Numerical Analysis of Wind Flow Influence on Thermal Loading Inside Dry Cooling Towers

SAEED-REZA SABBAGH-YAZDI
IMAN KHALILI-POOYA
Civil Engineering Department,
KN Toosi University of technology,
No.1346 Valiasr Street, Tehran-19697,
Iran
SYazdi@kntu.ac.ir
I_khalilipooya@yahoo.com
And
NIKOS E. MASTORAKIS
Military Institute of University Education (ASEI)
Hellenic Naval Academy
Terma chatzikyriakou 18539
Piraues, GREECE
Master@wseas.org

Abstract: In this paper, the set of 3-dimensional equations for the incompressible air flow is combined with the equation for the temperature field are solved using FEM (Finite Element Method). Commercial FEM software is applied to simulate the changes in the velocity components, pressure and temperature distribution inside a Natural Draught Dry-Cooling tower due to the wind flow at supercritical Reynolds number ($Re = 24 \times 10^7$). This Dry-Cooling tower is located in the SHAZAND power station, which is under operation in north part of IRAN (ARAK province). The utilized numerical model solves the governing equations for wind flow on a three-dimensional unstructured Finite Element mesh. Using unstructured meshes provides the merit of accurate geometrical modeling of the curved boundaries of the Dry-cooling tower. Satisfactory results are obtained by the use of proper boundary conditions.

Key word: Finite Element Method (FEM), Dry-Cooling tower, Computational Fluid Dynamics (CFD), Temperature Distribution in Air Flow

1. Introduction:
Natural-draught Dry-Cooling tower is an energy-efficient system and water-saving cooling equipment in power plants, widely used in the regions that there is lack of water but rich in coal or oil, such as, Middle East countries like IRAN. However, the performance of Dry-Cooling towers is highly sensitive to the environment condition. The conventional design of cooling towers operating system does not consider the impact of windy conditions which may exist most of the time of operation. Hence, it is important to investigate the influence of the wind effect on the performance of the cooling towers. Since the air flow patterns and related temperature distributions considering the thermal buoyancy effects are two main factors of such an investigation, appropriate improvements of the system may be proposed using accurate modeling of the phenomena.

Some studies have been performed in the area. Wei experimentally investigated the mechanism of unfavorable effect of wind on cooling efficiency of Dry-Cooling towers [1]. Su studied the thermal performance of a Dry-Cooling tower under cross-wind conditions by using computational fluid dynamics (CFD) technologies [2]. The tower had vertical heat exchangers placed around the bottom. The study indicated that the wind-caused tangential airflow at external sides of the heat exchangers is the major cause for the cooling efficiency reduction. Fu and Zhai numerical investigated
the effect of cross-wind on two in-line Dry-cooling towers, which represented a more realistic scenario [3]. Transfer pattern from the single tower study, especially when verified that the wind-induced around-flow destroys the radial inflow into the cooling towers and thus significantly deteriorates the heat transfer performance at lateral sides. As a consequence, a straightforward idea to improve the performance of cooling towers is to utilize wind-break walls to break the around-airflow at both sides of towers and force the surrounding air entering the towers. Bender investigated, in a wind tunnel, the utilization of wind-break walls to balance the airflow rate into cooling tower intakes to prevent ice formations due to cold and windy weather [4]. Du Preez and Kroger also carried out extensive experimental and numerical research on the performance of wind-break walls underneath Dry-Cooling towers that have heat exchangers horizontally placed in the inlet cross-section of towers. Both of them concluded that good arrangement of wind-break walls can lead to significant reduction in the adverse effects of cross-wind on Dry-Cooling towers.

In this work, the research interest has been consistently focused on the cooling towers with vertical heat exchangers around the bottom of towers, which generally confront the most significant impact from cross-winds using computational modeling the system. For this purpose, ANSYS 5.4 FEM 3-Dimensional solver is used for simulation of steady air flow and heat transfer around and inside a natural draught cooling tower.

2. Governing Equations:
In this numerical work, it is assumed that the flow is in fully turbulence condition (Re = 24 *10^7), incompressible and 3-dimensional. The Control Volume is considered to be homogeneous, isotropic and in local thermo-dynamics equilibrium with fluid (Air). In addition, the thermo-physical properties of the fluid and the solid are assumed to be constant. The governing conservation equations for the problem can then be written separately for the fluid region and porous region. The flow over the channel is governed by the incompressible Reynolds equations. The resulting dimensionless governing equation for steady state case can be written in vector form as:

$$\nabla V = 0$$

$$(V \cdot \nabla)V = -\frac{1}{\rho} \nabla P + \nabla \cdot \left( \frac{\sigma}{\rho} - \beta (T - T_a) \right) g + S_h$$  \hspace{1cm} (1)

$$\rho (V \cdot \nabla) T = -\nabla \cdot (\Gamma \nabla T) + Q_h$$

Where $V$ is velocity vector, $T$ the temperature, $\sigma$ is the stress tensor which can be calculated using the following formula:

$$\sigma_{ij} = \mu S_{ij}$$  \hspace{1cm} (2)

Where $i, j = 1, 2, 3 : \mu$ is the molecular viscosity. $\beta$ is the volume expansion coefficient of air, $\beta = -1/\rho \frac{\partial \rho}{\partial T}$. For the complete gas $\beta = 1/T$; $T_a$ is the atmospheric temperature. The total pressure loss of the fluid through the porous media is calculated by means of drag term in the momentum equation, $S_h$.

When air flows through the media, heat can effect the air flow from the bottom walls, and therefore, in the energy equation, a heat term should be added:

$$Q_h = \alpha_h (T - T_a)$$  \hspace{1cm} (3)

Where $\alpha_h$ denotes the heat transfer coefficient and it is determined from Darcy Numbers. $\Gamma$ is the molecular heat conduction coefficient, and, it can be calculated using the following formula:

$$\Gamma = \frac{\mu}{Pr}$$  \hspace{1cm} (4)

Where $Pr$ is Prandlt Number of the fluid flow, where $Pr = C_p \mu / k = \nu / \alpha$

The local Nusselt number and dimensionless pressure drop is calculated as:

$$Nu = \frac{q H}{(T - T_a) k_f}$$  \hspace{1cm} (5)

$$\Delta P = \frac{\Delta p}{1/2 \rho (V_{ave})^2}$$  \hspace{1cm} (6)

The average Nusselt number and pressure drop in the channel are obtained by the integration of the local values over the channel.
3. Numerical Method:

In the previous section, the equations of the fluid environment are presented. This section presents the method of discretization of the equations. In Finite element’s method, computation domain should be divided into smaller sub domain (Elements). In each element, depended variable are approximated using polynomial functions. Therefore, velocity, pressure and heat in each element can be presented as:

\[ V(x,t) = \psi^T V(t) \]
\[ P(x,t) = \alpha^T P(t) \]
\[ T(x,t) = \Phi^T T(t) \] (7)

In above relations, \( V, P, T \) are time dependent nodal values, \( \psi, \alpha, \Phi \) are in a general format element’s shape functions. Sub situating these relations in the governing differential equations, result in a general format is:

\[ f(\psi, \alpha, \Phi, u, P, T) = R \] (8)

In this equation, \( R \) is residual of approximated equation. In order to integrate the equation, the weight residual Galerkins’ method is used.

\[ \int (f \cdot W) \, d\Omega = \int (R \cdot W) \, d\Omega = 0 \] (9)

There, \( W \) is weight function of the element, and \( \Omega \) is the sub domain of interest. The quantities of mass, momentum, energy, can be considered as one transferring scalar equation. In this equation, there are four type of term; transient, advection, diffusion and source terms. Each of these terms have been explained later. Scalar’s transferring equation for variable \( q \) is:

\[ \frac{\partial}{\partial t}(\rho \cdot \alpha) + \frac{\partial}{\partial x}(\rho \cdot \alpha \cdot V_x) + \frac{\partial}{\partial y}(\rho \cdot \alpha \cdot V_y) + \frac{\partial}{\partial z}(\rho \cdot \alpha \cdot V_z) = \] \[ \frac{1}{\rho} \cdot \frac{\partial}{\partial x}(\Gamma_\alpha \cdot \frac{\partial \alpha}{\partial x}) + \frac{1}{\rho} \cdot \frac{\partial}{\partial y}(\Gamma_\alpha \cdot \frac{\partial \alpha}{\partial y}) + \frac{1}{\rho} \cdot \frac{\partial}{\partial z}(\Gamma_\alpha \cdot \frac{\partial \alpha}{\partial z}) + S_\alpha \] (10)

\( \Gamma_\alpha \) is diffusion coefficient and \( S_\alpha \) is source term.

Table (1) presents the variables (Degree of Freedom), coefficients and source terms of each equation. It is important to note that pressure is computed using a pressure connection equation which is explained later. With equations descritize by application of finite elements’ method, the general descritized is matrix from equation of FE model a process of elements’ is obtained as:

\[ \begin{bmatrix} A^\text{transient}_e & A^\text{advection}_e & A^\text{diffusion}_e \end{bmatrix} \begin{bmatrix} \psi_e \end{bmatrix} = \begin{bmatrix} f_e \end{bmatrix} \] (11)

For calculating elements’ integrals of, matrix formulation, weight residue Galerkins method is used.

First term of elements’ equation is transient term which is:

\[ A^\text{transient}_e = \int W \cdot \frac{\partial (\alpha)}{\partial t} \, d(\Omega) \] (12)

\( W \) is weight function of the which is considered as the shape function of the element.

Using the approximation of concentrated mass this term is written as:

\[ \int W \cdot \frac{\partial (\alpha)}{\partial t} \, d(\Omega) = \frac{\partial (\alpha)}{\partial t} \int W \cdot d(\Omega) \] (13)

For calculating transient derivative term of equation, following difference is used:

\[ \frac{\partial (\alpha)}{\partial t} = \frac{3(\alpha)_{n-2} - 4(\alpha)_{n-1} + 3(\alpha)_{n}}{2\Delta t} \] (14)

Next part of elements’ equation is advection term that is discretized base on this theory that pure advection transfer take place along specific stream lines. Flow field can be imaged as a collection of stream lines which are tangent in each point on velocity vectors. Therefore, advection term can be explained based on velocity of stream line. Since advection transfer take place along stream lines, it can be considered constant along a stream lines in elements.

\[ \frac{\partial (V_x \alpha)}{\partial x} + \frac{\partial (V_y \alpha)}{\partial y} + \frac{\partial (V_z \alpha)}{\partial z} = \frac{\partial (V_s \alpha)}{\partial s} \] (15)

Therefore:

\[ A^\text{advection}_e = \int \frac{d(V_s \alpha)}{ds} \cdot W \cdot d(\Omega) \] (16)

Derivative can be calculated using following difference:

\[ \frac{d(V_s \alpha)}{ds} = \frac{(V_s \alpha)_U - (V_s \alpha)_D}{\Delta s} \] (17)

Diffusion term is discretized after integration of related term over the sub domain after multiplying to the weight function:

\[ A^\text{diffusion}_e = \int W \cdot \frac{1}{\rho} \cdot \frac{\partial}{\partial x}(\Gamma_\alpha \cdot \frac{\partial \alpha}{\partial x}) \cdot d(\Omega) + \int W \cdot \frac{1}{\rho} \cdot \frac{\partial}{\partial y}(\Gamma_\alpha \cdot \frac{\partial \alpha}{\partial y}) \cdot d(\Omega) + \int W \cdot \frac{1}{\rho} \cdot \frac{\partial}{\partial z}(\Gamma_\alpha \cdot \frac{\partial \alpha}{\partial z}) \cdot d(\Omega) \] (18)
After integrating by partially and considering that:
\[
\frac{\partial \alpha}{\partial x} = W^e_x \alpha
\]
\[
W^e_x = \frac{\partial W^e_x}{\partial x}
\]
Diffusion term can be expressed as:
\[
\left[ \frac{\alpha_{\text{diffusion}}^e}{\partial x} \right] = \left[ \left( \frac{\partial^2 \alpha}{\partial x^2} + \frac{\partial^2 \alpha}{\partial y^2} + \frac{\partial^2 \alpha}{\partial z^2} \right) \right] (\Omega)
\]
In order to calculate source term, it must be multiplied to the weight function and then integrated over the sub domain as:
\[
S^e_{\alpha} = \int W^e_{\alpha} S_{\alpha} d(\Omega)
\]

4. Application of the Model:
One of under numerical experiment, Dry-Cooling tower with height of 130 m, base diameter of 112 m, outlet diameter 65 m, height of vertical heat exchangers 20m, is considered for present simulation (Fig.1). According to VGB 2005 Dry-Cooling towers should remain stable during hurricanes with the speed of 41.2(m/s). For air the density \( \rho = 1.205(kg/m^3) \), viscosity \( \mu = 1.8*10^{-5} \) base temperature 300\( ^\circ \)K is considered. So Reynolds number for flow around the tower is computed as:
\[
Re = \frac{\rho V_{\text{ref}} d}{\mu} = \frac{1.205(kg/m^3) * 41.2(m/s) * 87.3(m)}{1.8*10^{-5} (kg/m/s)} = 24*10^5
\]
The performance of this flow-solver is examined by solving turbulence flow outside tower in the SHAZAND (Arak) power station located in north part of IRAN. In this work, No-slip and no-penetration boundary conditions are imposed at the solid walls. At inlet, the incoming fluid has a uniform temperature, and a fully developed laminar flow profile or a uniform velocity is prescribed. At the downstream end of the computational domain, the non-dimensional pressure there is set to 101.3KPa.

5. Discussion on the Results:
In this study, a Finite Element model is used to investigate the interaction between temperatures induced airflow and wind effect inside and outside the natural draught Dry-Cooling towers. The results of numerical modeling investigations indicate that the windy condition may improve the heat exchange with the airflow at the wind front of the Dry-Cooling tower, while the heat...
exchange condition in down stream parts inside the Dry-Cooling tower may get worse. The results of numerical modeling investigations have been shown computed stream-line (Fig.4), distribution of temperature (Fig.5) and distribution of pressure (Fig.6) inside and outside of the natural draught Dry-Cooling towers in windy condition and without wind.

References


Figures:

Fig (1) : Schematic view of the boundary surfaces of computational around the -cooling tower, Using a symmetric plane (200*400*400 m)

Fig (2) : Unstructured mesh for computational domain (Left: General view Right: Close view)
Fig (3) : Velocity profile inlet of imposed upstream of the computational domain
At supercritical Reynolds number \( \text{Re} = 24 \times 10^7 \)

Fig (4): Computed stream-lines in the computational domain (Left: Without wind Right: With wind)

Fig (5) : Temperature contour in the computational domain (Left: Without wind Right: With wind)

Fig (6) : Pressure contour in the computational domain (Left: Without wind Right: With wind)