2nd World Conference and Exhibition on Biomass for Energy, Industry and Climate Protection
10-14 May 2004, Rome, Italy

ADVANCED CFD ANALYSIS OF LARGE FIXED BED BIOMASS BOILERS WITH SPECIAL FOCUS ON THE CONVECTIVE SECTION

Robert Scharler[1,2,3], Martin Forstner[2], Markus Braun[5], Thomas Brunner[1,2,3,4], Ingwald Obernberger[1,2,3,4]
[1]AUSTRIAN BIOENERGY CENTRE GmbH, Inffeldgasse 24d, A-8010 Graz, Austria, Phone: +43 (0)316 481300 33;
mailto: robert.scharler@abc-energy.at
[2]BIOS BIOENERGIESYSTEME GmbH, Sandgasse 47, A-8010 Graz, Austria
[3]Institute of Resource Efficient and Sustainable Systems, Graz University of Technology, Inffeldgasse 25, A-8010 Graz;
[4]Technical University of Eindhoven, Department of Mechanical Engineering, Section Process Technology, Den Dolech 2, P.O. Box 513, 5600 MB Eindhoven, the Netherlands
[5]FLUENT Deutschland GmbH, Birkenweg 14a, 64295 Darmstadt, Germany

ABSTRACT: The researchers have successfully designed and optimised several biomass grate furnaces using CFD analysis as an efficient tool. As a detailed CFD simulation of tube bank heat exchangers is currently impossible due to computational limitations, a finite cell based convective heat exchanger (CHE) model has been developed. A verifycation of the overall CFD model including a comparison with measurements from a large waste wood fired steam boiler was successfully performed. Measured CO emissions at boiler outlet and simulation results agreed very well. The comparison of simulation results with flue gas temperature measurements (suction pyrometer) in the first duct of the radiant boiler section showed an excellent qualitative agreement. The agreement between the CHE model results and temperature measurements at the inflow of the second superheater and the first economiser was very good as well. Concluding, the developed CHE model was successfully tested and it showed to be a powerful design and optimisation tool for convective heat exchangers. Furthermore, it forms an important basis for deposit formation models for biomass fired boilers, which are currently being developed.

Keywords: grate furnace, boiler, modeling.

1 INTRODUCTION

Due to the increasing performance of computers, Computational Fluid Dynamics (CFD) is gaining importance as a cost-efficient tool for the design and optimisation of industrial furnaces and boilers. Despite the high complexity of heterogeneous combustion of biomass in fixed beds, the researchers have successfully designed and optimised several biomass grate furnaces [5] using CFD analysis as an efficient tool. A detailed CFD simulation of the convective boiler section with a resolution of the heat exchanger tubes is still impossible because of the large variation of geometric length scales and due to transient flow phenomena like vertex shedding behind tubes. Therefore, a finite cell based convective heat exchanger (CHE) model has been developed in order to investigate the whole boiler by means of CFD simulations. Simulation results were compared with measurements of flue gas temperature and CO emissions in a large waste wood fired steam boiler (nominal fuel power related to the net calorific value = 19.5 MW) in order to test and verify the results of the overall CFD model. Furthermore, the performance of the furnace and the radiant section of the boiler, which had been designed based on CFD simulations, was investigated.

2 MODELLED BIOMASS FIRED STEAM BOILER

The investigated boiler (Figure 1) is designed for Low-NO_x combustion in order to reduce expensive secondary NO_x reduction measures. Therefore, the furnace is divided into two combustion zones (air staging technology). Whereas the primary combustion zone is operated as an air lean, hot reduction zone with sufficient residence time for the flue gas (0.6 – 0.8 s at nominal power) to reduce NO_x emissions by primary measures, the secondary combustion zone is designed as an air rich burnout zone. A further important feature is the staged flue gas recirculation (flue gas supply below and above the grate) in order to optimise temperature control and turbulent mixing in the primary combustion zone.

The boiler section consists of four vertical ducts, which are divided into two parts. The radiant section of the boiler includes the first and the second duct. The convective section covers the third duct with evaporator and superheater tube bundles as well as the economiser. In the lower part of the first boiler duct the finned walls are partly covered with refractory lining in order to ensure an appropriate reaction temperature concerning flue gas burnout and NO_x reduction by secondary measures.

Operating data of the waste wood fired boiler for a selected period of typical boiler operation were taken as a basis for the CFD simulation (average water content of the fuel 16.4 wt% w.b.; fuel power related to the net calorific value = 18.5 MW). The simulation results were then compared to detailed flue gas measurements performed during the selected operation period. Concentrations of CO (and O_2) were measured after the fabric filter by conventional flue gas analysis. Profiles of flue gas temperatures were measured at two positions in the first vertical duct (secondary combustion zone/radiant boiler section) by a suction pyrometer. Additionally, temperature measurements with thermocouples were performed at three positions in the convective section of the boiler.

3 MODELLING

An own-developed empirical model was used for the combustion of solid biomass on the grate. This model supplies velocities, species concentrations and temperatures of the flue gas above the surface of the fuel layer as boundary conditions for the subsequent CFD simulation of the turbulent reactive flow in the furnace and radiant boiler section.

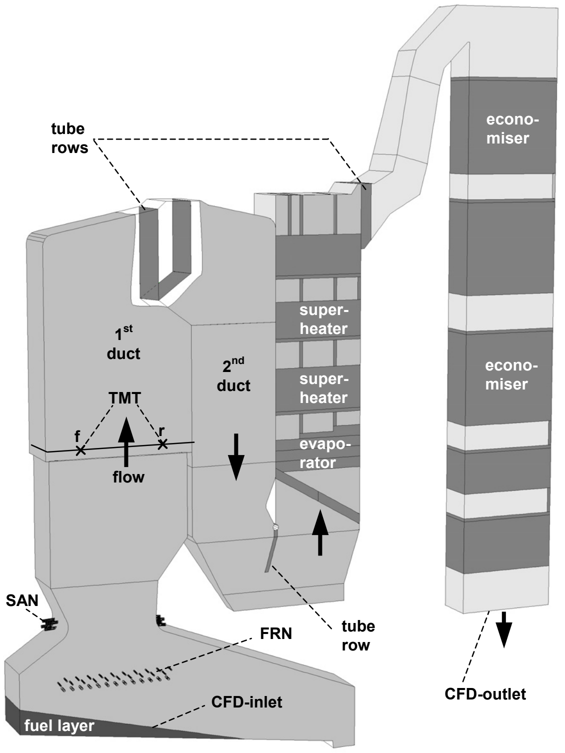


Figure 1: CFD model of the biomass furnace and boiler
Explanations: modeled tube bundles and rows are pictured dark gray; SAN…secondary air nozzles, FRN…flue gas recirculation nozzles, TMT… suction pyrometer temperature measurement traverses

In order to save computational time, the convective part of the boiler section was simulated separately, based on the results of the simulation of the furnace and radiant boiler section. The computational domains of both simulations were overlapping to correctly extract inlet boundary conditions for the simulation of the convective part. The CHE model was applied in order to calculate the flow as well as the convective and radiant heat transfer in the evaporator, superheater and economiser tube banks. Additionally it was applied to calculate the convective and radiant heat transfer in the single tube rows between the first and the second duct as well as in the tube rows at the entry to and the exit from the third duct.

3.1 The own developed fixed bed combustion model

A detailed mathematical treatment of biomass fixed bed combustion is too time-consuming for engineering applications and, furthermore, the underlying processes are not completely understood so far. An empirical model was thus applied for the thermal decomposition of the solid fuel on the grate. The model consists mainly of three parts. The definition of one-dimensional profiles along the grate concerning the degradation of the fuel components as well as fuel drying (part No. 1; based on assumptions and experimental data) forms the basis. In combination with the definition of conversion parameters (based on assumptions as well as experimental and literature data), which describe the formation of the most important flue gas components CH_4, CO, CO_2, H_2, H_2O and O_2 (part No. 2), the stepwise balancing of mass and energy fluxes released from the fuel bed is possible (part No. 3). For a more detailed description of the model see [4] and [5].

3.2 CFD modeling of the turbulent reactive flow

For the CFD modeling the Realizable k-ε model (turbulence), the Discrete Ordinates model (radiation) and the Eddy Dissipation model [3] (EDM - turbulent gas phase combustion) in combination with a global methane 3-step reaction mechanism of Brink [1] including the species CH_4, CO, CO_2, H_2, H_2O and O_2 were used.

The EDM in combination with reaction kinetics serves to calculate the reaction rate as the lowest (limiting) step of mixing rates proportional to the break-up of fuel, oxygen and product eddies as well as a kinetic rate without a consideration of the influence of turbulent fluctuations. The EDM is reasonably accurate for most industrial applications, numerically robust and applicable to premixed, non-premixed and partially premixed combustion but cannot account for strong coupling between turbulence and reaction kinetics. If reaction kinetic effects become important (e.g. NO_x formation or flame quenching) the EDM in combination with global kinetics is not suitable. Another disadvantage is the fact that the empirical model parameter (A_{mag}), which determines the mixing rate calculated with the EDM is not universally valid. Calculations performed for a lab-scale test case as well as comparison with measurements at a pilot-scale biomass grate furnace have shown that the originally proposed value of $A_{mag} = 4.0$ (see [3]) is too high for fixed bed biomass compustion applications (for details see [5]). A value of $A_{mag} = 0.6$ is therefore used for the simulation of biomass grate furnaces.

3.3 The convective heat exchanger (CHE) model

The tube banks (evaporator, superheater, economiser) were simulated with the newly developed CHE model. As the CHE model was originally developed for inhomogeneous flow fields, the economisers were included in the presented CFD simulation for validating purposes only. In contrast to standard heat exchanger modeling approaches, the CHE model considers the influence of the flow direction within a tube bank on flow resistance and convective heat transfer. Furthermore, a separate treatment of the first row of a tube bundle is applied because it is underlying peculiar flow patterns and highest local surface temperatures, which are of special importance concerning ash deposit formation. Peak values of surface temperatures and total heat fluxes of tube banks are calculated in a post-processing mode.

The calculation of the source terms for momentum and energy transport equations is mainly based on literature data ([2], [6] and [7]), which is available for tube banks of staggered or in-line arrangement and flow perpendicular to the individual tube rows. Transient 2D simulations were performed to calculate the flow resistance for flow directions other than the literature values. Thus the dependence of flow resistance on the flow angle within a tube bank can correctly be accounted for. Radiative heat transfer is considered by using the Discrete Ordinates model and by calculating source terms for the radiation intensity as well as by the energy transport equations. Peak surface temperatures are calculated based on correlations of the local heat transfer coefficient along the tube perimeter [7] as well as on experience-based assumptions for the local distribution of the thermal resistance of the deposit layer.

With this approach the influence of the tube bundle geometry and the deposit layers on pressure loss, flow-field and heat transfer are considered.

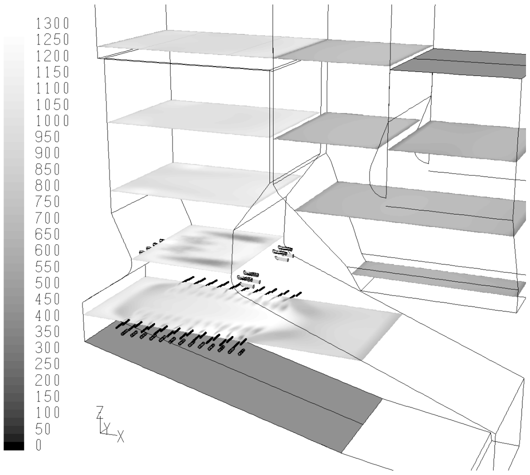


Figure 2: Temperature profiles in horizontal cross-sections of the furnace and boiler
Explanations: top cross-section…level where suction pyrometer temperature measurements were performed

4 DISCUSSION OF RESULTS

4.1 Simulation of the furnace and radiant boiler section

The comparison of calculated and measured flue gas temperatures for a selected test run in various boiler sections was performed for two main reasons. On the one hand the performance of the furnace should be checked and on the other hand the developed CFD model for biomass grate furnaces including the thermal boundary conditions (emissivity of furnace walls and thermal resistance of deposit layers) should be validated. In the first duct of the water tube steam boiler, measurements of the flue gas temperature were performed with a suction pyrometer along two parallel traverses at the SNCR ports (for secondary NO_x reduction).

The CO emissions measured at boiler outlet were very low (CO <30 mg/Nm^3 dry flue gas) and agreed well with CFD simulation results (+ 5-20 % dev.). This is a consequence of the optimised CFD furnace design, especially of the air and flue gas re-circulation nozzles. An even and complete penetration of the furnace cross-section with flue gas and air jets and an induction of a swirling flow in the secondary combustion zone could be achieved, which leads to a break-up of flue gas strains and an efficient mixing of the unburned flue gas with air. In Figure 2 the temperature profiles in horizontal cross-sections of the furnace and radiant boiler section including the measuring traverses are shown. One can see that the temperature profiles are very evenly distributed, which, again, is a result of the CFD design of the furnace.

Considering all uncertainties of measurements in grate fired furnaces which underlie fluctuations in operation conditions, the qualitative agreement with the calculations is excellent. However, especially the thermal resistance of the deposit layer on the finned walls, which is a priori unknown, has a certain influence on the absolute values of flue gas temperatures, and therefore, a sensitivity analysis based on literature data and experience was performed (variation of thermal resistance: 5-10 m^{2}K/kW). In Figure 3 the results for the best case with a value of 7.5 m^{2}K/kW for the thermal resistance are shown. The simulation matched the measurements very well (maximum deviation: 50 °C).

The developed CFD model for biomass grate furnaces and radiative boiler sections could be successfully tested for large biomass CHP plants. The validated thermal resistance value of the deposit layers in combination with the flow, temperature and species profiles at the end of the second radiant duct where used as boundary conditions for the simulation of the convective section of the boiler.

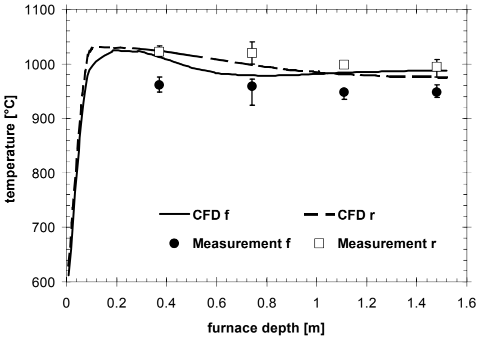
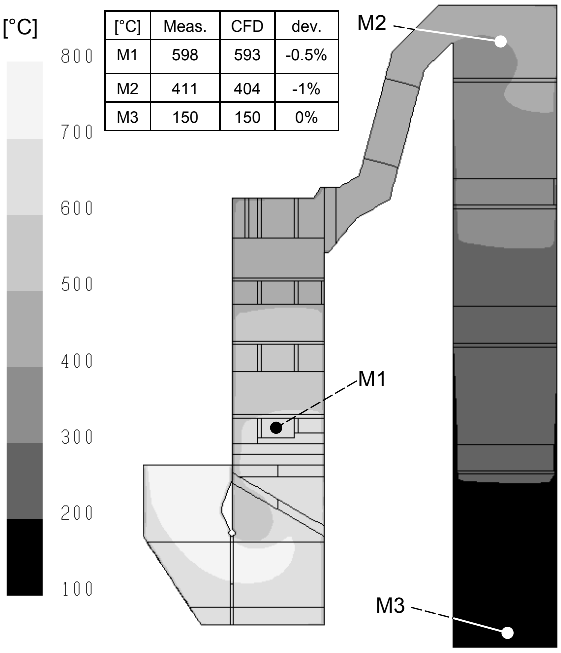


Figure 3: Measured and simulated flue gas temperature
Explanations: flue gas temperature profiles were measured with a suction pyrometer at two positions in the first duct of the first boiler duct (radiative section; see Figure 1)

4.2 Simulation of the convective heat exchanger section

As already mentioned, the thermal resistances of the deposit layer on the finned walls in the convective section were assumed to have the same value as in the radiant section. The same value was also taken for the evaporator and superheater tube banks. For the economisers, smaller values were assumed (25% of thermal resistance of the superheaters). With these boundary conditions, the agreement between the CFD simulation and the flue gas temperatures measured in the convective section was very good (table in Figure 4). The assumption of a uniform distribution of the ash deposit layer on the boiler walls and tube banks was reasonably accurate for the description of the operating behavior of the plant investigated.



[°C]	Meas.	CFD	dev.
M1	598	593	-0.5%
M2	411	404	-1%
M3	150	150	0%

Figure 4: Flue gas temperature in the symmetry plane - results of CFD simulation compared to measurements

Upstream of the evaporator the CFD simulation showed a large recirculation vortex section, which caused a very uneven distribution of flue gas temperatures (Figure 4) and velocities within the evaporator. The maximum velocity was about 11 m/s compared to a mean value of about 4 m/s. As a consequence, large differences regarding the the peak surface temperature (up to 200 °C) and a maximum value of about 650 °C in the first tube row of the evaporator (Figure 4) occur. This points out that by an optimised and more even flow distribution at the entrance to the evaporator the maximum surface temperature and, following, the risk of hard ash deposit formation can be lowered. Furthermore, it could be found, that in the superheaters and in most economiser tube banks the flue gas temperature profiles are uniformly distributed, with the exception of the first economiser, where another strong deflection of the flow takes place. As the temperatures of flue gas and tube surfaces are relatively low at the entrance to the economiser, the above mentioned non-uniform flow distribution is normally not critical from a deposition point of view.

The obtained results show, that compared to traditional 1D heat exchanger design and standard CFD modeling approaches, the CHE model gives a much deeper insight into the heat transfer process. Especially in regions of flow gradients, which are usually most relevant for ash deposit formation, the CHE model is superior.

However, further test runs and measurements at large biomass boilers are necessary and planned in order to validate the CHE model.

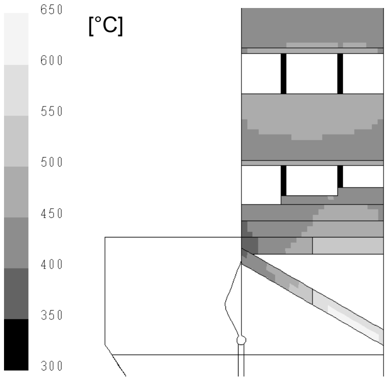


Figure 5: Predicted maximum surface temperatures in the evaporator and the first part of the superheater section (symmetry plane)

5 SUMMARY AND CONCLUSIONS

A convective heat exchanger (CHE) model was developed to enable the optimisation of biomass fixed bed boilers including the convective section via CFD analysis. Moreover, a comparison of simulation results with measurements of flue gas temperature and CO emissions at boiler outlet for a large waste wood fired steam boiler was performed in order to verify the overall CFD model used (model for biomass grate furnaces combined with the CHE model). Furthermore, the performance of the furnace and radiant section of the boiler, which had been designed based on CFD simulations, was evaluated.

The CO emissions measured were very low due to the CFD optimised mixing of flue gas and air and in good agreement with the CFD results. Furthermore, the temperature profiles in the furnace section were evenly distributed with low peak values. The comparison of CFD calculations with flue gas temperature measurements (suction pyrometer) in the first duct of the radiant boiler section showed an excellent qualitative and quantitative agreement. These results show, that the CFD analysis has been successfully applied during the design process and that the CFD model for biomass grate furnaces has been successfully validated for large-scale biomass boilers. The simulation results of the furnace and radiant boiler section were used as boundary condition for the subsequent simulation of the convective boiler section with the CHE model.

A comparison between CHE calculations and temperature measurements performed at three locations in the convective boiler section showed a good agreement. Further comparisons with measurements are ongoing in order to validate the model. The CHE model developed showed to be superior to conventional 1D heat exchanger calculations in regions where large flow gradients occur and gives a more realistic description of the flow field within the tube bundles compared to other CFD modelling approaches. Consequently, the newly developed CHE model has been successfully tested. Moreover, a CFD analysis of the convective boiler section was made possible and the CHE model showed to be a powerful design and optimisation tool for convective heat exchangers. Furthermore, it forms an important basis for deposit formation models for biomass fired boilers, which are currently being developed.

ACKNOWLEDGEMENTS

This work has been supported by the FWF Austrian Science Fund and the Austrian Bioenergy Centre GmbH.

REFERENCES

[1] Brink, A.: Eddy Break-Up based models for Industrial diffusion flames with complex gas phase chemistry. Ph.D. thesis, Abo Akademi University, Finland, (1998).

[2] Idelchik, I: Handbook of Hydraulic Resistance, Begell House, New York (1996).

[3] Magnussen, B.F, Hjertager, B.H.: On mathematical modeling of turbulent combustion with special emphasis on soot formation and combustion. In: Proceedings of the 16th Symp. (Int.) on Combustion, pp. 719-729, The Combustion Institute (Ed.), Pittsburgh, USA, (1976).

[4] Obernberger, I., Widmann, E., Scharler, R.: Entwicklung eines Abbrandmodells und eines NO_x-Post-prozessors zur Verbesserung der CFD-Simulation von Biomasse-Festbettfeuerungen. Reports from energy and environment research, report No. 31/2003, Ministry for Transport, Innovation and Technology (Ed.), Vienna, Austria.

[5] Scharler, R.: Entwicklung und Optimierung von Biomasse-Rostfeuerungen durch CFD-Analyse, Ph.D. thesis, Graz University of Technology, Austria (2001).

[6] VDI, VDI-Wärmeatlas, Berechnungsblätter für den Wärmeübergang, 8. Auflage, Springer-Verlag, Berlin, (1997).

[7] A. Zukauskas, R. Ulinskas, High-Performance Single-Phase Heat Exchangers, Hemisphere Publishing Corporation, Washington, DC (1989).