IV International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2011
M. Papadrakakis, E. Oñate and B. Schrefler (Eds)

# MASS, HEAT AND MOMENTUM TRANSFER IN NATURAL DRAFT WET COOLING TOWER WITH FLUE GAS DISCHARGE

**ADAM F. KLIMANEK* AND RYSZARD A. BIALECKI***

*Institute of Thermal Technology
Silesian University of Technology
ul. Konarskiego 22, 44-100 Gliwice, Poland
e-mail: ryszard.bialecki@polsl.pl, www.itc.polsl.pl

**Key words:** CFD, Coupled Heat, Mass and Momentum Transfer, Cooling Towers

**Abstract.** The paper presents CFD simulation results of a natural draught wet-cooling tower (NDWCT) with flue gas discharge. The problem considered is mixing of the flue gases with the rising plume and possible corrosion of the tower shell due to acid condensate. A previously developed CFD model of a NDWCT has been used in the analysis. No wind conditions have been assumed and the results have shown that under this condition the corrosion is unlikely to occur.

## 1 INTRODUCTION

Cooling towers are devices used to cool industrial water. In conventional coal-fired power plants, they cool the water coming from the condenser of the steam turbine. The aim of the tower is to keep the cooled water temperature low, since this temperature affects the efficiency of the power plant and thus the fuel consumption and pollutant emissions. The most frequently encountered natural draught cooling towers are massive structures whose height reaches up to 200 m [1, 2]. The water to be cooled is sprayed on the top of a heat and mass exchanger called a fill (packing) flowing downwards in a form of a thin film. Evaporation and heat exchange between the film and the air are responsible for the cooling effect of the water. Free-fall rain zone below the fill, allows airflow into the tower. Finally the droplets traveling downward the rain zone are collected in the water basin from where the water is pumped back to the condenser. Increased humidity and (and usually higher temperature) reduce the density of the air in the tower and form the draught of the tower. From the viewpoint of mass, heat and momentum transfer the three zones: spray, fill and rain are the most important regions of the cooling tower. In Figure 1 the three transfer zones are presented. Two groups of models are used to simulate the operation of natural draught towers:

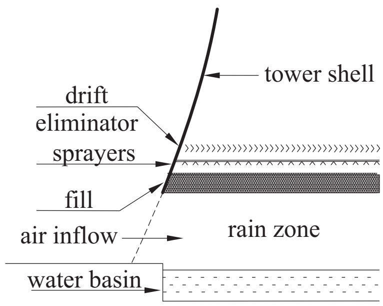


**Figure 1**: The heat, mass and momentum transfer zones in the cooling tower

- Zero or one-dimensional models (global models) used in both design calculations and performance tests. These models work on average flow and thermal parameters of cooling towers.

- Two or three-dimensional models making use of the Computational Fluid Dynamics (CFD) codes. These models can be used in both design calculations and performance tests, however since they are more computationally demanding, they are used when the local flow and thermal parameters of cooling towers need to be determined.

Combustion exhaust gases cleaned in wet type desulphurization plants are too cold (50 - 80 °C) to be introduced without preheating into the chimney of a power plant. These gases can either be reheated and directed to the chimney or introduced into the cooling tower and dispersed in the atmosphere with the plume. The latter solution is extremely interesting due to the lack of a reheating system, which reduces the investment cost of a newly built desulphurization plant. When this approach is used in newly built power plants, the stack does not need to be erected at all. Figure 2 shows a simplified diagram of a power plant equipped with the desulphurization plant transferring the cleaned gases to the cooling tower.

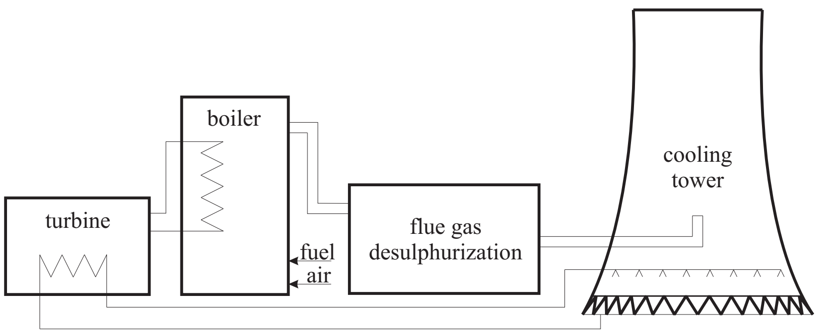


**Figure 2**: Schematic diagram of a power plant with flue gas discharge

The first plant using the flue gas discharge into the cooling tower was built in Völkingen, Germany in 1982 as a model (prototype) version of this solution [3]. Introduction of more strict regulations regarding $SO_2$ emissions required installation of desulphurization plants in both existing and newly built power plants in order to meet this regulations. In the last 30 years a couple dozen of retrofitted (e.g. RWE Niederaussem, Germany; Vattenfal Jänschwalde and Boxberg, Germany; PKE Jaworzno III, Poland) and newly built plants (e.g. RWE Niederaussem, Germany [1, 2]; PGE Belchatów, Poland; PKE Lagisza, Poland) used the concept of flue gas discharge into the cooling tower [4].

Despite the known aforementioned advantages, the problem of corrosion of the inner part of the tower shell due to acid condensate from the flue gas is of concern. This can be solved by coating the internal part of the shell by acid resistant material e.g. epoxy resin, however it generates additional cost. Also questions arose how the introduction of flue gases affect the cooling efficiency of the tower and how well the flue gases are mixed with the plume? There is also a risk that under wind conditions the flue gases would not mix properly with the rising plume. They could then flow along the tower shell and the remaining sulphur oxides would be the reason of corrosion of the tower shell if not protected. Another question is how the flue gases are dispersed in the atmosphere? This question is specifically important under strong wind conditions. Under normal and mild wind conditions the towers are known to disperse the gases even better than stacks [4]. The first applications of flue gas discharge into the cooling tower used specially designed mixers to increase diffusion of the flue gas in the plume [4]. In new cooling towers, however, the inflow is through a gas duct placed centrally a few meters above the fill.

To address these questions a previously developed and validated CFD model of a natural draught wet-cooling tower [5, 6, 7] has been adapted to take into account the flue gas discharge into the tower. In this paper, as a preliminary calculations, only the mixing effect of the introduced gases with the rising plume is examined.

## 2 CFD MODEL OF THE COOLING TOWER

The object under consideration is a 120 m high natural draught wet-cooling tower built in PKE Jaworzno III power plant in Poland. The flue gases are introduced in the center of the tower 25.86 m above the ground level by 7 m diameter duct as shown schematically in Figure 3 (left). The duct is introduced to the tower at the level of 18 m through an opening in the tower shell. The geometry of the model encompasses both the tower and the surrounding atmosphere comprising a 300 m high cylinder of 200 m diameter, as shown in Figure 3 (right). The model of the tower has been developed using a commercial CFD code ANSYS Fluent. The coupled heat, mass and momentum transfer of the multiphase flow in the cooling tower is solved by the use of built-in functions of the CFD code and additional models developed by the authors of this paper. The developed models are incorporated via the User Defined Functions mechanism. The need of special treatment of the heat, mass and momentum exchange in the spray, fill and rain zones is induced by the large scale difference of the surrounding atmosphere ($O(10^3)$ m) and

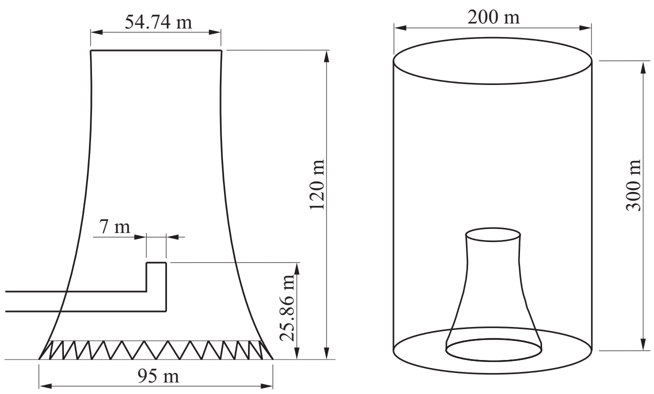


**Figure 3**: Main dimensions of the cooling tower (left) and the computational domain (right)

the internal equipment of the tower ($O(10^{-2})$ m). Since the modeling approaches were already published in [5, 6, 7] they will not be presented here in detail but will be briefly discussed.

The heat and mass transfer in the spray and fill zones are expressed in terms of two point boundary value problem [7, 8]. For boundary conditions that are determined from the CFD code, results of the equations in the fill are the distributions of mass and heat sources. These sources are plugged into appropriate transport equations of the CFD code. The momentum transfer in the spray and fill zone is accounted for by treating them as a porous medium. Substantial acceleration of the computations is achieved by employing an original technique based on Proper Orthogonal Decomposition Radial Basis Function (POD-RBF) network [7, 9]. The final result of this technique is a vector-matrix product defining the distribution of the heat and mass sources in the fill.

The heat, mass and momentum transfer in the rain zone is simulated using the Euler-Euler multiphase model available in the Fluent code. The standard functionality of this model has been extended to account for mass transfer.

The geometry of the model has been created using the Gambit preprocessor and the generated mesh consisted of 5.12 million structured and unstructured grid cells. The mesh has been verified in terms of quality.

## 2.1 Governing equations

The equations solved in the CFD code are the mass, momentum and energy transport equations. Additionally species transport equations are solved for $O_2$, $CO_2$, $SO_2$ and $H_2O$ (vapor). The concentration of $N_2$ is inferred from the concentrations of the remaining species. The multiphase flow in the rain zone of the cooling tower is solved using the Euler-Euler approach. This requires solution of the mass, momentum and energy transport equations for the liquid phase as well. The Standard $k - \epsilon$ turbulence model is used to close the system of equations. The option Dispersed is used to account for the turbulence of the secondary phase. The governing equations are summarized in Table 1 The governing

**Table 1**: Transport equations solved in Fluent

| Equation | # of equations | Applied to |
|---|---|---|
| Continuity | 2 | both phases – air and gas, liquid water |
| Species transport | 4 | primary phase – $O_2$, $CO_2$, $SO_2$ and $H_2O$ |
| Momentum | 6 | both phases – air and gas, liquid water |
| Energy | 2 | both phases – air and gas, liquid water |
| Turbulence | 2 | primary phase – air and gas |

equations are solved in a steady state with second order discretization schemes in space.

### 2.2 Boundary conditions

The pressure inlet and pressure outlet boundary conditions are assigned to the side boundaries and the top of the cylinder like domain, respectively. The pressure and temperature are assumed constant at that boundaries. This also infers that no wind conditions were assumed at this stage. The velocity inlet boundary condition is used for the flue gas introduction. The summary of the input data assigned at the boundaries is presented in Table 2.

**Table 2**: Air, water and flue gas parameters

| | | |
|---|---|---|
| **Atmospheric air parameters** | | |
| Air temperature | $T_a$, °C | 7.11 |
| Relative humidity | $\varphi$, % | 53.45 |
| Pressure | $p$, hPa | 996.1 |
| **Water parameters** | | |
| Hot water temperature | $T_w$, °C | 29.22 |
| Water mass flow rate | $m_w$, kg/s | 14876 |
| **Flue gas parameters** | | |
| Gas temperature | $T_g$, °C | 70 |
| Gas mass flow rate | $m_g$, kg/s | 649.2 |
| Composition: | | |
| $O_2$ | $z_{O_2}$, mol/mol | 0.06 |
| $CO_2$ | $z_{CO_2}$, mol/mol | 0.135 |
| $H_2O$ | $z_{H_2O}$, mol/mol | 0.08 |
| $SO_2$ | $z_{SO_2}$, mol/mol | $10^{-4}$ |
| $N_2$ | $z_{N_2}$, mol/mol | 0.7249 |

## 3 RESULTS AND DISCUSSION

In order to determine the effect of flue gas mixing with the rising plume the cooling process has been simulated using the methods described above. The obtained velocity and temperature fields are presented in Figure 4. As can be seen the velocity of the flue gases entering the tower dissipates quickly and then increases again within the plume. The initial raise of the plume velocity above the tower outlet, can be explained by the low pressure that is formed inside the tower. The lowest pressure is just above the fill. Due to buoyancy forces the plume rises towards higher pressure which is reducing its velocity. The rising plume accelerates to some altitude and then starts to decelerate until its inertia and buoyancy potential (difference in density of plume and air) is lost by mixing with the ambient air. This final reduction of velocity is not shown in the pictures, since the height of the domain is not large enough. The temperature of the flue gases gradually decreases, however, as can be seen from Figure 5 (right), the core of warm flue gases is encircled by air stream until approximately 210 m, where the profiles of inner and outer part of the jet become smooth. Similar behavior can be observed for concentration of the flue gas constituents. It can be seen from Figure 5 (left) that under no wind conditions the concentration of $SO_2$ is zero near the tower shell at any altitude. At the tower outlet the distance of the closest non-zero $SO_2$ concentration from the shell is ca. 12 m.

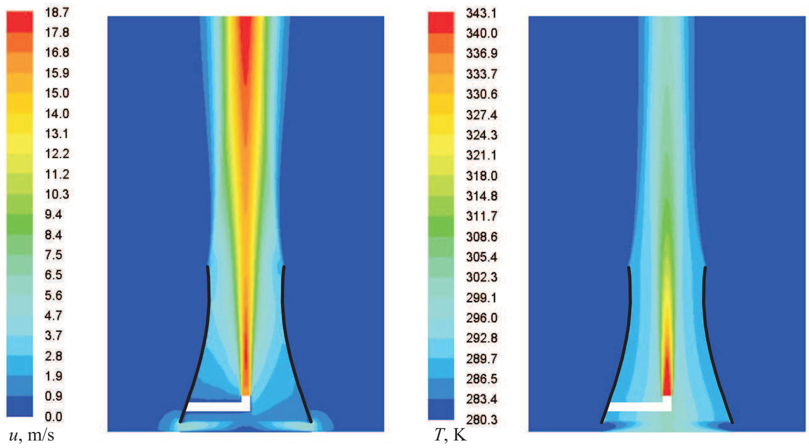


**Figure 4**: Contours of velocity (left) and temperature (right)

## 4 CONCLUSIONS

The preliminary results of simulation of the natural draught wet-cooling tower with flue gas discharge showed that under no wind conditions the mixing of the flue gases inside the tower is low enough to be sure that the gases do not touch the tower shell. This infers that no corrosion due to acid condensation from the remaining sulphur oxides will occur. It should be however stressed, that this situation may be change under wind conditions which

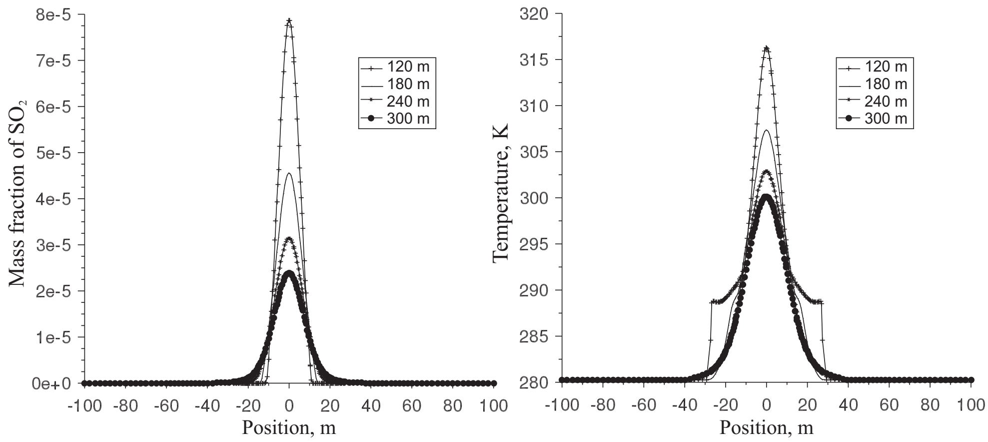


Figure 5: Profiles of $SO_2$ mass fraction (left) and temperature (right) at various elevations above the tower

will be the subject of our future research. The factors that may also have influence on the obtained results are the turbulence model used, and the assumptions made regarding the boundary conditions, specifically the constant temperature and pressure along the vertical boundaries.

## ACKNOWLEDGEMENT

The research has been supported by a grant of the Polish Ministry of Research and Higher Education. This financial assistance is gratefully acknowledged herewith.

## REFERENCES

[1] Busch, D., Harte, R., Krätzig, W.B. and Montag, U. New natural draft cooling tower of 200 m of height. *Engineering Structures* (2002) **24**:1509-1521.

[2] Roth, M., Gerber, R. and Niepel, A. Commissioning and operation of the world-wide largest Natural Draught Wet Cooling Tower of a lignite-fired power plant with flue gas discharge. *Natural draught cooling towers*. Mungan & Wittek (eds). Taylor & Francis Group, London, (2004).

[3] Ernst, G., Leidinger, J.G., Natusch, K., Petzel, H.K. and Scholl G. *Kühlturm und Rauchgasentschwefelungsanlage des Modellkraftwerkes Völkingen.* Fortschritt-Berichte VDI, Reihe 15, No. 45, (1986)

[4] Damjakob, H., Wöhler H.J. and Duddeck, G. Flue gas discharge via natural draught cooling towers - A survey. *Natural draught cooling towers.* Wittek and Krätzig (eds). Balkema, Rotterdam, (1994).

[5] Klimanek, A., Bialecki, R.A. and Ostrowski, Z. CFD two scale model of a wet natural draft cooling tower. *Numerical Heat Transfer, Part A* (2010) **57**:119–137.

[6] Klimanek, A. and Bialecki, R.A. A 3D CFD model of a natural draft wet-cooling tower. *Archives of Thermodynamics* (2009) **30**:119–132.

[7] Klimanek A. *Numerical Modelling of Heat, Mass and Momentum Transfer in Natural Draft Wet-cooling Tower*. Faculty of Energy and Environmental Engineering, Silesian University of Technology, PhD Thesis, (2010).

[8] Klimanek, A. and Bialecki, R.A. Solution of heat and mass transfer in counterflow wet-cooling tower fills. *Int. Commun. Heat Mass* (2009) **36**:547-553.

[9] Klimanek, A., Ostrowski, Z. and Bialecki, R.A. Reduced Order Model of Heat and Mass Transfer in Wet-cooling Tower Fills, *Recent Developments in Artificial Intelligence Methods, AI-METH Series, T. Burczyński, W. Cholewa, W. Moczulski (eds)*, Gliwice, (2009), 149–160.