TECHNICAL TRANSACTIONS | **CZASOPISMO TECHNICZNE**
CIVIL ENGINEERING | BUDOWNICTWO

5-B/2014

MATEUSZ SZUBEL, TOMASZ SIWEK*

THE APPLICATIONS OF NUMERICAL MODELING FOR THE OPTIMIZATION OF THE OPERATION OF ENERGY DEVICES ON THE EXAMPLE OF AN AIR DISTRIBUTION SYSTEM INSIDE THE BIOMASS BOILER

MOŻLIWOŚCI ZASTOSOWANIA MODELOWANIA NUMERYCZNEGO W PROCESIE OPTYMALIZACJI PRACY URZĄDZEŃ ENERGETYCZNYCH, NA PRZYKŁADZIE SYSTEMU DYSTRYBUCJI POWIETRZA KOTŁA NA BIOMASĘ

Abstract

The paper addresses the results of a numerical – based on experimental data – study of air distribution in the manifold responsible for air supply into a combustion chamber of a biomass boiler. An analysis of the possibilities of optimizing the air distribution system using a ommercial CFD program by changing the diameter of the feeding ducts was performed and described.

Keywords: Renewable energy; Straw combustion; CFD; Air distributor optimization

Streszczenie

Artykuł opisuje wyniki opartego na danych eksperymentalnych modelowania numerycznego kolektora powietrznego, odpowiedzialnego za dostarczanie powietrza do komory spalania kotła na biomasę. Przeprowadzono analizę możliwości optymalizacji systemu rozprowadzania powietrza z wykorzystaniem komercyjnego programu z grupy CFD poprzez zmianę średnic rur dolotowych.

Słowa kluczowe: energia odnawialna, spalanie słomy, obliczeniowa mechanika płynów, optymalizacja kolektora powietrznego

---

* M.Sc. Eng. Mateusz Szubel, M.Sc. Eng. Tomasz Siwek, AGH University of Science and Technology, Faculty of Energy and Fuels.

Designations

σ – standard deviation
N – total number of air feeding ducts
X_i – air mass flow of i duct

1. Introduction

1.1. The process of combustion of solid biomass in energy appliances

The growing interest in renewable energy resources and total pollution emissions reduction is raising the research efforts in the process of biomass combustion [1].

The oldest form of biomass, combusted in domestic devices is wood, but today the utilization of different biofuels, such as straw, many types of pellets, brickets or even sewage sludge [1, 2] is rising rapidly. Practices related to the use of biomass for electricity generation in large scale of co-combustion (power plants) have shown a lot of disadvantages of this form of biomass utilization, especially because of transport costs, energy density and difficulties related to the adaptation of the coal-fired boilers to new parameters of fuel [3].

The alternative way to utilize biomass effectively is gasification, pyrolysis or combustion in small heating and cogeneration systems, which are characterized by power equal maximally a few hundreds of kW. Most of popular small systems are based on the biomass batch or retort boilers with fixed bed or retort.

1.2. The energy appliances studies and optimization with use of numerical models, based on the computational fluid dynamics (CFD) tools

The application of CFD tools is a helpful method for the design and optimization of innovative concepts in power engineering. It is easy to find many examples of CFD simulations, which are directly related with the operation of some elements of heat generation systems or combustion and heat transfer processes.

Miltner et. al. used process simulation and computational fluid dynamics to achieve the maximization of the thermal efficiency and the reduction of gaseous and particulate matter emissions in an innovative baled biomass-fired combustion chamber [4].

Menghini et. al. described experimental and computational activities, which were performed to determine the possible improvements of total environmental impact of a wood-open fireplace. Results of experimental tests were used to validate the simulation model. Then, the simulation results have been adopted to hypothesize a new configuration of fireplace [1].

Many papers are devoted to the study of the air distribution process, into combustion chamber of fireplace or boiler. Bhasker discussed simulation of turbulent air flow distribution in a boiler furnace, where primary air is entrained through the inlet duct system called windbox [5]. Dong and Blasiak presented a numerical model, designed to analyze the advanced secondary/over fired air Ecotube system [6].

However, only small part of works is devoted to the air feeding systems of small-scale heating units. Therefore, it is reasonable to consider the example of such a type of system.

2. Experiments and simulations

2.1. The system of biomass boiler

The paper describes the results of numerical simulations, related to the function of air collector of the biomass boiler "EKOPAL RM40", which is intended for the combustion of straw. The boiler and manifold system is presented in Fig. 1. In the rest of the paper, mass air flows for individual ducts, which in Figure 1 are marked as $R1$-$R7$ (for front ducts) and $D1$, $D2$ (for side ducts) will be presented as X_i (i – the number of duct). The appliance which is studied and simulated is responsible for air distribution into the primary and secondary combustion chamber of the boiler.

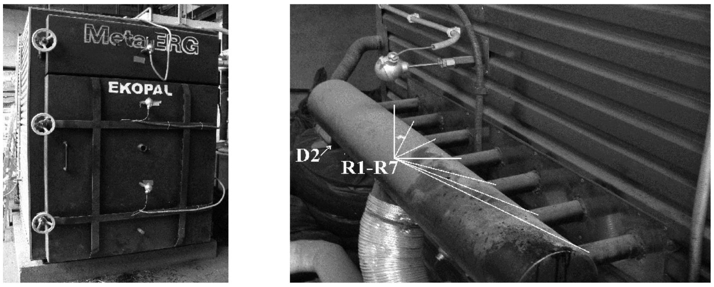

Fig. 1. View of the front of the biomass boiler "EKOPAL RM40" and air collector installed at the backside wall of the combustion chamber

The air flow distribution in the whole area of the primary chamber of boiler, affects the process of combustion. Various quantities of air (oxygen) in different areas of the chamber may cause incomplete combustion.

2.2. Results of experimental studies

The air collector is composed of seven front pipes and two curved side pipes, welded to the main duct. Pipes at the front of the main duct are numbered as $R1$–$R7$ (radius). The symbol of the pipe at the center of the main duct is $R4$. Two side curved pipes are numbered as $D1$ and $D2$ (diameter).

The results of the analysis with the use of a thermo-anemometer probe were used to characterize the distribution of air at the inlet pipe of the fan (diameter – 150 mm). Air velocity distribution inside the pipe was determined by measurements in 16 points along the radius. As a result of calculations, the air mass flow which is provided by fan was defined.

2.3. Numerical model

The air collector was analyzed with use of the'ANSYS CFX' tool. The spatial geometry (3D) of the modeled object has been prepared in Autodesk Inventor Professional 2013.

The designed geometry was imported into the ANSYS Design Modeler environment, and then transferred to the ANSYS Meshing where discretization has been carried out automatically. Because of the relative simplicity of the manifold design and size, it was decided that in the case of the domain interior it is not necessary to use advanced meshing options, but it was important to take into account the phenomenona near the boundary layer. A fine final grid was imposed by generating the inflation layer having the first element thickness of 0.0005 m and the growth rate of 1.2 for the next 6 sublayers. The total number of the computational grid elements was 61×10^4. The details of the applied computing grid are shown in Fig. 2.

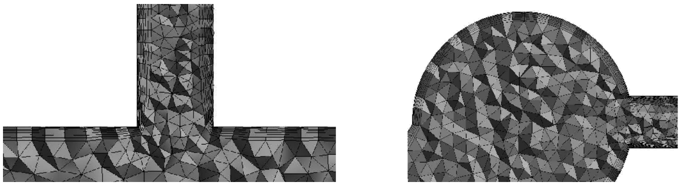

Fig. 2. The details of computing grid applied in the designed model

The SST (Shear Stress Transport|) model of turbulence was used. The air was assumed as an ideal gas (stationary domain). The air flow through the collector was solved by the commercial code ANSYS CFX, with the use of the finite volume method (FVM). The level of residues chosen for the simulation was 10^{-5}.

Because of certain details of the geometry of the secondary combustion chamber, the accurate measurement of the outlet air flow inside the boiler is difficult. Consequently, the distribution of air mass flows at the outlets of the collector ducts has been simulated by CFD.

The results of experiments show that the distribution of air along the outlets of feeding pipes, inside the chamber, is heterogeneous. To optimize (compensate) air distribution inside the combustion chamber, the objective function $\sigma(X)$ (Equations 1–3), that relates the diameter of the pipe with the air flow was defined. The function $\sigma(X)$, which is classic standard deviation of dispersion of the air mass flux X_i was chosen because, in the considered case, the standard deviation is a measure of divergence of individual mass flows. Minimization of the standard deviation means homogenization of the mass flow in the ducts along the manifold.

$$\sigma(X) = \sqrt{\frac{1}{N}\sum_{i=1}^{N}(X_i - X)^2} \tag{1}$$

$$X = \frac{1}{N}\sum_{i=1}^{N} X_i \tag{2}$$

$$\sigma(X) \rightarrow \min \tag{3}$$

The above mentioned function was implemented to the ANSYS CFX and results of standard deviation for all design points were displayed as a part of post processing.

3. Results and discussion

In the optimization process, the objective function $\sigma = 5^{-5}$kg/s was achieved. The air mass flow for the designed collector with new dimensions of feed ducts is equal to about 0.024 kg/s (for each pipe).

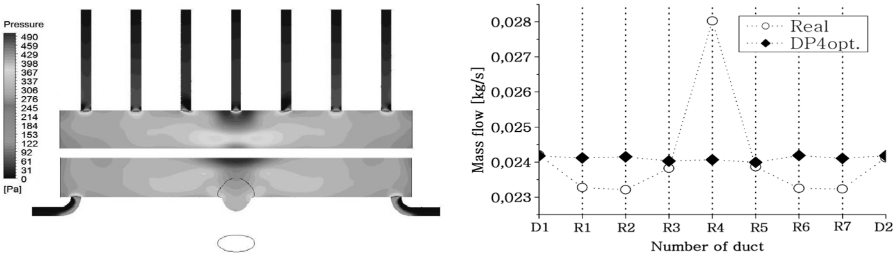

Fig. 3. Simulation of pressure distribution inside the optimized collector (left side) and chart, which presents the comparison of mass flow in pipes for real and optimized dimensions (right side)

Table 1

Presentation of the dimensions of ducts in given design points (DP) and major parameters of the device

Name	The radius (*R*)/diameter (*D*) of the given duct [mm]					Pressure [Pa]	Power [W]	σ [kg/s]
	*R*1 = *R*7	*R*2 = *R*6	*R*3 = *R*5	*R*4	*D*1 = *D*2			
Current	18.35	18.35	18,35	18.35	36.7	524	95.7	$18 \bullet 10^{-4}$
DP4	18.35	18.65	18.3	16.9	36.7	548	100.1	$5 \bullet 10^{-5}$

The comparison between the current and optimized air flow distribution is presented in Fig. 3. For the chosen air mass flow values, changes of pressure and fan power in function of air velocity were defined. It was found that both curves are described by a second degree polynomial function. Consequently, the loss of pressure inside the collector was calculated, according to the Darcy-Weisbach equation. Pressure loss inside air collector after optimization, for air mass flow 0.217 kg/s, rose from 523 to 548 Pa.

Because of it, the fan power which is required to overcome the increased flow resistance, is growing from 67 to 100W (Table 1). The distribution of pressure inside the optimized collector is showed on Fig. 3. Growth of pressure is particularly visible in the main duct, near to narrow inlets of ducts.

4. Conclusions

In the studies described in the paper, CFD model has been used to successfully simulate the physical properties of air inside the collector of the biomass boiler. The results of model calculations allow to design a device, which provides homogeneous distribution of air inside the combustion chamber of a boiler.

Based on reference [7], it can be concluded that proper construction of the air collector avoids too high emission of carbon monoxide (CO), which is a product of incomplete fuel combustion. Confirmation of the effect should be the subject of further works. Numerical and experimental analysis of the impact of proposed air manifold design changes on the CO emission effect will be performed and described in the separate paper. Moreover¸ in the considered case, it is suggested to analyze the possibility of using a different type of a collector – for example a device with equal lengths of the profiled feed ducts.

It is always important to compare growth of efficiency of combustion process and energy generation to the financial aspects of design changes. In the simulated appliance, modification is related only to a change of the pipes' diameter, so the implementation of the presented solution seems to be reasonable.

This paper was carried out under contract (11.11.210.217) AGH-Univeristy of Science and Technology, Faculty of Fuels and Energy, Cracow, Poland.

References

[1] Menghini D., Marchione T., Martino G., Marra F.S., C. Allouis, Berenta F., *Numerical and experimental investigations to lower environmental impact of an open fireplace*, Experimental Thermal and Fluid Science 31, 2007, 477-482.

[2] Szubel M., Sornek K., Robak B., Ciołek M., Filipowicz M., *Analysis of the possibility of co-combustion of sewage sludge in biomass boilers*, Archiwum Spalania, nr 4/2012, 263-271.

[3] Szubel M., *Fuel cells powered by hydrogen from biomass*, Zeszyty Naukowe Politechniki Rzeszowskiej, nr 283, Budownictwo i Inżynieria 2012, z. 59, nr 2, t. 2, 755-762.

[4] Miltner M., Miltner A., Harasek M., Friedl A., *Process simulation and CFD calculations for the development of an innovative baled biomass-fired combustion chamber*, Applied Thermal Engineering 27, 2007, 1138-1143.

[5] Bhasker C., *Simulation of air flow in the typical boiler windbox segments*, Advances in Engineering Software 33, 2002, 793-804.

[6] Dong W., Blasiak W., *CFD modeling of ecotube system in coal and waste grate combustion*, Energy Conversion and Management 42, 2001, 1887-1896.

[7] Rybak W., *Spalanie I współspalanie biopaliw stałych*, Oficyna Wydawnicza Politechniki Wrocławskiej, Wrocław 2006, 108-114.