CHEMICAL ENGINEERING TRANSACTIONS Volume 21, 2010
Editor J. J. Klemeš, H. L. Lam, P. S. Varbanov

DOI: 10.3303/CET1021183

Process-Simulation and Computational Fluid Dynamics for the Development of a Novel Solid Biomass-Fired Combustor

Christian Kuttner, Martin Miltner*, Michael Harasek, Anton Friedl
Vienna University of Technolgy, Institute of Chemical Engineering
Getreidemarkt 9/166-2, 1060 Vienna , Austria
martin.miltner@tuwien.ac.at

The aim of this work is to design and devise an innovative combustion chamber with a thermal output of 500kW by means of experimental analysis accompanied by process simulation and computational fluid dynamics (CFD). Various biomasses in form of compressed bales, such as straw, maize or similar energy crops are used for this combustor. As the combustion concept of the named oven is rather sophisticated and innovative, powerful simulation tools have to be applied in order to design the combustion chamber prior to the plant realization. It will be shown that the correct application of these tools, especially CFD, significantly supports basic and detailed engineering of novel combustion systems.

1. Introduction

The interest of decentralized energy production by means of biomass-fired combustion is increasing in the last few years. This is the result of an increasing CO2-awarness and high prices for fossil fuels. The use of biogenic combustibles provides many advantages particularly with regard to CO_2-emissions. The development of appropriate designs of combustion systems for these fuels is rather challenging, cost intensive and often time-consuming. Therefore the application of advanced simulation tools like process simulation and computational fluid dynamics is of vital importance in an early stage.
The aim of this work is to design and devise an innovative combustion chamber with a thermal output of 500kW operating with solid biofuel in form of compressed biomass bales. Process simulation is used for the balancing of the whole plant and for the validation of different process layouts and operational parameters like recirculation rate of flue gas and combustion air distribution. Several construction relevant facts have to be considered in order to maintain stable bale combustion and minimize gaseous emissions such as VOCs, carbon monoxide and nitrogen oxide. The determination of essential combustion parameters like residence time distribution, mixing behavior of flue gas with secondary air and temperature distribution is accomplished by means of CFD. With the result of the simulation, a suitable design of the individual combustion zones can be developed applying a repetitive approach. Additionally to the commonly used modeling approaches for CFD, more advanced models have to be used to

Please cite this article as: Kuttner C., Miltner M., Harasek M. and Friedl A. (2010), Process-simulation and computational fluid dynamics for the development of a novel solid biomass-fired combustion chamber, Chemical Engineering Transactions, 21, 1093-1098 DOI: 10.3303/CET1021183

characterize the heterogeneous combustion which comprise the description of the solid biofuel, the steps during the combustion (drying, devolatilization and char combustion) and the generation of nitrogen oxide emissions. These models have to be implemented by user-defined subroutines.Furthermore, experiments are carried out to enable the validation and calibration of the calculation models and their parameters. Regarding the improvement of the control engineering, prefabricated parts of the plant such as valves for the fresh and recycled air are investigated by CFD and supported by experiments. The results are used to design and improve the control of the fresh and recycled gas-flow, which is important to accomplish a stable combustion performance. Furthermore, a full-scale combustion pilot plant with 500 kW thermal power is currently in commissioning and comprehensive combustion experiments will be undertaken in the forthcoming period to validate the applied mathematical models and to sharpen the used model parameters.

2. Design and Engineering via CFD

CFD has proven to be a well-suited tool for the development and optimization of combustion processes as reported by recent publications (Agraniotis et al 2009, Venturini et al 2010, Yin et al 2010 and Yu et al 2010). A multitude of research papers has been published in this field. The main objective of this task is to support the construction of the combustion system by providing simulation results showing the impact of design deviations and modifications by parameter variation. This investigation is mainly accomplished prior to the final combustion plant assembly. Hence, construction costs are reduced and practical experiments supporting the development process are minimized. The combustor is composed of three compartments: afterburning grate, primary combustion-zone and secondary combustion-zone. In the primary zone the drying and devolatilization of the biomass takes place. The secondary zone is used to oxidize the combustible gases emanating from the primary zone. Since the drawer transports the biomass continuously, the non-volatile or low-volatile fractions of the biomass fuel end up on the small afterburning grate where a complete solid fuel burnout takes place. The walls of the grate and the primary zone are not lined with refractory like in conventional combustion systems. The inner core is surrounded by an outer shell flowed through with the combustion air, which therefore is preheated and cools the walls (Miltner et al, 2005). This design offers a fast start-up process, however it needs an accurate and fast controlling system. The engineering process is partitioned in three steps. First, a combustion calculation and mass- and energy balancing are required to start the design of the primary and secondary zone. Figure 1 shows a scheme of the combustion system and the combustion chamber. Secondly, the initial geometry design of the inner core, the outer shell and the bale drawer is implemented and the CFD simulation is accomplished. The third step comprises the evaluation of the simulation results and the re-design of the combustor. Thus, an optimized combustion chamber design can be elaborated applying a repetitive approach. Additional simulations on a test valve (Figure 2) have been accomplished in order to identify the characteristics of the gas-flow upstream and downstream the baffle and to calculate the drag coefficient. The combustion plant contains three streams of fresh air and recycled gas in which the gas volume flow has to be measured and controlled. For this purpose, an approach given by Jung (1956) is used. The gas volume flow is calculated by the pressure and the actual angle of aperture together with the pre-defined drag coefficient function. The results are used to measure and control the gas-flow in a single device so as to improve the control engineering and the responding behavior of the gas supply. For this purpose, several 3D-models of the valve with different angles of apertures have been implemented and analyzed by CFD.

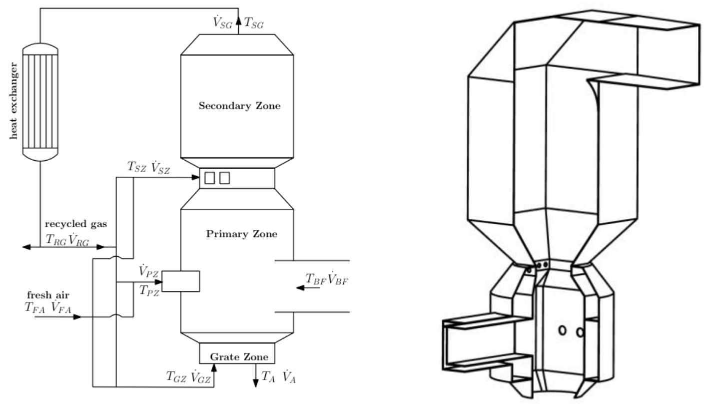

Figure 1: Process scheme of the combustion plant (left) and construction design of the combustion chamber (right)

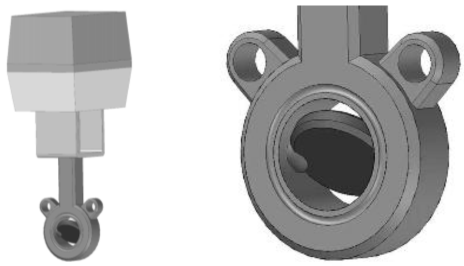

Figure 2: Test valve for gas volume flow measurement and control

2.1 Model Implementation

For the CFD simulations the commercial Navier-Stokes-solver FLUENT© (by Fluent Inc.) has been chosen, geometry implementation has been done using GAMBIT©. The gas is supposed to be incompressible and the density is assumed according to the ideal gas law. In addition to the continuity and Navier-Stokes-equation the turbulence of the gas-flow is considered by using the k-ε-model. The air is marked by using the species-model to show the mixing behavior of the secondary combustion zone. The boundary conditions for the simulations are as follows:

- Walls are supposed to be infinite thin, impermeable, adiabatic and have a noslip-boundary.
- The walls are defined to be adiabatic and have nonslip-boundary.

- The inlets are specified as “mass-flow-inlet”. The parameters are taken from the balance-calculation; the outlet is defined as “pressure-outlet” with surrounding pressure.
- The thermo-physical properties are taken from the FLUENT©-database.

It has to be mentioned, that in order to perform an accurate prediction of the flow pattern of the hot, reacting gas in the individual zones, several sophisticated models concerning the turbulence, heat conduction and radiation, combustion and gas-solid-interaction have to be implemented. Due to the early stage of the ongoing research project, most of these effects have been neglected in the first step. The results presented in this work have been obtained using an isothermal CFD-simulation of a hot, non-reacting, turbulent gas flow without radiation and without a solid dust phase. Convergence of the CFD-computation was assumed as soon as the residuals have stabilized at a low level and several monitors on process-relevant parameters (mass flows and gas velocities) did not significantly change anymore.

3. Results and Discussion

A contour plot of the gas velocity magnitude of the primary air nozzles is presented in Figure 3. This analysis has been used to determine the exact positioning of the two primary air nozzles during the design phase. As the high-turbulent impact of combustion air jets on the bale surface is one of the key factors of the applied combustion technique, this design determination is of high importance, especially if no experiments are to be conducted in this context. The high gas velocity and the free impinging jets are clearly observable. The results shows that the depth of impression, which depends on the inlet-diameter, is sufficient to reach the surface of the straw bale, thus an adequate supply of air for achieving a stable combustion is guaranteed and the elaborated positioning of the nozzles is adequate.

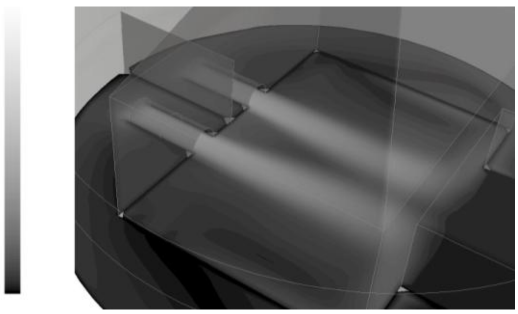

Figure 3: Contours of velocity magnitude in the primary zone in m/s

The inlets of the secondary air are positioned at the end of the primary zone to implement a staged combustion and to guarantee low emissions of nitrogen oxide and unburned combustibles. For this purpose, the exact position of these nozzles and the design of the narrow intersection of the primary and secondary zone had to be found. In order to promote the mixing of the flue gas emanating from the primary zone with the secondary air the cross sectional area, the position of the secondary air nozzles and the diameter of these nozzles have been optimized by application of CFD. Figure 4 (left) illustrates path lines leaving the secondary air nozzles. Due to the adjustment of the inlets a vortex is generated which improves the mixing behavior of the secondary air which is also shown in Figure 4 (right). After a short path the secondary combustion air is totally mixed in, which indicates the quality of the turbulent mixing process. Providing sufficient residence time and temperature, the complete burnout of CO and VOCs in the secondary combustion zone is assured.

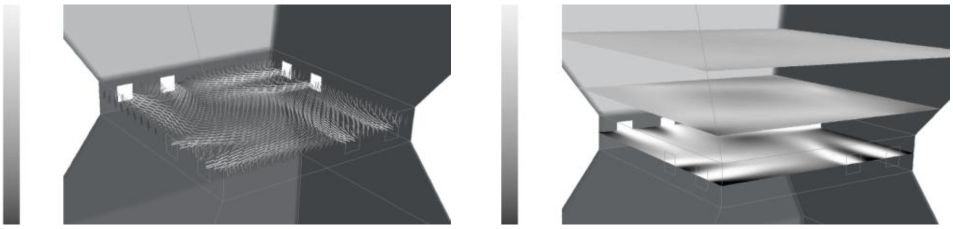

Figure 4: Path lines coloured velocity magnitude in m/s (left) and mixing behaviour of the secondary air (right)

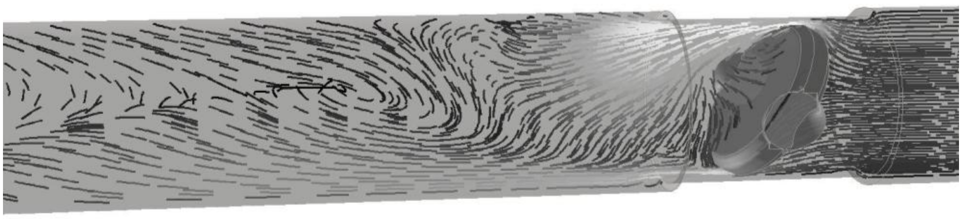

Figure 5: path lines at the cross-section of the valve coloured in velocity magnitude in m/s

Figure 5 shows exemplarily the complex flow patter in the vicinity of the investigated gas valve. The correct positioning of the pressure measurement device is important to get accurate measurement results that can be used to calculate the volume-flow of the gas. Therefore a minimum distance is strictly adhered. The occurred eddies downstream the baffle affects a disparate distribution of the absolute pressure along the pipe. It is observed that the flow pattern homogenizes within a distance of approximately 5 times of the pipe-diameter sufficiently for volume flow measurements according to the described procedure.

4. Outlook

The results of the simulation work are satisfying for the given tasks of a basic design of the combustion chamber and very promising for future improvements. However, further improvements in modeling accuracy have to be done. These improvements will involve the appropriate description of the gas phase reaction kinetics, the solid fuel combustion

containing all relevant process steps like drying, devolatilization and char burnout, and the adequate modeling of radiation heat transfer. Many of these effects will be modeled using user-defined subroutines as described by Miltner et al (2008).
Furthermore, combustion experiments will to be carried out to validate the modeling approaches and sharpen the used model parameters.

5. Conclusions

The results of this work show that process simulation and CFD are powerful tools for designing and optimizing a novel biomass combustor. Even if not a full set of sophisticated modeling approaches has been implemented yet, the design of the pilot combustion chamber was possible in a trusted way using already gained knowledge from former publications. CFD provides efficient and cost-saving assistance in the development and optimization of the novel apparatus and the controlling system.

Acknowledgments

We gratefully acknowledge the support of the providers of funds FFG, BMVIT and BMWFJ and our project partners WTI GmbH and Federspiel Oekotechnology Consulting GmbH.

References

Agraniotis M., Stamatis D., Grammelis P. and Kakaras E., 2009, Numerical investigation on the combustion behaviour of pre-dried Greek lignite, Fuel 88 (12), 2385-2391.

Jung R., 1956, The dimensioning of throttles for flow rate control, Brennstoff Waerme Kraft, 8 (12), 580-583 (in German).

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

Miltner M., Makaruk A., Harasek M. and Friedl A., 2008, Computational fluid dynamic simulation of a solid biomass combustor: modelling approaches, Clean Technologies and Environmental Policy 10, 165–174.

Venturini P., Borello D., Iossa C., Lentini D. and Rispoli F., 2010, Modeling of multiphase combustion and deposit formation in a biomass-fed furnace, Energy 35 (7), 3008-3021.

Yin C., Kaer S., Rosendahl L. and Hvid S., 2010, Co-firing straw with coal in a swirl-stabilized dual-feed burner: Modelling and experimental validation, Bioresource Technology 101 (11), 4169-4178.

Yu Z., Ma X. and Liao Y, 2010, Mathematical modeling of combustion in a grate-fired boiler burning straw and effect of operating conditions under air- and oxygen-enriched atmospheres, Renewable Energy 35 (5), 895-903.