CHEMICAL ENGINEERING TRANSACTIONS

VOL. 39, 2014

A publication of AIDIC
The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Peng Yen Liew, Jun Yow Yong
Copyright © 2014, AIDIC Servizi S.r.l.,
ISBN 978-88-95608-30-3; **ISSN** 2283-9216 DOI: 10.3303/CET1439152

Numerical Flow Field Prediction Downstream of Axial Guide Vane Swirl Generator

Jiří Vondál*, Jiří Hájek

[a]Institute of Process and Environmental Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, Brno, Czech Republic
vondal@fme.vutbr.cz

This paper utilizes current state of art in CFD simulations suitable for engineering application. The object of interest is an axial guide vane swirl generator often found in process burners. The aim is to validate available turbulence models suitable for flow field prediction in industrial combustors. The grid independence study on four grids supports choice of used grid. Influence of the outlet shape on the flow field inside the chamber and especially on recirculation zone shape is proved. Last, the comparison of the three turbulence models is provided. The most accurate is identified the Reynolds Stress turbulence model. Predicted velocity fields are compared with the measured data obtained from the literature. CFD predictions of all the turbulence models underestimate peak axial velocity and overestimate velocity in central recirculation zone on the chamber axis.

1. Introduction

Axial guide vane swirl generator is widely used in process furnaces as a flame holder for non-premixed gas and oil combustion. It generates swirling flow with recirculation zones which provide reactants with sufficient time to react and form products. Often it is introduced to the stream with higher momentum – typically the air stream. Swirler (swirl generator, flame holder) is a key burner design element that significantly influences the flow pattern in combustion chambers. The purpose of a swirler is to convert part of axial momentum of the flow to a tangential momentum. This type of flow is required in many burners (such as in cement industry (Mikulčić et al., 2012) or oxygen-enhanced (Belohradsky and Skryja, 2013)) and also in other applications. In burners, swirling flow is important for flame stabilization and as a primary measure to decrease NO_x emissions. Swirler generates a low pressure zone in the flame core, which for confined flames leads to two recirculation zones that dilute reacting fuel and air by inert combustion products (flue gas). This desirable process can in cases with increased tangential momentum lead to unstable oscillations called precessing vortex core (PVC). The phenomenon may cause undesirable acoustic emissions and in extreme cases it may even destabilize the combustion process. Swirling flows and PVC are both the subject of a significant research activity as documented e.g. by Syred (2006).
For the quantitative description of the relative strength of tangential momentum is used a nondimensional swirl number (S), which is defined as the ratio of axial flux of tangential momentum over axial flux of axial momentum (Gupta et al., 1984). In most cases published works provide values of swirl number calculated on the basis of swirl generator geometry as proposed by (Claypole and Syred, 1981). The geometric swirl number must however be used thoughtfully, as it is suitable only for specific swirler geometries. In spite of this, number of authors provides geometry-based swirl number as the only information about swirl intensity, e.g. Cortés and Gil (2007). Swirl number calculated from measured velocity profiles is encountered less frequently in the literature, e.g. in Coghe et al. (2004), but it is essential in the case of this work, as measured data are necessary for the validation of predictions.
There are two basic types of swirling flow – low swirl flows typically with swirl number lower than 0.6 and strongly swirling flows with higher value of swirl number. Precessing vortex core is encountered mainly in the case of strong swirl flows, with the exception of flow through sudden expansion (which is the case also

Please cite this article as: Vondál J., Hájek J., 2014, Numerical flow field prediction downstream of axial guide vane swirl generator, Chemical Engineering Transactions, 39, 907-912 DOI:10.3303/CET1439152

in most burners), where PVC has been observed even with lower swirl numbers (Ranga Dinesh and Kirkpatrick, 2009).

When we face the task to predict flame properties inside combustion chamber, the swirl generator i.e. inlet velocity profile has to be taken into account. In CFD simulation of combustion the inlet velocity profile has to be provided. One option is to physically measure the profile and then provide it as a boundary condition on air inlet. However, it is often impossible to measure velocity profile downstream of the swirler when burner is in operation. There has been even cast doubt on this approach by Wegner et al. (2004). Other procedure, favoured in practice, is to provide velocity profile upstream of swirl generator and predict the flow field by numerical analysis. This procedure has not received broad attention of scientific community yet even though it has been recommended in Wegner et al. (2004). Published work has not focused on verification of predicted flow field or authors utilize different type of swirl generator.

Such approach leads to increased computational requirements, which is however becoming acceptable. In the case of Large Eddy Simulations (LES) which have increased demands on the quality of data for inlet boundary conditions, such treatment is almost a necessity.

LES is already quite widely used, but application to more complex geometries is still very difficult. The reason, due to which LES of most practically relevant swirling nonpremixed flames is not feasible, is the excessive number of computational cells in a discretized model. This is caused by the great span of scales inherent to practical fired heaters, where gas nozzles are few millimetres in diameter, while combustion chambers have dimensions in meters or tens of meters. Due to the necessity to use uniform mesh cell size for the whole computational model in LES, such applications currently may only be simulated by moment closure turbulence models.

It could seem that it is late to validate RANS models due to the rising popularity of more advanced methods (LES, DES) but they still dominate in practical industrial applications (Pallarés et al., 2009). In the area of swirling nonpremixed combustion, several turbulence models are used that have been validated with relative success by measured velocity profiles. Specifically, it has been shown that the RNG k-ε turbulence model is acceptable for the prediction of low-swirl flows (up to 0.6), where it performs even better than the RSM model (Escue and Cui, 2010). For the modeling of higher-swirl flows, it has been shown that solving the anisotropic Reynolds stresses directly by RSM is a more fitting option. The work (Wegner et al., 2004) even shows that unsteady RANS model based on RSM is applicable for the description of the precessing vortex core. The authors also report, that for a high-quality prediction it is necessary to include swirler in the simulated domain, otherwise velocity profiles may be deformed. Previous work with slightly different geometry of swirler in water media was published by Vondál and Hájek (2011) and its application to the large scale combustion facility was made in Vondál and Hájek (2012).

2. Experimental data

Computational study in this paper is based on experimental measurement by detail description may be found in Lilley (1985) who used axial eight-guide vanes swirl generator. Measurement was taken by five-hole pitot tube. Axial and tangential velocity field was measured. Inlet bulk velocity was 10 m/s and Reynolds number 120,000 with an inlet diameter 150 mm and sudden expansion downstream of swirler into a diameter 300 mm. Measurement was performed in six axial positions from x/D=0 up to x/D=2.5.

3. Modelling

3.1 Turbulence model

Three turbulence models were compared to identify the most appropriate, namely the realizable k-ε (Shih et al., 1995), Reynolds Stress Model - RSM (Launder et al., 1975) and the Improved Delayed Detached Eddy Simulation - IDDES (Shur et al., 2008).

Realizable k- ε model was chosen for its good rate of accuracy over computational demand. It is able to satisfy some mathematical constraints on Reynolds stresses. Its modification is in alternative formulation of turbulent viscosity and modified transfer equation for dissipation rate ε with respect to the standard k-ε model.

Reynolds Stress Model in variant of Stress Omega Model for pressure-strain term is based on omega equation and LRR model according to (Wilcox, 2006). The closure coefficients are the same as for k-ω but the two additional closure coefficients.

The Improved Delayed Detached Eddy Simulation model is a hybrid of RANS SST k-ω model and advanced Large Eddy Simulation (LES). It is improved version of Detached Eddy Simulation (DES) which provides e.g. shielding against grid induced separation similar to DDES. IDDES is based on modification of k-equation to obtain information on local grid spacing. When grid is fine enough it allows switching to LES

mode, otherwise it keeps in SST k-ω mode. The attempt is made to keep the boundary layer in steady RANS mode and resolve it with SST model even on the fine grid.
Scalable wall function had been employed to model near wall flows where appropriate. It suppresses deterioration effect of standard wall function (Launder et al., 1975) under refined grid near the wall ($y^*<11$). Otherwise it is equivalent to the standard wall function.

3.2 Discretization

The second order upwind discretization scheme for momentum equation, SIMPLEC method for pressure-velocity coupling and the first order upwind discretization scheme for other turbulence quantities were used. Transient simulation in first order implicit formulation was run to be able to capture unsteady flow effects. Presented results in all figures are averaged velocity fields from unsteady calculations.
Hexahedral cells in whole domain were used for all grids. Four grid variants were created and the grid independence study was performed. The grids had following parameters: a base grid with 1.92×10^6 cells, a grid with 1.99×10^6 cells, a 7×10^6 cells grid and a 10×10^6 cells grid.

3.3 Geometry

Commercial CFD code ANSYS Fluent was utilized for all the simulations. Geometry was based on experimental setup of (Lilley, 1985).
Geometry includes 1×D combustion air duct upstream of swirl generator, axial guide vane swirl generator and 4×D of the duct downstream. Guide vanes are modelled as zero-thickness walls.

4. Results

4.1 Grid independence study

Obtained results of grid independence study are displayed in **Error! Reference source not found.**. The turbulence model chosen for this study was realizable k-ε which is the least demanding model on computational power. We can see that there are only negligible differences in the outer region close to the wall with increasing tendency toward the axis. On the central axis the trend is the same for all the grids – the prediction shows backflow and correctly creates central recirculation zone. Since all the grids perform similarly, the economical 1.99 million grid was chosen for all subsequent calculations.

4.2 Effect of outlet shape

Next study aimed at the comparison of outlet shape influence on the velocity field inside the chamber. One case had fully opened outlet of the same diameter as the chamber itself (300 mm). The second case had partially blocked outlet with inside diameter of 150 mm. Both were simulated with realizable k-ε turbulence model. Resulting flow fields are shown in **Error! Reference source not found.**. Measured data on **Error! Reference source not found.** are for fully opened outlet which corresponds to the predicted values with large central recirculation zone reaching up to the end of measured section at x/D=2.5. Difference between measured and predicted profiles can be seen in width of recirculation zone. Predicted recirculation zone spreads more toward the wall and therefore has larger volume. On the other side the partially blocked outlet significantly changes flow field and creates smaller recirculation zone extending to the x/D=0.5. Central axis is not affected by backflow which is in contrary to the case with the open outlet. Results of the closed outlet are in agreement with the measured data (Lilley, 1985) on the chamber with contraction nozzle at the position x/D=2. The partially blocked outlet had tendency to create an additional outer recirculation zone near the wall of the chamber. Those two cases are visually compared on the **Error! Reference source not found.** where central recirculation zone is displayed. Those are iso-surfaces of zero axial velocity coloured by velocity magnitude.

4.3 Turbulence model validation

Three turbulence models were compared as described in Chapter 3.1. All the models were validated with the measured data – see **Error! Reference source not found.** and **Error! Reference source not found.**. Trends of the velocity profile are captured well by predicted data, however absolute values may differ. The most problematic part is on the central axis of the chamber where simulation predicts higher backflow than was measured. The smallest deviation on the central axis produced RSM turbulence model. Surprisingly the most advanced IDDES model did not performed well. It over predicts central backflow and is even outperformed by the realizable k-ε model. In tangential profiles (**Error! Reference source not found.**) we can see significant deviation in the first position – x/D=0. It is however caused by grid resolution. There exists strong gradient between zero velocity at the wall and tangential velocity at the adjacent cell. Since the predicted velocity profiles were exported from cell centres it displays non-zero velocity at the wall. We can see that next profile at the position x/D=0.5 corresponds well with measured data.

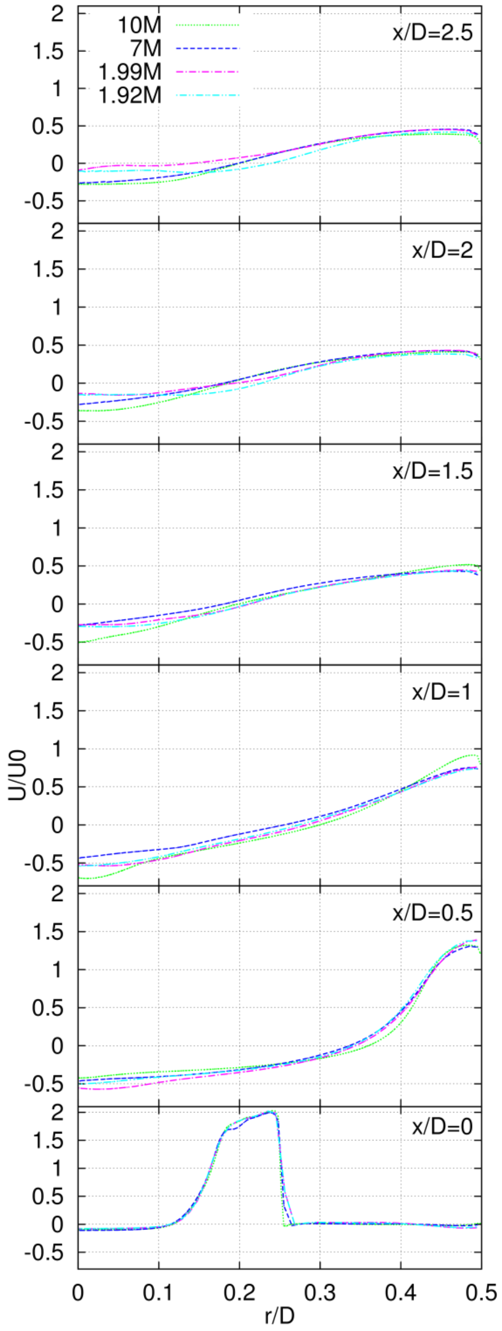

Figure 1: Velocity profiles of the grid independence study

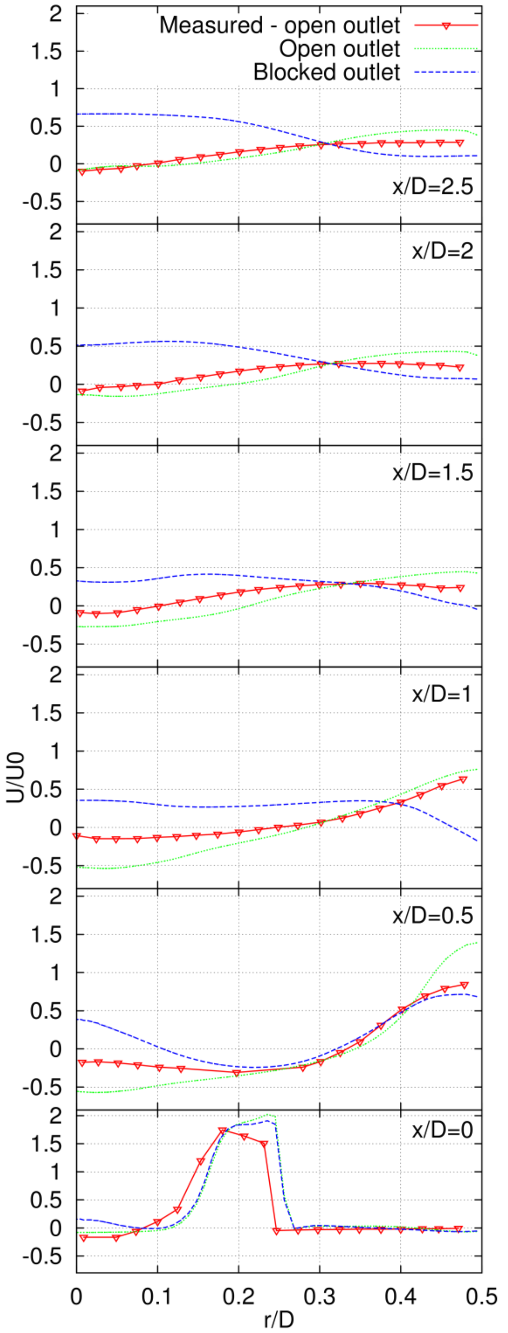

Figure 2: Influence of the outlet shape on the velocity profiles inside the chamber

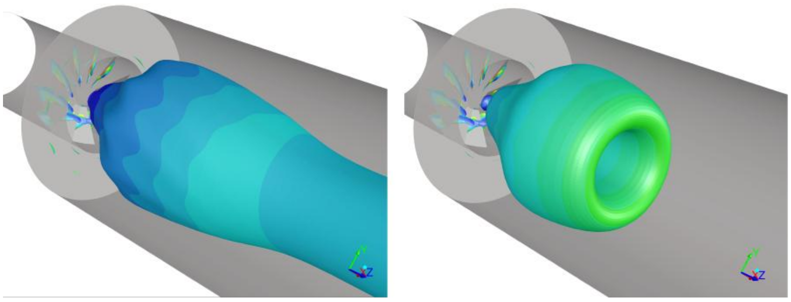

Figure 3: Internal recirculation zone of a) fully open outlet; b) partially blocked outlet

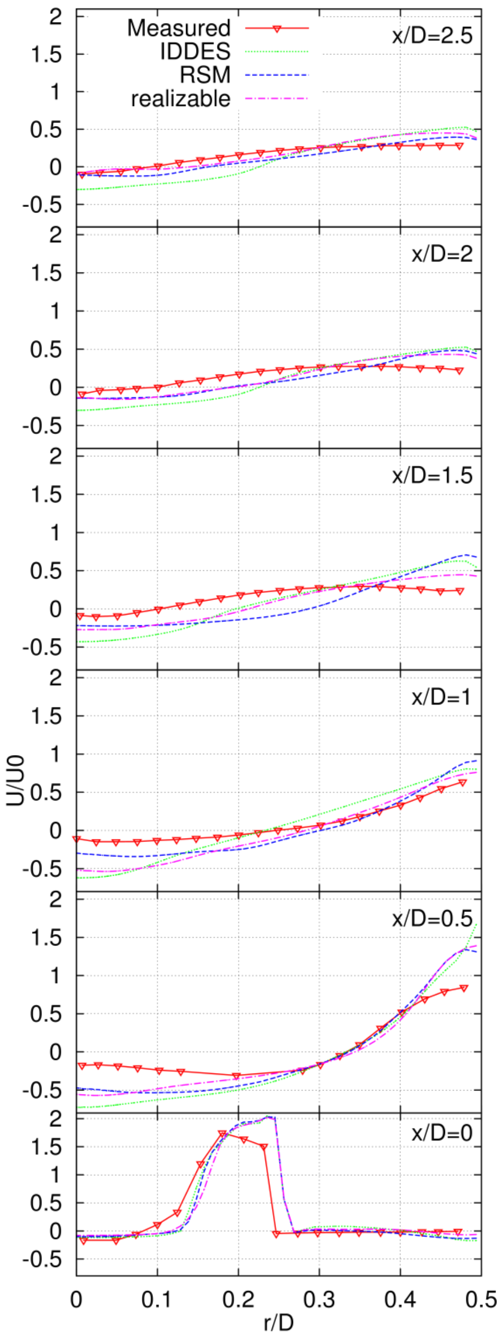

Figure 4: Axial velocity profiles of the three turbulence models validated with the measured data

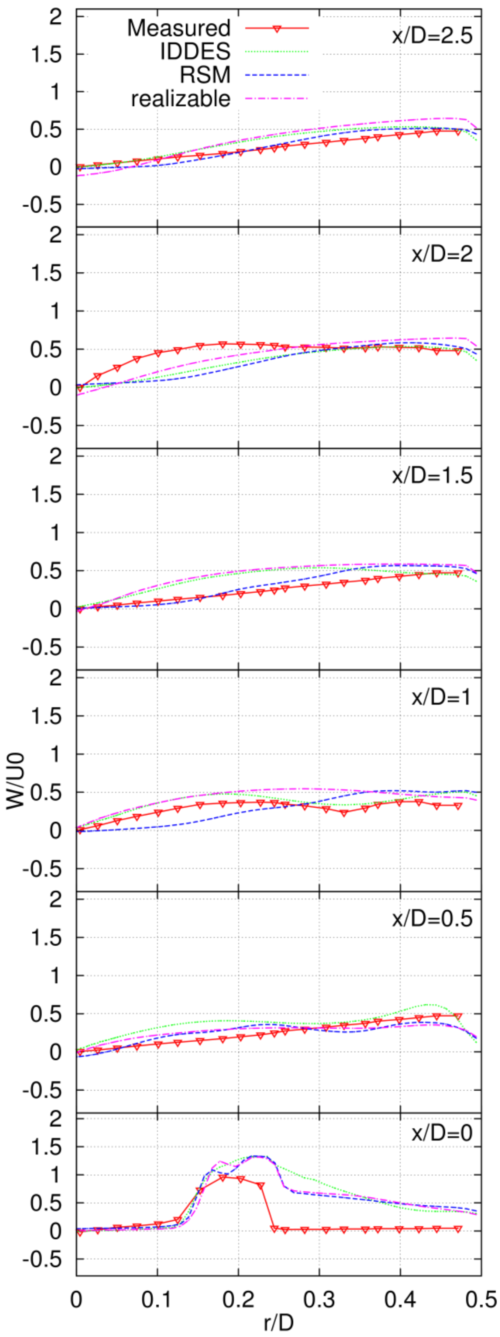

Figure 5: Tangential velocity profiles predicted by the three turbulence models validated with the measured data

5. Conclusions

RANS models are valuable and efficient tool for a CFD simulation of industrial scale combustors. Even though the new promising procedures such as LES emerged they are still far more computationally demanding beyond capability of many users. This paper proved that the trends of the velocity profiles can be captured by efficient RANS models even in complex flows such as swirling flows generated by the axial guide vane swirler. Validation pointed out several drawbacks of RANS predictions such as overprediction of backflow velocity near the swirler, however general trends are captured. The most accurate was identified RSM turbulence model which can account for flow anisotropy and provide velocity profiles closest to measured data. Results provide clear answer for research and industry about performance of selected turbulence models under swirling flows. It gives provides guidance in swirling flow modeling for broad audience and variety of industrial applications.

Acknowledgement

The authors gratefully acknowledge financial support within the project NETME CENTRE PLUS (LO1202) which was co-funded by the Ministry of Education, Youth and Sports within the support programme „National Sustainability Programme I. This work was also supported by the project CZ.1.07/2.3.00/30.0039 of Brno University of Technology.

References

Belohradsky P., Skryja P., 2013, Experimental study on the influence of oxygen fraction in the combustion air on the combustion characteristics, Chemical Engineering Transactions, 35, 1147–1152.

Claypole T. C., Syred N., 1981, The effect of swirl burner aerodynamics on NOx formation, Symposium (International) on Combustion, 18(1), 81–89.

Coghe A., Solero G., Scribano G., 2004, Recirculation phenomena in a natural gas swirl combustor, Experimental Thermal and Fluid Science, 28(7), 709–714.

Cortés C., Gil A., 2007, Modeling the gas and particle flow inside cyclone separators, Progress in Energy and Combustion Science, 33(5), 409–452.

Escue A., Cui J., 2010, Comparison of turbulence models in simulating swirling pipe flows, Applied Mathematical Modelling, 34(10), 2840–2849.

Gupta A.K., Lilley D.G., Syred N., 1984, Swirl Flows, Abacus Press, Tunbridge, Wells, UK.

Launder B.E., Reece G.J., Rodi W., 1975, Progress in the Development of a Reynolds-Stress Turbulence Closure, Journal of Fluid Mechanics, 68(3), 537-566.

Lilley D.G., 1985, Investigations of Flowfields Found in Typical Combustor Geometries, National Aeronautics and Space Administration, Scientific and Technical Information Branch. <ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/19850012857_1985012857.pdf>, accessed 27.6.2013

Mikulčić H., Von Berg E., Vujanović M., Priesching P., Tatschl R., Duić N., 2012, CFD analysis of a cement calciner for a cleaner cement production, Chemical Engineering Transactions, 29, 1513–1518.

Pallarés J., Gil A., Cortés C., Herce C., 2009, Numerical study of co-firing coal and Cynara cardunculus in a 350 MWe utility boiler, Fuel Processing Technology, 90(10), 1207–1213.

Ranga Dinesh K.K.J., Kirkpatrick M.P., 2009, Study of jet precession, recirculation and vortex breakdown in turbulent swirling jets using LES, Computers & Fluids, 38(6), 1232–1242.

Shih T.-H., Liou W. W., Shabbir A., Yang Z., Zhu J., 1995, A new k-[epsilon] eddy viscosity model for high reynolds number turbulent flows, Computers & Fluids, 24(3), 227–238.

Shur M.L., Spalart P.R., Strelets M.K., Travin A.K., 2008, A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities, International Journal of Heat and Fluid Flow, 29(6), 1638–1649.

Syred N., 2006, A review of oscillation mechanisms and the role of the precessing vortex core (PVC) in swirl combustion systems, Progress in Energy and Combustion Science, 32(2), 93–161.

Vondál J., Hájek J., 2011, Prediction of flow through swirl generator and validation by measured data, Journal of Physics: Conference Series, 318(2), 022026.

Vondál J., Hájek J., 2012, Swirling flow prediction in model combustor with axial guide vane swirler, Chemical Engineering Transactions, 29, 1069–1074.

Wegner B., Maltsev A., Schneider C., Sadiki A., Dreizler A., Janicka J., 2004, Assessment of unsteady RANS in predicting swirl flow instability based on LES and experiments, International Journal of Heat and Fluid Flow, 25(3), 528–536.

Wilcox D.C., 2006, Turbulence Modeling for CFD, 3rd ed., Dcw Industries, Incorporated, La Canada, California, USA.