Applied and Computational Mechanics 9 (2015) 67–78

Applied and Computational Mechanics

Wall heat transfer in gas-fired furnaces: Effect of radiation modelling

J. Vondál[a], J. Hájek[a,*]

[a] *Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, 616 69 Brno, Czech Republic*

Received 9 January 2015; received in revised form 5 May 2015

Abstract

The purpose of this work is to study heat transfer to cooled walls in a MW-scale laboratory furnace with a dominating thermal radiation component. Experiment is performed in a specially designed combustion chamber with segmental water-cooled walls and profile of absorbed heat flux is measured along the flame. Non-premixed natural gas flame is stabilized by a guide-vane swirler. The unsteady governing equations of turbulent flow are solved by a finite-volume code with a two-equation k-ε realizable turbulence model, a combination of first-order and second-order upwind schemes and implicit time integration. The coupling of pressure with velocity is treated by SIMPLE (semi-implicit method for pressure-linked equations) algorithm. Radiative heat transfer as the main heat transfer method is modelled with special care by discrete ordinates method and gas absorption coefficient is calculated by two alternatives of WSGGM (weighted sum of grey gases model). The predicted total heat transfer rate is found to depend strongly on method chosen for the computation of mean beam length. The results of numerical simulations show that overall heat transfer in a process furnace can be successfully predicted, while heat flux profile along the flame is more difficult to predict accurately. Good engineering accuracy is nevertheless achievable with reasonable computational resources. The trend of deviations is reported, which is useful for the interpretation of practical predictions of process furnaces (fired heaters).

Keywords: gas combustion, wall heat flux, swirling diffusion flame, radiative heat transfer, numerical modelling

1. Introduction

The use of gas combustion for the purpose of heating water, steam, crude oil and other media is a very old, yet very important technology till the present day. In all such applications, it is of great importance to ensure that heat loads (dominated by radiative heat fluxes) are distributed across the heat exchanging surfaces evenly. In crude oil heating, excessive heat loads may also lead to fouling on the inner side of tubes (coking). This requirement is important for several reasons: in order to maximize efficiency of the heater (process furnace, boiler), to maximize lifetime of the heater, and to improve safety of the plant operation.

There are essentially two ways how to optimize the design and operation of such fired heaters with respect to the distribution of heat flux. The first option is empirical, based on monitoring of operating parameters and an array of additional instrumentation, used to determine the thermal loading to the tubes. These additional measurements in practice include e.g. thermocouples on tubes, thermal imaging of tube surfaces by pyrometers, spot measurements by heat flux probes or radiometers, etc. They may provide hard evidence, but it is not feasible to make these additional measurements in industrial units in sufficient detail due to the sheer number of local measurements that would be necessary. Also the tuning of operating parameters is a difficult

*Corresponding author. Tel.: +420 541 144 943, e-mail: hajek@fme.vutbr.cz.

task that (in order to really optimize the process) would require impractical (too costly) level of automated control system of the heater.

Second option for the optimization of the fired heater is to use computational fluid dynamics (CFD). Modelling tools that are presently available enable detailed predictions of fired heaters including fuel combustion, radiative heat transfer to the walls, prediction of pollutants emissions and also fouling rates on the process side [9]. State of art combustion models like the one in [6] can provide an alternative to the empirical approach. Practicability of the predictions is however a limiting issue. Therefore in this work, it has been decided to use models and methods with acceptable CPU requirements that are practicable in engineering computations.

Swirling flame is used in the present study as a typical representative of flames used in industrial fired heaters. Swirl-stabilized flames are very popular, especially in the so-called power burners that are widely used in power and process industries [13]. They combine a 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.

It has been recognized for a long time that the prediction of swirling diffusion flames using moment turbulence closures is difficult, although partial successes were reported. For example, a 150 kW TECFLAM burner with movable block type swirler has been modelled by [18] by unsteady Reynolds-Averaged Navier-Stokes (U-RANS) model based on second-order moment turbulence closure, as well as by Large-Eddy Simulation (LES). Authors concluded that the U-RANS method employing a full Reynolds stress model is able to capture unsteady phenomena, such as precessing vortex core phenomenon both qualitatively and in parts also quantitatively. Another work based on the same burner has employed traditional RANS first-order moment closure and authors have shown that close attention to the boundary conditions and converging strategy lead to correct results in the studied case of a strongly swirling flame [5].

Recent progress achieved using large-eddy simulations coupled with advanced chemistry models is on the one hand very promising but on the other hand it is still far from being applicable to industrial problems due to excessive computational requirements. Real-world problems are tractable only using supercomputing facilities due to large 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. Successful predictions of in-flame properties were reported e.g. by [18] for movable block swirler, by [7] for multi-swirl gas turbine and by [11] for Sandia/Sydney swirl burners. All these studies however have been done for relatively simple burners (compared to industrial power burners) and yet they still entailed CPU demands on the order of weeks using tens of CPUs. Thus it is clear that the only viable alternative for practicable predictions of industrial fired heaters in the present as well as in the near future will depend on RANS or U-RANS models.

For the validation of a CFD code it is necessary to possess good-quality measured data as stressed e.g. by [10]. Wall heat flux distribution is one of the parameters that should be included in the validation due to its practical importance. It is however important to make a distinction between various measured values that are reported in the literature. Most of the measurements provide radiative heat flux incident on the measuring probe as in [8]. Some measurements report total heat fluxes, i.e. the probe is cooled and the measured value is the sum of radiative and convective heat flux absorbed by the probe as in [3]. Only rarely are reported values of local heat flux absorbed by the working fluid as in [16] or [21]. In a real application however, one would ideally want to know the distribution of the absorbed heat flux, as irradiation or total heat flux absorbed by several point probes are only qualitatively indicative of the real heat flux density.

Existing meters of absorbed heat flux have been developed mainly for coal-fired boilers, see e.g. [1], where their primary purpose is the monitoring of fouling on tubes, not optimization of boiler or fired-heater performance.

In order to measure the absorbed heat flux, it is best to measure the enthalpy increase of the process medium. This method is used in practice to monitor the total thermal output of a furnace or boiler. However, the same approach may be potentially used to measure local heat transfer using segments of the walls, on the condition that each segment is equipped by flow rate monitoring and temperature sensors. Heat flux profiles may thus be obtained with spatial resolution depending on the number and size of segments. Surprisingly, this method did not receive attention and indeed it has not been implemented until recently, when it was implemented in a large-scale laboratory combustion facility [13]. The method is naturally impracticable in most industrial applications, but it offers great advantages for research [2]. Therefore it has been used in the present work.

There is an appreciable gap between detailed studies of in-flame properties (e.g. [27]) and research done on large-scale furnaces and boilers, where experiments are limited by the restricted access and the sheer size of the units. Moreover, to the knowledge of the authors, laboratory combustion experiments so far did not investigate heat transfer to cooled walls, even though many experiments were done for confined flames. This is one of the main factors that motivated the present work, which aims at providing data from a MW-scale laboratory unit, resembling in many important aspects industrial fired heaters.

The present work thus focuses on measurement of local wall heat fluxes and comparative analysis of computationally manageable predictions. The motivation for such a numerical analysis is to provide validation of utilized numerical models. This is the key factor to trustworthy investigation of industrial furnace.

2. Experiment

The experimental facility is described in detail in [12] and details about the measurement precision can be found in [26].

2.1. Experimental facility

The measurements were performed in experimental facility located at Brno University of Technology. The combustion chamber has a form of a water-cooled horizontal cylinder with 1 m internal diameter and 4 m length (see Fig. 1). The shell of the chamber is divided into seven sections each of which has a separate water inlet and outlet and is equipped with a water flow meter and temperature sensors, allowing for accurate heat transfer rate measurement. A low-NOx staged-gas burner was employed (shown in section 3 below) and fired by natural gas. Flame ignition was performed by a small (25 kW) premixed natural-draft pilot burner. Flame was stabilized by a guide-vane axial swirl generator.

Each section consists of double shell with cooling water inlet at the top and outlet at the bottom. Flow velocity and heat transfer rate inside each section is enhanced by a helical rib which distributes water flow evenly on the flame side wall as shown in [23]. Consequently it suppresses dead zones with potential for increased temperature.

2.2. Wall heat flux measurement

Four experiments were carried out with focus on wall heat flux measurement. Two were done at firing rate of 745 kW (Case 1) and other two at 1 120 kW (Case 2). All the experiments

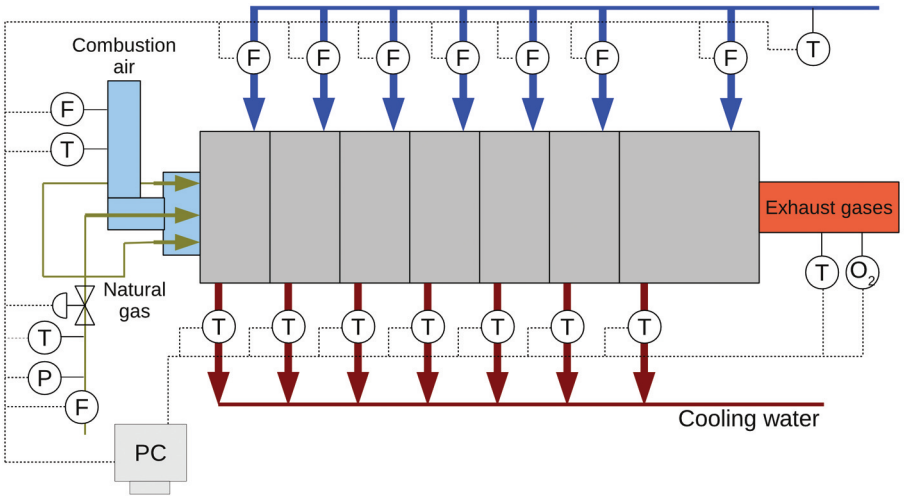

Fig. 1. Combustion facility with data acquisition system

utilized excess air ratio of 1.1 and the same burner. Repeated measurements were performed to verify repeatability of the results. Stabilization of the heat fluxes was established with respect to local wall heat fluxes in all sections of the furnace, which were monitored continuously. After reaching a steady state, the measurement procedure began and data were collected for about 30 min. The resulting heat flux profiles are displayed in Fig. 2, demonstrating excellent repeatability and accuracy (maximum error estimates are shown by error bars).

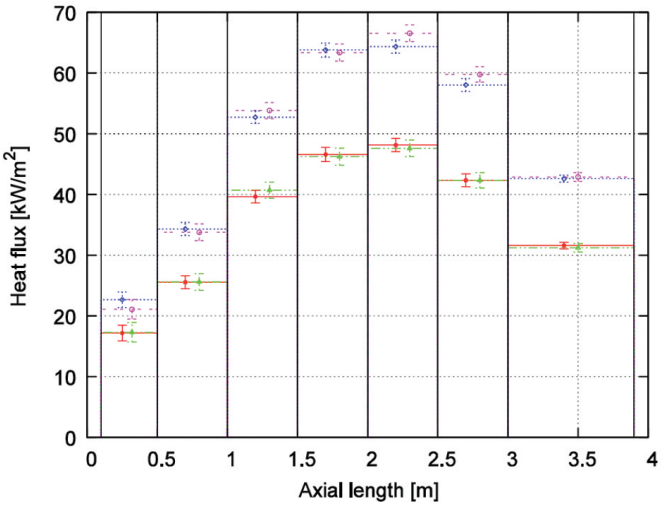

Fig. 2. Measured heat flux profiles along the furnace (burner is on the left end): — and - · - · - represent Case 1, - - - - and · · · · · represent Case 2

Note that natural gas utilized during experiment was in the simulations substituted by methane. The flow rates in simulation were corrected to compensate the difference in heating values between natural gas (used in experiment) and methane (used in simulation). Total mass flow rate of natural gas to the burner was measured during experiment. To be able to identify mass flow rates through the two gas stages (nozzles) individually a separate preliminary CFD simulation of gas distributor inside of the burner was carried out. Heat transfer rate to the cooling water was calculated from calorimetric equation for each section based on measured flow rate and temperature difference.

3. Modelling

For each of the two cases several combustion simulations were performed. The problem was carefully set up taking into account recent results of a related investigation [24]. Thermal boundary condition on the water-cooled walls was treated by prescribed temperature of 80 °C (water side of inner steel shell) based on previous work published in [23], while adiabatic condition was used for all the remaining walls, which are in reality covered by fibrous aluminosilicate thermal insulation. For swirl generation, this work adopts a common practice used in engineering computations, which means that air supply duct is included into the computational domain along with all important features, including the swirler.

3.1. Flow domain and computational grid

The geometry of the modelled flow domain includes several meters of air supply duct, the burner, and combustion chamber. It was necessary to include the air duct as air enters the burner from a side and the burner does not incorporate any measures (e.g. honeycomb) that would homogenize the flow. In preliminary cold-flow computations it was confirmed that due to this inlet asymmetry air flow in the burner is far from axially symmetrical.

Gas enters the computational domain at nozzle orifices. The diameter of nozzles has been increased in order to compensate nozzle area reduction after discretization. Grid was created with great care to ensure high quality. Most of the volume of computational domain was meshed by hexahedral cells and only in the vicinity of nozzles were used tetrahedral elements, as illustrated by Fig. 3. Total number of grid cells was approximately 1 300 000.

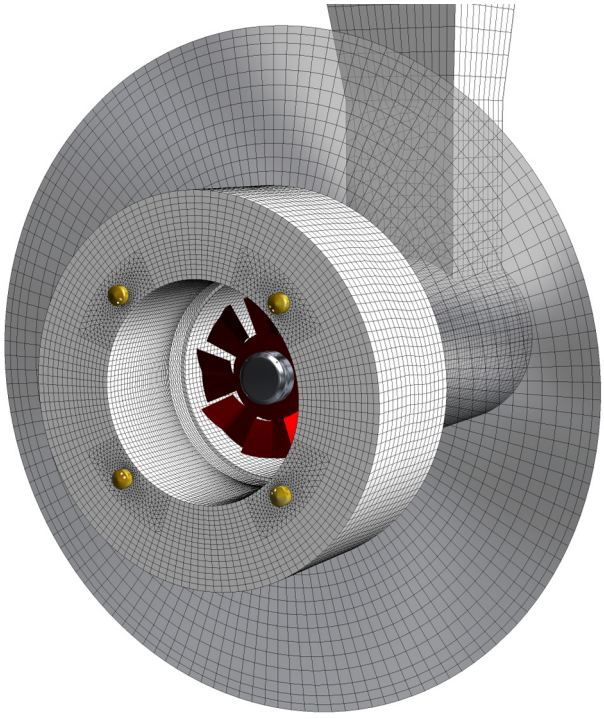

Fig. 3. Detail of mesh around the burner orifice

3.2. Computational model setup

The objective of this work is to validate the performance of a CFD model that has acceptable computational requirements and thus can be applied in the industrial practice. Therefore an industry-standard two-equation turbulence model was applied, namely k-ε realizable model. Simulations were run in unsteady mode since combustion in this complex geometry displays large-scale transient fluctuations (due to precessing flame vortex). To account for turbulence-chemistry interactions and combustion, eddy-dissipation model [14] has been employed. This model falls into the family of eddy breakup models and therefore combustion occurs as soon as fuel and oxidants are mixed by turbulence. Due to this simplifying assumption of the chemistry model, single-step reaction mechanism was appropriate.

Solver settings were kept the same for all of the test cases as specified in Tables 1 and 2. Time step was set to 2 ms in order to allow the solver to perform from ten to twenty iterations per time step before reaching convergence. Instantaneous data were averaged over more than 2 s of simulated physical time to calculate time-averaged results.

Table 1. 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 2. Discretization methods used in all simulations

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

3.3. Radiative heat transfer modelling

Compromise in accuracy of modelling of absorption coefficient between simplified grey gas model and narrow band models is traditionally represented by the weighted-sum-of-grey-gases model (WSGGM), with coefficients that were first published in [19]. The WSGGM approach is suitable for overall heat transfer predictions in furnaces as noted by [17]. Detailed comparison of WSGG model to the correlated k-distribution model (CK) and wide band correlated k-distribution (WBCK) model was published by [20]. It shows that WBCK model is significantly

more accurate over WSGG and is not as CPU intensive as CK. Overall error due to WSGG may reach up to 30 % [4]. However, that work utilized WSGG model with the cell-based mean beam length, which may be responsible for a great part of the observed deviation, as shown below.

Significant difference between cell-based and domain-based WSGG was observed by [20]. The term cell- and domain-based describes the way how the mean beam length is determined. The author shows that cell-based WSGG significantly over-predicts both the total wall heat fluxes and radiative source term when compared to domain-based WSGG and non-grey wide band correlated-k method (WBCK).

In the present computations was applied a new set of parameters for WSGGM, adopted from [28]. Contrary to the standard WSGGM, the new coefficients are suitable even for oxy-fuel combustion. The authors show that their model predicts wall heat fluxes more accurately; the validation was made with experimentally obtained in-flame temperature. In the cell-based method, a characteristic beam length is calculated for each cell. This procedure is straightforward and well suited to unstructured grids. On the other hand the domain-based procedure is mesh independent, but requires the characterization of domain size by a single value. The procedure from [28] defines the domain size parameter as $L = V_{\mathrm{domain}}/A_{\mathrm{domain}}$, where L is mean beam length, V_{domain} is volume of the computational domain and A_{domain} is the surface area of the domain. In the present analysis, the mean beam length was equal to 3.6 m.

4. Results and discussion

The results of combustion experiment are discussed first to emphasize the properties of the measured data. Predictions are then discussed from various aspects on the basis of the hard evidence provided by the measured data.

4.1. Discussion of measured data

It is important to note that the present measurements constitute a significant technical challenge because of the size and complexity of the combustion facility. In such large-scale furnace (roughly 1 MW flames) it is much more difficult to ensure repeatability of all boundary conditions and thus also of the measured values than in small-scale laboratory experiments (typically less than 150 kW). This challenge has however been tackled quite successfully, as proven by the repeatability shown in Fig. 2. In repeated experiments, values of the global heat flux differed by less than 1 % and repeatability of the local heat fluxes was within 3 %. The source of the existing small differences was related mostly to the uncertainty in measuring air flow rate (about 10 %) and much less to the uncertainty of heat flux measurement (on average about 3 %). Those uncertainties of local wall heat fluxes are plotted in Figs. 2, 4, 5 and 6 as error bars. Furthermore, the heat balance of the furnace has been closed within 1 % margin of unaccounted heat losses. Accuracy of the measurement has been improved by decreasing flow rate of the cooling water, yet keeping it sufficiently high to prevent boiling, as follows from error propagation analysis that was done in [25].

Along with good accuracy, the present segmental measurement of heat flux provides several advantages. Firstly it is the inherent averaging of heat flux over each of the segments and to some extent also over the measurement time period. The measured value of local heat flux in each segment of the furnace is in fact spatial average over each individual segment. It is also a floating average in time thanks to heat capacity of the steel shell and cooling water. Positive effect of the spatial and temporal averaging is that it eliminates the influence of numerous sources of error and irregularities (including non-symmetry of the flame, flux non-uniformity

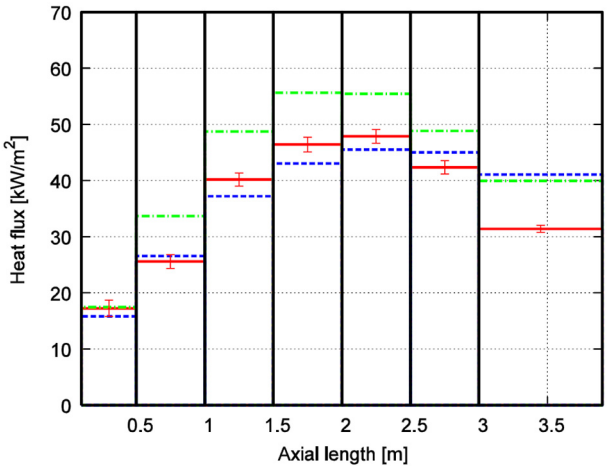

Fig. 4. Effect of the different methods for absorption coefficient calculation – Case 1: — measurement, - · - · cell-based and standard WSGG, - - - - domain-based and standard WSGG

and fluctuations) that influence most point methods of heat flux measurement (e.g. ellipsoidal radiometers, Schmidt-Boelter heat flux sensors, etc.).

Previous studies that reported point measurements of heat flux in combustion chambers like [8] are unable to relate the measured heat fluxes to the overall balance of the furnace. Such measurements must therefore be interpreted mostly qualitatively and are useful mainly in adiabatic combustion chambers or for the monitoring of fouling in boilers. The present method however produces data, which provide a heat flux profile that inherently conserves the global energy balance. It is therefore practically very useful.

In Case 1, the total heat transferred to cooling water was 58 % of the total firing rate and in Case 2, the transferred heat was 52 % of the firing rate. Measured heat flux profiles are displayed in Fig. 2.

4.2. Effect of mean beam length

Absorption coefficient is the parameter that defines to a large extent radiative heat transfer in the combustion chamber. The WSGG model needs as one of the inputs (besides gas temperature and composition) also a mean beam length. This geometrical parameter is typically calculated by one of two methods as described above in section 3, but it normally receives very little attention in most combustion simulation studies. Industrial analysts and researchers therefore have very little guidance on the choice between the domain-based and cell-based beam length calculation approach. The present work addresses this issue and provides data for informed choice of this important parameter.

The beam length enters calculation of absorption coefficient in two places [28]. First is the emissivity, which grows with rising beam length asymptotically to unity. Emissivity is calculated from empirical absorption coefficients of imaginary gases and multiplicative parameters that represent the influence of temperature and composition. The resulting total emissivity is used to calculate a representative absorption coefficient. The absorption coefficient is inversely proportional to the mean beam length. Greater beam length therefore means higher emissivity (optical thickness), but only the simulation results show (in Fig. 4) that this actually leads to lower absorption coefficient and lower radiative heat transfer to the cooled walls.

Fig. 4 shows that the prediction of Case 1 using a cell-based beam length calculation method leads to a substantially more intensive heat transfer to the walls (total transferred heat about

17.6 % above the measured total heat flux). To explain this behaviour, we can look at the average value of absorption coefficient (volume averaging over the entire combustion chamber). The average value of absorption coefficient is almost an order of magnitude higher for the cell-based beam length approach than for the domain-based approach (about 2.3 and 0.4, respectively). High absorption coefficient in the cell-based approach makes the flame more luminous and this enhances heat transfer to the walls.

It is therefore clear that the present work leads to a recommendation to use a domain-based beam length calculation method, which predicts total heat transfer to the walls 1.7 % and 2.4 % above the measured values for Case 1 and 2, respectively. In both cases the total heat transfer to cooled walls is slightly over-predicted. Local heat transfer rates in individual segments display larger deviations, generally the predicted profiles are flatter and peak loads are shifted slightly farther from the burner.

4.3. *Effect of WSGGM coefficients*

In order to further analyse the effect of the gas radiative properties, standard WSGG method that uses empirical coefficients published by [19] has been substituted in simulation of the two Cases 1 and 2 by a newer set of coefficients published by [28]. The work of [28] compared WSGG predictions with a significantly more accurate spectral method (exponential wide band model), but employed the cell-based beam length and did not offer any validation by measured heat fluxes.

The application of this new set of WSGG coefficients together with the domain-based approach has resulted in a further decrease of predicted heat transfer to the furnace walls by about 7 %. The predicted values of total heat flux in both Case 1 and 2 are slightly below the measured values by 5 % and 2.7 %, respectively. See Figs. 5 and 6 for segmental profiles of heat transfer.

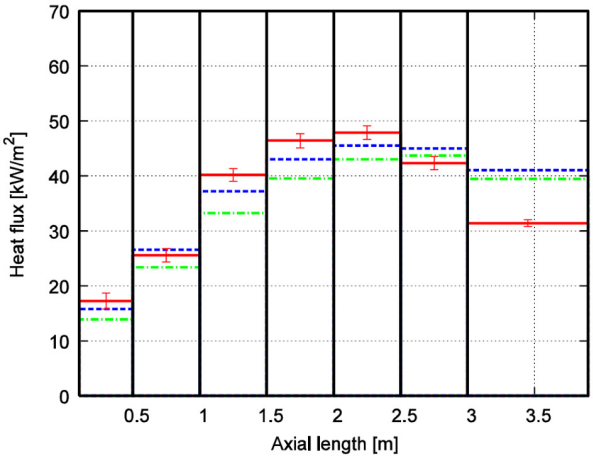

Fig. 5. Effect of the different WSGG coefficients – Case 1: — measurement, - - - - domain-based and standard WSGG [19], - · - · domain-based and new WSGG [28]

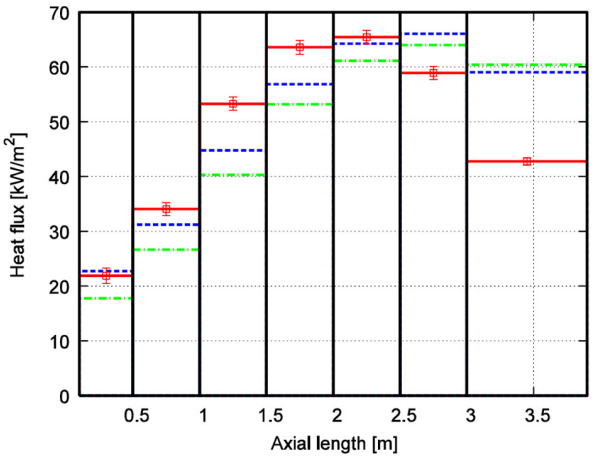

Fig. 6. Effect of the different WSGG coefficient calculation – Case 2 — measurement, - - - - domain-based and standard WSGG, - · - · - domain-based and new WSGG

4.4. *Flame length*

The main qualitative difference of the predictions provided by the three simulation models, as may be observed from Figs. 4 and 5, is the flame length as reflected by the position of peak heat

Table 3. Predicted characteristic values in the furnace volume

		Volume averaged temperature [°C]	Maximum temperature [°C]	Outlet temperature [°C]	Absorption coefficient [m^{-1}]	Total wall heat flux [kW]
Case 1	Cell-based	695.84	2 045.93	683.47	2.32	515.30
	Domain-based	829.24	2 040.82	875.04	0.44	445.70
	Domain-based, new WSGGM	888.93	2 039.04	924.47	0.30	416.13
Case 2	Domain-based	938.00	2 094.70	949.20	0.41	608.50
	Domain-based, new WSGGM	915.90	1 953.70	1 011.20	0.29	577.90

flux along the furnace. We may even generalize this observation slightly by including results obtained in the case of the same flames for different values of wall emissivity 2. Decreasing wall emissivity had a similar effect of shifting the peak of heat flux profile towards the furnace exit. These two observations lead to the conclusion that similar effect (lower and more flat heat flux profile) occurs upon decreasing radiative heat transfer to the walls by any means (acting globally). For an overview of global predicted parameters see Table 3.

The flame shape and consequently shape of the heat flux profile is however greatly influenced by turbulence model. In the present analysis, two-equation turbulence model has been used as is typical in practical simulations. The present problem however features a very complex flow with swirl generated by a vane swirler along with a sudden expansion of the flow channel on entrance into the combustion chamber. Such swirling flows are well known to be very difficult to predict with moment closure methods, see e.g. [18]. Neither eddy-viscosity models nor full Reynolds-stress models are able to predict the solid-body rotation occurring in the central region of similar strongly swirling flows [15]. We may thus conclude that flattening of the heat flux profile and prolongation of the flame is probably a feature shared by predictions employing moment-closure turbulence models and should not be attributed to the radiation model. Rather, radiation model is primarily responsible for transferring the right amount of heat to the cooled walls.

It is worth noting that in the last section (at the outlet side of the furnace, widest of all segments), the difference of measured and predicted heat flux is the largest. In line with the previous arguments the authors believe, that this is the consequence of the turbulence model, which does not predict the correct flow pattern in the combustion chamber and is probably responsible for the greatest part of discrepancies that are contained in the simulation results. But of course, there are also other factors contributing to the qualitative differences, like simplified chemistry model or numerical errors due to false diffusion.

5. Conclusions

The present work has analysed combustion and heat absorption to walls in a water-cooled large-scale combustion facility equipped with a staged-gas burner. Computationally manageable CFD model has been set up to examine predictive capabilities for similar industrial applications. The main outputs of the work may be summarized as follows:

- Heat flux profiles were measured by monitoring the process medium (water) with a good accuracy and repeatability, both about 3 %. Measured heat flux profiles were obtained for two firing rates (745 kW and 1 120 kW).

- A computationally manageable model has been shown to yield satisfactory overall heat transfer rate predictions.
- Method of computing mean beam length has been shown to have a major impact on the overall heat transfer rate. The domain-based method has been shown to give superior results when compared to the cell-based method.
- New set of WSGGM coefficients adopted from [28] has been validated. It led to slight under-prediction of the total measured heat transfer rate (by 3 and 5 %), decreasing total heat transfer by about 7 % as compared to standard WSGGM coefficients from [19]. The standard WSGGM coefficients yielded slightly more accurate overall heat transfer rates (over-prediction by about 2 %).
- Flame length was slightly over-predicted in all of the simulations and predicted heat flux profiles were flatter than the measured, which has been attributed to modelling simplifications.

Acknowledgements

The research leading to these results has received funding from the Ministry of Education, Youth and Sports under the National Sustainability Programme I (Project LO1202). This work was also supported by the project CZ.1.07/2.3.00/30.0039 of Brno University of Technology.

Reference

[1] Beckmann, M., Krüger, S., Spiegel, W., A method for non-invasive heat flux detection in membrane walls of steam generators, Proceedings of the Annual International Conference on Incineration and Thermal Treatment Technologies, Savannah, USA, Air and Waste Management Association, 2006, pp. 531–550.

[2] Bělohradský, P., Skryja, P., Hudák, I. Experimental study on the influence of oxygen content in the combustion air on the combustion characteristics, Energy 75 (2014) 116–126.

[3] Butler, B. W., Denison, M. K., Webb, B. W., Radiation heat transfer in a laboratory-scale, pulverized coal-fired reactor, Experimental Thermal and Fluid Science, 9 (1) (1994) 69–79.

[4] Coelho, P. J., Numerical simulation of radiative heat transfer from non-gray gases in three-dimensional enclosures, Journal of Quantitative Spectroscopy and Radiative Transfer 74 (3) (2002) 307–328.

[5] Frassoldati, A., Frigerio, S., Colombo, E., Inzoli, F., Faravelli, T., Determination of NO_x emissions from strong swirling confined flames with an integrated CFD-based procedure, Chemical Engineering Science 60 (11) (2005) 2 851–2 869.

[6] Fureby, C., LES of a ulti-burner annular gas turbine combustor, Flow, Turbulence and Combustion 84 (3) (2010) 543–564.

[7] Fureby, C., Grinstein, F. F., Li, G., Gutmark, E. J., An experimental and computational study of a multi-swirl gas turbine combustor, Proceedings of the Combustion Institute 31 (2) (2007) 3 107–3 114.

[8] Hayes, R. R., Brewster, S., Webb, B. W., McQuay, M. Q., Huber, A. M., Crown incident radiant heat flux measurements in an industrial, regenerative, gas-fired, flat-glass furnace, Experimental Thermal and Fluid Science 24 (1–2) (2001) 35–46.

[9] Hu, G., Wang, H., Qian, F., Van Geem, K. M., Schietekat, C. M., Marin, G. B., Coupled simulation of an industrial naphtha cracking furnace equipped with long-flame and radiation burners, Computers & Chemical Engineering 38 (2012) 24–34.

[10] Jaluria, Y., Challenges in the accurate numerical simulation of practical thermal processes and systems, International Journal of Numerical Methods for Heat and Fluid Flow 23 (1) (2013) 158–175.

[11] James, S., Zhu, J., Anand, M. S., Large eddy simulations of turbulent flames using the filtered density function model, Proceedings of the Combustion Institute 31 (2) (2007) 1 737–1 745.

[12] Kermes, V., Bělohradský, P., Biodiesel (EN 14213) heating oil substitution potential for petroleum based light heating oil in a 1 MW stationary combustion facility, Biomass and Bioenergy 49 (2013) 10–21.

[13] Kermes, V., Bělohradský, P., Oral, J., Stehlík, P., Testing of gas and liquid fuel burners for power and process industries, Energy 33 (10) (2008) 1 551–1 561.

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

[15] Mitrofanova, O. V., Hydrodynamics and heat transfer in swirling flows in channels with swirlers (analytical review), High Temperature 41 (4) (2003) 518–559.

[16] Paist, A., Poobus, A., Tiikma, T., Probes for measuring heat transfer parameters and fouling intensity in boilers, Fuel 81 (14) (2002) 1 811–1 818.

[17] Saario, A., Oksanen, A., Effect of computational grid in industrial-scale boiler modeling, International Journal of Numerical Methods for Heat & Fluid Flow 19 (1) (2009) 93–117.

[18] Sadiki, A., Maltsev, A., Wegner, B., Flemming, F., Kempf, A., Janicka, J., Unsteady methods (URANS and LES) for simulation of combustion systems, International Journal of Thermal Sciences 45 (8) (2006) 760–773.

[19] Smith, T. F., Shen, Z. F., Friedman, J. N., Evaluation of coefficients for the weighted sum of gray gases model, Journal of Heat Transfer 104 (4) (1982) 602–608.

[20] Ströhle, J., Spectral modelling of radiative heat transfer in industrial furnaces, PhD Thesis, University of Stuttgart, Shaker Verlag GmbH, Aachen, 2004.

[21] Taler, J., Duda, P., Weglowski, B., Zima, W., Gradziel, S., Sobota, T., Taler, D., Identification of local heat flux to membrane water-walls in steam boilers, Fuel 88 (2) (2009) 305–311.

[22] Vondál, J., Computational modeling of turbulent swirling diffusion flames, PhD Thesis, Brno University of Technology, 2012.

[23] Vondál, J., Hájek, J., Boundary condition evaluation and stability issues in swirling flame gas combustion, Proceedings of the conference ThermaComp2009, Naples, Italy, 2009, pp. 314–317.

[24] Vondál, J., Hájek, J., Experimental and numerical investigation of swirling non-premixed gas flames in industrial-scale furnace, Proceedings of the 9th European Conference on Industrial Furnaces and Boilers, Estoril, 2011.

[25] Vondál, J., Hájek, J., Experimental data of the local wall heat flux for combustion codes validation in non-premixed swirling gas flames, Proceedings of the ASME-ATI-UIT 2010 Conference on Thermal and Environmental Issues in Energy Systems, Sorrento, 2010, pp. 931–936.

[26] 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, Chemical Engineering Transactions 21 (2) (2010) 1 123–1 128.

[27] Xiouris, C. Z., Koutmos, P., Fluid dynamics modeling of a stratified disk burner in swirl co-flow, Applied Thermal Engineering 35 (2012) 60–70.

[28] Yin, C., Johansen, L. C. R., Rosendahl, L. A., 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, Energy & Fuels 24 (12) (2010) 6 275–6 282.