

Research paper

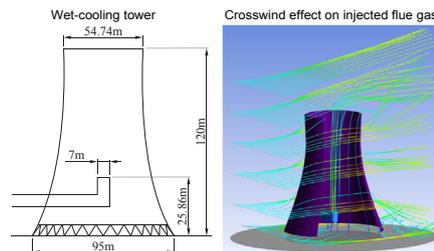
3D CFD modeling of natural draft wet-cooling tower with flue gas injection

Adam Klimanek ^{a,*}, Michał Cedzich ^b, Ryszard Biatecki ^a^a Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, 44-100 Gliwice, Poland^b SBB ENERGY S.A., Łowicka 1, 45-324 Opole, Poland

HIGHLIGHTS

- We developed a CFD model of a natural draft cooling tower with flue gas injection.
- The model was validated against experimental data.
- Influence of flue gas injection on tower performance was examined.
- The effect of crosswind on plume and tower performance was studied.
- Increased risk of tower shell corrosion under crosswind was observed.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:

Received 10 February 2015

Accepted 31 August 2015

Available online 9 September 2015

Keywords:

Cooling towers

Natural draft

Wet-cooling tower

NDWCT

Flue gas injection

Heat and mass transfer

Numerical modeling

ABSTRACT

The paper presents a study on numerical modeling of a natural draft wet-cooling tower with flue gas injection. A previously developed 3D CFD model of a natural draft wet-cooling tower is adopted to account for the flue gas injection. The model is validated against experimental data and then applied to verify the effect of flue gas injection on the cooling tower performance. The effect of wind speed on the rising plume and injected gas is also studied. It is concluded that the injected flue gas has insignificant influence on the cooled water temperature. The increasing wind speed is a reason of recirculation regions formation near the tower outlet which leads to flue gas flow near the tower shell increasing the risk of corrosion.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

Cooling towers are devices used to cool industrial water. In power plants, water cooling steam turbine condenser is introduced to the cooling towers and is cooled by atmospheric air flow. If the air is in direct contact with the water the cooling tower is called

wet-cooling tower. For dry-cooling towers the water flows through an air–water heat exchanger [1]. Since the cooled water temperature affects the efficiency of the power plants and thus the fuel consumption and pollutant emissions, it is important to keep the temperature low. The most frequently encountered natural draft cooling towers are large structures whose height reaches up to 220 m [2]. In wet-cooling towers the water to be cooled is sprayed on the top of a heat and mass exchanger called a fill (packing) and flows downwards in a form of a film. Free-fall rain zone below the fill, allows airflow into the tower. Finally the droplets traveling

* Corresponding author. Tel.: +48 32 2372974; fax: +48 32 2372872.

E-mail address: adam.klimanek@polsl.pl (A. Klimanek).

down the rain zone are collected in the water basin from where the water is pumped back to the condenser. Humidity and temperature of the air inside the cooling tower increase and density decreases. The natural draft occurs due to the difference in the density between the air inside and the ambient air outside the tower. Evaporation and heat exchange between the water and the air are responsible for the cooling effect.

Due to stringent environmental regulations introduced by many countries, coal combustion exhaust gases require deep desulphurization before they can be introduced to the atmosphere. The most widely used method of flue gas desulphurization (FGD) is limestone wet scrubbing with removal efficiencies reaching 92–98% [3]. During the desulphurization process the flue gas is cooled to low temperatures (50–80 °C) and is saturated with water vapor. If an existing plant is retrofitted with a wet scrubbing FGD the gases need to be reheated in a gas–gas heat exchanger before they can be introduced to the existing stack. To avoid the reheating, which is associated with additional investment cost and energy losses in the flue gas, frequently wet stacks are built. This is an additional cost for the investment and the wet stacks require an expensive corrosion resistant liner. Another possibility is to introduce the wet and low temperature flue gas into the cooling tower and disperse it in the atmosphere with the cooling tower plume. Fig. 1 shows a simplified diagram of a power plant equipped with the FGD system transferring the cleaned flue gas to the cooling tower. This solution is interesting due to avoidance of the capital cost associated with the reheating system or the wet stack, which need to be erected for retrofitted and newly built power plants. Unfortunately due to corrosive properties of the cleaned flue gas condensates, the internal and part of the external shell of the cooling tower is also covered with corrosion resistant coatings e.g. epoxy resin, which is associated with additional high capital costs due to large area of the tower shell surface.

The first plant using the flue gas discharge into the cooling tower was built in Völklingen, Germany in 1982 as a model (prototype) version of this solution [4]. In the last 30 years in a couple dozen of retrofitted, e.g. RWE Niederaussem, Germany; Vattenfall Jämschwalde and Boxberg, Germany; PKE Jaworzno III, Poland and newly built plants, e.g. RWE Niederaussem, Germany [5,6]; Vattenfall Schwarze Pumpe, Germany; PGE Bechatów, Poland; PKE Łagisza, Poland, the concept of flue gas injection to the cooling towers was used.

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 and could then flow along the tower shell increasing the risk of tower shell corrosion. 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 [7]. 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 [7]. In recently built cooling towers, however, the gases are introduced through a gas duct placed centrally above the fill.

In recent years extensive work has been published focusing on thermal performance improvement of natural draft wet-cooling towers. The need for detailed studies of the thermofluid fields and the increasing computer power lead to development of many numerical models based on computational fluid dynamics (CFD) software. Al-Waked and Behnia [8] developed a 3D CFD model of a natural draft wet-cooling tower, which was used to predict cooling tower performance under different operating and crosswind conditions. In Ref. [9] the authors studied the effect of installing wind break walls near the tower inlet and in the rain zone, and concluded that this modification can improve tower performance under crosswind. The same model has been later used [10] to study the interaction of two cooling towers and power plant structures. Williamson et al. [11] performed CFD simulations using a 2D axisymmetric model developed with an emphasis on accurately predicting the flow, heat and mass transfer inside the tower under no-wind conditions. The model has been compared with a 1D Merkel type approach [12]. The authors further extended the 1D model to a zonal method which, together with the 2D axisymmetric CFD model, has been used in an optimization of fill shape and water flow distribution [13]. Klimanek et al. developed a 2D axisymmetric [14,15] and a 3D [16] CFD model. This approach is applied in the current study and will be described in more detail in Section 2.3. Reuter and Kröger [17] introduced a 2D axisymmetric model capable of predicting performance of wet-cooling towers packed with all types of fills including trickle and splash fills, which have anisotropic flow resistances. The model has been successfully validated and applied to predict, among others, the effects of droplet sizes in the rain zone and of rounding the tower inlet. Recently Jin et al. [2] presented a 3D CFD model of a 220 m high wet-cooling tower. The model has been validated and applied to tower performance evaluation under different operating conditions and non-uniform water mass flow rate. The latter has been shown to slightly improve the cooling performance.

Limited number of publications related to the associated effects of flue gas injection in both the dry-cooling and wet-cooling towers can be found in the literature. In the work of Schatzmann et al. [18] wind tunnel study of cooling tower with flue gas discharge under various wind conditions was conducted. It was concluded that the dispersion of flue gas with cooling tower plume is better than dispersion from stack under no wind and mild wind conditions. Under strong wind conditions, due to tower downwash the cooling tower dispersion was worse. Han et al. [19] presented and economic evaluation of 300 MW power plant equipped with conventional cooling tower and with flue gas injection concluding that substantial annual savings can be achieved when the latter is applied. The authors also performed a multi-objective optimization of cooling tower with flue gas injection showing possibility of further running cost decrease. They also reported a slightly lower (~0.3 K) cooled water temperature to be achieved by cooling tower with flue gas discharge. Jahangiri and Golneshan [20] developed a 3D CFD model of a natural draft dry-cooling tower with flue gas injection and studied the effect of injection vertical position, diameter, gas temperature and mass flow rate on thermal effectiveness and tower draft under no wind conditions. For the best of the analyzed cases they obtained 0.7% improvement of thermal effectiveness. This was obtained for very small diameter of the flue gas duct of 2 m and gas inlet velocity of 91.2 m/s. Eldredge et al. [21] developed a 2D axisymmetric CFD model of a natural draft wet-cooling tower with flue gas injection and examined the effect of flue gas flow rate, temperature, rate of liquid entrainment in the

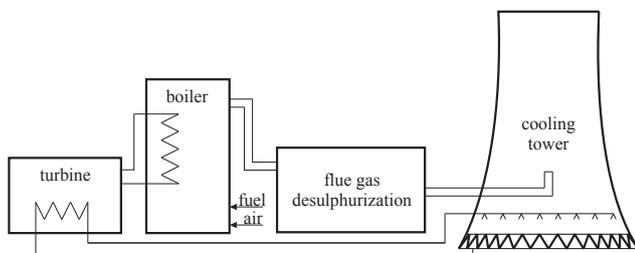


Fig. 1. Schematic diagram of a power plant with flue gas discharge.

flue gas, radial location and direction of the injection. They concluded that the injected gas temperature had the largest effect on the cooled water temperature. The highest considered gas temperature of 65.6 °C increased considerably the tower draft which allowed reducing the cold water temperature when compared to a base case without flue gas injection. Recently Ma et al. [22] developed a 3D CFD model of a natural draft dry cooling tower with flue gas injection coupled with the model of turbine condenser. The study focussed on the thermal performance of the tower at various ambient temperatures and under cross wind. The influence of the flue gas injection and its behavior has not been examined.

In this study a previously developed and validated 3D CFD model of a natural draft wet-cooling tower [14–16] has been adapted to take into account the flue gas injection. The gas is injected vertically through a centrally placed flue gas duct as shown schematically in Fig. 2 (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 Fig. 2 (right). The dimensions of the internal equipment of the cooling tower is presented in Fig. 3 Geometry of the model has been created using the Design Modeler of ANSYS software. In this study a full 3D model of the cooling tower has been used. It should be stressed that under no wind conditions, and if the geometry of the flue gas duct in the tower is neglected, the model could be developed as axis-symmetric. For wind conditions, due to symmetry half of the model could be used.

2. Numerical modeling

2.1. Geometry of the model

The object under consideration is a 120 m high natural draft wet-cooling tower built in PKE Jaworzno III power plant in Poland. Nominal design parameters of the cooling tower under consideration are summarized in Table 1 [23].

The flue gas is introduced in the center of the tower 25.86 m above the ground level through a 7 m diameter duct as shown schematically in Fig. 2 (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 Fig. 2 (right). The dimensions of the internal equipment of the cooling tower is presented in Fig. 3 Geometry of the model has been created using the Design Modeler of ANSYS software. In this study a full 3D model of the cooling tower has been used. It should be stressed that under no wind conditions, and if the geometry of the flue gas duct in the tower is neglected, the model could be developed as axis-symmetric. For wind conditions, due to symmetry half of the model could be used.

2.2. The mesh

The computational mesh has been created using the ANSYS Meshing application. The generated mesh was composed of 5.12 million tetrahedral and hexahedral cells and has been verified in terms of quality by examining mesh quality parameters. The applied mesh sizing (varying in various regions of the cooling tower

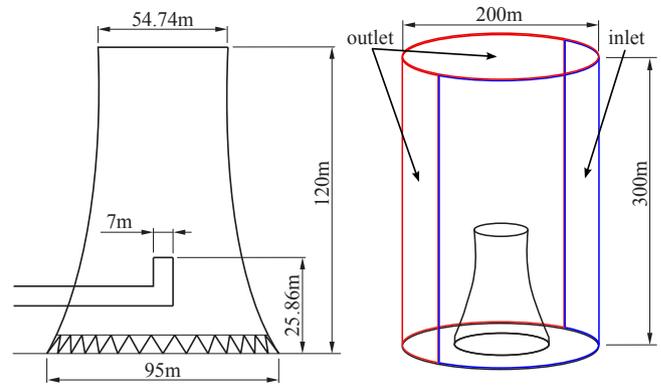


Fig. 2. Main dimensions of the cooling tower (left) and the computational domain (right).

and atmosphere) provides mesh independent results which is verified by mesh adaptation and sensitivity study presented in Section 3.2. The mesh configuration applied in this study is presented in Fig. 4.

2.3. Numerical model

The model of the tower has been developed using a commercial CFD code ANSYS Fluent 14.0. 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 additional 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 the internal equipment of the tower ($O(10^{-2})$ m). Since the modeling approaches used had already been published in our previous works [15,14,16] 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 a two point boundary value problem [15,24]. The fill is divided radially and angularly into sections. In each section the heat and mass transfer problem is solved. Boundary conditions for each section (local air humidity, temperature and mass flow rate under the fill) are determined from current solution of the CFD model. Solution of the

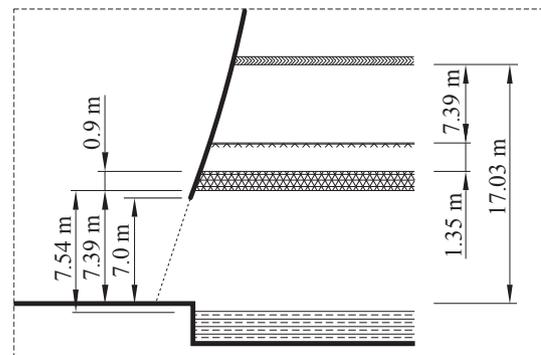


Fig. 3. Main dimensions of the internal equipment of the cooling tower (dimensions in m).

Table 1
Design parameters of the analyzed cooling tower.

Parameter	Quantity	Unit
Water flow rate, V_w	61,000	m^3/h
Heat rejection rate, Q	602	MW
Inlet water temperature, T_{wi}	35.2	°C
Cooled water temperature, T_{wo}	26.7	°C
Ambient air temperature, T_a	15	°C
Relative humidity, φ	70	%
Atmospheric pressure, p	990	hPa

governing equations of the heat and mass transfer model produces distributions of mass and heat sources in each section of the fill and spray zones. These sources are plugged into appropriate transport equations (species transport and energy equations) of the CFD code and the CFD solution is updated. This procedure is repeated until convergence is reached. Substantial acceleration of the computations is achieved by employing a technique based on Proper Orthogonal Decomposition Radial Basis Function (POD-RBF) network [14,15]. The final result of this technique is a vector-matrix product defining the distribution of the heat and mass sources in the fill and spray zones.

2.4. Transport equations

The equations solved in the CFD code are the mass, momentum and energy equations. Additionally species transport equations are solved for O₂, CO₂, SO₂ and H₂O (vapor). The concentration of N₂ 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 Realizable *k*– ϵ 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 solved in a steady state with second order discretization schemes in space. The governing equations are summarized in Table 2.

2.5. Momentum transfer in the fill, spray zone and drift eliminator

The pressure drop in the fill (*fi*), spray zone (*sz*) as well as in the drift eliminator (*de*) was taken into account by introducing sources to the momentum equations using the built in porous media model.

Table 2
Transport equations solved in ANSYS Fluent.

Equation	No. 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 ₂ O
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

Table 3
Flue gas parameters.

Temperature	<i>T_g</i> , °C	70
Mass flow rate	<i>m_g</i> , kg/s	645.8
Composition:		
O ₂	<i>z_{O₂}</i> , mol/mol	0.06
CO ₂	<i>z_{CO₂}</i> , mol/mol	0.135
H ₂ O	<i>z_{H₂O}</i> , mol/mol	0.08
SO ₂	<i>z_{SO₂}</i> , mol/mol	10 ^{–4}
N ₂	<i>z_{N₂}</i> , mol/mol	0.7249

The momentum source *S_i* in zone *i* (pressure drop per unit length) is given in the form

$$S_i = C_i \frac{1}{2} \rho u^2 \tag{1}$$

where ρ and *u* are inflowing air density and velocity, respectively and *C_i* in m^{–1} for the fill zone is given by Ref. [25]

$$C_{fi} = (6.8 + 0.8G_w) / H_{fi} \tag{2}$$

and for the spray zone [26]

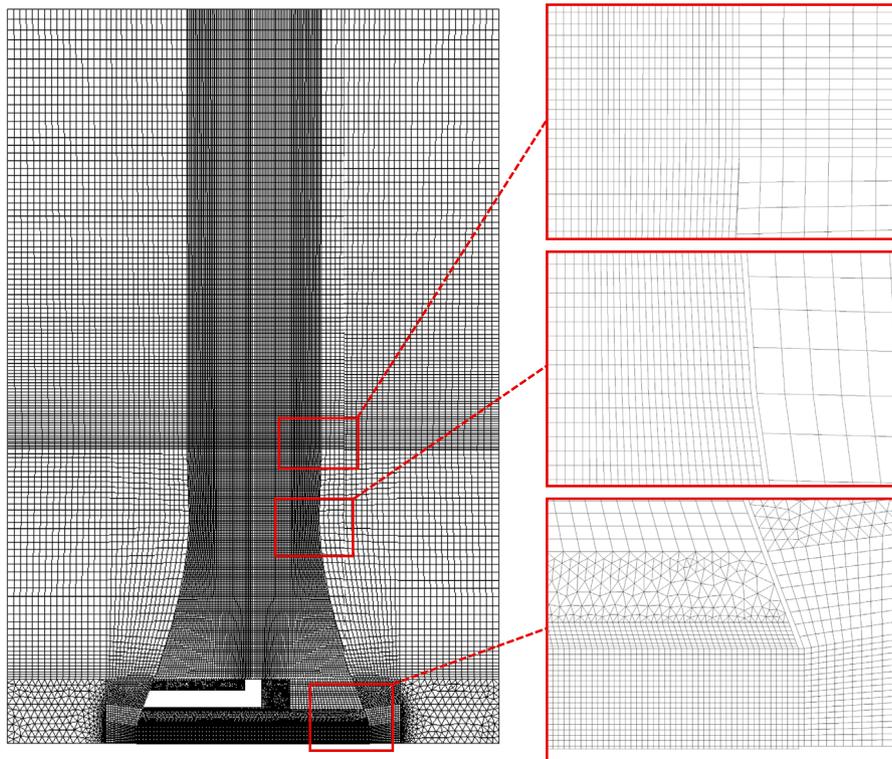


Fig. 4. The applied mesh configuration – cross section in the cooling tower center.

Table 4
Air and water parameters.

Parameter	Symbol	c1	c2	c3
Dry bulb air temperature	$T_a, ^\circ\text{C}$	7.1 ± 0.4	9.2 ± 0.2	9.1 ± 0.4
Wet bulb air temperature	$T_{awb}, ^\circ\text{C}$	3.4 ± 0.6	4.1 ± 0.4	4.1 ± 0.4
Atmospheric pressure	p, hPa	996.1	995.3	995.3
Wind speed (at $z_0 = 2 \text{ m}$)	$u_0, \text{m/s}$	1.60	1.49	1.55
Water mass flow rate	$m_w, \text{kg/s}$	$14,876 \pm 281$	$14,876 \pm 281$	$14,876 \pm 281$
Hot water temperature	$T_{wh}, ^\circ\text{C}$	29.2 ± 0.2	30.0 ± 0.2	29.7 ± 0.2

Table 5
Comparison of measured and predicted cold water temperatures.

Cold water temperature	Symbol	c1	c2	c3
Measured	$T_{wcm}, ^\circ\text{C}$	19.2 ± 0.2	19.9 ± 0.2	19.8 ± 0.2
Computed	$T_{wcc}, ^\circ\text{C}$	19.8	20.0	19.9
Absolute difference	$ \Delta T_{wc} , ^\circ\text{C}$	0.6	0.1	0.1

$$C_{sz} = 0.52(G_a/G_w)^{-1.32} \quad (3)$$

where G_a and G_w are air and water mass fluxes in $\text{kg/m}^2\text{s}$, respectively and $H_{fi} = 0.9 \text{ m}$ is the fill depth. The loss coefficient for the drift eliminator is [25] $C_{de} = 12.5 \text{ m}^{-1}$.

2.6. The Merkel number in the fill and spray zones

A film fill of total area $A = 6400 \text{ m}^2$ is installed in the tower. The modified Poppe approach presented in Refs. [15,24] has been adopted in this study to account for heat and mass transfer in the fill and the spray zones. The Merkel number according to the Merkel theory in the fill is given by [25]

$$Me_{fi} = 1.5(G_a/G_w)^{0.6} \quad (4)$$

The Merkel number in the spray zone is [15].

$$Me_{sz} = 0.2H_{sz}(G_a/G_w)^{0.5} \quad (5)$$

where $H_{sz} = 1.35 \text{ m}$ is the spray zone depth. The same procedure as described in Ref. [15] has been applied to account for the differences of the Merkel number in the Merkel and Poppe approaches.

2.7. The rain zone

The rain zone is treated as multiphase region where the Euler–Euler multiphase model is applied. The heat and momentum transfer between spherical water droplets and the air is modeled internally within the multiphase model, where the correlation of Ranz and Marshal [27] is used to determine heat transfer and correlation of Schiller and Naumann [27] is used to determine the drag coefficient. The mass transfer is determined applying analogy between heat and mass transfer as described in Ref. [15]. A constant droplet diameter in the rain zone of 5.5 mm , typical for film fills [28] has been assumed.

2.8. Boundary conditions

The calculations have been performed taking into account the wind. For validating computations the measured wind velocity has been used to define the velocity profile of air in the atmosphere. The velocity profile in the atmosphere can be described by Ref. [29,30].

$$u = u_0(z/z_0)^m \quad (6)$$

where u is the velocity at elevation z and u_0 is the measured velocity at elevation z_0 . The index m varies during the day and the season. A value of $m = 0.1$ has been used in all the calculations [29]. The wind speed measurements were taken at $z_0 = 2 \text{ m}$ above the ground level. The pressure outlet boundary condition has been used at the top and at the outlet of the domain, as shown in Fig. 2.

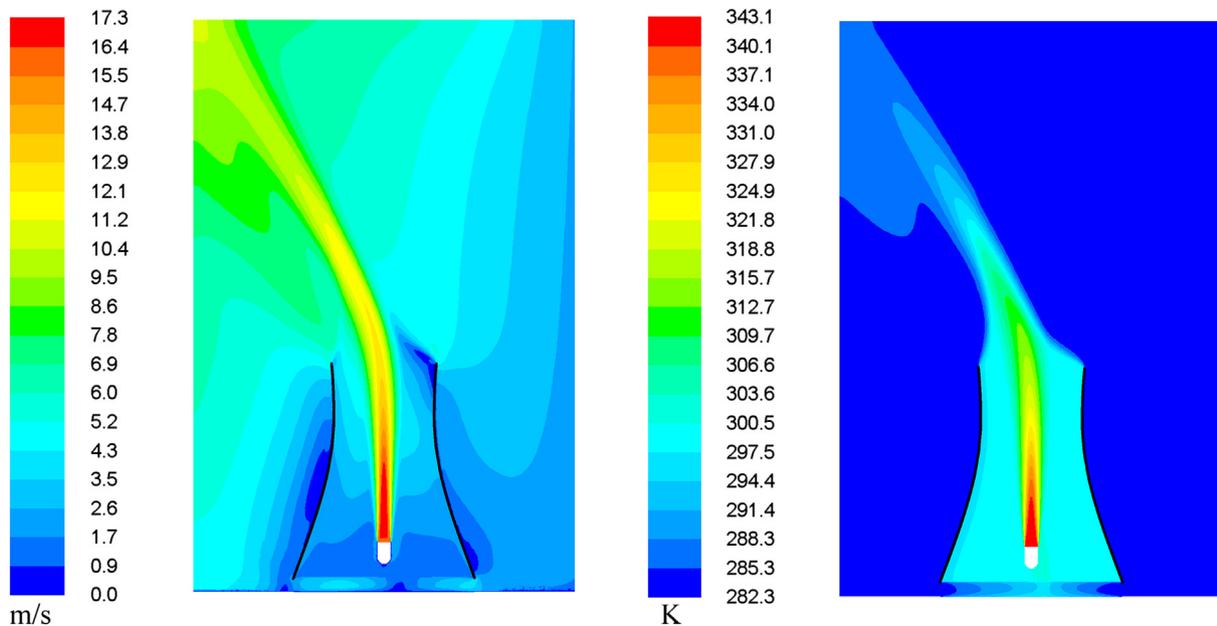


Fig. 5. Contours of velocity magnitude (left) and temperature (right) for case c3 (wind speed $u_0 = 1.55 \text{ m/s}$).

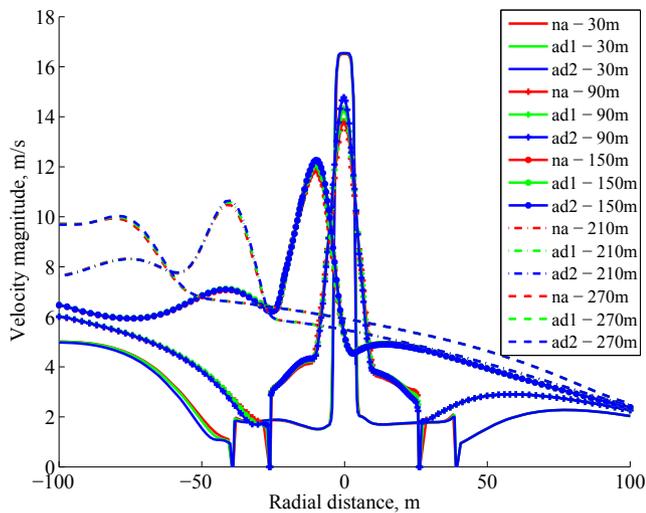


Fig. 6. Velocity magnitude profiles for the original mesh (na), single (ad1) and double (ad2) mesh adaptation at various elevations above the ground level for case c3.

The atmosphere has been assumed to be isothermal. The velocity inlet boundary condition is used for the flue gas injection. It has been assumed that the parameters of the injected flue gas are the same in all analyzed cases. Summary of the input flue gas parameters is presented in Table 3. As can be seen from Fig. 2 the shortest distance from the boundary to the tower is 52.5 m, therefore influence of the boundary conditions on the obtained results can be expected. This effect has not been examined in this study.

3. Results and discussion

3.1. Validation of the model

In order to validate the developed model, cold water temperatures obtained from three experiments (c1, c2 and c3) [23] were compared with cold water temperatures predicted by the model. The measured values obtained from the experiments are presented in Tables 4 and 5. It can be seen that the experimental data vary just slightly, however due to large uncertainty in the measured water mass flow rates, all three cases (c1, c2 and c3) were used for the comparison. The uncertainties given in Tables 4 and 5 are the expanded uncertainties with a coverage factor of 2 providing a level of confidence of 95.4%. The fact that only the cold water temperatures could be compared with the computations limits the credibility of the validation. These are however the only data available. The data presented in Table 4 were used as model inputs. In Table 5 the measured and computed cold water temperatures are presented along with the absolute difference between the measurements and model predictions. It can be seen that the model predicts similar cooled water temperatures to the measured values. In Fig. 5 contours of velocity and temperature inside the cooling tower and in the surroundings for case c3 are presented. It is visible how the rising plume is bent due to the wind and that the air under the tower is heated due to the water flow in the rain zone.

Table 6

Comparison of mean cold water temperatures and total gas mass flow rates for case c3 with the original and adapted meshes.

Mesh adaptation	No	Single	Double
Cold water temperature, °C	19.82	19.89	19.89
Total mass flowrate, kg/s	10,134.6	10,096.1	10,086.5

Table 7

Comparison of predicted cold water temperatures for cases with and without flue gas injection.

	c3	c4	c5	c6
Flue gas injection	Yes	No	Yes	No
Wind speed u_0 , m/s	1.55	1.55	0.0	0.0
Cold water temperature, °C	19.89	19.90	19.41	19.43
Flue gas mass flowrate, kg/s	645.8	0.0	645.8	0.0
Total mass flowrate, kg/s	10096.1	9426.3	10404.7	9721.2

3.2. Mesh sensitivity study

The applied mesh has been validated by conducting a sensitivity study. The influence of mesh density on the simulation results for case c3 has been examined by double mesh adaptation and verification of the results. The mesh has been adapted in regions of the highest gas velocity gradients. The applied mesh adaptation strategy was that all cells with velocity gradient values higher than 10% of maximum gradient have been refined. The original mesh built out of 5.12 million elements has at first been adapted to obtain 5.63 million elements and then adapted further to obtain 6.68 million elements. In Fig. 6 profiles of velocity magnitude for the three analyzed meshes are presented at various elevations above the ground level. As can be seen the profiles coincide virtually at all positions except for those near the tower shell regions and in the center of the rising plume at the lines placed 90 and 150 m above the ground level, where for the adapted meshes the predicted maximum velocities were slightly higher. These discrepancies are however small and do not affect the mean quantities like mean cold water temperature and the total gas mass flow rate through the tower. The predicted cold water temperatures and total gas mass flow rates through the tower obtained for the three meshes are presented in Table 6. Due to small influence of mesh density on the results, all results presented in this study stem from single mesh adaptation.

3.3. Cooling improvement by flue gas injection

The model has then been used to verify the influence of the flue gas injection on cooled water temperature and air flow through the tower. For this purpose case c3 from Table 4 has been used as reference. Additional computations have been done for the same settings as in c3 but without flue gas injection (case c4). Furthermore cases c5 and c6 were considered in which no-wind conditions were assumed. In case c5 the flue gas was introduced to the tower and in case c6 it was not. The predicted cooled water temperature and air flow through the tower are presented in Table 7. As can be seen the cold water temperatures are virtually identical and no positive effect of the flue gas injection is visible. The total mass flowrate presented in Table 7 is composed of the air and flue gas mass flowrates. The air mass flowrate through the cooling tower outlet is 9450.3 kg/s in case c3 and 9758.9 kg/s in case c5. This shows that the increase of air mass flow rate through the tower due to flue gas injection is only 24.0 kg/s in case c3 and 37.7 kg/s in case c5. Nonetheless the injected flue gas is hot (70 °C) this is too little to

Table 8

The effect of wind on cold water and air flow through the tower.

Wind speed, m/s	0	0.5	1.55	3.0	4.5
Total mass flowrate, kg/s	10404.7	10187.3	10096.1	9229.3	9070.8
Cold water temperature:					
Rain zone inlet, °C	21.76	22.00	22.33	23.36	23.65
Rain zone outlet, °C	19.41	19.63	19.89	20.75	20.80

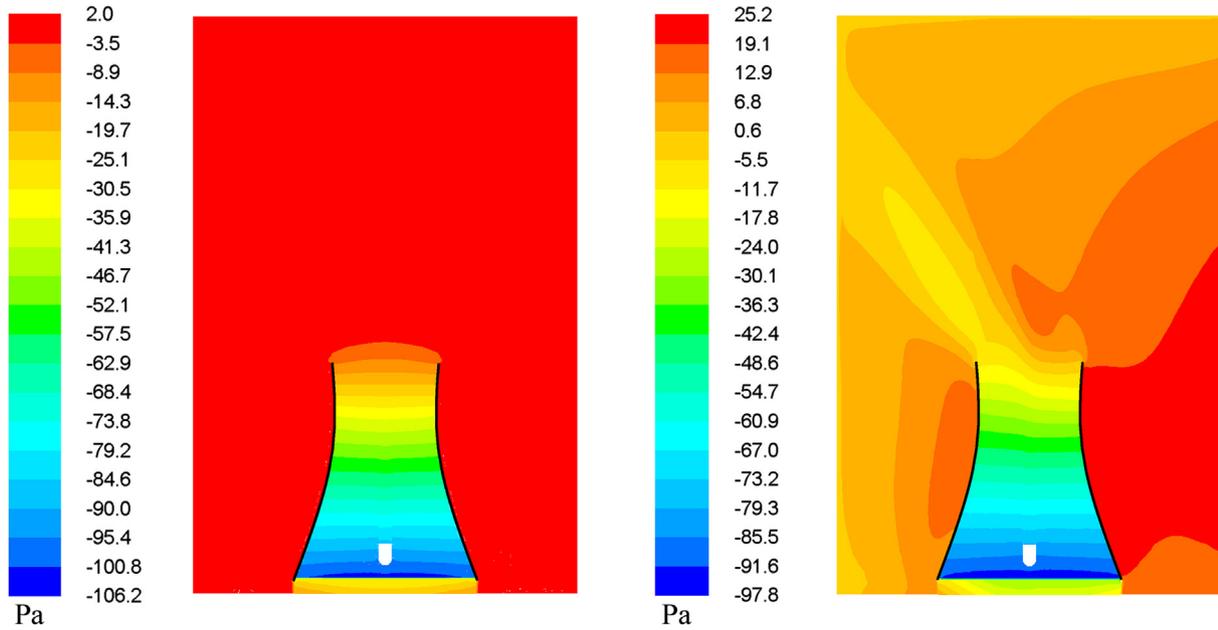


Fig. 7. Contours of pressure for no wind conditions (left) and wind speed $u_0 = 1.55$ m/s (right).

obtain considerable cold water temperature decrease. Therefore it can be concluded that the flue gas injection, due to low air mass flow rate increase through the tower, does not improve the cooling considerably.

3.4. The effect of wind

The effect of wind on cooled water temperature and gas mass flowrate through the tower has also been examined. The wind speeds u_0 of 0, 0.5, 1.55, 3.0 and 4.5 m/s at $z_0 = 2$ m have been assigned by means of the velocity profile given by Eq. (6) to the boundary of the computational domain. The obtained results are summarized in Table 8, where gradual increase of mean water

temperatures at the rain zone inlet (fill zone outlet) and rain zone outlet (water basin inlet) are observed. As can be seen the difference between the mean rain zone inlet and outlet temperatures increases with increasing wind speed. This is attributed to the intensified heat and mass transfer in the rain zone at higher wind speeds. This intensification is however associated with reduction of the total gas mass flowrate through the tower and of heat and mass transfer in the fill zone. The total mass flowrate presented in Table 8 is the sum of air and flue gas mass flowrates evaluated at the tower outlet.

In Figs. 7 and 8 contours of static relative pressure (relative to the measured atmospheric pressure for case c3 of 995.3 hPa) for various wind speeds are presented. It can be seen that for

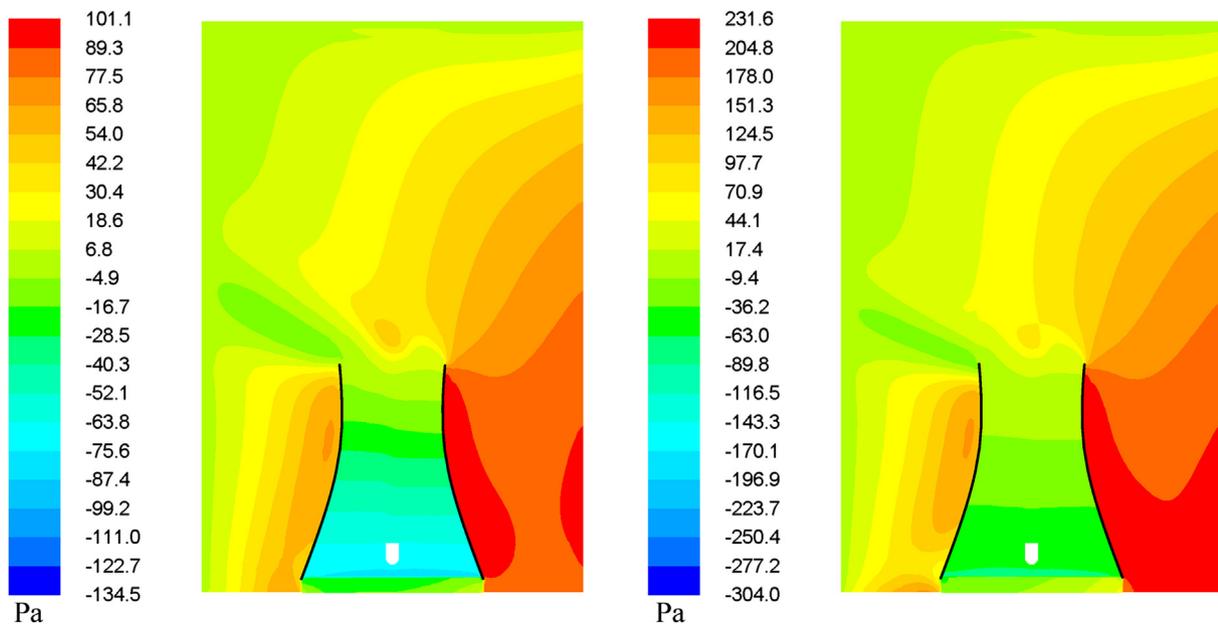


Fig. 8. Contours of pressure for wind speed $u_0 = 3.0$ m/s (left) and $u_0 = 4.5$ m/s (right).

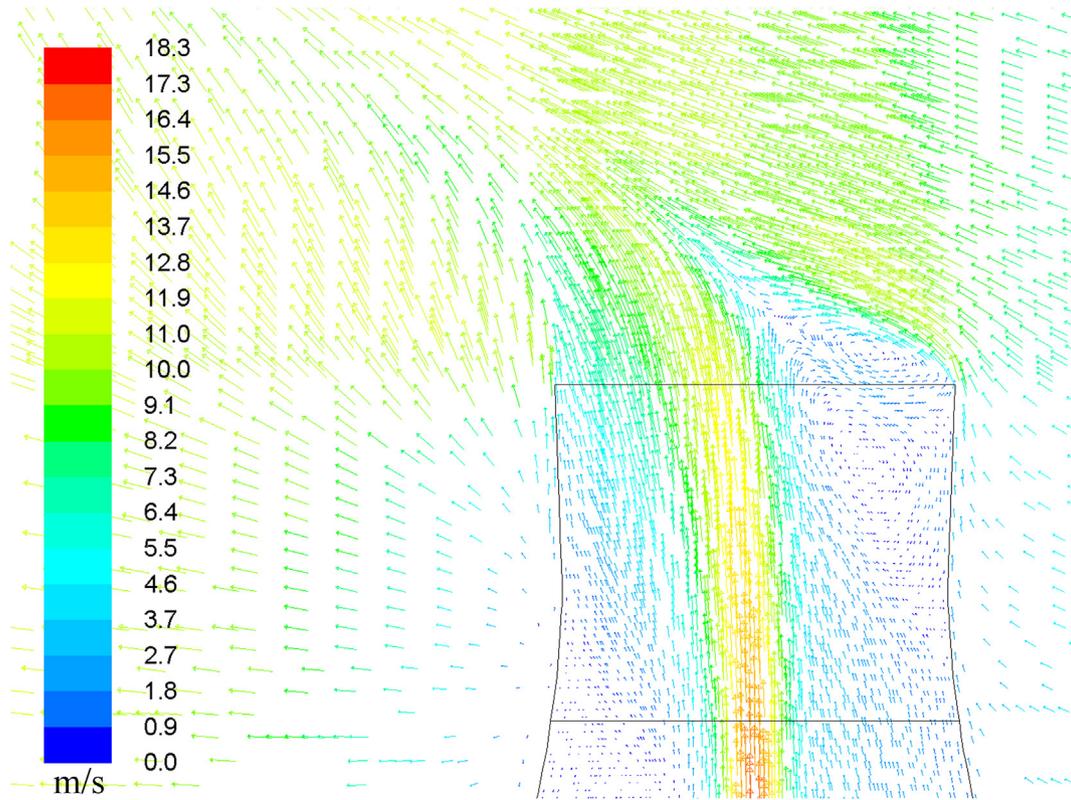


Fig. 9. Vectors of velocity magnitude near the tower outlet for wind speed $u_0 = 3.0$ m/s.

increasing wind speed the increasing pressure, as well as the increasing strength of the air stream, form a cover above the tower outlet reducing the flow through the tower. It can be seen that the pressure increase is partly a result of formation of stagnation region on the right side of the tower. The increasing wind speed forms also a recirculation region near the tower outlet reducing the flow through the tower. This is observed already for wind speed $u_0 = 3.0$ m/s which is presented in Fig. 9 as vectors of velocity magnitude. Reduction of air velocity is also observed just above the fill on the left side of the tower. The wind speed affects the rising plume not only above the tower but already inside and is a reason

of formation of recirculation regions. Therefore the injected flue gas can touch the tower shell. In Fig. 10 mass fraction of SO_2 present in the flue gas at the cooling tower outlet plane is presented for three wind speeds ($u_0 = \{1.55, 3.0, 4.5\}$ m/s). It can be seen that already for wind speed 3.0 m/s the flue gas mixed with air can flow close to the tower shell. It can be therefore concluded that the tower shell should be covered with corrosion resistant liner. In Figs. 11–13 streamlines released at various heights colored by velocity magnitude are presented. It can be seen that even for the highest velocity the bent plume does not form strong tower downwash, although a recirculation region behind the rising plume is

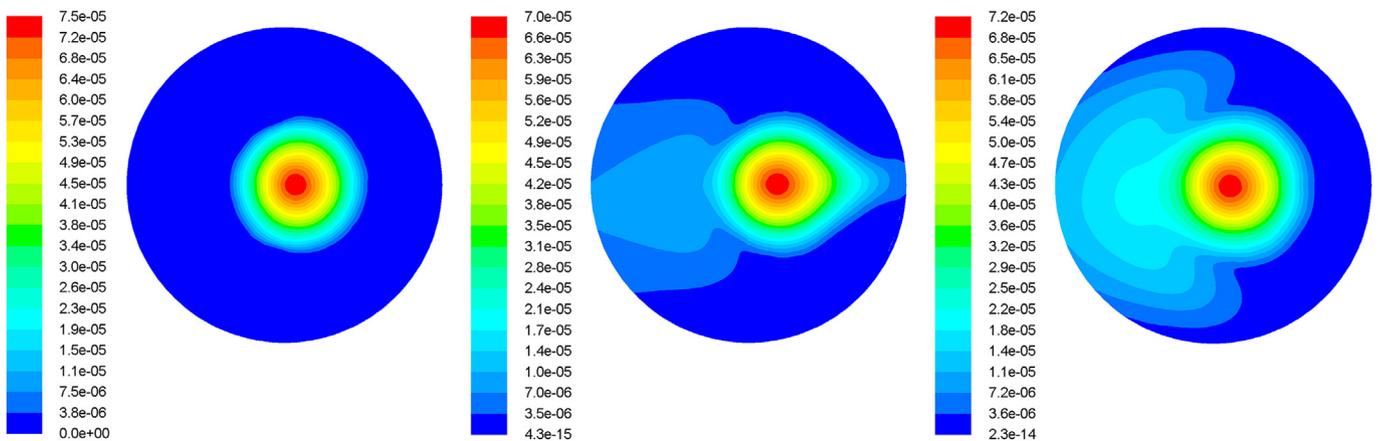


Fig. 10. Contours of SO_2 mass fraction (g/g) at tower outlet (120 m) for various wind speeds: 1.55 m/s (left), 3.0 m/s (middle) and 4.5 m/s (right).

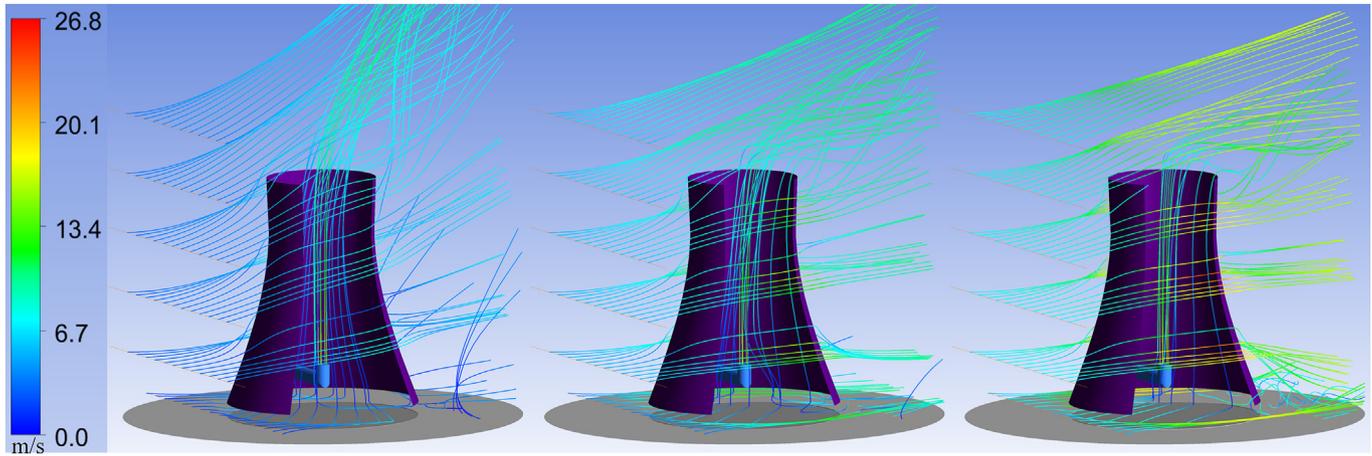


Fig. 11. Streamlines colored by velocity magnitude released at various elevations for $u_0 = 1.55$ m/s (left), $u_0 = 3.0$ m/s (middle) and $u_0 = 4.5$ m/s (right). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

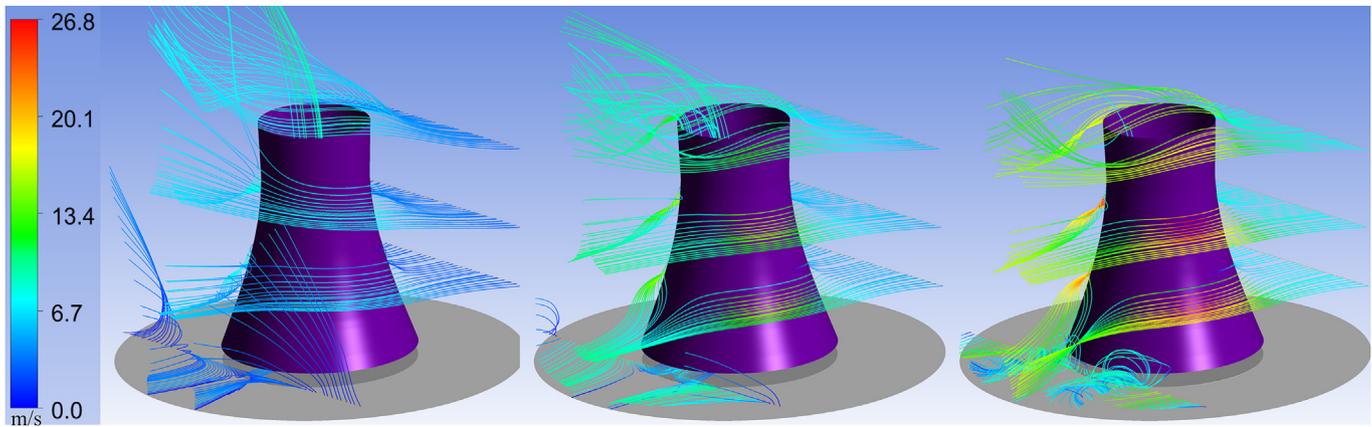


Fig. 12. Streamlines colored by velocity magnitude released at various elevations for $u_0 = 1.55$ m/s (left), $u_0 = 3.0$ m/s (middle) and $u_0 = 4.5$ m/s (right). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

observed. Increased downwind flue gas concentrations are also not observed at elevations much lower than the tower outlet. The increasing wind velocity affects also the air flow through the rain zone and the amount of air sucked by the low pressure inside. It can be seen in Figs. 12 and 13 that in all analyzed cases some of the air is

sucked to the tower from the downwind region. This is specifically visible in Fig. 13, where the streamlines are released at various locations 5 m above the ground level. This effect is however considerably reduced for the highest wind velocity. Trailing vortices are also formed on both sides at the top of the tower for

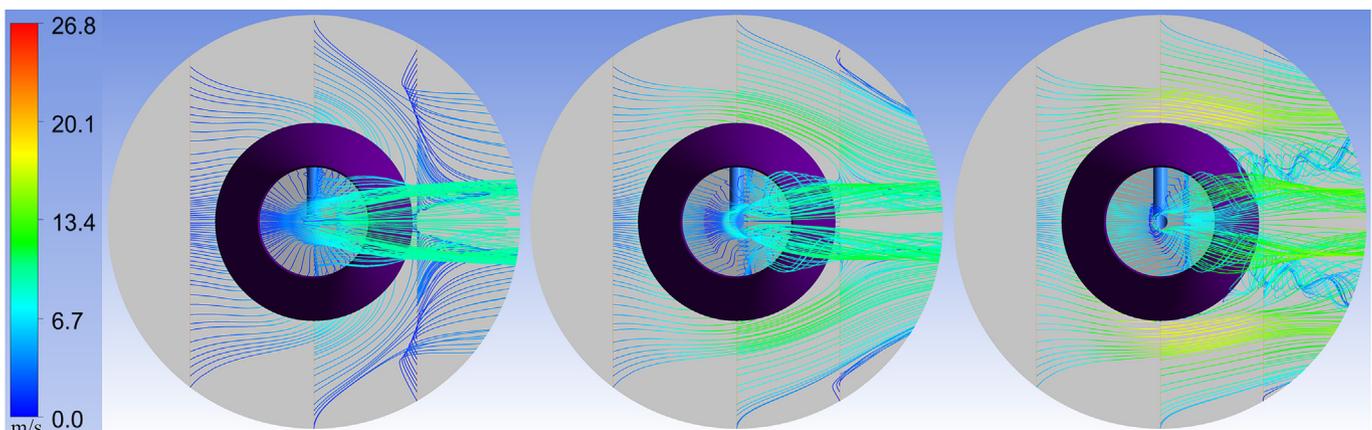


Fig. 13. Streamlines colored by velocity magnitude released at various positions 5 m above the ground level for $u_0 = 1.55$ m/s (left), $u_0 = 3.0$ m/s (middle) and $u_0 = 4.5$ m/s (right). (For interpretation of the references to colour in this figure legend, the reader is referred to the web version of this article.)

$u_0 = 3.0$ and $u_0 = 4.5$ m/s. For the latter case, they are also observed at the bottom, where the air passing the rain zone leaves the tower and is sucked to the low pressure downwind region.

4. Conclusions

A previously developed CFD model of a natural draft wet-cooling tower has been adopted to account for flue gas injection. The adopted model has been validated against experimental data, however only the mean cooled water temperature could be compared. The predicted water temperatures agree well with the measured values. The model has then been used to study the influence of flue gas injection on cooled water temperature and air mass flowrate through the tower. It has been shown that the injection has positive, but insignificant effect on cooled water temperature due to a slight increase of air flow rate through the tower. Finally operation of the tower with flue gas injection under wind conditions has been studied. The very well known effect of cooled water temperature increase with increasing wind speed has been confirmed. It was also shown that under strong wind conditions, the centrally introduced flue gas can flow close to the tower shell due to inflow of cold air and bending of the gas (air and flue gas) stream. This can increase the risk of corrosion of the tower shell and therefore application of corrosion resistant liner is required.

Acknowledgements

This study has been supported by the statutory research fund of the Silesian University of Technology, Faculty of Energy and Environmental Engineering, Institute of Thermal Technology.

References

- [1] D. Kröger, Air-cooled Heat Exchangers and Cooling Towers, Penn Well Corporation, Oklahoma, USA, 2004.
- [2] T. Jin, L. Zhang, K. Luo, J. Fan, Three-dimensional numerical study on thermal performance of a super large natural draft cooling tower of 220m height, *J. Therm. Sci.* 22 (3) (2013) 234–241.
- [3] P. Córdoba, Status of flue gas desulphurisation (FGD) systems from coal-fired power plants: overview of the physico-chemical control processes of wet limestone FGDs, *Fuel* 144 (2015) 274–286.
- [4] G. Ernst, J. Leidinger, K. Natusch, H. Petzel, G. Scholl, R. 45, Kühlturm und Rauchgasentschwefelungsanlage des Modellkraftwerkes Völklingen, *Fortschr.-Ber. VDI*, vol. 15, VDI Verlag, Düsseldorf, Germany, 1986 (in German).
- [5] D. Busch, R. Harteb, W. Krätzig, U. Montag, New natural draft cooling tower of 200 m of height, *Eng. Struct* 24 (12) (2002) 1509–1521.
- [6] M. Roth, R. Gerber, A. Niepel, Commissioning and operation of the world-wide largest natural draught wet cooling tower of a lignite-fired power plant with flue gas discharge, in: I. Mungan, U. Wittek (Eds.), *Natural Draught Cooling Towers*, Proceedings of the Fifth International Symposium on Natural Draught Cooling Towers, Istanbul, Turkey, 20–22 May 2004, CRC Press, London, UK, 2004, pp. 73–78.
- [7] H. Damjakob, G. Duddeck, H. Wöhler, Flue gas discharge via natural draught cooling towers - a survey, in: U. Wittek, W. Krätzig (Eds.), *Natural Draught Cooling Towers*, Proceedings of the 4th International Symposium of Natural Draught Cooling Towers, Kaiserslautern, Germany, 29–31 May 1996, A.A. Balkema, Rotterdam, Netherlands, 1994, pp. 45–54.
- [8] R. Al-Waked, M. Behnia, CFD simulation of wet cooling towers, *Appl. Therm. Eng.* 26 (2006) 382–395.
- [9] R. Al-Waked, M. Behnia, Enhancing performance of wet cooling towers, *Energy Convers. Manag.* 48 (2007) 2638–2648.
- [10] R. Al-Waked, Crosswinds effect on the performance of natural draft wet cooling towers, *Int. J. Therm. Sci.* 49 (2010) 218–224.
- [11] N. Williamson, S. Armfield, M. Behnia, Numerical simulation of flow in a natural draft wet cooling tower - the effect of radial thermofluid fields, *Appl. Therm. Eng.* 28 (2008) 178–189.
- [12] N. Williamson, M. Behnia, S. Armfield, Comparison of a 2D axisymmetric CFD model of a natural draft wet cooling tower and a 1D model, *Int. J. Heat Mass Transf.* 51 (9–10) (2008) 2227–2236.
- [13] N. Williamson, M. Behnia, S. Armfield, Thermal optimization of a natural draft wet cooling tower, *Int. J. Energy Res.* 32 (2008) 1349–1361.
- [14] A. Klimanek, R. Białecki, Z. Ostrowski, CFD two scale model of a wet natural draft cooling tower, *Numer. Heat. Transf. Part A Appl.* 57 (2010) 119–137.
- [15] A. Klimanek, Numerical modelling of natural draft wet-cooling towers, *Arch. Comput. Methods Eng.* 20 (1) (2013) 61–109.
- [16] A. Klimanek, R. Białecki, A 3D CFD model of a natural draft wet-cooling tower, *Arch. Thermodyn.* 30 (4) (2009) 119–132.
- [17] H. Reuter, D. Kröger, A new two-dimensional CFD model to predict the performance of natural draught wet-cooling towers packed with trickle or splash fills, in: *14th International Heat Transfer Conference, IHTC14*, ASME, Washington, DC, USA, 2010, pp. 589–598.
- [18] M. Schatzmann, A. Lohmeyer, G. Ortner, Flue gas discharge from cooling towers. wind tunnel investigation of building downwash effects on ground-level concentrations, *Atmos. Environ.* 21 (8) (1987) 1713–1724.
- [19] Q. Han, D. Liu, F. Chen, Z. Yang, The energy-saving benefit and economic evaluation analysis of cooling tower with flue gas injection, in: *Proceedings of the 1st International Conference on Sustainable Power Generation and Supply (SUPERGEN)*, Power Network Technology Press, Beijing, China, 2009. S08P0643.
- [20] A. Jahangiri, A. Golneshan, Feasibility study of the thermal performance improvement of natural draft dry cooling towers due to flue gas injection, *World Appl. Sci. J.* 12 (4) (2001) 568–575.
- [21] T. Eldredge, D. Benton, J. Hodgson, An investigation of the effects of flue gas injection on natural draft cooling tower performance, *J. Eng. Gas. Turbines Power* 119 (2) (1997) 478–484.
- [22] H. Ma, F. Si, L. Li, W. Yan, K. Zhu, Effects of ambient temperature and crosswind on thermo-flow performance of the tower under energy balance of the indirect dry cooling system, *Appl. Therm. Eng.* 78 (2015) 90–100.
- [23] A. Drączyk, E. Magiera, Report on Measurements of Cooling Tower No. 2 at Jaworzno III S.a. Power Plant, vol. 20/99(04), Zakłady pomiarowo-badawcze energetyki Energopomiar Sp. z o.o., Gliwice, Poland, 1999 (in Polish).
- [24] A. Klimanek, R. Białecki, Solution of heat and mass transfer in counterflow wet-cooling tower fills, *Int. Commun. Heat. Mass* 36 (6) (2009) 547–553.
- [25] A. Stechman, Procedures for Winter Operation of Wet-cooling Towers (PhD thesis), Faculty of Energy and Environmental Engineering, Silesian University of Technology, Gliwice, Poland, 2006 (in Polish).
- [26] R. Rish, The design of a natural draught cooling tower, in: *Proceedings of 2nd International Heat Transfer Conference*, Boulder, Colorado, USA, 1961, pp. 951–958.
- [27] ANSYS FLUENT, *Fluent 14.0 Theory Guide*, ANSYS, Inc, 2011. Software documentation.
- [28] R. Terblanche, H. Reuter, D. Kröger, Drop size distribution below different wet-cooling tower fills, *Appl. Therm. Eng.* 29 (2009) 1552–1560.
- [29] J. Juda, S. Chrościel, Air Protection, WNT, Warsaw, Poland, 1974 (in Polish).
- [30] R. Gryboś, S. Tomaszek, Climate Change Processes above Industrialized Area, Wydawnictwo Politechniki Śląskiej, Gliwice, Poland, 1997 (in Polish).