Computational Fluid Dynamics of System Hydrodynamics and Erosion Behavior in Internally Circulating Fluidized Bed Reactor with Inserting Pipe

D. Thiemsakul, R. Piemjaiswang, P. Piumsomboon, and B. Chalermsinsuwan

Abstract—**Internally circulating fluidized bed reactor (ICFB) is the system with combining the function of reactor, cyclones and loop seal of a conventional circulating fluidized bed reactor (CFB) into a single reactor column. In this type of reactor, the reactor column is separated into two sections (riser and downer) by baffles and is linked together via connecting ports. This system is then considered as compact operation when comparing with the conventional CFB reactor. However, the simplicity of the ICFB reactor is trade-off with a gas leakage which takes place between the two sections through the connecting ports. In addition, the solid particle movement inside the system can cause the erosion on the inserting pipes which are used for heating or cooling this ICFB reactor column. In this study, the system hydrodynamics and erosion behavior inside ICFB reactor with inserting pipe were investigated by computational fluid dynamics (CFD) using two-dimensional Eulerian-Eulerian model. The adjusted Gidaspow drag model was applied to compute the interaction between the gas and solid particle phases. Then, the system hydrodynamics was obtained and the wall shear stress was calculated in the existent of the erosion at the surface region of the inserting pipes. The results from this simulation were used to design the inserting pipe arrangement inside this ICFB reactor.**

Index Terms—**Internally circulating fluidized bed reactor, computational fluid dynamics, hydrodynamics, erosion.**

I. Introduction

The gas and solid particle multiphase flow system is conventionally used in many industrial processes such as chemical, petrochemical, environmental and power production processes [1]. The CFB reactor is commonly used for gas and solid particle multiphase flow system due to its many properties such as reactor efficiency, operational flexibility and overall productivity [2].

The CFB reactor is firstly applied for alumina calciners [3]. The conventional CFB reactor or externally CFB consists of a riser in which a gas-solid particle suspension is transported upward. The mixture is separated by the cyclone. The solid particles are then recycled back to the system via the downer (standpipe) and the other additional specific equipment such as stripper, regenerator, external heat exchanger and etc [4]. Nevertheless, the complex operation between the two reactor columns is difficult to operate.

To simplify the operation, the ICFB reactor is the new reactor system alternative [5]. The ICFB reactor consists of a single reactor column dividing into several vertical sections. Therefore, the solid particles can be circulated in a single reactor column of ICFB by applying this reactor column with unequal fluidizing velocities. Usually, the two reactor columns of the ICFB reactor are also referred as the riser and the downer. This ICFB reactor is proposed to have a much higher mixing potential and heat transfer rate [6]. Zhou et al. [7] investigated the solid particle flow behavior and biomass gasification in a clapboard-type ICFB reactor. The effect of fluidization velocity on solid particle circulation rate and pressure distribution in the ICFB reactor was found. Both the fluidization velocities in the riser and downer sections were the main operational parameters controlling the solid particle circulation rate. Osman *et al.* [8] studied the operating performance of a novel ICFB reactor applying for chemical looping combustion. The investigation showed that the ICFB reactor concept could be a promising candidate for scale-up and commercialization of pressurized chemical looping technologies. The results provided the valuable insights for designing of larger-scale units. Hassan *et al.* [9] investigated the circulation characteristics of binary mixture of solid particles in ICFB reactor. The result showed that the ICFB reactor exhibited a higher solid mixing comparing with a conventional reactor. Osman *et al.* [10] investigated the syngas production using chemical looping reforming system in the ICFB reactor. The methane (CH_4) and water vapor (H_2O) were fed into the fuel reactor of chemical looping reforming system. The main characteristic of chemical looping reforming system was the circulation of an oxygen-carrier (metal-oxide solid particle) between two interconnected reactors namely the air reactor and the fuel reactor. The successful chemical looping reforming system of CH_4 and H_2O in an ICFB reactor was obtained. To apply the ICFB reactor for chemical looping reforming system, the temperature between air reactor and fuel reactor should not be the same. Therefore, the heat sources such as heating pipe should be added to the system for occurring the endothermic chemical reaction. However, for a system with solid particles, having an inserting pipe across the reactor column will cause the erosion. Therefore, the erosion behavior should be investigated to identify the location of inserting pipes that have high risk and to prevent or reduce the occurrence of

Manuscript received January 30, 2020; revised September 13, 2020. This work was supported in part by the Chulalongkorn University Graduate Scholarship to Commemorate the 72nd Anniversary of His Majesty King Bhumibol Adulyadej, the 90th Anniversary of Chulalongkorn University Fund and the Grant for Research (CU-GR_62_34_23_11) from Ratchadaphiseksomphot Endowment Fund of Chulalongkorn University.

D. Thiemsakul, P. Piumsomboon, and B. Chalermsinsuwan are with Department of Chemical Technology, Faculty of Science, Chulalongkorn University, Bangkok 10330, Thailand (e-mail: 6172177023@student.chula.ac.th, pornpote.p@chula.ac.th, benjapon.c@chula.ac.th).

R. Piemjaiswang is with Environmental Research Institute, Chulalongkorn University, Bangkok 10330, Thailand (e-mail: ratchanon.p@chula.ac.th).

doi: 10.18178/ijesd.2020.11.11.1298

erosion inside the system.

Recently, the computational fluid dynamics (CFD) has been considered as a promising method for researchers and engineers. The CFD uses the numerical method to analyze transport phenomena and flow behaviors in the considered system [11]. The governing equations, including continuity equation, momentum equation, and energy equation, are solved instantaneously [12]. For the solid particle phase, the granular temperature is applied to calculate the fluctuating kinetic energy of the solid particles [13]. There are several unique advantages of CFD such as easily uses for new system design, reduces of time and cost of new system design, be able to study system where controlled experiment is difficult to perform and practically gives unlimited level of detail of results.

In this study, the development of a CFD model of an ICFB reactor with inserting pipes was investigated to explore system hydrodynamic and erosion behavior. The wall shear stress was used to represent to the erosion behavior at the surface region of the inserting pipes. The results from this simulation then could be used as a guideline for designing the inserting pipe arrangement inside the ICFB reactor.

II. Experimental

A. CFD Setup

In this study, the CFD simulation was performed based on a two-dimensional cold flow Eulerian-Eulerian model using commercial CFD program, ANSYS Fluent R1. With this model, the governing equations, so called conservation equations, including mass and momentum, for each phase, were solved instantaneously as can be seen in Eqs. (1) and (2).

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

Conservation of momentum:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho \vec{g} + \vec{F} \quad (2)$$

where the nomenclature of each symbol can be found in the ANSYS Fluent 18 user's guide [12].

The modified Gidaspow drag coefficient correlation or drag model was applied to calculate the interaction between gas and solid particle phases [14]. The phase coupled SIMPLE algorithm was used for pressure–velocity coupling, while the spatial discretization scheme was set as second-order upwind. The pressure-based first-order implicit unsteady solver was employed for time-temporal discretization. A time step was set as 0.001 s. The CFD simulations were performed for the total simulation time of 25 s to ensure the quasi steady state operation.

B. ICFB Reactor Base Case

The schematic diagram of the ICFB reactor base case is shown in Fig. 1. The height and width of the freeboard zone were 80 cm and 75 cm, respectively. The height of the circulating zone was 100 cm. The width of the circulating zone was 30 cm which divided into two sections, the riser and downer sections, by the baffle. The widths of the riser and downer were 20 cm and 10 cm, respectively. The top and the bottom of the circulation zone were linked with 2 cm diameters connecting ports. The ICFB reactor had two gas outlets with 1 cm length. The first outlet was located at the top of the freeboard zone. The setup of the outlet position was equivalent to the gas outlet of the riser section in the conventional CFB reactor. The second outlet was at the top of the circulating zone to allow the gas to exit from the downer section.

The employed density and diameter of the solid particles were 2,500 kg/m^3 and 200 µm, respectively, which was used as the bed material. The gas density was fixed at 1.21 kg/m^3 and the gas viscosity was set as 1.79 x10^{-5} kg/m·s. Initially, the solid particles were packed in to riser and downer sections with 18.37 cm height. The gases were fed into the riser and downer sections with superficial gas velocities of 1.37 m/s and 0.3 m/s, respectively.

For the result validation, the pressure profile from the experimental data of Zaabout *et al.* [15] was selected to compare with the obtained CFD simulation data from this study.

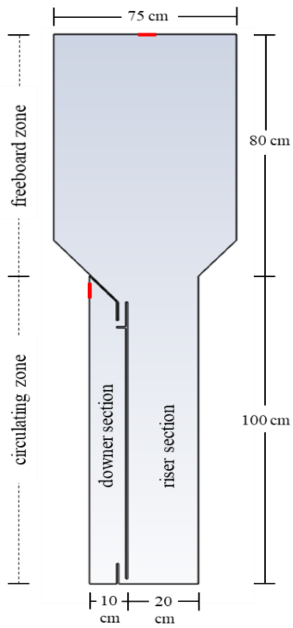


Fig. 1. The schematic diagram of the ICFB reactor used in this study.

C. ICFB Reactor with Inserting Pipe

After the model validation, the ICFB reactor was modified by introducing the inserting pipes of 0.8 cm diameter into a downer section. As stated in the introduction, the inserting pipes will use as the heat source for occurring the endothermic chemical reaction. The 75 inserting pipes were added at the height of 20 cm up to 80 cm from the bottom of the reactor column. In this study, two inserting pipe arrangements, model A and model B, were studied. The difference between these two inserting pipe arrangements is depicted in Fig. 2. For the model A and model B, the inserting pipes were arranged in a square pitch and in a triangular pitch, respectively. The other system operating conditions were fixed as the base case. After the modification by introducing the inserting pipe, the shear stress observation was explored to obtain the erosion behavior. The collisions between solid particles and inserting pipe was the cause of the shear stress on the surface of the inserting pipes. Besides, the system hydrodynamics investigation was obtained. The solid volume fraction and the pressure distribution in the ICFB reactor were plotted and explained the solid particle and gas flow behavior inside the system.

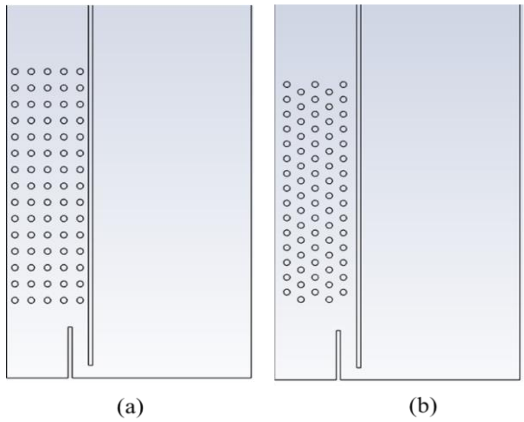


Fig. 2. The inserting pipe arrangements of the ICFB reactor with (a) square pitch pipe and (b) triangular pitch pipe used in this CFD simulation.

III. Results and Discussion

A. Grid Independence and Time Independence

For the general CFD study, the large number of grid or computational cell increases the accuracy of the obtained result, however, the large number of computational cell also consumes the simulation time. Therefore, the grid independence test should be performed to optimize the number of computational cell.

Here, three numbers of computational cell were simulated with 15,000 (coarse), 30,000 (medium) and 45,000 (fine) computational cells. The obtained pressure results from each computational cell are shown in Fig. 3. The calculated results were close to each other for 30,000 and 45,000 computational cells. To save the computational cost and preserve the accuracy of the result, the 30,000 computational cells was used in the rest of the study.

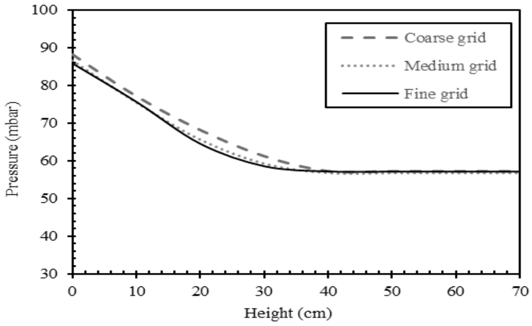


Fig. 3. Pressure profile from simulation with coarse, medium and fine computational cells.

The time independence test gives the simulation time that the system reaches quasi steady state condition. Fig. 4 shows the pressure profile in the downer section at 40 cm. At the beginning simulation time, the pressure fluctuated significantly because of the start-up operating condition. After 15 s simulation time, the pressure slightly oscillated around a specific value. Therefore, in this study, the average results between 15-25 s were collected for the comparison and analysis.

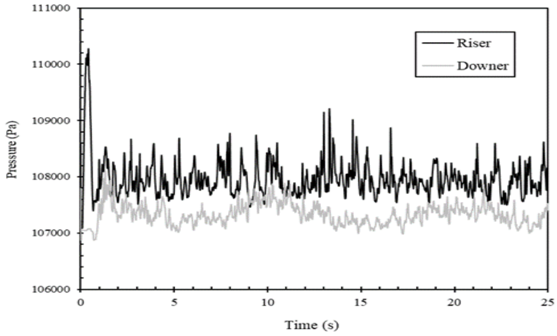


Fig. 4. Pressure variation with simulation time in riser and downer sections at the height of 40 cm.

B. Model Validation

The validation between the experiment and simulation results was conducted to confirm that the developed CFD simulation model could represent the ICFB reactor.

As stated in the experimental, the developed CFD model was validated against system hydrodynamics of the ICFB reactor of Zaabout *et al.* [15]. The base case operating condition was operated with 1.37 m/s of the fluidization velocity (U_r) in the riser section and 0.3 m/ of the fluidization velocity (U_d) in the downer section.

Fig. 5 compares the pressure profiles between the obtained simulation results and the experimental results of Zaabout *et al.* [15]. The pressure profiles along the reactor height were consistent with each other after increasing of 1.7 times the conventional Gidaspow drag coefficient model. The modified Gidaspow drag coefficient was adjusted in two dimensional model to obtain higher accuracy as the result of three dimensional model [14]. Accordingly, the developed CFD model was reliable for the ICFB reactor hydrodynamics investigation.

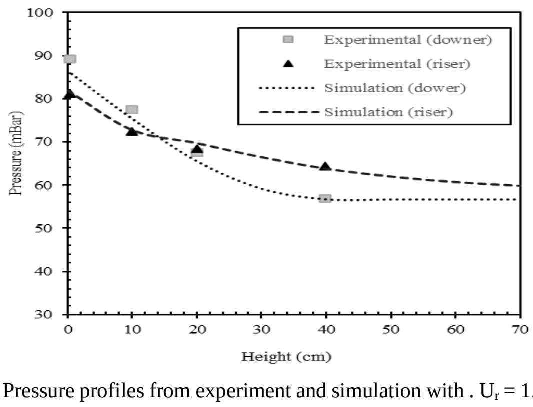
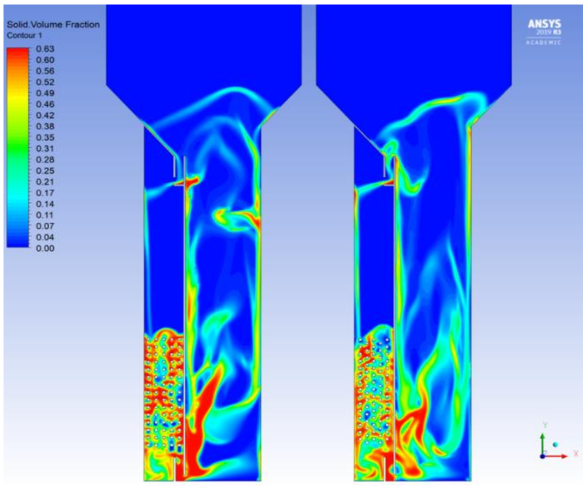


Fig. 5. Pressure profiles from experiment and simulation with . U_r = 1.37 m/s and U_d = 0.3 m/s.

C. System Hydrodynamics and Erosion Behavior of ICFB with Inserting Pipe

The introduction of the inserting pipes was a cause of the increasing of gas velocity in the downer section. It was because the cross-sectional area of the reactor column was decreased. With this condition, more than 50% of solid particles in the system was transferred out of the reactor column. Therefore, the 0.2 m/s fluidization velocity at the downer section was selected for the investigation.

Fig. 6(a) and Fig. 6(b) illustrates the solid volume fraction contours for the inserting pipe arrangements of model A and model B, respectively. The solid particles in the downer section was expanded to approximately 90 cm height above the bottom of reactor column with average solid volume fraction of 0.31 and 0.25 for the inserting pipe arrangements of model A and model B, respectively. The hydrodynamics in the downer section was similar to the bubbling fluidization regime. In this regime, the solid particles were expanded, which helped the solid particles to exchange the heat with the inserting pipes better than the fixed bed operation. For the hydrodynamics in the riser section, the solid particles were behaved in fast fluidization regime. In this regime, the solid particles were rapidly carried upward inside the system column.



(a) (b)

Fig. 6. Solid volume fraction contours in ICFB reactor with (a) square pitch pipe and (b) triangular pitch pipe.

Here, the possible erosion behavior was explained through the wall shear stress. Fig. 7 and Fig. 8 show the contour plots of wall shear stress in downer section with square pitch pipe and triangular pitch pipe, respectively. At the ‘a1’ position, in Fig. 7 and Fig. 8, the wall shear stresses had high values on the top of inserting pipes. This is because the recirculation of solid particles from the riser section is mainly impacted at this location. When comparing between the inserting pipe’s row, row ‘1’ had high wall shear stress than the other rows. This is because this row is the top inserting pipe row.

Fig. 9 shows the wall shear stress acting on inserting pipes. For both the inserting pipe arrangement, near the reactor column wall, e.g. ‘a’ and ‘e’ positions, had higher wall shear stress than the column in the middle. According to the system hydrodynamics in downer section, the gas bubbles occur in the middle of reactor column causing the solid particles to move to the reactor column wall and to erode the inserting pipes in that location. Also, the triangular pitch pipe had less overall erosion than square pitch pipe due to the available space for solid particle.

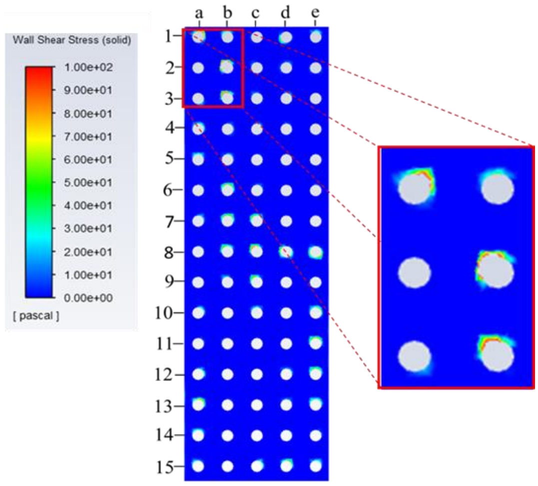


Fig. 7. Contour plot of wall shear stress in downer section with square pitch pipe.

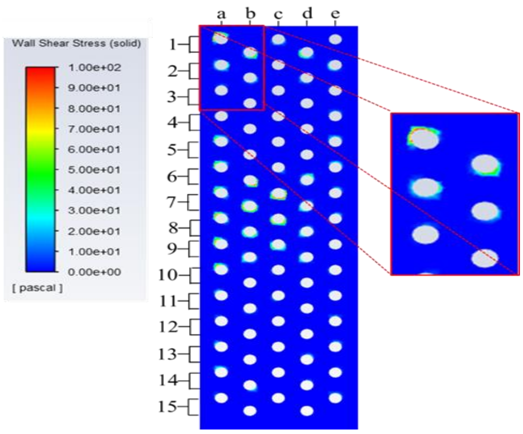


Fig. 8. Contour plot of wall shear stress in downer section with triangular pitch pipe.

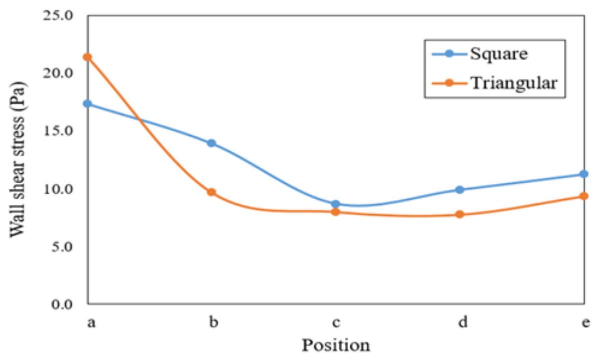


Fig. 9. Time-averaged wall shear stress at various reactor column positions for both square and triangular pitch pipes.

IV. Conclusion

In this study, the system hydrodynamics and erosion behavior in the ICFB reactor with inserting pipes were investigated by the CFD simulation. The riser and downer sections were operated in the fast and bubbling fluidization regimes, respectively. The erosion possibility was shown in term of the wall shear stress at each inserting pipe. According to the system hydrodynamics, the positions that had high risk to erode were the top and near reactor wall regions for both the pipe arrangement. The triangular pitch pipe had less overall erosion than square pitch pipe due to the available space for solid particle movement. In future, the other inserting pipe arrangement and reactor modification to reduce the erosion should be considered to reduce the system erosion and system maintenance.

CONFLICT OF INTEREST

The authors declare no conflict of interest.

AUTHOR CONTRIBUTIONS

D. Thiemsakul conducted the research. All authors had analyzed the data, wrote the manuscript and approved the final version.

ACKNOWLEDGMENT

This work was supported in part by the Chulalongkorn University Graduate Scholarship to Commemorate the 72nd Anniversary of His Majesty King Bhumibol Adulyadej, the 90th Anniversary of Chulalongkorn University Fund and the Grant for Research (CU-GR_62_34_23_11) from Ratchadaphiseksomphot Endowment Fund of Chulalongkorn University.

REFERENCES

[1] L. Reh, "New and efficient high-temperature processes with circulating fluid bed reactors," *Chem. Eng. Technol.*, vol. 18, no. 2, pp. 75-89, 1995.

[2] J. R. Grace, C. J. Lim, C. M. H. Brereton, and J. Chaouki, "Circulating fluidized bed reactor design and operation," *Sadhana*, vol. 10, no. 1-2, pp. 35-48, 1987.

[3] L. Reh, *Circulating Fluid Bed Reactor — AKey To Efficient Gas/Solid Processing*, Pergamon Press Canada Ltd, 1986.

[4] M. Van de Velden, J. Baeyens, B. Dougan, and A. McMurdo, "Investigation of operational parameters for an industrial CFB combustor of coal, biomass and sludge," *China Particuology*, vol. 5, no. 4, pp. 247-254, 2007.

[5] H. S. Ahn, W. J. Lee, S. D. Kim, and B. H. Song, *Solid Circulation and Gas Bypassing in an Internally Circulating Fluidized Bed with an Orifice-Type Draft Tube*, 1999.

[6] F. Zhang, L. Zhu, and D. Rao, "The evaluation of a methane autothermal chemical looping reforming experiment based on exergy analysis," *RSC Adv.*, vol. 9, no. 38, pp. 22032-22044, 2019.

[7] Z. Q. Zhou, L. L. Ma, X. L. Yin, C. Z. Wu, L. C. Huang, and C. Wang, "Study on biomass circulation and gasification performance in a clapboard-type internal circulating fluidized bed gasifier," *Biotechnol. Adv.*, 2009.

[8] M. Osman, A. Zaabout, S. Cloete, and S. Amini, "Mapping the operating performance of a novel internally circulating fluidized bed reactor applied to chemical looping combustion," *Fuel Process. Technol.*, vol. 197, p. 106183, Jan. 2020.

[9] M. Hassan, M. P. Schwarz, F. Yuqing, M. Rafique, P. Witt, and L. Huilin, "Numerical investigation of solid circulation flux in an internally circulating fluidized bed with different gas distributor designs," *Powder Technol.*, vol. 301, pp. 1103-1111, 2016.

[10] M. Osman, A. Zaabout, S. Cloete, and S. Amini, "Internally circulating fluidized-bed reactor for syngas production using chemical looping reforming," *Chemical Engineering Journal*, Elsevier B.V., 2018.

[11] W. Zhong, A. Yu, G. Zhou, J. Xie, and H. Zhang, "CFD simulation of dense particulate reaction system: Approaches, recent advances and applications," *Chem. Eng. Sci.*, vol. 140, pp. 16-43, 2016.

[12] ANSYS Fluent Tutorial Guide 18, "ANSYS fluent tutorial guide 18," *ANSYS Fluent Tutor. Guid. 18*, vol. 15317, no. April, pp. 724-746, 2018.

[13] Y. Feng, T. Swenser-Smith, P. J. Witt, C. Doblin, S. Lim, and M. P. Schwarz, "CFD modeling of gas-solid flow in an internally circulating fluidized bed," *Powder Technol.*, vol. 219, pp. 78-85, Mar. 2012.

[14] E. Esmaili and N. Mahinpey, "Advances in engineering software adjustment of drag coefficient correlations in three dimensional CFD simulation of gas – Solid bubbling fluidized bed," *Adv. Eng. Softw.*, vol. 42, no. 6, pp. 375-386, 2011.

[15] A. Zaabout, S. Cloete, and S. Amini, "Innovative internally circulating reactor concept for chemical looping-based CO_2 capture processes: Hydrodynamic investigation," *Chem. Eng. Technol.*, vol. 39, no. 8, pp. 1413-1424, 2016.

D. Thiemsakul is a student of Department of Chemical Technology at Faculty of Science, Chulalongkorn University, Thailand. He finished his B.Eng. in chemical engineering from Thammasat University, Thailand. His main research interest is computational fluid dynamics (CFD) simulation in circulating fluidized bed and internally circulating fluidized bed reactor systems.

R. Piemjaiswang is a researcher at Environmental Research Institute, Chulalongkorn University, Thailand. He received his M.S. and Ph.D degrees in chemical technology, Faculty of Science, Chulalongkorn University. In addition, he finished his bachelor degree of engineering in petrochemical and polymeric material from Silpakorn University, Thailand. His main research topics are focusing on the environmental problems in applied with computational fluid dynamics (CFD). Besides, the human health related to the change of the environment is also his concerned.

P. Piumsomboon is a professor of Department of Chemical Technology at Faculty of Science, Chulalongkorn University. He hold a B.Sc in chemical engineering from Chulalongkorn University, M.E. in chemical engineering and industrial engineering from Lamar University in USA and Ph.D degree in chemical engineering from the University of New Brunswick in Canada. His research interest relates to proton exchange membrane fuel cell and circulating fluidized bed technology.

B. Chalermsinsuwan is an associate professor of Department of Chemical Technology at Faculty of Science, Chulalongkorn University. He hold a B.Sc. in chemical engineering from Chulalongkorn University and Ph.D degree in chemical technology from Chulalongkorn University. His research interest relates to the several topics including: computational fluid dynamics (CFD) simulation, experimental design and analysis, multiphase flow/fluidization technology and carbon dioxide (CO_2) capture and utilization.