A 3D CFD MODEL OF A NATURAL DRAFT WET-COOLING TOWER

Adam Klimanek*, Ryszard A. Białecki*, Ziemowit Ostrowski*

*Institute of Thermal Technology, Silesian University of Technology
Konarskiego 22, 44-100 Gliwice, Poland
e-mail: {adam.klimanek, ryszard.bialecki, ziemowit.ostrowski}@polsl.pl

ABSTRACT

A 3D CFD model of a natural draft wet-cooling tower is presented in this paper. A commercial code Fluent has been used to simulate the transport phenomena inside the 100 m high tower and in the surrounding environment (300 m high cuboid with 200 m by 200 m base). A total number of 4.84 million hybrid grid cells define the discretized flow domain. The three-dimensional model is an extension of a previously developed 2D axisymmetric model that has been successfully validated against experimental data [1]. There is a large scale difference in the phenomena occurring in the tower. The geometrical scales of the environment are of the order of hundreds meters whilst the heat and mass in the fill zone is transferred within a few centimeters. The most challenging task of the study is providing a link between these scales by building sub-models of heat, mass and momentum transfer in the smaller scales. The developed model is therefore supplied with a low dimensional representation of heat and mass transfer in the fill, whose purpose is to determine the heat and mass rejected from the cooling water to the air. The heat and mass transfer is well described by a set of four ordinary differential equations comprising the so called two point boundary value problem [2]. In order to reduce the computational time, the heat and mass transfer is then represented by an original statistical technique called the Proper Orthogonal Decomposition. The Euler-Euler multiphase model was used to simulate the flow, heat and mass transfer in the rain zone. The RNG k-ε model with the dispersed option is used for the turbulence modeling of multiphase flow. Application of all required models infers that a total number of 14 transport equations are solved in Fluent. This makes the solution process demanding and increases the overall computational time additionally to the large mesh size. The obtained results showed considerably different flow fields obtained from the 3D simulation than those from the 2D axisymmetric model mainly above the tower. Due to rapid mixing and buckling of the plume the draft is reduced in the 3D model showing lower average velocities and higher cooled water temperature. The model can be used to predict the three dimensional phenomena like the effect of cross-wind, dispersion of desulphurized flue gas introduced to the plume, etc., however it should be validated against large scale experimental results first.

References

[1] A. Klimanek, R.A. Białecki, On a Numerical Model of a Natural Draught Wet-cooling Tower. *Archives of Thermodynamics*, 29 (2008), No. 4, pp. 63-72.

[2] A. Klimanek, R.A. Białecki, Solution of heat and mass transfer in counterflow wet-cooling tower fills. *International Communications in Heat and Mass Transfer*, 36 (2009), No. 6, pp. 547-553.