Flow Analysis Of Cooling Tower Using Cfd

[1]Ch.Indira Priyadarsini, [2]A.Akhil[2], [3]V.Lakshmi Shilpa
[1]Faculty, Mechanical Engg.Dept., Chaitanya Bharathi Institute of Technology, Hydearabad
[2]M.E Student, Mechanical Engg.Dept., Chaitanya Bharathi Institute of Technology, Hydearabad
[3]M.E Student, Mechanical Engg.Dept., Chaitanya Bharathi Institute of Technology, Hydearabad

Abstract : This paper deals with a natural draft wet-cooling tower with various inlet conditions. A commercial code FLUENT has been used to simulate the transport phenomena inside the tower. In this a 50 tons cooling capacity model has been taken as reference model. The developed model is analyzed with two air flow rates in vertical direction and by combining air inlet temperature and water inlet temperature. The developed model has been updated and the height of the water inlet is increased from the basin height and the same analysis is done by using the two flowrates of air and water into the system. It is observed that the temperature and humidity inside the tower are the main influence factors on the performance of cooling tower. It is also observed that due to increase in height to an optimum condition the performance of the cooling tower is increased and further increase in the height decreased its performance. Analysis shown that due to temperature of fluid inlet, cooling capacity of the tower has been improved with increase in air airflow rate when compared with natural aspirated air. A turbulence model of k-ε with energy equation is used for simulation

***IndexTerms* – Cooling tower, CFD, FLUENT**

I. INTRODUCTION

Cooling tower operation is based on evaporative cooling as well as exchange of sensible heat. During evaporative cooling in a cooling tower, a small quantity of the water that is being cooled is evaporated in a moving stream of air to cool the rest of the water. Also when warm water comes in contact with cooler air, there is sensible heat transfer whereby the water is cooled. The major quantity heat transfer to the air is through evaporative cooling while only about 25% of the heat transfer is through sensible heat. Figure 1. taken from Mulyandasari [4] shows the schematic of a cooling tower. In 1925, Merkel [7] was one of the first to propose a theory to quantify thecomplex heat transfer phenomena in a counterflow cooling tower. Merkel made severalsimplifying assumptions so that the relationships governing a counterflow cooling towercould be solved much more easily. Benton [2] and Kloppers and Kroger [8] list theassumptions of the Merkel theory as follows:The saturated air film is at the temperature of bulk water, The saturated air film offers no resistance to heat transfer, The vapor content of the air is proportional to the partial pressure of water vapor, The force driving heat transfer is the differential enthalpy between saturated andbulk air. Baker and Shryock[9] give a detailed explanation of the procedure of arriving at the final equations of theMerkel theory and also list some of the shortcomings of the Merkel theory and suggestsome corrections. Bourillot [10] developed a program called TEFERI to predict theperformance of an evaporative cooling tower in 1983. Benton [11] developed the FACTSmodel in 1983 and compared it to test data. Benton [2] states that the FACTS model iswidely used by the utilities to model cooling tower performance. Majumdar [12] reviewed the then existing methods of cooling tower performance evaluation and developed a new mathematical model that is embodied in a computer code called VERA2D. Majumdar [12] also gives a more detailed list of available mathematical models for analyzing wet cooling towers. In 1989 Jaber and Webb [13] developed equations to apply the ϵ–NTU method of heat exchanger design to design cooling towers. The Merkel method and ϵ–NTU method with modifications are the methods generally used to predict tower performance. Bergsten [14] states that the ϵ–NTU method (with some modifications) is used in well known and wide spread building simulation programs such as TRNSYS, Energy Plus and the ASHRAE Primary HVAC Toolkit package. Poppe and Roegener [15] came up with the Poppe model also known as the exact model in 1991 which does not make the simplifying assumptions of Merkel’s theory and is therefore more accurate. Kloppers and Kroeger [16] critically evaluate the Merkel theory by comparing it with the Poppe method. Kloppers and Kroger [8] give a detailed derivation of the Merkel, Poppe and Entu methods, their comparison and how to solve the governing equations in each of the methods. They conclude that the Poppe method is more accurate than the Merkel and ϵ– NTU methods and that the Merkel and ϵ–NTU methods give identical results since they are based on the same simplifying assumptions. With the advancement of computing power, computational Fluid Dynamics (CFD) models have been created to simulate performance of cooling towers [17].

II. THE COOLING TOWER

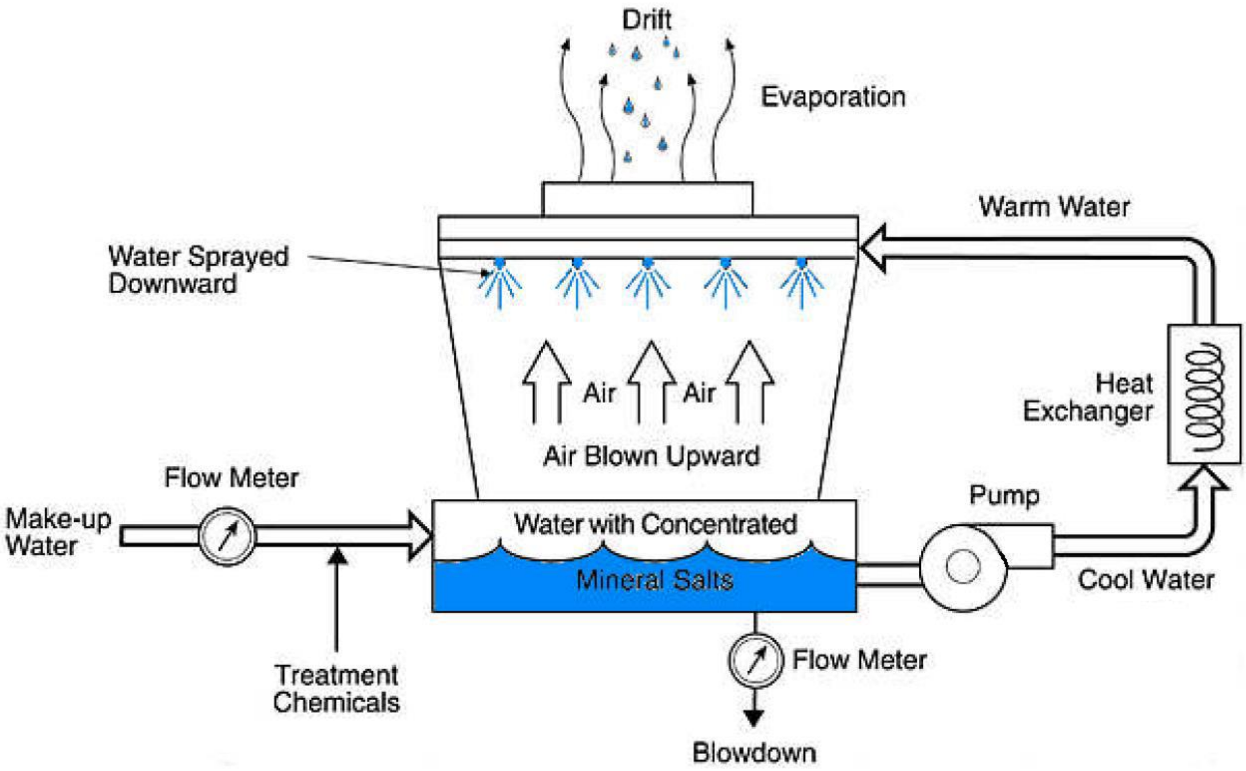


Figure 1 Schematic of a Typical Mechanical Draft Cooling tower

Fig. 1: Airfoil terminology

III. COMPUTATIONAL FLUID DYNAMICS

Governing Equations

The instantaneous equations of mass, momentum and energy conservation can be written as follows in a stationary frame:

- **The continuity Equation:**

$$\frac{\partial y}{\partial x} + \nabla.(\rho U) = 0$$

- **The Momentum Equations**

$$\frac{\partial(\rho U)}{\partial t} + \nabla.(\rho U \otimes U) = -\nabla p + \nabla.\tau + S_M$$

Where the stress tensor, τ, is related to the strain rate by

$$\tau = \mu\left(\nabla U + (\nabla U)^T - \frac{2}{3}\delta\nabla.U\right)$$

- **The Total Energy Equation**

$$\frac{\partial(\rho h_{tot})}{\partial t} - \frac{\partial p}{\partial t} + \nabla.(\rho U h_{tot}) = \nabla.(\lambda\,\nabla T) + \nabla.(U.\tau) + U.S_M + S_E$$

ANSYS Fluent provides comprehensive modeling capabilities for a wide range of incompressible andcompressible, laminar and turbulent fluid flow problems. Steady-state or transient analyses can beperformed.

IV. METHODOLOGY

Dimensions for modeling

• 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 10 m
- Water flow rate 0.055 kg/s; 0.099 kg/s
- Air flow rate 0.0404kg/s; 0.077 kg/s
- Water inlet temperature 329 K,320K
- Ambient air temperature 300 K
- Ambient air humidity 55 %
- Ambient pressure 101 kpa

Mesh:

After geometry, mesh is generated. During meshgeneration much attention is to be paid with meshquality requirement recommendation in FLUENT 18. In order to have an appropriate resolution of the flowfield inside the cooling tower the computationaldomain is define into a large number of finite volumecells.

- Different parts are meshed with differentelement 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 themodel, have identical shapes, so the mesh sizes on thetwo sides of the walls can be same. In order to have an appropriate resolution of the flowfield in the vicinity of and inside the cooling tower,the computational domain was discretised into a largenumber of finite volume cells. The generated mesh with 1386 nodes and 1300 elements is shown below with naming.

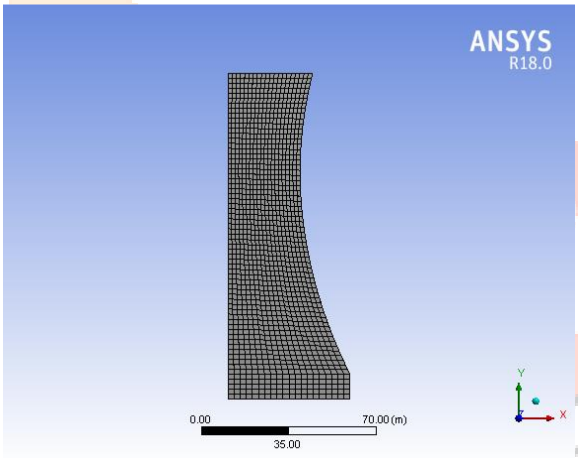


Figure 2: Mesh structure of the cooling tower

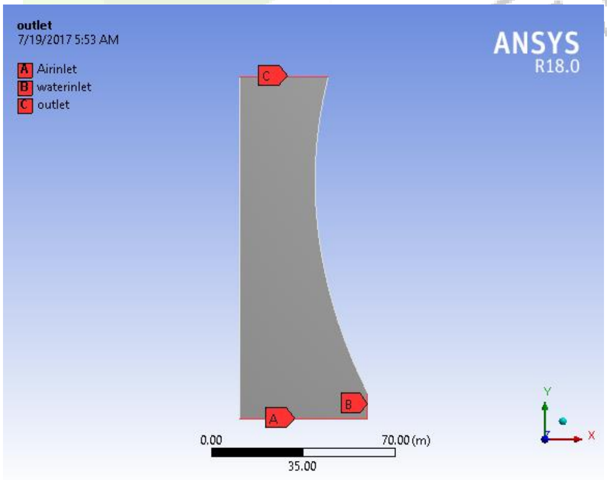


Figure 3: Names of the cooling tower

Cell zone Condition:

In cell zone surface body is considered as fluid. The operating pressure is 101325 Pa in upstream from the centerline of the cooling tower. The gravitational acceleration is 9.81 m/s2. Operating temperature is 288.16 K and operating density is 1.22 kg/m3 entered

Boundary Conditions:

The operating condition is, at a point 130 m upstreamfrom the centre line of the cooling tower at groundlevel and acceleration due to gravity is specified as9.81 m/s2. For this approach an operating temperatureof 288.16 K and an operating density of 1.2 kg/mwere entered. At walls zero heat flux boundarycondition is applied (adiabatic walls). For momentumequation no slip shear condition is prescribed and awall roughness height is specified. In FLUENT 12 anequivalent sand grain roughness height should beused with the default roughness constant of 0.5 When determining the equivalent sand-grainroughness height for the physical roughness height ofdifferent walls, recommendations in literature areapplied. Velocity inlet boundary condition is used todefine the inlet velocity and other properties of air. Velocity magnitude of air takes normal to theboundary of inlet [10]. Turbulence is taken asintensity and length scale. Thermal condition andspecies in mole fraction is defined. Outlet is definedas pressure out-let of air. Other zones are also definedlikewise [5,8].

V. RESULTS AND DISCUSSION

The generated geometry models are solved using the boundary conditions, the solution is initialized and the temperature derivatives, pressure, velocity, turbulence kinetic energy contours and velocity vectors are obtained after the solution convergence criteria get reached upto its minimum value. The analyzed cooling tower models are displayed below,

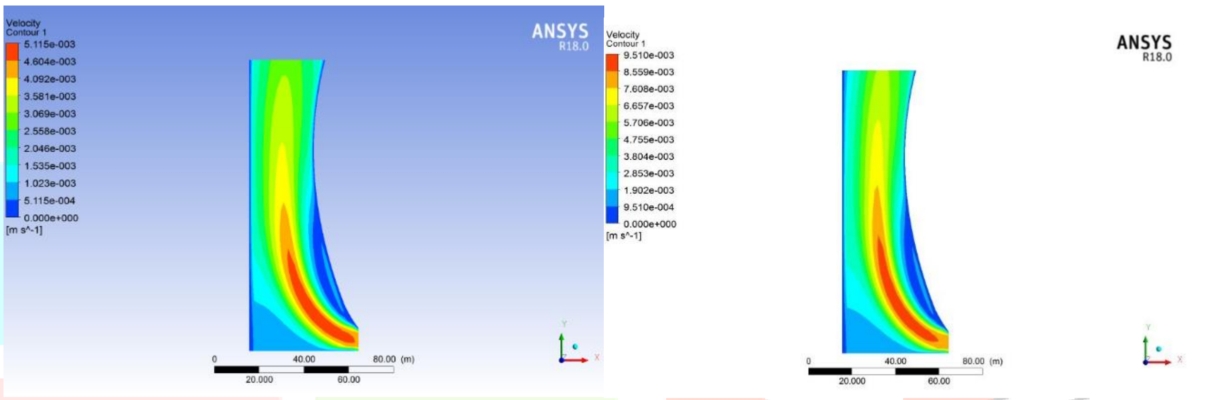
Considering the following conditions with its boundary conditions:

1.Water Inlet temperature (T1)-329K, Mass flowrate of water, Mw-0.055kg/s, Mass flowrate of air, Ma – 0.0404kg/s

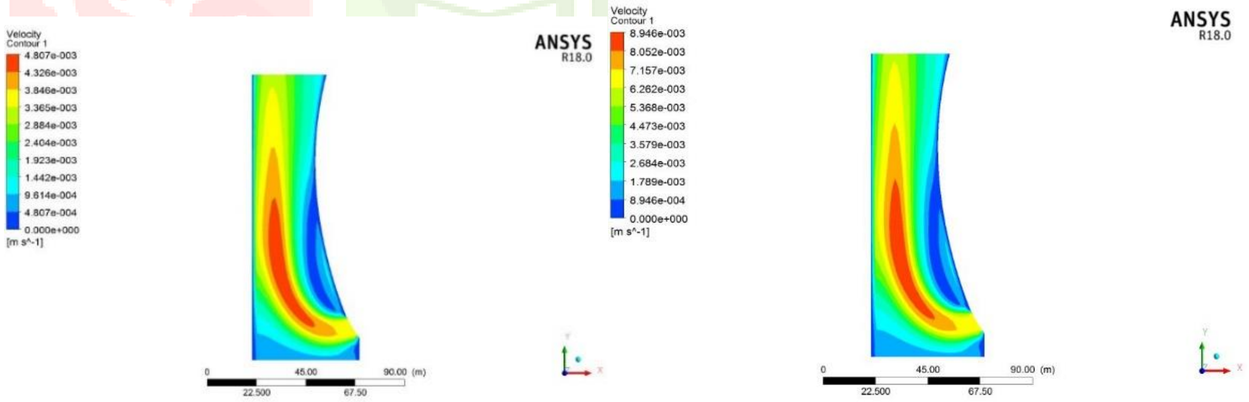
2.Water Inlet temperature (T1)-329K, Mass flowrate of water, Mw-0.099kg/s, Mass flowrate of air, Ma – 0.077kg/s

The two designs are solved using the above two boundary conditions. And the results were shown below:

Velocity



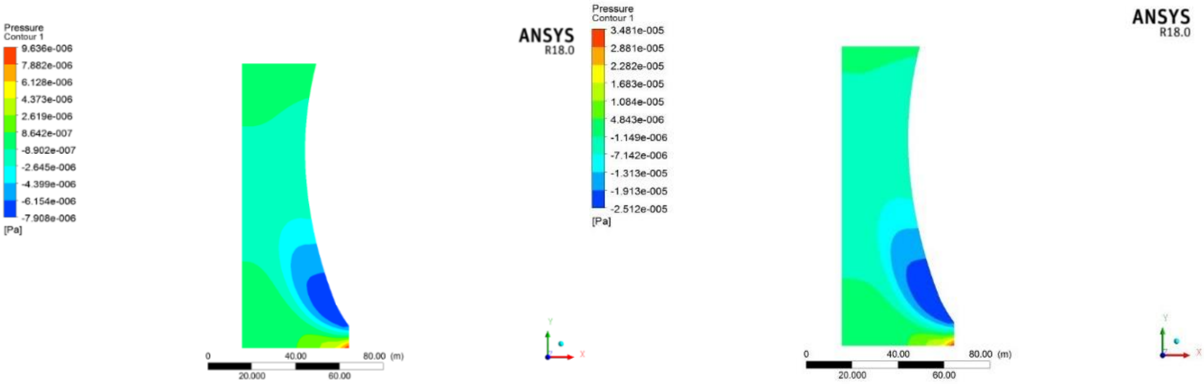
a) **Design 1**



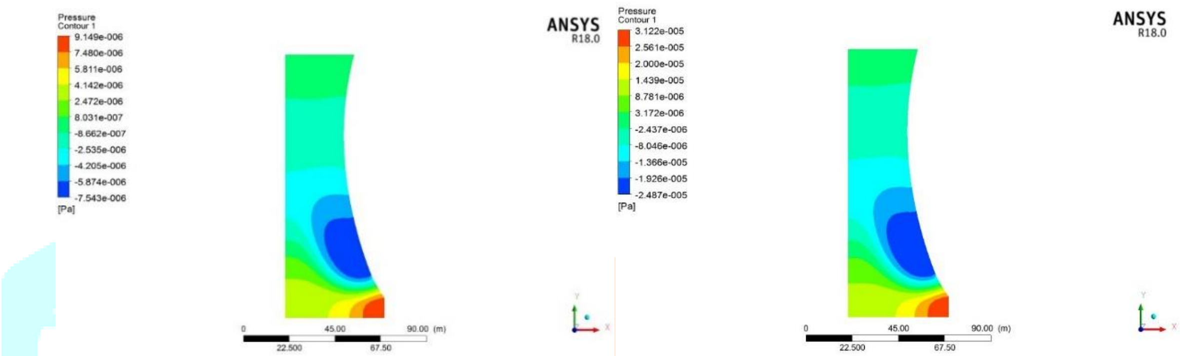
b) **Design 2**

From the above figure it is clearly stating that the velocity function within the cooling tower at low flow rates is lesser than the higher flowrates. It is also stated that in the second type of design consideration the velocity distribution within the tower is high and it is moved towards the center axis of tower and this is due to the water inlet position and also the velocity function is maximum at the throat section in second design consideration.

Pressure Contour



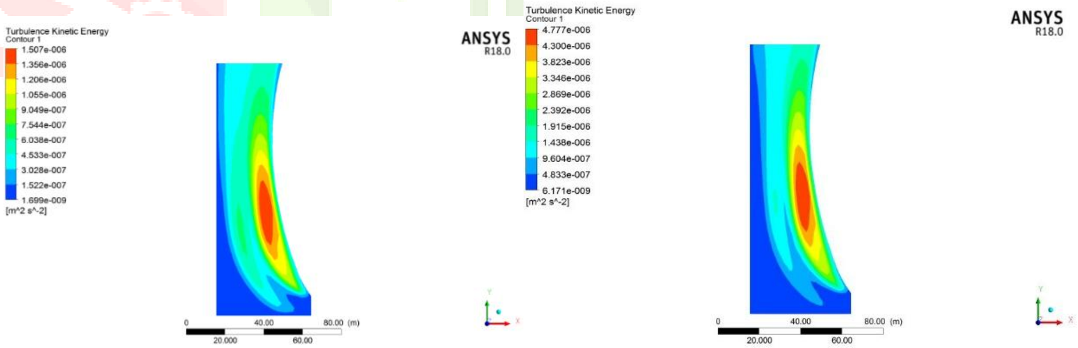
a) **Design 1**



b) **Design 2**

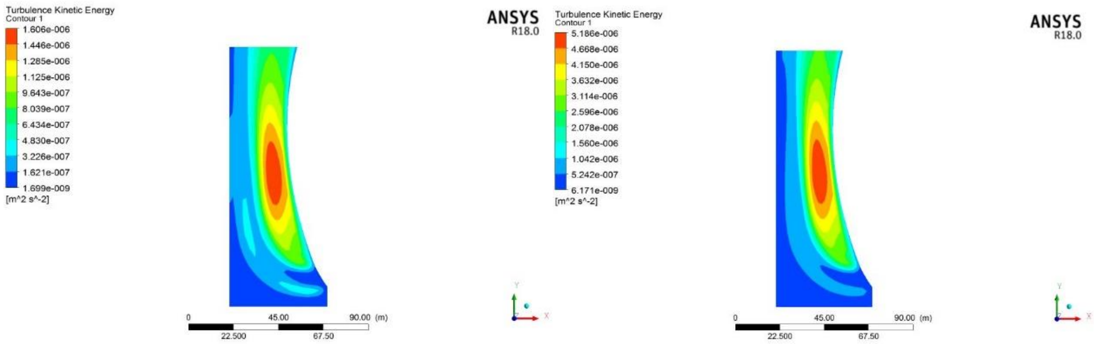
It is clearly shown that the max pressure is created at below the water inlet in the second design consideration. And in the first case as the water is directly in contact with the air which is at ambient temperature and there is no scope of creating maximum pressure. Which leads in the pressure difference and helps the hot air to flow out of the tower at a faster rate and providing maximum heat transfer rate.

Turbulence Kinetic energy contour



a) **Design 1**

The turbulence kinetic energy in the cooling tower is at the desired at second design consideration. The turbulence in other conditions is created at the wall before the throat section. Where as in the second condition it is created at near the throat where it causes the hot air to pass away from the cooling tower in all other cases the hot the air is distributed throughout the cross section.



b) Design 2

VI. CONCLUSIONS

The temperature of natural draft wet cooling tower at inlet of tower the temperature of cold ambient air is 300k., 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. Total pressure suddenly falls to fill area at 20 m than slight increases according to height. The thermal conductivity is very high at and near the axis because of high temperature and low density and very poor near wall. Density is high near wall and low near axis. Turbulence intensity changes very randomly up to the 20m that is fill zone and then takes its smooth values. Every thermodynamics characteristics changes after rain zone either increases or decreases. Temperature is having its high value in middle line and lower near the wall. Pressure decreases to the value from max to zero at fill zone area than increase slightly according to height. Highest value of thermal conductivity is near axis. Turbulent intensity increases up to rain zone than decreases, turbulent viscosity decreases to rain fill zone than increases. Stream function is linearly constant for axis and decreases according to height for middle line and line near wall. From the above results analyzed from the two design considerations at two varied flow rates and at a constant temperature, it is shown that the second design consideration is having the best output results.

REFERENCES

[1] Saidur, R., Abdelaziz, E., Hasanuzzaman, M., 2010, "A Study of Energy Efficiency, Economic and Environmental Benefits of a Cooling Tower," International Journal of Mechanical and Materials Engineering, 5(1) pp. 87-94.
[2] Benton, D. J., Bowman, C. F., Hydeman, M., 2002, "An Improved Cooling Tower Algorithm for the Cool Tools TM Simulation Model," ASHRAE Transactions, 108.
[3] Elovitz, K. M., 1994, "Can Your Plant Benefit from Free Cooling?" Plant Engineering, 48(5) pp. 76-78.
[4] Mulyandasari, V., 2011, "Cooling Tower Selection and Sizing (Engineering Design Guideline)," KLM Technology Group.
[5] Stanford, H.W., 2012, "HVAC Water Chillers and Cooling Towers: Fundamentals, Application, and Operation," CRC Press.
[6] Hill, G.B., Pring, E., and Osborn, P.D., 1990, "Cooling Towers: Principles and Practice," Butterworth-Heinemann.
[7] Merkel, F., 1925, "Verdunstungs kühlung," Verein Deutscher Ingenieure Verlag.
[8] Kloppers, J. C., and Kröger, D., 2005, "A Critical Investigation into the Heat and Mass Transfer Analysis of Counterflow Wet-Cooling Towers," International Journal of Heat and Mass Transfer, 48(3) pp. 765-777.
[9] Baker, D. R., and Shryock, H. A., 1961, "A Comprehensive Approach to the Analysis of Cooling Tower Performance," Journal of Heat Transfer, 83(3) pp. 339-349.
[10] Bourillot, C., 1983, "TEFERI: Numerical model for calculating the performance of an evaporative cooling tower," Electricite de France, 78-Chatou. Thermal Transfer and Aerodynamic Dept. [11] Benton, D.J., 1983, "A numerical simulation of heat transfer in evaporative cooling towers," Tennessee Valley Authority, WR28-1-900-110.
[12] Majumdar, A., Singhal, A., and Spalding, D., 1983, "Numerical Modeling of Wet Cooling towers—Part 1: Mathematical and Physical Models," Journal of Heat Transfer, 105(4) pp. 728-735.
[13] Jaber, H., and Webb, R., 1989, "Design of Cooling Towers by the EffectivenessNTU Method," Journal of Heat Transfer, 111(4) pp. 837-843.
[14] Bergsten, B., 2009, "Evaporative Cooling Tower and Chilled Beams. Design Aspects for Cooling in Office Buildings in Northern Europe." Institutionen För Energi OchMiljö, Installation steknik, Chalmers Tekniska Högskola, pp. 978-991.
[15] Poppe, M., and Rögener, H., 1991, "Berechnung Von Rückkühlwerken," VDI Wärmeatlas, Pp.Mi.
[16] Kloppers, J., and Kröger, D., 2004, "Cooling Tower Performance: A Critical Evaluation of the Merkel Assumptions," R&D Journal.
[17] Al-Waked, R., and Behnia, M., 2006, "CFD Simulation of Wet Cooling Towers," Applied Thermal Engineering, 26(4) pp. 382-395.