BRNO UNIVERSITY OF TECHNOLOGY
Faculty of Mechanical Engineering
Institute of Process and Environmental Engineering

Ing. Jiří Vondál

COMPUTATIONAL MODELING OF TURBULENT SWIRLING DIFFUSION FLAMES

POČÍTAČOVÉ MODELOVÁNÍ TURBULENTNÍHO VÍŘIVÉHO DIFÚZNÍHO SPALOVÁNÍ

Short version of PhD Thesis

Branch: Design and Process Engineering
Supervisor: doc. Ing. Jiří Hájek, Ph.D.
External examiner: prof. Ing. Jiří Klemeš, CSc.
Ing. Antonín Tuček, CSc.
Date of defence: 1. 6. 2012

Klíčová slova

CFD, Spalování, vířivé proudění, turbulence, přestup tepla, radiace, experiment.

Keywords

CFD, combustion, swirling flow, turbulence, heat transfer, radiation, experiment.

Místo uložení dizertační práce

Ústav procesního a ekologického inženýrství, FSI, VUT v Brně

ISBN 978-80-214-4604-5
ISSN 1213-4198

Contents

1. INTRODUCTION

Emissions are studied namely to ensure compliance with legislative regulations (e.g. [1]). One of the common emission reduction techniques is swirling combustion. Swirl-stabilised flames are very popular, especially in the so-called power burners that are widely used in power and process industries [2]. They combine the wide range of operating conditions with stable flame and low emission levels. The basic idea is to introduce swirl motion to the stream of air, fuel or both. This not only improves stability of the flame but also intensifies mixing.

However, it has been recognized for long time that the prediction of swirling diffusion flames using moment turbulence closures is extremely problematic, in spite of partial successes i.e. predictions of in-flame properties reported in [3–5]. Recent progress achieved using large-eddy simulations coupled with advanced chemistry models is on one hand very promising but on the other hand it is still far from being applicable to industrial problems due to excessive computational requirements, tractable only using supercomputing facilities due to huge dimensions of the combustors (on the order of 10 m) and the need to resolve fine features like gas nozzles with diameters on the order of 1 mm.

Industry primarily requires predictions of wall heat fluxes (typically for membrane walls or tubes). In spite of that, combustion modeling research almost exclusively focuses on the details of flame core structure and wall heat loads are typically disregarded. The present work thus focuses on validation of computationally manageable Reynolds-averaged Navier-Stokes (RANS) models by accurately measured local wall heat fluxes. The experiments were performed at a modern experimental facility of the Institute of Process and Environmental Engineering, FME, BUT.

1.1 MOTIVATION

Our institute is equipped with the experimental facility containing industrial size combustion chamber. This allows us to test not only new fuels from renewable sources but also new combustion equipment such as low-NO_x burners as well. Since we want to provide deeper insight into the process of combustion at the facility the suitable tool is Computational fluid dynamic (CFD) simulations. Even though it has been developing for many years there are still many concerns for use in specific areas such as combustion.

Testing facility was designed with accurate heat flux measurement and therefore this work benefits from it. It is the first step to verify ability of the computational software to accurately predict wall heat fluxes. This ability is highly demanded among design engineers. The local wall heat fluxes are crucial in the design of process or power industry equipment. It is required by material and stress analysts and designers who needs to know real heat loads of the walls. Other area is in already built plants for failure analysis, retrofit design and life cycle analysis.

1.2 WORK OBJECTIVE

First task is to provide set of accurate and well documented measurements focused primarily on the local wall heat fluxes (heat transfer rates) in a water cooled combustion chamber equipped with natural gas turbulent swirling diffusion flame burner. Accuracy of the measurements and exact operating conditions have to be reported. This will serve as a basis for validation of simulations.

When reliable data are gathered the focus will move to simulations. The models shall be examined and the best settings found with respect to pre-defined computational resources. Results will be helpful in industrial scale combustor simulations and providing guidelines for model selection.

2. MEASUREMENTS AT LARGE-SCALE COMBUSTOR FACILITY

2.1 EXPERIMENTAL FACILITY OVERVIEW

The construction of the semi-industrial experimental combustion facility for burners up to 2 MW enables variable length adjustment of the combustion chamber and accurate heat flux and emission measurements. The main feature distinguishing the test facility at Brno University of Technology from others is the ability to measure local heat transfer rates to the cooled walls, which is enabled by the segmental design of the combustion chamber.

Several measurements previously performed at the same testing facility, although with different objectives than in this work, were described in [2, 6, 7].

2.1.1 Combustion chamber and burner geometry

There are up to seven water cooled segments of the combustion chamber, see Fig. 2.1. All internal segments are of diameter 1 m and length 0.5 m and have the same flame-facing area of 1.57 m^2, whereas the first and seventh section have length 0.4 m (area 1.26 m^2) and 1 m (area 3.14 m^2), respectively. The last three segments are removable which allows adjustment of the combustion chamber length.

The burner was a low-NO_x design with staged gas supply and axial swirl

Figure 2.1: Industrial-scale combustor

generator, fired by natural gas. It was possible to adjust burner geometry thanks to the six adjustable geometrical parts.

Geometry settings of the burner and a few other parameters were derived from the set of measurements as described in [6] and [7]. The primary parameter was stability of the flame and low emissions. The final assemble consists of swirl generator with diameter 240 mm made of 8 vanes with pitch angle of 35° acting as a flame holder. Gas inlet includes twelve primary nozzles and eight secondary nozzles. Eight of the primary nozzles have diameter 2.6 mm and the other four 3.0 mm. All the primary nozzles are drilled in a nozzle head located on the burner axis. Secondary gas injection is performed by four additional nozzle heads located in regular intervals around annular air channel which surrounds the primary nozzle head. Each of the four secondary nozzle heads has two nozzles with a diameter of 3.3 mm and head angle of 20°. Secondary nozzle angle is 0° which means that it faces directly to the axis of the chamber and toward the center of the burner. Axial position is 0 mm therefore with the lowest possible extension to the chamber and radial position is 0 mm as well which states for the closest radial position to the axis of chamber and the burner. Throttling element had diameter of opening 5.5 mm. This settings were utilized in all the measurement in this thesis.

2.2 MEASUREMENT UNCERTAINTY AND ITS PROPAGATION IN CALCULATED DATA

Using the information from sensor manufacturers, the uncertainty of derived parameters can be calculated. To do this, the theory of uncertainty propagation [8] was utilized. E.g. the standard deviation σ_P of product P of uncorrelated variables A and B which have standard deviations σ_A and σ_B may be calculated from the following formula:

$$\left(\frac{\sigma_P}{P}\right)^2=\left(\frac{\sigma_A}{A}\right)^2+\left(\frac{\sigma_B}{B}\right)^2. \tag{1}$$

Equation 1 is general form of uncertainty propagation in summations. The accuracy of the measurement of heat transfer rate in a segmental experimental combustion chamber with water cooling may be determined using the calorimetric equation

$$Q=\dot{m}c_p\Delta T/A\,, \tag{2}$$

where Q is the heat transfer rate [kW/m^2], m is mass flow rate of the cooling water [kg/s], c_p is specific heat capacity [kJ/kg-K], ΔT is temperature difference [K] and A is the heat exchanging surface area of a section [m^2]. By applying the theory of error propagation [8] we can easily show that for the variance of calculated heat flux holds the following relation:

$$\left(\frac{\sigma_Q}{Q}\right)^2=\left(\frac{\sigma_{\dot{m}}}{\dot{m}}\right)^2+\frac{\sigma_{T_{in}}^2+\sigma_{T_{out}}^2}{\Delta T^2}. \tag{3}$$

2.3 RESULTS OF WALL HEAT FLUX MEASUREMENTS

Total of four long term experiments were carried out at the testing facility focused on wall heat flux measurement. Two at the firing rate 745 kW (referred as

Case 1) and other two at 1120 kW (referred as Case 2). All the experiments utilizes the excess air ratio 1.1 and the same burner geometry. Repetitive measurements were performed to verify repeatability of experiments.

Based on my recommendation the cooling water flow through each duplicator was reduced to the minimum. It improved uncertainty of the heat flux measurement by 50 %, e.g. from uncertainty 16.1 % for original flow rate to the 8.4 % at the lowered flow rate. The reason is that due to the lower flow rate the outlet temperature increases and so the difference between outlet and inlet temperature increases. Based on the calculation of propagation of uncertainty the heat flux measurement uncertainty is inversely proportional to the temperature difference. The fact significantly improved measurement results.

Main operating parameters characterizing the two cases for all four runs are summarized in Table 1. Note that natural gas utilized during experiment was for CFD simulation purposes substituted by methane and the flow rates were corrected to compensate the difference in heating values. Mass flow rates through particular gas stages were not measured since we do not have flow meters there, but are rather calculated in CFD simulation. Model of the entire gas-staged fuel piping system was created earlier in our research group which I used for flow rate calculations.

Table 1: Operating conditions in the Case 1 (745 kW)

		Measure-ment A	Error estimate [%]	Measurement B	Error estimate [%]	Average
Total thermal duty	[kW]	745.7	1.62	748.0	2.7	746.9
Natural gas flow rate	[kg/s]	0.01517	1.62	0.01522	2.7	0.01520
Calculated methane mass flow rate	[kg/s]	0.01486	1.62	0.01492	2.7	0.01489
Air mass flow rate	[kg/s]	0.289	9.8	0.290	10.1	0.290
Fuel temperature	[°C]	20.11	1.5	12.5	2.6	16.31
Air temperature	[°C]	19.23	1.5	4.26	1.9	11.75
Total extracted heat flux	[kW]	438.5	4.3	437.8	5.27	438.1
Mass flow rate at primary gas stage	[kg/s]					3.84E-3
Mass flow rate at secondary gas stage	[kg/s]					1.10E-2

The measured heat flux densities are summarized in Table 3, which includes the average of measured values from the two data sets available for each of the cases 1 and 2. These average values (and corresponding averaged operating conditions) were used in simulations reported in the following sections. The table also includes mean fluctuations observed in the measurement during the period of steady operation, i.e. standard deviation of the measured data. Additionally, there are provided also error estimates calculated by error propagation theory according to accuracy of the involved sensors.

Table 2: Operating conditions in the Case 2 (1120 kW)

		Measurement A	Error estimate [%]	Measure-ment B	Error estimate [%]	Average
Total thermal duty	[kW]	1115.0	1.6	1124.2	2.4	1119.6
Natural gas flow rate	[kg/s]	0.02270	1.6	0.02286	2.4	0.02278
Calculated methane mass flow rate	[kg/s]	0.02224	1.6	0.02240	2.4	0.02232
Air mass flow rate	[kg/s]	0.435	9.8	0.438	10	0.436
Fuel temperature	[°C]	20.58	1.5	13.08	2.3	16.83
Air temperature	[°C]	20.55	1.4	8.53	1.6	14.54
Total extracted heat flux	[kW]	591.5	3.3	596.7	4.2	594.1
Mass flow rate at primary gas stage	[kg/s]					5.79E-3
Mass flow rate at secondary gas stage	[kg/s]					1.65E-2

Table 3: Measured heat fluxes – averaged values from two measurements

	Case 1 (745 kW)			Case 2 (1120 kW)		
	Heat flux	Mean fluctuation	Error estimate	Heat flux	Mean fluctuation	Error estimate
	[kW/m^2]	[kW/m^2]	[%]	[kW/m^2]	[kW/m^2]	[%]
Section 1	17.25	0.15	8.4%	21.88	0.21	6.4%
Section 2	25.57	0.16	4.8%	34.05	0.27	3.5%
Section 3	40.17	0.14	2.9%	53.28	0.26	2.3%
Section 4	46.41	0.15	2.8%	63.58	0.24	2.0%
Section 5	47.87	0.16	2.6%	65.45	0.27	1.9%
Section 6	42.33	0.17	2.8%	58.9	0.29	2.0%
Section 7	31.4	0.21	2.0%	42.74	0.19	1.6%

2.4 SUMMARY

Modern industrial-scale facility allows us to accurately measure local wall heat fluxes with uncertainty of measured data better than 8.5 %. The method of propagation of uncertainty was utilized to identify error estimate of measured data. The acquisition system is able to collect data with frequency 1 Hz except for the cooling water flow measurement with the frequency 0.17 Hz. Other measuring technique is in development – i.e. inflame temperature measurement which shows us promising results. Wall emissivity was identified to be about 0.9 based on the measurement of the wall sample with identical coating of the chamber.

The data processing and evaluation with my in-house software allows us to quickly use measured data for CFD analysis. Underlying features of the software were described in the fulltext of this thesis and the software was included on CD-ROM.

3. GAS COMBUSTION MODELING AND SIMULATION

This chapter summarizes modeling approaches for reactive flows with radiative heat transfer. Focus is on the turbulence modeling and its effect on wall heat flux predictions. Several comparisons of predicted wall heat fluxes with two measured cases (firing rate 745 kW and 1120 kW) are provided. All discussed models and methods are either available in ANSYS Fluent® commercial solver or source code is given for its implementation via User defined function (UDF).

Details of geometry model, chemistry modeling, radiation model, absorption coefficient and mean beam length calculation and material properties are provided in the fulltext of this thesis. Nevertheless, all the results are discussed in Summary (Chapter 3.2) and Conclusions (Chapter 5).

3.1 TURBULENCE MODELING

Turbulence is the major concern in modeling of swirling diffusion flames. Turbulence is the main reason why simulation of this type of flames is so difficult. Swirling flows generally (including non-reacting flows e.g. in cyclones) pose a challenging problem for CFD simulations and the additional complexity caused by large density variations, turbulence-modified chemical reaction rates and radiative heat transfer make the simulations even more challenging.

The objective of this work is to analyze the performance of CFD models that have acceptable computational requirements and thus can be applied in the industrial practice. Therefore the considered turbulence models comprise a selection of turbulence modeling approaches that are available in most commercial CFD codes.

The first group of models is applicable to the calculation of time-averaged (Reynolds-averaged) flow field properties. This group of models includes so-called moment turbulence closures and they are collectively classified as Reynolds-averaged Navier-Stokes models (RANS).

When turbulent fluctuations are too severe and simulations using RANS models do not converge, it is possible to include accumulation terms into the model equations and simulate the problem as transient. The resulting models are collectively denoted as unsteady RANS models (URANS).

3.1.1 Effect of turbulence models on wall heat flux

Four RANS turbulence models available in ANSYS Fluent® were investigated for their influence on wall heat flux predictions. Namely it was realizable k-ε, RNG k-ε, SST k-ω and RSM turbulence model. Solver settings were kept the same for all of the test cases as specified in Table 4 and 5. Simulations were run in unsteady mode since combustion in such complex geometry is physically transient phenomena. Time step was chosen according the convergence to allow the solver to perform from ten to twenty iterations per time step. New method for WSGGM absorption coefficient calculation was utilized [9].

Table 4: Solver settings of all cases

Model	Settings
Turbulence model	realizable (or RNG) k-ε or SST k-ω or RSM
Radiation model	Discrete ordinates
Species transport	EDM with global one step mechanism
Pressure-velocity coupling	SIMPLEC
Skewness correction	1
Time step [s]	0.002

Table 5: Discretization scheme of all cases

Variable	Scheme
Pressure	PRESTO!
Density	QUICK
Momentum	QUICK
Turbulent Kinetic Energy	First Order Upwind
Specific Dissipation Rate	First Order Upwind
CH_4	First Order Upwind
O_2	First Order Upwind
CO_2	First Order Upwind
H_2O	First Order Upwind
Energy	First Order Upwind
Discrete Ordinates	First Order Upwind

Comparison has been made for both measured cases (see Chapter 2.3). The first one for the Case 1 with firing rate 745 kW and the second for the Case 2 with firing rate 1120 kW. All the boundary conditions remain identical except for mass flow inlets, i.e. combustion air inlet and all the fuel inlets.

Turbulence model comparison for the Case 1 (745 kW)

Results show negligible effect of turbulence model on overall heat transfer which differs by less than 5 % from measured value. This is given by fact that all fuel has enough time to completely mix with oxygen, burn and release heat no matter what turbulence model is used. After that it is just upon the radiative properties of gas and walls how much heat is transferred into walls and how much is carried out of the chamber by the gas. Turbulence may only affect locations within the chamber, where the heat is released.

Presented results are the best that were achieved so far. Overall heat flux for the SST k-ω model deviates just by 0.3 % from the measured value. The profile of wall heat fluxes along the axial length of chamber fits well to the measured profile – see Fig 3.1.

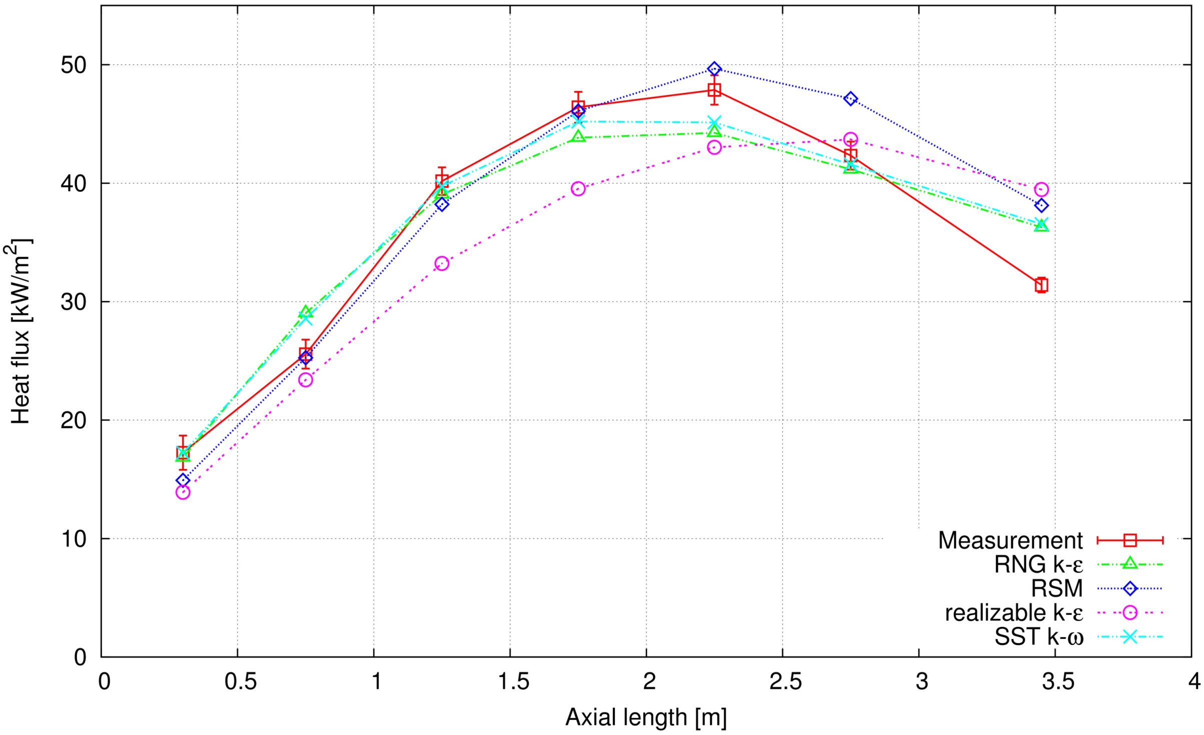

Figure 3.1: Turbulence model comparison and its effect on predicted wall heat flux

Surprisingly the worst predictions give one of the modern turbulence models realizable k-ε. The overall transferred heat into wall has acceptable deviation of 5.0 %. However the profile of the heat flux is inaccurate. First five sections are underestimated (up to 19 %) while the last seventh section is overestimated by 25 %.

Displayed values in Figure 3.1 are averaged over at least 2 seconds of physical time in simulation.

Turbulence model comparison for the Case 2 (1120 kW)

Even at higher firing rate the results confirm accurate predictions of total extracted heat. Good agreements show realizable k-ε turbulence model with deviation just 0.12 %. RSM turbulence model indicated the highest deviation from measured total extracted heat (3.8 %). On the contrary the local wall heat fluxes monitored by separate sections shows higher deviations than in preceding simulations of Case 1, see Figure 3.2. In the case of SST k-ω turbulence models deviation reaches up to 41 % in last, seventh section when compared to the measurement. All the turbulence models, but one, amplified previously observed trend in overprediciton of local wall heat fluxes in the last section. Only exception was RSM turbulence model that gives consisting deviation in all the sections for both cases 1 and 2.

Obvious underprediction of the local wall heat fluxes in the first four sections and significant overprediction in the last two sections indicates low turbulence mixing of fuel and oxidizer. Turbulence models seems to have problems with the swirling flow enhanced mixing and heat release. Therefore, it takes longer time than in reality.

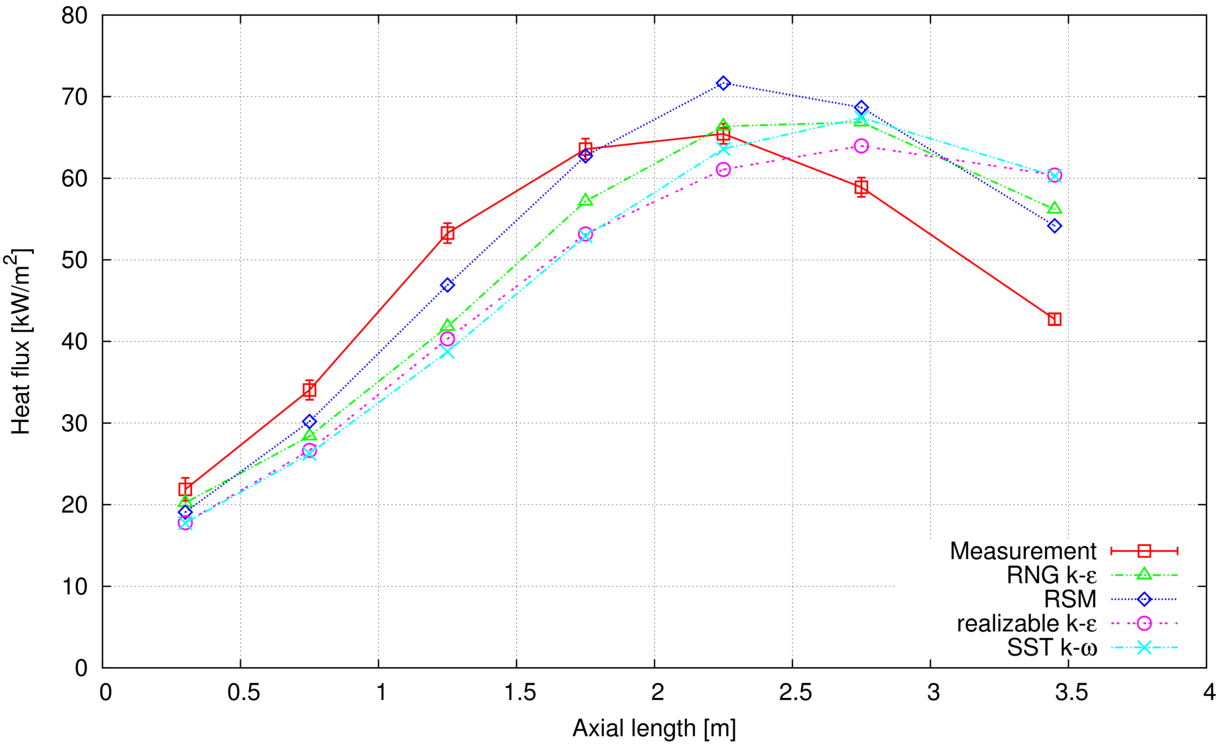

Figure 3.2: Turbulence model comparison for the Case 2

3.1.2 Summary of turbulence modeling

SST k-ω turbulence model along with new implementation of WSGGM and domain-based mean beam length predicts local wall heat fluxes in the Case 1 the most accurately. Overall heat flux deviates by 0.3 %. However it still tends to overpredicts heat fluxes in the last section. In this case it is over 16 %. Very good results gives RNG k-ε model as well as with deviation of total heat fluxes at 1 % and maximum local deviation no more than 15.5 %.

Predictions for the Case 2 show higher deviations from measured values. Total extracted heat is still in a good agreement (from 0.12 % to 3.8 %) but the local wall heat fluxes in last section tend to significantly overpredicts measured data (from 26.7 % to 41 %). I assign this behavior to the problematic predictions of swirling flow affecting mixing and therefore even heat release. However, such problems are only amplified for higher firing rates with more intensive swirl and higher velocity through the swirl generator.

3.2 SUMMARY

Several aspects of gas combustion modeling and simulations were shown in this chapter. Focus was on models and properties of the simulation influencing the wall heat flux predictions. In the Table 6 is in descending order summarized effect on total wall heat flux. The maximum change in total wall heat flux represents e.g. in emissivity of walls change of total extracted head between emissivity 0.6 and 0.9.

Table 6: Influence of solver models or material properties on total extracted heat

Parameter	Maximum change in total wall heat flux [%]
Absorption coefficient/mean beam length	19.2
Emissivity of walls	12
Turbulence model	7.6
Chemistry model	1.8

From the Table 6 can be seen that the most sensitive is the total wall heat flux to the mean beam length calculation which decrease heat flux by 19.2 % when changed from cell-based to domain-based method. Significant is also emissivity of walls which can be however identified from literature or measured. Turbulence models has surprisingly a little effect (just 7.6 % change). The highest deviation (7.6 %) was between RSM and realizable k-ε turbulence model.

4. SWIRLING FLOW PREDICTION AND MODELING

Modeling of the combustion chemistry via simple eddy dissipation model, which utilizes the strategy mixed-is-burned relies on the accurate turbulence prediction more than any other chemistry model. The reason is that turbulence is the driving factor for mixing and therefore also for chemistry and heat release. The importance of the turbulence modeling is therefore amplified.

4.1 INTRODUCTION

Following discussion of swirling flows and more details were already introduced in my previous work [10, 11]

Swirl-stabilised non-premixed flames are frequently used in industrial burners, but they represent one of the most difficult problems to predict computationally. Only with the advances in large eddy simulations (LES), successful predictions of in-flame properties were reported [3–5]. The LES approach is unfortunately still too computationally expensive for the simulation of large-scale fired heaters due to their huge dimensions (on the order of 10 m) and the need to resolve fine features like gas nozzles with diameters on the order of 1 mm. The only viable alternative for practical predictions in the present as well as for a number of years to come thus consists of models based on first or second-order turbulence closures.

The key question in predicting swirling diffusion flames is, whether the prediction of swirl using geometry of swirl generator is dependable. In the literature, only scarce instances may be found of measurements suitable for the validation of such swirl generation predictions [12, 13]. In most cases of advanced predictions of swirling flows including those mentioned above, boundary conditions on the inlet are typically specified using measured velocities and velocity fluctuations. Predictions validated by experimental data are almost nonexistent in peer-reviewed journals. Occasionally, swirl is even specified by geometric swirl number, i.e. by inclination of swirl generator vanes (helixes) [14]. Neither of these approaches is however suitable for most cases of practical predictions of swirl-stabilised gas and liquid fuel burners, due to the large variety of swirl generator designs used by burner vendors and due to the unavailability of detailed measurements.

For the quantitative description of the relative strength of tangential momentum a nondimensional swirl number (S) is used, which is defined as the ratio of axial flux of tangential momentum over axial flux of axial momentum [15]. In most cases published works provide values of swirl number calculated on the basis of swirl generator geometry as proposed by [16]. The geometric swirl number must however be used thoughtfully, as it is suitable only for specific swirler geometries, e.g. when guide vanes cover the whole cross-section of air flow tube and there are no short-cut currents. In spite of this, number of authors provides geometry-based swirl number as the only information about swirl intensity, e.g. [17], [13]. Swirl number calculated from measured velocity profiles is encountered less frequently in the literature, e.g. in [18] or [19], 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 in most burners), where PVC has been observed even with lower swirl numbers [20].

4.2 EXPERIMENTAL DATA FOR VALIDATION OF CFD SIMULATION

Since I was aware of the importance of the swirling generation and propagation on the combustion process the investigation was initiated to find capabilities of the utilized software ANSYS Fluent®. The task was to identify published experiment with the same swirler to ours'. However, the only experiment with guide vane swirler was found while the burner at our facility is equipped with combination of bluff-body and guide vane swirl generator. The combustion air flows partly around our swirler since the duct has bigger diameter than the swirler and partly through the swirler and along the guide vanes.

In our case after the swirler there is about one diameter long constant cross-section channel followed by the sudden change of diameters from supply air round duct into combustion chamber itself. Similar concept was adopted in many works, e.g. [12, 13, 21, 22]. However, in the first stage we focused on the flow field prediction just behind the swirler and before a sudden expansion. Our aim is to see ability of the solver to predict flow through guide vanes.

After a literature survey the most proper source of measured validation data was found in a work of [12]. They utilized axial guide vane swirler. The geometry of experimental setup was further clarified in personal communications with one of the authors [23]. Measured data were in a suitable form however many details necessary for validation of the CFD simulation were missing. The main problem was with uncertain geometry specification, which was cleared by personal communication [23]. Nevertheless, doubts still remains since the authors had just a few records about the measurement which took place several years ago. This is common problem among many published articles with experimental data. Nearly none of them provide complete geometry specification, which would allow to create reliable model for CFD computation.

Figure 4.1: Sudden expansion chamber with swirler [12]

The measurements were performed for a vane swirl generator by optical method (particle image velocimetry, PIV). Geometry of the computational domain including the swirl generator is displayed in Figure 4.1. Inclination of the guide vanes in the present case is 45°. The experimental work was focused on analyzing flow features in a sudden expansion and its deeper analysis by proper orthogonal decomposition (POD), but they measured also velocity components just above the expansion ($x/D = -0.44$) in order to determine accurately the amount of swirl in the expanding flow. These velocity measurements above the expansion were used in the present work to validate computational predictions. Working medium was water.

4.3 RESULTS

Since we wanted to compare predicted data with the validation data from the study of Mak et al.[12] it was necessary to make the same control plane at the x/D = -0.05. At this plane the line was created and data from the line were exported. All the results are averaged values over several seconds of physical time.

4.3.1 Flow fields predictions

Three turbulence models were tested for ability to predict flow field in three-dimensional domain. Figure 4.2 bellow shows comparison of results from the three turbulence models. For axial velocity profiles the decrease was predicted in the center by all the models however only SST k-ω and RNG k-ε turbulence model on a rough mesh predicted reversed flow. Since the results were not confirmed on a finer meshes it might be rejected as scatty results.

Problem in predictions of axial velocity is caused on one hand by radial momentum transport from the swirl effect and on the other hand in contrary by jet penetration downstream from the short-circuit through the center of guide vane swirler.

Other effect is caused by guide vanes which generates vortex shedding. Those vortices are then pushed toward the wall by radial transport of momentum, travel downstream and influence near wall velocity profile.

Radial velocity profile near the axis is predicted well by all the models. However at the near-wall region strongly deviates from the measured data. It might be caused

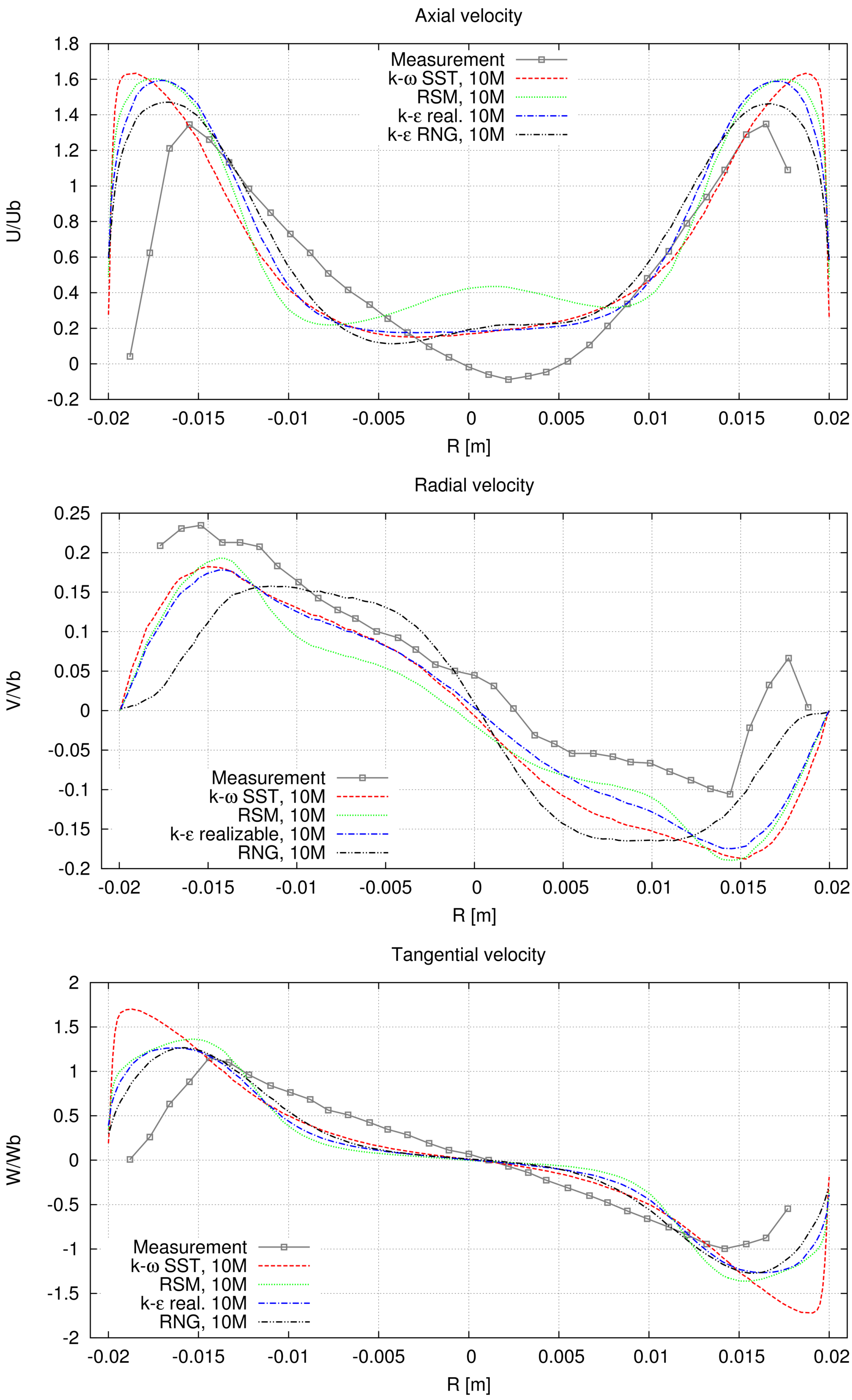

Figure 4.2: Profiles of axial, radial and tangential velocity in the highest density mesh for four turbulence models

by vortex shedding mentioned earlier which affects flow field near wall and RANS turbulence models cannot describe it.

Near-axis tangential velocity and its gradient is in all cases underpredicted. While in the near wall region is tangential velocity significantly overpredicted. This leads us to hypothesis that swirling tangential momentum is pushed toward the wall while in the center of the stream dominates non-swirling jet, penetrating further downstream.

Our data implies that none of the models is able to predict the solid body rotation of the core swirling flow, which is observed in the measured data. Moreover, the SST k-ω model shows an unexpected behavior in the tangential momentum transport behind the swirler, as monitored by the swirl number. Discussion of the results is confronted with previously published observations on this topic. The aim is to critically evaluate the applicability of computations to determine inlet boundary conditions for swirling air in industrial combustors.

4.4 SUMMARY

I was not able to find any well documented measurement with the axial guide vane swirl generator combined with bluff-body as our swirler utilized in the burner in the literature.

Results show that prediction of swirling flow in the given geometry is problematic. One key factor is combination of jet-like flow combined with guide vane swirl generator influenced flow. When interaction of these two flows is involved turbulence models fail to predict velocity flow fields in the near-wall region no matter what turbulence model is utilized from common set of commercially available turbulence models.

5. CONCLUSIONS

This thesis was focused on the phenomena of heat transfer within combustion chamber with turbulence swirling diffusion flame. The fuel was natural gas converted for the purpose of CFD simulation to the equivalent flow rate of methane. Oxidizer was air at ambient conditions. Measurements were performed at experimental facility with industry-size combustor. The facility is equipped with modern data acquisition system. Key feature distinguishing this experimental facility from others is segmental design with accurate wall heat flux measurement. Measurement of local wall heat fluxes was performed for two firing rates 745 kW and 1120 kW. Measurement uncertainty of the local wall heat fluxes was calculated as better than 8.4 % in (Section 1) for the Case 1 (firing rate 745 kW). All other parameters (flow rates, temperatures, pressure, etc.) were acquired as well. Calculated uncertainty was usually better than 2 % for all parameters but for air mass flow rate, where it reaches 9.8 % for the Case 1. Such a data was then utilized for validation of CFD predictions.

Settings of various industry-standard models utilized for combustion and underlying physical phenomena were investigated. Boundary conditions were set according to processed data from measurements. Simulations were focused on effects of individual models and their settings on total and local wall heat flux predictions. Investigated models were turbulence models, chemistry models and radiative properties of involved materials. Their effect on wall heat flux was assessed and it was found that predictions are most sensitive to the evaluation of mean beam length which is related to the absorption coefficient of the gas in combustion chamber. Different evaluation of this parameter may change the total wall heat flux prediction by 19.2 %. Therefore it is essential to properly choose this parameter.

Predictions of wall heat fluxes for the Case 1 with lower firing rate was found to be very accurate. Total wall heat flux was predicted with deviation from measurement by just 0.3 %. Predicted local wall heat fluxes deviated the most in last (seventh) section by 16 %. However, in other sections the deviations were around 5 %. In the Case 2 with the higher firing rate predictions of total extracted heat was very good as well. Predictions deviated from measurement by just 0.1 %. However, the turbulence models were found to be limiting factor in such simulations since deviation in local heat flux measured at individual sections reached up to 27 %. The whole profile of heat fluxes was shifted toward the outlet of the combustion chamber. First four sections were underestimated while last three sections were overestimated. Such profile shift was predicted by all turbulence models. This reveals problems in swirling flow predictions and swirl generation in the guide vane swirl generator which was included in computational model.

Prediction difficulties of swirling flow are discussed in the last chapter of this thesis with focus on effect of turbulence models. Well documented measurement with a swirl generator similar to ours was utilized from literature. First of all a grid independence study has been performed on prepared computational models. Even though the model included sudden expansion chamber, my attention was paid to the first measuring plane between swirl generator and sudden expansion. This plane captured velocity fields of the generated swirling flow. Decay of swirl and momentum transport was thus monitored without additional geometrical disturbances. The aim was to verify ability of turbulence model to predict flow field generated by the guide-vanes swirl generator. Results confirm that utilized turbulence models have trouble to accurately predict such flow. The main problem seems to be in interaction of the jet at the center of the swirl generator and swirling flow generated by the guide vanes.

REFERENCES

[1] *Directive 2001/80/EC of the European Parliament and of the Council on the limitation of emissions of certain pollutants into the air from large combustion plants* [online]. 23 October 2001. [Accessed 30 November 2009]. Available from: http://eur-lex.europa.eu/LexUriServ/LexUriServ.do?uri=CELEX:32001L0080:EN:HTML.

[2] KERMES, V., BĚLOHRADSKÝ, P., ORAL, J. and STEHLÍK, P. Testing of gas and liquid fuel burners for power and process industries. In: *Energy*. October 2008, Vol. 33, no. 10, pp. 1551–1561. DOI 10.1016/j.energy.2008.07.013. 10th Conference Process Integration, Modelling and Optimisation for Energy Saving and Pollution Reduction, Ischia Isl, ITALY, JUN 24-27, 2007.

[3] FUREBY, C., GRINSTEIN, F.F., LI, G. and GUTMARK, E.J. An experimental and computational study of a multi-swirl gas turbine combustor. In: *Proceedings of the Combustion Institute*. January 2007, Vol. 31, no. 2, pp. 3107–3114. DOI 10.1016/j.proci.2006.07.127.

[4] SADIKI, A., MALTSEV, A., WEGNER, B., FLEMMING, F., KEMPF, A. and JANICKA, J. Unsteady methods (URANS and LES) for simulation of combustion systems. In: *International Journal of Thermal Sciences*. August 2006, Vol. 45, no. 8, pp. 760–773. DOI 10.1016/j.ijthermalsci.2005.11.001.

[5] JAMES, S., ZHU, J. and ANAND, M.S. Large eddy simulations of turbulent flames using the filtered density function model. In: *Proceedings of the Combustion Institute*. January 2007, Vol. 31, no. 2, pp. 1737–1745. DOI 10.1016/j.proci.2006.07.160.

[6] BĚLOHRADSKÝ, P., KERMES, V. and STEHLÍK, P. Design and analysis of experiment for low-NOx burners design for process industries. In: *8th European Conference on Industrial Furnaces and Boilers*. Vilamouria, Portugal: 2008.

[7] KERMES, V., BĚLOHRADSKÝ, P. and STEHLÍK, P. Influence of burner geometry with staged gas supply on the formation of nitrogen oxides. In: *8th European Conference on Industrial Furnaces and Boilers*. Vilamouria, Portugal: 25 March 2008.

[8] BRAEMBUSSCHE, V. Den (Ed.). *Measurement Techniques in Fluid Dynamics: An Introduction*. Rhode Saint Genése: von Karman Institute for Fluid Dynamics, 2001.

[9] YIN, C., JOHANSEN, L.C.R., ROSENDAHL, L.A. and KÆR, S.K. New Weighted Sum of Gray Gases Model Applicable to Computational Fluid Dynamics (CFD) Modeling of Oxy−Fuel Combustion: Derivation, Validation, and Implementation. In: *Energy & Fuels*. 16 December 2010, Vol. 24, no. 12, pp. 6275–6282. DOI 10.1021/ef101211p.

[10] VONDÁL, J. and HÁJEK, J. Experimental and numerical investigation of swirling non-premixed gas flames in industrial-scale furnace. In: *Proceedings of the 9th European Conference on Industrial Furnaces and Boilers*. Estoril, Portugal: 26 April 2011.

[11] VONDÁL, J. and HÁJEK, J. Prediction of flow through swirl generator and validation by measured data. In: *Journal of Physics: Conference Series*. 2011, Vol. 318, no. 2, pp. 022026. DOI 10.1088/1742-6596/318/2/022026.

[12] MAK, H. and BALABANI, S. Near field characteristics of swirling flow past a sudden expansion. In: *Chemical Engineering Science*. December 2007, Vol. 62, no. 23, pp. 6726–6746. DOI 10.1016/j.ces.2007.07.009.

[13] FERNANDES, E.C., HEITOR, M.V. and SHTORK, S.I. An analysis of unsteady highly turbulent swirling flow in a model vortex combustor. In: *Experiments in Fluids*. 2006, Vol. 40, no. 2, pp. 177–187. DOI 10.1007/s00348-005-0034-4.

[14] SO, R.M.C., AHMED, S.A. and MONGIA, H.C. Jet characteristics in confined swirling flow. In: *Experiments in Fluids*. 1985, Vol. 3, no. 4, pp. 221–230.

[15] GUPTA, A.K., LILLEY, D.G. and SYRED, N. *Swirl Flows*. Abacus Press, 1984. ISBN 0-85626-175-0.

[16] CLAYPOLE, T.C. and SYRED, N. The effect of swirl burner aerodynamics on NOx formation. In: *Symposium (International) on Combustion*. 1981, Vol. 18, no. 1, pp. 81–89. DOI 16/S0082-0784(81)80013-6.

[17] CORTÉS, C. and GIL, A. Modeling the gas and particle flow inside cyclone separators. In: *Progress in Energy and Combustion Science*. 2007, Vol. 33, no. 5, pp. 409–452. DOI 16/j.pecs.2007.02.001.

[18] KHEZZAR, L. Velocity measurements in the near field of a radial swirler. In: *Experimental Thermal and Fluid Science*. March 1998, Vol. 16, no. 3, pp. 230–236. DOI 10.1016/S0894-1777(97)10027-9.

[19] COGHE, A., SOLERO, G. and SCRIBANO, G. Recirculation phenomena in a natural gas swirl combustor. In: *Experimental Thermal and Fluid Science*. September 2004, Vol. 28, no. 7, pp. 709–714. DOI 10.1016/j.expthermflusci.2003.12.007.

[20] RANGA DINESH, K.K.J. and KIRKPATRICK, M.P. Study of jet precession, recirculation and vortex breakdown in turbulent swirling jets using LES. In: *Computers & Fluids*. 2009, Vol. 38, no. 6, pp. 1232–1242. DOI 10.1016/j.compfluid.2008.11.015.

[21] AHMED, S.A. and NEJAD, A.S. Swirl effects on confined flows in axisymmetric geometries. In: *Journal of Propulsion and Power*. March 1992, Vol. 8, no. 2, pp. 339–345. DOI 10.2514/3.23483.

[22] WANG, P., BAI, X.S., WESSMAN, M. and KLINGMANN, J. Large eddy simulation and experimental studies of a confined turbulent swirling flow. In: *Physics of Fluids*. 2004, Vol. 16, no. 9, pp. 3306. DOI 10.1063/1.1769420.

[23] BALABANI, S. *Personnal communication*. 4 October 2010.

CURRICULUM VITAE

Personal data

Name	Vondál, Jiří
Date of birth	15. 7. 1982
Marital status	Married
Address	Staňkova 16, 606 00 Brno

Contact

Cell phone	+420 777 128 907
E-mail	vondal@upei.fme.vutbr.cz

Education

2008 – present	Institute of process and environmental engineering, Faculty of Mechanical Engineering, Brno University of Technology
2001 – 2008	Department of Environmental Engineering, Faculty of Mechanical Engineering, Czech Technical University in Prague
2006	Technische Universität München, ATHENS – Building Energy Simulation course, Munich, Germany
2005 – 2006	University of Nebraska – Lincoln, College of Engineering & Technology, Department of Architectural Engineering, Energy System Laboratory, IAESTE Internship

Work Experience

2008 – present	Institute of process and environmental engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technical staff
2008 – 2011	TechSoft Engineering, spol. s r.o., External consultant for UDF programming

2007-2010	Ing. Jiří Vondál, PROVO, design engineering in field of heating, water supply, natural gas supply and waste water – Junior design engineer
2003 – 2005	IAESTE – The International Association of the Exchange of Students for Technical Experience, PR Manager of the IAESTE Czech Republic, „Week of Companies 2004“ Project Coordinator at CTU

Foreign Languages

English

Computer Skills

C/C++ programming, ANSYS Fluent, Gambit, OpenOffice/Microsoft Office, AutoCAD, SolidWorks, Linux OS.

Other Skills

Driving license class. B

Personal Interests

Basketball, skiing (downhill & crosscountry), cycling, hiking.

AUTHOR'S OWN PUBLICATIONS

LACINA, K., VONDÁL, J., SKLÁDAL, P. A novel approach to the uniform distribution of liquid in multi-channel (electrochemical) flow-through cells. In *Analytica Chimica Acta*. 2012. Vol. 727, pp. 41–46. DOI 10.1016/j.aca.2012.03.046.

VONDÁL, J. and HÁJEK, J. Boundary Condition Evaluation and Stability Issues in Swirling Flame Gas Combustion. In: *ThermaComp2009* [online]. Naples, Italy: 10 September 2009. pp. 314–317. Available from: http://www.thermacomp.com/uploads/ThermaComp2009_Proceedings_web.pdf.

VONDÁL, J. and HÁJEK, J. Experimental and numerical analysis of wall heat transfer in non-premixed gas combustor. In: *Chemical Engineering Transactions*. 10 May 2009, no. 18, pp. 587–592. DOI 10.3303/CET0918095.

VONDÁL, J. and HÁJEK, J. Experimental and numerical investigation of swirling non-premixed gas flames in industrial-scale furnace. In: *Proceedings of the 9th European Conference on Industrial Furnaces and Boilers*. Estoril, Portugal: 26 April 2011.

VONDÁL, J. and HÁJEK, J. Experimental data of the local wall heat flux for combustion codes validation in non-premixed swirling gas flames. In: *Proceedings of the ASME-ATI-UIT 2010 Conference on Thermal and Environmental Issues in Energy Systems*. Sorrento, Italy: 16 May 2010. pp. 931–936.

LIU, G., VONDÁL, J., LIU, M. In-Situ Internal Heat Gain and Air Infiltration Measurement using Genetic Algorithms. In *Proceeding of the 13th Information processing and management of uncertainty in knowledge-based systems*. Paris, France, 2006.

VONDÁL, J., HÁJEK, J., KERMES, V. Local wall heat fluxes in swirling non-premixed natural gas flames in large-scale combustor: Data for validation of combustion codes. In *Chemical Engineering Transactions*. 2010. Vol. 21, no. 2, pp. 1123–1128. DOI 10.3303/CET1021188.

HÁJEK, J., VONDÁL, J., ŠARLEJ, M., STEHLÍK, P. Numerical and experimental analysis of turbulent swirling diffusion gas flames. In *11th Conference* *on Process Integration, Modelling and Optimisation for Energy Saving and Pollution Reduction*. Prague, Czech Republic, 2008.

CHO, Y.H., VONDÁL, J., WANG, G., LIU, M. Optimal Terminal Box Control for Single Duct Air Handling Units. In *Proceedings of international conference for enhanced building operations*. Shenzen, China, 2006.

VONDÁL, J. and HÁJEK, J. Prediction of flow through swirl generator and validation by measured data. In: *Journal of Physics: Conference Series*. 2011, Vol. 318, no. 2, pp. 022026. DOI 10.1088/1742-6596/318/2/022026.

WANG, Z., ZHENG, J., VONDÁL, J., XU, W., WANG, G., LIU, M. Variable Air Volume System Application in Museums. In *Proceedings of Energy Sustainability*. Long Beach, California, 2007.

AUTHOR'S OTHER PRODUCTS

Patent under registration procedure - "Zařízení rozdělující tok tekutiny do více kanálů a jeho použití", Administration number: PV 2010-954, Date of submission: 21. 12. 2010. License was already sold to RNDr. Dagmar Jílková, IČO65171331.

Software TepToky – Stand alone graphical application. Attached on CD-ROM to the fulltext of this thesis.

Software CFD Bilance – UDF routine for ANSYS Fluent. Attached on CD-ROM to the fulltext of this thesis.

ABSTRACT

The ability to predict local wall heat fluxes is highly relevant for engineering purposes as these fluxes are often the main results required by designers of fired heaters, boilers and combustion chambers. The aim of this work is to provide reliable data measured by an innovative method for the case of swirling diffusion natural gas flames and consequently utilize the data for validation of Computational Fluid Dynamics simulations represented by commercial solver ANSYS Fluent® 12.1. The subject is a large-scale combustion chamber with a staged-gas industrial type low-NO_x burner at two thermal duties, 745 kW and 1120 kW. Attention is paid to the evaluation of boundary conditions via additional measurement or simulation, such as wall emissivity and wall temperature. Several in-house software codes were created for computational support. Remarkable results were obtained for low firing rate where prediction reached accuracy up to 0.2 % in total extracted heat and better than 16 % in local wall heat flux in individual sections. However, for high firing rate the accuracy significantly decreases. Consequently, close attention was paid to the confined swirling flow phenomena downstream of the swirl generator. Several problematic points were identified in the prediction capabilities of utilized computationally capable, industry-standard models.

ABSTRAKT

Schopnost predikovat tepelné toky do stěn v oblasti spalování, konstrukce pecí a procesního průmyslu je velmi důležitá pro návrh těchto zařízení. Je to často klíčový požadavek pro pevnostní výpočty. Cílem této práce je proto získat kvalitní naměřená data na experimentálním zařízení a využít je pro validaci standardně využívaných modelů počítačového modelování turbulentního vířivého difúzního spalování zemního plynu. Experimentální měření bylo provedeno na vodou chlazené spalovací komoře průmyslových parametrů. Byly provedeny měření se pro dva výkony hořáku – 745 kW a 1120 kW. Z měření byla vyhodnocena data a odvozeno nastavení okrajových podmínek pro počítačovou simulaci. Některé okrajové podmínky bylo nutné získat prostřednictvím dalšího měření, nebo separátní počítačové simulace tak jako například pro emisivitu, a nebo teplotu stěny. Práce zahrnuje několik vlastnoručně vytvořených počítačových programů pro zpracování dat. Velmi dobrých výsledků bylo dosaženo při predikci tepelných toků pro nižší výkon hořáku, kde odchylky od naměřených hodnot nepřesáhly 0.2 % pro celkové odvedené teplo a 16 % pro lokální tepelný tok stěnou komory. Vyšší tepelný výkon však přinesl snížení přesnosti těchto predikcí z důvodů chybně určené turbulence. Proto se v závěru práce zaměřuje na predikce vířivého proudění za vířičem a identifikuje několik problematických míst v použitých modelech využívaných i v komerčních aplikacích.