Institute of Heat Engineering
WARSAW UNIVERSITY OF TECHNOLOGY

Open Access Journal

**Journal of Power Technologies 99 (2) (2019) 98–103**

`journal homepage:papers.itc.pw.edu.pl`


JOURNAL OF
POWER
TECHNOLOGIES
Vol. 91
2011
BULLETIN
OF INSTITUTE
OF HEAT ENGINEERING

# Hybrid techology of flue gas denitrification system. Part 1—Preliminary studies of flow turbulence and pressure drop in the elements of rotary air heater baskets

Andrzej Kwiczala[a,*], Robert Wejkowski[b], Katarzyna Jagodzińska[b]

[a] SBB ENERGY S.A., Łowicka 1, 45-324 Opole, Poland
[b] Silesian University of Technology, Institute of Power Engineering and Turbomachinery Konarskiego 20B, 44-100 Gliwice, Poland

## Abstract

The paper presents the results of physical and numerical tests of fluid flow through the filling of a rotary air heater (RAH). A laboratory-scale test bench was used to measure flow resistance across a fragment of a RAH. Seven types of RAH modules were tested---one steel and six ceramic (as catalyst carriers). The relationship between pressure drop and velocity (Re number) of flow was used to deduce the flow characteristics for each of the RAH modules tested.

Measurements carried out on the test bench were used to create a substitute mathematical model, which in the CFD code Ansys Fluent enables accurate mapping of pressure drop and velocity distribution full fit to the real flow conditions.

Numerical calculations were used to validate measurements for an alternative model, to create guidelines for the substitute model of the porous zone and to optimize application checking the correctness of created guidelines for simplified calculations. Flow simulations were performed for various turbulence models. Results were compared to the test-bench measurements to determine the best adjustment for this specific type of reverse flow inside the air duct.

This research is part of an ongoing research project: “Hybrid Technology of Flue Gas Denitrification System in Steam and Hot Water Boilers”. The aim of the project is to investigate the concept of using rotary air heater fillings as a carrier for catalytic coatings to reduce nitrogen oxides. In the further part of the research project, the replacement porous zone substitute models will make it possible to precisely calculate the entire RAH and will significantly reduce the calculation time as the basis for further project work.

*Keywords:* Hybrid DeNOx System; turbulence models; backward facing step; rotary air heater; CFD

## 1. Introduction

Due to the Industrial Emissions Directive (IED) and Best Available Techniques Conclusions (BAT) [1, 2] more attention is being paid to emissions of nitrogen oxides ($NO_x$). The energy industry is faced with the challenge of fulfilling exacting legal requirements by August 2021---reducing $NO_x$ emission to below 150 $mg/m_n^3$, referred to flue gases with a 6 % content of $O_2$, for existing power units producing more than 100 MW from fossil fuel combustion.

Until now, measures to reduce $NO_x$ emissions have included: Selective Catalytic Reduction (SCR) and Selective Non-Catalytic Reduction (SNCR), combined with primary methods (e.g., an Over-Fire Air system). However, the continual tightening up of regulations requires the development of more efficient technologies; for example, Hybrid Denitrification System (HDS) which combines SNCR with SCR.

Previous research with catalytic metal elements added to the RAH has shown promising $NO_x$ reduction results [3–5]. The combined SNCR and SCR method, with SCR located in the RAH, has been subject to research for roughly 30 years. Analysis performed within [3, 4] concerns mostly the chemistry of reactions and the temperature dependence of the catalyst selectiveness as well as its structure along the temperature profile, while in [5] full-scale tests of installation were performed in an existing Swedish power plant, resulting in 90% $NO_x$ reduction. That was obtained thanks to the $NO_x$ reduction process occurring in parallel with partial oxidation of ammonia at the catalyst surface located in the RAH. Oxidation means more reactant can be used in SNCR installations, which also improves $NO_x$ reduction at the SNCR stage. Unfortunately, erosion problems impacting metal sheets coated with catalyst have stopped these technologies from being installed in coal-fired boilers [6].

Commercially available HDS technologies consist of the SNCR stage followed by the SCR stage, which is usually an integral part of the boiler secondary duct. These systems

---

[*] Corresponding author
*Email address:* `a.kwiczala@sbbenergy.com` (Andrzej Kwiczala)

are characterized by high $NO_x$ reduction levels (up to 90%) with relatively small operating costs compared to a traditional SCR and/or SNCR, due to the lower ammonia requirement. When the SNCR and SCR are combined, the furnace $NH_3$ slip from the SNCR is used as the reductant in the downstream SCR. This also results in the simultaneous reduction of $NH_3$ emissions. Due to the SNCR NOx reduction, the SCR size can be reduced, lowering catalyst maintenance costs and flue gas flow resistance [7].

The greatest obstacle to implementing hybrid systems in existing boiler units is lack of space for the SCR system, even though its size has been significantly reduced. That became the basis of a project realized by the Consortium of SBB ENERGY S.A. and Institute of Power Engineering and Turbomachinery, Silesian University of Technology. On the basis of previous studies performed at laboratory and then semi-technical scale [8, 9] concerning the heat transfer and reduction efficiency of catalytic elements, the concept of placing them in a rotary air heater (RAH) was proposed to enable HDS to be installed economically in most existing boilers.

This paper presents the first stage of this investigation and concerns analysis of pressure drop across metal and catalytic elements in relation to their porosity and shape. Following an initial numerical analysis of the air flow distribution in the duct, the laboratory-scale research facility required a redesign. The previous laboratory-scale RAH system introduced measurement uncertainty due to radial and axial leaks and the influence of temperature changes of fluids on the dynamics of pressure changes.

The main aim of the study was to determine the equations expressing flow resistance, turbulence and pressure drop dependence on the type of catalytic input, based on the measurement results. These equations were then used to simplify a numerical analysis of $NO_x$ reduction and heat transfer in the full-scale RAH, thereby reducing the computational time while maintaining acceptable accuracy of the results.

## 2. Methodology

### 2.1. Laboratory-scale stand description

Pressure drop measurements were performed on the laboratory-scale stand which is illustrated in Figure (1). The stand is a pie-shape 12 m length duct (cross-sectional area = 0.066 $m^2$), supplied with air by a centrifugal fan (1). It is equipped with a flow straightener (3) before which the stub pipe is located (2) enabling the injection of hot gas from a gas burner into the channel. About 3 m after the flow straightener the pie-shaped catalytic or metal investigated elements (Figure (2)a) are mounted (5).

The measurement system is equipped with pressure and velocity devices (4 and 6). Before the insert placement the velocity, pressure and temperature of the fluid were measured and the pressure after the placement as well. All data was collected and remotely communicated to the main control and acquisition unit (Figure (2)b).

Measurement of the pressure drop at the facility enables precise characterization of any type of RAH inner elements, while avoiding measurement inaccuracy associated with radial and axial fluid leaks.

Exponent of the medium. The equation shows that the only parameter of engine performance is the compression ratio.

### 2.2. Numerical simulation

The second stage of the research was to conduct a series of numerical calculations using the FVM finite volume method in the ANSYS Fluent program. Numerical calculations were investigated in order to:

- validate the measurement results,
- determine an alternative model,
- create guidelines for the substitute model of the porous zone and optimization of its application,
- check the correctness of the guidelines for simplified calculations.

Flow simulations were performed for various turbulence models. Results were compared to measurements to determine the best adjustment for this specific type of reverse flow inside the air duct. The following turbulence models were evaluated:

- $k-\Omega$ / SST
- $k-\Omega$ / standard
- $k-\varepsilon$ / standard
- $k-\varepsilon$ / RNG
- $k-\varepsilon$ / realizable
- Reynolds stress

By considering the results for the various turbulence models, the overall correctness of the numerical model could be evaluated. Moreover, these comparisons allowed a correction to be introduced to match the numerical calculations to the measurements. A series of simulations were carried out to best match the numerical model to the results of measurements and to determine the influence of the mesh refinement. An example of the catalyst insert shape and its discretized model are shown in Figure (4)a and Figure (4)b.

## 3. Results

### 3.1. Results of physical investigation

During physical tests at this stage of the research project, 6 types of catalytic fillings (Element 1 - 6) and 1 steel (Element 0) were tested (Figure (5)).

The laboratory installation makes it possible to measure the relationship between fluid velocity and pressure drop for any flow and shape of insert. While the relationship between velocity and pressure drop can be calculated, it is better to use empirical results over CFD calculations as regards the corrugated metal insert. This is due to the advanced and complicated boundary layer (Figure (5)).

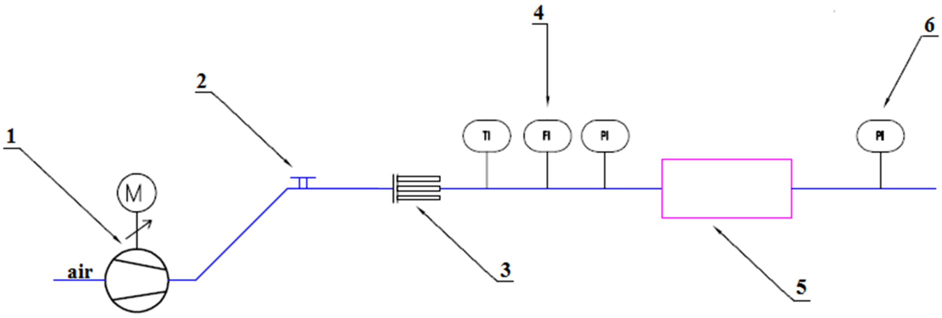

Figure 1: Scheme of laboratory stand: 1---fan, 2---stub pipe, 3---flow straightener, 4 and 6---measuring points, 5---investigated elements placement

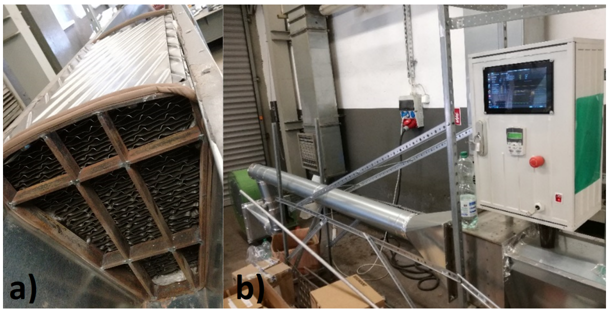

Figure 2: a) One of the exanimated steel elements, b) The main control and data acquisition unit

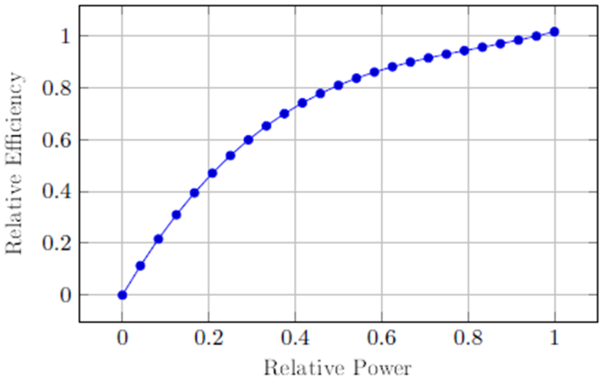

Figure 3: a) Engine performance according to [5]

*3.2. Optimization of lab-stand performance conditions*

The first measuring series provided for steel insert nr 0 gave a correlation between fluid velocity and pressure drop on Figure (6).

Analysis of the results shown on Figure (6)a indicates a slight analytical error, because the relationship between velocity and pressure drop does not converge to zero. Two possible sources for this non-realistic result are: 1. problems with the measurement devices and 2. problems with the design of the ductwork.

Potential error provided by measurement devices was quickly excluded by replacement with a newly calibrated device. To determine the reason for a non-physical result it was crucial to calculate flow at all ducts within the flow straightener (Figures: (7), (8), (9)).

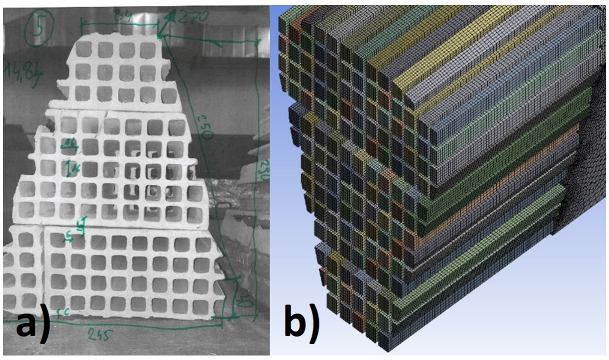

Figure 4: a) One of the investigated catalytic elements, b) Mesh generated for the catalytic element

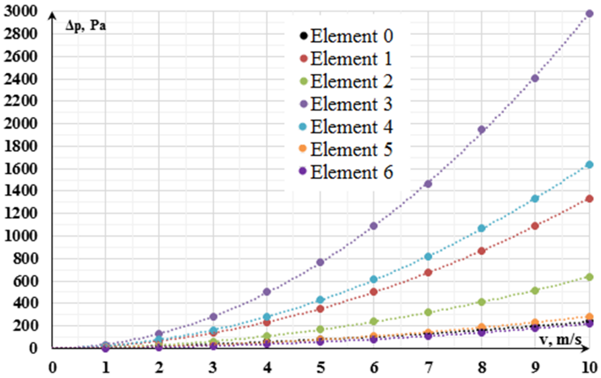

Figure 5: Pressure drop depending on fluid velocity for 7 types of fillings

The relationship shown on Figure (5) was used to create a porous zone model. The porous zone model is commonly used in CFD calculations due to its simplicity. A set of equations were created for each insert tested.

Superficial Velocity Porous Formulation is part of ANSYS Fluent code. This formulation allows users to calculate the pressure drop for calculated mixture velocities based on the volumetric flow rate in a specially declared porous region. Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. For the case of homogeneous porous media the viscous loss term is included on the first term on the righthand side of the Eq. (1),

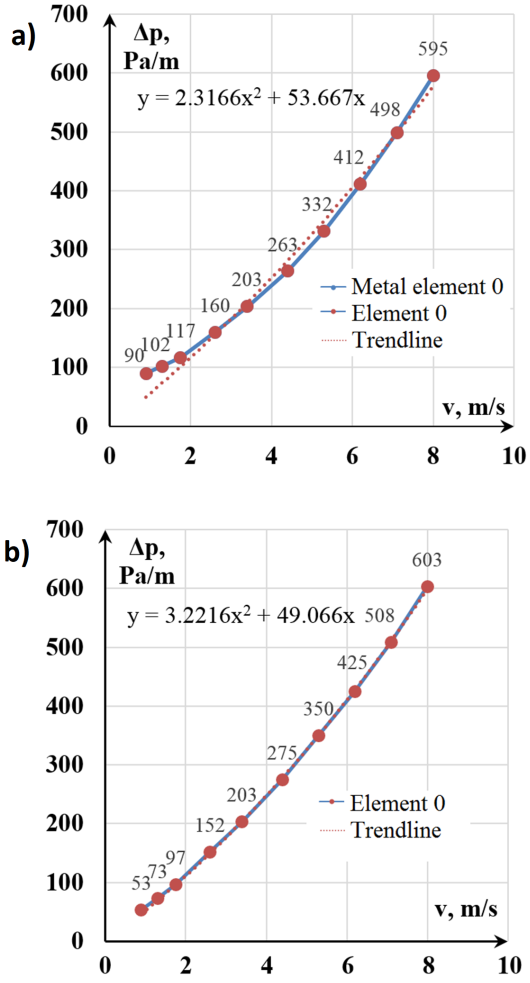

Figure 6: Pressure drop for element nr 0 (metal insert) depending on fluid velocity---before the re-design of the lab-stand

and internal loss term on the second term on the righthand side of Eq. (1).

$$
\begin{aligned}
S_i &= -\left(\frac{\mu}{\alpha} vi + C_2 \frac{1}{2} \rho \, |v| \, v_i\right) S_i \\
S_i &= -\left(\frac{\mu}{\alpha} vi + C_2 \frac{1}{2} \rho \, |v| \, v_i\right)
\end{aligned}
\tag{1}
$$

where: $S\,i$---source term for the i th (x, y, or z); $v_i$---direction velocity; $v$---magnitude of the velocity, alpha, $C_2$---internal resistance factor.

Alpha and $C_2$ are calculated from the laboratory measurements. In indicated cases of fluid flow in the catalytic layer, the above equation has to be implemented in the fluid main flow direction. In other directions the values are enlarged (e.g., 100 000) to stop or to reduce flow at real catalytic input.

All calculated cases presented show a vortex at measurement point 1 after the flow straightener. This result was common to all cases, independent of the turbulence model and mesh refinement.

Examples of results for tested insert nr 0 are shown on Figure (7) to Figure (10). Series of measurements were taken at Measurement Point 1 (on Figure (7)) as a representative place to investigate best correlation between the theoretical calculated model and real measurements. This measuring domain is located 50 mm from the flow straightener at the center of the tested duct.

Figure (10) shows a comparison between (i) the cross-section velocities obtained (Measuring Point 1) for different turbulence models, and (ii) data from real measurements. Values closest to those measured during the measurements at the station are shown in the model k - $\varepsilon$ / realizable. This model was adopted as a reference.

In tests [10, 11] for the parameter verifying the convergence of the model with the results of measurements or literature data, the reattachment flow length (RFL) was adopted---the distance between the backflow-step and the position. In [10] it was shown, on the basis of measurements in the wind tunnel, that for backflows characterized by low Re values (from 133 to 3693, of which Re = 380 can be regarded as laminar), there was relatively high convergence of the k - $\Omega$ / SST model with experimental results, with none of the k - $\varepsilon$ models correctly predicting RFL.

In [11] on the other hand, for Re equal to 9000, the k - $\Omega$ / SST model is also recommended, comparing the results to the literature data, with the k - $\varepsilon$ models being found to have very low convergence. For higher Re (13200) both model k - $\Omega$ / SST, and k - $\varepsilon$ / standard show satisfactory convergence [12]---model k - $\Omega$ / SST for RFL, while model k - $\varepsilon$ / standard for mean velocity profile. In [13], for Re = 38,000, the k - $\Omega$ / SST model was also recommended as being closer to the measurement results.

However, the station tests described in this paper indicate greater accuracy of the k - $\varepsilon$ / realizable model. The speed profiles obtained in numerical simulations were compared with data obtained during the measurements. This confirms the thesis, contained in [10x], that k - $\varepsilon$ models reproduce the velocity of the fluid quite accurately.

An additional result of the numerical calculations was finding the cause of measurement errors during the first series tests. These errors resulted from the design defect of the test stand.

Completed calculations and detailed measurements of individual inserts (catalytic and steel) were used to determine accurate and validated replacement models. In the subsequent part of the HDS research project, the replacement porous zone models will enable precise calculation of the entire regenerative rotary air heater RAH and will significantly reduce the calculation time as the basis for further project work.

## 4. Conclusion

The aim of the laboratory work and the numerical calculations based on them was to create a corrected and simplified model of mathematical catalytic contribution based on real flow measurements. Measurements carried out on the test bench enabled the creation of a substitute mathematical model, which in the computing environment of Ansys Fluent

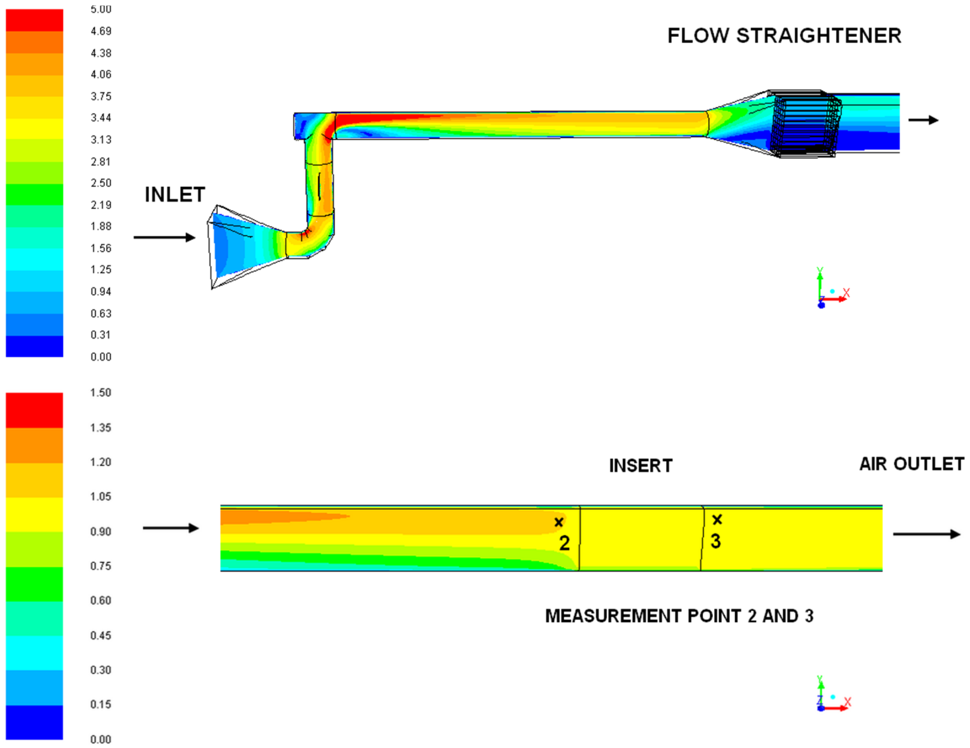

Figure 7: Results of numerical analysis for element nr 0---air velocity

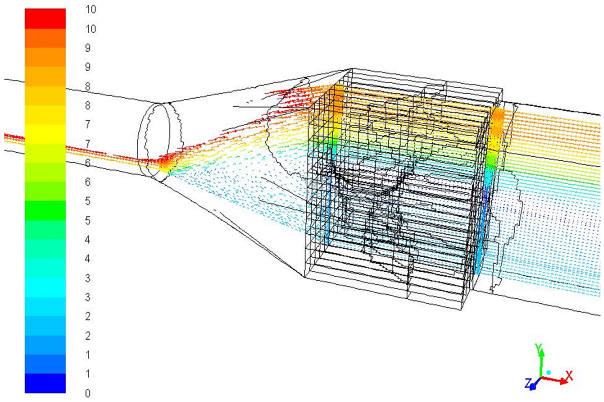

Figure 8: Velocity vectors in the flow straightener for Element 0

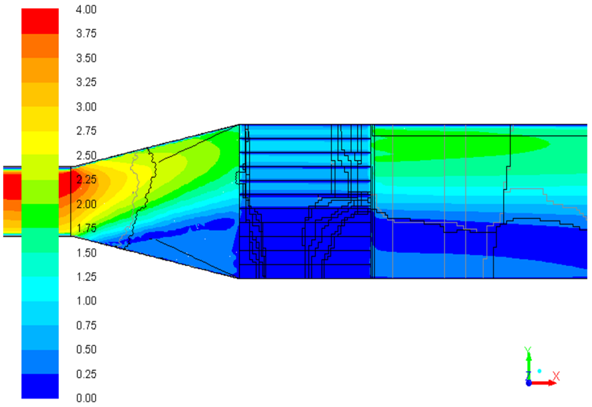

Figure 9: Velocity contours in flow straightener for Element 0

makes it possible to accurately map pressure drop and velocity distribution while maintaining the model's full fit to the actual flow conditions.

A substitute mathematical model is at the stage of being implemented into a full scale numerical model and is being tested and evaluated using measurements from RAH.

*Acknowledgements*

The research presented in this paper was performed as part of the Hybrid Technology of Flue Gas Denitrification System in Steam and Hot Water Boilers Project co-funded by the European Union, realized by the Consortium of SBB ENERGY S.A. and Institute of Power Engineering and Turbomachinery, Silesian University of Technology.

## References

[1] C. Directive, Directive 2010/75/eu of the european parliament and of the council, Off. J. Eur. Union L 334 (2010) 17–119.

[2] Decision (eu) 2017/1442 of 31 july 2017 establishing best available techniques (bat) conclusions.

[3] M. Kotter, H.-G. Lintz, T. Turek, Katalytische stickoxid-reduktion in einem rotierenden wärmeübertrager, Chemie Ingenieur Technik 64 (5) (1992) 446–448.

[4] K. Veser, Regenerativ-wärmetauscher in der umwelttechnik, betriebserfahrungen mit dem gasvorwärmer an nassentschwefelungsanlagen, mit pilot-anlagen denox-gerechter luft-und gasvorwärmer an entstickungsanlagen und anordnungskriterien für solche; entwicklungsstand des denox-luvo/denox-gavo, VGB Kraftwerkstechnik 66 (12) (1986) 1123–1130.

[5] Scandenox combined sncr and scr, http://www.scandenox.dk/kontakt1.

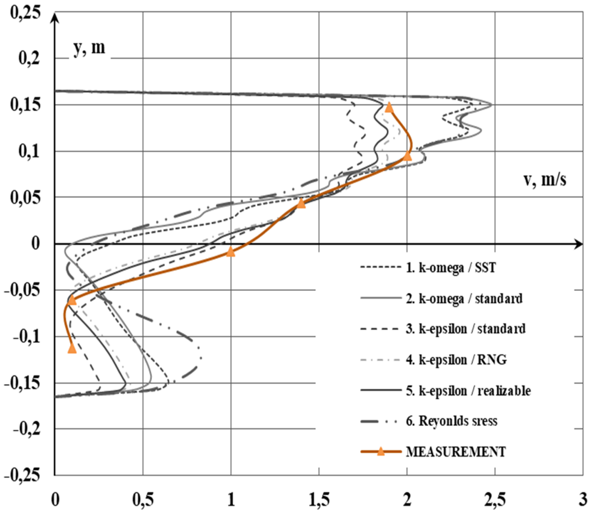

Figure 10: Horizontal velocity composition---comparison of measurement results to 3 different turbulence models

[6] M. J. FRANK, H. GUTBERLET, J. BRANDENSTEIN, Betrieb von rauchgas-entstickungsanlagen mit denox-katalysatoren: Ein übersichtsbeitrag über technologie und betriebspraxis von denox-katalysatoren, VGB powertech 86 (4) (2006) 72–77.

[7] B. K. Gullett, P. W. Groff, M. L. Lin, J. M. Chen, Nox removal with combined selective catalytic reduction and selective noncatalytic reduction: pilot-scale test results, Air & waste 44 (10) (1994) 1188–1194.

[8] R. Wejkowski, W. Wojnar, Selective catalytic reduction in a rotary air heater (rah-scr), Energy 145 (2018) 367–373.

[9] M. Kułażyński, M. Pronobis, A. Walewski, R. Wejkowski, W. Wojnar, Selektywna redukcja katalityczna (scr) tlenków azotu w regeneracyjnym obrotowym podgrzewaczu powietrza (selective catalytic reduction scr in rotary air heater - in polish), Rynek Energii (6) (2008) 82–87.

[10] F. A. Anwar-ul Haque, S. Yamada, S. R. Chaudhry, Assessment of turbulence models for turbulent flow over backward facing step, in: Proceedings of the World Congress on Engineering, Vol. 2, 2007, pp. 2–7.

[11] P. P. Araujo, A. L. T. Rezende, Comparison of turbulence models in the flow over a backward facing step, International Journal of Engineering Research and Science 3 (1).

[12] D. Jehad, G. Hashim, A. Zarzoor, C. N. Azwadi, Numerical study of turbulent flow over backward-facing step with different turbulence models, Journal of Advanced Research Design 4 (1) (2015) 20–27.

[13] M. K. Isman, Investigation of inlet effects on backward-facing step flow prediction, Transactions of the Canadian Society for Mechanical Engineering 40 (3) (2016) 317–329.