long tail of the RTD curve induced by the local hydrodynamic of the jet clarifier as shown by the recirculation loop present in the flocculation zone that exhibiting by the CFD technique. Characteristic time scales related to macromixing are then extracted. Based on CFD data processing, local and instantaneous shear rate are estimated. The analysis of space averaged velocity gradient *G* is presented. The range of *G* is 2-15 s^{-1} whereas the residence time decreases from 4 - 1 hr. Based on the hydrodynamic analysis; the parameter *Gt* is shown to be constant around 30,000 for different jet flow rates. Efficiency of such jet clarifier can thus be foreseen in term of the flocculation, but the settling time of the pilot also depends on liquid flow rate.