

Improving the capacity of a MSW incineration plant.

Per Ankersen, Paw Andersen, J.V. Nielsen.

Babcock & Wilcox Vølund ApS, Falkevej 2, DK-6705 Esbjerg Ø,
Denmark

Abstract

A CFD case study of the Orvade Municipal Solid Waste (MSW) incinerator in Saran in France was conducted to investigate the possibility of improving the capacity of the plant and to ensure compliance with European Union (EU) environmental legislation, in particular retention time and CO emission requirements. Two cases were analysed, the pre-modified state (operation prior to modification) and the modified state (capacity increase of 12%). The CFD model predicted that the pre-modified state did not meet retention time requirements. The modified state demonstrated the ability to meet the requirements of retention time and a reduction in CO.

Keywords: CFD analysis, retention time, improving waste incineration capacity.

1 Introduction

Babcock & Wilcox Vølund (BWV), a subsidiary of McDermott Inc., has since 1931 been designing, constructing and commissioning waste-to-energy plants and other energy conversion systems for the global market.

In 1996, BWV began using CFD (Fluent) as a tool in the design of new plants and in troubleshooting and service projects at existing plants.

BWV's many years of hands-on experience within operation and troubleshooting means that our use of CFD is linked to a vast amount of know how. As a large proportion of the predictions obtained from CFD analysis rely on this experience, BWV engineers have a clear idea of what to expect from this modelling in terms of quality vs. quantity. This issue is discussed further in the introduction to section 3.1.

Typical service projects cover issues such as emission control, corrosion problems, increases in incineration capacity, slag and fouling problems and changes of waste specifications such as calorific value.

This paper will present some of the results of a CFD analysis which was part of a service project for a French client.

2 Background

The Orvade MSW incinerator, which is located in Saran in France, was originally designed for a steam production of approximately 17t/h per boiler. The client's experience was however that steam production was only approximately 15t/h. The aim of this service project was therefore to modify the plant to increase steam production to approximately 17t/h, whilst ensuring that the plant maintained its compliance with the EU requirement of a minimum retention time of 2 seconds at 850°C (Directive 2000/76/EC).

3 The service project

A complete CFD service project typically consists of two cases:

- 1) A base case, which is a CFD analysis of the incinerator in its current mode of operation and design.
- 2) A modified case, where a selection of features is tested to determine their ability to solve the problems under investigation.

3.1 General model features

The engineering approach applied to the present problem in terms of grid generation and simplifications of boundary conditions and sub-models is not believed to enable prediction of chemistry and flow field with great precision and a high level of detail. The analysis therefore focuses on trends. Examples of such trends include posing the following questions: Can the modifications improve flow conditions in the furnace? Does CO escape from the furnace? Is retention time prolonged and at what margin does it exceed the requirements etc.?

3.1.1 Boundary conditions

3.1.1.1 The bed

The top of the waste layer is modelled as a series of velocity inlets. Refer to the left view of Figure 2.

The waste bed was modelled using the external bed model developed by Jørgensen and Swithenbank (1997), which is based on waste specification, calorific value, the waste mass flow and primary air flow.

The output from the bed model includes values for gas temperature, mass flow of gas, species concentrations, particle mass flow and temperature. These are read into the fluent grid as boundary conditions for the velocity inlets for the top of the waste layer on each grate. The gas from the bed model is a mixture of O₂, CH₄, CO, H₂O and CO₂.

3.1.1.2 Walls

Heat transfer to the walls is dependant on the quantity and quality of ash deposits. Heat transfer properties of furnace and boiler walls take this into account through adjustment of heat transfer coefficients based on BWV's modelling experience.

3.1.2 Sub models

The following Fluent sub models are normally applied (Table 1):

3.1.2.1 Heat transfer by radiation

Radiation is modelled using the Discrete Ordinates Model (DO).

Fluent provides a number of models for radiation heat transfer, two of which support particle radiation. Of these two, the DO model is insensitive to optical thickness and less sensitive to local heat sources (Fluent, 2001).

The Fluent concept of the DO model is used to evaluate radiation intensity for a finite number of fixed directions in space. The directions are defined by the user. This means that radiation intensity is calculated on every cell face where there is incoming or outgoing radiation from or to the surrounding gases, particles or surfaces.

3.1.2.2 Turbulence

The RNG k- ϵ model is applied to flow modelling. The k- ϵ model is selected for flow modelling to overcome the so-called *closure problem* of the Reynolds averaged Navier-Stokes equations (RANS) (Versteeg & Malalsekera, 1995).

Fluent provides a number of turbulence models, the RNG k- ϵ model being one of these. The concept of the k- ϵ model is to substitute the unknown Reynolds stresses in the RANS by means of turbulent kinetic energy (k) and the dissipation rate of the turbulent kinetic energy (ϵ). In Fluent 6, the k- ϵ model is based on work carried out by Launder and Spalding (1974).

Fluent has, however, included the principles of the so-called *Renormalization Group* (RNG) in the k- ϵ model, which should provide better accuracy particularly for the dissipation rate.

We have used the Fluent default model parameters.

3.1.2.3 Particle transport

A Lagrangian approach has been applied to the modelling of particle transport. To account for the impact turbulence in the flow field has on particles trajectories, Fluent integrates the instantaneous fluid velocity ($\bar{u} + u'(t)$) at any current position of the particle. By calculating $u'(t)$ as function of the turbulent kinetic energy (k) and a normally distributed random number, Fluent argues that particle tracking is linked to the turbulent nature of the flow (Fluent, 2001).

3.1.2.4 Boundary layer

To save computational resources and because detailed information of conditions in the boundary layer is not essential in the analysis, near wall phenomenon are treated using a wall function instead of dissolving the boundary layer region in a fine mesh.

The Fluent Standard Wall Function, which is based on work carried out by Launder and Spalding (1974), provides a connection between variables in the near-wall cells and their counterpart on the wall.

The variables in the near-wall cells will in general become a function of the distance from the cell node to the wall and of the turbulent properties (k and ε) for that cell. Refer to Fluent (2001) for further details.

3.1.3 Reactions

Following the principles presented in Jørgensen and Swithenbank (1997), which are based on the work carried out by Sharifi (1990) and Dvirka (1976), a simple methane combustion scheme is considered sufficiently productive:

The main gas components produced from gasification of waste are CH₄, CO and H₂. A large and complex 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 reaction scheme involving the elements C, H and O will be sufficient. Methane combustion is, in addition, well documented (Turns, 1996).

The homogeneous reactions are therefore simplified to:



The reaction rate will in general be controlled by a combination of an Arrhenius expression and the eddy-dissipation model, the latter as described by Magnusson and Hjertager (1976). In the eddy-dissipation model, reaction rate is a function of turbulent kinetic energy (k) and turbulent dissipation rate (ε). The flow must therefore be turbulent for a reaction to occur. Reaction rates for (1) and (2) are the lower of either the reaction rate obtained by the

Arrhenius expression or the reaction rate computed by the eddy-dissipation model. Fluent default parameters are applied to the reaction rate calculations. The reaction rate for (3) is controlled only by the Arrhenius expression.

3.2 *The base case*

3.2.1 Information for the base case

A BWV CFD engineer visits the client to determine and define the current mode of operation. The mass and energy conservation data required is obtained from readings recorded in the control system history log. General information from the client also provides an understanding of special operational conditions such as waste type and the location of molten slag and ash deposits. The client must submit blueprints for the grid to be created.

In the current example, the base case is burning 6.25t/h MSW with a lower calorific value of 8.1 MJ/kg. More details are listed in Table 2.

3.2.2 The base case: geometry and grid

The geometry of the furnace and boiler in the pre-modified state is shown in Figure 1. The orientation of the furnace/boiler is usually

based on the direction of the waste flow. In the left view of Figure 1, waste is transported from left to right, i.e. in the x direction. The wall to the left, when looking in the x-direction, is designated *the left side* (of the furnace/boiler) and vice versa. The left view in Figure 1 therefore shows the right side of the furnace and boiler. Predictions are analysed in selected cross sections ('data output planes') as shown in the right view of Figure 1. The red plane is the geometrical symmetry plane of the furnace and the boiler. The green plane is parallel to the symmetry plane but at a distance of 500 mm from the left furnace wall.

The computational grid for the base case geometry is shown in Figure 2.

The furnace is meshed with tetrahedral cells and the boiler is meshed using hexahedral cells. The mesh is dense close to the secondary air nozzles. During computation, the mesh is adapted with respect to y^+ and gradients of temperature and velocity. The grid counts approximately 150,000 cells.

Figure 3 shows how the secondary air nozzles are positioned and their direction in the pre-modified state, i.e. the base case. Position and direction of these nozzles is very important as they control the flow field in the furnace. It can be seen in Figure 3 that there are

two rows of nozzles positioned in the roof of the furnace. One row is positioned on the inclined section of the roof, pointing towards the rear of the furnace (*SA1*) and one row is positioned on the horizontal part of the roof aimed at the grate (*SA2*). A further system of nozzles is located on each side of the furnace, just before the entrance to the 1st pass. These side nozzles operate in pairs. The nozzle pairs on the left side of the furnace, *SA3L* and *SA4L*, face their counterparts on the right side of the furnace, *SA3R* and *SA4R*. The designation indicates that the nozzles inject secondary air (SA) and the 'L' and 'R' identify whether they are located on the left (L) or the right (R) side of the furnace.

The distribution of secondary air in the base case is shown in Table 3. The amount of cooling air through the air cooled walls is 7,375 Nm³/h. The location of the air cooled walls can be seen in Figure 2.

3.2.3 The base case: predictions and analysis

A CFD analysis produced by BWV is usually based on a selection of predicted species concentrations, temperatures and velocities examined in a large number of planes. Given the number of planes analysed, it generally is unnecessary and of little value to present and describe each plane to the client. We therefore usually present

a more general picture to our clients, such as predictions in the symmetry plane such as the right view in Figure 4.

The right view in Figure 4 shows the predicted temperature distribution in the symmetry plane. The interpretation of this is as follows:

- Main combustion zone is located on grate 2.
- Relatively low flue gas temperatures are found in front of the furnace. Drying of the waste on grate 1 is therefore inefficient.
- Temperature distribution above grate 3 is influenced by the secondary air nozzles (SA4L and SA4R).
- High temperatures are found along the deflector.
- Temperature distribution in the boiler reflects the flow distribution (refer to Figure 5).

The temperature distribution in the left side of the furnace indicates (Figure 4 left view) the following:

- Large influence of the cooling air injected through the cooled walls. Cooling air is approximately $430\text{Nm}^3/\text{h}$ pr. m^2 cooled wall. BWV normally uses $300\text{Nm}^3/\text{h}$ pr. m^2 cooled wall.
- Large volume of the furnace is at relatively low temperatures, therefore less space for efficient combustion, i.e. poor utilisation.

The flow pattern in the symmetry plane is shown in Figure 5:

- Low velocities in front of furnace, i.e. poor drying of waste on grate 1.
- Large re-circulation zone with relatively low velocities along roof of furnace, i.e. poor utilisation of furnace volume.
- Main flow in the 1st, 2nd, 3rd and top of 4th boiler passes along the back wall. Heat transfer is therefore less effective.
- High velocities along some of the boiler walls may cause erosion/corrosion.

Figure 6 shows oxygen concentration in the symmetry plane.

- As expected, oxygen concentration is low in the area where combustion is most intense. Refer also to the temperature distribution in Figure 4.
- Influence of re-circulation zone is clearly shown.
- Area with low O₂ concentration in bottom of 1st boiler pass, indicates that combustion is not completed within the furnace.

The CO₂-concentration is expected to be high in the main combustion zone where the oxygen concentration is low. This corresponds well with the picture in Figure 7.

The variation in CO₂ concentration in the 1st boiler pass indicates poor mixing of the flue gasses in the furnace.

Figure 8 shows CO concentration in the symmetry plane.

- As expected, concentration is highest immediately above the grates.
- CO is relatively high after the last injection of secondary air (i.e. the secondary air nozzles on either side of the furnace just before the entrance to the 1st pass. Refer to Figure 3). This makes the furnace/boiler vulnerable to variations and instabilities that can result in large variations in the CO-concentration. In a pre-modified state, the plant experienced variations of $\pm 17\%$ and it is expected that large variations in the CO emissions from the plant can be found. Because the model is a steady state model, these variations cannot be captured and the CO emission is underpredicted.

Before turning to the retention time problem, we will look at the analysis of the modified furnace.

3.3 The modified case: changing the secondary air system

The analysis of the base case demonstrates that the secondary air system in the pre-modified state produces a large re-circulation zone in the furnace which tends to push the combustion gasses in

the direction of the boiler inlet. This significantly reduces the efficient furnace volume.

The strategy behind furnace modification therefore was to change the secondary air system in such a way that gasses are retained in the furnace for complete burnout.

BWV has achieved good results with a different type of secondary air concept, which uses a smaller number of nozzles positioned in the furnace roof. The modified case uses this secondary air system. Refer to Figure 9. No changes in the geometry were made other than changing the secondary air system.

3.3.1 The modified case: operational data

A number of operational modes were applied to the geometry in Figure 9 and analysed using CFD in the complete service project. Due to the limited space available in this paper, just one example has been selected for comparison with the base case. The example chosen is based on the data in Table 2.

3.3.2 The modified case: predictions and analysis

Using the modified secondary air system and the modified operational mode as shown in Table 2, predictions were analyzed using the analysis method in the base case.

Compared to isothermals in the base case (Figure 1) the modified case (Figure 10) shows the following improvements:

- Temperatures near the furnace wall are less affected by the cooling air, which has been reduced by approximately 25%.
- Gasses are hot in the first part of the furnace which provides fast drying of the waste.
- Temperatures close to the deflector are lower, making the deflector less subject to thermal stresses.

Figure 11 shows the velocity vectors. The flow field in the furnace is completely changed: the recirculation zone in the furnace is eliminated due to the two jets from the secondary air nozzles now being located only on the inclined part of the roof.

The main flow in the boiler is, however, still concentrated along the back walls. Optimising the flow in the boiler was, however, not a part of the service project.

Figure 12 shows the concentration of oxygen.

- Concentration of oxygen is low in the first part of the furnace because combustion is most intense in this region. Refer to the temperature distribution in Figure 10.
- There are no local variations in oxygen concentration in the last part of the furnace. This indicates improved burnout.
- There are no local variations in oxygen concentration in the boiler. This indicates good mixing.

The distribution of CO₂ can be seen in Figure 13. High concentrations of CO₂ indicate high combustion intensity in the front of the furnace. This corresponds well with the temperatures predicted in Figure 10 and the distribution of oxygen in Figure 12.

From the prediction of CO distribution in Figure 14, it is clear that the modified secondary air system provides good control of combustion in the furnace compared with the base case secondary air system (Figure 8):

- CO does not escape from the furnace but is fully burnt before the flue gas enters the boiler.

3.4 Retention time: comparing the base case and the modified case

The modified secondary air system gives better combustion control than the pre-modified system. However, it has not been checked whether the retention time requirement can be met. The requirement in EU Directive 2000/76/EC specifies that flue gasses must remain above 850°C for at least 2 seconds after the final injection of combustion air.

One way of using CFD to demonstrate the ability of the incinerator to meet the retention time requirement is to define a control volume for the flue gasses to travel through. The volume inlet is a cross sectional plane positioned after the secondary air nozzles. The outlet volume is a cross section where the mean flue gas temperature is at least 850°C. If it takes at least 2 seconds for the flue gas to travel from the inlet to the outlet, the requirement is considered to be fulfilled. The control volume is therefore applied to a plug flow model, in which flue gas volume flow is based on the mean flue gas density inside the volume.

This method uses the same principles as are used when measuring the retention time on site (17. BImSchV). Note that the planes used to define the control volume have no resemblance to the data output planes in Figure 1.

The planes defining the control volumes used in the calculation of the retention time are shown in Figure 15. The left view shows the planes from the base case and the right view shows the planes from the modified case. The inlet planes are positioned after the last injection of combustion air. In the modified case, the last injection of combustion air is injected from the nozzles in the inclined part of the furnace roof and from the grates. In the base case, the side nozzles are located just below the boiler. The control volume inlet for the base case is therefore positioned much higher than the control volume inlet for the modified case. The outlet planes are located where the flue gas mean temperature is 850°C. Flue gas mean temperature is a non-weighted average, which is consistent with 17. BImSchV.

The data in Table 4 is used to calculate the retention time in the modified case. According to the 'source' column in the table, the data is extracted from the CFD model or calculated by using formulas (4) – (7). For the Modified Case $t_{retention} = 2.9$ s. For the Base Case $t_{retention} = 1.9$ s, using the same calculation.

Using data from the CFD models to calculate retention time therefore shows that the pre-modified case may not satisfy the

retention time requirement. Modifying the secondary air system can however solve this problem.

$$T_{volume} = \frac{T_{in} + T_{out}}{2} \quad (4)$$

$$\rho_{volume} = \rho_{gas} \cdot \frac{T_{gas}}{T_{volume}} \quad (5)$$

$$\dot{V}_{volume} = \frac{\dot{m}_{gas}}{\rho_{volume}} \quad (6)$$

$$t_{retention} = \frac{V}{\dot{V}_{volume}} \quad (7)$$

3.5 Remarks

The predictions are products of simplifications which are likely to introduce a level of uncertainty. Except maybe for grid dependence, estimating the level of uncertainty each simplification introduces is not possible.

It is imperative that, for a project with this length of time frame, we can draw on our experience from other client case CFD studies and our in-house know how. This CFD model performed as expected and corresponded well with experience from previous service projects. In addition, the results are reasonable when compared to operation data and observations made on site.

The interpretation of the results is furthermore based on trends rather than a detailed analysis of local variations. The solution has therefore not been checked for grid dependence.

4 Conclusion

BWV has used CFD since 1996 as a tool in design and troubleshooting in the waste-to-energy business.

BWV has successfully applied CFD in solving operational problems for this client, enabling the client to improve capacity by 12% and also comply with EU retention time requirements.

5 Acknowledgements

We would like to thank the management of the Orvade MSW incinerator for allowing us to use their incinerator as a case study for this paper.

6 Nomenclature

k	turbulent kinetic energy (m^2/s^2)
\dot{m}	mass flow (kg/s)
t	time (s)
T	mean temperature ($^{\circ}\text{C}$)

\bar{u}	mean velocity component (m/s)
$u'(t)$	fluctuating velocity component (m/s)
V	volume (m ³)
\dot{V}	mean volume flow (m ³ /s)
y^+	non-dimensional distance to wall (-)

Greek letters

ε	rate of dissipation of turbulent kinetic energy (m ² /s ³)
ρ	density (kg/m ³)

Subscripts

<i>gas</i>	flue gas
<i>in</i>	inlet surface
<i>out</i>	outlet surface
<i>retention</i>	referring to retention time
<i>volume</i>	referring to control volume for calculation of retention time

7 References

1. The European Parliament and the Council (2000)

Directive 2000/76/EC of the European Parliament and of

the Council of 4 December 2000 on the Incineration of Waste. Article 6.1

2. Jørgensen, K., Swithenbank, J. (1997) *CFD Simulation of Flow and Combustion in the Vestforbrænding 5 Furnace and Boiler*. Internal BWV document
3. Fluent Inc. (2001) *Fluent User's Guide for Fluent 6.0*
4. Länderausschusses für Immissionsschutz (April 1994) *Fachliche Erläuterung zur: Bundeseinheitlichen Praxis bei der Überwachung der Verbrennungsanlagen nach 17. BImSchV*, Erich Schmidt Verlag
5. Magnussen, B.F., Hjertager, B.H. (1976) *On mathematical models of turbulent combustion with special emphasis on soot formation and combustion*. In 16th Symp. (Int'l.) on combustion. The Combustion Institute.
6. Versteeg, H. and Malalsekera, W. (1995). *An introduction to computational fluid Dynamics. The finite volume method*. Longman.
7. Launder, B.E. and Spalding, D.B. (1974). *The Numerical Computation of Turbulent Flows*. Computer Methods in Applied Mechanics and Engineering, vol 3. pp. 269-289.
8. Turns, S.R. (1996). *An Introduction to Combustion, Concepts and Applications*. McGraw-Hill.

9. Sharifi, V.N. (1990). *Optimization Study of Incinerator in a MSW Incinerator with a Vertical Radiation Shaft*. Ph.D. thesis, University of Sheffield.
10. Dvirka, M. (1976). *Practical Application of Incinerator Burning Rate Equations*. Proc. of the National Waste Process Conference, 7th Bienn., in Boston, Mass., May 23-26 1976, pp. 547-563. Published by ASME, Solid Waste Process Div., New York, US.

8 Biographical notes:

M. Sc. Eng. and BA Per Ankersen is CFD Engineer in the Department of Technology Design. E-mail: pea@volund.dk.

M. Sc. Eng. Paw Andersen is a CFD and Combustion Engineer in the Department of Technology Design. E-mail: paa@volund.dk.

B. Sc. Eng. J. V. Nielsen is manager of the Department of Technology Design. E-mail: jvn@volund.dk.

Phenomena	Sub model
Radiation	Discrete Ordinates Model (DO)
Turbulence	k- ϵ (RNG)
Particle transport	Discrete Random Walk Model
Boundary layer	Standard wall functions
Chemical reaction	Eddy-Dissipation Model or Arrhenius reaction rate

Table 1. Applied sub models (Fluent Inc., 2001).

	Modified case (all values calculated)	Base case
Waste flow (t/h)	7,000	6,250
Lower calorific value (MJ/kg)	8.374	8.135
Primary air flow (Nm ³ /h)	18,891	16,491
Secondary air flow (Nm ³ /h)	5,032	4,581
Secondary air velocity (m/s)	> 50	See Table 3
Cooling air flow (Nm ³ /h)	5,700	7,375
Flue gas flow (Nm ³ /h)	35,609	Calculated: 32,678
Oxygen volume fraction, wet (%)	8.00	Calculated: 9.32

Table 2. Selection of data for modified and pre-modified operation. In pre-modified operation (the base case) flue gas flow and oxygen content is calculated from mass and energy conservation based on the data obtained from the control system history log. All data in the modified case are design values.

SA nozzles	Number of nozzles	SA flow (Nm ³ /h)	SA velocity (m/s)
SA1	6	1,374	22
SA2	6	1,374	22
SA3L	2	458	22
SA4L	2	458	22
SA3R	2	458	22
SA4R	2	458	22
Total	-	4,580	-

Table 3. Distribution of secondary air to secondary air nozzles. Base case.

Parameter	Value	Source
Mean inlet gas temperature, T_{in} (°C)	1028	CFD model
Mean outlet temperature, T_{out} (°C)	850	CFD model
Control Volume, V (m ³)	126.57	CFD model
Flue gas mass flow, \dot{m}_{gas} (kg/s)	12.3	CFD model
Flue gas density, ρ_{gas} (kg/m ³)	0.691	CFD model
Flue gas temperature T_{gas} (°C) in the gas density calculation	219.45	CFD model
Flue gas mean temperature, T_{volume} (°C)	939	eq. (4)
Flue gas density, ρ_{volume} at T_{volume} (kg/m ³)	0.281	eq. (5)

Flue gas mean volume flow, \dot{V}_{volume} (m ³ /s)	43.80	eq. (6)
Retention time, $t_{retention}$ (s)	2.9	eq. (7)

Table 4. Data used for calculation of retention time. Modified case.

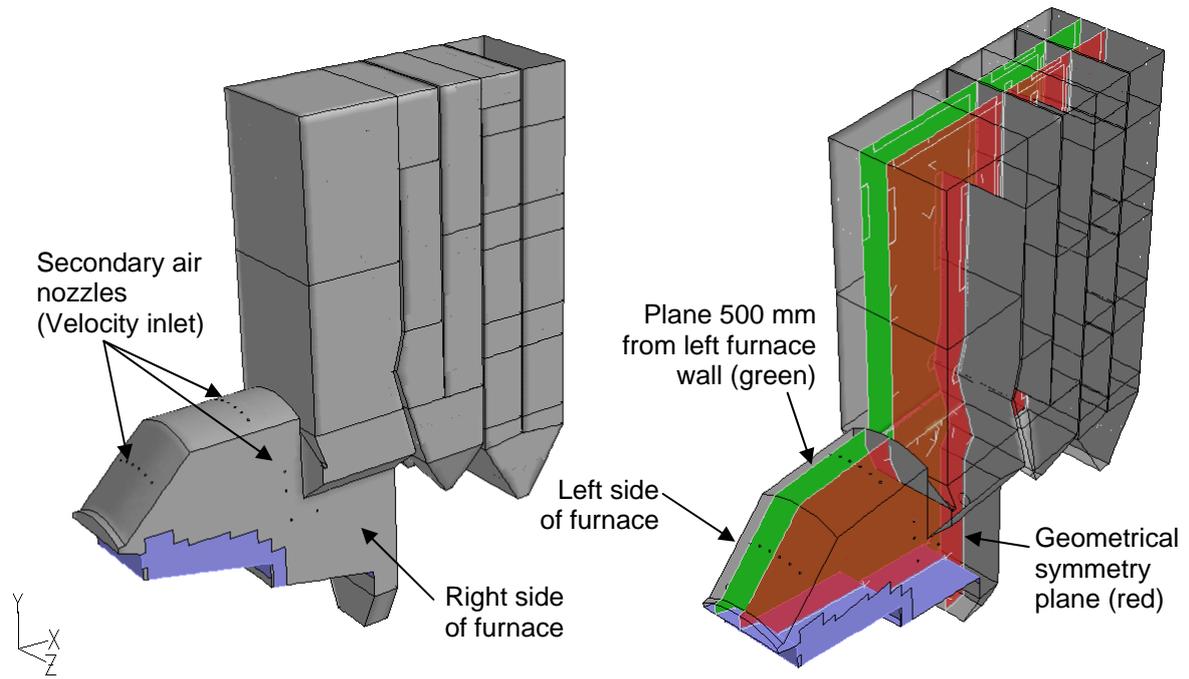


Figure 1. The base case geometry and data output planes (right view). Blue surfaces are velocity inlets.

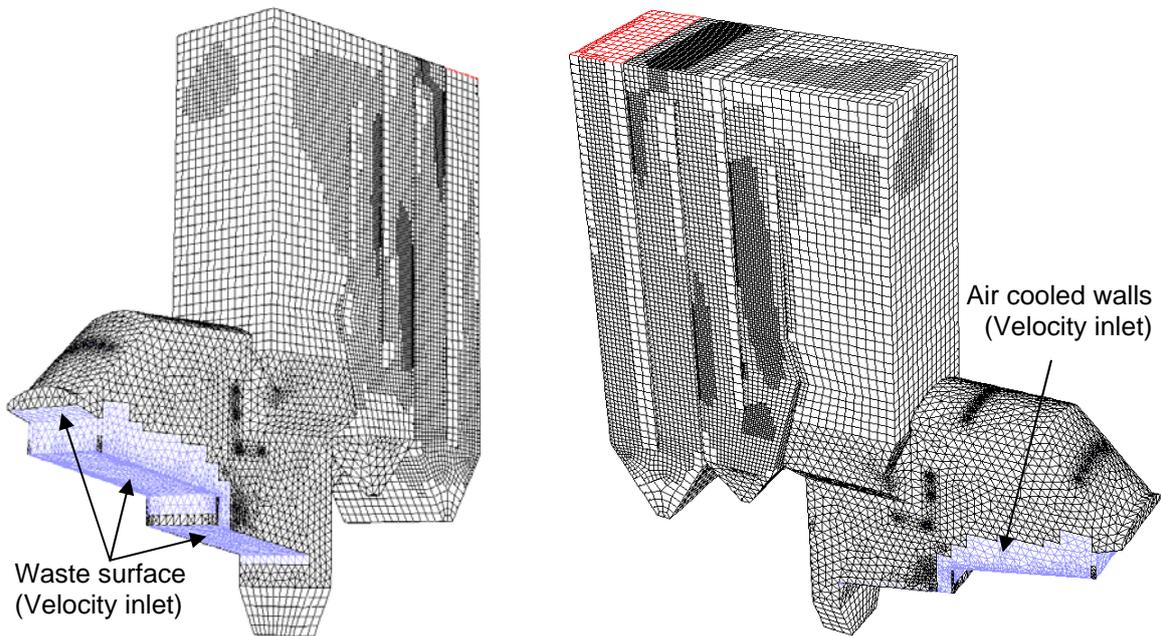


Figure 2. Computational grid for the base case geometry. Blue surfaces are velocity inlets.

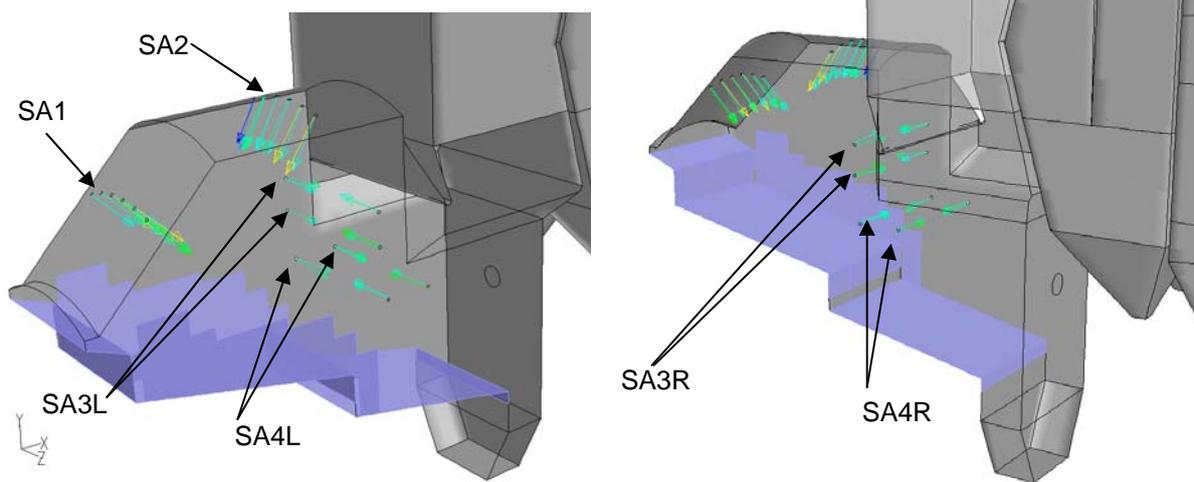


Figure 3. Secondary air nozzles in the base case geometry. Vectors indicate direction of secondary air flow and location of nozzles.

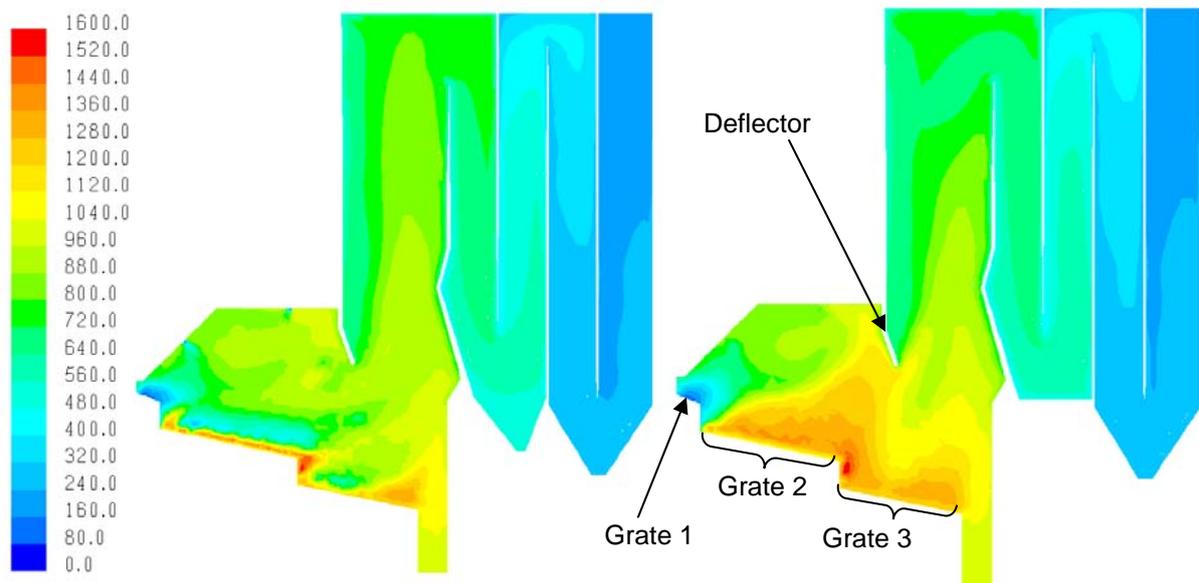


Figure 4. Isothermals (°C) from the base case. Temperatures 0.5 m from the left furnace wall (left view) and in the symmetry plane (right view).

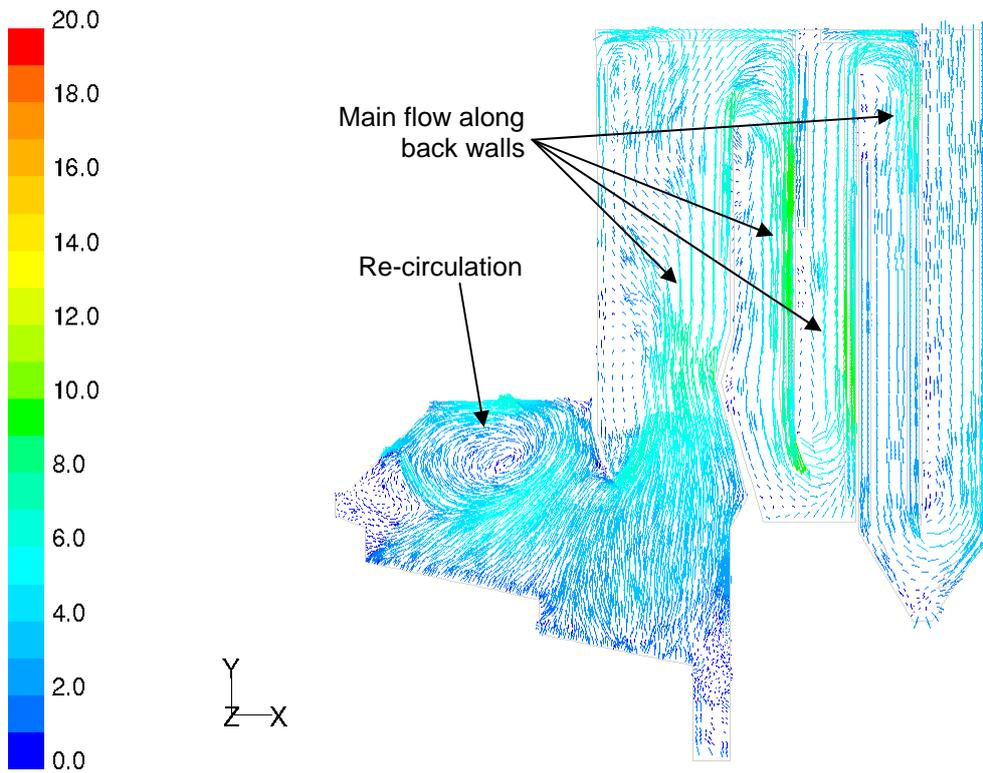


Figure 5. Velocity vectors (m/s) in symmetry plane of the base case.

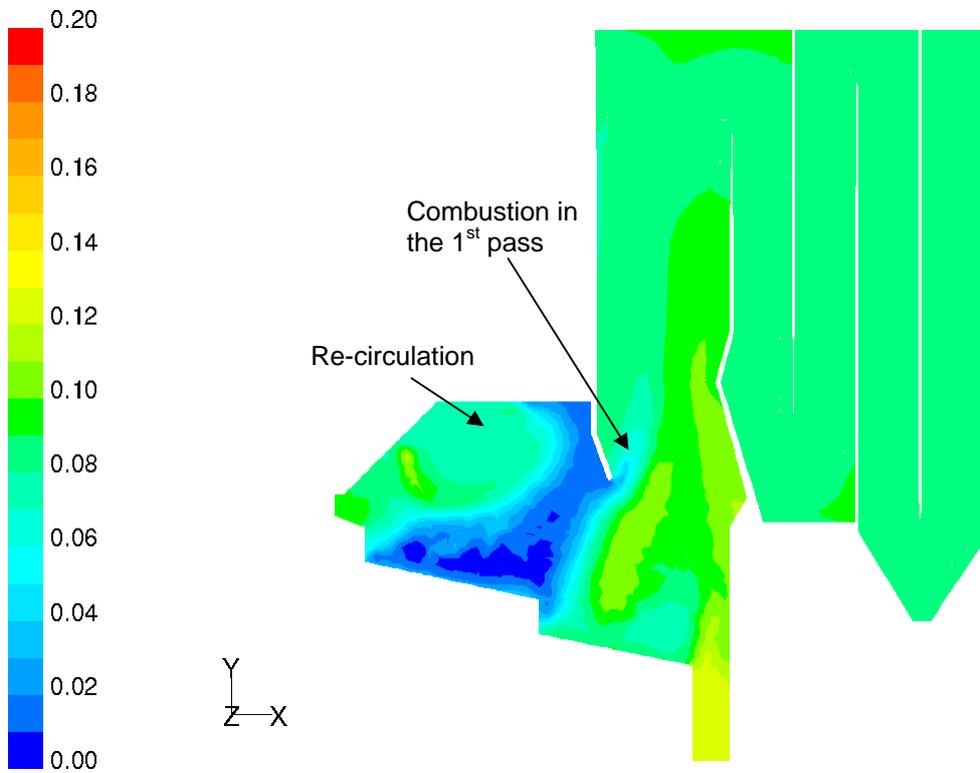


Figure 6. Mole fractions of O₂ in symmetry plane of the base case.

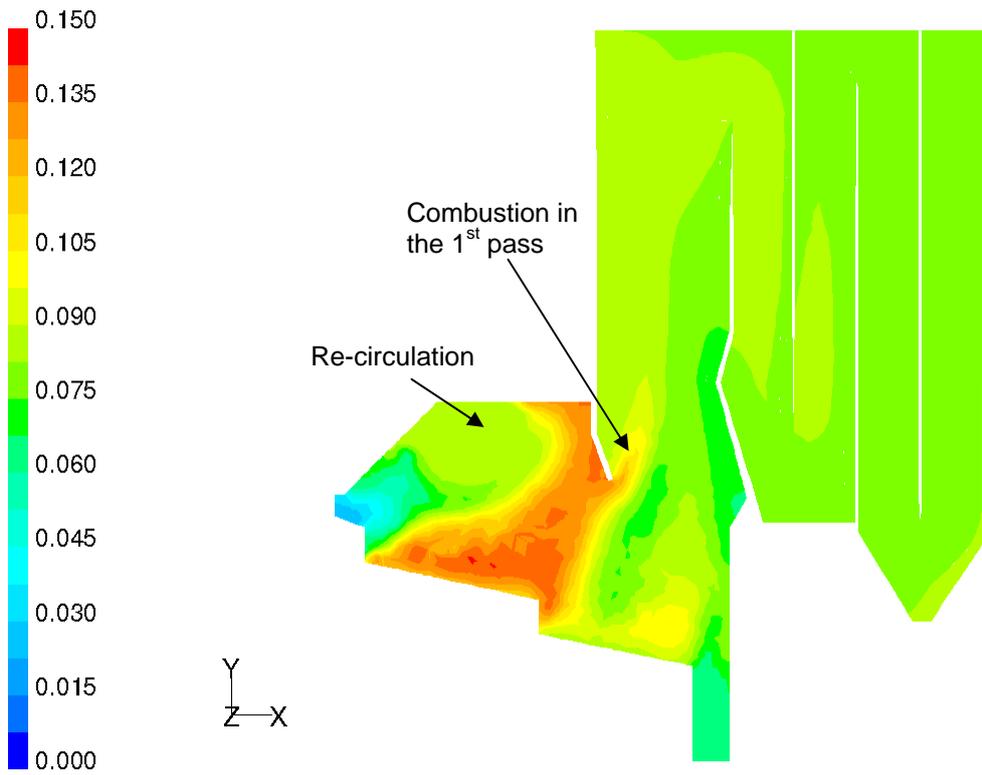


Figure 7. Mole fractions of CO₂ in symmetry plane of the base case.

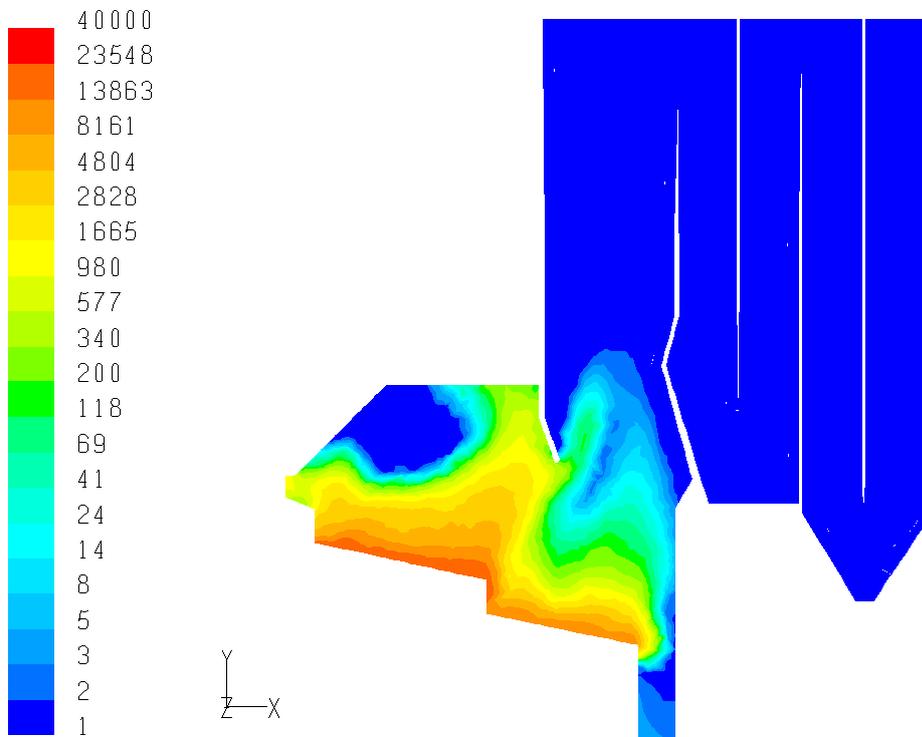


Figure 8. Concentration (ppm) of CO in symmetry plane of the base case. Note: Logarithmic scale.

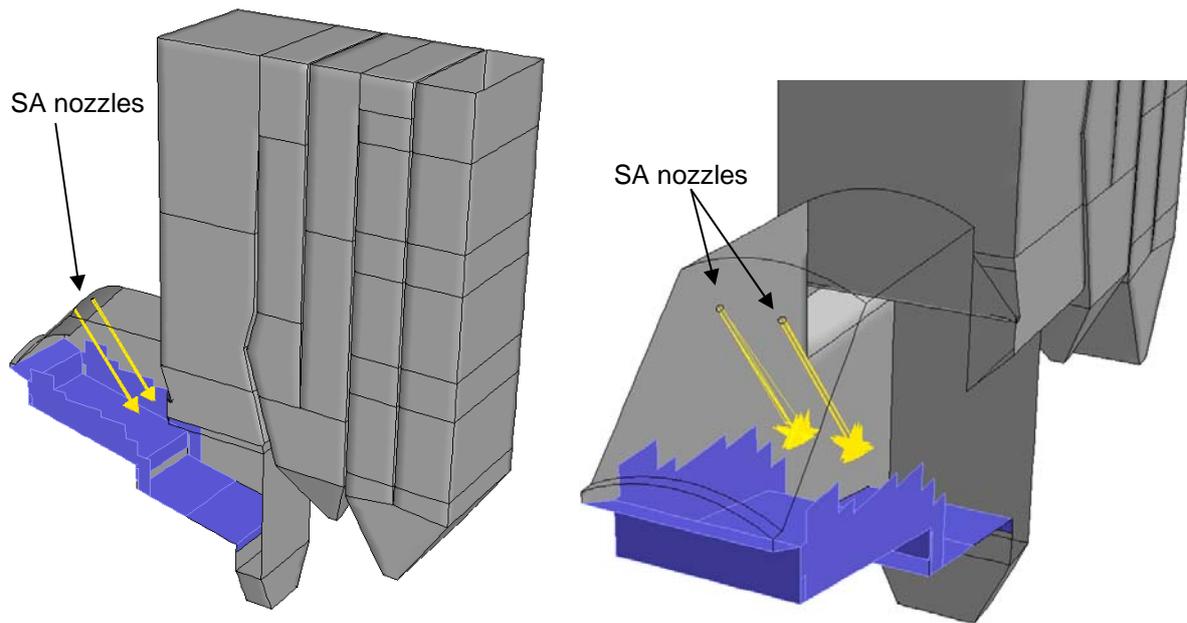


Figure 9. Secondary air nozzles in the modified case geometry. Vectors indicate direction of secondary air flow and location of nozzles.

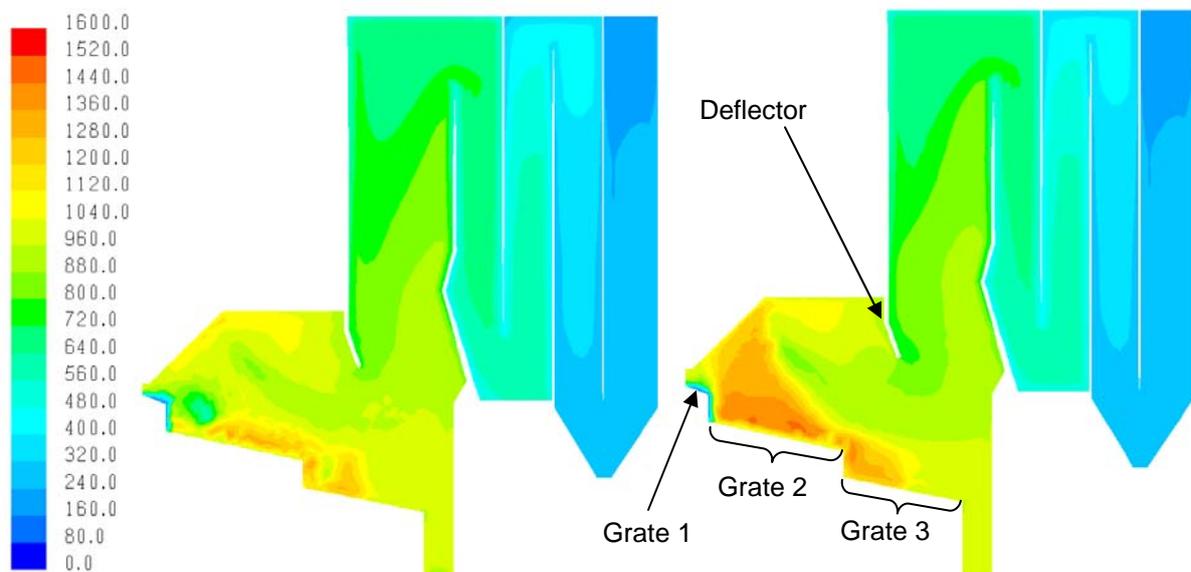


Figure 10. Isothermals ($^{\circ}\text{C}$) from the modified case. Temperatures 0.5 m from the left furnace wall (left view) and in the symmetry plane (right view).

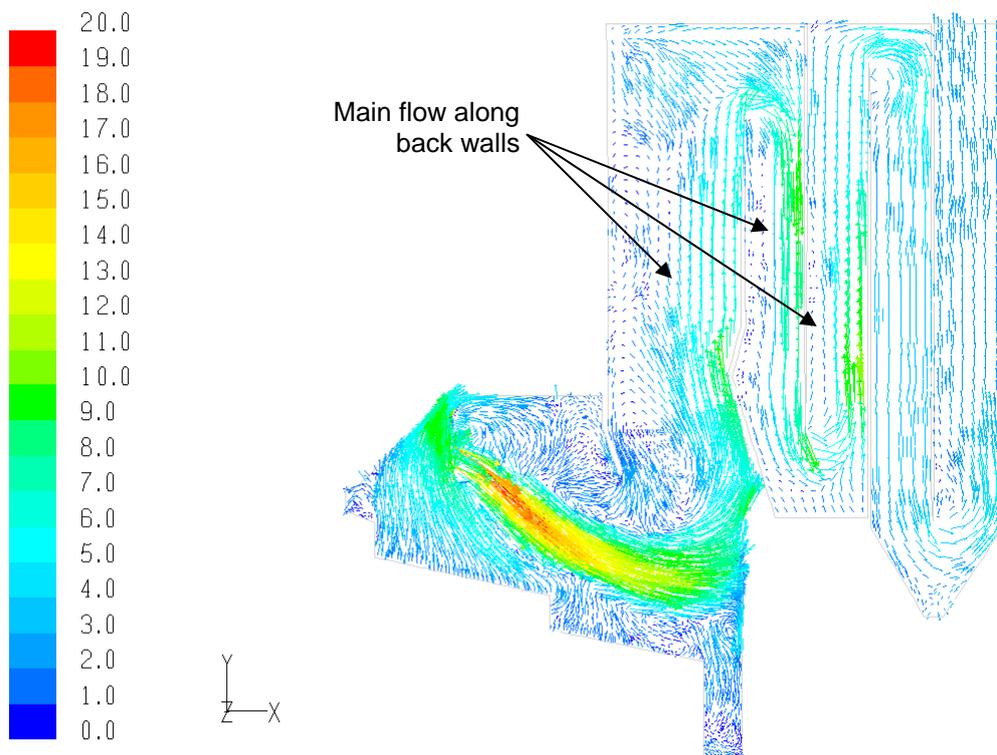


Figure 11. Velocity vectors (m/s) in symmetry plane of the modified case.

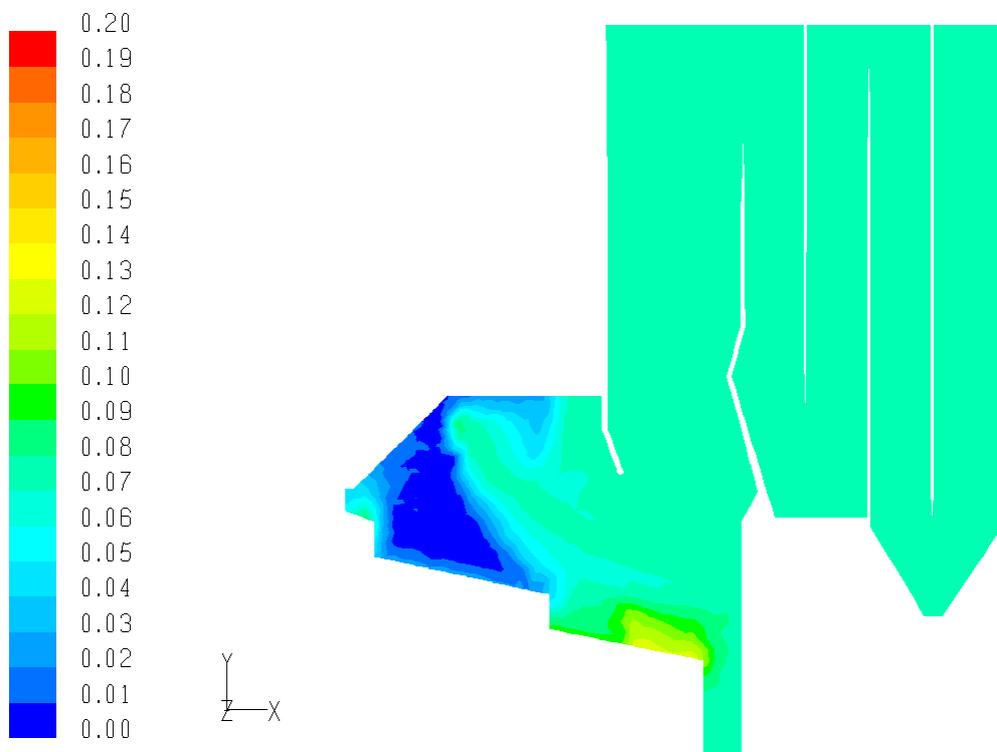


Figure 12. Mole fractions of O₂ in symmetry plane of the modified case.

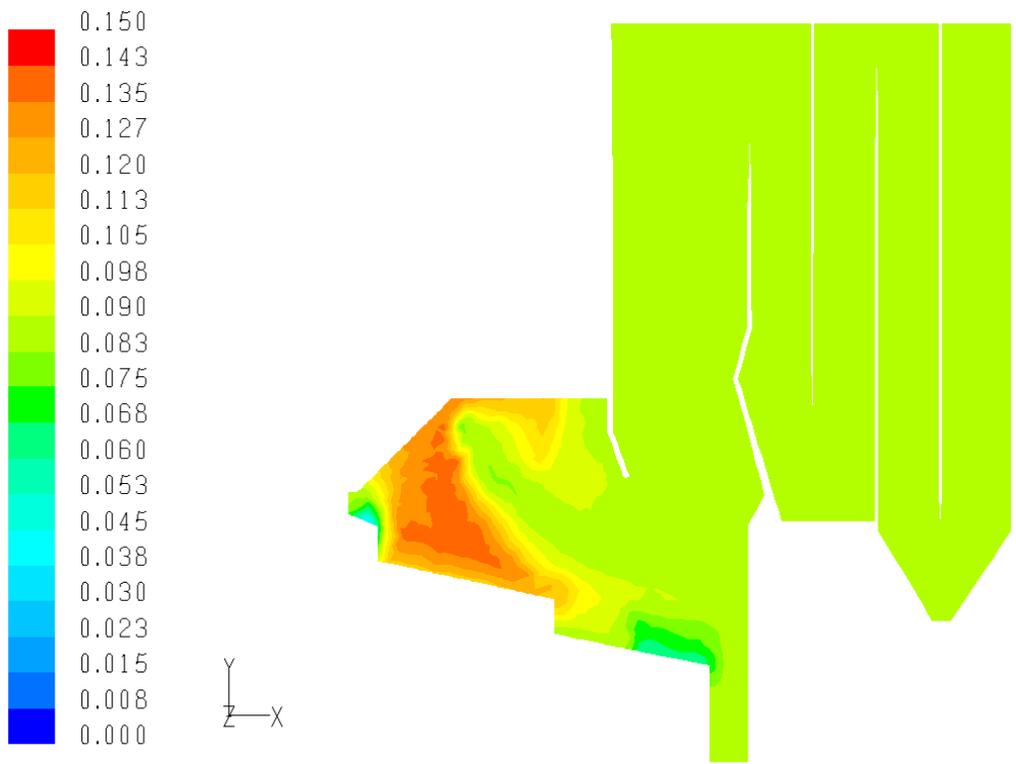


Figure 13. Mole fractions of CO₂ in symmetry plane of the modified case.

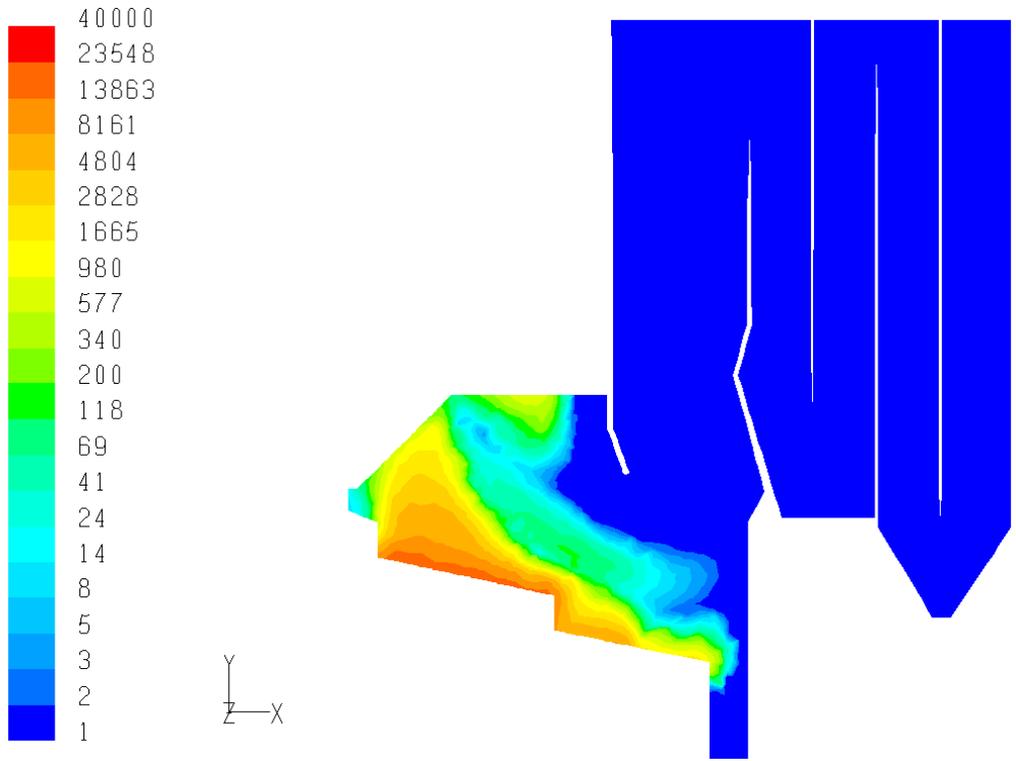


Figure 14. Concentration (ppm) of CO in symmetry plane of the modified case. Note: Logarithmic scale.

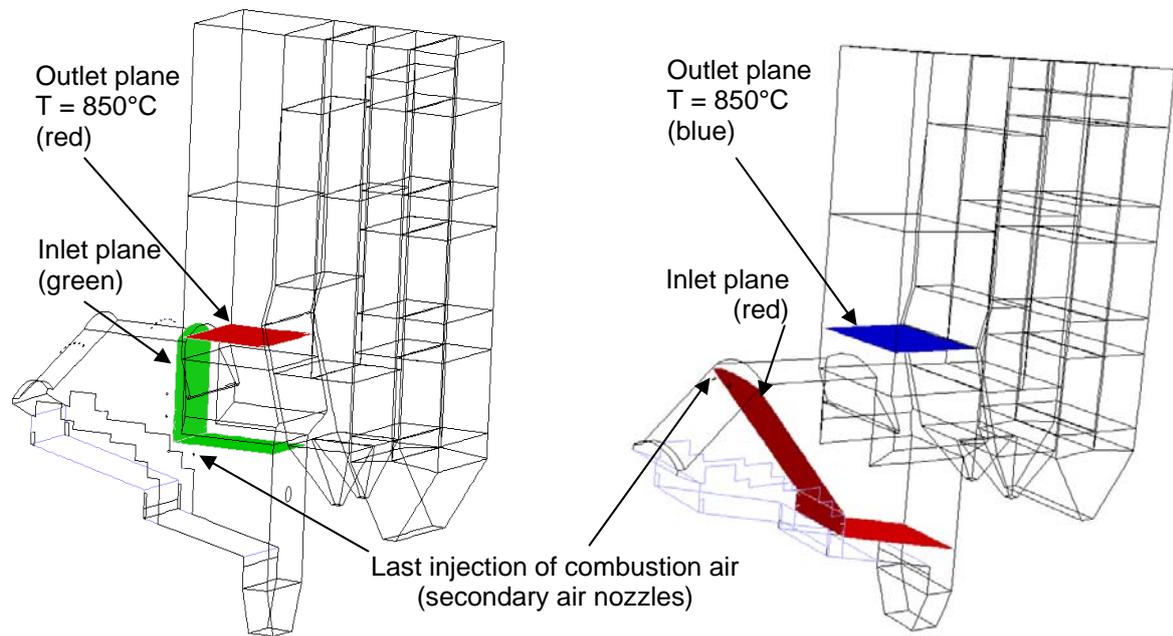


Figure 15. Inlet and outlet planes of control volumes for calculation of retention time. The base case (left view) and the modified case (right view).