2014 3rd International Conference on Informatics, Environment, Energy and Applications
IPCBEE vol.66 (2014) © (2014) IACSIT Press, Singapore
DOI: 10.7763/IPCBEE. 2014. V66. 32

Numerical Simulation of Plume Motions for Cooling Tower in an Inland Nuclear Power Plant

Chunlai Tian+, Shan Zhou and Liyong Han

State Nuclear Power Technology Corporation Research & Development Center, Beijing, 100190, China

Abstract. In order to analyze the plume motions, the computational fluid dynamics model of the air flow field around the cooling tower is established in this paper. The super big scale cooling tower is specially designed for the inland nuclear power plant. The free open source computational fluid dynamics solver software named OpenFOAM is used. The velocity, temperature and pressure fields are demonstrated. The minimum speed appears near the top lee side of the tower. The highest temperature appears at the tower exit and the temperature field could predict the plume motions. With the increase of the distance, the plume rises to 250 m at the position 1000 m away from the cooling tower and would rise continually. This paper provides a low-cost reliable numerical simulation method to predict the plume and air flow characteristics around the cooling tower.

Keywords: cooling tower, CFD, plume, inland nuclear power plant, OpenFOAM.

1. Introduction

Cooling towers are important heat transfer devices which are commonly used to dissipate heat from power plants. In cooling towers, atmospheric air cools warm water by direct contacting. Distributing water is operated over a heat transfer surface across which a stream of air is passing while the cooling tower is running. Water droplets are incorporated in the air stream and will be taken away from the units with high temperature and humidity[1]. Many big scale cooling towers have been designed and would be built in the future with the inland nuclear plants. They are designed in type of big scale natural draft cooling towers with more spray area and higher height[2].

Plume motions coming from cooling towers affect plant areas and buildings. It is necessary to pay attention to the plume performance in consideration of environmental protection and radiological safety nearby the nuclear power plant. Study of fluid flow characteristics with plume dispersion has been carried out through computational fluid dynamics (CFD) simulation by many researchers[3-5]. Their works focused on the normal cooling tower in the power plant. However the normal cooling tower is different from the big scale one, which is specially designed and applied in the inland nuclear power plant[6].

The affected areas of the plume from the big scale cooling tower are different from that from the normal one. In order to obtain the effects of the plume on fluid flow characteristics, a computational fluid dynamics model for predicting the plume flow is established in this paper. The velocity, pressure and temperature are solved to predict the plume flow with the wind field over the cooling tower. This work is carried out with a free open source CFD software OpenFOAM and a post-process software ParaView[7].

2. Modeling

2.1. Mathematical CFD model theory

+ Corresponding author. Tel.: +86-10-59367633; fax: +86-10-59367716.
E-mail address: tianchunlai@snptrd.com.

The incompressible Navier-Stokes equations are used to describe this three dimensions fluid flow problem[8]. The governing equations include the mass continuity equation, the momentum equations and the energy equation. They are

$$\frac{\partial \rho}{\partial t} + div\left(\rho U\right) = 0 \ , \tag{1}$$

$$\begin{aligned} &\frac{\partial u}{\partial t} + \mathrm{div}\left(uU\right) = \mathrm{div}\left(\nu \mathrm{grad} u\right) - \frac{1}{\rho}\frac{\partial p}{\partial x} \\ &\frac{\partial v}{\partial t} + \mathrm{div}\left(vU\right) = \mathrm{div}\left(\nu \mathrm{grad} v\right) - \frac{1}{\rho}\frac{\partial p}{\partial y} \\ &\frac{\partial w}{\partial t} + \mathrm{div}\left(wU\right) = \mathrm{div}\left(\nu \mathrm{grad} w\right) - \frac{1}{\rho}\frac{\partial p}{\partial z} \end{aligned} \ , \tag{2}$$

$$\frac{\partial \left(\rho T\right)}{\partial t} + \mathrm{div}\left(UT\right) = \mathrm{div}\left(\frac{\lambda}{\rho c}\mathrm{grad} T\right) + \frac{S}{\rho} \ , \tag{3}$$

where u is the flow velocity on x direction, v is on y direction and w is on z direction, U is the flow velocity field, ρ is the density, p is the pressure, ν is the kinematic viscosity, T is the temperature, λ is thermal conductivity, c is the heat capacity and S is the source item.

The standard k-ε turbulent model is applied here as

$$\begin{aligned} &\rho\frac{\partial k}{\partial t} + \rho u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(u + \frac{u_i}{\sigma_{\mathrm{k}}}\right)\frac{\partial k}{\partial x_j}\right] + G - \rho\varepsilon \\ &\rho\frac{\partial \varepsilon}{\partial t} + \rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(u + \frac{u_i}{\sigma_{\varepsilon}}\right)\frac{\partial k}{\partial x_j}\right] + c_1\frac{\varepsilon}{k}G - c_2\rho\frac{\varepsilon^2}{k} \end{aligned} \ , \tag{4}$$

where σ_{k} and σ_{ε} are the turbulent Prandtl numbers of the k equation and ε equation, c_1 and c_2 are constant values and G is described as

$$G = \frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \ . \tag{5}$$

These equations described the problem and would be solved through the semi-implicit method for pressure-linked equations (SIMPLE).

2.2. Configurations and CFD model

A big scale hyperbolic natural draught cooling tower is employed in this work. It is modelled with a height of 210 m, a base diameter of 85 m and an exit diameter of 65 m. The CFD simulation was performed on a domain with a length of 3000 m, a width of 1000 m and a height of 750 m. As shown in Fig.1, the CFD model is built by tetrahedral meshes with 160975 elements and 28383 nodes. The surface mesh of the cooling tower is refined with a growth ratio of 1.5.

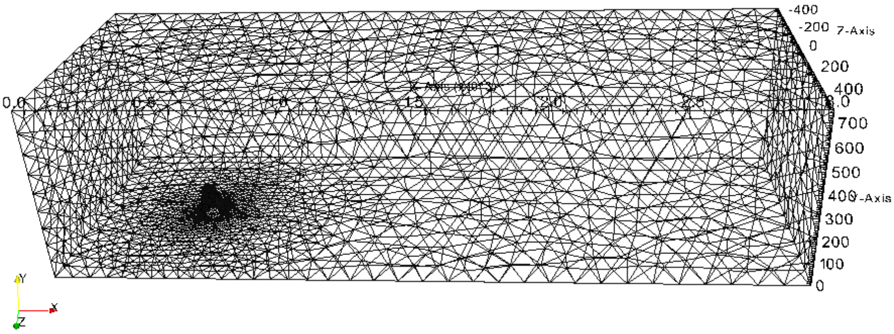

Fig.1: CFD model with tetrahedral elements.

2.3. Boundary conditions

The boundary conditions required for this work include the inlet, the outlet, the wall and the symmetry conditions. The air velocity 8 m/s and temperature 293 K were set to constant values as the inlet condition for domain inlet. The pressure outlet boundary condition is selected for domain exit. Conditions imposed on cooling tower exit are set to constant values for air velocity and temperature. The velocity is 4.5 m/s and the temperature is 300 K. Wall boundary conditions with non-slip are selected for the ground and for the cooling tower surface. Symmetry boundary conditions are selected for the domain sides and top.

3. Solver setup

The problem is solved by the basic CFD solver 'simpleFoam'. It is a steady-state solver using the SIMPLE algorithm for incompressible turbulent flow, which is distributed with the OpenFOAM. The solver configurations are set by the 'fvSolution' file, the 'fvSchemes' file and the 'controlDict' file in the case system folder. The Gauss upwind method is composed of 'divSchemes' and the Gauss linear corrected is composed of 'laplacianSchemes' in the 'fvSchemes' file. And in the 'fvSolution' file, the 'relaxationFactors' are defined.

4. Results and discussions

Results coming from the solver are post-processed by ParaView software. The contours of the X direction velocity and Y direction velocity are demonstrated in Fig.2. The minimum velocity on the X direction appears near the backside of the cooling tower. The maximum velocity on the Y direction is 5.07 m/s at the exit of the cooling tower.

(a)

(b)

Fig.2: X and Y velocity contour.

The temperature field result is shown in Fig.3. The maximum temperature is 316 K at the tower exit. The temperature decreases as the plume leaves the cooling tower. It is usefully to predict the plume's motion. As shown in Fig.4, the pressure contour is ranged with the maximum pressure of 39.4 Pa. Compared with the velocities in Fig.2, the area of the maximum pressure is near the minimum velocity.

Fig.3: Temperature contour.

Fig.4: Pressure contour.

The plume rise graph in log scale is shown in Fig.5. The position is from the cooling tower centre and the plume rise is calculated from the cooling tower exit. As shown in Fig.5, the plume rises to 250 m at the position 1000 m. Moreover, the plume continues rising. The further motion of the plume at the position more than 1000 m would be predicted in the future analysis.

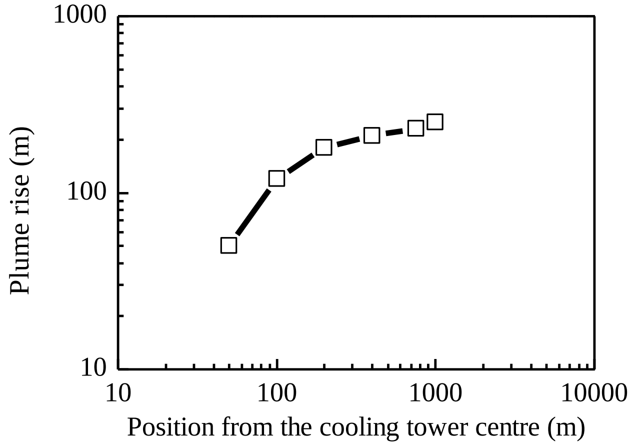

Fig.5: Plume rise prediction.

5. Conclusions

The CFD model of the plume from big scale cooling tower is built in this paper. Plume motions are predicted with the numerical solver in the free open source CFD software OpenFOAM. The results of velocities, temperature and pressure fields are shown. The plume rise with the centre position is demonstrated. This work supports a low-cost reliable method for CFD analysis of the air fluid field with the plume from the cooling tower, which is applied in the inland nuclear power plant.

6. Acknowledgements

The authors appreciate the support of Staff Independent Inovation Fund of SNPTC (State Nuclear Power Technology Company, China) with Grant No. SNP-KJ-CX-2013-10.

7. References

[1] Lucas M, Martinez P J, Ruiz J, et al. On the influence of psychrometric ambient conditions on cooling tower drift deposition. *International Journal of Heat and Mass Transfer*. 2009, **53**(2): 594-604.

[2] Zhou L, Ma S, Gong X, et al. Structure optimization for cooling tower of an AP1000 nuclear power plant. *Journal of Chinese Society of Power Engineering*. 2012, **32**(12): 985-989.

[3] England W G. *Cooling tower plumes defined and traced by means of computer simulation models*. Cooling Tower Institute, Houston, 1973.

[4] Bergstrom D J. Derksen D, Rezkallah K S. Numerical study of wind flow over a cooling tower. *Journal of Wind Engineering and Industrial Aerodynamics*. 1993, **46**(8): 657-664.

[5] Meroney R N. CFD prediction of cooling tower drift. *Journal of Wind Engineering and Industrial Aerodynamics*. 2006, **94**(6): 463-490.

[6] Wang X. Wind field simulation in an inland nuclear power plant based on a CFD model. *Journal of Meteorology and Environment*. 2012, **28**(6): 54-60.

[7] OpenFOAM User Guide. *http://www.openfoam.org/docs/user*.

[8] Anderson J D. *Conputational fluid dynamics*. McGraw-Hill, New York, 1995.