EXPERIMENTAL STUDY ON A NEW FCC SPENT CATALYST DISTRIBUTOR

Yongmin Zhang, State Key Laboratory of Heavy Oil Processing, China University of Petroleum, China
zym0876@gmail.com
Songyuan Yu, Chao Zha, State Key Laboratory of Heavy Oil Processing, China University of Petroleum, China

Abstract: In the regenerator of an industrial fluid catalytic cracker (FCC), uniform distribution of its solids reactant, i.e. spent catalyst, plays a crucial role in obtaining better regenerator performance. In traditional FCC unit designs, there was usually no spent catalyst distributor or some intuitive designs with simple structures, i.e. boat or pipe distributors in most China's FCC units (1). In this study, we built a large cold experimental installation to evaluate the performances of various spent catalyst distributors. Distribution uniformity and solids flow resistance were the main target indices for distributor performance evaluation. The experimental results indicate that the boat distributor has the poorest performance, as solids flows preferentially through the few front openings. At high gas flowrates, the pipe distributor can obtain a relative uniform solids distribution, but its flow resistance is also higher. Good flowability of solids that is difficult to maintain throughout the distributor was found to be the root cause of their bad distribution performance. Referring to the idea of an air-slide solids transportation system (2-4), a new slot spent catalyst distributor was proposed. Its performance was systematically evaluated in a large cold model unit. It was found that the new slot distributor has a critical superficial gas velocity, beyond which good solids distribution uniformity and high solids transportation capacity can be both maintained. Compared with traditional boat and pipe spent catalyst distributors, the new slot distributor is much more advantageous comprehensively, e.g. in solids distribution uniformity, solids transportation capacity and operating flexibility.

Key Words: spent catalyst distributor; FCC; experimental; distribution

References:

1. J. Chen. Catalytic Cracking Process and Engineering (2nd ED). China Petrochemical Press, Beijing, China, 2005.

2. M. Ishida, H. Hatano, T. Shirai. The flow of solid particles in an aerated inclined channel. Powder Technol., 1980, 27(1): 7-12.

3. S. K. Gupta, V. K. Agarwal, S. N. Singh, V. Seshadri, D. Mills. An experimental investigation on a fluidized motion conveying system. Powder Technol., 2006, 167(2): 74-84.

4. S. K. Gupta, V. K. Agarwal, D. A. Mills. A model for fluidized motion conveyor transporting fly ash. Exp. Therm. Fluid Sci.. 2010, 34(8): 1042-1048.

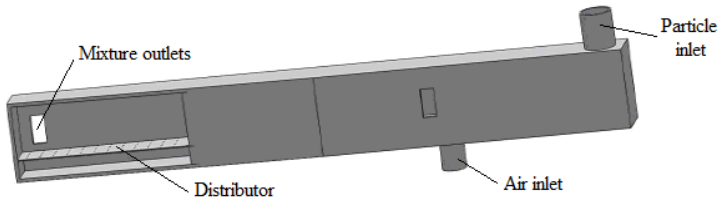


Fig. 1 Schematic of the mechanism of the tested model of the new slot catalyst distributor

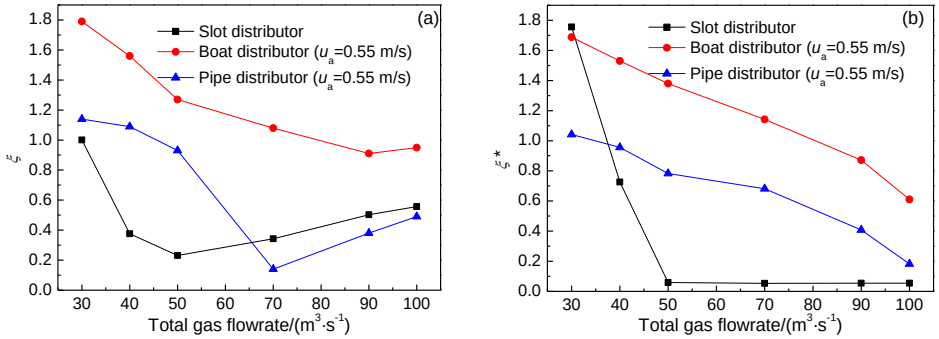


Fig. 2 Comparison of heterogeneity indices of the three spent catalyst distributors

Poster Number 1

A NEURAL-BASED MODELLING APPROACH TO ESTIMATE RESIDENCE TIME AND SOLIDS FLOW RATE IN A CONICAL SPOUTED BED WITH AND WITHOUT DRAFT TUBES

Idoia Estiati, Universidad del País Vasco, Bilbao, Spain
Aitor Atxutegi, Universidad del País Vasco, Bilbao, Spain
Haritz Altzibar, Universidad del País Vasco, Bilbao, Spain
Fábio Bentes Freire, Universidade Federal de São Carlos, São Carlos, Brazil
bentes@ufscar.br
Roberto Aguado Zarraga, Universidad del País Vasco, Bilbao, Spain
Martin Olazar Aurrekoetxea, Universidad del País Vasco, Bilbao, Spain

Key Words: Spouted bed, neural network, draft tube, fluidodynamics

Spouted beds are an alternative contact method to the fixed and fluidized beds especially for handling coarse particles. Since its discovery in 1954 by Gishler and Mathur during grain drying in a fluidized bed, spouted beds have been widely studied and applied in different industrial processes, such as, particle coating, catalytic reactors, pyrolysis and drying of solids, pastes and suspensions. One of the main advantages of spouted beds over fluidized beds is its high degree of mixing between gas and solids within the bed due to a cyclical and uniform movement of particles, leading to high heat and mass transfer rates and also a homogeneous final product. Despite its advantages, the conventional configuration of the spouted bed still has some limitations; such as a high pressure drop for stable spouting, the existence of a maximum height for a stable spout with the corresponding scaling-up limitations and the highly dependable gas flow rates on stable spouting. In order to minimize these operational limitations, several spouted bed configurations have been proposed, the main ones being the spouted bed with a draft tube, the two-dimensional spouted bed and the spouted bed with supplementary aeration, among others. Solids residence time and flow rates are key parameters in the modeling, optimization and control of any process. Although predictions of solids residence time and flow rate are important issues that reflect the accuracy of the whole process simulation, difficulties arise in the use of either purely mechanistic or empirical approaches. In developing a reasonable theoretically based model, the only task is to estimate the values of the coefficients in the model, but, taking into account all relevant phenomena involved this model can prove to be impractical. Additionally, the seemingly overwhelming choice of structural options for empirical models may considerably hinder the modeling step. Artificial neural networks may be an effective alternative in that they can represent highly nonlinear processes, with flexibility and robustness against input noise and once they have been developed and their coefficients determined, they can provide a rapid response for a new input [1]. This neural network based model (MatLab® R2013a, Mathworks) uses the particle diameter dp, bed density ρb, cone angle γ, inlet diameter D0 and draft tubes characteristics as inputs whilst the maximum, mean and minimum residence times along with the solid flow rates are predicted with reasonable agreement with the experimental data, as shown in Figure 1.

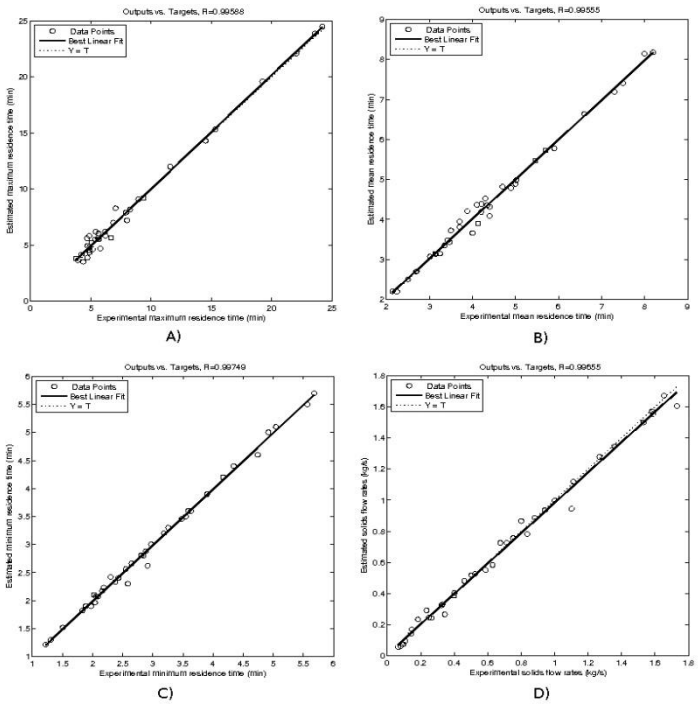


Figure 1: Calculated vs Experimental values for; A) maximum residence time, B) mean residence time, C) minimum residence time , D) solids flow-rate.

[1] D. M. Himmelblau, “Accounts of experiences in the application of artificial neural networks in chemical engineering,” *Ind. Eng. Chem. Res.*, vol. 47, no. 16, pp. 5782–5796, 2008.

CFD MODELLING OF ELECTROSTATIC CHARGE GENERATION IN GAS-SOLID FLUIDIZED BED-A PRELIMINARY WORK

Fahad Chowdhury[a], Andrew Sowinski[a], Alberto Passalacqua[b*] and Poupak Mehrani[a]
[a]Department of Chemical and Biological Engineering, University of Ottawa, 161 Louis Pasteur Street, Ottawa, ON, Canada, K1N 6N5
[b]Department of Mechanical Engineering, Iowa State University, 2025 Black Engineering Building Ames, Iowa, USA, 50011-2161
*T: 1-515-294-5047 E: albertop@iastate.edu

Gas-solid fluidized beds have been developed for a large variety of industrial applications, which include polymerization, combustion, drying, etc. The solid particles in this flow system tend to generate electrostatic charges due to particle-particle and particle-wall interactions. Particularly in the case of polymerization fluidized beds, the electrostatic charge generation results in particles collecting on the reactor walls. This accumulation of particles might instigate wall fouling (known as “sheeting”) and consequently force a reactor shutdown for clean-up. Although the fluid bed electrification has been experimentally investigated, its computational fluid dynamic (CFD) modeling has received limited attention. Previously, in a work conducted by Rokkam et al. (1) an Euler-Euler multi-fluid and electrostatic model was used to simulate laboratory-scale experiments on electrostatics. In that work, the CFD model used experimentally measured particles charge-to-mass ratio (q/m) as an input for the simulation. In the present work, the electrostatic model is modified to simulate charge generation due to particle interactions. Single particle contact experiments are conducted to obtain charge generation values and used as an input to an Euler-Lagrange model accounting for electrostatics. The goal is to obtain simulated values of electrostatic charge of particles which are comparable to measurements from laboratory-scaled fluidized bed experiments.

REFERENCES

1. R. G. Rokkam, A. Sowinski, R.O. Fox, P. Mehrani, and M.E. Muhle, Computational and experimental study of electrostatics in gas-solid polymerization fluidized beds. Chemical Engineering Science, 92: 146–156, 2013.

HPC LARGE SCALE SIMULATION OF AN INDUSTRIAL FLUIDIZED BED AND APPLICATIONS TO CHEMICAL ENGINEERING PROCESSES WITH NEPTUNE_CFD

Hervé Neau, Université de Toulouse; INPT, UPS; IMFT; CNRS, France
herve.neau@imft.fr,
Pascal Fede[a,b], Cyril Baudry[c] and Olivier Simonin[a,b]
[a] Université de Toulouse; INPT, UPS; IMFT; F-31400 Toulouse, France
[b] CNRS ; Institut de Mécanique des Fluides de Toulouse;
Allée du Professeur Camille Soula, F-31400 Toulouse, France
[c] EDF Recherche & Développement; DMFEE; F-78401, Chatou, France

Key Words: HPC, parallel computation, large scale, industrial fluidized bed, rotating parts

Fluidized beds and more broadly dilute and dense particle-laden reactive flows are encountered in a wide range of industrial chemical engineering applications such as catalytic polymerization, coal combustion … Nowadays, it is possible to perform realistic 3D simulations of industrial configurations using an unsteady Eulerian multi-fluid approach for polydisperse reactive flows with a good physical modelling. Hence CFD is a powerful tool for studying the optimization of chemical processes, new designs and scaling-up.

To obtain numerical results in an acceptable CPU time, it is important to check the feasibility of CFD simulation of fluidized bed flows in complex geometries at industrial scale. Also we need to estimate HPC capacities of CFD tools. Numerical simulations have been performed with the solver NEPTUNE_CFD: parallelized unstructured code (MPI) using unsteady Eulerian multi-fluid approach. NEPTUNE_CFD is based on the same numerical methods than Code_Saturne. Code_Saturne is an open source CFD software code ready to run on petascale machines. NEPTUNE_CFD's high parallel computing performances for particle-laden flows have been demonstrated over last years. Recent developments allow overtaking NEPTUNE_CFD's limitations making it fit for massive parallel computing.

In the present study, the massively parallel computing performances of NEPTUNE_CFD are investigated. We show the results of a numerical simulation of a 3D dense fluidized bed reactor at industrial scale including more than 80 tons of particles. The geometry is about 5 meters in diameter and 30 meters in height. The mesh has more than one hundred million hexahedral cells leading to cells of $1cm^3$. The parallel numerical simulation has been performed using up to 12 000 cores on a massive parallel supercomputer. This kind of very large simulation is able to capture the different scales of such a complex multiphase flow: from the large flow structures to the small ones (particle clusters, bubbling …). It is important to capture these small structures as they are essential features of reactive flows. The results of this study constitute a reference for the development of sub-grid models to be used on coarser meshes.

We also focus on reactors featuring rotating parts. They are common in chemical engineering processes. It is essential to take these rotating parts into account, even with complex industrial geometries, as they enhance chemical reactions. NEPTUNE_CFD's rotating mesh numerical methodology is now effective. The method is based on splitting the domain into static and rotating parts and information is passed thanks to a non-conformal mesh matching technique. Methodology and numerical results are presented.

Acknowledgments:
This work received funding from the European Community through the 7th Framework program for the projects SUCCESS (Grant agreement No 608571) and NANOSIM (Grant agreement No 604656). This work was granted access to the HPC resources of CalMip supercomputing center under allocation 2015-0111.

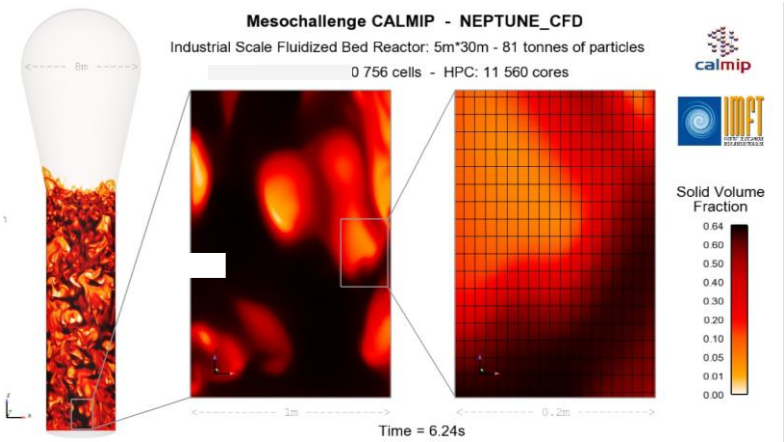


Figure 1 - NEPTUNE_CFD simulation of an industrial 3D fluidized bed reactor

THE EFFECT OF BUBBLE SIZE ON THE PERFORMANCE OF EBULLATED BED HYDROPROCESSORS

Connor Gregory Farrell, Centre for Catalysis Research and Innovation, Department of Chemical and Biological
cfarr069@uottawa.ca
Chris D. Lane , Adam Donaldson, Department of Process Engineering & Applied Science, Dalhousie University, Halifax, Canada
Arturo Macchia, Engineering, University of Ottawa, 161 Louis Pasteur, Ottawa, ON, Canada, K1N 6N5

Keywords: Three-phase fluidization, Bubble size, Reactor modelling, Resid hydroprocessing

A recent cold-flow study has revealed that modifying gas distributor design in three-phased fluidized beds can have a significant effect on overall phase hold-ups and regime transition velocities, even at equivalent phase velocities. It is conjectured that this can be attributed to changes in the bubble-size distribution within the reactor. This study aims to develop a complete kinetic-hydrodynamic model of a resid hydroprocessing ebullated bed reactor with internal recycle in order to study the effects of bubble size distribution on performance metrics of industrial significance. The model consists of combined catalytic and ther-mal reaction models, phase separation efficiency correlations obtained through CFD modelling, catalyst fouling and deactivation models, boiling-point based Vapour-Liquid Equilibria (VLE) relations, and specialized phase hold-up corre-lations developed for resid hydroprocessing applications.

A preliminary hydrodynamic model comparing monodisperse bubble sizes between 0.5 mm and 4 mm has been performed. At the lowest bubble size, gas entrainment (and hence gas hold-up) were maximal, while ideal phase separation was achieved at the largest bubble size. Bed gas hold-ups ranged from 10 % to 40 %. Increased gas entrainment was also associated with a decrease in internal liquid recycle ratio required for design ebullation height. The effect of bubble size was most pronounced near a diameter of 1 mm.

Ongoing analysis into the effect of bubble recycle on bed gas phase com-position in a reactive system is being performed. The effect of bubble size on liquid residence time and hence conversion will be studied with the objective of determining the optimal bubble size for maximizing key conversion parameters.

NUMERICAL AND EXPERIMENTAL STUDY OF A MULTIPHASE COLD BUBBLING BED REACTOR

Tricomi, Leonardo, Industrial Research Chair on Cellulosic Ethanol and Biocommodities , Univesité de Sherbrooke, Sherbrooke, Québec, Canada
Leonardo.Tricomi@Usherbrooke.ca
Melchiori, Tommaso, Chiaramonti, David & Lavoie Jean-Michel , Industrial Research Chair on Cellulosic Ethanol and Biocommodities , Univesité de Sherbrooke, Sherbrooke, Québec, Canada

A lab scale bubbling fluidized bed reactor was simulated using a CFD model, aiming at investigating and ultimately matching its overall fluid dynamic behaviour and mostly the bubbles “footprints” on the pressure drop signal.

In order to validate the model, a cylindrical transparent reactor (PVC) has been assembled. This system, operating at ambient condition, was filled with aluminum oxide (Geldart group B particles) and homogeneously fluidized using compressed air flowing through a porous plate. A high frequency differential pressure gauge was used to measure the pressure drop across the bed. In addition, videos were recorded using a commercial camera and the generated dynamic slow motions frames combined with the still frames analysis contributed to the investigation of this system. Using Fluent-Ansys software, a 2D-Cartesian Two Fluid Model (TFM) was implemented, numerically verified and validated against experimental data using a Power Spectral Density (PSD) analysis of the pressure drop signal. Empirical data also suggested a minimum data capture time for pressure drop records and results showed that a 40 seconds sampling time was could be considered as a minimum threshold to capture a representative PSD distribution of the pressure oscillations. Based upon this finding, several simulations were performed to investigate the model sensitivity to the variation of some parameters specific to granular flows.

Specifically, the 2D TFM sensitivity was tested by varying the formulation for the solid pressure terms, the values of the experimental minimum fluidization velocity (used as a calibration point for the parametric drag law used in this study), and restitution coefficients for particles collisions. Related results have been plotted against each other along with the experimental ones and afterwards analyzed.

Comparison with the experimental data of the 2D-TFM model has proved to be satisfactory in matching the time-averaged pressure drop, the pseudo stationary bed expansion, its bubbles shape and ultimately capturing the total pressure drop “power”, evaluated as the integral of its Power Spectral Density (PSD).

The 2D model showed limitations in reproducing correctly the experimental PSD in the 0-10 Hz range, where the effect of bubbles are more significant. To overcome this limit, a 3-D version of this TFM model was implemented and tested. The analysis of the pressure drop spectrum showed how the 3D TFM model could overcome the aforementioned limitation of the corresponding 2D version. However, the quite coarse mesh size used to reduce the computational time did not allowed capturing the total power of the pressure drop oscillation. Our current efforts address the implementation and validation of a 3D-Dense Discrete Particle Model (DDPM) linked to this lab scale system such as to evaluate the performance and accuracy of this new approach with regards to the TFM previously tested. Such an approach would also contribute to reduce computational costs, this introducing the parcels concept. Collisions between these parcels is accounted throughout the soft-sphere (DEM) model where a linear spring-dashpot model is used. The effect of parcels number as well as DEM parameters variation on the pressure drop has also been investigated and analyzed.

THE UNDERSTANDING OF SILICON SEQUENTIAL ELUTRIATION BEHAVIOUR

Nicolas ESTIME, Bluestar Silicones France SAS, 55 avenue des Frères Perret, 69192 Saint-Fons Cedex, France
nicolas.estime@bluestarsilicones.com
Eloi KEWES , Franck DAHLEM, Sandrine BEC, Jean-Luc LOUBET, Laboratoire de Tribologie et Dynamique des Systèmes, UMR CNRS 5513, 36 avenue Guy de Collongue, 69134 Ecully Cedex, France
Jacques GROLLEMUND, Bluestar Silicones France SAS, 55 avenue des Frères Perret, 69192 Saint-Fons Cedex, France

During the fluidization of broad PSD (Particle Size Distribution) powders, elutriation can not be avoided, but has to be process controlled. Batch elutriations of continuous PSD powders were studied in a laboratory scale fluidized bed. The reference sample was metallurgical-grade silicon powder, with non-spherical shape.

The smallest elutriable fines, namely superfines (<10 µm) are entrained first. However, the largest elutriable particles (Ut ~ Ug) do not begin to be entrained simultaneously, but only after a delay that is as long as the time required for the superfines to leave the bed, thus inducing sequential elutriation (Figures 1). When no superfines were present, the entrainment was not delayed. This peculiar phenomenon was observed at all of the tested gas velocities (0.05-0.2 m/s). The superfines thus seem to strongly limit the elutriation of the larger elutriable particles. This sequential behaviour is particularly interesting to separate particles according to a small and narrow PSD (Figure 2).

These phenomena are related to interparticle interactions within the bed and/or the freeboard and confirm the importance of polydispersity in the elutriation behavior. Thanks to the elutriation mathematical models developed in this study, the behavior that was thought to be explained by Silicon attrition can now be explained by sequential elutriation.

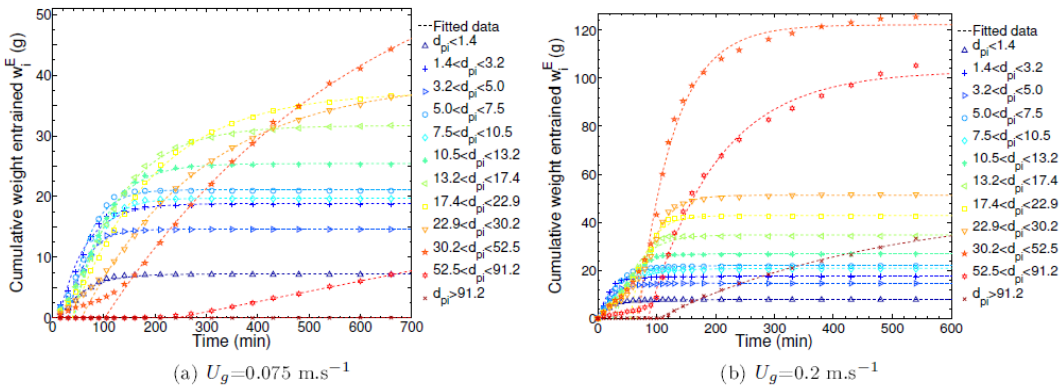


Figure 1 : Cumulative entrained weight per size interval for all elutriating fractions at 0,075 m/s (a) and 0,2 m/s (b)

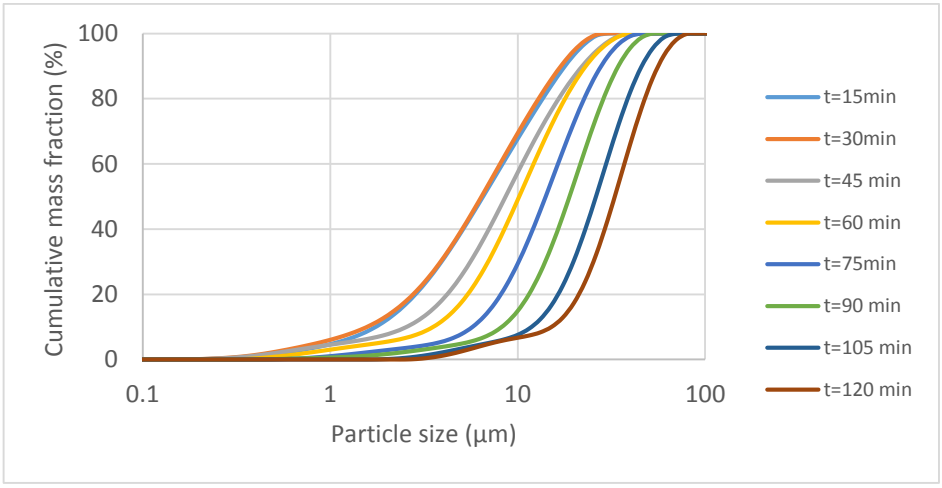


Figure 2 : Evolution of cumulative PSD (Particle Size Distribution) of elutriated particles collected in function of time

THE THREE-DIMENSIONAL SIMULATION OF VORTEX ON THE BOTTOM OF A PARTICLE IN GAS-LIQUID TWO-PHASE FLOW

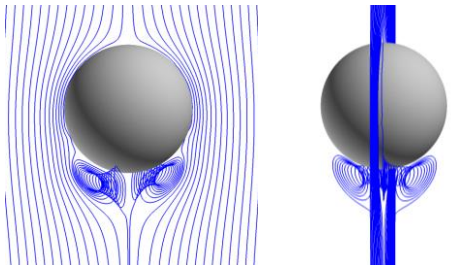
Guangxiang He, Bo Chen, Xiaoyan Guo, Suohe Yang, Haibo Jin
Department of Chemical Engineering
Beijing Institute of Petrochemical Technology
Beijing 102617, China

ABSTRACT

The sophisticated industrial applications demand deep knowledge of local flow hydrodynamics on a particle surface in fixed bed reactors in order to improve process efficiency, particularly micro- (or particle-) scale of gas-liquid two-phase flow, which the study of vortex on the bottom of a particle is much more meaningful. The three-dimensional numerical simulations of gas-liquid two phase flow passing a spherical particle were investigated using Computational Fluid Dynamics (CFD) methodology with the volume of fluid (VOF) method. The effects of gas velocity, liquid velocity, liquid-solid contact angle, surface tension coefficient, and liquid viscosity on the interface status of fluid flow were presented in this paper. The simulation conditions were particle diameters of 10mm, water and air as liquid and gas respectively, droplet size 4mm, and an atmospheric pressure.

Results show that the droplet movement and gas streamlines are greatly dependent on the air flow conditions and the liquid physical conditions. To better observe the flow field around the liquid droplet, the 3D plot is drawn for the contact angle of 50° with the gas velocity of 1.0m/s at the time instant of 0.015s as shown in Figure 1. Vortices are produced on the bottom of a particle known as Von Kármán vortex street, which have the opposite direction of rotation and the double row arrangement vortex. In addition, vortices are found to be more obvious and farer from the particle at higher gas and/or liquid velocities, and the droplet moves faster with the increasing gas velocity (0.2m/s-2.0m/s) and liquid velocity (0.24m/s-0.465m/s). Moreover, the generated vortices can be clearly observed at downstream of particle, and the droplet shape varies with the flow time. Vortex radius changes from 0.5mm to 2.5mm with the time going at the gas velocity 0.5m/s and liquid velocity 0.24m/s. The liquid-solid contact angle between 40° and 80° mainly affects particle surface wettability, which results in the different droplet shape and flow gas streamlines. At the beginning, the shape of the droplet is spherical and thereafter it changes to appropriate shape according to the surface properties and the contact angle. When the liquid-solid contact angle is 50°, vortex is comparatively density which radius is only 0.5mm initially, and then changes to 3.5mm approximately with an increase of the flow time.

Meanwhile, the surface tension can affect the contractile properties and the gas streamlines. With increasing the surface tension coefficient, vortices produced at the same time are density initially, and then change to rarefaction tardily. And the radius of vortex firstly increases to 3mm, then decreases until the droplet leaving the particle surface at the surface tension coefficient of 35dyn/cm. Viscosity is generally one important parameter of the fluid properties. However, it is also shown that as compared to the above factors, the influence of liquid viscosity is negligible in this simulation.



(a) (b)

Figure 1. Streamlines around the particle with gas velocity of 0.5m/s and liquid velocity of 0.24m/s at

REFERENCES

[1] Al-Dahhan M H, Larachi F, Dudukovic M P, et al. High-pressure trickle-bed reactors: A Review. Industrial and Engineering Chemistry Research, 1997, 36: 3292-3314.

[2] Bittagopal Mondal, Kui Jiao, Xianguo Li, Three-dimensional simulation of water droplet movement in PEM fuel cell flow channels with hydrophilic surfaces, Int. J. Energy Res. 2011;35:1200-1212.

[3] May J. Cheah, Ioannis G. Kevrekidis, Jay B. Benziger, Water Slug Formation and Motion in Gas Flow Channels: The Effects of Geometry, Surface Wettability, And Gravity, Langmuir 2013, 29, 9918-9934.

[4] Wei Du, Dongsheng Feng, Jian Xu, Weisheng Wei, Computational Fluid Dynamics Modeling of Gas-Liquid Two-Phase Flow around a Spherical Particle. Chemical Engineering Technology, 2013, 36 (5), 840-850.

PREPARATION OF SUPPORTED SKELETAL NI CATALYST AND ITS CATALYTIC HYDROGENATION PERFORMANCE OF C_9 FRACTION FROM COKING PROCESS

Daming Sun,Ranran Ren,Xin Xu,Guohua Luo*,Kaili Gao,Xue Yang,Jing Ma
Department of Chemical Engineering
Beijing Institute of Petrochemical Technology
Beijing 102617,China

Currently, the inferior compressive strength of traditional Raney-Ni catalyst restricts its application in fixed-bed reactor. To approach this problem a series of supported skeletal Ni catalysts were prepared by mixing pseudo boehmite and Ni-Al alloy powder. In the process,the calcination temperature and atmosphere, mass ratio of pseudo boehmite to Ni-Al alloy powder and the sodium hydroxide solution concentration were investigated. The catalysts characterized by intelligent granule intensity tester(IGIT), scanning electron microscopy(SEM), X-ray photoelectron spectroscopy(XPS), X-ray diffraction (XRD),low temperature nitrogen adsorption, temperature programmed reduction of hydrogen (H_2-TPR), and thermogravimetric-differential thermal analysis (TG-DTA).The results were shown that the calcination atmosphere had a considerable impact on the compressive strength of the catalyst. Compared with air atmosphere, the compressive strength of the catalyst increased from 12.62 N/mm to 23.96N/mm, obviously, in argon atmosphere, which was almost twice as much as the former.The inherent reason for this was that the argon obviously inhibited the transform of $NiAl_3$ to Ni_2Al_3 in which the latter was the key factor to improve compressive strength. Additionally, coke-oven C_9 hydrogenation was used to evaluate the performance of the catalyst and the results indicated that the conversion of indene, the key component of coke-oven C_9, was as high as 90% in 1000h under the optimum reaction conditions:T=220^{o}C, P(H_2)=2.5MPa, H_2/oil=200(v/v), LHSV=3.0h^{-1}. Our data demonstrated that the supported skeletal Ni catalyst have a good industrial prospect in the fixed-bed reactor in future.

*Corresponding author.*Tel./fax*:+86 10 81292074
E-mail address:luoguohua@bipt.edu.cn(G.Luo).

EFFECT OF PRESSURE AND GAS VELOCITY ON RESIDENCE TIME OF PARTICLES SUSCEPTIBLE TO ENTRAINMENT IN GAS-SOLID FLUIDIZED BEDS

Poupak Mehrani, Department of Chemical and Biological Engineering, University of Ottawa, 161 Louis Pasteur Street, Ottawa, ON, Canada, K1N 6N5
Marc-Andre Seguin, Arturo Macchi, Department of Chemical and Biological Engineering, University of Ottawa, 161 Louis Pasteur Street, Ottawa, ON, Canada, K1N 6N5
Scott Champagne, Robin W. Hughes, Natural Resources Canada, CanmetENERGY, 1 Haanel Drive, Ottawa, ON, Canada, K1A 1M1
Mark A. Fitzsimmons, Gas Technology Institute, 5945 Canoga Ave., Woodland Hills, CA, USA, 91367

In relation to pressurized fluidization processes such as oxyfuel coal combustion, understanding the influence of pressure on bed hydrodynamics and in turn their effect on parameters including feed residence time and entrainment rate is essential. The main focus of the work presented here was to evaluate the impact of pressure and gas velocity on particle elutriation rates and residence times. Experiments were conducted under cold flow conditions in a pilot-scale pressurized fluidized bed with an inner diameter of 0.15 m. The bed material was relatively large glass beads (0.8 to 1.2 mm in diameter) while the feed material was simulated with smaller glass beads (37 to 106 micron in diameter), susceptible to entrainment. Operating pressures and fluidization velocities tested were between atmospheric and 1200 kPa(a) and 0.4 and 1.1 m/s, respectively. Preliminary experiments carried out in batch mode resulted in particle elutriation rates increasing with fluidization velocity in a power law relationship. To simulate coal combustors, experiments were then conducted in a continuous mode where the finer material was continuously fed to the fluidized bed of large particles over a desired period of time, without recycling of fines. This work thus presents particle entrainment results for both batch and continuous operations.