Aalborg Universitet

AALBORG UNIVERSITY
DENMARK

CFD modeling and experience of waste-to-energy plant burning waste wood

Rajh, B.; Yin, Chungen; Samec, N.; Hribersek, M.; Kokalj, F.

CFD MODELING AND EXPERIENCE OF WASTE-TO-ENERGY PLANT BURNING WASTE WOOD

B. RAJH*, C. YIN**, N. SAMEC*, M. HRIBERŠEK* AND F. KOKALJ*

Faculty of Mechanical Engineering, University of Maribor, 2000 Maribor, Slovenia
**Department of Energy Technology, Aalborg University, 9220 Aalborg East, Denmark*

SUMMARY: Computational Fluid Dynamics (CFD) is being increasingly used in industry for in-depth understanding of the fundamental mixing, combustion, heat transfer and pollutant formation in combustion processes and for design and optimization of Waste-to-Energy (WtE) plants. In this paper, CFD modeling of waste wood combustion in a 13 MW grate-fired boiler in a WtE plant is presented. As a validation effort, the temperature profiles at a number of ports in the furnace are measured and the experimental results are compared with the CFD predictions. In the simulation, a 1D model is developed to simulate the conversion of the waste wood in the fuel bed on the grate, which provides the appropriate inlet boundary condition for the freeboard 3D CFD simulation. The CFD analysis reveals the detailed mixing and combustion characteristics in the waste wood-fired furnace, pinpointing how to improve the design and operation in order to achieve better WtE efficiency. The results also show that neglecting buoyancy effects may lead to appreciable errors.

1. INTRODUCTION

Nowadays thermal treatment technologies enabling energy recovery from waste are highly developed. Grate-firing is one of the main technologies that are widely used in Waste-to-Energy (WtE) plants for CHP (combined heat and power) production from waste combustion/incineration. However, some plants are still reported to suffer from comparatively high loss of ignition (LOI), low efficiency, and high emissions, which need to be optimized during design and/or commissioning/operation phase. CFD (Computation Fluid Dynamics) is a powerful tool to aid in optimization of WtE plants to achieve higher efficiency and lower emissions.

WtE processes represent an important step towards the utilization of available energy sources and reducing the dependence on energy imports (especially fuels from politically unstable regions). For energy utilization only the waste that has calorific value can be used. Energy utilization is according to the legislation defined as utilization of renewable energy sources, where a significant role plays especially protection of environment and air against the accumulation of green-house gases that arise by using fossil fuels.

Waste wood that is not suitable for recycling is regarded as waste and can be used to generate electricity or heat. Untreated wood can be used as a fuel in virtually any biomass heating plant, but wood that has been treated (painted, coated, impregnated or otherwise contaminated) can only be fired only in plants with the right combustion processes and exhaust scrubbers (WtE plants).

The objective of this paper is to establish a reliable CFD numerical model for a 13 MW WtE plant, which can be used for better optimization and design of this type WtE plants. The CFD analysis is based on the commercial code ANSYS CFX. Firstly, a sensitivity analysis is done on the basis of the design conditions of the WtE plant to evaluate the effects of different factors in CFD modeling of grate-firing boilers. Additionally, a 4-h measuring campaign is carried out to measure the local gas temperatures on different measuring ports to collect the operating parameters, and to observe the overall mixing and combustion pattern in the WtE plant. On the basis of the sensitivity analysis and the experimental measurements, a CFD numerical model is finally defined. The CFD results are compared to the measured data and observations in the real WtE plant.

2. THE WtE PLANT DESCRIPTION

WtE plants of capacities less than 30.000 tons/yearly is usually performed in two or more phases. Commercially successful incinerators of this type usually have two combustion chambers, as shown schematically in Figure 1. The WtE plant under this study is designed for waste wood combustion (Figure 1a) with nominal steam generation rate 13.7 t. The maximum capacity of the boiler is 13 MWth. Grate-firing system is equipped with air-cooled forward acting grates which are located at the bottom of the primary combustion chamber, as sketched in Figure 1b.

PA₀, PA₁, PA₂, PA₃, PA₄: primary air under the grate
FGR₀: recirculated flue gases under the grate
FGR₁: recirculated flue gases above the grate

A, B, C, D, E, F: measuring ports of temperature (left, right side)

a) The WtE plant under study and locations of measuring ports

b) close view of the grate

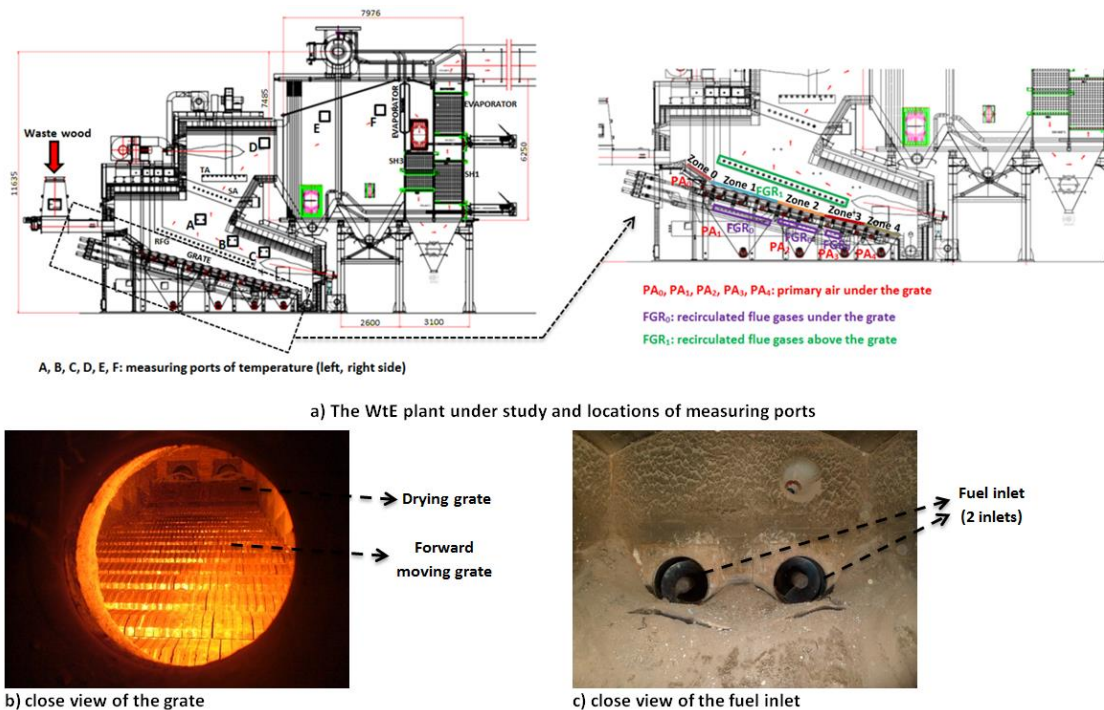c) close view of the fuel inlet

Figure 1. A detailed sketch of the WtE plant under study with indicated measuring ports, close view of the grate and fuel inlet.

In the primary chamber, the combustion process is managed with an air deficiency (about 70% of the stoichiometric value), so the pyrolysis-gasification process dominates (Kokalj et al., 2005). Volatiles and flue gases are then travelled into the secondary combustion chamber for complete combustion. The temperature of the created gases leaving the fuel bed from the primary combustion chamber are typically between 650-850 °C, and a large amount of generated heat is used in endothermic pyrolytic processes.

In the secondary combustion chamber supply of secondary air (SA) in the mixing zone generates an optimum combustible mixture of air and volatile gases. In this zone mixture is ignited. Complete combustion is assured by correct mixing procedure and by supplying tertiary air (TA). The temperature in secondary combustion chamber is around 850-1.100 °C with residence time of at least two seconds. These conditions ensure the complete combustion of the organic substances, as well as PCB's, PAH's and PCDD/F possibly formed in the primary combustion chamber.

In this WtE plant 44 RFG nozzles are located near the fuel bed in the primary combustion chamber and 54 individual SA and TA air nozzles in the secondary combustion chamber of different orientations. Some of the RFG, SA and TA nozzles are indicated in Figure 1a. The SA and TA jets of different momentums and orientations play a very important role in mixing of flue gases, combustion quality and emission characteristics.

The profile of primary air (PA) flux lengthwise along the grate is controlled in design by adjusting the air flow rate out of the four wind-boxes which are located under the grate and using non-uniform distribution of small holes on the grate. Additionally the flux of the PA is mixed with recirculated flue gases (RFG) from beneath the grates based on the combustion settings and fuel type. Additionally, the RFG are supplied above the grate through individual nozzles (Figure 1a). The main purpose of using RFG, which is not often used in real WtE plants, is reducing risk of slagging and lowering combustion temperature. Complete combustion process takes place inside the secondary combustion chamber. In the boiler is only heat transfer between secondary chamber and boiler (there is no combustion at all inside the boiler). There is certainly dominant radiant and convective heat transfer. Based on the PA and RFG flux, the grate may be divided lengthwise into four zones: pre-grate (zone 0), zone 1, zone 2, and zone 3, zone 4 as indicated in Figure 1a.

The majority of moisture in the waste wood is expected to be released in the first half of the grate (zones 0-3). Most of the devolatilization process is released in zone 1 and zone 2 and the rest released in the zone 0. Most of the char oxidation is assumed to occur in zone 1 and zone 2 (40% and 35%) because the the fuel bed is mostly reduced in these two zones whilst the rest is split onto the zone 0, zone 3 and zone 4 (Figure 3).
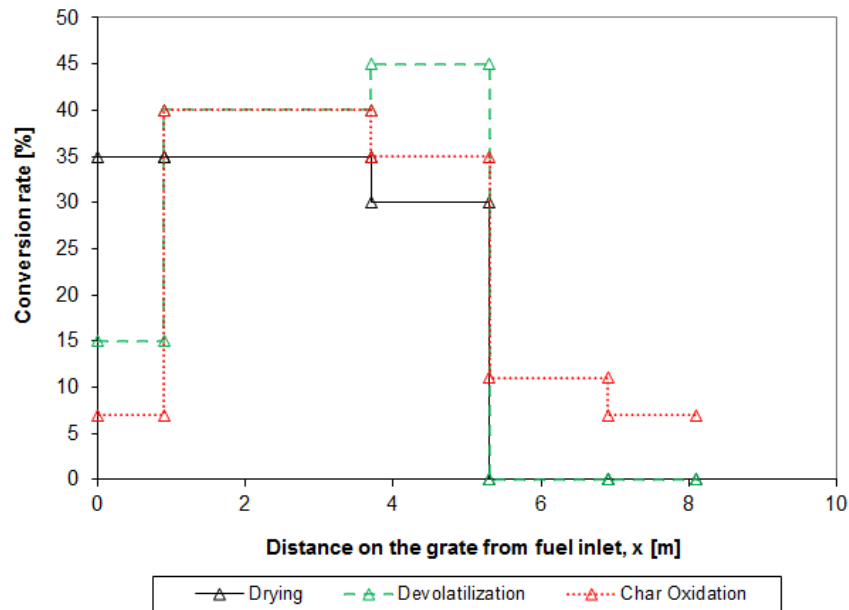

Figure 2. Waste wood

Figure 3. Expected and estimated fuel conversion rates along the grate.

The 4h experimental measurements are conducted mainly to determine the averaged gas temperature on selected measuring ports as sketched in Figure 1a. The boiler was operating comparatively steadily under a condition close to 50% load during the observations. RFG is extracted from the point after filter and it returns back into primary combustion chamber (below and above the grate). Higher % of oxygen in RFG is due to continuous filter cleaning with air, and due to air leakages into the system. Heat transfer coefficients on various walls are estimated and calculated based on the known composition of each wall. Table 1 summarizes the operating conditions averaged over the 4-h measurements.

3. CFD MODELING OF THE WtE PLANT

Modeling of waste combustion in a grate-fired boiler consists of two parts: modeling of waste conversion in the fuel bed on the grate and CFD simulation of gas phase combustion in the freeboard. These two processes are strongly coupled by the combustion gas leaving the fuel bed into the freeboard and the radiative heat flux emitted by the flame and furnace walls onto the fuel bed. The coupled modeling methodology needs to iteratively switch between the in-bed fuel conversion modeling and the freeboard CFD simulation, until there is no substantial change in either the combustion gas leaving the fuel bed or the radiative heat flux incident onto the fuel bed. This method has been successfully widely used in fixed bed combustion and grate-firing modeling (Yin et al., 2008). CFD simulations are also done for the condition given in Table 1.

Table 1. The operating conditions averaged over the 4-h measurements.

1) Waste wood: Feeding rate and the composition

Feeding rate	Moisture	Ash	C	H	O	N	Volatiles	Fixed carbon	Lower heating value
[kg/s]	%wt (as received)				%wt (dry)				MJ/kg (as received)
0.55	27.12	17.16	46.95	6.1	27.38	1.81	57.54	24.7	11.1

2) Primary air (PA): distribution along the lengthwise direction of the grate and the composition

2.1) Composition of PA in %wt

O_2		H_2O		N_2	
23		0		77	

Five lengthwise zones	Pre-grate (Zone 0)	Zone 1	Zone 2	Zone 3	Zone 4
Length [m]	0.9	2.8	1.6	1.6	1.2
PA flow rate [kg/s]	0.0481	1.2915	0.5329	0.322	0.214
PA temperature [°C]			32		

3) Recirculated flue gases (RFG): distribution along the lengthwise direction of the grate and the composition

3.1) Composition of RFG in %wt

O_2	H_2O	CO_2	N_2
12.1	4.7	8.1	75.1

Five lengthwise zones	Pre-grate (Zone 0)	Zone 1	Zone 2	Zone 3	Zone 4
Length [m]	0.9	2.8	1.6	1.6	1.2
RFG flow rate [kg/s]	0	0.6	0.3	0.1	0
RFG temperature [°C]			165		

4) Flow rate distribution of RFG above the grate, Secondary air (SA) and Tertiary air (TA):

RFG flow rate [kg/s]	1.56
SA flow rate [kg/s]	1.08
TA flow rate [kg/s]	0.36
RFG temperature [°C]	165
SA temperature [°C]	30
TA temperature [°C]	50

5) Wall heat flux

No.	Wall	Estimated Heat Transfer Coefficient [W/m² K]	Outside Temperature [°C]
1	Primary combustion chamber	2	20
2	Secondary combustion chamber	2	20
3	Boiler – first part	10	20
4	Boiler – second part	39	20
5	Bottom half of the boiler	0.367	20

3.1 Fuel bed conversion modeling

Different bed models for waste combustion on the grate will lead to very different lengthwise profiles of velocity, species and temperature of the gas mixture leaving the fuel bed and entering the freeboard (i.e., profile along the grate length). Such profiles will be used as the inlet boundary conditions for the subsequent freeboard simulation. The big differences in the profiles

resulted by different models will be virtually restricted to the vicinity of the fuel bed due to strong turbulent mixing in the freeboard induced by advanced SA supply system. This means that a reliable freeboard simulation can still be achieved, even with a very simple bed model, as long as the simple bed model can feed a correct flux of gas species, velocity and temperature into the freeboard.

In our case the waste conversion in the fuel bed is modeled by an empirical 1D bed model, based on the feeding rate and composition of the waste wood, the flux of the PA which is mixed with RFG from beneath the grates and the incident radiative heat flux at the top of the fuel bed. The bed model provides profiles of temperature, species concentrations and velocity of the gas leaving the fuel bed into the freeboard, which serve as the grate inlet conditions for the freeboard CFD simulation.

Based on the expected and estimated fuel conversion rates as a function of the lengthwise position on the grate (Figure 3), the combustible gas released from the fuel bed into the freeboard can be calculated from the overall heat and mass balances.

In this study, the waste wood volatiles are lumped into one single "species" $CH_{3.27}O_{0.924}N_{0.07}$. It's enthalpy of formulation is determined from the waste wood analysis data to be -11.533 kJ/kg. Char in the fuel bed is assumed to be oxidized into Carbon Monoxide (CO) and it is the only product of the in-bed char oxidation. Finally six species (i.e., $CH_{3.27}O_{0.924}N_{0.07}$, O_2, H_2O vapor, CO, CO_2 and N_2) are considered in the gas released from the fuel bed into the freeboard. The calculated lengthwise profiles at the fuel bed top are then used as the inlet boundary condition for the freeboard CFD simulation. Figure 4 shows the profiles of gas species, velocity and temperature at the fuel bed top when the coupled simulation is converged. The final net radiative heat transfer incident onto the fuel bed top is converged to 0.62 MW, which is used in the calculation of the profiles in Figure 4.
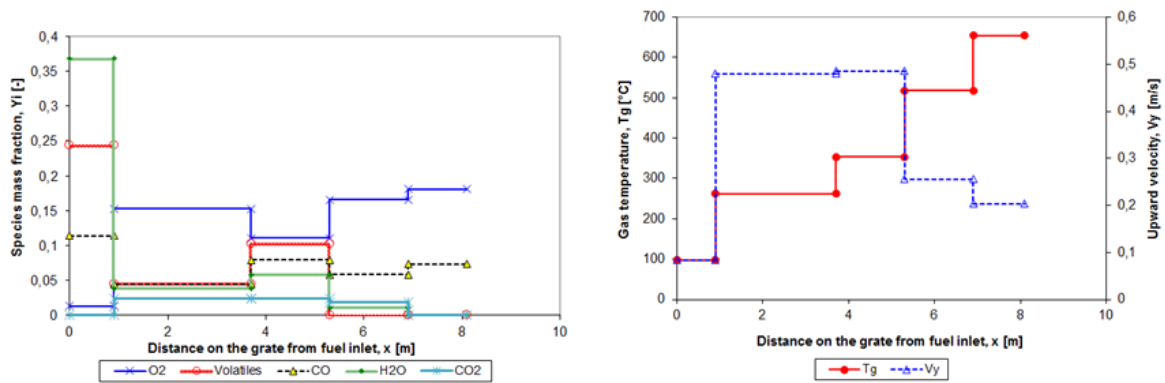

Figure 4. Lengthwise profiles of temperature, species concentrations and velocity at the fuel bed top when the coupled simulation is converged.

3.2 CFD modeling in the freeboard

The 3D CFD simulation in the freeboard is typically gas-phase combustion modeling. The grate inlet conditions for the freeboard CFD simulation and RFG, SA and TA inlet conditions at different nozzles are defined by Figure 1 and Table 1. The boundary conditions at different walls are estimated with heat transfer coefficient and outside temperature (Table 1). The 3D CFD simulation is done using commercial package ANSYS CFX.
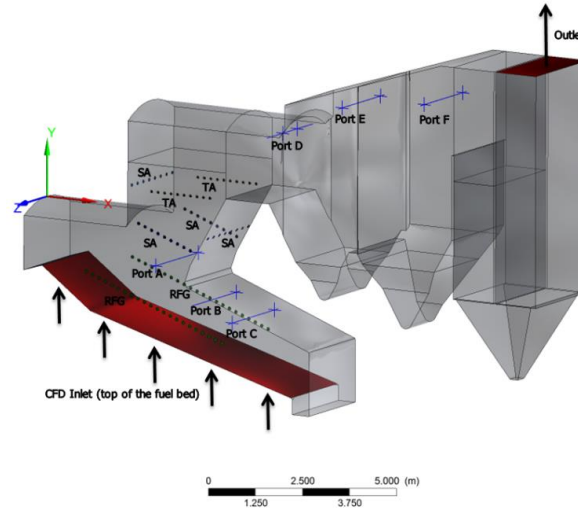
Figure 5. CFD model.

The gas flow in the freeboard is taken as the ideal gas mixture including volatiles, O_2, CO, H_2O vapour, CO_2 and N_2. These components are assumed to mix at the molecular level, sharing the same mean velocity, pressure and temperature (Ansys, 2011). Temperature-dependent specific heats and enthalpies in NASA format are used for all the gas species to correctly calculate the gas flow velocity and temperature.

Turbulence is modeled by the most widely used standard k-ε model. Thermal radiation through the gas phase is modeled using Discrete Transfer (DT) Model. The DT assumes that the spatial radiation gradients are relatively small and the radiation is emitted isotropically from the surfaces. The user defined parameter 'number of rays' in CFX determines the degree of spatial discretisation of the hemisphere above each finite surface element for radiation emission. Higher values mean a better representation of the reality and a higher accuracy, but at the same time will result in a significant increase in computational effort. The paths of rays are calculated only once, at the beginning of the simulation, and are then stored and re-used which leads to significant savings of computer memory (Zitzmann et al., 2007).

A large and complex chemical reaction scheme is required to obtain a high level of detail in the combustion process. However, as this analysis is not focussed on detailed chemistry, a chemical reaction scheme involving the elements C, H, O and N will be sufficient. A two-step global chemical reactions with CO as the intermediate species is used for combustion of volatiles and eddy-dissipation (ED) model with modified mixing parameters (A = 0.6, B = 0.5). ED model based on an assumption that chemical reactions are faster in comparison to diffusion processes.

$$CH_{3.27}O_{0.924}N_{0.07} + 0.8555O_2 => 1CO + 1.635H_2O + 0.035N_2 \quad (1)$$

$$CO + \frac{1}{2}O_2 => CO_2 \quad (2)$$
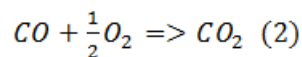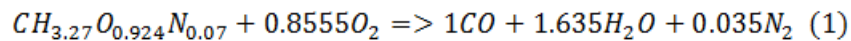
The selection of suitable kinetic reaction rates for chemical reactions is a very difficult task due to a shortage of data as well as ambiguity and inconsistency in scientific resources. After careful review the data of Yin et al. (Yin et al., 2012) have been used, which are defined as:

$$\frac{d[CH_{3.27}O_{0.924}N_{0.07}]}{dt} = 5 \times 10^{12} \cdot exp\left(-\frac{2 \times 10^8}{R_u T}\right)[CH_{3.27}O_{0.924}N_{0.07}]^{0.2}[O_2]^{1.3} \quad (3)$$

$$\frac{d[CO]}{dt} = 2.24 \times 10^{12} \cdot exp\left(-\frac{1.7 \times 10^8}{R_u T}\right)[CO][O_2]^{0.25} \quad (4)$$

It is necessary to set up the correct orientations of the SA, TA and RFG nozzles in order to ensure and represent the actual operating conditions of WtE plant (Figure 6).
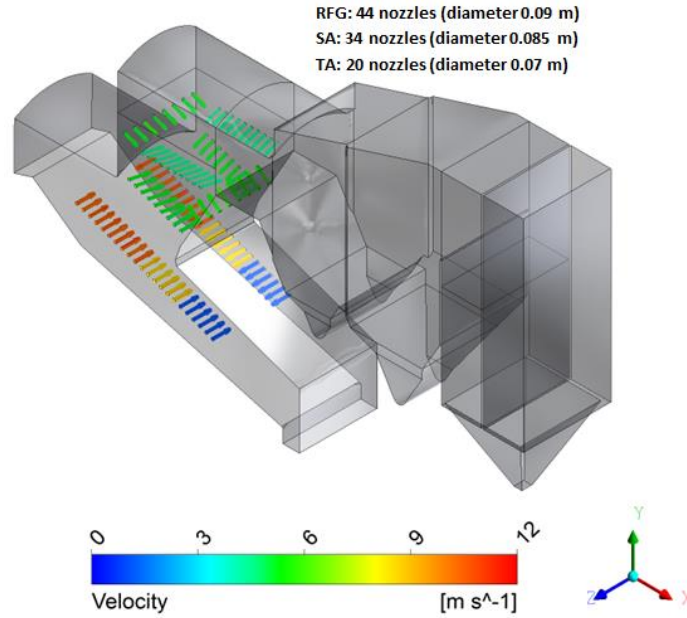

Figure 6. Orientations of RFG, SA and TA nozzles.

Buoyancy effects are considered in CFD model with Full Buoyancy model, so a source term is added to the momentum equations, as follows:

$$S_{M,buoy} = (\rho - \rho_{ref}) \cdot g \quad (5)$$

where ρ_{ref} is calculated with an expression as averaged density inside the WtE plant:

$$\rho_{ref} = volumeAve(\rho)@Domain \quad (6)$$

The mass, momentum, chemical species and energy equations are discretised using the finite volume approach. The discretised gas continuity and momentum equations are solved in a fully coupled manner. The convergence criterion for gas phase properties is 10^{-4} for the RMS residuals.

Great attention was given to appropriate mesh selection. For the freeboard CFD simulation, the majority of the efforts were taken to generate a fine and high-quality mesh. The mesh which is used in this study is unstructured tetrahedron-dominant mesh in which local boundary layer and advance size functions are used (Figure 7). The quality specifications of the mesh and number of elements are summarized in Table 2. Mesh always plays a crucial role in CFD simulations; it largely affects the convergence and significantly influences the reliability of the CFD results.
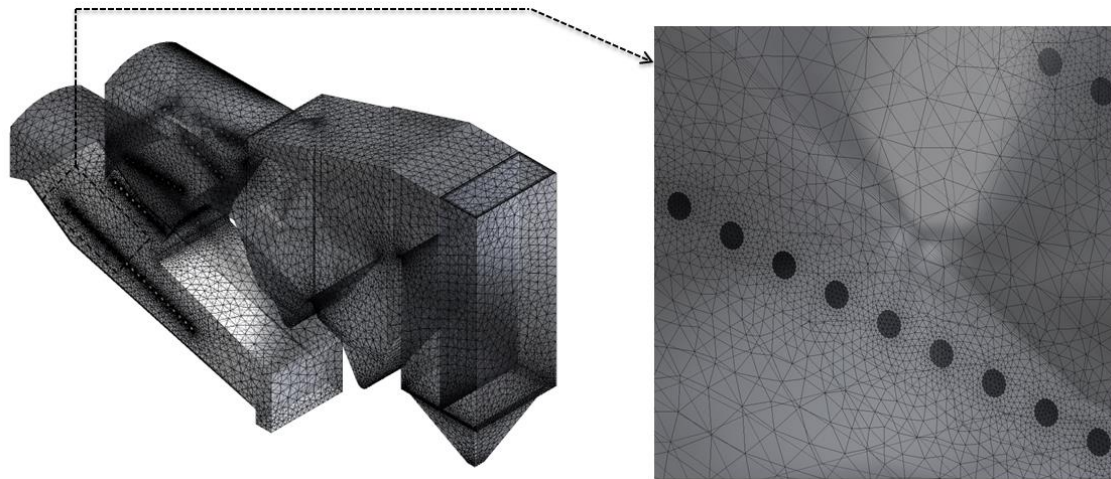
Figure 7. Discretized geometry and detailed view illustrating the RFG nozzles.

Table 2. Details of the quality of mesh

Quality-type specification	The mesh of this grate boiler: 2.048.923 elements in total; - Tetrahedra: 1.574.298 - Pyramids: 13.606 - Wedges: 461.019	
	Range	Average
Skewness	[0, 0.88]	0.254
Aspect ratio	[1.164, 3267.5]	5.633
Jacobian Ratio	[1, 1226.2]	1.05
Parallel Deviation	[0, 160.18]	1.637
Ortogonal Quality	[0, 0.98]	0.84
Warping Factor	[0, 0.264]	0.009

4. RESULTS AND DISCUSSION

The predicted CFD temperature distribution for the middle plane between side walls is shown in Figure 8 for the two cases: without buoyancy effects (left) and with buoyancy effects (right). The results show that neglecting buoyancy effects may lead to appreciable errors. As expected, the temperature distribution mainly affects the flow field when buoyancy is considered in the calculations. Relatively high flue gas temperatures are found in front of the primary combustion chamber. Drying of the waste on the first part of the grate is therefore efficiently. Just as expected, the highest temperatures are found in the burnout zone in the secondary combustion chamber where SA and TA are mixed with the combustible gases released from the primary combustion chamber.
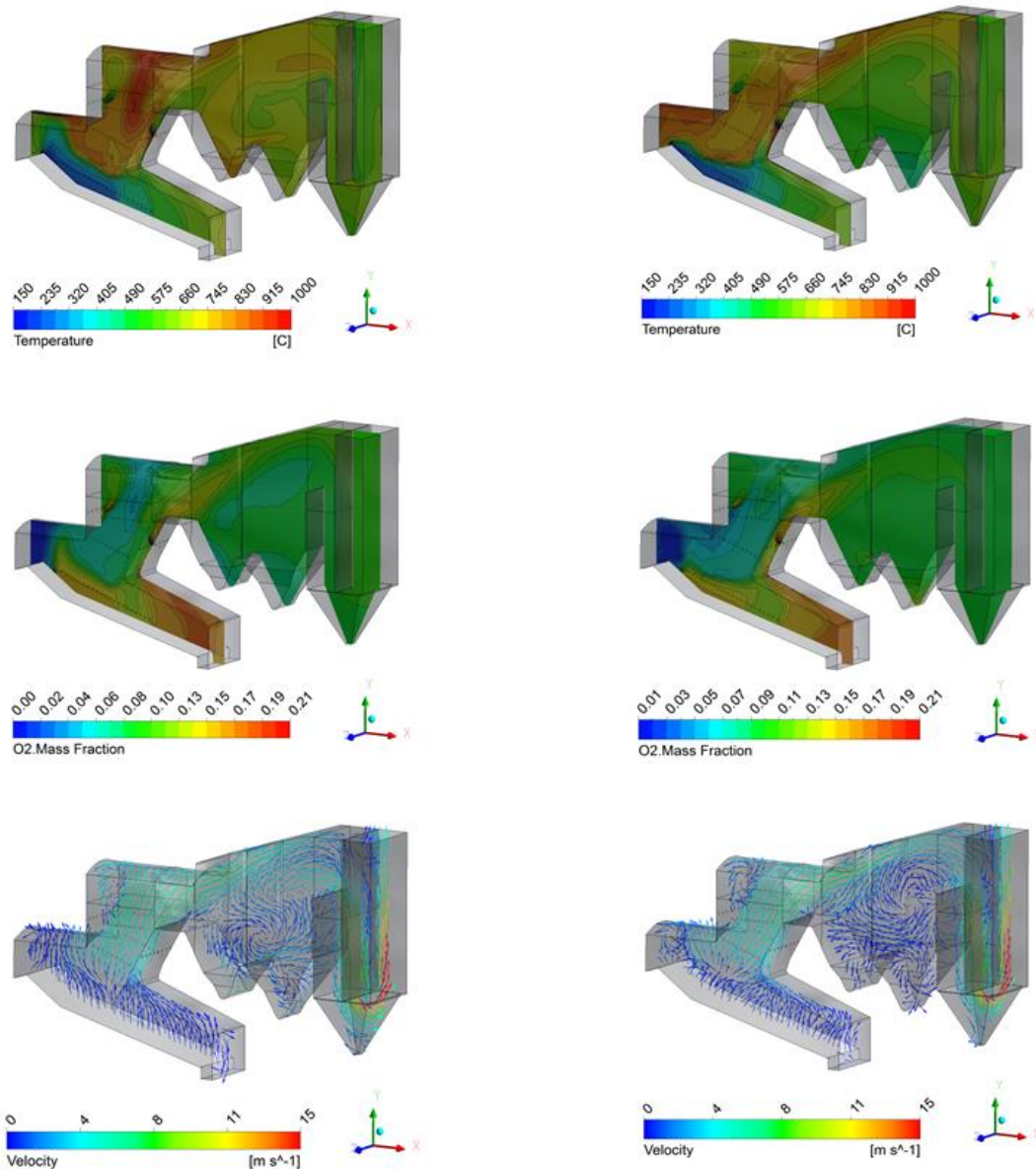
Figure 8. Gas temperature distribution, O_2 mass fraction and velocity vectors in the middle plane between two side walls for without buoyancy effect (left) and with buoyancy effect (right).

The velocity field shows relatively low velocities in primary combustion chamber. Large recirculation zone is located on the upper left edge of the secondary combustion chamber with relatively low velocities (poor utilization of furnace volume). It is also visible that poor mixing of flue gases is applied and need to be optimized with a new concept of SA and TA nozzles to provide better mixing. Averaged inlet velocity at individual air nozzle of SA and TA is 5 m/s (Figure 6), which is too low for effective mixing of flue gases. High velocities along some of the boiler walls may cause erosion and corrosion problems.

The comparison of CFD results of temperature on measuring ports with measured during the experimental campaign in Figure 9 shows that the numerical results with the buoyancy effects are better and more realistic. Finally, these results are in good agreement (deviation is less than ± 50 °C) with experimentally measured.
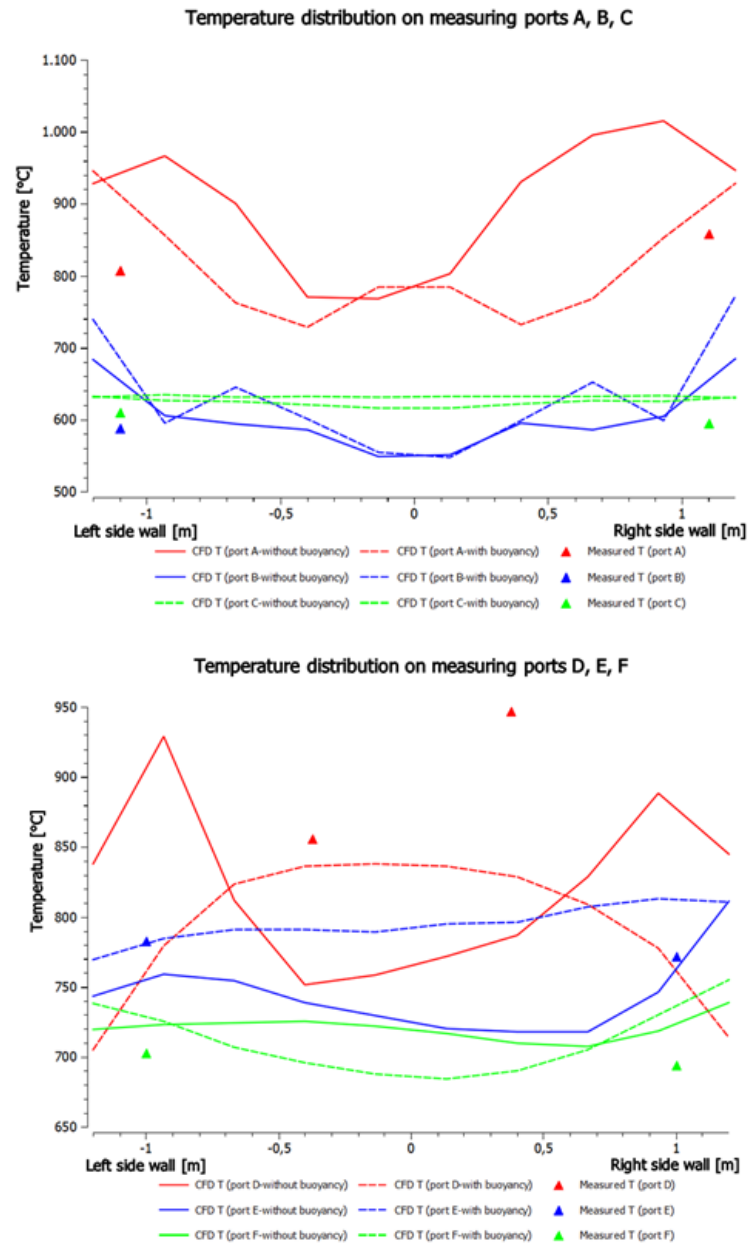
Figure 9: Comparison of CFD temperature with experimentally measured on measuring ports.

Based on the CFD analysis it was found (Figure 10) very high temperatures exist on certain walls (especially in secondary combustion chamber), which are very likely to cause severe problems with melted slag on refractory walls. The melted slag on refractory walls in primary and secondary combustion chamber is shown in Figure 11.
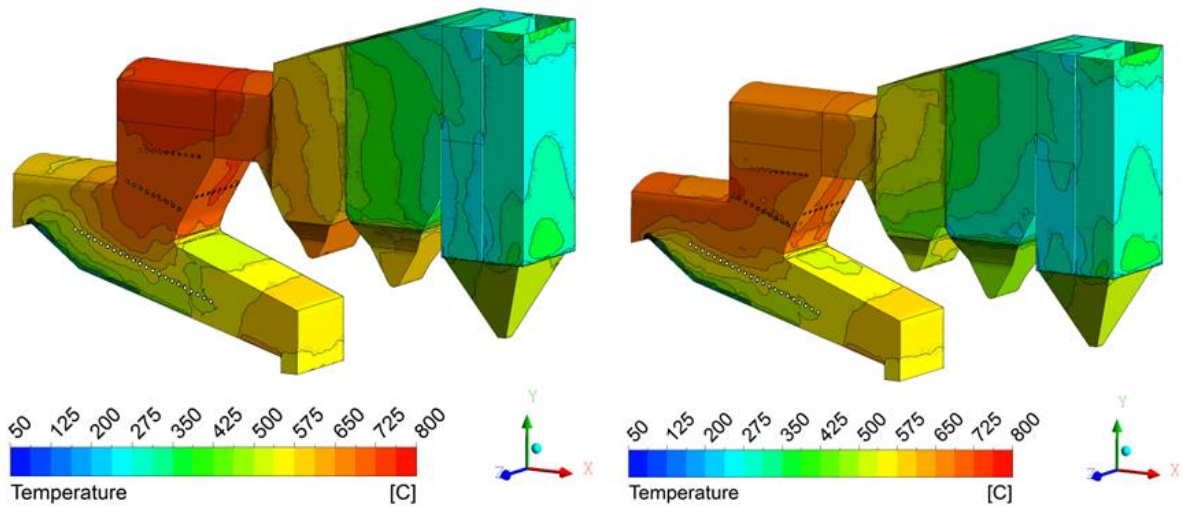
Figure 10. Temperatures on walls for without buoyancy effects (left) and with buoyancy effects (right).

Figure 11: Melted slag on refractory walls.

5. CONCLUSIONS

To sum up, this paper illustrates that our CFD modeling methodology of the WtE plant is appropriately accurate. The empirical 1D bed model can correctly predict the inlet boundary conditions for the freeboard 3D CFD simulation. The overall mixing in the WtE plant is comparatively poor. The air supply of SA and TA nozzles and RFG nozzles need to be optimized and improved. In future, fly ash particles and ash deposition to investigate the slagging behavior inside the WtE plant need to be included in the freeboard CFD numerical model. The comparison between the CFD predictions and experimental data demonstrates that the CFD model is reliable and suitable for the freeboard modelling in WtE plants if the buoyancy effects are considered.

In industry it is possible to use CFD as a basic design tool to optimize the WtE plant and their boiler design with respect to a large number of critical factors such as temperature distribution, velocity field, oxygen level, wall temperatures and emissions for better performance. Additionally, the positions and jet momentum of various air injection streams, the geometry of

the furnace and boiler, wall heat transfer properties are therefore all investigated thoroughly during the design phase using CFD as one of the main tools. Therefore a 3D CFD simulation will help to better plan/design and optimize the operation of the WtE plants.

REFERENCES

Yin C., Rosendahl L., Kær S. K. (2008). Grate-firing of biomass for heat and power production. Progress in Energy and Combustion Science, vol. 34, 725–754.

Yin C., Rosendahl L., Kær S. K., Clausen S., Hvid S. L. (2012). Characterizing and modeling of an 88 MW grate-fired boiler burning wheat straw: Experience and lessons. Energy, vol. 41, 473–482.

Kokalj F., Samec N., Škerget L. (2005). An Analysis of the Combustion Conditions in the Secondary Chamber of a Pilot – Scale Incinerator Based on Computational Fluid Dynamics. Journal of Mechanical Engineering, vol. 51, 280-303.

Zitzmann T., Pfrommer P., Cook M. J. (2007). Dynamic thermal building analysis with CFD-modelling radiation. Proceedings: Building Simulation, Beijing, China, 31-38.

ANSYS CFX-Solver Theory and Modeling Guide (2011). Release 14.0, ANSYS, Inc.