

# Validation of CFD model for mixture flow in FGD reactor

JAN NOVOSÁD, PETRA DANČOVÁ, TOMÁŠ VÍT  
 Department of Power Engineering Equipment  
 Technical University of Liberec  
 Studentská 1402/2, 461 17 Liberec  
 CZECH REPUBLIC  
 jan.novosad@tul.cz

*Abstract:* This article describes the flow of gas mixture inside the flue gas desulfurization (FGD) reactor. This reactor is self-constructed for using as experimental verification tool in research of modeling the desulfurization process. Inside the reactor space there is situated a spray nozzle for injecting water droplets into the gas. During the experimental study the velocity field in the reactor spray zone were measured. For CFD modelling the modification of standard OpenFOAM solver is used. CFD simulations were performed to model flow of multicomponent gas. Simulation results (velocity fields) are confronted with experimental data. Based on the comparison the good accordance with the real state was found. The conclusion is that the developed OpenFOAM solver could be used for future research, but further validation is recommended.

*Key-Words:* desulfurization, CFD, OpenFOAM, mixture, gas

## 1 Introduction

In the Czech Republic the most electricity is produced by coal power plants. Amount of pollutants, which are harmful for the environment (e.g.  $\text{NO}_x$ ,  $\text{SO}_2$ ...), has to be reduced under the legal limits. For reducing sulfur dioxide the devices called wet scrubbers are used. Process inside the scrubber is called flue gas desulfurization (FGD). Principle of wet FGD is based on absorption of sulfur dioxide into a droplets of limestone slurry. Inside the scrubber there is multicomponent multiphase flow. It could be simplified to flow of ideal gas mixture with limestone slurry droplets [1].

The aim of our work is to develop the numerical model of FGD process implemented into OpenFOAM computation software. This model should solve different physical problems like gas flow, injection of discrete phase (droplets of limestone slurry) and chemical reactions.

In previous work [2] we found that we need to build a scaled model of FGD reactor to verify the numerical results. The testing measurement and comparison of its results with results from initial CFD simulations were presented in work [3]. Based on this comparison, the recommendations for improving quality of measurement results were defined.

The CFD model was modified to reach more precise results with better compliance with experimentally measured data. Aim of this work is to show the comparison between velocity fields, measured inside the experimental reactor in

comparison with velocity fields gained from modified numerical solver.

Because of needs for simulate the injection of droplets, new piping and nozzles for spraying water were mounted inside the experimental reactor.

Based on these modification, new geometry of CFD model was created. The CFD solver (described in [3]) has been modified to allow modelling of multiphase flow using the Euler-Lagrange approach, that have been successfully used by Marocco and Inzoli [4] for simulations of processes in wet flue gas desulphurization technology. Before multiphase flow modelling, the solver for flow inside the reactor has to be validated.

Development of the fully worked solver should be divided into several steps:

1. Modelling of simple flow of air.
2. Modelling of mixture flow.
3. Modelling of multiphase flow.
4. Modelling of chemical reactions.

This article deals with step 2 aiming on the mixture flow modelling.

## 2 Experimental reactor

The scaled model for simulating the operation of flue gas desulphurization reactor was built. Based on the actual needs for our investigation, some modifications of its design were performed.







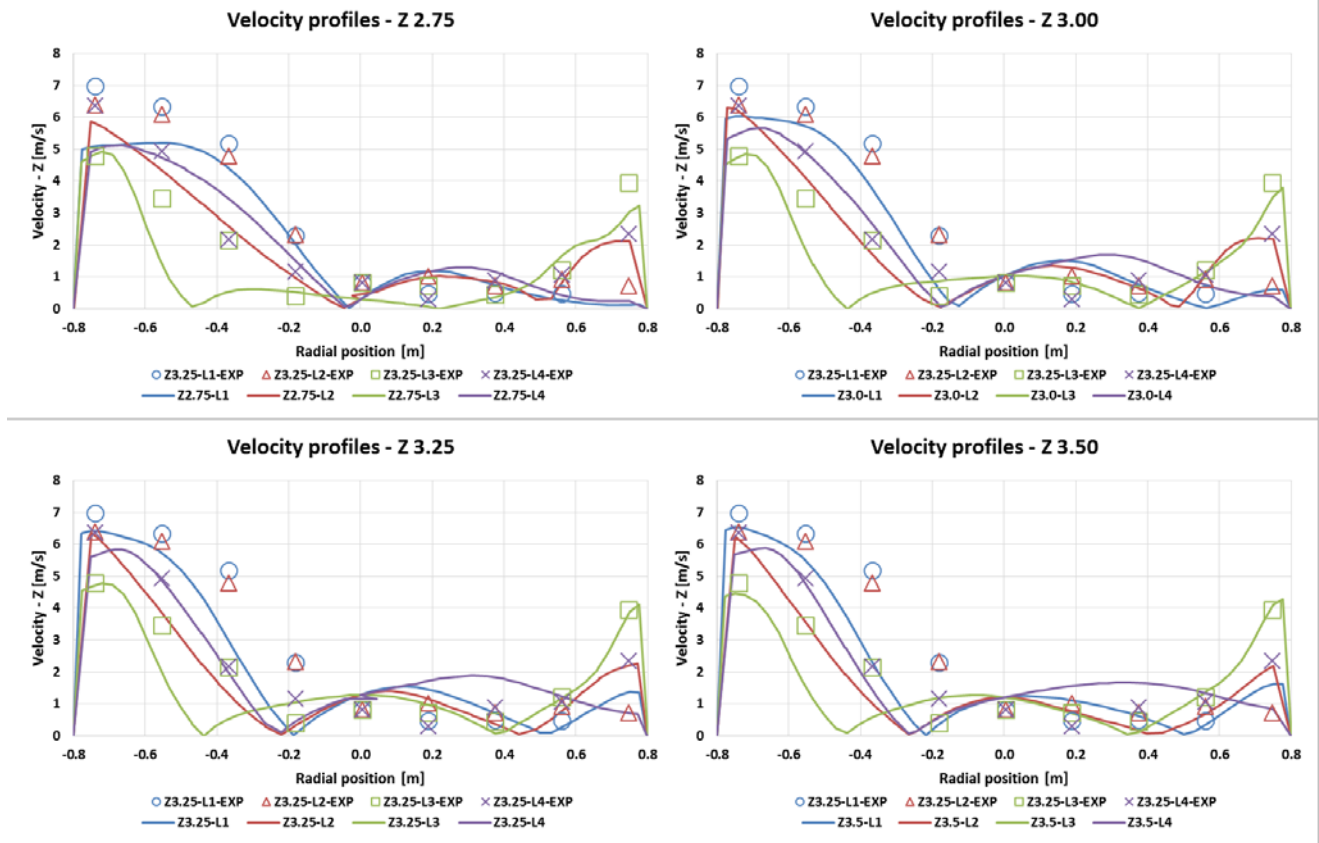


Fig. 8 – Velocity profiles compared with experimental results-  $Z = 3.25$  m

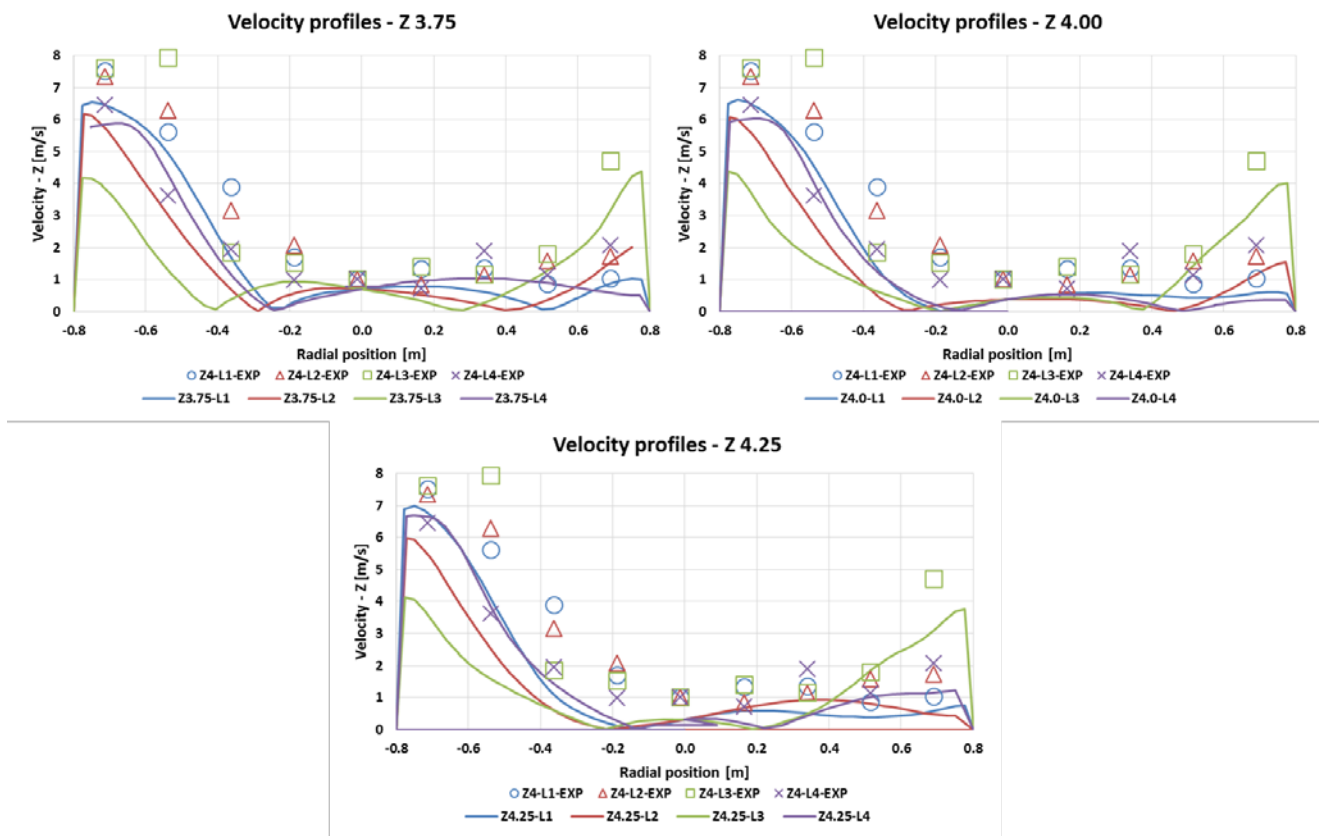


Fig. 9 – Velocity profiles compared with experimental results –  $Z = 4.00$  m

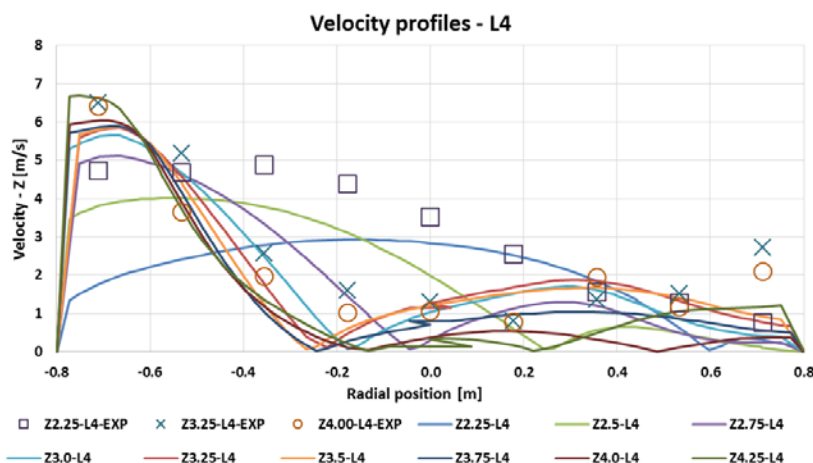


Fig. 10 – Velocity profiles compared with experimental results -  $Z = 4.00$  m

## 4 Conclusion

In this work the investigation of velocity fields inside the scaled model of FGD reactor by experimental and numerical methods have been performed. Initial conditions and parameters of CFD model, especially velocities at inlet area, were set to correspond well with experimental setup.

Experimental part was performed with using the possibilities and components of experimental reactor (e.g. nipples as entrances for inserting Prandtl probe) and available equipment. Experimental device consists of Prandtl probe and TESTO 480 data logger. Measuring points have been divided into three groups (each with 33 points) which represents three different height level inside the spray zone of the reactor. Experiments were performed by one flow regime (40% of the fan power).

As we can see from the results published in figures 7-9, the velocity field is very variable across the section of reactor. The typical feature inside the profiles is that velocity increases very significantly with the distance from the center. Maximum speeds are located in regions that are opposite to the inlet. It is probably caused by the inertia of flow from the inlet rectangular channel. Velocity in planes with  $Z$  coordinate close to the inlet area (shown in figures 7,10) have a worse match with experimental data. It could be caused by the effects of other velocity components ( $X$  and  $Z$ ), that are connected to turbulence phenomena and wake in these areas.

The turbulence effects are not included in experimental data, because the velocity values are time-averaged.

Generally it could be said that the velocity profiles obtained from CFD simulations are in a

good accordance with experimental data. In the regions, where the main flow stream is located, the difference between experimental and numerical data do not exceed over 15 %.

Finally it can be recommended to the future work to validate the CFD solver using experimental data for different flow regime (power of fan) and then continue with development of the solver by adding the solution for multiphase flow with water droplets.

## Acknowledgement

This publication was written at the Technical University of Liberec as part of the project "21135" with the support of the Specific University Research Grant, as provided by the Ministry of Education, Youth and Sports of the Czech Republic in 2017. Author gratefully acknowledges financial support by Czech Technological Agency under the project TACR TA04021338.

## References:

- [1] Clayton T. Crowe, John D. Schwarzkopf, Martin Sommerfeld, Yutaka Tsuji, *Multiphase flows with droplets and particles*, CRC Press Taylor & Francis Group, 2012.
- [2] Jan Novosád, Tomáš Vít, Numerical simulation of flow in the wet scrubber for desulfurization, *EPJ Web of Conferences*, Vol. 92, 2015, pp. 02055p.1-p.4.
- [3] Jan Novosád, Petra Dančová, Tomáš Vít. Experimental and Numerical Study of Velocity Profiles in Fgd Reactor. *WSEAS TRANSACTIONS on APPLIED and THEORETICAL MECHANICS*, Vol. 11, 2016, pp. 148, p.1-p.6. E-ISSN: 2224-3429
- [4] Luca Marocco, Fabio Inzuli. Multiphase Euler-Lagrange CFD simulation applied to Wet Flue Gas Desulphurisation technology. *International*

*Journal of Multiphase Flow*. Vol. 35, 2009, p.  
185-194.