A STUDY OF DILUTE TO DENSE FLOW IN A CIRCULATING FLUIDIZED BED.

Claus H. Ibsen*, Bjørn H. Hjertager and Tron Solberg

Group of Chemical Fluid Flow Processes
Chemical Engineering Laboratory, Aalborg University Esbjerg
Niels Bohrs Vej 8, DK-6700 Esbjerg, Denmark
*e-mail: ci@aue.auc.dk

This work concerns measurement carried out with LDA/PDA equipment in a lab-scale CFB system. The focus is put on how the amount of particles influences the flow in the CFB. Experiments are performed with an increasing amount of particles, whereby data of axial velocity, RMS velocity and diameter is obtained. The experimental data is used to evaluate an in-house multiphase CFD code.

INTRODUCTION

Modeling the hydrodynamics of circulating fluidized beds (CFB) is very much influenced by the distribution of particles. Therefore proper understanding of the influence of the particles is needed when modelling CFBs. This work continues the work by Mathiesen et al[1] who showed a significant increase of mean diameter in the wall region with a similar CFB system as presented here. Their study however, included a dilute flow and only two distinct particle sizes.

The objective of this work is to investigate the influence of hydrodynamics of the CFB system by increasing the amount of particles in the system and using particles having a full size range. Furthermore it will investigate the occurrence of a possible increase of mean diameter in the wall region as found by Mathiesen et al[1]. Finally measurement data will be given using Laser/Phase Doppler Anemometry for validating multiphase CFD codes.

EXPERIMENTAL SETUP

The riser has an internal diameter of 0.032 m, is 1.0 m high and made of glass. The primary gas inlet is placed at the bottom of the riser. At the top of the riser the suspended particles enter a glass cyclone where the solids are separated from the gas and recycled via a return loop. The Supply of secondary air is placed 0.05 m above the primary air inlet and feeds the solids back into the riser. The secondary air inlet has a diameter of 0.008 m. See Figur 1.

The measurements were carried out with a Laser/Phase Doppler Anemometer from Dantec.
The settings for the LDA/PDA system:

Laser wavelength	514.5nm
Scattering angel	30°
Scattering mode	Refraction
Transmitting lens	f=400mm
Receiving lens	f=400mm

Settings for the CFB system:

Superficial velocity	1.0 m/s
Glass particles, diameter	d_p=125-175μm
Glass particles, density	ρ_p=2400kg/m^3
Initial bed height over Riser height	h_i/H=0.02-0.16

NUMERICAL MODEL

The numerical part is based on a multiphase Computational Fluid Dynamics (CFD) model, FLOTRACS. It is an in-house code that uses an Eulerian description for the gas and solid phases. The conservation equations for the solid phases are based on the kinetic theory of granular flow.

The parameters for the numerical model: The code uses a 2D geometry with 29 x 109 control volumes, see figure 2. The flow which are solved are modeled with one gas phase and 3 particle phases, d_p=134 (20%), 151(60%) and 168(20%)μm. The code simulated 16 seconds of real time and the results are time averaged from the last 6 seconds. A simulation takes approximately 24 hours on a SGI Origin 2000.

RESULTS

Figure 3a shows the axial particle velocity versus height. It can be seen that the centerline velocity decreases until it reaches a more constant value. Figure 3b shows that for axial velocity as function of radial position, the local maximum velocities increase as the loading increases and that the thickness of the annulus region remains constant.

For the diameter profiles in figure 4a it can be seen that there is a decreasing axial segregation of particle size as a function of increased loading. In figure 4b is it shown that there is a radial segregation of particles by size when the loading increases.

In figure 5a and 5b it can be seen that the FLOTRACS code tends to underpredict the axial particle velocities and that the code does not predict the decreasing centerline velocity.

CONCLUSION

The experiments show how the flow structure in the CFB system changes with increased loading. The radial segregation of particle size was found but was less pronounced than the data of Mathiesen et al[1].

The data obtained will be of great value when validating CFD models. When used for validating the FLOTRACS code it was shown that the code is capable of predicting some characteristics of the flow structure in the CFB system. But further work is still needed to improve the predictability of the FLOTRACS code.

REFERENCE

Mathiesen, V., Solberg, T., Hjertager, B.H., An experimental and computational study of multiphase flow behavior in a CFB, Int. J. Multiphase Flow, 26, p. 387-419, 2000.

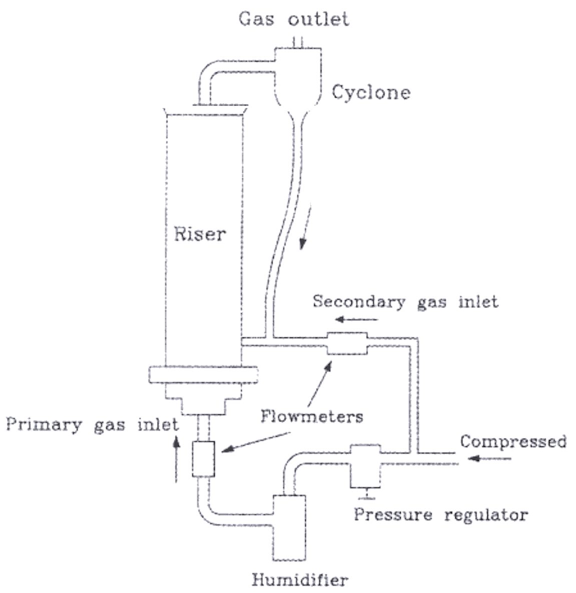

Figure 1: A schematic sketch of the CFB

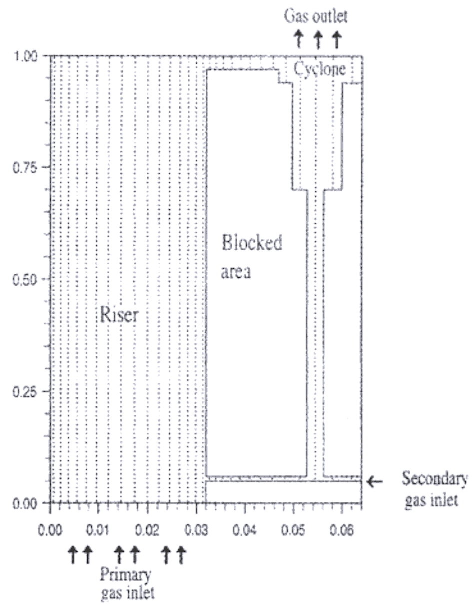

Figure 2: The calculation domain.

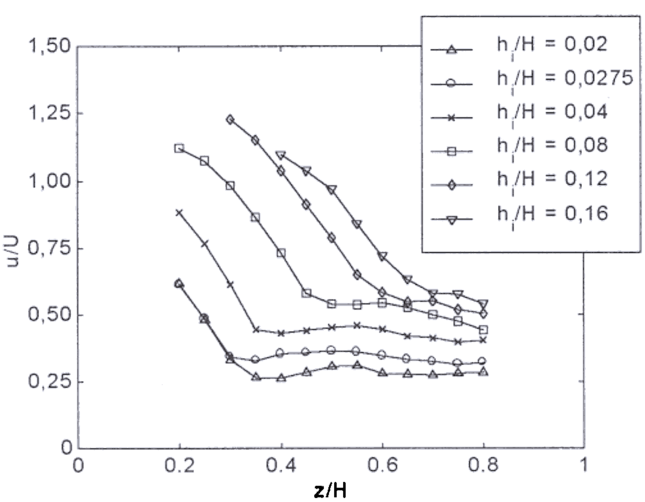

Figure 3: Axial particle velocity as function of Axial Position (y/R = 0)

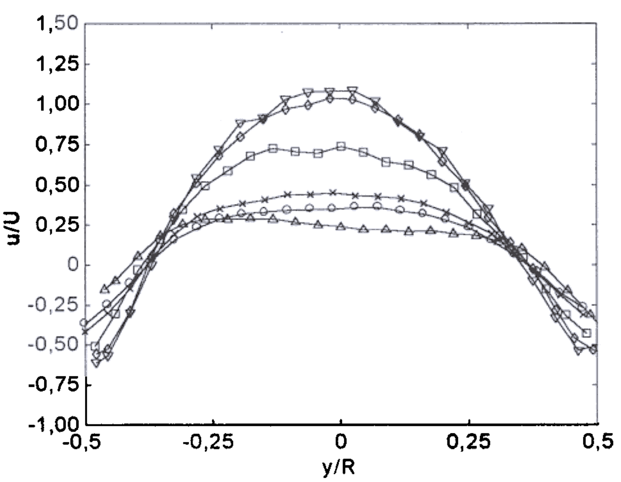

Figure 3a: Axial particle velocity as function of Radial position (z/H = 0,4)

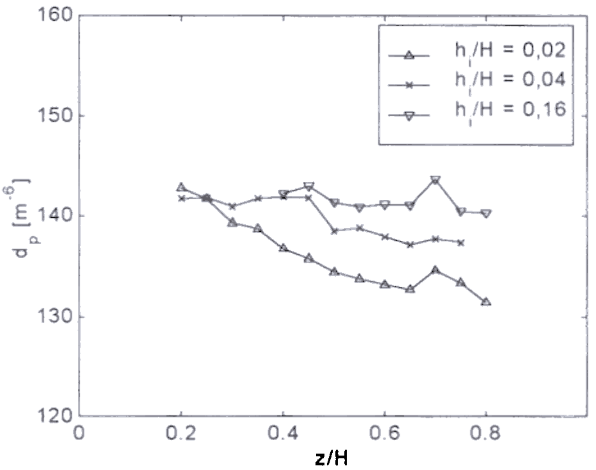

Figure 4: Particle diameter as function of Axial Position (y/R = 0)

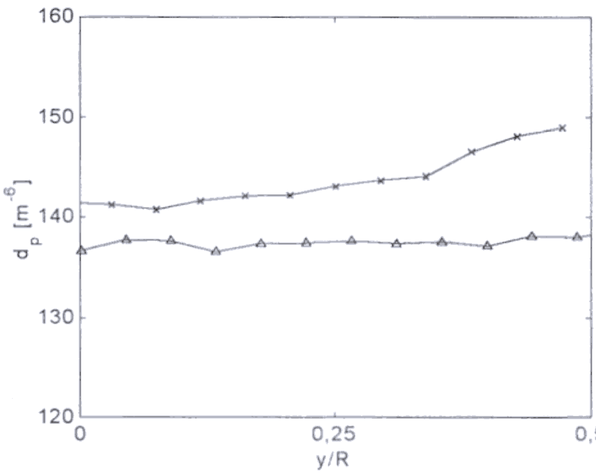

Figure 4a: Particle diameter as function of Radial Position (z/H = 0,4)

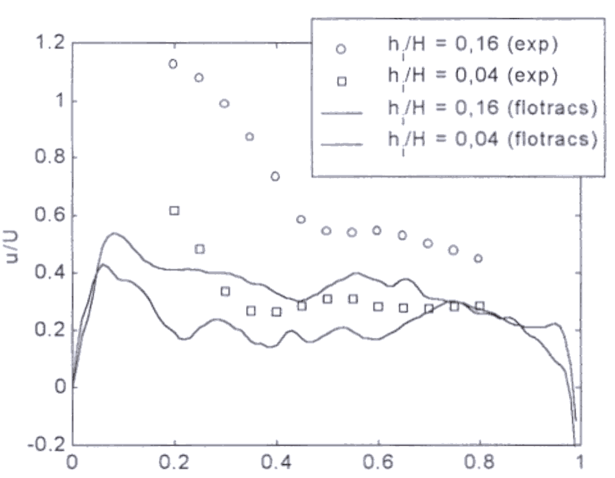

Figure 5: Experiments compared with Numerical models as function of Axial position (y/R = 0)

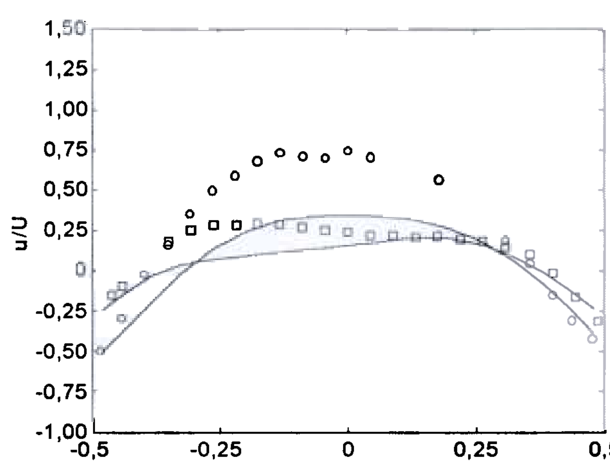

Figure 5a: Experiments compared with Numerical models as function of Radial position (z/H = 0,4)