CHEMICAL ENGINEERING TRANSACTIONS Volume 21, 2010
Editor J. J. Klemeš, H. L. Lam, P. S. Varbanov
 ISBN 978-88-95608-05-1 ISSN 1974-9791
DOI: 10.3303/CET1021186

Wall Heat Fluxes in Swirling Combustion of Extra Light Fuel Oil in Large Scale Test Combustor: Experiment and Modelling Using Eddy Dissipation Model

Jakub Broukal*, Jiří Hájek
Institute of Process and Environmental Engineering, Faculty of Mechanical Engineering, Brno University of Technology, VUT - UPEI, Technická 2, 616 69 Brno, Czech Republic
broukal@upei.fme.vutbr.cz

This paper presents a preliminary study which aims to compare wall heat fluxes in swirling combustion of fuel-oil in large-scale combustor obtained both by experimental measurement and computer simulation. Prediction of wall heat fluxes is of crucial importance when designing industrial furnaces. The first part of the paper presents the experimental facilities and the method used to gather and analyze the measured data.
In the second part a Computational Fluid Dynamics (CFD) simulation of the spray combustion is performed. The liquid phase is modelled using Discrete Phase Model (DPM) and the flow is solved by Reynolds-averaged Navier-Stokes (RANS) models. Various submodels are used to account for turbulence, radiation, fuel atomization and evaporation. The obtained results, i.e. means of wall heat fluxes, are finally compared with the experimental data and deviations together with possible explanations and improvements are discussed. Results show discrepancies which need to be addressed in future research.

1. Introduction

When dealing with spray combustion, one has to deal with two very challenging tasks – spray formation and combustion. Liquid sprays can be generated by various atomizers. For combustion purposes, as in this case, effervescent atomizers (twin-fluid atomizer with internal mixing) are gaining on popularity. This type of atomizer was introduced by Lefebvre (1988).
The most common spray modelling approach nowadays is the Euler-Lagrange method, which is used in many studies, e.g. (Broukal et al., 2010; Calay and Holdo, 2008; Qian et al., 2009; Wang and Baek, 2007; Xiong et al., 2009). In this approach the gas phase is modelled as a continuum whereas the liquid phase is treated as a system of discrete particles (droplets) that are tracked in the gas flow field. It is therefore necessary to use appropriate models for primary and secondary breakup (to determine initial droplet parameters like diameter, velocity and direction) as well as for all other processes concerning the droplets, like momentum, heat and mass transfer (evaporation). This is

Please cite this article as: Broukal J. and Hájek J., (2010), Wall heat fluxes in swirling combustion of extra light fuel oil in large scale test combustor: Experiment and modelling using Eddy Dissipation Model, Chemical Engineering Transactions, 21, 1111-1116 DOI: 10.3303/CET1021186

the approach adopted in the present work. A review of models and advanced methods used in spray modelling can be found in (Jiang et al., 2010).
Concerning practical combustion applications many papers can be found. In their work, de Jager and Kok (2004) present a simple model which assumes complete fuel pre-vaporization. A more complex approach is employed by Hai-Wen Ge (2006), where turbulence-chemistry interactions are treated in detail using PDF approach. The troubling issue of radiation and absorption coefficient in spray combustion is addressed in (Baek et al., 2002; Choi and Baek, 1996). One of the biggest concerns in the combustion industry is to be able to predict wall heat fluxes. This issue has been recently investigated for the case of methane combustion e.g. in (Vondál and Hájek, 2009), but in the case of spray combustion more research is needed.
The scope of this work is to perform a preliminary study on the viability of basic spray and combustion models that are going to be used in future research in the area of vegetable oil combustion.

2. Data Analysis and Experiments

This work reports data obtained from two different experiments. In the first the effervescent atomizer was analyzed by means of drop size distribution. This information later served as a starting point in defining the spray initial conditions in the CFD model. The purpose of the second experiment was to collect wall heat flux data in a large scale combustion chamber.

2.1 Spray measurement and Data Processing

The measured spray of extra-light fuel-oil was generated using the effervescent atomizer and operating conditions described in (Jedelský et al., 2009) as configuration E38. Drop sizes and drop velocities were measured using a Dantec phase/Doppler particle analyzer (P/DPA) in 6 radially equidistant sampling points at 150 mm from the atomizer orifice. The drawing in Figure 1 shows the measurement points in a half-angle of the spray (between the axis and the farthermost measurement point). A detailed description of the measurement can be found in (Jedelský et al., 2009).

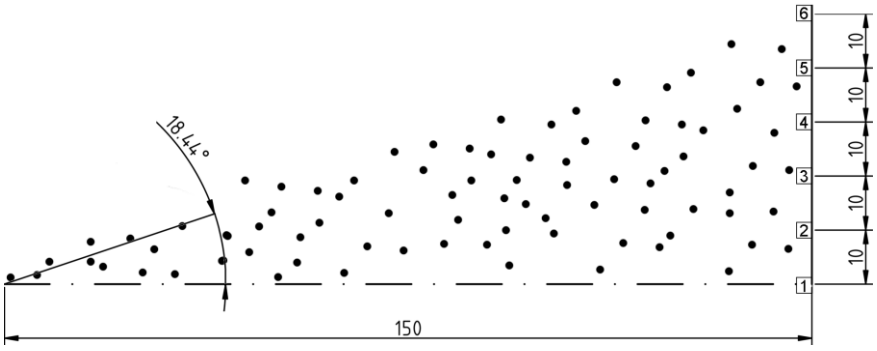

Figure 1 Schematics of the spray measurement

For the purpose of data analysis a software with graphical user interface was created using MATLAB programming environment. The software was designed for the processing of experimental data from multiple measuring points as generated by the

measuring device. It can display drop size and mass histograms together with representative diameters and the user can choose from a variety of analytical functions to fit the experimental data. Detailed results of the experimental analysis and fitting of the data have been published in (Broukal et al., 2010). Rosin-Rammler distribution was used despite the fact it is not the best unimodal fit, because Ansys Fluent (used for flow modelling in this work) is equipped with a pre-prepared procedure to discretize this particular distribution function (Broukal et al., 2010).

2.2 Large-scale Combustion Facility

The combustion experiment has been performed in a water-cooled horizontal combustion chamber (1 m internal diameter and 4 m length). 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. The experimental facility is described in detail by (Kermes and Bělohradský, 2008; Kermes et al., 2007). The fuel was atomized in an effervescent atomizer described in the previous section.

In order to reduce the fuel consumption, the combustion chamber was preheated using methane gas. Then the fuel and air parameters were set according to Table 1.

Table 1 Experiment parameters

Fuel mass flow	78.48 [kg/h]
Atomizing air mass flow	7.85 [kg/h]
Gas-Liquid ratio (GLR)	10 %
Combustion air mass flow	1280 [m^3/h]
Air equivalence ratio	1.46
Fuel density	820.7 [kg/m^3]

The steadiness of the experiment was judged according to the stability of local wall heat fluxes in all sections of the furnace. After reaching a steady state, the measurement procedure began and data were collected for about 30 min.

3. Modelling

The modelling part of the work was performed using commercial CFD code Ansys Fluent (Ansys Fluent, 2009). The main goal of these simulations was to predict wall heat fluxes of the combustion chamber. For the purposes of numerical analysis a mesh was constructed in the software Gambit. The total number of computational cells (97 % of which are hexahedral) was nearly 1,200,000. Four different boundary conditions were applied – mass flow inlet (for combustion air), pressure outlet, prescribed temperature on the water-cooled walls and adiabatic condition for the remaining walls.

The flow field was obtained by solving the RANS equations together with Eddy Dissipation Model (EDM) to account for turbulence chemistry interactions (Magnussen and Hjertager, 1977). Turbulence was modelled using k-ε realizable model. Radiation was accounted for by the Discrete Ordinates Model (DOM). The absorption coefficients were obtained using the domain-based approach of the Weighted Sum of Grey Gases Model (WSGGM). The fuel droplets were modelled using the DPM as discrete

Lagrangian entities – particles. Ansys Fluent offers a variety of atomizer models and injections. Unfortunately, Ansys Fluent does not offer any atomizer model that corresponds to the atomizer used in the experiments; therefore it was decided to use a so-called solid cone injection instead. The Rosin-Rammler distribution parameters of the injection where found using the previously mentioned MATLAB code. 40 drop sizes were chosen for the simulations and to each size 200 particle streams have been assigned. For detailed information about the injection setup and spray discretization see (Broukal et al., 2010). The particles were tracked in an unsteady fashion. To predict the particle trajectory, one has to integrate the force-balance equation, which can be written (for the *x* direction in Cartesian coordinates) as follows:

$$\frac{du_p}{dt} = F_D\left(u - u_p\right) + \frac{g_x\left(\rho_p - \rho\right)}{\rho_p} + F_x\,, \tag{1}$$

where u_p is the particle velocity, u the surrounding air flow velocity, F_x and g_x is an additional acceleration in x direction and gravity respectively. $F_D(u- u_p)$ is the drag force per unit particle mass. The shape of drops is assumed to be spherical and the drag force was calculated using the formula that reads (Ansys Fluent, 2009)

$$F_D = \frac{18\mu}{\rho_p d^2}\frac{C_D\,\mathrm{Re}}{24}\,, \tag{2}$$

where d is drop diameter, μ is the molecular viscosity of the fluid (air) and

$$C_D = a_1 + \frac{a_2}{\mathrm{Re}} + \frac{a_3}{\mathrm{Re}^2}\,. \tag{3}$$

The constants a_1, a_2, a_3 apply to smooth spherical particles over several ranges of Re given by Morsi and Alexander (Morsi and Alexander, 1972).
In order to take into account the turbulent flow effects on particle motion, the Discrete Random Walk (DRW) model has been applied. The DRW model simulates interactions of a particle with a succession of discrete stylized fluid phase turbulent eddies.
The discrete phase exchanges momentum, mass (evaporation) and energy with the continuous phase. Secondary atomization, drop collisions and coalescence have not been included similarly as in (Broukal et al., 2010).

4. Results and Discussion

In Figure 2 can be seen that the wall heat fluxes obtained from simulations do not agree well with the experimental measurements. The simulation peak occurs between the 5th and 6th section while the experiment suggests the peak is around the 4th section. The simulation also under predicts the maximal wall heat flux. One of the main reasons of these discrepancies is probably the representation of the effervescent spray. The figure

suggests that the smallest drops might be missing and therefore it takes longer for the spray to evaporate and subsequently burn, thus moving the peak further downstream.
Another possible cause could be the simplification of the effervescent atomizer. The used model does not take into account the atomizing air exiting the atomizer nozzle together with the liquid drops. Although the flow rate of the atomizing air is very small compared to the combustion air (0.5 %), it might have important effects on the mixing process of the evaporated fuel with air. This issue is closely related to turbulence modelling, which could also have major effects on the predicted wall heat fluxes as shown in (Vondál and Hájek, 2009).

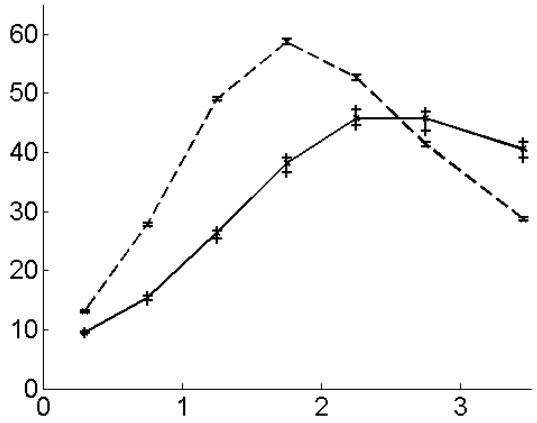

Figure 2 Dependence of wall heat flux [kW/m^2] on axial distance [m]. Solid line represents Fluent simulation and dashed line represents experimental results.

5. Conclusion and Future work

A preliminary study has been presented whose scope was to test simple CFD models in order to predict wall heat fluxes in fuel-oil fired combustion chambers. A MATLAB code was developed to analyze and discretize experimental spray data for combustion modelling purposes and a CFD simulation was performed. The discrepancies between prediction and experiment suggest that more complex models need to be used.
In the oncoming research emphasis will be placed on atomization models that are able to predict small drop diameters and more complex chemistry (Steady Flamelet Model) and turbulence models (Reynolds Stress Model) will be employed.

6. 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".

References

Ansys Fluent, 2009, Ansys Fluent 12.1 Documentation.

Baek, S., H. Kim, M. Yu, S. Kang, and M. Kim, 2002, Application of the extended weighted sum of gray gases model to light fuel oil spray combustion, Combustion Science and Technology, 174(7), 37-70.

Broukal, J., J. Hájek, and J. Jedelský, 2010, Effervescent atomization of extra-light fuel-oil: Experiment and statistical evaluation of spray characteristics, in ILASS Proceedings, Brno, Czech Republic. 23.

Calay, R., and A. Holdo, 2008, Modelling the dispersion of flashing jets using CFD, Journal of Hazardous Materials, 154(1-3), 1198-1209.

Hai-Wen Ge, 2006, Probability Density Function Modeling of Turbulent Non–reactive and Reactive Spray Flows, PhD Thesis, Ruprecht–Karls–Universitat Heidelberg, Germany

Choi, C., and S. Baek, 1996, Numerical analysis of a spray combustion with nongray radiation using weighted sum of gray gases model, Combustion Science and Technology, 115(4-6), 297-315.

de Jager, B., and J. B. W. Kok, 2004, Application of the first combustion model to spray combustion, Applied Thermal Engineering, 24(10), 1481-1489.

Jedelský, J., M. Jícha, J. Sláma, and J. Otáhal, 2009, Development of an Effervescent Atomizer for Industrial Burners, Energy & Fuels, 23 (12), 6121–6130

Jiang, X., G. Siamas, K. Jagus, and T. Karayiannis, 2010, Physical modelling and advanced simulations of gas-liquid two-phase jet flows in atomization and sprays, Progress in Energy and Combustion Science, 36(2), 131-167.

Kermes, V., and P. Bělohradský, 2008, Testing of gas and liquid fuel burners for power and process industries, Energy, 33, 1551-1561.

Kermes, V., P. Skryja, and P. Stehlík, 2007, Up to date experimental facility for testing low-NOx burners, Chemical Engineering Transaction, 12, 549 - 554

Lefebvre, A., X. Wang, and C. Martin, 1988, Spray Characteristics of Aerated-Liquid Pressure Atomizers., Journal of Propulsion and Power, 4(4), 293-298.

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.

Morsi, S., and A. Alexander, 1972, Theoretical Low-Speed Particles Collision with Symmetrical and Cambered Aerofoils., ASME Pap, (72 -WA/FE-35).

Qian, L., J. Lin, and H. Xiong, 2009, Simulation of Droplet-gas Flow in the Effervescent Atomization Spray with an Impinging Plate, Chinese Journal of Chemical Engineering, 17(1), 8-19.

Vondál, J., and J. Hájek, 2009, Experimental and numerical analysis of wall heat transfer in non-premixed gas combustor, Chemical Engineering Transactions, 18, 587-592.

Wang, T., and S. Baek, 2007, Numerical modelling of unsteady spray behaviour in moderately high-pressure regime, Combustion Theory and Modelling, 11(1), 21-46.

Xiong, H., J. Lin, and Z. Zhu, 2009, Three-dimensional simulation of effervescent atomization spray, Atomization and Sprays, 19(1), 75-90.