ISSN 1848-0071
683.97+662.65=111
Recieved: 2013-09-04
Accepted: 2014-01-23
Professional paper

INFLUENCE OF AIR DISTRIBUTOR ON FLOW FIELD IN FREEBOARD AREA IN A BUBBLING FLUIDIZED BED BOILER

JAN OPATŘIL, JAN HRDLIČKA

Faculty of Mechanical Engineering, Czech Technical University Prague, Czech Republic
e-mail: jan.opatril@fs.cvut.cz

One of the basic functions of the air distributor is to provide uniform fluidization and homogenous flow field above the bed. Thus demand of CFD simulation arises from needs to understand behaviour of the fluidized bed. Situation in this case is complicated by the trough-shaped air distributor. The paper deals with 2D CFD simulations of fluidized bed and confronts it with experimental measurements on pilot scale unit. Simulations and measurements are done under ambient conditions without combustion. Simplified geometry was built up in Gambit and CFD simulations are done by ANSYS FLUENT.
Key words: fluidized bed, air distributor, CFD.

Utjecaj razdjelnika zraka na polje strujanja u slobodnom ložišnom području i mjehurastom fluidiziranom ložištu kotla. Jedna od temeljnih funkcija razdjelnika zraka je pružiti jednoliku fluidizaciju i homogeno polje strujanja iznad ložišta. Tako potražnja za CFD simulacijom proizlazi iz potrebe razumijevanja ponašanja fluidiziranog ložišta. Situacija se komplicira u slučaju kanalnog oblika razdjelnika zraka. Rad se bavi 2D CFD simulacijom fluidiziranog ložišta i uspoređuje ju s eksperimentalnim mjerenjima na pilotnoj jedinici. Simulacije i mjerenja su provedene u uvjetima okoline bez izgaranja. Napravljena je pojednostavljena geometrija u Gambit-u i provedena je CFD simulacija u ANSYS FLUENT-u.
Ključne riječi: fluidizirano ložište, razdjelnik zraka, CFD.

INTRODUCTION

The bubbling fluidized bed combustion is a perspective technology for application in the field of small and medium scale energy systems. Main advantages isflexibility for input fuel, possibility of combustion of low rank fuels and alternative fuels or pollutant emissions control [1], [2].

Development and optimization of a new boiler is a time-consuming task. In most of the cases it is impossible to look into the ongoing processes in details, usually due to their physical essence, time reasons or costs. These difficulties can be overcome by CFD modelling, which offers wide possibilities to investigate the respective phenomena.

EXPERIMENTAL DEVICE

The aim of this work is simulation of a flow field inside the furnaceof fluidized bed boiler and comparison of velocity profiles in the freeboard area with measured data. Knowledge of gas phase behaviour in the freeboard is essential for emission control since in the freeboard take place key CO and NOx conversion reactions. A pilot scale boiler with bubbling fluidized bed has been used for simulation and experimental purposes. Nominal capacity load of the boiler is 500 kW. Cylindrical combustion chamber is designed as adiabatic, thus facilitates combustion of low grade fuels with high moisture content. The boiler furnace diameter is 0,65 m and the height is 2,2 m. The furnace is divided into two regions, the bed and the freeboard.Scheme of the furnace is shown in Fig. 1.

The trough-shapedair distributor consists of thirty six nozzles immersed in the fluidized bed. Nozzles are allocatedon two parallel sides and placed horizontally in three cascade rows (see Fig. 1). Design of the distributor brings uncertainties in behaviour of the boiler as well as makes very hard prediction of flow above the bed including CFD simulations.

For numerical simulation, the geometry is too complicated and must be simplified.Also the domain was simplified to save number of cells and calculation time.Modelled central area of the bed is assumed to play a dominant role in the velocity profile development. Thus it was decided not to simulate whole bed area.The space below the nozzles was partially neglected as well as space between wall and nozzles. This simplification can be considered as satisfactory since fluidization in these regions almost does not occur.

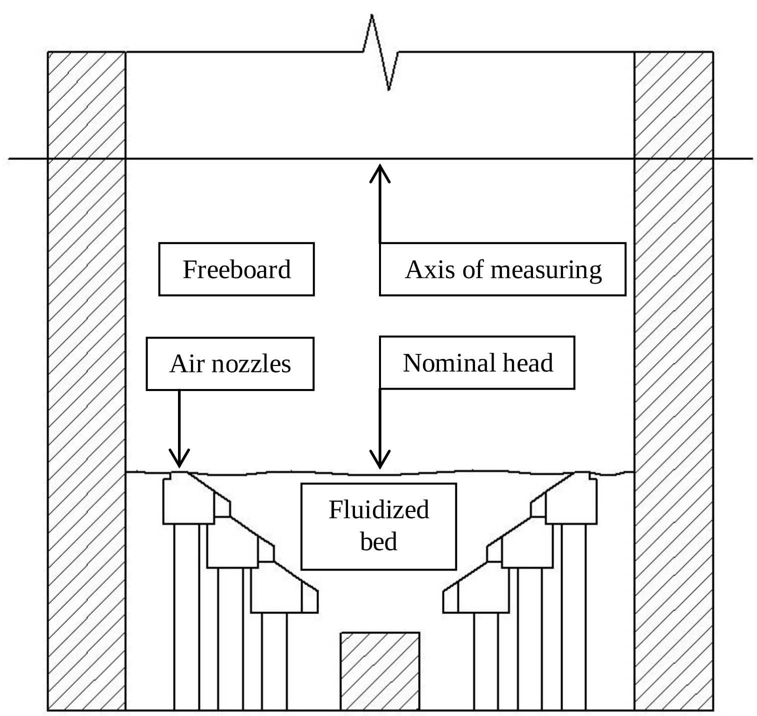

Figure 1. Shematic representation of cross-section of the furnance
Slika 1. Presjek razdjelnika zraka

The velocity profiles in freeboard obtained by simulation were compared with measurement. The experiment was carried out at “cold” conditions, i.e. the boiler was not in operation and only air was present in the freeboard area. Measured data of velocity profile were obtained at measurement axis that was placed 0,4 m above the topmost row of nozzles, as shown in Figure 1. A hot bulb probe was used for velocity measurement. This probe allows measurement of absolute velocity magnitude without measurement of the flow direction. Therefore for the comparison are used absolute values of the velocities calculated by the CFD model. The velocity profiles in freeboard are composed of thirteen points with 5 cm spacing. First and last points are in the distance of 2,5 cm from the freeboard wall. At each of the measurement points were acquired 15 velocity values in time interval of 2 seconds. From these values were calculated averages for minimization of measurement error. These averages are used in evaluation of the results. Similar approach was used in CFD simulation, thus CFD profiles are also averaged.

The bed is represented by inert material, specifically expanded clay aggregate. Also it is referred as lightweight ceramic aggregate (LWA) or exclay.For combustion applications, the main advantages are low density, volume stability, temperature stability (up to 1050°C), and high sphericity [3]. In numerical model homogenous particles are used and thus the diameter equals mean diameter obtained by sieve analyses (d_m = 3,39 mm).

CFD SIMULATIONS OF THE FLUIDIZED BED

For modelling was chosen a 2D model due to high computational demands for two phase flow in the case of 3D modelling.The main problem of this case is in symmetry and a third dimension. The air distributor is plane symmetric, whereas surrounding wall is axisymmetric. In Fluent, the third dimension is considered equal 1. Thus the case is sensitive on inlet boundary conditions, which are modelled as a velocity inlet. The modelled domain involves processes only in two dimensions XY. As a matter of fact, spreading of stream occurs also in thethird dimension, but the 2D model cannot involve this effect. Therefore, simulated velocity profilesin the freeboard should be above the measured profiles. Respectively, the mean freeboard velocity of simulation is supposed to behigherthan the measured velocity.

It is obvious that in the 2D case is not possible to get exactlythe same results obtained by measurement. However, similar trends in behaviour and shape of velocity profiles can be expected.

Uniform hexahedral mesh size, with spacing 5mm, was done by meshing scheme called PAVE. A coarser grid was also processed, but the solution was notwell convergent, thus the final grid is result of an optimization process.

The setting up of a model was done according to recommendations from literature [4] and [5], as well as from papers focused on related issues. Critical is choice of models such as model of turbulence and model of two phase flow. The k-ε turbulence model was employed, because from previous results of measurement is obvious, that significant swirls occur. The general recommendations, for cases with highly swirling flows, are held to get better fitting results. In case of two phase flow, the Eulerian approach is right choice, where both phases, air and LWA, are treated as fluids.

Inlets of air are set up as velocity inlet and outlet is modelled as pressure-outlet. For velocity inlets is necessary to set up turbulence intensity together with hydraulic diameter. Input values were calculated according recommendations in literature [4].

It is necessary to pay attention on convergence of solution. Generally recommended approach to get or improve convergent solution was employed. It means that obtaining initialization flow field is necessary. Without initial solution the calculation becomes divergent. So at first, initial flow field is calculated only for primary phase (air), and then full Eulerian model is solved. Also very small time steps are recommended, especially this case allows time step 0.001 s.

Other sensitive parameters are under relaxation factors, which had to be significantly decreased for momentum as well as for volume fraction.

Indeed the convergence of initial flow field has really good behaviour, values of scaled residuals fall on order 1e-8 and some of them lower. However, situation changes in the moment of switching volume fraction equation on. As in many cases, the continuity equation had the worst convergence behaviour. Thus that is the reason for gradually decreasing and it provides value of continuity equation residual about 1.5e-5. However, after implication of volume fraction equation, the residuals do not fall so far.

RESULTS AND DISCUSSION

Three levels of fluidized bed depth were simulated under same settings and air inlet velocity at nozzles (v = 8,17 m/s). Fluidized bed levels were measured for conditions of fixed bed. For simplification, levels of depth are labelled as follows:

- S1-the nominal depth, where the free surface of the bed is at the same level as the topmostrow of nozzles (see Fig. 1);
- S2-35 mm above the nominal depth;
- S3-70 mm above the nominal depth.

Figures Fig. 2 (S1), Fig. 4 (S2) and Fig. 3 (S3) represent results obtained by simulation of 2D multiphase flow compared to data obtained by measurement.

In the Fig. 2, the results of simulation S1 did not meet expectation mentioned above,that mean freeboard velocity of simulation is greater than velocity given by measurement. In contrast with S2 and S3, see Fig. 4 and Fig. 3, the simulated velocity profile is not situated under the measured data. It means that the flow is significantly more swirling in the real situation compared to the simulation. However, the nominal depth is distinguished itself by sharp changes in velocity magnitude for both of cases (measured data and simulation). Evaluation of velocity deviations is listed in Tab. 1.

Quality of combustion significantly depends on mixing in combustion chamber and in this case it is secured by distributor together with fluidized bed. It is obvious, that good velocity profiles above the bed, respectively required uniform air distribution, depends on velocity of air at inlet and height of fluidized bed.

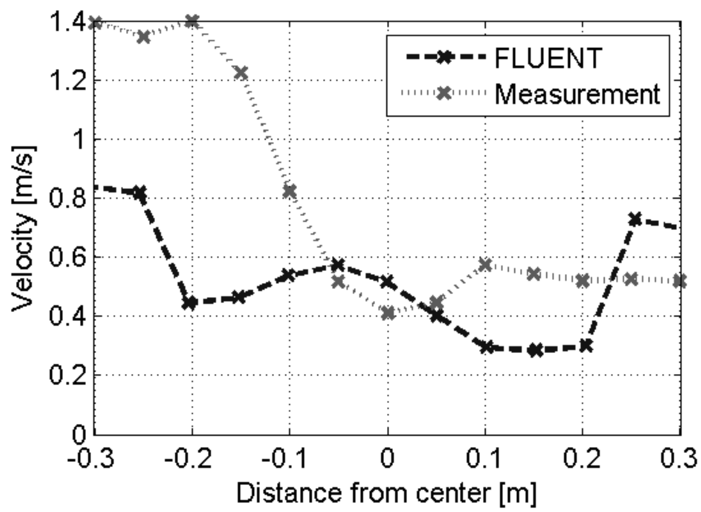

Figure 2. Data comparison, nominal head S1
Slika 2. Usporedba podataka, nominalne visine S1

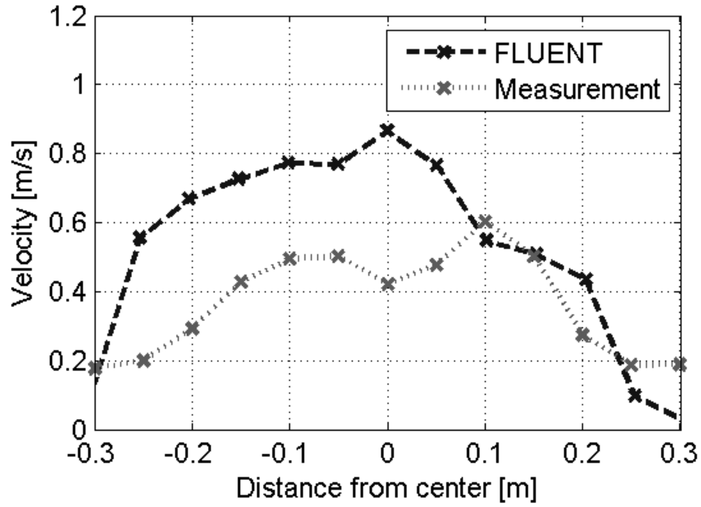

Figure 3. Data comparison, high head S3
Slika 3. Usporedba podataka, maksimalne visine S3

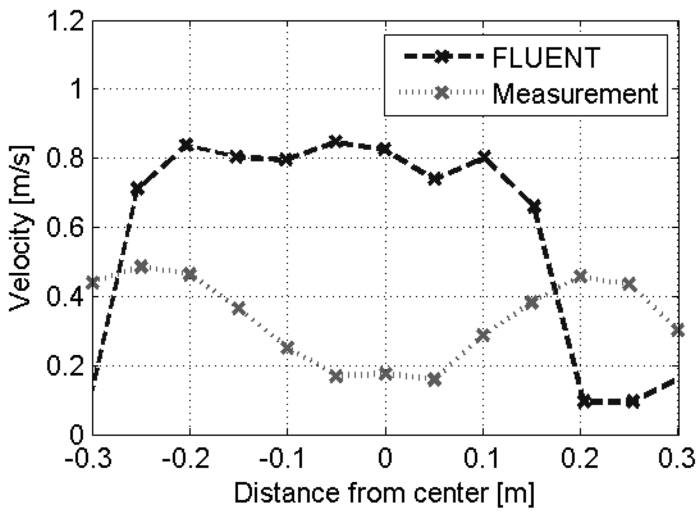

Figure 4. Data comparison, medium head S2
Slika 4. Usporedba podataka, srednje visine S2

S1 – sharp changes in velocity magnitude between adjacent points
S2 – two peaks closed to the wall region and linear behaviour in central area
S3 – velocity peak close to the chamber centre with gradual decreasing of velocity toward to the wall region

Results for case of nominal depth (S1) are questionable and it is better not to consider for validation of fluidized bed CFD model. From the point of view of uniform fluidization, the nominal depth is the worst case, because high velocity differences occur between adjacent points. Measured data analysis leads to an idea that it is caused by insufficient height of the fluidized bed. Respectively pressure drop of the bed is too low and hence behaviour of the bed is sensitiveto generation of air flow channels due to high inlet velocity at nozzles, which seems to be playing a dominant role on flow in the case S1. Another issue is influence of geometric inaccuracy of the air distributor as well as non-uniformity of the freeboard cylinder, which is lined by the brick lining. It is not possible to involve the imperfections in the CFD model. In other words, the fluidized height is not sufficient to provide uniform fluidization and behaviour of the air distributor becomes hard predictable.

On the other side, cases involving higher fluidized bed are more "stable" and simulated velocity profiles are similar to the measured profiles. Thus is reasonable to consider that 2D CFD simulations are suitable for simulations of the trough-shapedair distributor with sufficiently high fluidized bed. Also velocity changes become smoother and so fluidization becomes more uniform.

Table 1. Velocity deviations
Tablica 1. Brzina odstupanja

Case	Deviation of mean measured and simulated velocity at the given level [%]	Mean velocity deviation at single points [%]
S1	-32,51	-15,35
S2	70,56	121,32
S3	44,03	40,49

CONCLUSION

The primary aim of this paper was validation of the simplified 2D CFD model based on comparison with experimental data. Results show that the actual model is very well suitable for case with the highest fluidized bed height, respectively the biggest pressure drop. Other cases are better to not consider as validated. However, similar trends in behaviour were observed.

It is obvious that accuracy of simulation primarily depends on fluidized bed height. A model without simplification of geometry was also tested, butthere were no significant differences in results. Therefore is necessary to pay attention for shallow fluidized beds, where different physical phenomenon becomes more dominant.

The pilot scale unit with power input approx. 500 kW is dimensionally close to the small industrial units, so the results could be transferred to a full scale unit. Thus, based on measured data is possible to observe that fluidized boiler must be operated with sufficient bed height to avoid of sharp changes in velocity profiles. However, this conclusions are valid only for similar shaped air distributor.

REFERENCES

[1] J.R., Howard. Fluidized Bed Technology: Principles and Applications. New York: Taylor & Francis Group, 1989.

[2] Ravelli S., Perdichizzi A., Barigozzi G. „Description, applications and numerical modelling of bubbling fluidized bed combustion in waste-to-energy plants.“ Progress in Energy and Combustion Science, Volume 34, Issue 2, April 2008: Pages 224-253, ISSN 0360.

[3] http://www.liapor.cz/cz/zakladni-vlastnosti-kameniva

[4] ANSYS FLUENT Theory Guide, Release 14.0.November 2011.

[5] ANSYS FLUENT User's Guide, Release 14.0.November 2011.