CHEMICAL ENGINEERING TRANSACTIONS Volume 18, 2009
Editor Jiří Klemeš
 ISBN 978-88-95608-04-4 ISSN 1974-9791
DOI: 10.3303/CET0918095

Experimental and numerical analysis of wall heat transfer in non-premixed gas combustor

Jiří Vondál, Jiří Hájek
Institute of process and environmental engineering, Brno University of Technology (UPEI VUT), Technická 2, 616 69 Brno, Czech Republic, tel. +420 541 142 374, vondal@upei.fme.vutbr.cz

Prediction of wall heat transfer in combustion chambers attracts attention of design engineers in process industry due its implications for the construction of combustors and boilers. Firstly, this paper addresses the reliability and accuracy of an experiment performed in a large-scale experimental combustion chamber equipped with staged-gas non-premixed 745 kW burner. The experiment serves as means to determine boundary conditions and validate predictions by computational fluid dynamics (CFD) software. Several accuracy issues of the used measuring gauges and instruments are discussed with focus on wall heat fluxes and stabilization of the measured data.
Second part of the paper presents CFD simulations of the combustor using Reynolds-averaged Navier-Stokes (RANS) models and validates them by the measured data. Overall model configuration, boundary conditions and key sub-models for turbulence and chemistry are summarised. Deviations of the predictions from the experiment are discussed and possible explanations are offered.

Introduction

Diffusion swirling gas combustion is a favoured solution in many industrial applications. Thus a lot of recent research work was focused on its analysis in several ways. Experiments seem to be an irreplaceable reliable method for verification of computational models. Most authors dealing with computational fluid dynamical (CFD) simulations of swirling diffusion flames use experimental data in their work for comparison e.g. German and Mahmud (2005) and Khelil et al. (2008). These papers however use detailed data describing the internal structure of the respective flames. Such a complete information is however rarely available.

Many experiments follow narrow particular objectives, e.g. to determine pollutant formation rates and species concentration fields, temperature field, or velocity field. The respective experiment set up is then suitable only for a specific objective, e.g. Kermes et al. (2008). The view adopted in this work is more practical and holistic as we are interested namely in the prediction of wall heat fluxes.

Numerical simulations presented in this paper use Reynolds-Averaged Navier-Stokes (RANS) equations. This approach is preferred in practical applications due its reasonable CPU requirements, unlike more advanced approaches as Large Eddy Simulations or Direct Numerical Simulations. However, numerous works show that the prediction of swirling diffusion gas flames using RANS equations is a difficult task, see e.g. Warnatz et al. (1996) or Weber et al. (1995). Therefore the correct use of RANS

Please cite this article as: Vondal J. and Hajek J., (2009), Experimental and numerical analysis of wall heat transfer in non-premixed gas combustor, Chemical Engineering Transactions, 18, 587-592 DOI: 10.3303/CET0918095

models for practical swirling diffusion flames in a large-scale furnace with experimental validation is the main objective of the present work.

Experimental set-up

The measurements have been performed at a burner testing facility of the UPEI VUT Institute, which has been described in detail by Bělohradský et al. (2008), Kermes et al. (2008) and Kermes et al. (2007). The combustion chamber has 1 m internal diameter and is 4 m long, water cooled. The burner used in the experiments is a gas-staged 745 kW low-NO_x type (see Fig. 1). The shell of the chamber is divided into seven sections, enabling evaluation and monitoring of heat flux in each section. The length of the first six sections is 0.5 m and the last one is 1 m long.

Previous research in the facility was focused on parametric studies of the effect of burner parameters on pollutant formation as documented by Kermes et al. (2007). In the present work, the objective is heat transfer analysis, which required adjustments of the measuring procedures. The main difference is in the time scope because pollutants formation analysis is much less time consuming, since stabilization of the flue gas species composition is mainly related to the flame temperature (outlet flue gas temperature). Flue gas temperature has been a sufficient indicator of a steady state condition for the NO_x formation analysis, whereas it was observed that heat fluxes need much longer time for stabilization (see Fig. 2) and thus the outlet temperature could not be used as an indicator of stabilisation any more.

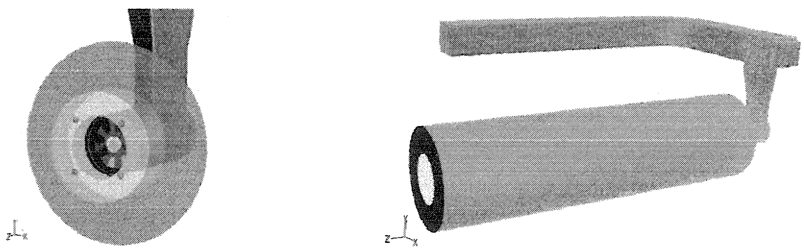

Figure 1: a) Gas-stage burner *b) Combustor*

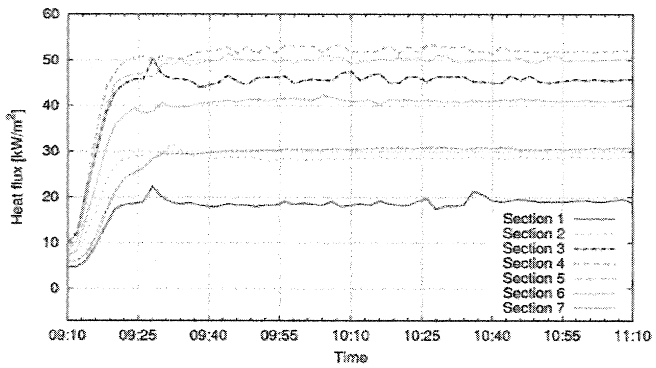

Figure 2: Heat fluxes during the long-term experiment

It was necessary to perform experiment with a stabilizing period long enough to achieve steady state condition in terms of the wall heat fluxes. Thus stable heat fluxes in each individual section had to be achieved in order to obtain reliable data. Fig. 2 shows start-up and stabilization process of the experiment.

Error analysis

The main focus of this paper is on the heat fluxes. Thus error analysis concentrates on three related quantities, namely inlet cooling water temperature, outlet cooling water temperature and cooling water flowrate.Calculations are based on the methodology of propagation of uncertainty according to Braembussche (2001). For the heat flux calculation is used the following simple equation:

$$Q=\rho V c_p \Delta t / A \qquad (1)$$

where Q is heat flux rate [kW/m^2],
ρ is density [kg/m^3],
V is water flow rate [m^3/h],
c_p is specific heat [kJ/kg],
Δt is temperature difference [K] and
A is area of the section [m^2].

Therefore two basic equations for error propagation have to be used:

$$\Delta t = t_{out} - t_{in} \quad : \quad \sigma_{\Delta t}^2 = a^2 \sigma_{t_{in}}^2 + b^2 \sigma_{t_{out}}^2 \qquad (2)$$

$$Q=\rho V c_p \Delta t / A \quad : \quad \left(\frac{\sigma_Q}{Q}\right)^2 = \left(\frac{\sigma_V}{V}\right)^2 + \left(\frac{\sigma_{\Delta t}}{\Delta t}\right)^2 \qquad (3)$$

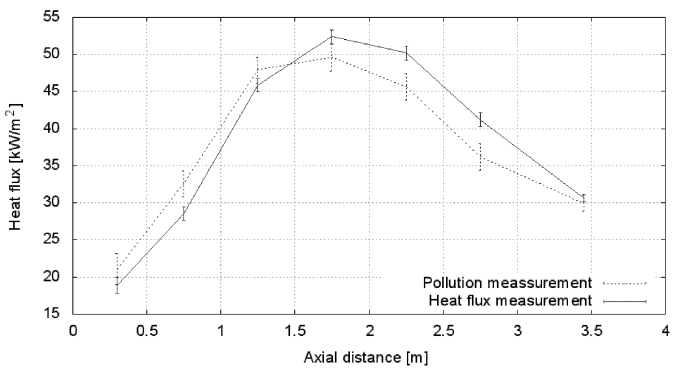

Figure 3: Comparison of two measurements with different objectives

Table 1 and Fig. 3 show the results of error analysis and also comparison of measurements. Objective of the first was pollutant emission analysis with a demand only on stabilisation of outlet flue gas temperature and emission levels and the other focuses on stabilised heat fluxes. Finally an obvious and clearly observable, but very important fact needs to be stressed, namely that the uncertainty discussed above does not include deviations due to poorly stabilised wall heat loads.

Tab. 1: Heat fluxes from the two experiments focused on emission characteristics vs. wall heat loads

	Objective: Pollution measurement			Objective: Heat Fluxes		
	Q	Absolute Error	Relative Error	Q	Absolute Error	Relative Error
	kW/m^2	kW/m^2	%	kW/m^2	kW/m^2	%
Section 1	21.05	2.71	12.87	18.88	1.37	7.23
Section 2	32.59	1.92	5.88	28.53	1.11	3.88
Section 3	47.89	1.98	4.13	45.84	1.06	2.32
Section 4	49.59	2.09	4.21	52.39	1.08	2.05
Section 5	45.61	2.02	4.43	50.20	1.07	2.13
Section 6	36.17	1.99	5.51	41.19	1.11	2.70
Section 7	29.92	1.21	4.06	30.64	0.61	1.98

Computational set-up

The aim of the numerical study was the prediction of steady-state heat fluxes through the water-cooled walls of the combustion chamber. Since the case involves swirling non-premixed combustion, the problem is at the utmost limits of current state-of the art RANS models as discussed in the introduction and a proper validation is thus required.
For numerical analysis the meshed model consisted mostly of hexahedral cells (97 %). The total number of computational cells was 1,156,788. Simulation-relevant details were modelled including air supply ductwork, swirl generator and fuel jets (see Fig. 1a,b). The jet diameters were adjusted to compensate nozzle area reduction in the meshed model, as compared to the real cross-sectional area.
Four different types of boundary conditions have been used. Namely mass flow inlets (gas and air), pressure outlet, Dirichlet condition for walls of the segmental combustion chamber (constant cooling water temperature) and all other walls were approximated as adiabatic. The boundary condition used in the present simulations for the water-cooled segmental wall is slightly idealised. The assumption was made that cooling is highly efficient and temperature boundary layer can be neglected. Thus logarithmic average of the inlet and outlet temperature of each section was deemed adequate as an approximation of the wall temperature.
All computations performed were using RANS equations coupled with Eddy Dissipation Model (EDM) (Magnussen and Hjertager, 1977). For RANS coupled with Flamelet model see Hájek et al. (2008). Three different turbulence models were tested. The most computationally costly was the Reynolds Stress Model (RSM) (Fluent, 2006), but it displayed high unsteadiness when used with second order discretization scheme. Other two turbulence models were two-equation models: SST k-ω (Menter, 1994) and realizable k-ε (Shih et al., 1995). All models were used with the third order

discretization scheme QUICK (Leonard and Mokhtari, 1990) for momentum and density equations and Pressure Staggering Option (PRESTO!) (Patankar, 1980) for the pressure (continuity) equation.

Results and discussion

As can be seen in Fig. 4, the results show that all simulations overestimated the wall heat fluxes. RSM and k-ε overpredict the total heat flux by 10 %. These two models give similar results, acceptably agreeing with measured values in the first three sections, but strongly overestimating heat fluxes in the last three sections. This leads to a conclusion that EDM when used with k-ε or RSM models is unable to predict length of the flame accurately. This is in agreement with Warnatz et al. (1996) who show that EDM highly overpredicts the temperature in the end of the flame.
However, the k-ω model when coupled with EDM predicts qualitatively reasonably well the heat flux profile over the whole length of the chamber and fails only in the identification of absolute values. The overall difference of extracted heat is again 10 %.

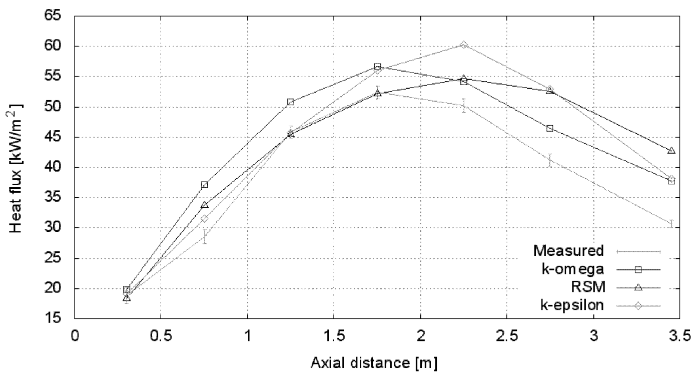

Figure 4: Turbulence model comparison

Conclusion

The reported work presents a rigorous method of identification of boundary condition values for numerical analysis. It points out the importance of correct experimental set-up and discusses achieved accuracy for the reported experiments. Error analysis is performed to increase reliability and credibility of the measurements.
The results of the CFD simulations are not completely satisfactory, but the k-ω turbulence model coupled with simple eddy-dissipation chemistry model predicts qualitatively well the overall profile of wall heat fluxes. This combination of models performs better than eddy-dissipation chemistry coupled with the RSM and the k-ε turbulence models.

Acknowledgement

The authors gratefully acknowledge financial support of the Ministry of Education, Youth and Sports of the Czech Republic within the framework of project No. 2B08048

″Waste as raw material and energy source″ and research plan No. MSM 0021630502 "Waste and Biomass Utilization focused on Environment Protection and Energy Generation".

References

Bělohradský, P., V. Kermes, and P. Stehlík, 2008, Design and analysis of experiment for low-NOx burners design for process industries, in 8th European Conference on Industrial Furnaces and Boilers, Vilamouria, Portugal.

Braembussche, V. D. (Ed.), 2001, Measurement Techniques in Fluid Dynamics: An Introduction, von Karman Institute for Fluid Dynamics, Rhode Saint Genése.

Fluent, 2006, FLUENT 6.3.26, User`s Guide,

German, A., and T. Mahmud, 2005, Modelling of non-premixed swirl burner flows using a Reynolds-stress turbulence closure, Fuel, 84(5), 583-594.

Hájek, J., J. Vondál, M. Šarlej, and P. Stehlík, 2008, 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.

Kermes, Vít et al. (2008): „Testing of gas and liquid fuel burners for power and process industries“. V: *Energy*. 33 (10), s. 1551-1561.

Kermes, Vít; Skryja, Pavel; Stehlík, Petr (2007): „Up to date experimental facility for testing low-NOx burners“. V: *10th Conference on Process Integration, Modelling and Optimisation for Energy Saving and Pollution Reduction PRES 2007*. Ischia Porto, Italy s. 1-6.

Khelil, A., H. Naji, L. Loukarfi, and G. Mompean, 2008, Prediction of a high swirled natural gas diffusion flame using a PDF model, Fuel, 88(2), 374-381.

Leonard, B. P., and S. Mokhtari, 1990, ULTRA-SHARP nonoscillatory convection schemes for high-speed steady multidimensional flow, NASA; Institute for Computational Mechanics in Propulsion, Lewis Research Center, Case Western Reserve University.[online] Available from: http://purl.access.gpo.gov/GPO/LPS67518

Magnussen, B., and B. Hjertager, 1977, On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion, Symposium (International) on Combustion, 16(1), 719-729.

Menter, F. R., 1994, Two-equation eddy-viscosity turbulence models for engineering applications, AIAA Journal, 32, 1598-1605.

Patankar, S. V., 1980, Numerical Heat Transfer and Fluid Flow, Taylor & Francis, Great Britain.

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

Warnatz, J., R. W. Dibble, and U. Maas, 1996, Combustion: Physical and Chemical Fundamentals, Modelling and Simulation, Experiments, Pollutant Formation, Springer, Berlin.

Weber, R., A. Peters, P. Breithaupt, and B. Visser, 1995, Mathematical-Modeling of Swirling Flames of Pulverized Coal - What Can Combustion Engineers Expect from Modeling, Journal of Fluids Engineering-Transactions of the Asme, 117(2), 289-297.