Velocity fields in FGD reactor by different flow regimes

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 gas flow 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. During the experimental study the velocity field at the inlet of the reactor and inside the spray zone were measured. Flow inside the FGD reactor could be modified by controlling power of the radial fan. This control is provided by frequency converter connected to the electric fan engine. For measuring of velocity Prandtl probe connected to TESTO 480 data logger is used. Obtained values of velocity at the inlet were used to set boundary conditions for CFD modelling. For CFD modelling the self-made modification of standard OpenFOAM solver is used. CAD model of the reactor was edited to the simplified geometry of fluid inside and several different computational meshes with different element size were created. After that CFD simulations were performed to study the effect of mesh size to results of simulations (velocity fields). These results are confronted with experimental data to assume the useful mesh. Whole process was made for two different flow regimes with 40% and 70% of maximal power of the fan. Based on the comparison the good agreement with the real state was found. The conclusion is that the mesh with generated elements could be used for further simulations.

Key-Words: desulfurization, reactor, flow, velocity profile, CFD, OpenFOAM

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. NO_x, SO_2…), has to be reduced under the legal limits. For reducing sulfur dioxide the devices called wet scrubbers are used.

These scrubbers looks like several tens of meters tall and few meters wide cylinders. 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 with limestone slurry droplets [1].

The whole chemical process is very complicated, description of particle steps needs tens of chemical equations. There is an effort to simplify the description to several equations for using in CFD modelling.

Main chemical reactions are described with following equations [2]:

$$\mathrm{SO_2 + CaCO_3(s) \rightarrow CaSO_3 + CO_2} \tag{1}$$

$$\mathrm{CaSO_3 + 0.5O_2 \rightarrow CaSO_4(s)} \tag{2}$$

In fact the main products of these reactions are CO2, which is exhausted to atmosphere, and gypsum CaSO4, which could be used for making materials for civil engineering.

Bravo et al. in [3] described the division of the main reactions (1), (2) and physical steps of absorption into several steps. These steps include diffusion, dissolution and dissociation of SO2 and dissolution of CaCO3. Reaction kinetics of these processes may be affected by various influences, e.g. operating temperature, concentration of reactants, partial pressure etc.

Complex mathematical model of FGD process including multiphase flow with limestone slurry droplets and chemical interaction between phases was clearly and concisely described by Goméz et al. in [4] and Marocco et al. in [5],[6]. They described the approaches to CFD simulations of desulfurization process applicable to real power plant equipment.

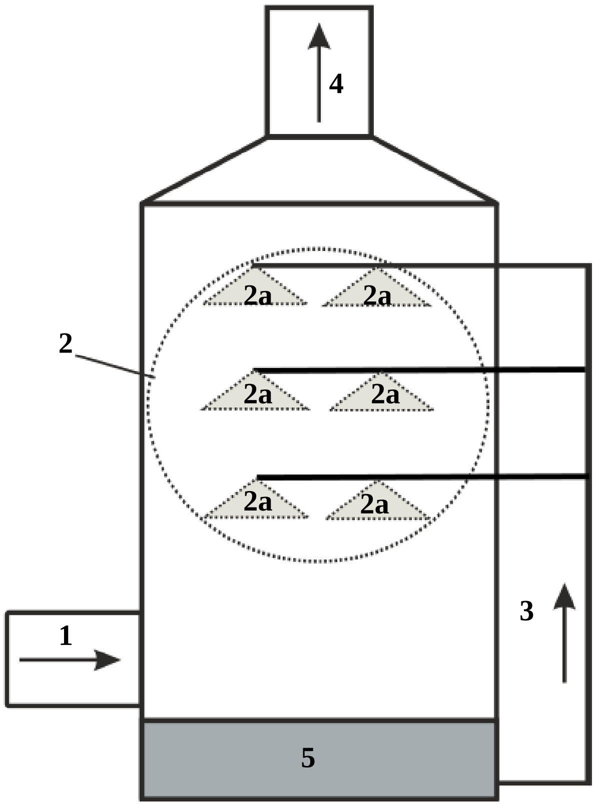
The aim of our work is to develop the numerical model of FGD process implemented into OpenFOAM computation software.

In work [7] we found that we need to build a scaled model of FGD reactor to verify the numerical results.

Aim of this work is to show velocity fields measured inside the experimental reactor in comparison with velocity fields gained from numerical simulation.

2 Experimental FGD reactor

Based on the cooperation with specialists from company *DIZ Bohemia* Ltd., the experimental FGD reactor was built. It represents a scaled model with typical proportions of FGD reactors, which are implemented in Czech coal power plants. The reactor also includes parts, which are designed to be modified based on our needs for future research.



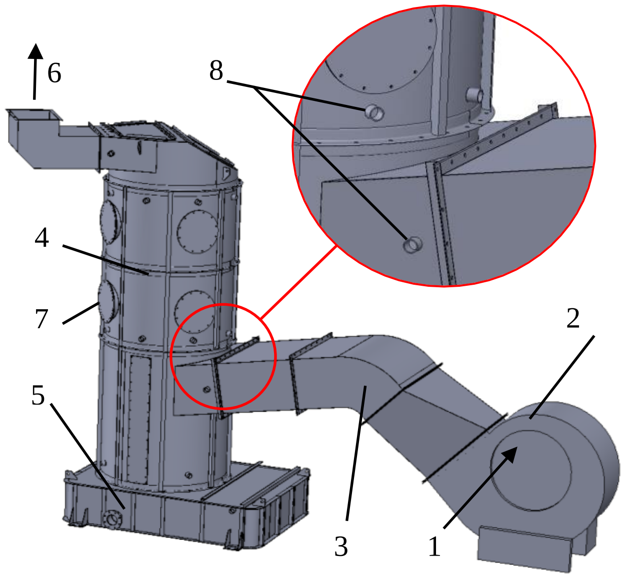
1 – inlet of flue gas
2 – spray zone
2a – spraying nozzles
3 – circulation of slurry
4 – outlet (cleaned gas)
5 – slurry tank

Fig. 1 Schematic sketch of FGD

A schematic sketch of the FGD reactor is shown in figure 1. In spray zone (2) limestone slurry is sprayed by spraying nozzles (2a) into the flow of flue gas. Sulfur dioxide is absorbed by limestone slurry droplets. The cleaned gas flows to the outlet and gypsum particles as a product of chemical reactions of limestone and SO2 fall down by gravity to the tank (5) in the bottom part of reactor.

The whole design of scaled model built by DIZ Bohemia is shown in figure 2 and figure 3. In principle it coincides with the scheme shown in figure 1. The main cylindrical part of the reactor is 5 m height and it´s diameter is 1.6 m. This reactor is currently designed for use air from surroundings instead of flue gas. Air is blown into the reactor by radial fan. Amount of air flown to the reactor could be controlled by changing rounds of the fan. This is allowed by frequency converter connected to the fan engine. Control element of the converter could be set from 0% to 100% of the engine power with step 10%.

For the experimental study of velocity fields inside the reactor, the spraying nozzles and devices for circulation of slurry have not been installed yet. Outlet of the reactor is opened to the surroundings.



1 – suction of air
2 – radial fan
3 – inlet pipe
4 – spray zone
5 – tank
6 – outlet of air
7 – transparent flange
8 – nipples

Fig. 2 Assembly of reactor and fan – CAD model

Fig. 3 Assembly of reactor and fan – photo

2.1 Experimental setup

Flow of gas inside the FGD reactor is very important for studying the desulfurization process. Because of quite huge dimensions of the scaled reactor (several meters in order) it is not possible to find the full velocity field inside.

2.1.1 Area of interest

The most important area for us is the spray zone. Through this area under the area for spray nozzles

there were installed several nipples in three different levels Z1, Z2, Z3 (see figure 4).
In each level 33 measuring points across the section were defined. Scheme of point´s position same for all levels is shown in figure 5. Labeling of points is based on numbers of point´s row and number of point in row (increasing in radial direction from the center).

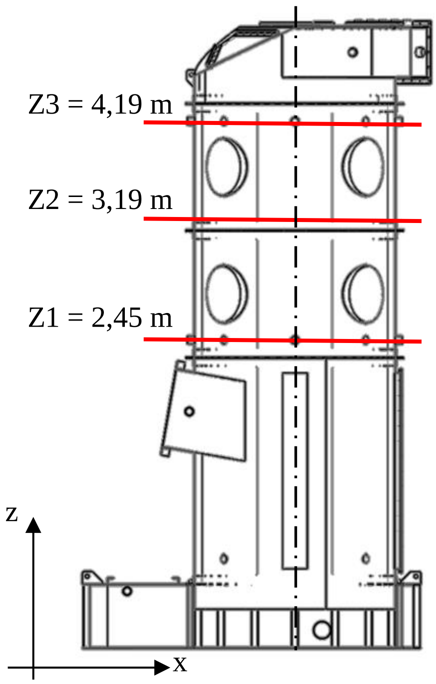


Fig. 4 – Reactor – Position of section planes

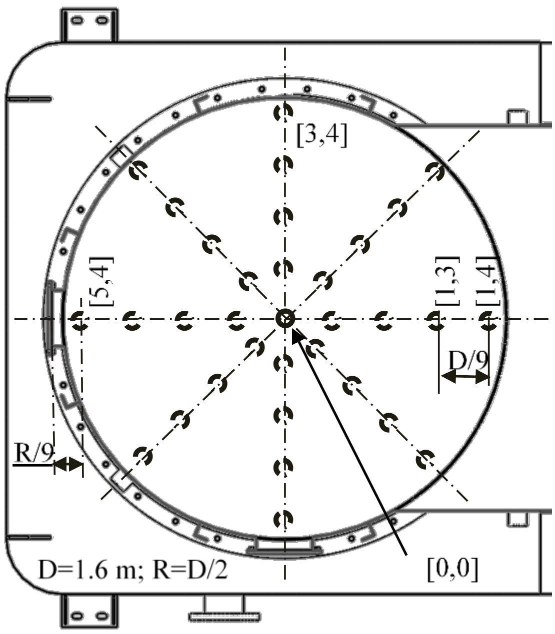
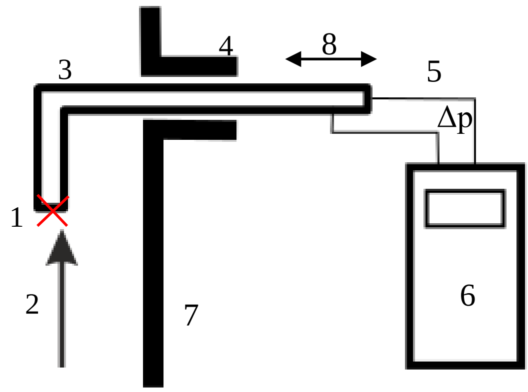


Fig. 5 – Measuring points – scheme and labeling

2.1.2 Measuring devices and setup

For measuring velocity the system with Prandtl probe was used. The whole device consists of the Prandtl probe with 1m length (see fig. 7), connecting silicone hoses and Testo 480 climate measurement meter and data logger. This device connected to the Prandtl probe measure differential pressure and convert that directly to the value of velocity shown on the display. Values are also stored to the inner memory of the device from which could be exported to PC for evaluation.

The function scheme of the apparatus described in the previous paragraph is shown in figure 6 . Prandtl probe is inserted through the concrete nipple and is traversed by hand only in horizontal direction to measure velocity in different points (see figure 5).



1 – measuring point
2 – velocity vector
3 – Prandtl probe
4 – nipple
5 – silicone hoses
6 – TESTO 480
7 – wall of reactor
8 – traversing direction

Fig. 6 – Block scheme of experimental setup

Fig. 7 – Prandtl probe with hoses and detail of measuring part

It should be mentioned that the end of Prandtl probe was oriented in –Z direction, so only the z-component of velocity was measured.

Measurement was provided two times by different flow regimes. All conditions were maintained at same values except the changes of flow rate (velocity) at the inlet area. Flow rate has been set to match the operating mode of the fan, which is represented by 40% (regime **1**) and 70% (regime **2**) of engine power.

2.1.3 Evaluation of measured values

The base principle of measuring velocity by Prandtl probe is to measure pressure difference between static and total pressure in the measuring point. Velocity should be evaluated by equation:

$$\mathrm{v} = \sqrt{\frac{\Delta p}{\rho}},$$

where v (m/s) is measured velocity,

Δp (Pa) is pressure difference between total and static pressure measured by Prandtl probe,

ρ (kg/m3) is gas density before Prandtl probe.

TESTO 480 logger used inner algorithm for evaluating directly the velocity from measured pressure difference. The algorithm is not known (especially evaluation of gas density), so there is no possibility to evaluate correctly the errors of measured values. The only known error is for measuring of pressure which is ±(0.3 Pa + 1% of measured value).

2.2 CFD

For CFD simulation of flow inside the FGD reactor the OpenFOAM software have been used. The goal of simulation is to numerically simulate the whole desulfurization process. Our investigation aimed to development of the suitable solver. It is a self - developed solver based on the coalChemistryFoam solver included in OpenFOAM.

2.2.1 Geometry and mesh

Geometry is based on the CAD model of experimental reactor. Before making computational mesh the whole geometry was simplified, some features like chamfers, small holes etc. were deleted and the volume extraction was performed to gain the volume of fluid inside the reactor (see figure 7). For these modifications ANSYS Space Claim Direct Modeler was used.

After that the computational meshes were created using Ansys Meshing utilitiy. For next study of influence of element size on results, four hexahedral meshes with different element size (quality) were created. All variants of meshes with element size are shown in figure 9. Boundary layer has not been modeled because of neglected dimensions of that in comparison with whole reactor dimensions. Meshes were exported from ANSYS Meshing to *.msh format and then imported to OpenFOAM environment by function „fluent3DMeshToFoam“.

2.2.2 Boundary conditions

Boundary conditions were set in agreement with the experiment. Inlet velocity was obtained from measurement, it is different for each regime - 1) 40%, 5 m/s, 2) 70%, 7,5 m/s. Other boundary conditions are shown in figure 8.

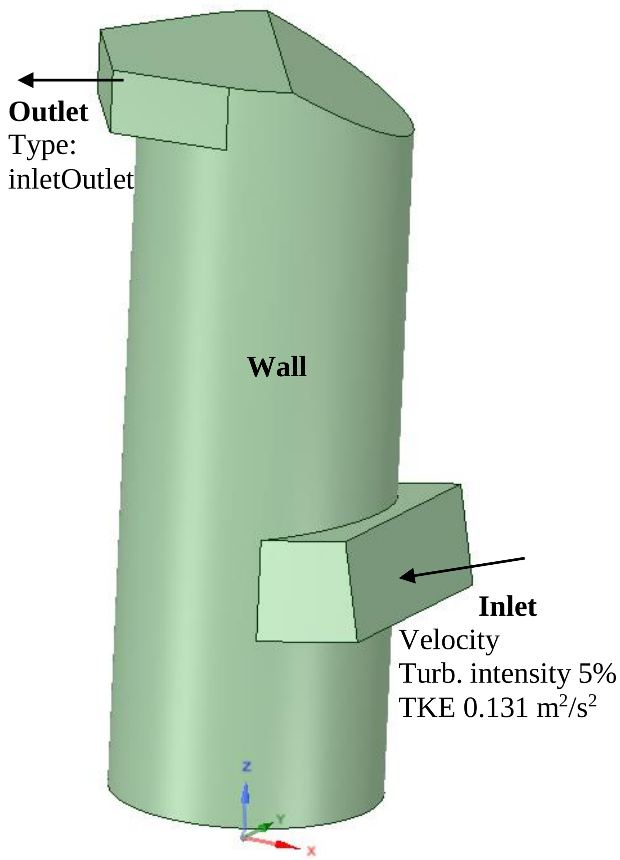


Fig. 8 – Fluid volume with boundaries

2.2.3 Calculation setup

Calculation is conceived as transient flow of air through the reactor. Air is considered as ideal gas with constant properties. K-ω SST model was used. Residuals of momentum and turbulent quantities were set to 10^{-3}, residuals of energy to 10^{-6}.

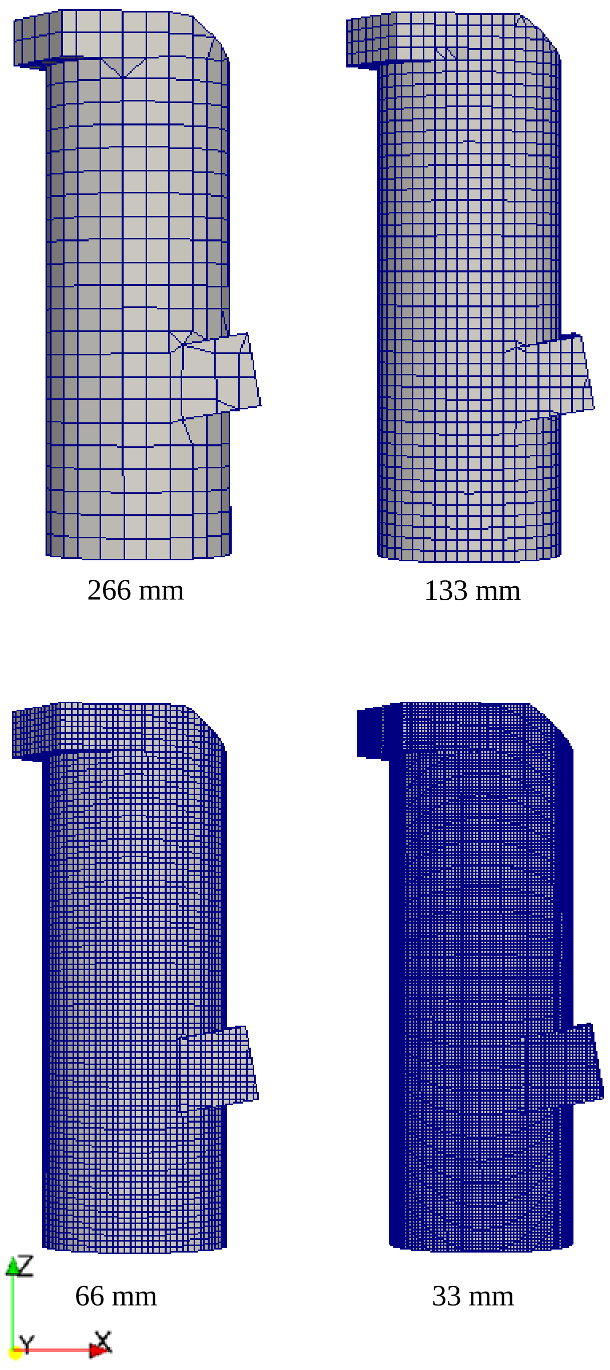


Fig. 9 – Computational meshes labeled by different element size

3 Results assessment

For both methods (experiments and CFD) the velocity fields in previously described areas were obtained.

3.1 Experimental data

From measurement with Prandtl probe the velocity fields in three levels of the scrubber totally in 99 measuring points were obtained. Data saved in data logger TESTO were analyzed. As was written in chapter 2.1.2, the velocity values in z- direction were measured. Results of measuring velocity for different flow regimes are shown in figure 10 and figure 11.

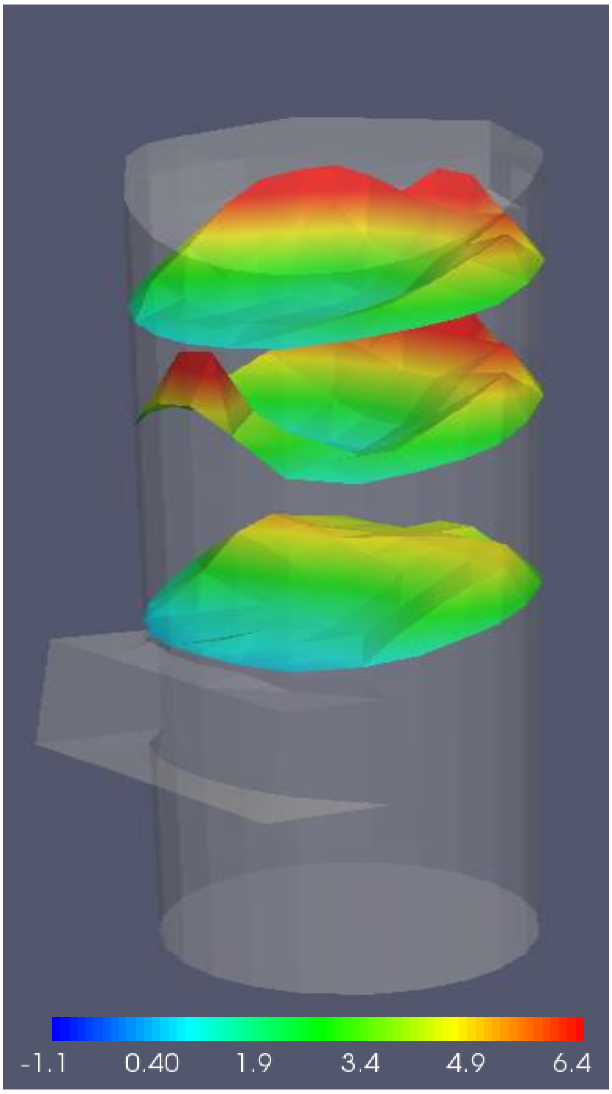


Fig. 10 – Velocity field (exp.) – regime 40% z-velocity (m/s)

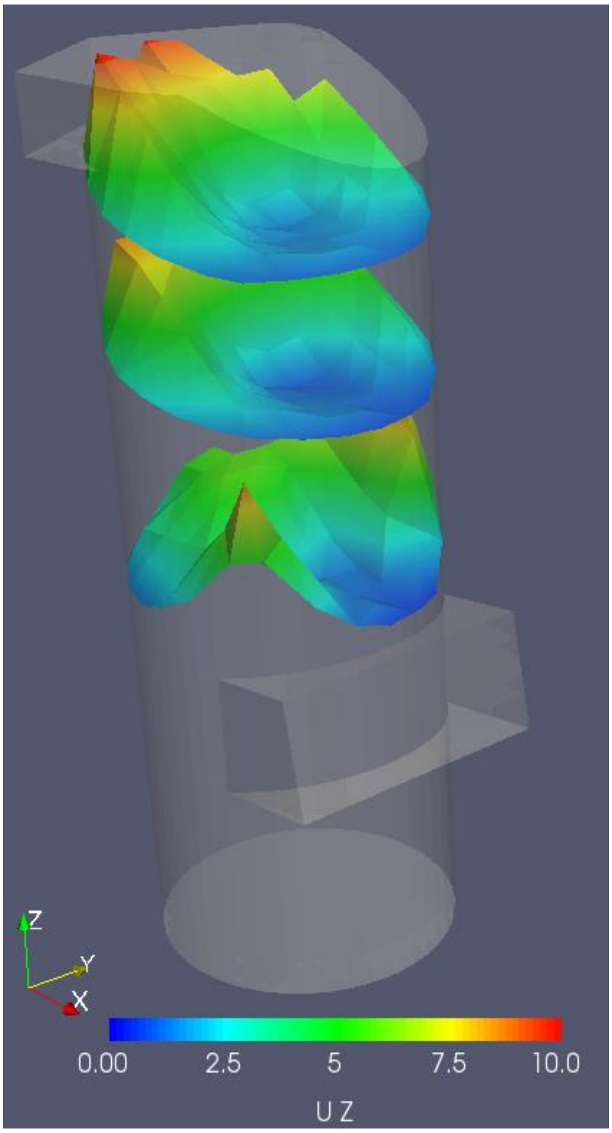


Fig. 11 – Velocity field (exp.) – regime 70% z-velocity (m/s)

3.2 CFD analysis

From CFD analysis were obtained velocity fields for different mesh types. Because only the z-direction velocities was obtained from the experiment, the results from CFD are shown in the same way, only velocities in z-direction (see figure 12). For better illustration of differences between experiment and CFD, results from experiment and CFD for the finest mesh are shown side by side in figure 13.

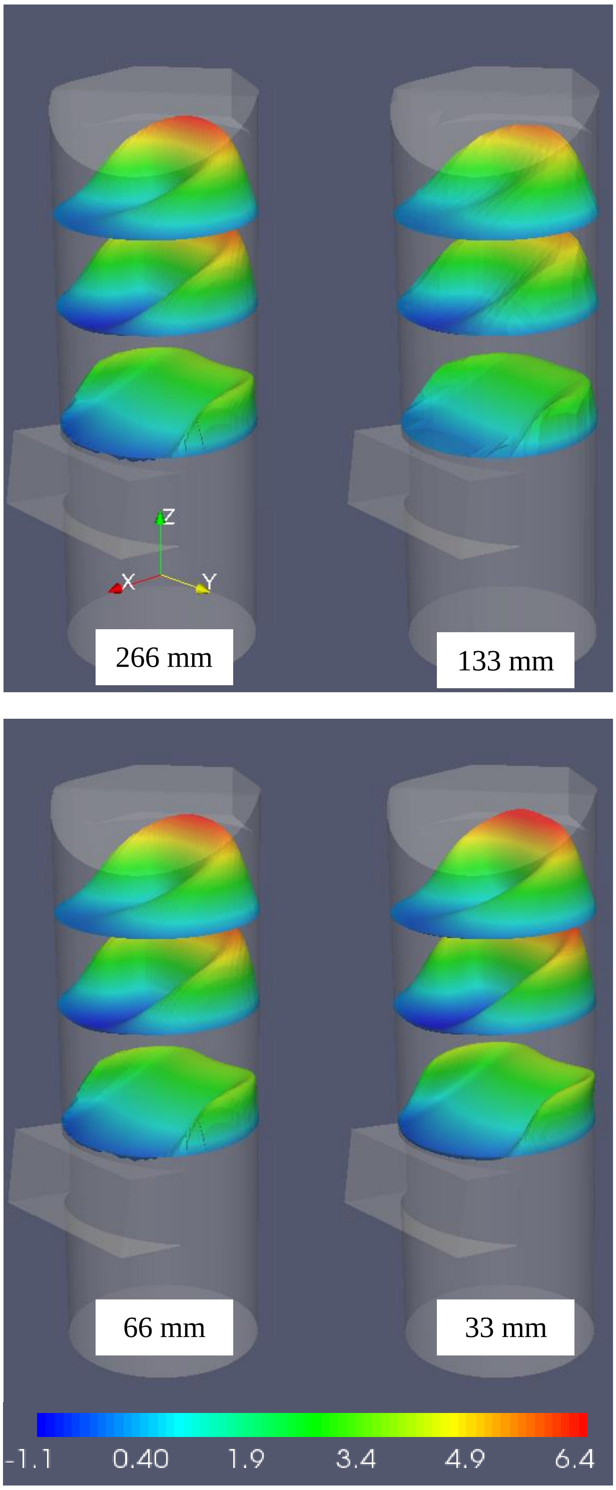


Fig. 12 – Velocity fields (CFD) – regime 40% z-velocity (m/s) for meshes with different element size

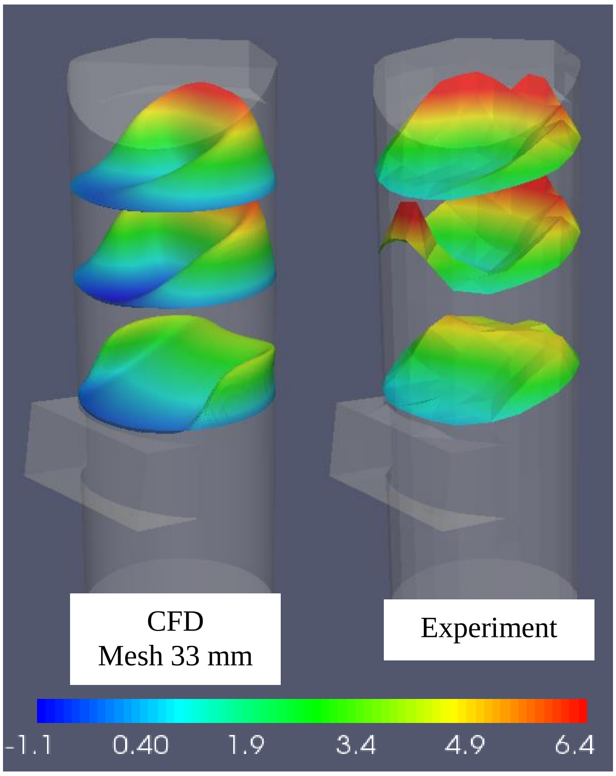


Fig. 13 – Velocity (comparison) – regime 40%, z-velocity (m/s)

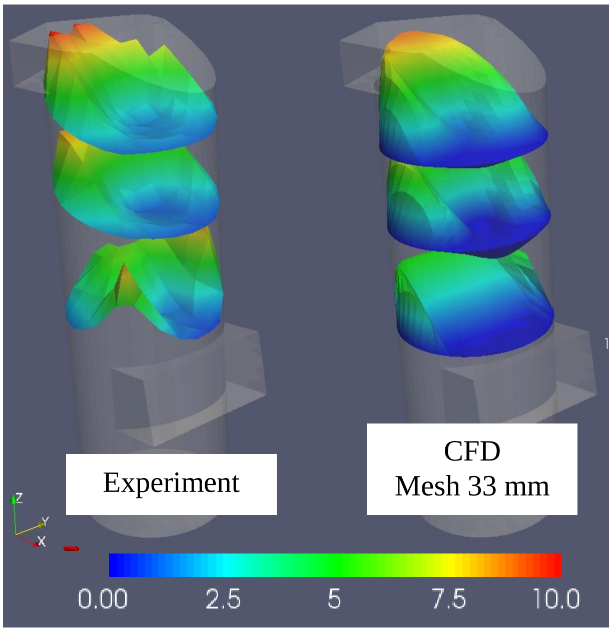


Fig. 14 – Velocity (comparison) – regime 70%, z-velocity (m/s)

4 Conclusion

In this work the investigation of velocity fields inside the FGD reactor by experimental and numerical methods was performed. FGD reactor assembly consist of the radial fan, the inlet channel, main part of the reactor (spray zone) and the outlet. The geometry of real reactor except the radial fan was transformed to the CFD model. 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 inside the spray zone) and available equipment. Experimental setup consists of Prandtl probe and TESTO 480 data logger. Main goal was to measure the velocity in specified points inside the reactor volume. Measuring points were divided to three groups (each with 33 points) which represents three different height level inside the spray zone of the reactor. Experiments were performed by two different regimes of flow – 40% and 70% of maximal power of the fan engine.

As we can see from the results published in figure 13 and figure 14, in the velocity profile in the middle level is several points with non-consistent values of velocity (higher) than in neighboring points. It is probably the measurement error. It could be caused by worse positioning of the probe, or because we have measured only the z- component of velocity and in these area the velocity vector may not be in accordance with this direction. In this case it is not possible to measure the turbulence effects, which could affect the flow around the probe. For the future the possibility of using device for traversing the probe should be considered. The second important thing is that we should measure more than one component of the velocity.

In numerical methods several types of computational meshes were used. From the obtained results we can say, that the element size did not affect the velocity fields in the spray zone a lot. For all images shown in figure 12 could be said that the flow is dominant on the right side, which is the side far from the inlet. That is because the flow which enters the reactor is forced to change the direction from x-direction to dominantly z-direction flow. The blue areas shown probably the backflow area. For the future these areas could be affected by adding the nozzles into the spray zone and if not, it will be good to think about changes of the geometry of inlet pipe to gain better distribution of air in the spray zone.

From figures 13 and 14 it could be said that very good accordance of results was reached for all cases.

Some differences are in the area with lower velocities, it could be caused by measuring device accuracy and by flow fluctuations, because the measured values were time averaged.

For future work it should be recomended to improve the quality of measured values, which could be done in two different ways. First one is to measure more points in each level, but it could be complicated because of precision of positioning the probe inside the reactor and take longer time. The second way to improve results is to neglect values which seems to be wrong significantly. After applying these recommendations then the CFD mesh with element size between 33mm and 66 mm is enough for making the numerical simulations with good agree with the real state (experiment).

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 2016. 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] Charlotte Brogren and Hans Thomas Karlsson. Modeling the absorption of SO_2 in a spray scrubber using the penetration theory, *Chemical Enqineering Science*, Vol. 52, No. 18, 1997, pp. 3085-3099.

[3] R. V. Bravo, R. F. Camacho, V. M. Moya, L. A. I. García, Desulphurization of SO_2–N_2 mixtures by limestone slurries, *Chemical Engineering Science*, Vol. 57, Issue 11, 2002, pp. 2047–2058.

[4] Antonio Gómez, Norberto Fueyo, Alfredo Tomás, Detailed modelling of a flue-gas desulfurisation plant, *Computers and Chemical Engineering* 31, 2007,pp. 1419–1431.

[5] Luca Marocco, Fabio Inzoli, Multiphase Euler–Lagrange CFD simulation applied to Wet Flue Gas Desulphurisation technology, *International Journal of Multiphase Flow* 35, 2009, pp. 185–194.

[6] Luca Marocco, Modeling of the fluid dynamics and SO_2 absorption in a gas–liquid reactor, *Chemical Engineering Journal* 162, 2010, pp. 217–226.

[7] 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.