CFD MODELLING OF BIOMASS COMBUSTION IN A HEATING BOILER

Răzvan MAHU, Florin POPESCU, Ion V. ION

"DUNAREA DE JOS" UNIVERSITY OF GALATI, Galați, Romania

INTRODUCTION

To optimize the efficiency of the boiler using agricultural residues, an important task is to study its behaviour in different conditions. The general behaviour depends on a number of operational factors (combustion air flow, humidity, circulation, and so on), dimensions and characteristics of biomass briquettes (moisture content, size, raw material, density, friability, etc.).

The large variation of the properties of biomass solid fuels, particularly agricultural residues, significantly influence the thermal efficiency and pollutant emissions of combustion plants in which they are used.

This paper describes the numerical simulation of the biomass combustion process in a heating boiler (Termofarc FI-GS 40) using the Ansys CFD. It was considered that the biomass volatilization occurs in the upper chamber of the boiler and volatiles combustion occurs in the lower chamber. After the volatiles composition generated in the upper chamber has been calculated the combustion simulation in the lower chamber was performed using ANSYS CFD (Fluent).

THE NUMERICAL MODEL

The numerical simulation was performed following the steps below:

- Setting the goals of the simulation was the stage where we have analyzed the problem and set the significant physicochemical quantities to be calculated;
- Defining the analysis field: we design the geometry of the computational field;
- Creating the mesh for the computational field;
Defining the mathematical models: choosing the set of equations that model the characteristic phenomena for the problem. In the global set of equations we considered also the constitutive equations which describe the physical properties of substances;

- Defining the initial and the boundary conditions;
- Solving the problem numerically to find the unknown quantities;
- Post-processing: the derived results are calculated using the numerical solution obtained in the above stages and a number of graphical representations of the results are drawn.

**Pre-processing phase**

We designed the geometry of the Termofarc FI-GS 40 boiler in Catia (Fig. 1). We considered the computational domain for the lower chamber of the boiler (Fig. 2).

The computational domain consists of the lower chamber of the heat exchanger (in this case the inner surface), the gas inlet ducts (denoted by I, II and III in Fig. 2), the secondary air inlet ducts (denoted from 1 to 6 in Fig. 2) and flue gas exhaust channel (denoted by O in Fig. 2).

After defining the computational field, we discretized it using a fully structured mesh. There resulted 1,497,136 cells and 1,572,681 nodes (Fig. 3).

For gas flow modelling the RANS model was chosen (Reynolds Averaged Navier-Stokes). For turbulence modelling SST k-ω model (shear stress transport) was chosen, so named because the definition of the turbulent viscosity has been adjusted to take into account the main transport turbulent shear stress. Modelling chemical reactions and chemical species transport was made using a predefined Fluent namely the firing of the mixture fraction (non-premixed combustion model). Heat transfer was modelled using the energy conservation equation for conduction and convection. For radiation P-1 model was chosen. It is considered that the heat transfer to the chicanes and the side walls is made by conduction and radiation heat flux is of 13179.6 W/m².

The inlet boundary conditions have been set, specifying the mass flow rate of the gas mixture.

On the outlet boundary the pressure was considered to be equal to atmospheric pressure.
Wall was considered a condition "no-slip" which means that the relative velocity was considered 0 in the walls.

The resulted flow rate of the fuel gas in the primary chamber was 0.0376 kg/s at a temperature $t=875^\circ\text{C}$. The resulted flow rate for the secondary air has been found to be 0.0038 kg/s at a temperature $t=23^\circ\text{C}$.

The numerical solver used was a Pressure Based Solver Coupled. It solves in a coupled manner the conservation equations and momentum equations while the rest of the equations are segregated solved. To accelerate the convergence of stationary solution a pseudo-time stepping algorithm was used.

To discretize the transport equations a second-order model was chosen (Second Order upwind) for better accuracy for all equations.

RESULTS AND DISCUSSIONS

The simulation results were compared with experimental data, observing a very good correlation. The temperature resulted from the numerical simulation of volatiles' combustion at the point where the temperature was measured was 850°C. The difference between the measured temperature (836°C) and the results from numerical modelling (850°C) was 16°C (Fig. 7). The difference between the measured CO concentration (1.11%) and simulated CO concentration (1.13%) was 0.02% (Fig. 10).

Flue gas velocity is higher inside the jet and the flow follows a correct configuration according to the boiler geometry (Fig. 4).

The turbulence increases in the areas where the jet strikes the first baffle and flame vortex appears (Fig. 5).

It can be seen in Figures 6 and 7 that the temperature of the walls is greater in the upper side and decreases as the combustion gases is approaching the exhaust.
As a consequence it is necessary to protect the boiler upper bed with refractory material against hot gases (Fig. 7).

The heat transfer is mainly achieved by radiation (Fig. 8). The radiation intensity is higher in the inlet area of the secondary chamber and decreases as the exhaust gas approaches the outlet.

![Fig. 8 Radiation intensity [W/m²].](image)

![Fig. 9 Radiation absorption coefficient [1/m].](image)

The radiation absorption coefficient of the flue gas depends on its composition and the local temperature. The absorption radiation coefficient is higher as the combustion gas approaches the outlet (Fig. 9).

In Figure 10 mass fraction of carbon monoxide CO has a higher concentration in the entrance area and as combustion occurs, the amount decreases when moving to the exit. This result is consistent to the experimental results where the CO concentration is higher in the entrance area of the secondary chamber of the boiler.

![Fig. 10. CO mass fraction.](image)

**CONCLUSIONS**

Modeling and numerical simulation synergistically complements the experiments and is a low-cost approach to design, allowing optimization of biomass combustion plants to produce heat or electricity. Modeling and numerical simulation provides an advanced understanding of flow processes occurring during thermochemical conversion of solid biomass and therefore it is recognized as a powerful tool for faster development, less risky and more accurate technologies.

The simulation results of a down-draught boiler with a power of 42 kW for domestic heating which uses briquettes made from agricultural residues are in very good agreement with experimental data. The results indicate the following: flue gas velocity is higher inside the jet and the flow follows boiler geometry; turbulence increases in areas where flame jet strikes the first row of heating tubes and eddies are formed resulting the need to protect the tubes with refractory material against the hot gases.

**REFERENCES**


