Improvement of a Combustion Unit based on a Grate Furnace for Granular Dry Solid Biofuels using CFD Methods

C. Jordan, M. Harasek
Vienna University of Technology; Institute for Chemical Engineering; Getreidemarkt 9/1662
A-1060 Vienna; tel. +43 1-58801-15924, e-mail: jordan@mail.zserv.tuwien.ac.at.

Introduction

Considering the current efforts to reduce the emissions of green house gases and the need to satisfy the growing energy demand new and improved technologies for combustion are required. Despite known problems with emissions and ash melting free flowing granular dry solid biofuels like wood, straw pellets, and grain have an outstanding potential to replace fossile fuels.

The aim of this ongoing project is to develop a small scale combustion unit based on grate firing with a thermal power output in the range of 350-5000 kW, which overcomes the above mentioned drawbacks. In cooperation with the Austrian company Polytechnik GmbH, the Austrian Bioenergy Centre (ABC) and the Österreichisches Forschungsinstitut für Chemie und Technik (OFI) a prototype will be constructed, which will be optimized using computational fluid dynamics (CFD) simulations.

Possible measures to overcome the problems with ash melting are suitable primary air distribution, temperature control using flue gas recirculation and the guidance of the hot flue gas flow over the fuel bed. CFD methods can be used to predict the behaviour of the combustion chamber if these measures were implemented and therefore it is possible to optimize the boiler geometry very cost efficient.

Based on the calculated results improvements of the existing geometry have been suggested and will be included in the design of the prototype which is planned to start operation in autumn 2008.

Large Scale Experiments in a 1 MW grate fired boiler

At the start of the project a measurement campaign was carried out – for different operating conditions flow, temperature and composition were measured. During the experiments thermographic images were taken.

Combustion Unit: The firing unit used for the experiments is a 1 $MW_{(th)}$ grate furnace. The solid fuel is fed using a push rod onto the grate. The primary air is split in several zones, the flow rate of each zone can be adjusted. Secondary air is fed directly into the combustion area, flue gas recycling can be used for the post combustion zone. The hot flue gas is fed to a hot water boiler (not included in the simulation). The main control is based on the fuel feed rate and on the primary air.

Primary air is adjusted in a way that the bed combustion takes place at $\lambda < 1$ (understochiometric combustion), the produced CO and volatiles are burnt consuming the supplied secondary air.

The remaining ash is discharged on the lower end of the grate into an ash bunker, the flue gas after the heat exchanger is passed through a bag filter to reduce the dust emissions.

Measurements: To provide sufficient input and validation data for the CFD simulation of the combustion unit measurements on the 1 $MW_{(th)}$ unit are carried out. Gas flow rates of the primary and secondary air as well as the recycle gas were measured using Pitot tubes or hot wire anemometers, all of the air and flue gas temperatures as well as solid bed temperatures were recorded using thermocouples. The pressure difference between the combustion chamber and the surrounding induced by the draught fan and flue gas composition (including O_2, CO and NO_x emissions) were also measured. Furthermore the fuel feed rate and the ash amount were recorded during the experiments. A mass balance was set up to account for losses and leak air. Experiments using different fuels (wood chips, grain) at full and partial load conditions have been carried out in order to cover the dynamic range of the boiler and to analyze the emissions.

Thermographic Imaging: The inner wall temperatures of the combustion chamber and the surface temperature of the burning fuel bed on the grate were measured after approx. 1 h of steady operation using an infrared camera (Infratec VarioCam, featuring an uncooled microbolometer array with a resolution of 320x240 pixel, calibrated up to 1700°C in the spectral range of 7-14 µm, [Infratec, 2008]). Images were taken through a maintenance lid at the lower end of the grate.

CFD Models and Boundary conditions

Based on the measured data a CFD simulation was set up, the full geometry of the combustion unit used for the measurements was implemented with all necessary details and meshed. The steady flow solutions were analyzed and used to suggest improvements for the boiler unit.

Grid generation: The first step for each CFD simulation is the discretization of the considered geometry. In this case, for the implementation of the full and simplified 3D geometries and the generation of the computational grid the pre-processor GAMBIT 2.3 [Fluent.Inc, 2001-2006] was applied. The grid implementation of the furnace covered not only the combustion chamber but also the primary and secondary air pipes and the distribution zones, the flue gas recirculation ducts, the post combustion zone and the refractory of the furnace. In total, approximately 1 million hexahedral finite volume cells were used for discretizing the simulation zones. An outline of the implemented geometry can be found in figure 1.

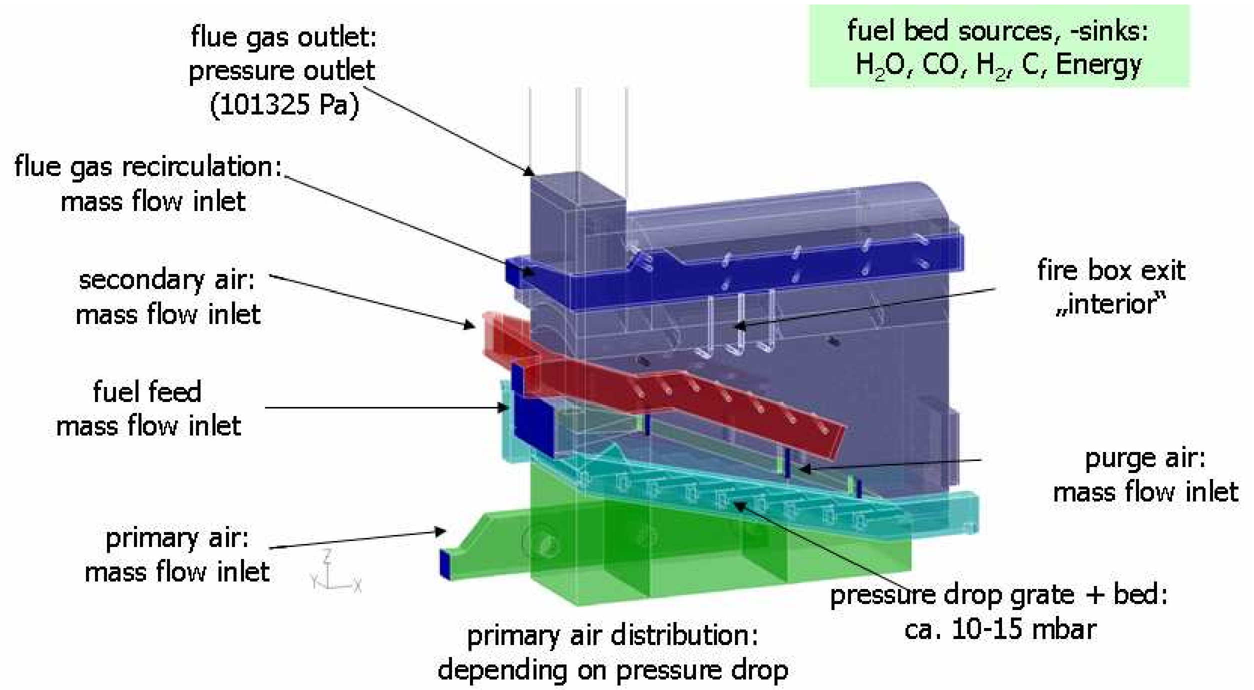

Figure 1: Overview of the geometry and the boundary settings

CFD Method: A commercial CFD solver code was used - FLUENT 6.3 [Fluent.Inc, 2001-2006] - a state of the art general purpose code with included post processing tools. FLUENT is based on finite volume discretization, for each finite control volume the physical balance equations are solved for momentum (Navier-Stokes-Equations), energy, mass (Continuity Equation) and species. Also common turbulence and radiation models are available. Since some of the equations are highly nonlinear, an iterative solving procedure is applied. One important feature of the CFD code is the possibility to include new models by utilizing compiled user defined functions (UDFs), which can be hooked into predefined interfaces of the solver.

To ensure highest numerical accuracy, sufficient computational performance and good convergence behaviour all grids used for this work consist only of (unstructured) hexahedral elements. Also second or higher order discretization schemes were applied for all equations.

Boundary conditions: All inlet boundaries were set to constant mass flow rate (mass flow inlets, constant mass flux over the area), all exit boundaries were set to constant static pressure (pressure outlet with possible back flow) – see also Figure 1. The most important inlets and outlets are listed and their location.

Following sub-models of the FLUENT package were applied to ensure for an accurate description of the blast furnace:

- *Fluid properties:* The density of the gas phase is described using the ideal gas equation to account for the elevated absolute pressure in the furnace. Other properties of the mixture (thermal conductivity, heat capacity, viscosity) are calculated by ideal gas mixing rules based on data from VDI Wärmeatlas [VDI, 2002] and NIST [National Institute of Standards, 2007], if not available, kinetic theory approximations have been used.
- *Solid properties:* Temperature dependent thermal conductivity data for the refractory have been provided by Polytechnik GmbH, the heat capacity and the density were considered constant.

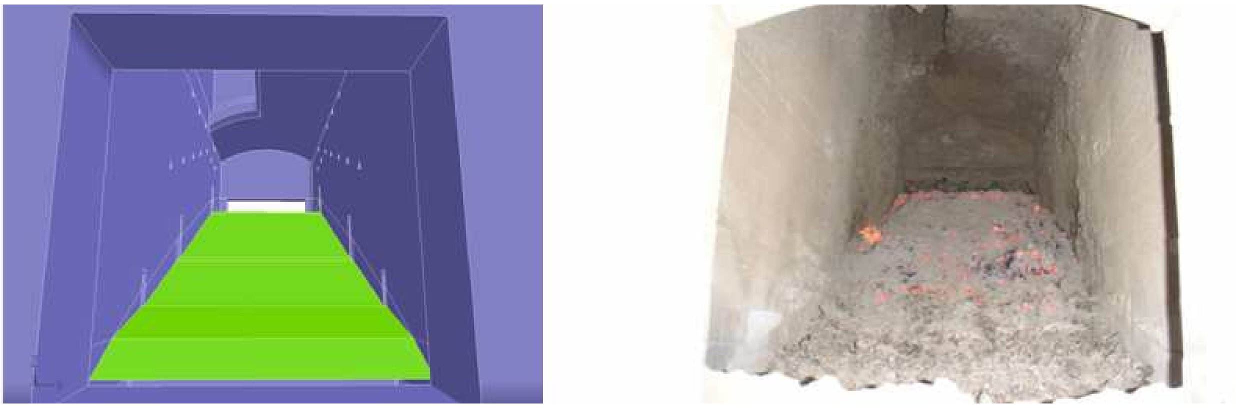

Figure 2: Fuel bed on the grate – left: model, right: photograph of the combustion unit

- *Grate combustion*: A simplified zonal grate combustion model for wood chips with a fixed bed geometry (Figure 2) was used for the combustible solids, including a devolatilization reaction depending on the longitudinal position on the grate. The solid bed was modelled using the porous media approach with a given pressure drop. The most important regions are the drying region, the aforementioned devolatilization region and the coke combustion region. The simplified biomass volatiles can consist of CO, H_2, the solid coke reaction is modelled with a zonal carbon source.

- *Gas phase reaction*: A simplified combustion mechanism with only 15 species (O_2, N_2, CO, CO_2, H_2, H_2O, C_s and radicals including OH, O, H) and approx. 30 reactions has been constructed. The mechanism was based on the work by Saxena [Saxena, 2006]. To include turbulence interaction, the well known eddy dissipation concept (EDC, [Magnussen, 1981]) was used (also [Harasek, 2004]. NO_x formation was not covered in this simulation.
- *Turbulence*: In this simulation the SST-k-ω-Model by Menter [Menter, 1994] was applied since it provided better convergence than the available k-ε-Models but required less computational power than RSM. The model has already been successfully applied to a wide range of technical flow problems [Jordan, 2005, Maier, 2008, Miltner, 2007]. Standard wall functions were used for the solid boundaries, wall roughness was included.
- *Radiation heat transfer*: Due to the operating temperatures of the grate furnace combustion, radiation is a very important heat transfer mechanism. An appropriate model for radiation heat transfer, which can be applied for optically dense media P1 model. The radiation absorptivity and emissivity of the gas phase were described using the weighted sum of grey gases (WSGG) approach [Barlow, 2003].

All simulations were carried out using cluster computers of the ZID of the Vienna University of Technology (www.zid.tuwien.ac.at) or of the Institute of Chemical Engineering (Research group Thermal Process Engineering and Simulation, www.cfd.at):

- ICP5: number of computing nodes: 54x IBM p5-520/52A each with 2x CPU power5+ @ 1,9 GHz/36 MB L3 cache, 16 GB RAM, InfiniBand HCA interconnection - operating system: AIX 5L V5.3
- PHOENIX: number of computing nodes: 40x AMD Opteron 250 @ 2,4 GHz, 8 GB RAM - operating system: Fedora Core Linux).

The calculation of the steady state solutions required approx. 2-3 weeks CPU time each, memory consumption was in the range of 2 GB. To analyze the possible low frequency fluctuations and oscillating flow features also unsteady simulations were carried out for a representative setup.

Results of the Simulation

The numerical simulation demonstrated that the overall concept of the grate furnace unit works very well. The secondary air system and the flue gas recirculation ducts provide a good gas distribution, the position of the secondary air inlets above the grate proves to support the CO combustion. No unexpected significant wakes or dead zones in the flow could be detected (figure 3), the big recirculation zone above the grate towards the lower end was predictable.

The primary air distribution system is sufficient but a improved control system could provide increased performance, especially in partial load conditions when a equal coverage of the grate with coke and fuel bed cannot be guaranteed and channelling may occur.

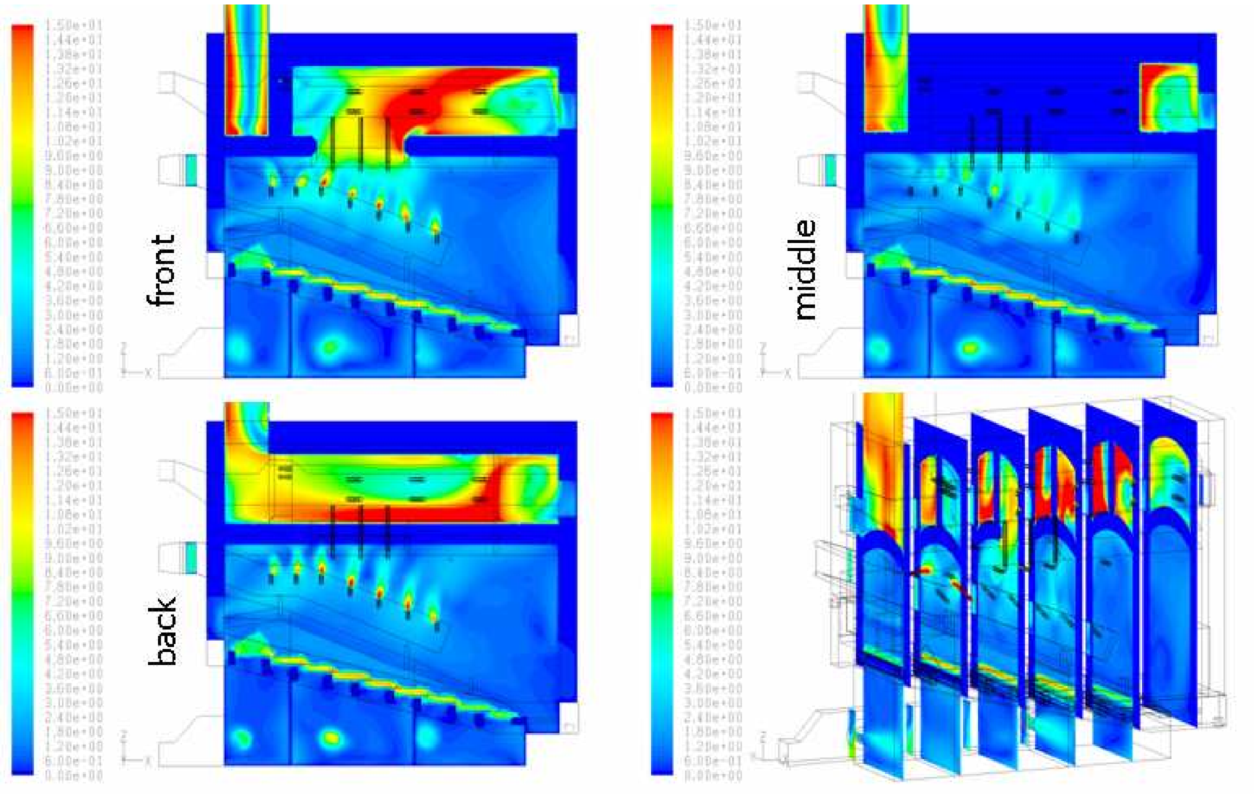

Figure 3: Velocity magnitude in m/s in various cross sections and planes

The wall temperature distribution in the simulation agrees reasonably well with the thermographic images (figure 4 and 5). Usually infrared thermography in the 7-14 µm range is not affected by gas flames (having the emission maxima between 2-5 µm), unless the flame is optically thick and has a high soot content. The radiative contribution from the soot has to be taken into account to calculate the correct wall temperature, the primary air flow has to be reduced while taking the IR-photo. In addition the CFD simulation has to be adapted using an appropriate soot model.

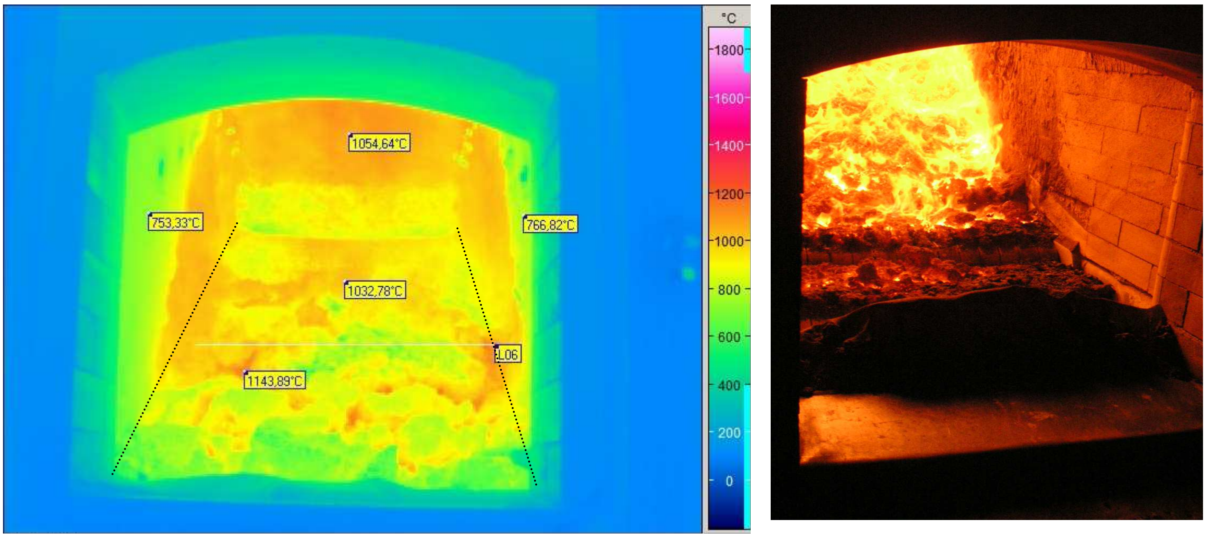

Figure 4: Left – infrared image of the bed and the side walls (temperature in °C, surface radiation emission overlaid with flame structures, soot is also visible in this wavelength range) – the dotted lines show the position of the grate edge; Right: normal photograph of the flame structure on the fuel bed.

The calculated coke/fuel bed temperatures are in good agreement with the measurements (Figure 4): Anyhow, the measured bed temperature of approx. 1050°C in the upper part of the grate during the experiments using wood chips would be much too high for straw pellets or grain – the ash melting temperature can be as low as 700-800°C. Therefore additional cooling using recycle flue gas is also recommended in the primary air zone thus reducing the oxygen mass fraction.

The measured and calculated temperatures are summarized in table 1, the uncertainty in the measurements is approx. ± 20 K (depending on the emission factor)

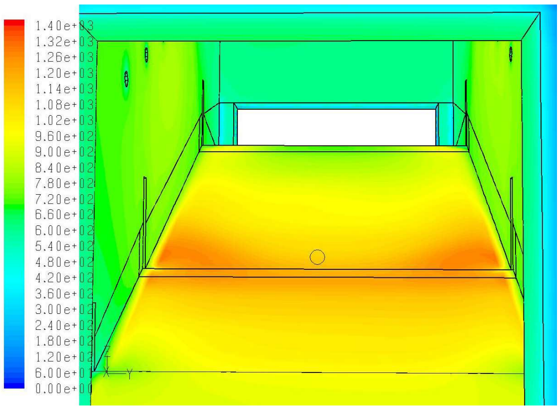

Figure 5: CFD result (plain surface temperature in °C)

Table 1: Overview Temperatures measured/calculated

Position	Measured	CFD
coke bed surface (IR and thermocouple)	1030°C	1100°C
wall temperature left	750°C	740°C
wall temperature right	770°C	760°C

The gas phase temperature distribution is shown in figure 6 and 7 – the high temperature at the fuel feed zone (figure 7, left) most probably results from the leak air coming in with the solid fuel. The unsteady simulation revealed a pulsating shape of the temperature iso surfaces (like in figure 6). Figure 8 shows the inner and outer surface temperature of the refractory.

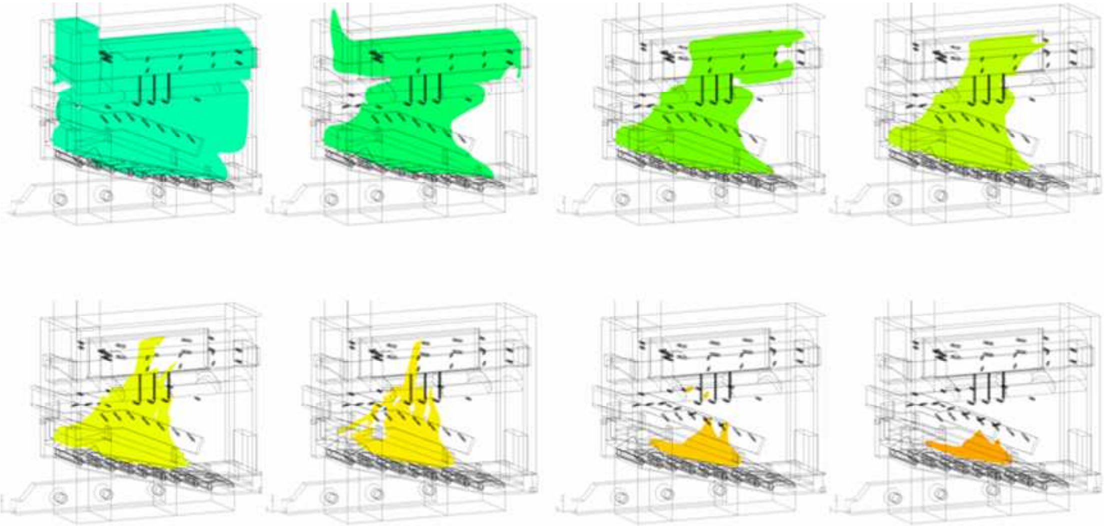

Figure 6: Iso surfaces of constant temperature (in °C) – 1st row: 600/700/800/900°C, 2nd row: 1000/1100/1200/1300°C

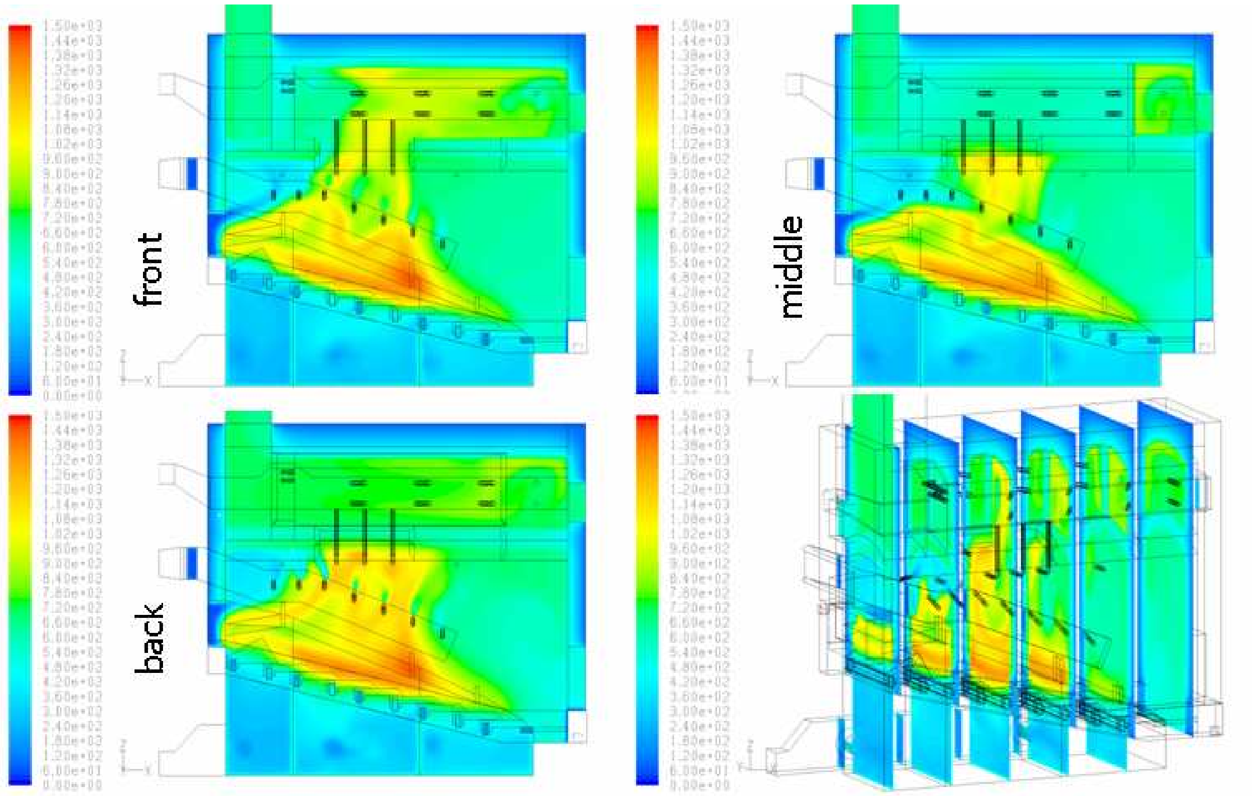

Figure 7: Gas phase temperature in °C (various planes)

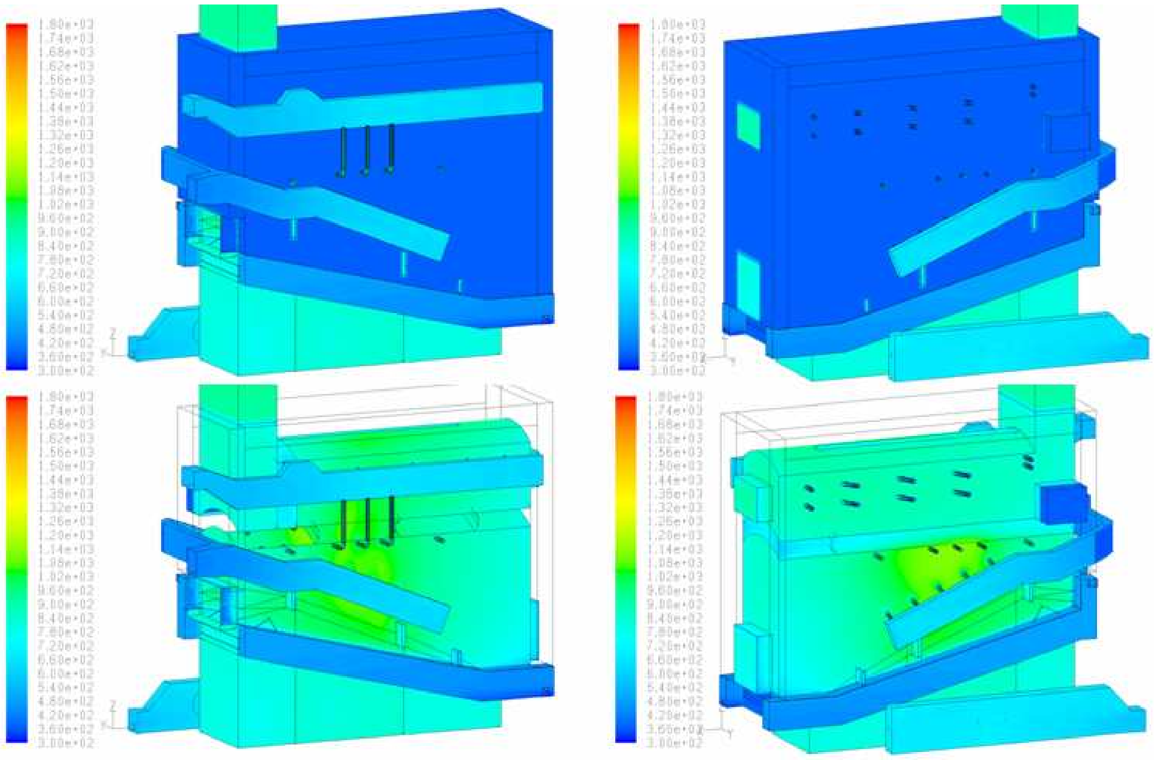

Figure 8: Inner and outer surface temperature of the refractory or the air distribution ducts (temperature in K)

Furthermore some other improvements regarding the position of the entrance to the post combustion zone from the fire box and the shape of several refractory elements to avoid slow large eddies in the gas flow which can cause ash deposition and accretion have also been suggested and are being considered for the implementation into the new prototype geometry.

Outlook, further work

Simulation and Measurements:

Additional measurements for longer periods of constant operation would provide a more reliable foundation for validation of the simulation. Further simulations will be done for other fuels (especially grain). also other load conditions will be included. The inclusion of a soot model and a more detailed bed combustion model will be considered.

Prototype:

One of the next steps would be the detailed planning of the prototype, during the summer period (when the demand for heating is limited) the new grate furnace is about to be set up at the production facility of the Polytechnik GmbH. Installation of modern control hardware would allow for continuous measurements. Detailed fuel analyzes would be required to close the mass and energy balances.

Acknowledgements

The authors also want to thank the Österreichisches Forschungsinstitut für Chemie und Technik (OFI), Austrian Bioenergy Centre ABC and Polytechnik GmbH for their financial support, their valuable contributions and their work.

References

[Infratec, 2008] http://www.infratec.de/. Infratec, 2008

[Fluent.Inc, 2001-2007] FLUENT: http://www.fluent.com/. FLUENT Inc., 2007.

[VDI, 2002] VDI [2002] VDI Wärmeatlas Berechnungsblätter für den Wärmeübergang, VDI-Gesellschaft Verfahrenstechnik und Chemieingenieurwesen; Springer-Verlag.

[National Institute of Standards, 2007] National Institute of Standards, USA, NIST Chemistry Webbook [online]. Available from: http://webbook.nist.gov [Accessed 15 Oct. 2007].

[Saxena, 2006] Saxena P., Williams, F. A.: Testing a small detailed chemical-kinetic mechanism for the combustion of hydrogen and carbon monoxide. Combustion and Flame 145 (2006) 316–323

[Magnussen, 1981] Magnussen, B.: On the structure of turbulence and a generalized eddy dissipation concept for chemical reaction in turbulent flow. In AIAA, Aerospace Sciences Meeting. 1981.

[Menter, 1994] Menter, F. R.: Two-equation eddy-viscosity turbulence models for engineering applications. AIAA, 32:1598–1605, 1994.

[Jordan, 2005] Jordan, C., Miltner, M., Potetz, A. and Harasek, M.: Modellierung turbulenter Freistrahlen mit numerischer Strömungssimulation. Chemie Ingenieur Technik, 77(8):1061 – 1062, 2005.

[Maier, 2008] Maier, C., Jordan, C., Bolhar, M., Kaiser, S., Harasek, M.: CFD-Simulation of the Gas Phase Combustion in Stationary Fluidized-Bed Firing Systems. 9th International Conference on Circulating Fluidized Beds CFB9; Hamburg, Germany. May, 13 – 16, 2008

[Miltner, 2007] Miltner, M., Miltner, A., Harasek, M. and Friedl, A.: Process simulation and CFD calculations for the development of an innovative baled biomass-fired combustion chamber. Applied Thermal Engineering, 27(7):1138–1143, 2007.

[Harasek, 2004] Harasek, M., Miltner, M., Jordan, C. and Friedl, A.: CFD-Simulation of a baled biomass-fired furnace: Combustion modelling and design optimisation for the reduction of emissions. In Proceedings of the Second World Conference and Technology Exhibition on Biomass for Energy, Industry and Climate Protection, Rome, Italy, pages 1391–1393. 2004.

[Barlow, 2003] Barlow, R.; Sandia National Laboratories [2003], International Workshop on Measurement and Compuation of Turbulent Nonpremixed Flames [online]. Available from: public.ca.sandia.gov/TNF/radiation.html [Accessed 15 Oct. 2007].