# Effect of Air Flow Rate on Performance of Natural Draft Wet Cooling Tower

[1]Dr.T.Ratna Reddy, [2]Ch.Indira Priyadarsini, [3]A Akhil

[1]*Associate professor, Mechanical Engg. Dept., CBIT, Gandipet, Hyderabad,*

[2]*Assistant professor, Mechanical Engg. Dept., CBIT, Gandipet, Hyderabad,*

[3]*M.E Student, Mechanical Engg. Dept., CBIT, Gandipet, Hyderabad*

**ABSTRACT**

*Cooling tower is an important heat exchanger that is to be used in power plants for efficient working of cycle. Main object of work is to determine the performance of a natural draft wet-cooling tower with various inlet conditions. A commercial tool FLUENT is used to simulate the transport phenomena inside the tower. 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 height of the water inlet is increased from the basin height and the same analysis is done by using the two flow rates 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. 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. Simulation 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.*

***Keywords— Cooling tower, CFD, Fluent, Heat exchanger***

## I. INTRODUCTION

Cooling tower operation is based on evaporative cooling as well as exchange of sensible heat, 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.

According to Hill [6] the factors influencing the performance of a cooling tower are:The cooling range, The approach, The ambient wet bulb temperature, The flow rate of water through the tower, The flow rate of air over the water, The ambient temperature, The type of fill in the tower and Total surface area of contact between water and air

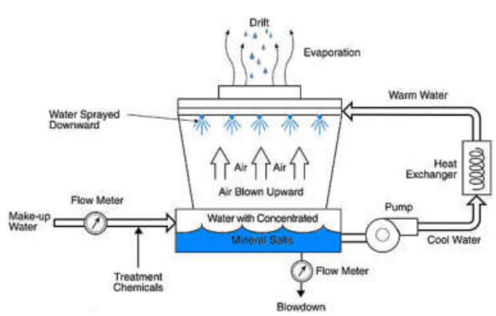



**Figure 1 Schematic of a Typical Mechanical Draft Cooling tower**

The Some important terms relating to cooling towers as described by Stanford [5] are:

- Approach- It is the difference between the temperature of water leaving the cooling tower and the wet bulb temperature. It is used as an indicator of how closeto wet bulb temperature the water exiting the tower is.
- Range- It is the difference between the temperature of water entering the towerand temperature of water leaving the tower.
- Capacity- The total amount of heat a cooling tower can reject at a given flow rate,approach and wet bulb temperature. It is generally measured in tons.
- Cell- It is the smallest tower subdivision that can operate independently. Eachindividual cell of a tower can have different water flow rate and air flow rate.
- Fill- The heat transfer media or surface designed to maximize the air and watersurface contact area.
- Make up water- The additional water that needs to be added to offset water lost toevaporation, drift, blowdown and other losses.
- Dry bulb temperature – It is the temperature of air measured by a thermometer freely exposed to the air but shielded from moisture and radiation. In generalwhen temperature is referred to, it is dry bulb temperature.
- Wet bulb temperature – It is the temperature of air measured by a thermometer whose bulb is moistened and exposed to air flow. It can also be said to be the adiabatic saturation temperature. The wet bulb temperature is always lesser than the dry bulb temperature other than the condition of 100% relative humidity whenthe two temperatures are equal.
- Free Cooling or Waterside Economizer Operation – It is the operation of the cooling tower in conditions where just the cooling tower is able to provide the required temperature cold water for HVAC or process needs without needing mechanical cooling from the chiller. This saves energy because while the chiller may utilize about 0.7 kW/ton, the tower is now able to provide the same coolingat about 0.2 kW/ton.

Figure 2 shows the configuration of the cooling tower for forced draft and induced draft fans as taken from Stanford [5].

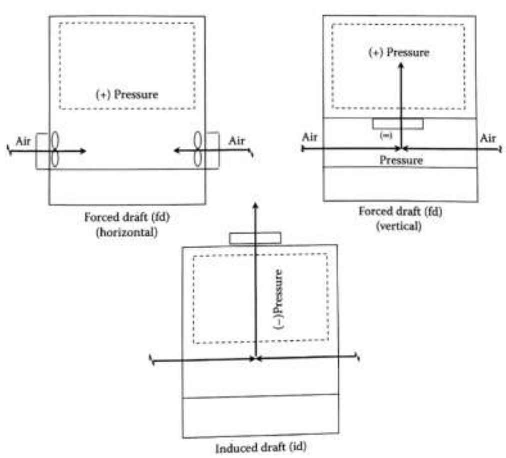



**Figure 2 Schematic of Forced Draft and Induced Draft Cooling Towers**

### 1.1 Problem Description and Objectives

The water is introduced into the tower through spray nozzles approximately 8-10m above the basin. The primary function of the spray zone is simply to distribute the water evenly across the tower. The water passes through a small spray zone as small fast moving droplets before entering the fill. There are a range of fill types. Generally they tend to be either a splash bar fill type or film fill type.The splash bar type acts to break up water flow into smaller droplets with splash bars or other means. A film fill is a more modern design which forces the water to flow in film over closely packed parallel plates. This significantly increases the surface area for heat and mass transfer.

Cooling tower works on the Principle of water evaporation. Based on rate of evaporation, the hot water could be cooled more effectively. The rate of evaporation of hot water by,

- Increasing time of contact of air with water
- Increasing the air velocity
- Increasing the area of contact of air and hot water.

In this project it is studied the temperature effect of hot water along with the inlet temperature of air at two water inlets with two flow rates.

## II. LITERATURE

Besides 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 and bulk air, The specific heat of the air water vapor mixture and heat of vaporization are constant, The loss of water by evaporation is neglected, The air exiting the tower is saturated with water vapor and is characterized only by its enthalpy. (This assumption regarding saturation has a negligible influenceabove ambient temp of 68°F but is of importance at lower temperatures), The Lewis factor relating heat and mass transfer is equal to 1. (This assumptionhas a small influence but affects results at low temperatures), This model has been widely applied because of its simplicity. 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.

## III THE MATHEMATICS OF CFD

The set of equations that describe the processes of momentum, heat and mass transfer are known asthe Navier-Stokes equations. These partial differential equations were derived in the early nineteenthcentury and have no known general analytical solution but can be discretized and solved numerically

Equations describing other processes, such as combustion, can also be solved in conjunction with theNavier-Stokes equations. Often, an approximating model is used to derive these additional equations,turbulence models being a particularly important example.

There are a number of different solution methods that are used in CFD codes. The most common, and the one on which FLUENT,CFX is based, is known as the finite volume technique.

In this technique, the region of interest is divided into small sub-regions, called control volumes. The equations are discretized and solved iteratively for each control volume. As a result, an approximation of the value of each variable at specific points throughout the domain can be obtained. In this way, one derives a full picture of the behaviour of the flow.

### 3.1 Governing Equations of CFD

The cornerstone of computational fluid dynamics is the fundamental governing equations of fluid dynamics—the continuity, momentum and energy equations.

These equations speak physics. They are the mathematical statements of three fundamental physical principles upon which all of fluid dynamics is based:

(1) mass is conserved;

(2) F = ma (Newton's second law);

(3) energy is conserved.

The purpose of this chapter is to derive and discuss these equations.

The purpose of taking the time and space to derive the governing equations of fluid dynamics

### 3.2 Modelling of Cooling Tower

Computation Fluid Dynamic Modeling: In order to analysis different conception of cooling tower behavior in wind first the computational fluid dynamic modeling of cooling tower is developed. The CFD code "FLUENT 18" is used for modeling. This package has been employed in this study to develop a two dimensional steady state simulation of NDWCT.

**Geometry:**

In the first step geometry is created in 2 D using reference data (4) providing different parts of cooling tower considering important details. The structure of whole model imagined in advance, because the possibilities in the subsequent steps depended on the composition of different geometrical shapes .Assumptions were made to take into account the main features of real construction.

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 18 at the exit planes.

**3.3 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 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

**3.4 Mesh**

After geometry, mesh is generated. During mesh generation much attention is to be paid with mesh quality requirement recommendation in FLUENT 18.

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 thetwo 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. The generated mesh with 1386 nodes and 1300 elements is shown below with naming.

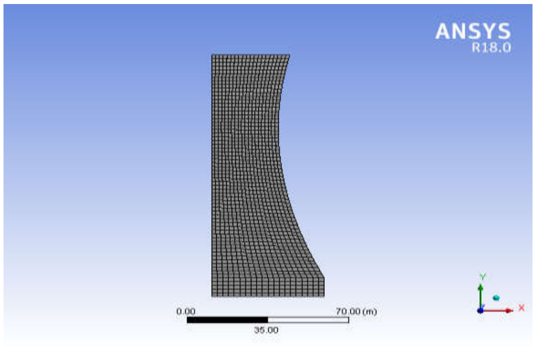



Figure 3 Mesh structure of the cooling tower

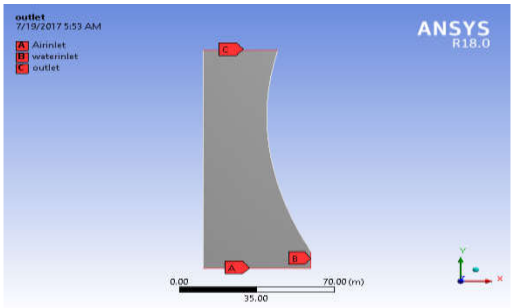



Figure 4 Names of the cooling tower design 1

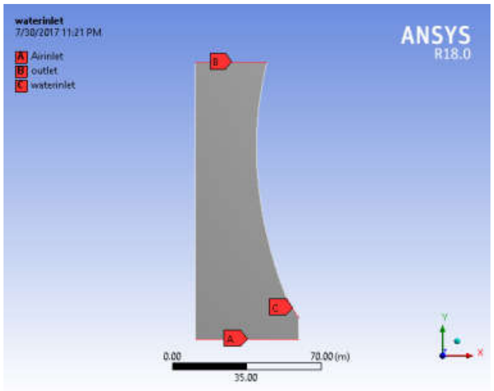



Figure 5 Names of the cooling tower design 2

**3.5 Cell zone/Boundary 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

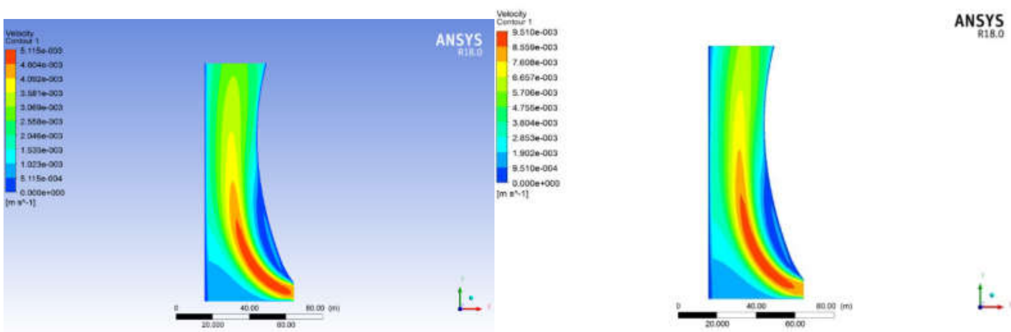
**Boundary Conditions:**

The operating condition is, at a point 130 m upstream from the centre line of the cooling tower at ground level and acceleration due to gravity is specified as9.81 m/s2. For this approach an operating temperature of 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 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]. Considering the boundary conditions as water Inlet temperature (T1)-329K, mass flowrate of water, Mw-0.055kg/s, mass flowrate of air, Ma – 0.0404kg/s, Water Inlet temperature (T1)-329K, Mass flowrate of water, Mw-0.099kg/s and Mass flowrate of air, Ma – 0.077kg/s

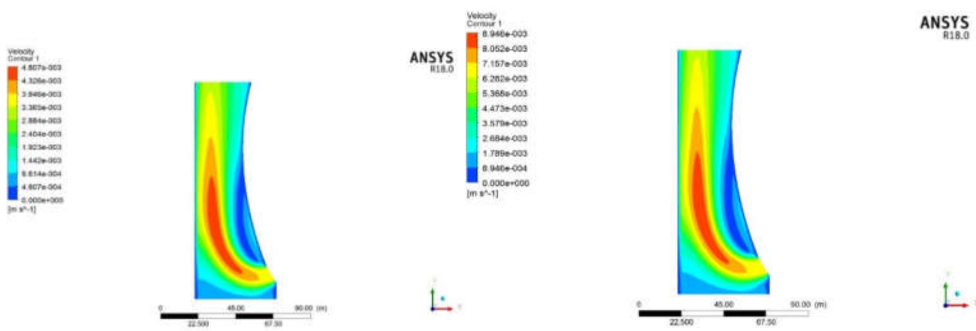
## IV RESULTS AND DISCUSSION

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.

### 4.1 Velocity Contour




**a) Design 1**

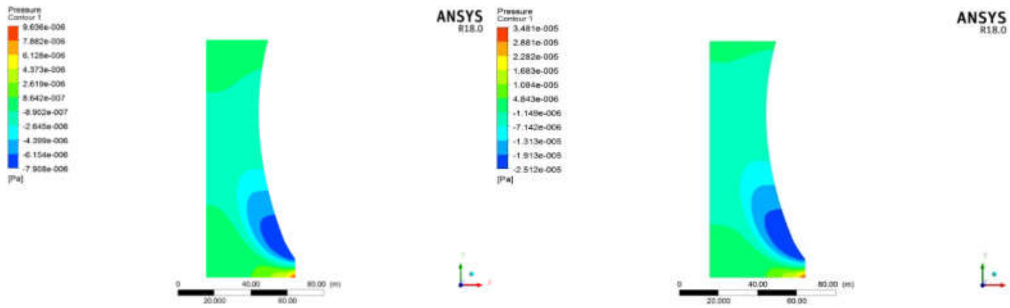



b) Design 2

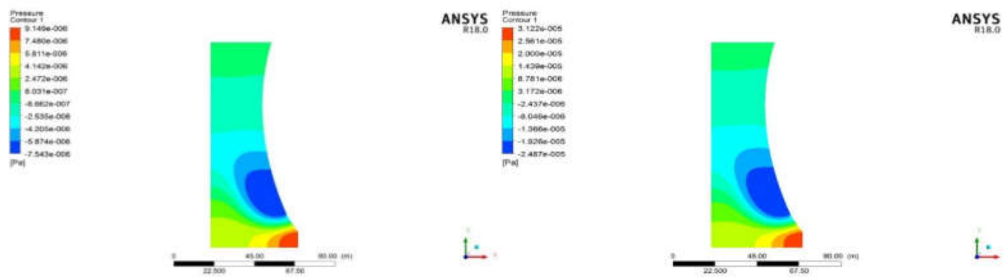
**Figure 6 Velocity contours**

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.

**4.2Pressure Contour**



a) Design 1

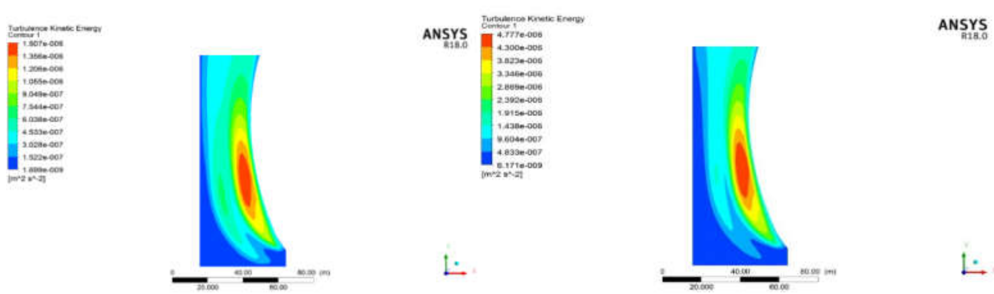


b) Design 2

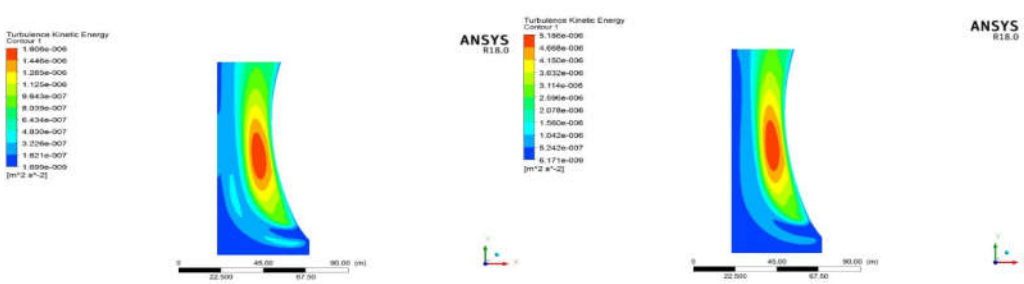
Figure 7 Pressure Contours

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.

**4.3 Turbulence Kinetic energy contour**



a) Design 1

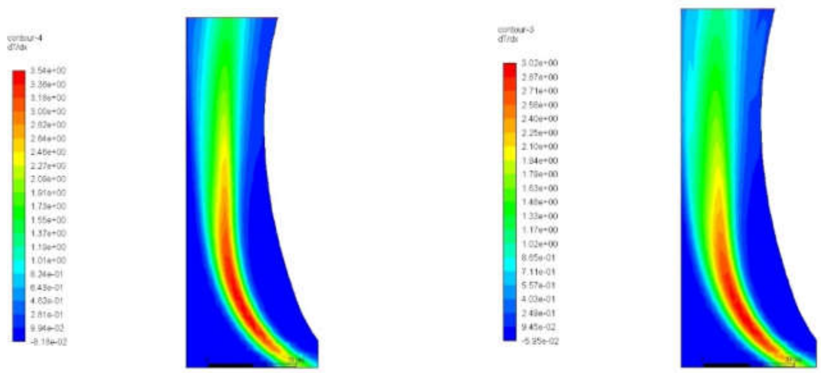


b) Design 2

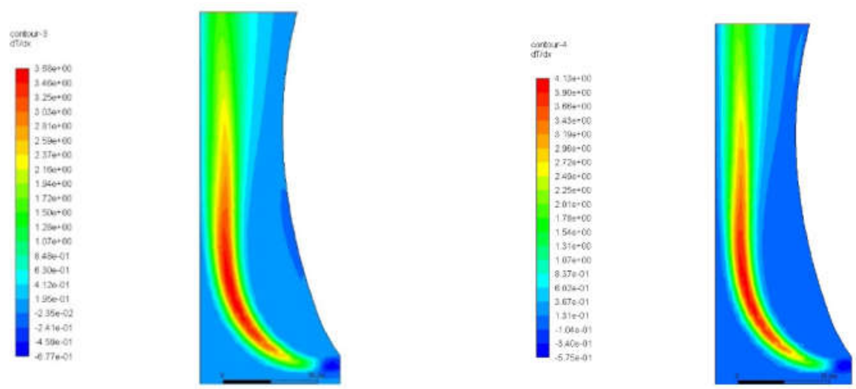
Figure 8 Turbulence kinetic energy contour

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.

**4.4 dT/dX derivative**



a) Design 1



a) Design 2

Figure 9 Temperature derivative contour

The temperature distribution within the cooling tower can be seen from the above contour plots. Where in the first design consideration the mixing of the temperature is carried away from the throat and the temperature is minimum in that area and later it is distributed throughout the surface as it reaches the outlet portion. In the second consideration it is shown that the least temperature is visible only at the air inlet portion and the temperature distribution is lot better when compared with the first design.

## V 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 ofair becomes high. As hot air crosses the spray zone itstarts go up due to pressure difference inside thetower and out- side ambient air. 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. By conducting 2 dimensional CFD simulation for investigating the thermal performance of wet cooling tower the overall changes take place in three zone. 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. The distribution of the temperature and the turbulence is developed properly and distributed within the cooling tower which leads the maximum heat transfer rate and also increase in the cooling capacity of the tower.

## 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 CoolToolsTM 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, "Verdunstungskühlung," VereinDeutscherIngenieureVerlag.

[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.