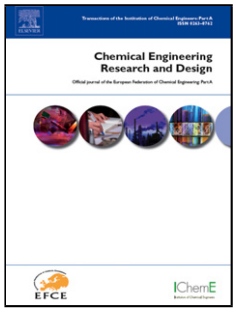

Title: Comparative analysis on flocculation performance in unbaffled square stirred tanks with different height-to-width ratios: Experimental and CFD investigations

Author: Weipeng He Lianpeng Xue Beata Gorczyca Jun Nan Zhou Shi

PII: S0263-8762(18)30065-0
DOI: https://doi.org/doi:10.1016/j.cherd.2018.01.055
Reference: CHERD 3029

To appear in:

Received date: 18-7-2017
Revised date: 12-12-2017
Accepted date: 29-1-2018

Please cite this article as: He, W., Xue, L., Gorczyca, B., Nan, J., Shi, Z.,Comparative analysis on flocculation performance in unbaffled square stirred tanks with different height-to-width ratios: Experimental and CFD investigations, *Chemical Engineering Research and Design* (2018), https://doi.org/10.1016/j.cherd.2018.01.055

Comparative analysis on flocculation performance in unbaffled square stirred tanks with different height-to-width ratios: Experimental and CFD investigations

Weipeng He [a,*], Lianpeng Xue [a], Beata Gorczyca [b], Jun Nan [c], Zhou Shi [a]

[a] College of Civil Engineering, Hunan University, Changsha 410082, China

[b] Department of Civil Engineering, University of Manitoba, Winnipeg R3T 5V6, Canada

[c] School of Municipal and Environmental Engineering, Harbin Institute of Technology, Harbin 150090, China

* Corresponding author. Tel.: +86 731 8882 1491; Fax: +86 731 8882 1441.

Postal address: Department of Water Engineering and Science, College of Civil Engineering, Hunan University, 2 Lushan South Road, Changsha 410082, China (Weipeng He)

E-mail address: heweipengwater@163.com (W. He).

Abstract:

The effect of liquid-level height (*H*) on floc growth during flocculation has been investigated in unbaffled square stirred tanks with a fixed bottom width (*D*). Firstly, flocculation tests were performed by using an in-situ recognition system for floc morphology to evaluate flocculation performance within each tank at three typical shear rates of G_{ave} = 10, 30 and 70 s^{-1}. Then, turbulent flow fields generated under all flocculation-test conditions were predicted by Computational Fluid Dynamics (CFD) simulations, followed by a detailed discussion based on