CFD Computation of Natural Draft Cooling Tower Flow

T. Hyhlík
Czech Technical University in Prague, Faculty of Mechanical Engineering, Department of Fluid Dynamics and Thermodynamics, Prague

Abstract

The model of the cooling tower is created by using commercial CFD code Fluent. Produced CFD model includes natural draft and hydraulic losses. The influence of the hydraulic losses to the character of the flow is observed. Natural draft is generated in the model by means of the source term in the momentum balance equation in the direction perpendicular to the bottom of the cooling tower. Hydraulic losses are included in the model by means of the internal boundary conditions and by means of the sinks in the individual balance equations. In the case of hydraulic losses inside the rain zone the users defined function creating sink in the momentum balances is programmed.

Introduction

Natural draft cooling towers are generally used in thermal power plants to remove heat absorbed in circulating cooling water. The performance of cooling tower can be affected by effect of crosswind condition. The knowledge of three dimensional flow structures can help us to design internal part of cooling tower. This is important especially in the case where flue gas goes directly to the cooling tower. Natural draft cooling tower can operate in different working regimes. Spreading of flue gas inside the cooling tower is very important from the construction service life point of view.

Current design procedure is based on one dimensional analytical model [1] which is based on experimental correlations for loss coefficients, heat transfer coefficient and mass transfer coefficient.

Thesis [2] proposed one dimensional model based on the system of ordinary differential equations. Eulerian approach is used both for the gas and liquid phase. Designed model allows to solve time evolution of the state in the natural draft cooling tower.

CFD model developed in the work [3] solves multi-phase steady state flow in three dimensional domain using commercial code Fluent. Air flow is solved as continuous phase using Eulerian approach whereas droplet trajectories are solved as dispersed phase using the Lagrangian approach [3]. The influence of crosswind condition to the thermal performance of natural draft wet cooling tower was also investigated in this work. Crosswind conditions has been found to affect significantly thermal performance of cooling tower. Both increase of the outlet water temperature and decrease of outlet water temperature was observed for different crosswind velocity [3].

CFD model developed in this work is not so complex as the model used in the article [3] nevertheless allows to solve complicated flow structures inside and outside cooling tower unlike the model used in the work [2].

Numerics

Single phase air flow is solved in Fluent code. The effect of liquid phase is included through the source and sink terms in the governing equations. The code was used to solve Reynolds Averaged Navier-Stokes equations closed with realizable k-ε model of turbulence. Pressure based on segregated implicit solver was used. Pressure correction method SIMPLE was utilized. Convective terms were discretized using second order upwind scheme.

Pressure Losses

Pressure losses are included through the radiator condition in Fluent code. Pressure loss in the case of radiator condition can be expressed as

$$\Delta p = k_i \frac{1}{2} \rho V_P^2, \tag{1}$$

where k_i represent non-dimensional loss coefficient of the individual part of the cooling tower, ρ is the density and V_P is the component of the velocity perpendicular to the surface of the individual part of the cooling tower. Pressure loss was used at the inlet to the cooling tower and at the fill inlet. Both pressure loss in the fill and the pressure loss in the drift eliminators were prescribed at the fill inlet.

Source Terms

Draft of the cooling tower can be then expressed as a source term in the momentum equation in the direction perpendicular to the bottom of the cooling tower. This idea is based on Boussinesq natural convection model [4]. The source term is generally given as

$$(\rho_\infty - \rho) g, \tag{2}$$

where ρ_∞ is the density from the standard atmosphere, ρ is actual density in the natural draft cooling tower and g is gravitational constant. The source term was applied only in the fill as constant driven by prescribed mass flow rate through the cooling tower. The value of required mass flow rate was taken from the model based on [1].

Pressure loss through the rain zone was modeled through the sink in momentum equations. This momentum sink can be expressed as

$$S_m = -K_i \frac{1}{2} \rho V^2, \tag{3}$$

where K_i is rain zone loss coefficient per meter, ρ is the density and V is the the velocity magnitude.

Boundary Conditions

In the case without influence of crosswind velocity the total pressure is prescribed at the inlet boundary condition. In the case with crosswind velocity the velocity profile at the inlet boundary condition [3] was prescribed as

$$\frac{U}{U_{ref}} = \left(\frac{y}{y_{ref}}\right)^{0.2}, \tag{4}$$

where U is the inflow velocity magnitude, U_{ref} is reference velocity, y is axis perpendicular to the bottom of the cooling tower in the upward direction and y_{ref} is the reference distance. Reference distance was set to $10m$ and reference velocity was set to $10ms^{-1}$ in the calculated case.

Pressure outlet boundary condition was used in both cases, i.e. in the case of crosswind velocity and in the case without influence of crosswind.

Computational Domain

Numerical computation was done in the cylindrical domain with radius of 1000 m and height 500 m. One-quarter model was solved in the case where influence of crosswind was not taken into consideration.

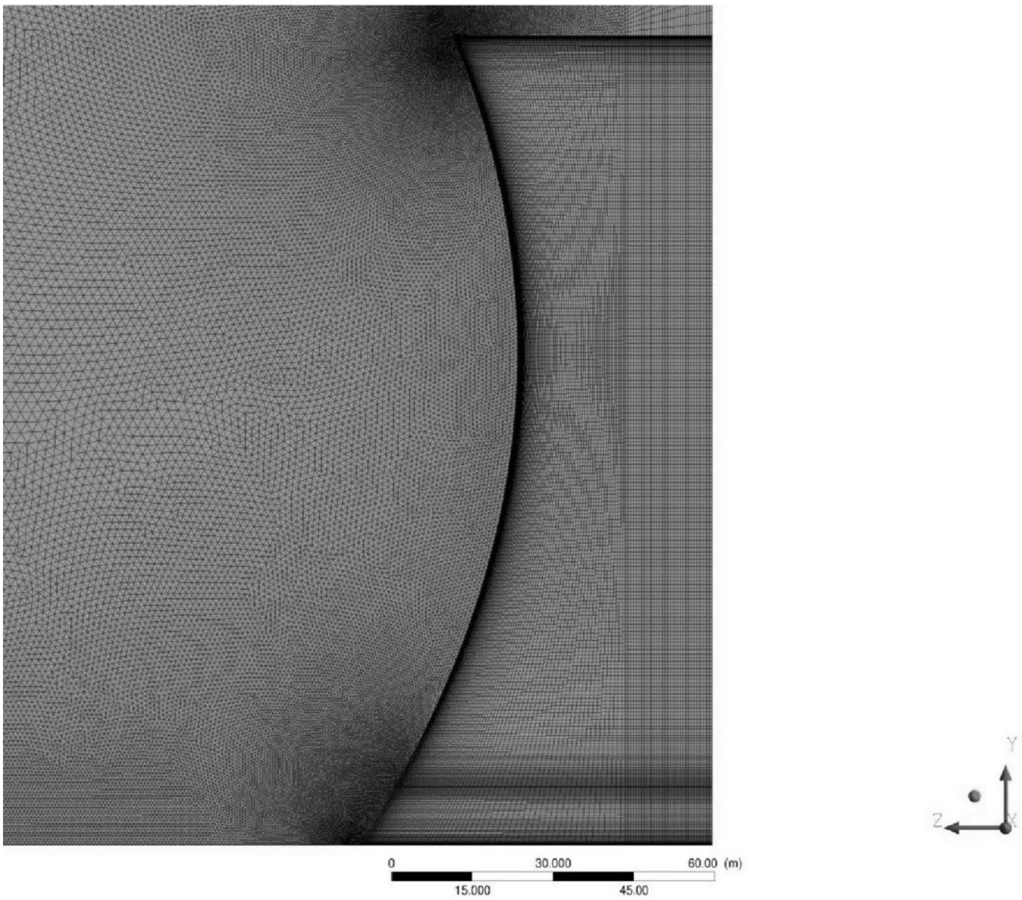

Fig. 1. Detail of the computational mesh

In the case of the simulation where crosswind was taken into account one-half model was solved. Computational mesh has approximately 25 millions of cells in the one-half model. Detail of computational mesh is shown in figure 1.

Results

Complex flow structure with vortexes is visible in figure 2. Vortex structures are created both in the leeward side of the cooling tower and inside the cooling tower close to the top. Vortex structures in the leeward side are present at the bottom and at the top of the tower.

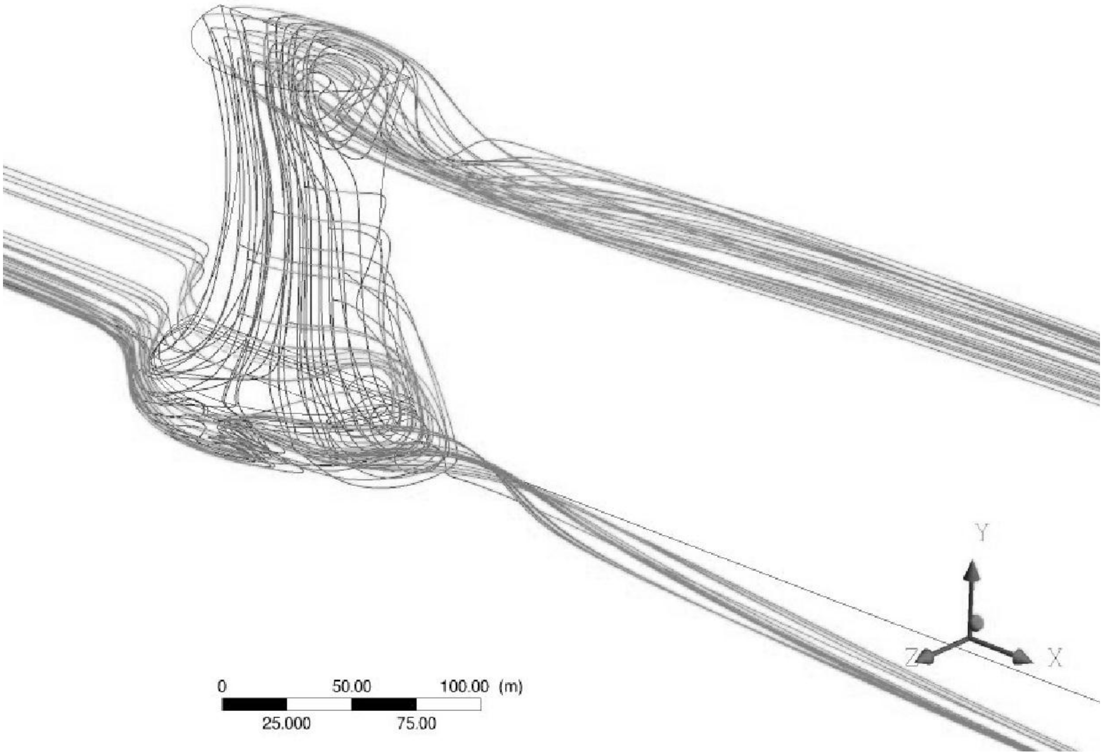

Fig. 2. Streamlines in the case of crosswind configuration

Conclusion

The simulation of the flow through and around natural draft cooling tower was performed using commercial code Fluent. Developed CFD model allows us to obtain relevant results with knowledge of loss characteristics of individual parts of cooling tower. Crosswind velocity has been found to affect significantly the structure of the flow field inside and outside the cooling tower.

Acknowledgement

The work has been supported by Ministry of Education, Youth and Sports of the Czech Republic within project No. 1M06059.

References

[1] Kröger, D. G. : *Air-Cooled Heat Exchangers and Cooling Towers.* Penn Well Corporation, 2004

[2] Zuniga-Gonzales, I. : *Modeling Heat and Mass Transfer in Cooling Towers with Natural Convection.* Ph.D. Thesis, CTU in Prague, 2005

[3] Al-Waked, R., Behnia, M. : CFD simulation of wet cooling towers. *Applied Thermal Engineering*, 26, pp. 387-395, 2006

[4] Bejan, A. : *Convection Heat Transfer.* John Wiley & Sons, Inc., 2004