International Journal of Engineering and Modern Technology ISSN 2504-8856 Vol. 3 No. 2 2017
www.iiardpub.org

The KTGF-Transport Modelling and the Flow Phenomena on 35^0 inclined Circulating Fluidised Bed (CFB) Riser Reactor using ANSYS CFD Modelling

M. N. Idris[1, 2, 3], T. Mahmud[2] & B. M. Gibbs[3]
[1]Clean Energy and Process Refining Technology,
Department of Chemical Engineering,
Faculty of Engineering,
University of Maiduguri,
Maiduguri, Nigeria
[2]Institute of Particle Science and Engineering,
IPSE, SPEME, University of Leeds,
United Kingdom
[3]Energy Resources Research Institute,
ERRI, SPEME, University of Leeds,
United Kingdom

ABSTRACT

The mixing pattern of fluids in the entrance and exit of the CFB riser reactor is a major factor that determine the quality, efficiency and meeting the requirements and standard specifications of clients, especially in the oil and gas industrial applications.

Commercial experience has shown that, in addition to feed atomization and feed distribution, the feed injection angle also plays a significant role in mixing with catalyst and hence the temperature profile in the riser reactor. Most of modern FCC units have the feed nozzles installed through riser shroud at a fixed angle, a new feed nozzle design have been presented by (Chen et al., 2006). This enables an FCC unit to optimize mixing of feed and catalyst by adjusting the injection angle to achieve the best performance of the unit.

In this paper, CFD was applied to validate the use of aerated solid flow in 35^0 inclined injection system which leads to a better entrainment and improve radial mixing, this differ in the case of non-aerated solid inlet region that is absolutely not good for an operation of FCC riser reactor systems.

Keywords: *ANSYS CFD, atomisation, 35^0 inclined FCC and Fluidisation pattern*

1.0 INTRODUCTION

Circulating fluidized beds (CFBs) have been studied intensively during the past two decades in order to continuously enhancing the performance of industrial processes such as CFB combustion and fluid-catalytic cracking (FCC) processes. The CFB is an advantageous alternative for combustion of solid fuels, because the fuel flexibility is high and the good possibility to control the combustion temperature. Due to the significance of CFB in industry and their complex fluid dynamics, more researches on CFBs and FCC riser reactors remain a subject matter in the literatures. The performance of a CFB boiler is influenced by the mixing of gas and particles. A high mixing rate contributes to an effective distribution of reactants, whereas an insufficient mixing can lead to hydrocarbon and CO-emissions. Therefore, an adequate understanding of the mixing behaviour is important to ensure a high combustion efficiency and emission control. Knowledge on the mixing characteristics is also useful for validation of computer simulations of CFB risers Mustafa (2005). It is therefore obvious that the study of solid concentration and particle velocity in a CFB riser system remains of great interest in the technology of FCC unit improvement (Idris and Burn, This is because FCC units have been considered the most successful applications of CFBs all over the world in recent decades. Records show that about 46% of the global gasoline productions come either directly from FCC units or indirectly from combination with downstream units, such as alkylation Idris and Burn (2008). The feed injection system is by far the most critical breakthrough of modern FCC reactor design. Four recent developments have made the feed injection system increasingly important as presented by (Idris et al., 2016).

1.1 Problem Statement

Optimal mixing of constituted feed into a riser reactor systems to achieving the desirable products that suite the clients' couple with addressing the environmental implications, remain a subject of discuss. There are numerous threats due to incomplete combustion which in turn cause ozone depletions when the flowing fluids are not having a qualitative mixing regime.

1.2 Significance of the Study

An excellent performance of a CFB-FCC riser and regeneration systems are influenced by the quality of mixing of the constituted feed into the injection units. A good mixing rate contributes to an effective distribution of reactants, achieving complete combustion of hydrocarbon and CO-emissions.

2.0 BACKGROUND ON FLUIDISATION REGIMES

At least six different fluidization regimes for gas-solid fluidized beds: fixed bed (FB), bubbling fluidization (BF), slugging fluidization (SF), turbulent fluidization (TF), fast fluidization (FF), and pneumatic conveying (PC). The fluidisation regimes are detailed in Figure 1 below.

A schematic profiles of the existent fluidization regimes in as gas-solid fluidized bed. In the FB regime the air flowing across the particle does not have enough velocity to move the particles. As the superficial gas velocity (U_g) increases, the system reaches the bubbling fluidization regime. In this regime, bubbles start to form and coalesce causing solid mixing; the velocity at which bubbles appeared is known as the minimum bubbling velocity (U_{mb}). From Figure 1, there is a slugging regime which appears in beds where the bed height (H) over the bed diameter (D) is larger than 2. Figure 1 represent the distribution of solids in the various contacting regimes (Levenspiel, 2007).

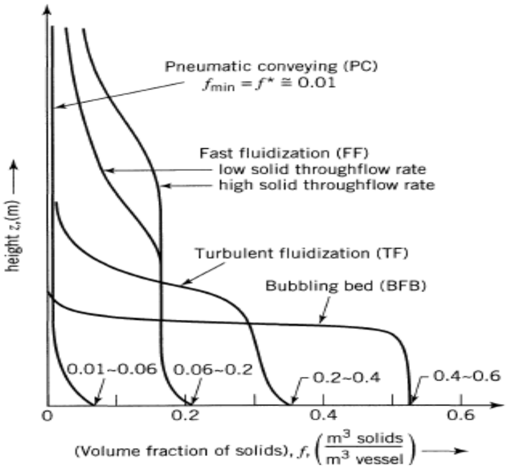


Figure 1 Distribution of solids in the various contacting regimes (Levenspiel, 2007).

3.0 SIMULATION METHODOLOGY

3.1 Simulation Data

The operating conditions of operations of the CFB-FCC riser were presented on Table 1 as shown below.

Table 1 Summary of operating conditions with corresponding numerical values

Operating conditions	(Huang et al., 2007)				(Van engelandt et al., 2007)		
	Case 1.1	**Case 1.2**	**Case 1.3**	**Case 1.4**	**Case 2.1**	**Case 2.2**	**Case 2.3**
G_s (kg/m^{2}s)	100	50	100	200	0.5	3	4.5
U_g (m/S)	3.5	5.5	5.5	8.1	5.3	6.4	7.4

Case 1.1 – 1.3 have previously been discussed at (Idris and Burn 2008).

3.2 Simulation Considerations

In this paper, Case 1.4 is been discussed here as a result of heavier solid loading and the dispersion regime that are captured in the CFD simulation.

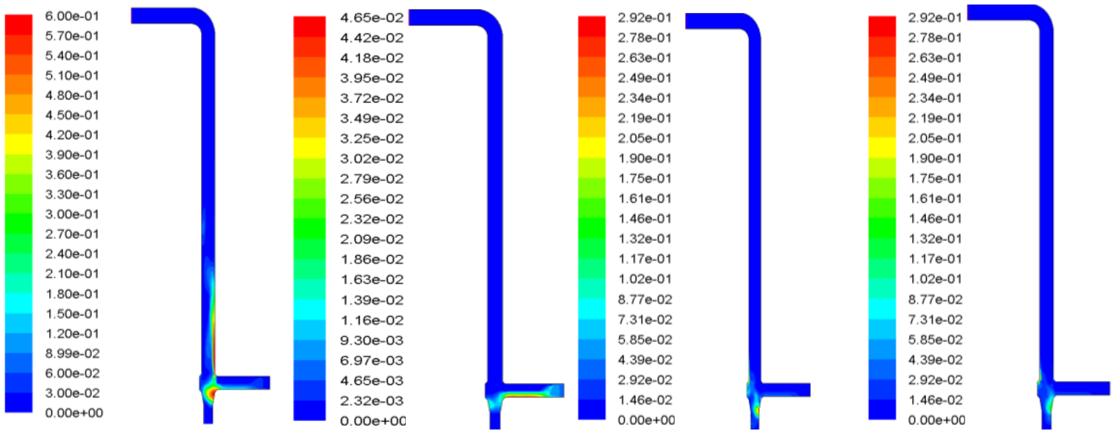
4.0 RESULTS AND DISCUSSION

4.1 Transient Simulation (KTGF-Transport Model) of Case 1.4 (*Ug* = 8.1 m/s at *Gs* = 200 kg/m^{2}s)

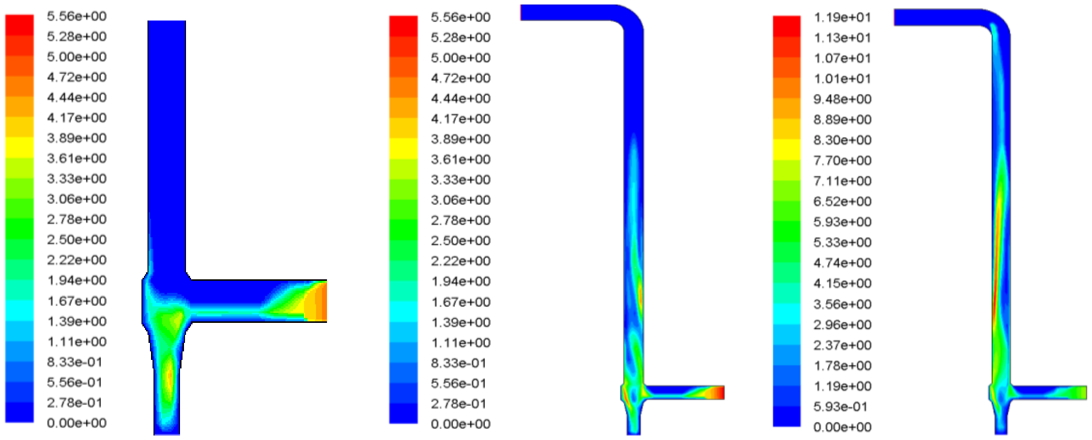
In this case, it was observed that the convergence criteria for the fully develop flow was achieved at time-step equals 8-seconds in Figure 2(d). In comparing cases 1.1, 1.2 and 1.4, it is observed that, in the lower region of the riser, the solids concentration is higher than in the upper section of the riser at all radial positions and is lower in the centre than in the wall region of the riser at all axial locations. Generally, in the FCC riser bottom, the solids concentration increases significantly towards the wall. Towards the exit or riser top, the solids concentration in the wall region decreases. As the solid circulation rate increases, that is, overall solids flux, G_s, and/or decreasing the superficial gas velocity, U_g, leads to increases in the solids holdup in the riser wall region throughout the riser. The studies of (Yan and Zhu 2004) show similar phenomena under the same operating conditions in the riser of height 10 m.

Conversely, flow development is mostly represented by the reduction of solids concentration towards the riser top at r/R from about 0.5 – 1.0. This is similar to most previous studies (Yan et al., 2004). In an FCC riser of this kind, the volume fraction at all radial positions increase with increasing solids flux, while they decrease with increasing gas velocity. Previous studies have shown that increase in the superficial gas velocity accelerates the flow development, but is seen to be more slower as the solids concentration rate increased from *Gs* = 50 kg/m^{2}s to 100 kg/m^{2}s and then to *Gs* = 200 kg/m^{2}s. Similar trends were observed as shown in studies (Yan et al., 2004).

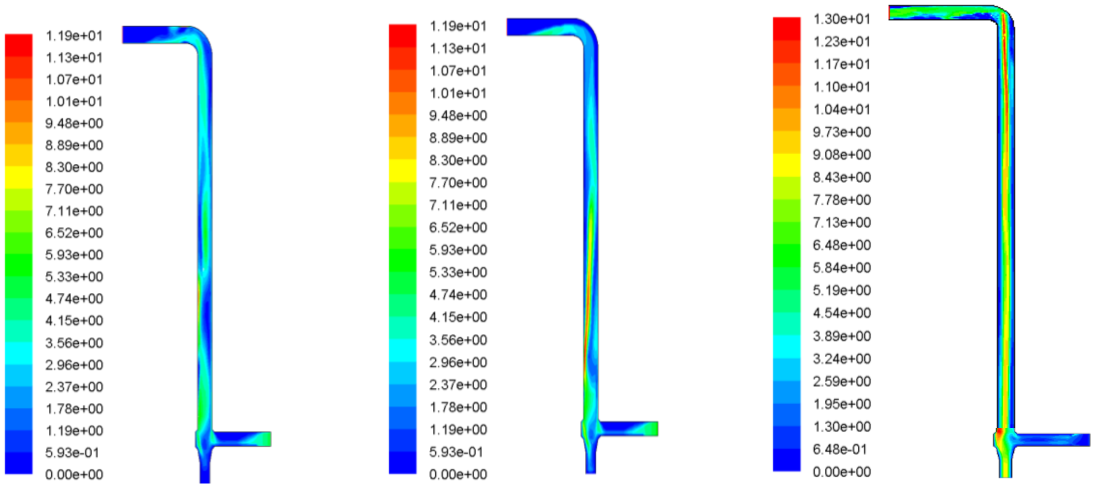
Figure 2 shows some close similarity in the 2-approaches used for the fluent solver. Although in plot 1 (z = 0.95 m) and plot 2 (z = 2.59 m), over-predictions occur in both cases, and the inlet region, there is under-prediction in plot 3 (z = 4.51 m). There were generally under-predictions in plot 4 – 6, that is, at z = 8.16 m – 14.08 m along the riser length. The reason may be due to the effect of high solid concentration, which was not in the case as in cases 1.1 and 1.2 at (Idris and Burn 2008). In this section only the fluent solver results was reported.



(a) FCC V F at t = 1 sec (b) FCC V F at t = 2sec (c) FCC V F t = 6 sec (d) FCC V F t = 8 sec



(e) At t = 1 sec (f) At t = 2 sec (g) At t = 4 sec



(h) At t = 5 sec (i) At t = 6 sec (j) At t = 8 sec

Figure 2: Contour of FCC particles distribution showing various modelling time-steps along the riser reactor. The grid size 578,240 cells were used in fluent, case 1.4 (Huang et al., 2007)

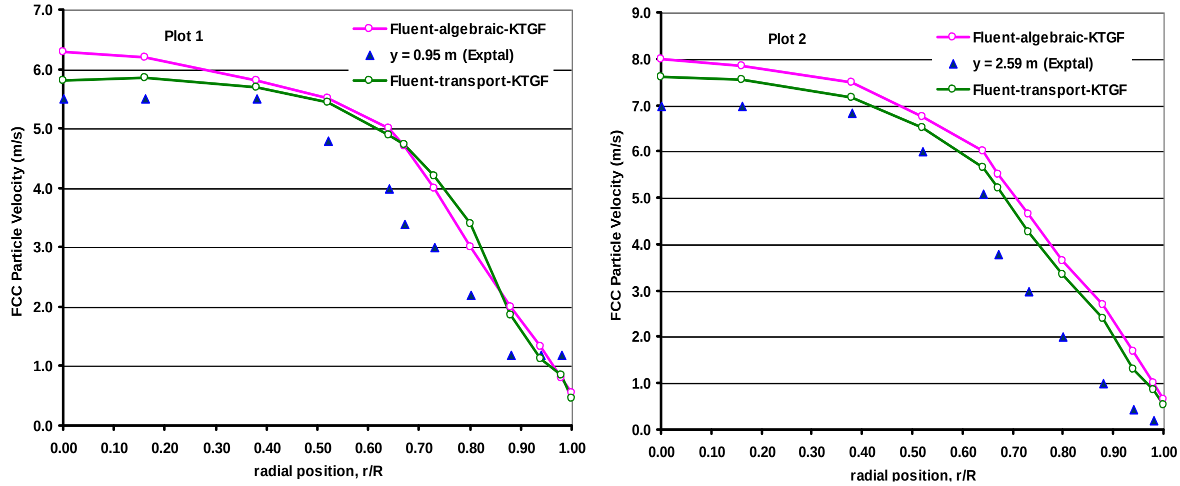


Fig 3(a): Profile of particle velocity at y = 0.95m   Fig 3(b): Profile particle velocity at y = 2.59m

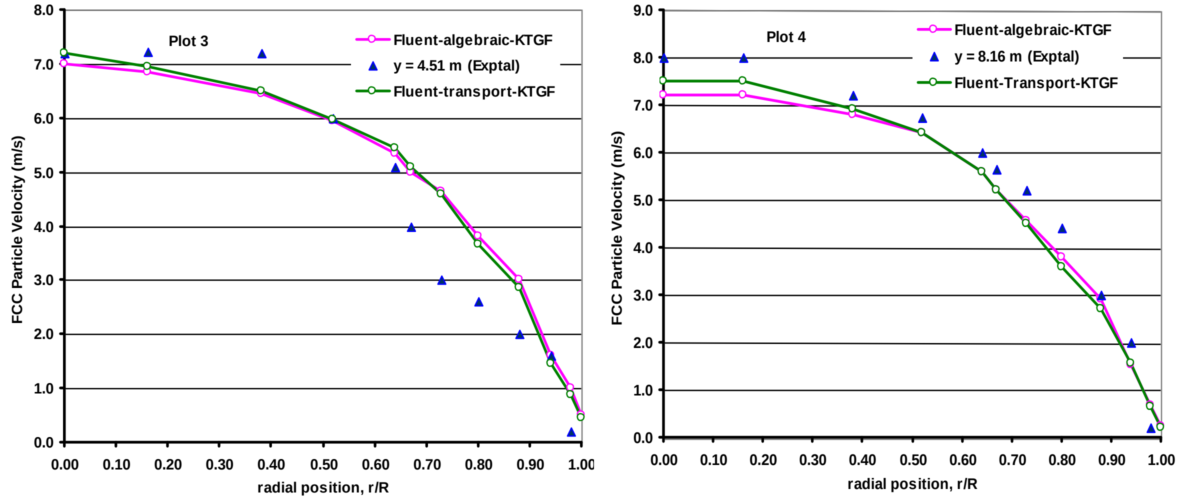


Fig 3(c): Profile of particle velocity at y = 4.51m   Fig 3(d): Profile particle velocity at y = 8.16m

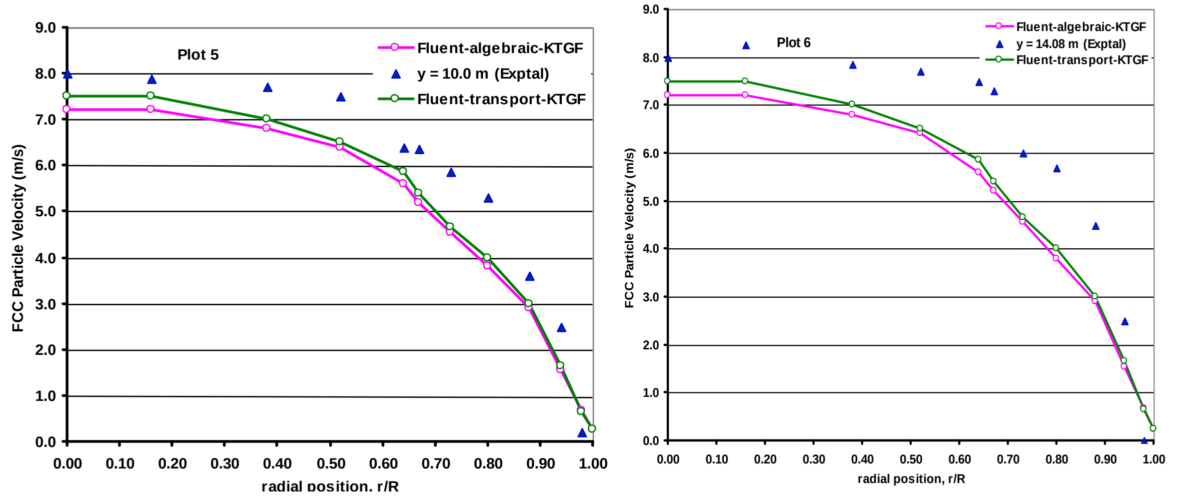


Fig 3(e): Profile of particle velocity at y = 10.0m   Fig 3(f): Profile particle velocity at y = 14.08m

Figure 3 (a - f): Radial profiles of FCC particle velocity along the FCC riser reactor length (z = 0.95 – 14.08 m), that is Plot 1 - 6. The grid size 578,240 cells were used in fluent, case 1.2 (Huang et al., 2007)

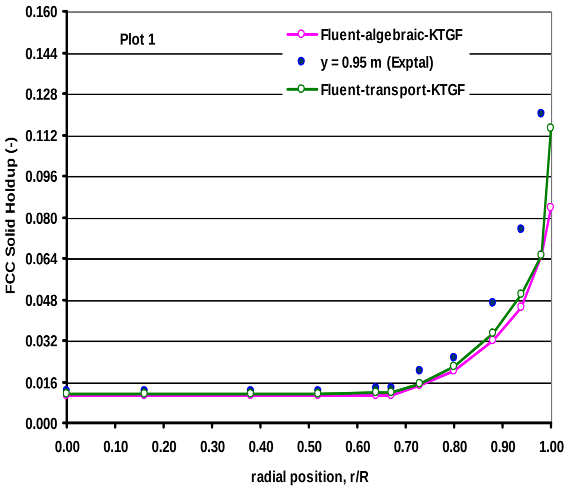


Fig 4 (a): Profile of solid holdup at y = 0.95m

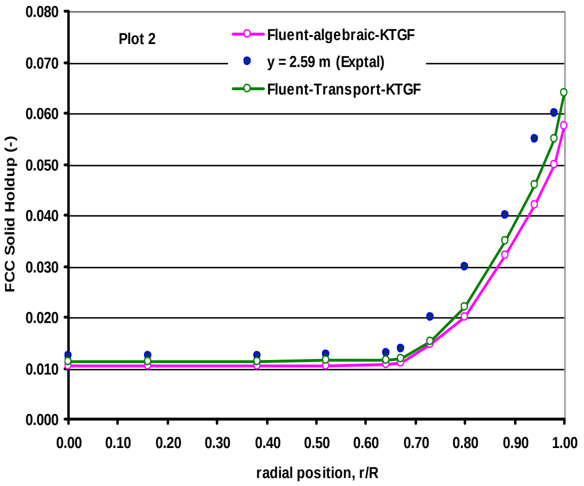


Fig 4 (b): Profile solid holdup at y = 2.59m

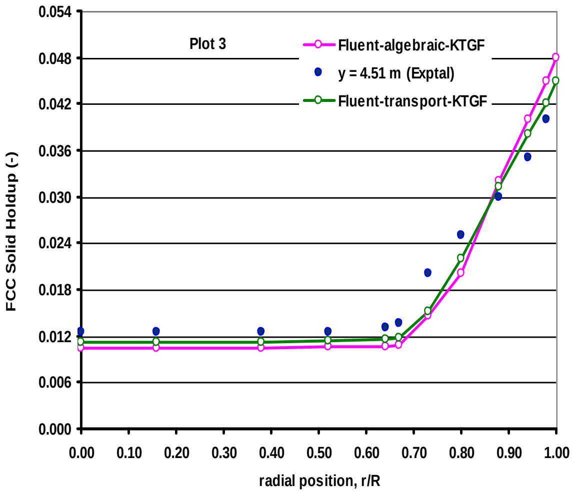


Fig 4 (c): Profile of solid holdup at y = 4.51m

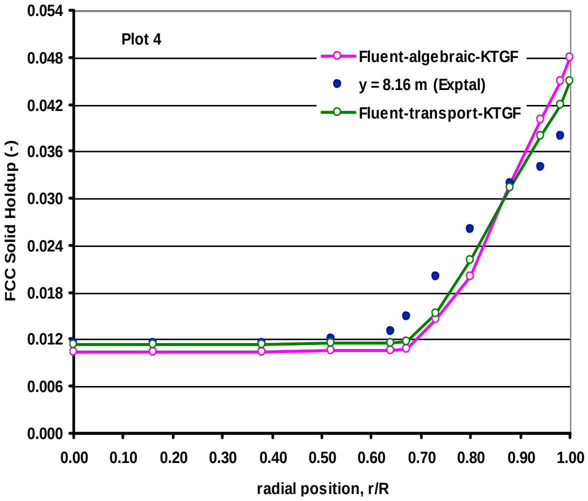


Fig 4 (b): Profile solid holdup at y = 8.16m

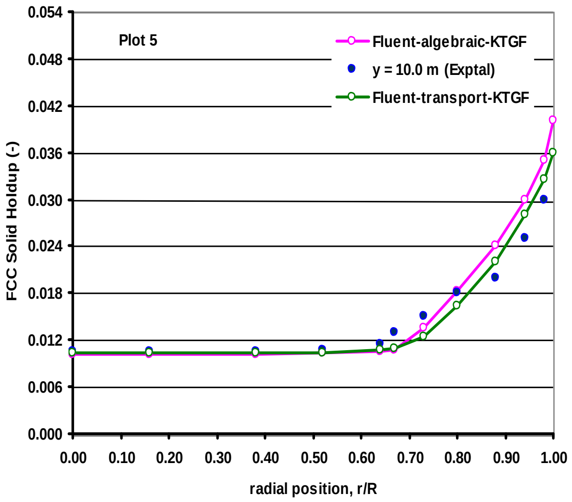


Fig 4 (e): Profile of solid holdup at y = 10.0m

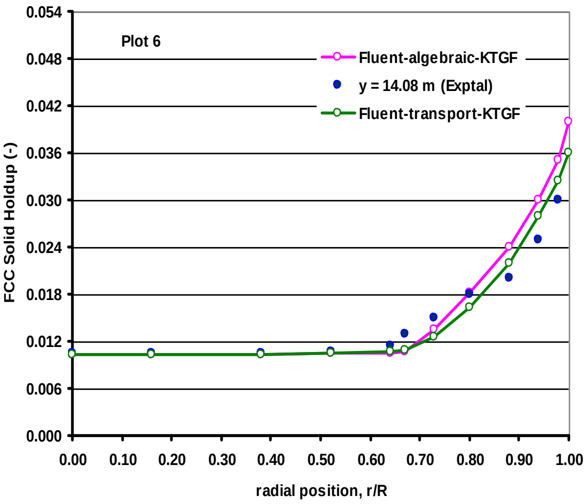


Fig 4 (f): Profile solid holdup at y = 14.08m

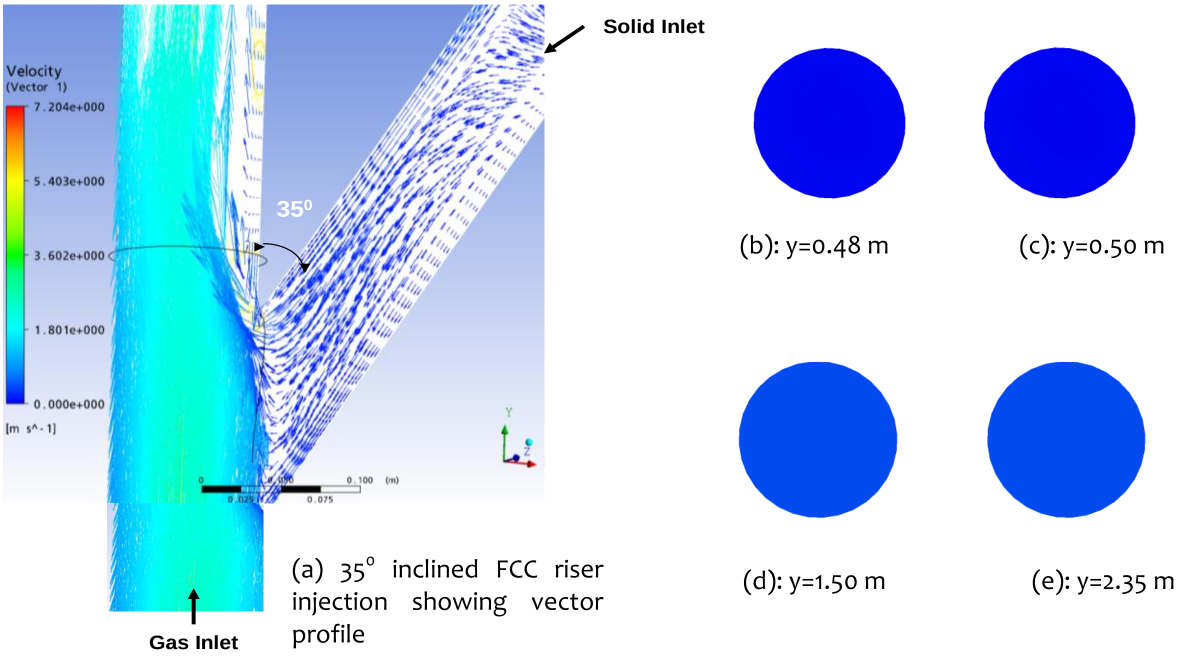
Figure 4: Radial profiles of FCC particle volume fraction along the FCC riser reactor (z = 0.95 – 14.08 m), that is Plot 1 - 6. The grid size 578,240 cells were used in fluent, case 1.2 (Huang et al., 2007)

From Figure 4, there are close similarity between the two modelling methods (fluent algebraic-KTGF and transport-KTGF) used. However, the predictions have shown that with increasing solid circulation rate, at constant superficial gas velocity, the flow development becomes very slow to reach fully developed stage. This is not the case in relation to the studies from Taghipour and Almuttahar (2008). In their own case, the flow pattern was non-uniform and from the 10 m riser length, they only record approximately 3.8 m, particles move down the wall of the riser. These study agree with the work of (Malcus et al., 2002; Manyele et al., 2002), which shows that the coexistence of up flow near the bottom and downward flow in the upper regions at the wall of the risers has a higher gas velocity and solid mass flux.

4.2 Simulation of Experimental Case Study 2 (Van engelandt et al., 2007) using ANSYS CFD: Aerated Simulation of Case 2.3

The study of Van engelandt et al., (2007) involves case 2.1 (5.3 m/s and 0.5 kg/m^2s), case 2.2 (6.4m/s and 3 kg/m^2s) and case 2.3 (7.4 m/s and 4.5 kg/m^2s). The choices of case 2.3 were used in this modelling simulation, because it is similar to the flow condition in an FCC riser reactor. Equally, data for the validation of the predictions of this case is available from the literature. Although, all the three cases (2.1, 2.2 and 2.3) are of low density, which characterises fast fluidisation. For the case 2.3, the volume fraction was determined to be about 4.737624E-04. The differences of this study with the previous simulation were on the solid particle inclined injection at angle of 35^0. The previous modelling studied was based on injection that is at right angle to the riser wall. The details of the simulation of this case are represented in Table 1. Since the modelling of aerated flow is applicable in FCC riser operation, because it give better entrainment, less pronounced broader bypass zones, improved radial mixing and a good reduction of the penetration depth. Therefore, the inclusion of non-aerated flow situations was not discussed in this study. In this simulation, the solid particles size of 77 μm and density of 1550.0 kg/m^3 was used. The CFD simulation were used only for case 2.3 (4.5 kgm^2s^{-1} and 7.4 m/s) reported from Van engelandt experimental study.

In aerated case study, gas flows to aid the pneumatic transportation of solids particle in the solid-entry zone into the riser system as shown in the schematic detail of Figure 5. The aerated gas flow was modelled at volumetric flow rate of 1.0 m^3/h (or 2.7778 m^3/s), as specified in the experimental setup. The setup of boundary condition for this modelling was based only on gas from the gas-inlet-zone, and gas flow with the FCC solid particles from inclined solid-zone. In this simulation, the effect of the recirculation (re-attachment) zone and eddy current was observed in the predicted profiles. It might be possible to assume that, the capturing of recirculation zones in CFX were possible may be because, and these experimental cases (2.1, 2.2 and 2.3) is of very low solid loading. The inlet into the riser pipe also show the effect of recirculation, and the effect is shown on the solid velocity vector along the axial flow of the riser system.



(a) 35° inclined FCC riser injection showing vector profile

(b): y=0.48 m (c): y=0.50 m

(d): y=1.50 m (e): y=2.35 m

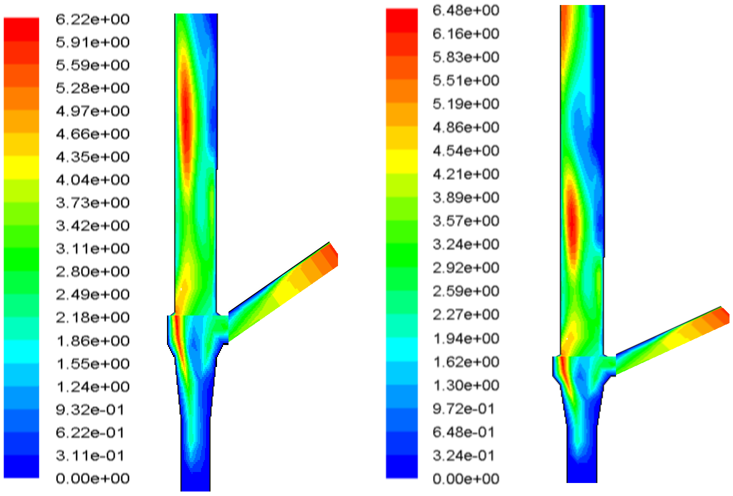
Fig 5(a)-(e): FCC Particle velocity profile showing the vector flow field, grid size is 598,570 cells (CFX-3D) case 2.3 (Van engelandt et al 2007)

The modelling process was carried out at ambient conditions of temperature and pressure. Figure 5 represents the flow field of the FCC particle and solid holdup (a) – (e) predicted in ANSYS CFX model. The mixing effect of the gas-solid flow from the injection zone into the riser was well captured by the solver.

This Simulation prediction confirm the asymmetrical profiles from the reported experimental data (Van engelandt et al., 2007), which must have been the attribution of the aeration effect side gas-inlet from the inclined injection zone. Secondly, fluent solver was used for the simulation as depicted in Figures 6(a) and 6(b). The predictions show some deviations in these figures indicate that the boundary conditions of the gas-solid flow model may have over-predicted the wall shear stress.

At corresponding riser heights of y = 0.48m, 0.50 m and 0.58 m as respectively shown in Figures 6(a), (b) and (c). The simulated axial velocities in the XZ-plane are in good agreement with the experimentally observed values. These figures indicate that the simulated solids velocities in the bypass zone are slightly low as compared to the measurement reported. While the predictions from the centre zone showed some close similarity with that from the experimental data. When proceed up the FCC riser (at z = 2.35 m), that is, at higher positions greater than 1.0 m, the solvers predictions are also of good agreement with the reported data.

However, there were slight deviations, which may be due to the viscous stresses from the gas-solid flow. The slight deviations are between flow velocity of 1.0 – 2.0 m/s, but decrease at the bypass-zone of the riser reactor. This finding is also in agreement with the work of (De Wilde et al., 2005). In riser system operations, the feeding conditions and the type of the solids affect the bottom operation and gas-solid mixing largely. However, for the case of FCC particles the acceleration zone in the riser is to a large extend. Low gas velocity and or high solid concentration result in an increased penetration depth of the solids jet and in explicit bypass zones in the plane facing the riser inlet. The radial mixing quickly dissipates the non-uniformities introduced by the FCC solids inlet. These statements also support the experimental studies of (Van engelandt et al., 2007).



(a): t = 4.5 sec (b): t = 5 sec (c): t = 6 sec

Fig 6(a)-(c): FCC Particle velocity profile showing the vector flow field (FLUENT-3D), grid of size 578,550 cells, case 2.3 (Van engelandt et al 2007)

A quantitative comparison between the two solvers (CFX and Fluent-3D) with the experimental data is shown in Figure 7 (a) – (d). The predictions in the both cases have shown that the radial component (x) of the FCC solids velocity increases, the maxima of the axial (z) solids velocities become clearly pronounced. Therefore, the position of the off-centred maxima shifts towards the riser wall.

Figure 6 (a) and (b) prove the effect of positive gravitational flow of the aerated gas-FCC solid in the riser reactor. It was observed that the flow in the inclined 35^0-injection system has a quick response to the flow situation before reaching the mixing zone. The effect of aeration as stated by Arastoopour (2001) is that, the local aeration in the standpipe resulted in less accumulation in the inlet and in more uniform inlet mixtures into the riser.

In this simulation, the KTGF-algebraic was used in CFX, while transport-PDE modelling approach was applied in fluent code. On the overall predictions, the results from the fluent code give a better agreement when compared with that from CFX and experimental data used.

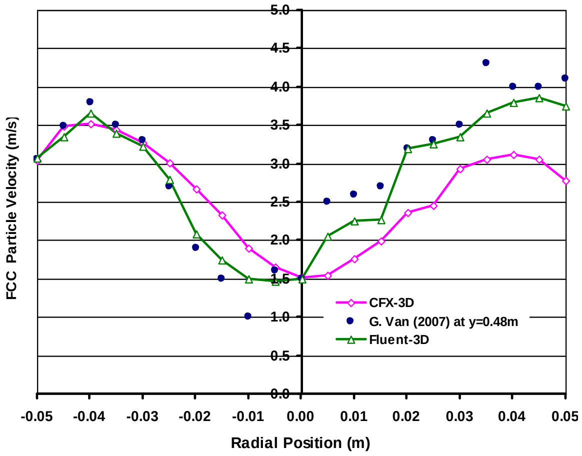


Fig 7(a): Profile of particle velocity at y = 0.48m

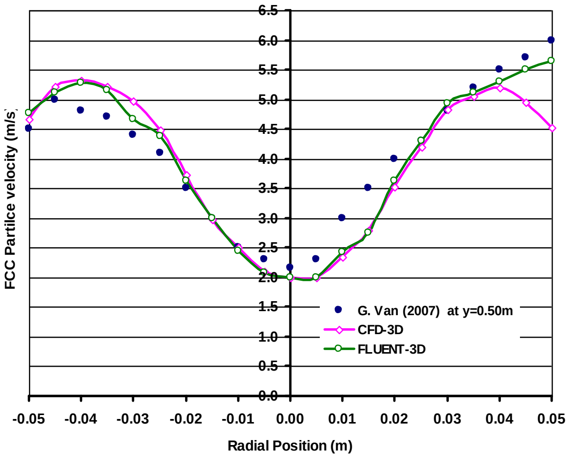


Fig 7(b): Profile of particle velocity y = 0.50m

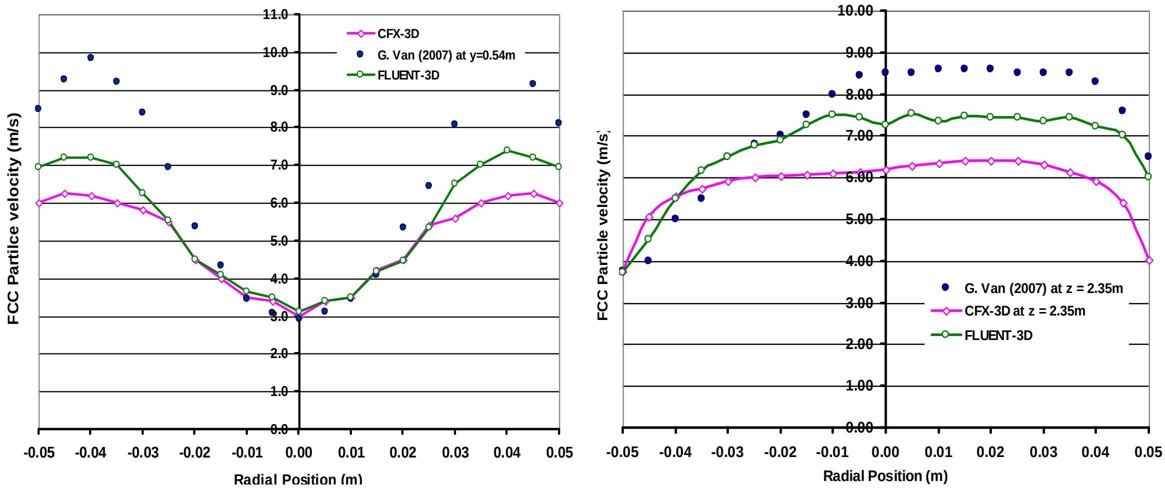


Fig 7(c): Profile of particle velocity at y = 0.54m   Fig 7(d): Profile of particle velocity y = 2.35m

Fig 7: Comparison of experimental and FCC velocity profile in radial axis (XZ-Plane), grid size is 598,570 cells (CFX-3D) and grid of size 578,550 cells, case 2.3 (Van engelandt et al 2007).

5.0 CONCLUSIONS

A comprehensive modelling strategy was adopted to study the hydrodynamics of gas-solid flow in an FCC riser reactor system using 3D grid. Primarily, the model was based on the E-E approach, using continuity and momentum modelling equations, together with constitutive modelling equations, using turbulent k-ε turbulence model for dilute assemblies of the particulate solid, and Gidaspow drag model for the CFX 11.0 and Fluent. In each stage predictions obtained using non-KTGF (Miller-Gidaspow 1992), KTGF-algebraic and KTGF-transport modelling approach have been described and experimentally validated. However, there are noticeable deviations, in terms of FCC particle velocity profiles, solid-holdup distributions, obtained by the first two approaches, which was finally improved using transport-KTGF modelling approach in fluent solver. However, the overall agreement of these predictions with the reported experimental data was reasonably good enough to attest the limitation of the solver capability. In addition, the use of only the kinetic-collisional contribution in the granular stress tensor is another limitation to the solver predictions. The mixing effect was well accounted for, in inclined FCC solid injection system. More so, the mixing effect was not only observed in fluent solver to have better predictions using Fluent solver, equally the solver was able to capture the re-circulation of flow in the wall region along the riser reactor length.

The gas-solid hydrodynamics show a degree of sensitivity to grid dependency with regard to the FCC particle velocity, solid hold up, and drag coefficient. But in this study, we found that with a value of the coefficient of restitution c, of 0.99, the coexisting dilute and dense phases in turbulent fluidised reactor can be predicted accurately and correctly. There are no significant differences in the FCC solid holdup predicted along the riser length by Gidaspow, Syamlal O'Brien or Arastoopour drag law models. In this study, it was found that increasing the overall solids circulation rate, G_s, and decreasing the superficial gas velocity, U_g, increases the solids holdup in the wall region throughout the FCC riser length, which also agrees with previous studies.

The novelty of this study was the success of achieving CFD predictions for a riser of length 15.1 m, whereas the literature has shown that in previous studies the length was about 10 m. In addition, 3D geometry with refined grids was used in ANSYS CFD 12.0 (CFX-3D and

Fluent 12.0) which differs from previous studies that use mostly 2D and coarse grids. Transient calculations including KTGF were used, although some of the previous studies also apply these concepts in 2D, but no report yet on the KTGF-transport modelling in 3D geometry. Fully turbulent flow simulation was achieved in this study, which differs from many previous studies that mostly used laminar flow calculations.

More so, the use of aerated solid flow in an inclined injection system lead to a better entrainment and improve radial mixing, which differ for the case of non-aerated solid inlet region, which is not good for an operation of FCC riser reactor system.

Symbol and Abbreviation

BF	Bubbling Fluidization
CF	Slugging Fluidization
CFB	Circulating Fluidised Bed
FB	Fixed Bed
FCC	Fluid Catalytic Cracking
FF	Fast Fluidization
PC	Pneumatic Conveying
TF	Turbulent Fluidization

Acknowledgement

The authors acknowledge the support of the petroleum technology development fund (PTDF), the Univation Aberdeen, and the acceptance to publish this article by the International Institute of Academic Research and Development (IIARD).

REFERENCES

Chen Y. M, (2006), '*Recent advances in FCC technology*' Powder technology 163, pp. 2-8

De Wilde J., Van engelandt G., Heynderickx G. J., and Marin G. B (2005), '*Gas-Solids Mixing in the inlet zone of a dilute Circulating Fluidized Bed*'. Science direct Powder Technology Journal pp. 96 - 116

Huang, W., Yan, A., and Zhu, J., (2007), *'Hydrodynamics and Flow Development in a 15.1m Circulating Fluidised Bed Riser'*, Chem. Eng. Techno. 30, No. 4, pp. 460 – 466

Kunii D. and Levenspiel O., (1991), '*Fluidization Engineering'* 2nd Ed., Butterworth-Heinemann Boston M. A. London, 459, pp. 36 – 354.

M. N. Idris and A. Burn (2008), *'Hydrodynamics Studies on an Injection System as Applicable to Modern Circulating Fluidized Bed (CFB) Riser using Computational Fluid Dynamics (CFD)'*, AIChE centenary journal, pg. 36-46 . (AIChE 2008 Conference Paper) 761c, pp. 661.

M. N. Idris, O. Stephen and A. J. Aduma (2016), *'Modelling and Simulation of Flow Hydrodynamics in a Circulating Fluidised Bed (CFB) Riser Reactor'*, Journal of Research and Innovations in Engineering Volume 1, Number 1, University of Uyo, Nigeria.

Malcus, S., Pugsley, T. S (2002), '*Hydrodynamic Model for High and Low-Density CFB Risers. In Circulating Fluidized bed Technology VII'*; Grace, J. R., J. X., De Lasa, H. I., Eds.; Canadian Society of Chemical Engineering: Ottawa, Ontario, Canada; p 395

Manyele, S. V., Parssinen, J. H., Zhu, J. X (2002), *'Characterising Particle Aggregates in a high-density and high-flux CFB riser'*, Chemical Engineering Journal 88, 151 – 161

Miller, A and D. Gidaspow (1992) '*Dense, Vertical, Gas-Solid Flow Field'*, J. Fluid Mech. 140, 223 – 256

Mustafa A. (2005) '*Simulation Studies of Gas-Solid in the Riser of a Circulating Fluidized Bed*' Conference Sponsors: Advanced Energy Systems ISBN: 0-7918-4183-9 FBC2005-78014, pp. 201-207

Taghipour, F and Almuttahar, A., (2008), '*Computational Fluid Dynamics of High Density Circulating Fluidized Bed Riser: Study of Modelling Parameters'*, Powder Technology 185, pp. 11 - 23

Van engelandt G, Wilde De J, Heynderrickx G. J., And Martin G. B., (2007), '*Experimental Study of Inlet Phenomena of 35^0 Inclined non-Aerated and Aerated Y-inlets in a Dilute Cold-Flow Riser'*, Chemical Engineering Science pp. 339 - 355

Yan, A. J, and Zhu J. X., (2004), *'Industrial Engineering Chemical Resources'*, 43, 5810

Yang, N., Wang, W., Ge, W., Wang, L., Li. J., (2004), *'Simulation of Heterogeneous Structure in a Circulating Fluidized-Bed Riser by Combining the Two-Fluid Model with EMMS Approach'*. Industrial and Engineering Chemistry Research 43, pp. 5548 - 5561.