Performing Computational Fluid Dynamics (CFD) simulations to analyze the flow passing through a Cooling Tower

[1]Aman Raj, [2]Prof. Sachin Baraskar, [3]Dr. Rashmi Dwivedi
M. Tech Scholar[1], Assistant Professor[2], Head of Dept.[3]
Department of Mechanical Engineering
School of Engineering, SSSUTMS Sehore, M.P., India

ABSTRACT: - The current study involves conducting a CFD analysis on the flow through a cooling tower, with a specific focus on the rain zone. Throughout this thesis work, the mass flow rate of water, air inlet temperature, and water inlet temperature in the rain zone are maintained at constant values of 15000kg/s, 295K, and 303K, respectively. To facilitate analysis, three different geometries with varying rain zone heights of 8.577m, 6.777m, and 4.977m have been created. By utilizing the Rosin-Rammler distribution, the diameter of water droplets has been adjusted, allowing for the examination of temperature drop at different rain zone heights. Furthermore, the rate of temperature drop in relation to varying droplet diameter for each rain zone height has been compared.

Keywords:- Computational Fluid Dynamics (CFD), Cooling Power, Droplets, Temperature

I. INTRODUCTION

Evaporative coolers are air–water heat exchangers. They are commonly used in industrial processes and in Heating, Ventilation, Air Conditioning/Refrigeration (HVACR) systems in order to cool the water used as a refrigerant. Cooling occurs thanks to the heat absorbed by water droplets, which evaporate in the air stream. Cooling towers are the most common type of evaporative coolers. In these systems, the air flows vertically through the tower in counter-flow with the water spray. The major contribution to the heat transfer is given by the latent heat released by the evaporation of about 1 % of the water flow rate. Cooling towers differ in the way air is fed into the entire system. There are, therefore, *natural draft towers,* where the air flow is created by the difference in density of the air–water vapor mixture, and *mechanical draft towers*, where the air flow is moved by means of fans. The latter are classified according to the type of fan. There are *centrifugal towers* equipped with centrifugal fans located at the bottom of the tower and upstream of the filling, and *axial or induced-draft towers* equipped with an axial fan placed at the top of the tower and downstream of the filling. A second classification is made with reference to the contact between water and air which takes place during the heat exchange. In particular, in the *open-circuit towers*, the water from the industrial process is sprayed through nozzles over the filling and is in direct contact with the air; in the *closed-circuit towers,* the fluid is not in direct contact with the air stream [1]. In the latter case, it circulates inside smooth tubes and the outside of the tubes is cooled by the evaporation of water sprayed against the air moved by a fan. In the present work, an induced-draft, open-circuit tower is examined.

The control of the outlet water temperature is not easy to perform because it depends on many factors, such as inlet air conditions and fluid flow rates. For example, when the flow rate of process water is reduced, the risk is to obtain water at a lower temperature than required. The development of simulation models is useful to predict these effects under different operating conditions, including part-load operation. Being able to simulate the heat exchange process in detail is a very complex challenge. Different concentrated-parameter theories have been developed for the study of cooling towers [2], but to adequately simulate the evaporation process, CFD analysis is recommended [3], [4].

The first major theoretical study on heat transfer in cooling towers comes from Merkel [5], who developed a theory based on the energy balance equation on the infinitesimal interface area of heat transfer. In 1961, Baker and Shryock [6] improved Merkel's theory, providing the final expression, which is still used today. The theoretical studies carried out on evaporative towers mainly concern the determination of the parameters of the Merkel equation and the physical quantities that influence the operation of any type of evaporative cooler. In this regard, Leeper [7] demonstrated that the tower characteristic, C, and consequently the Number of Transfer Units (NTU) of the heat exchanger, is independent of the wet bulb temperature. It is exclusively a function of the ratio between the two flow rates and of the physical characteristics of the filling. In addition, they formulated equations for estimating the electrical power of the motor-fan as a function of the air flow rate passing through the tower. From the experimental analyses carried out by the Cooling Tower Institute [8] on different types of heat exchanger packages, it was found that the value of the exponent of the Merkel equation for the tower characteristic can be assumed to be −0.6 in most applications. They also conducted several experiments to obtain precise correlations for the Merkel number against the air mass flow rate [9, 10], and performed an inverse

optimization via Golden Section Search Method (GSSM) to estimate the air required to meet the demand constraints.
This is the last part of CFD. Result is analyzed by the user and collects the data shown by the CFD software. The results may be displayed as vector plots of vector quantities like velocity, contour plots of scalar variables, for example pressure and temperature, streamlines and animation in case of unsteady simulation. Appropriate formulas can be utilized to compute global parameters such as the skin friction coefficient, lift coefficient, Nusselt number, and Colburn factor, among others. These data from a CFD post-processor can also be exported to visualization software for better display and to software for better graph plotting.

II. RESEARCH METHODOLOGY

Geometry Creation

A rain zone model in 2-dimension has been constructed using the design modeler GAMBIT 2.4.6. A line with a length of 98m was formed by connecting two points (-49, 0) and (49, 0) on the x-axis, resulting in the axis being positioned on the y-axis. Two additional points (-49, 6.77) and (49, 6.77) were chosen to represent a rain zone with a height of 6.777m. Similarly, for a rain zone with a height of 8.577m, the points (-49, 8.577) and (49, 8.577) were selected. Lastly, for a rain zone with a height of 4.977m, the points (-49, 4.977) and (49, 4.977) were utilized. Subsequently, facing was applied to all geometries.

Mesh Generation

After finish meshing is compulsory, in meshing whole face is divided into equal size domain. The mesh of the model is shown in fig. 1. It shows that the domain was meshed with rectangular cells. Operational mesh is used because it gives high correctness.

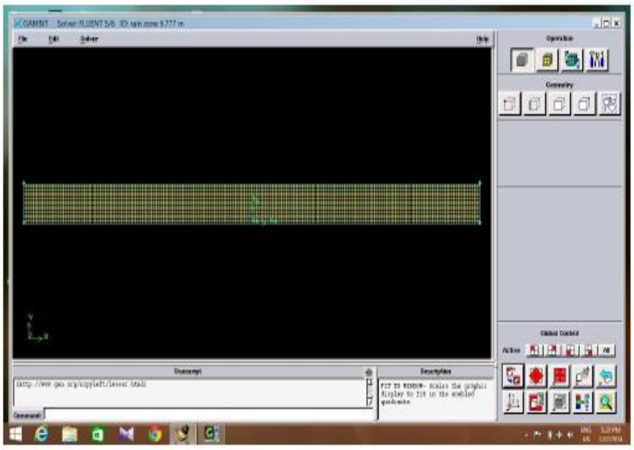

Fig. 1: Mesh

The work I have presented draws inspiration from N. Williamson et al.'s [7] study titled "Comparison of a 2D axisymmetric CFD model of a natural draft wet cooling tower and a 1D model".
The wet cooling tower was utilized to investigate the same parameters and geometry. The study was further enhanced to incorporate the impact of varying diameters in different heights of rain zones. The injection of droplets was accomplished using the discrete phase model, with the air serving as the continuous phase and the water droplets as the secondary phase.
To validate the findings, three distinct heights of rain zones were selected, all operating under identical conditions with a water droplet diameter of 2.8mm. The results obtained were consistent across the three heights of rain zones: 4.977m, 6.777m, and 8.577m. The temperature drop observed is presented below:

Table 1: Represent the percentage error with actual results

Rain zone height(m)	Actual temperature drop(k)	Predicted temperature drop(k)	% Error
8.577	3.4	3.54	4.1
6.777	2.8	2.857	2.0
4.977	2.5	2.3	8.0

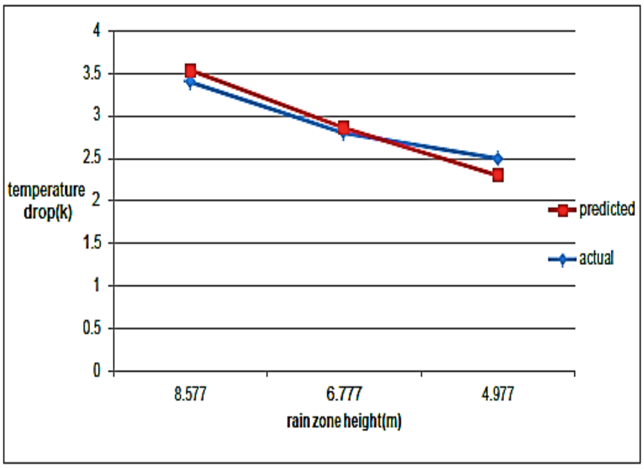

Fig. 2: Comparison between actual temperature drop with predicted temperature drop

Solution Procedure

1. The ambient air is made to flow in the solver FLUENT 6 at its ambient temperature and pressure which are assumed to be 295K and 1.01325 bar respectively. The turbulence intensity in the air is assumed 10%. The gravitational acceleration is taken as 9.81m/s2.standard k-epsilon (2-eqn) viscous model is used.
2. The species transport function is enabled in Fluent 6 and the species transport is specified for the air and water.
3. The water is introduced into the system through the utilization of specific regulations. Two regulations employed are inert heating and vaporization. Presently, the temperature, mass flow rate, and other parameters are precisely defined. The Lagrange particle tracking technique is employed to monitor the movement of the droplets. The stochastic tracking method utilizes the discrete random walk model.
4. The water-air interaction is now activated, and unsteady particle tracking is employed. The analysis

incorporates turbulence coupling and coupled heat and mass transfer. Additionally, an automated tracking scheme is enabled. To improve the solution's accuracy, droplet collision and droplet breakup are incorporated.

III. RESULT AND DISCUSSION

The analysis has been done for the variable diameters of the water droplets in the rain zone of the cooling tower and the cooling range has been determined for the design variables i.e. rain zone height. The efficiency of the cooling tower depends on the sensible heat transfer and vaporization which in turns depends on the saturation pressure, vapor pressure and the time available for this intimate contact. These factors are strongly depends on rain zone height, droplet diameters, base diameter and the type of flow. I have observed the variation in cooling effect for the different height of rain zones by varying water droplet diameter with Rosin-Rammler distribution and also compare the relative cooling rate among different heights of rain zones. Work presented in this project, the conditions are same unless otherwise specified.

Effect of variable droplet diameters for rain zone of 4.977m height

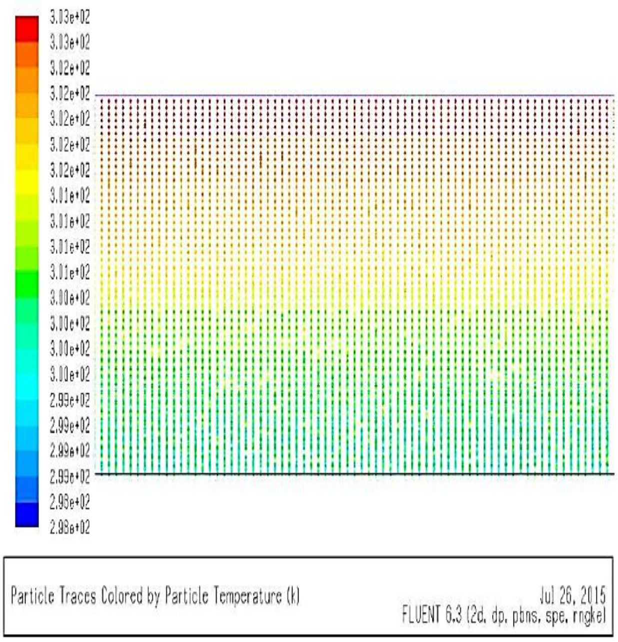

Fig. 3: Water temperature variation contours for maximum diameter 2.6mm and minimum diameter 1.8mm

Fig. 3 shows the temperature variation in the rain zone of the cooling tower for maximum diameter 2.6mm and minimum diameter 1.8mm. In this picture the water temperature is decreasing from 303 K to 298.924K.

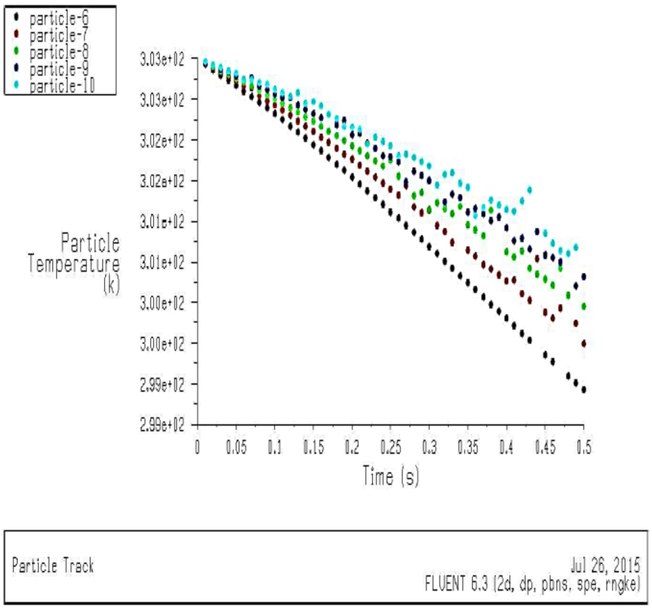

Fig. 4: Water temperature variation plot for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig. 4 represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 298.924 K.

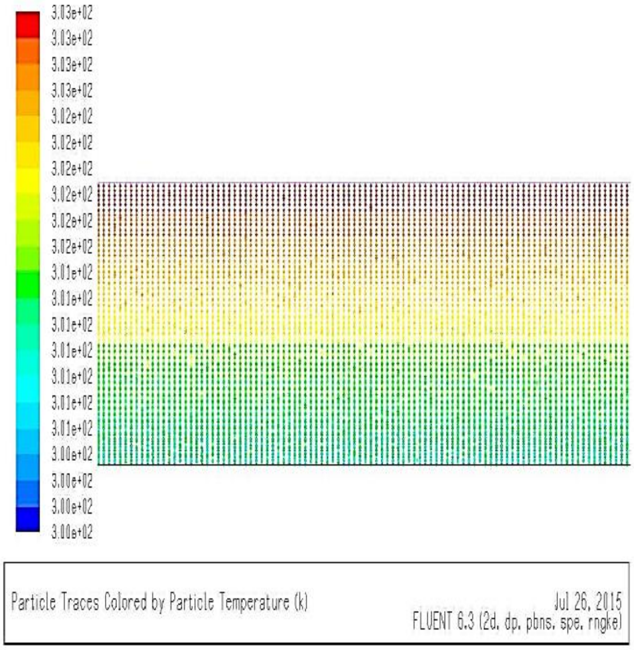

Fig. 5: Water temperature variation contours for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig. 5 shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.2mm and minimum diameter 2.4mm. In this picture the water temperature is decreasing from 303 K to 300.315K.

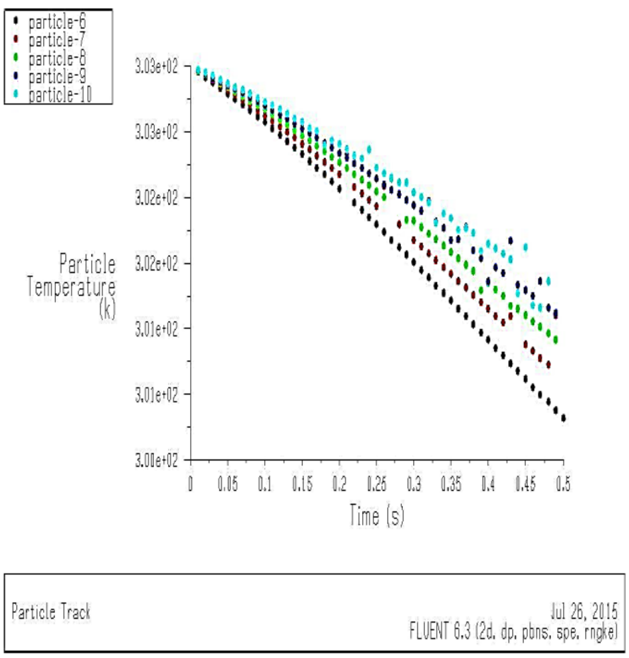

Fig. 6: Water temperature variation plot for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig. 6 represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 300.315K.

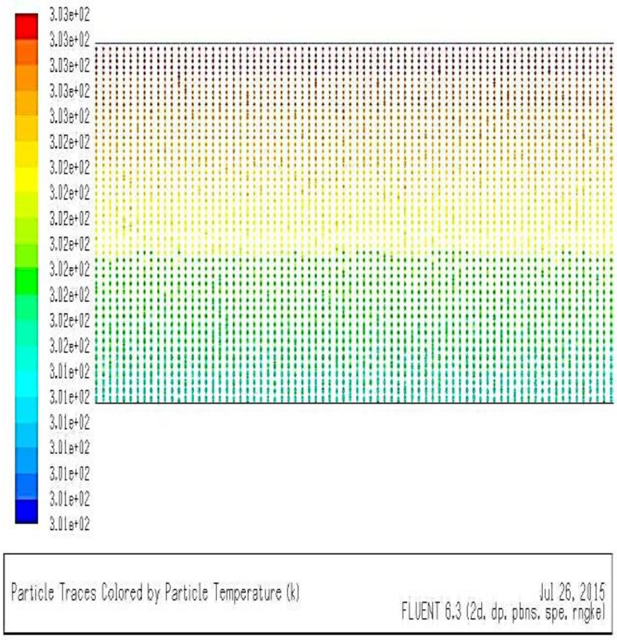

Fig. 7: Water temperature variation contours for maximum diameter 3.8mm and minimum diameter 3mm.

Fig. 7 shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.8mm and minimum diameter 3mm. In this picture the water temperature is decreasing from 303 K to 301.116K.

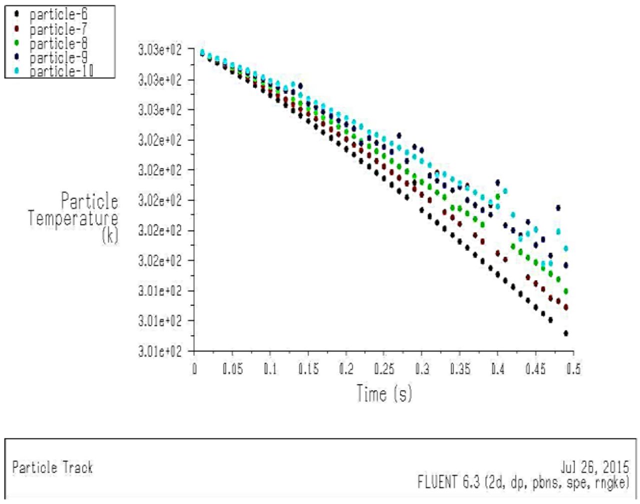

Fig. 8: Water temperature variation plot for maximum diameter 3.8mm and minimum diameter 3mm.

Fig. 8 represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 301.1163K.

Table 2: Represent temperature drop for 4.977m height rain zone

Droplet diameter	Water inlet temperature	Water outlet temperature	Temperature drop
1.8mm – 2.6mm	303	298.924	4.076
2.4mm – 3.2mm	303	300.315	2.685
3.0mm – 3.8mm	303	301.116	1.884

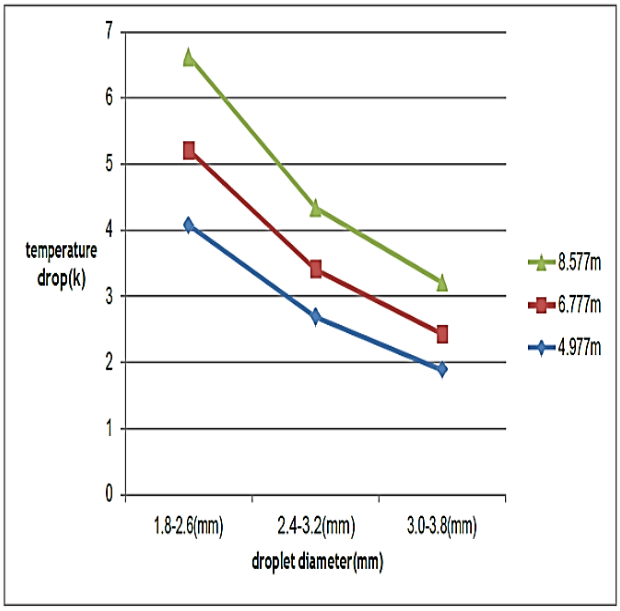

Fig. 9: Comparison of temperature drop between different heights and different diameters.

IV. CONCLUSION

On the basis of analysis done in this thesis, following are the conclusions:

1. Tables 1, 2, and 3 demonstrate that the cooling range expands as the diameter of water droplets decreases, regardless of the height of the rain zone.
2. The cooling rate is further enhanced as the height of the rain zone increases.
3. The rate of temperature drop increases as the rain zone height increases when the diameter of the water droplet remains constant.
4. The temperature drop rate increases as the droplet diameter decreases, as depicted in Figure 9, when considering a larger height of the rain zone.

REFERENCES

[1] H.C.R. Reuter, D.G. Kroger "Computational Fluid Dynamics Analysis Of Cooling Tower Inlets", Journal of Fluid Engineering 133 (8) 081104-081104-12.

[2] Abdullah alkhedhair, Hal Gurgenci, Ingo Jahn, Zhiqiang Guan, Suoying He, "Numerical simulation of water spray for pre-cooling of inlet air in natural draft dry cooling towers", Applied Thermal Engineering 61 (2013) 416-424.

[3] Rafat Al-Waked, Masud Behnia, "CFD Simulation of Wet Cooling Towers", Applied Thermal Engineering 26 (2006) 382-39.

[4] N.J. Willamson "Numerical Modeling of Heat And Mass Transfer And Optimisation of a Natural Draft Wet Cooling Tower" PhD Thesis, The School of Aerospace, Mechanical and Mechatronics Engineering, The University of Sydney, 2008.

[5] D.J. Viljoen "Evaluation And Performance Prediction of Cooling Tower Spray Nozzles" PhD Thesis, University of Stellenbosch.

[6] .G. Kroger "Air Cooled Heat Exchangers And Cooling Towers" Pennwela Corp. Tulsa, USA, 2004.

[7] D. Radosavljevic "The Numerical Simulation of Direct Contact Natural Draught Cooling Tower Performance Under The Effect of Cross Wind", PhD Thesis, University of London, 1990.

[8] A.S. kaiser, M. Lucas, A. Viedma, B. Zamora, "Numerical model of evaporative cooling processes in a new type of cooling tower", International Journal of Heat and Mass Transfer 48 (2005) 986–999.

[9] Alok Singh, S P S Rajput, "Application of CFD In Natural Draft WetCooling Tower", International Journal of Engineering Research And Applications 2 (2012) 1050-1056.

[10] G. Gan, S.B. Riffat, L. Shao, P. Doherty, "Application of CFD To Closed Wet Cooling Towers", Applied Thermal Engineering 21 (2001) 79-92

[11] H. Lowe, D.G. Christie "Heat Transfer And Pressure Drop Data On Cooling tower Packings And Model Studies of The Resistance of Natural Draught Cooling Towers To Airflow", International Heat Transfer Conference, Colorado.

[12] M.N.A Hawlader, B.M. Liu "Numerical study of The Thermal-Hydraulic Performance of Evaporative Natural Draft Cooling Towers", Applied Thermal Engineering 22 (2002) 41-59.

[13] R. Suresh Kumar, S.R. Kale, P.L. Dhar, "Heat And Mass Transfer Process Between Water Spray And Ambient Air Experimental data" Applied Thermal Engineering 28 (2008) 349-360.

[14] N. Williamson ,M. Behnia, S. Armfield, "Comparison of A 2D Axisymmetric CFD Model of A Natural Draft Wet Cooling Tower And 1D Model", Elsevier, International Journal of Heat And Mass Transfer 51 (2008) 2227-2236.