PERFORMANCE ANALYSIS OF NATURAL DRAFT WET COOLING TOWER AT OPTIMIZED INJECTION HEIGHT

[1]ALOK SINGH, [2]SANJAY SONI, [3]R. S. RANA

[1]Assistant Professor, [2]Associate Professor, [3]Mechanical Engineering Department
Maulana Azad National Institute of Technology, Bhopal, (M.P.)
E-mail: er_aloksingh@rediffmail.com

Abstract- Cooling tower is an integral part of thermal power generation plant. Basically cooling tower are heat rejection devices used to transfer waste heat to the atmosphere. Investigation involves the two-dimensional computational fluid dynamics model based on actual reference conditions. Temperature and humidity inside the tower are having main influence on the performance of natural draft cooling tower. At optimum injection height tower cooling range (difference of temperature between hot water inlet and cold water outlet) is about 1.1k, which sufficiently capable of increasing the effectiveness of cooling tower. This leads to increase of effectiveness by 12.15%, which results in improved efficiency of power plants. The resulting optimal injection height reduces the relative humidity by 8.5% at the outer radius of tower.

Keywords- Natural draft wet cooling tower, CFD.

I. INTRODUCTION

Cooling towers are main part of various industrial process as well as thermal power plants. They are often used in power generation plants to cool the condenser feed-water [1]. Here, the cooling tower uses ambient air to cool warm water coming from the condenser in a secondary cycle. There are many cooling tower designs or configurations. In dry cooling tower water is passed through finned tubes forming a heat exchanger so only sensible heat is transfered to the air. In wet cooling tower the water is sprayed directly into the air so evaporation occurs and both latent heat and sensible heat are exchanged. In hybrid tower a combination of both approaches is used. Cooling towers can further be categorised into forced or natural draft towers. Forced units tend to be relatively small structures where the air flow is driven by fans. In a natural draft cooling tower the air flow is generated by natural convection only. The draft is established by the density difference between the warm air inside the tower and the cool dense ambient air outside the tower. In a wet cooling tower, the water vapor inside the tower contributes to the buoyancy and tower draft. A further classification is between counter-flow and cross-flow cooling towers. In cross-flow configuration, the air flows at some angle to water flow, where as in counter-flow the air flows in the opposite direction to water flow [1,2].

II. NATURAL DRAFT WET COOLING TOWER

This paper is concerned with natural draft wet cooling towers (NDWCT) in counter-flow configuration. These structures are most commonly found in power generation plants. The main components of natural draft wet cooling tower are nozzle, fill, drift elimenator and water basin. The warm water is sprayed a grid of nozzle over the packing. The packing or fill is a multi-layered lattice with large specific air to water contact surface, which obstructs the free fall of water, thereby extending the heat and mass transfer time [3]. The fill breaks up water flow into droplets, increases the contact area and contact time with air, and therefore improves the heat transfer rate and efficiency of cooling tower.

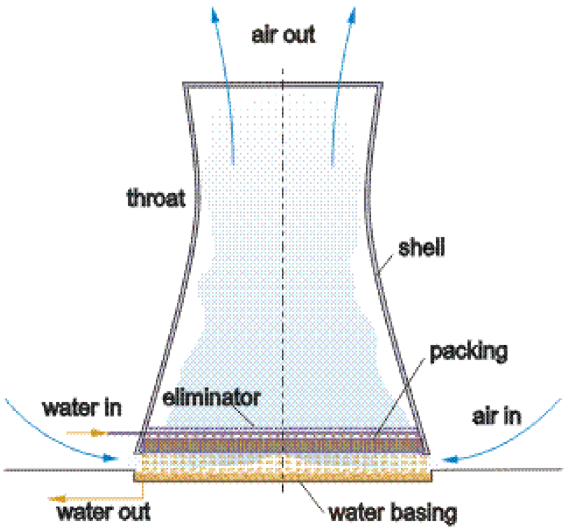

Figure :1 Natural draft wet cooling tower [1]

The air enters the tower horizontally through the rain zone where it initially flows in a partly counter flow and partly cross flow manner before being drawn axially into the fill and up into the tower. The air leaving the fill is generally supersaturated.

III. MODELING OF COOLING TOWER

In order to analysis the different parameter of cooling tower first the 2-D computational fluid dyamics model is develoded as actual reference condition of panipat power generation plat. The CFD code “FLUNT 12” is used for modeling of tower.

IV. GEOMETRY

Initially 2-D, CFD model of natural draft cooling tower is created considering important details. The structure of cooling tower model imagined in advance due to possibilities in the subsequent steps depended on the composition of different geometrical shapes. Some assumption were made to take into account the main features of real construction of cooling tower [4,7].

- 2-D symmetric model is developed; fixing the fill corresponding to real arrangement.
- Inlet and outlet space is created at bottom and top of the tower
- Cooling tower shell is considered as a wall with zero thickness and its profile is formed by curve with three point including throat.
- Assuming symmetrical thermal and flow field in the model, only one half of the cooling tower is modeled with a symmetry boundary condition.
- The effect of cooling water piping is modeled by porous zone boundary condition with appropriate pressure loss coefficient in the air flow.
- The outlet of the peak cooler cells is created with rectangular cross-section in the model without the transition piece to circular cross-section. The fans were modeled by the fan model of FLUENT 12 at the exit planes.

Reference conditions

- Tower height 130 m
- Air inlet height 10 m
- Fill depth 1 m
- Tower basin diameter 98 m
- Fill base diameter 95 m
- Tower top diameter 68 m
- Spray zone height 12 m
- Water flow rate 10,000 lit/s
- Water inlet temperature 311 K
- Ambient air temperature 299 K
- Ambient air humidity 55 %
- Ambient pressure 101 kpa

After geometry, mesh is generated. During mesh generation much attention is to be paid with mesh quality requirement recommendation in FLUENT 12.

In order to have an appropriate resolution of the flow field inside the cooling tower the computational domain is define into a large number of finite volume cells.

- Different parts are meshed with different element sizing.
- Fill zone are fine meshed.
- By using mapped face meshing the model with appropriate element sizing is created.
- After mesh generation naming of different parts of cooling tower is done.

The inner and outer surface of the wall inside the model, have identical shapes, so the mesh sizes on the two sides of the walls can be same.

In order to have an appropriate resolution of the flow field in the vicinity of and inside the cooling tower, the computational domain was discretised into a large number of finite volume cells. Detail from the mesh is illustrated from bottom region of the cooling tower quadrilateral face mesh elements.

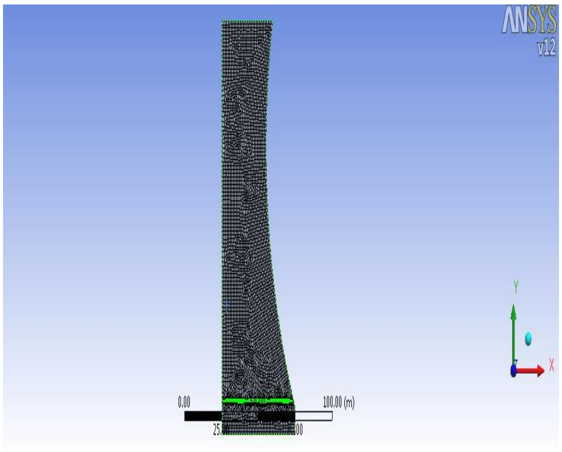

Figure:2 Half Meshed view of cooing tower

V. BOUNDARY CONDITIONS

The operating condition is, at a point 130 m upstream from the centre line of the cooling tower at ground level and acceleration due to gravity is specified as 9.81 m/s^2. For this approach an operating temperature of 288.16 K and an operating density of 1.2 kg/m were entered. At walls zero heat flux boundary condition is applied (adiabatic walls). For momentum equation no slip shear condition is prescribed and a wall roughness height is specified. In FLUENT 12 an equivalent sand grain roughness height should be used with the default roughness constant of 0.5. When determining the equivalent sand-grain roughness height for the physical roughness height of different walls, recommendations in literature are applied. Velocity inlet boundary condition is used to define the inlet velocity and other properties of air. Velocity magnitude of air takes normal to the boundary of inlet [10]. Turbulence is taken as intensity and length scale. Thermal condition and species in mole fraction is defined. Outlet is defined as pressure out-let of air. Other zones are also defined likewise [5,8].

VI. GOVERNING EQUATIONS

The numerical model has been built within FLUENT 12. This study examines only the flow in the tower under constant wind conditions.

The governing equations for incompressible steady fluid flow can be written as:

$$\nabla \cdot (\rho u \emptyset - \Gamma_{\emptyset} \nabla \emptyset) = S_{\emptyset} \quad (1)$$

where ρ is the air density (kg/m^3), u is the fluid velocity (m/s), ∅ is the flow variable and $\Gamma_{\emptyset}$ is the diffusion coefficient for ∅ and $S_{\emptyset}$ the source term. These equations can be expanded into the individual momentum and transport equations which, together with the continuity equation give the Navier-Stokes Equations.

The continuity equation for conservation of mass in Cartesian coordinates for transient flow can be given as,

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho v) = S_m \quad (2)$$

where S_m is the mass source term. The steady equation is obtained by simply neglecting the transient terms ∂/∂t, from the left hand side.

The equation for conservation of momentum can be written as,

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}\rho u_i u_j = -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] + S \quad (3)$$

where S is now a source term for momentum. The source term for buoyancy can be written as,

$$S_{b=(\rho-\rho_{ref})g} \quad (4)$$

The transport equation for a scalar ∅ can be written as:

$$\frac{\partial}{\partial t}(\rho \emptyset) + \frac{\partial}{\partial x_j}(\rho \emptyset u_j) = \frac{\partial}{\partial x_j}\left[\rho \Gamma \left(\frac{\partial \emptyset}{\partial x_j}\right)\right] + S_{\emptyset} \quad (5)$$

The cooling tower geometry and specifications are based on a NDWCT located at Panipat Power station, Panipat (Haryana). This is a coal fired power plant operating with 2x210MW Units. Each unit is cooled by a NDWCT. The two NDWCTs at this site have a history of underperforming, primarily due to wind effects. The plant operators have been actively involved in a research program to improve the performance of the NDWCTS. The towers at this site were chosen as the basis for this study as they are both of a typical NDWCT design and the design and operating data were readily available. The design parameters are given with the reference conditions used in this study.

VII. RESULT AND DISCUSSION

The actual height of injection is 12 m (1m from fill top) from the water basin, is taken as a base data to be optimized by increasing and decreasing the injection height with 0.25 m interval, with the use of ANSYS software tool FLUENT 12 we make the model of different injection height (12.25m, 12.0m, 11.75m, 11.5m) to optimize the injection height and analyze the effect of injection height on the performance of natural draft wet cooling tower taking all the parameter as constant for same structure of tower, finally we get the optimum injection height is 11.75m.

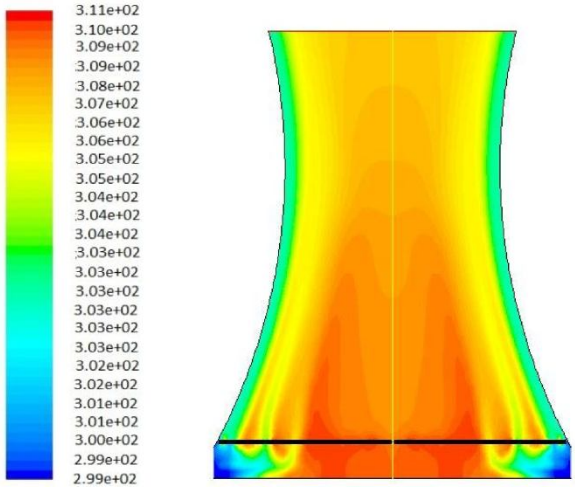

Figure:3 Contour of Temperature (k)

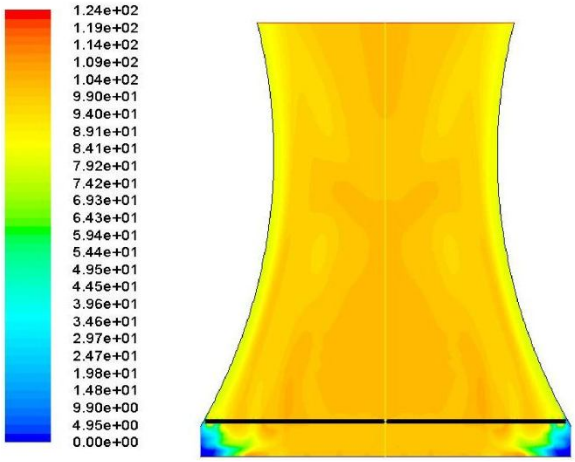

Figure:4 Contour of Relative Humidity (%)

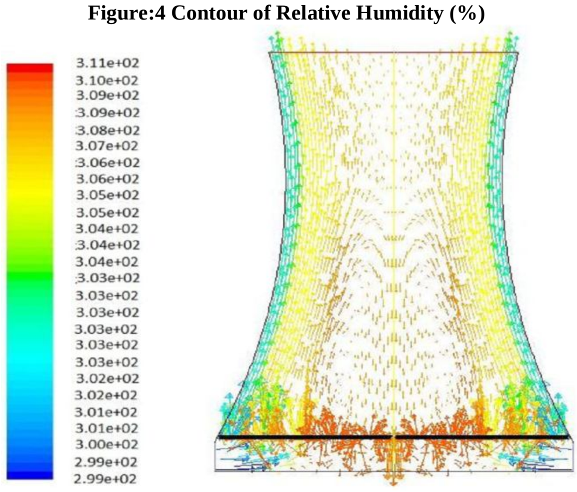

Figure:5 Velocity Vector of Temperature (k)

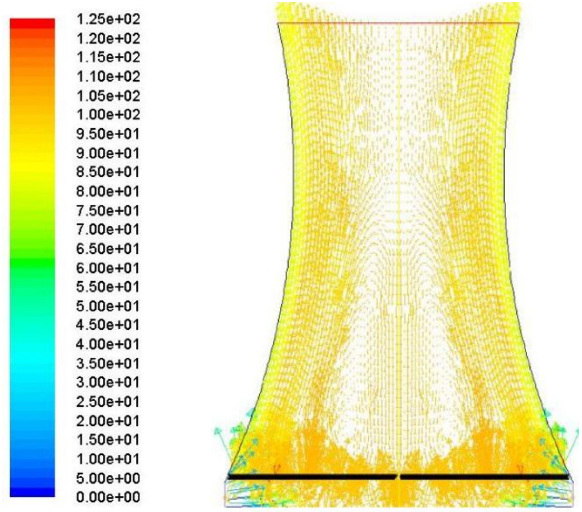

Figure:6 Velocity Vector of Relative Humidity (%)

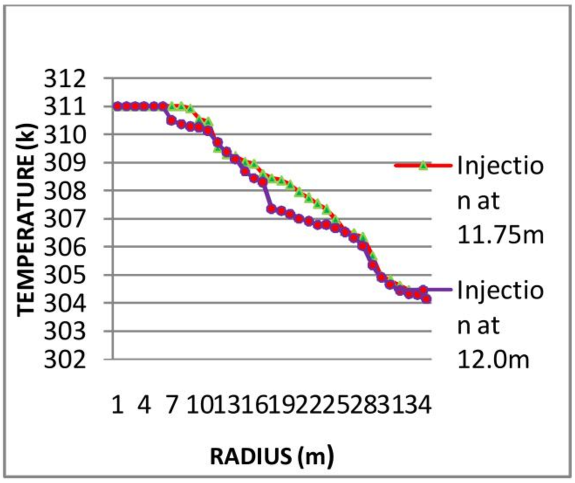

Figure :7 Comparison between temperature and radius at outlet

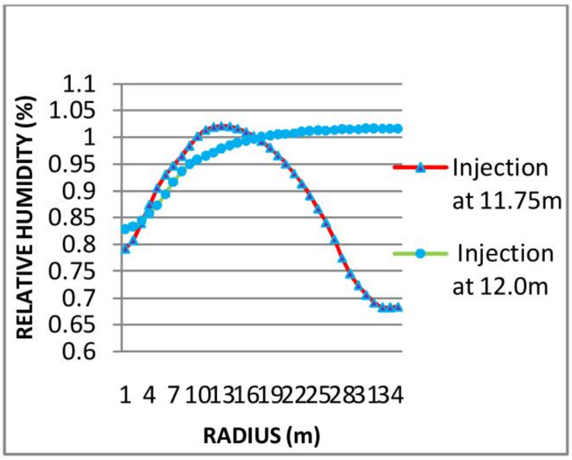

Figure:8 Comparison between relative humidity and radius at outlet.

In the contour of temperature of natural draft wet cooling tower at inlet of tower the temperature of cold ambient air is 299k., when it comes in contact with hot water in the rain zone suddenly temperature of air increases. Near the axis of tower the temperature of hot air and water particle remain high due to choking of air around axis. The highest heat transfer takes place in fill zone and the temperature of air becomes high. As hot air crosses the spray zone it starts go up due to pressure difference inside the tower and out- side ambient air. Near the wall portion its temperature becomes about 303K due to high density of hot air. At outlet of tower the average temperature of hot humid air about 308.2K and the relative humidity becomes 88.5% due to fine suspended water particles.

The result shows that near wall relative humidity is low because of low temperature and high density. With optimum injection height, reduction of average relative humidity is achieved by 8.5% at the outlet of tower, which results in reduction of make-up water added to water basin.

CONCLUSION

An analysis was conducted for the influence of injection height with key design and constant operating parameters, the fill depth, tower inlet height, water flow rate, ambient air temperature and humidity and the initial water droplet diameter and distribution in the rain zone. In particular, the radial uniformity of heat transfer and air flow due to inside geometric effects and overall gradients in air temperature and air humidity and flow rate are examined. The results show that with the exception of a small inlet affected region, the air flow is quite uniform through the fill and spray zones under the range of parameters considered in this analysis. A part of the objective of this study is to provide designers with insights into the flow within a typical NDWCT and how cooling may be improved with respect to optimum injection height. By demonstrated that decreasing the nozzle height by 0.75 m instead of 1m from fill top significantly reduces the average moisture content of hot air leaving the tower. Increased effectiveness and decreased cold water temperature by 1.1K generates 6-8MW more power towards rated production of electricity [12]. Average moisture content leaving the tower reduced by 8% in case of optimum injection height.

REFERENCES

[1] Alok singh, S.P.S. Rajput, " Application of CFD in Natural Draft Wet Cooling Tower Flow", International Journal of Engineering Research and Applications Vol. 2, Issue 1,Jan-Feb 2012,

[2] Alok singh, S.P.S. Rajput,"Thermal Performance Evolution of 2-D CFD Model of Natural Draft Wet Cooling Tower", International Journal of Scientific and Engineering Research, Volume 3, Issue 3, March 2012,

[3] Behnia, M. and Al-Waked,R.," The effect of windbreak walls effect on thermal performance of natural draft dry cooling tower". Heat Transfer Engineering, 26(8):p.50–62, [2005].

[4] Williamson, N., Armfield, S., Behnia M.", Numerical simulation of flow in a natural draft wet cooling tower –The

effect of radial thermofluid fields" Applied Thermal Engineering, vol. 28, no. 2-3, p.178-189. [2008].

[5] Cooper J., Grindle E., Lawson R." Improving Natural Draft Cooling Tower Performance With Heat Injection", International Joint Power Generation Conference, Phoenix, AZ, USA, [2010].

[6] Heinrich claude storm", CFD investigation of flow in and around a natural draft cooling tower", Msc. Engg. University of stellenbosh, south Africa, [2010].

[7] Williamson, N., "Numerical modeling of heat and mass transfer and optimization of a natural draft wet cooling tower" PhD. Thesis, The University of Sydney, [2008].

[8] Kloppers J.C. and Kroger D.G.," A critical cooling tower performance evaluation "In 12th IARH Symposium in Cooling Tower and Heat Exchangers, UTS, Sydney, Australia, 11-14 November [2001].

[9] Viljoen D.," Evaluation and Performance Prediction of Cooling Tower Spray Zone"s, MSc.Engg. Thesis, Department of Mechanical Engineering, Stellenbosch, University of Stellenbosch, South Africa, [2006].

[10] Kapas Nimrod", Wind effects on natural draught dry cooling towe"r, PhD. Department of energy engineering, Budapest University of technology and economics, [2005].

[11] Al-Waked R.,"Development of Performance-Improving Structures for Power Station Cooling Tower",. PhD thesis, University of New South Wales, Sydney, Australia,[2005].

[12] Samali B., and Madadnia J.," Wind simulation in an environmental wind tunnel for both structure and performance studies" 12th IAHR Symposium in cooling tower and heat exchanger page M-105, p. 44-52, Australia, 11-14 November [2001].

★★★