Fluid Dynamics to analyze the flow passing through a Cooling Tower: A Study

[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:- A cooling tower is a crucial component in power plants and heating ventilation air conditioning systems. It functions by expelling waste heat from hot water into the environment. As the hot water interacts with the air, it undergoes sensible heat transfer and the vaporization of certain water droplets, resulting in a decrease in water temperature. The analysis of the results indicates that as the droplet diameter decreases in any given height of the rain zone, there is an increase in the temperature drop. Furthermore, it is observed that the rate of temperature decrease is higher for the rain zone at a height of 8.577m compared to the other two rain zones.

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

I. INTRODUCTION

Industrial cooling systems are used to reject waste heat from power plants to the environment systems effectively that require heat rejection in refrigeration, process, chemical, combustion and power generation plants. In the past, the appropriate method of cooling was by hydrosphere, which involves water from a natural resource being passed through a heat exchanger and comeback to the source at an increased temperature. Several countries have legislation which restricts the increase in temperature limit of the cooling water used due to the harmful impact on the environment. This environmental issue, the lack of natural resources and the rising cost of water has bounded the use of natural water for once-through cooling [1, 2].

Cooling towers are essentially heat removing devices that removes waste heat to atmosphere. They are widely used in a wide range of areas like oil refineries, petrochemical and other chemical plants, thermal power stations and HVAC systems for cooling buildings. A natural draft wet cooling tower uses natural air drift for movement of air. It is mainly used in power stations. A lot of research goes around too make these cooling towers more efficient. The importance of NDWCT efficiency can be drawn from the fact that a single degree rise in outlet water in a power plant substantially increases the power production cost [3, 4].

Computational fluid Dynamics offers a platform to perform simulation of a working cooling tower and checking its various parameters without actually modelling an actual one. Currently only 1d and 2d simulations of NDWCT is being used in cooling tower design. This study aims to study the various models that are currently used to design cooling towers and to do a literature research work that has been done in the CFD simulation of different cooling tower designs and to find out the prospects of future work that can be done in order to get a better, more efficient and commercial livable design. The main contribution of the project is to answer several important questions relating to natural draft wet cooling tower (NDWCT) modelling, design and optimization [4, 5].

Specifically, the current work aims to conduct a detailed analysis of NDWCT and basic knowledge of Computational Fluid Dynamics (CFD). A general study of various cooling towers along with their common parts has been done. Followed by specific and detailed study of NDWCT along with its main parts like structure, fills and its types, drift eliminators, water basin and water spray system etc.

II. COOLING TOWER

There are several cooling tower designs. When water is passed through finned tubes forming a heat exchanger only sensible heat is transferred to the air such type of cooling tower is called dry cooling tower. In wet cooling tower water is sprayed directly into the air so evaporation occurs and both latent heat and sensible heat are exchanged. In a hybrid cooling tower a combination of both Methods are used. Cooling towers can further be divided into forced draft, natural draft and induced draft cooling towers. Forced draft and induced draft units are relatively small structures where fan drives the air flow [1].

In a natural draft cooling tower the air flow is developed by only natural convection. The draft is established by the density difference between the hot air inside the tower and the cold ambient air outside the tower.

A characteristic of natural draft wet cooling tower (NDWCT) is the spine of the cooling framework being used in expansive present day thermal power plants. In NDWCT, a blend of heat and mass exchange impacts are utilized to cool the water coming from the turbine's condenser. The boiling hot water, originating from the condenser, is showered on top of sprinkle bars or film fills keeping in mind the end goal to uncover an expansive bit of water surface to the cooling encompassing air [2, 3]. The humidity of the cooling air is less than the humidity of saturated air at the boiling hot water temperature, which brings about evaporation an measure of water. The energy needed for evaporation is extracted from the remaining water, henceforth diminishing its temperature. The cooled water is then gathered at the bowl of the NDWCT and pumped once again into the condenser. It is shown in fig: 1.

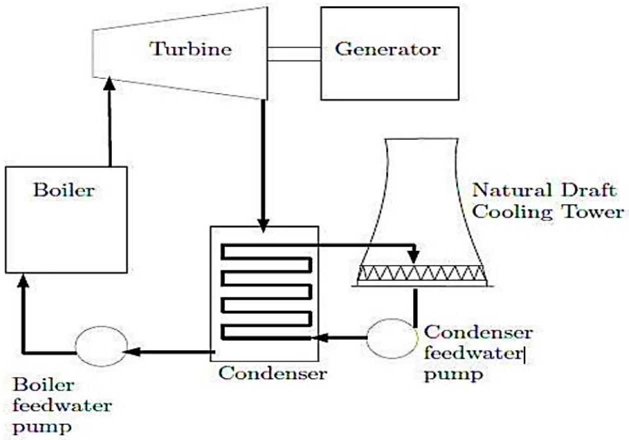

Figure 1: Power station cycle with cooling tower

III. LITERATURE REVIEW

In [1], have developed a numerical model to analyze the performance of a system that involves the two-phase flow of humid air and water droplets. The computational flow dynamics approach is employed, with the gas flow phase being treated using the Eulerian approach and the water droplet flow phase being treated using the Lagrangian approach. The two phases are coupled in a two-way manner. The researchers have demonstrated that the efficiency of the system is strongly influenced by the average size of the water droplets. Additionally, they have investigated the effects of other variables such as wet bulb temperature, the ratio of water mass flow to air mass flow, and the temperature difference between the water inlet temperature and the wet bulb temperature.

In [2], have developed a model using the commercial code FLUENT to analyze a natural draft wet cooling tower. Their model adopts a two-dimensional computational fluid dynamics (CFD) approach to determine the heat and mass transfer from water to air. The Eularian multiphase model and the RNG K-ε model are utilized to simulate the flow and turbulence characteristics of the multiphase flow, respectively. The researchers have observed a low density value along the axis of the cooling tower, with higher values near the wall. The highest thermal conductivity value is found near the axis. Furthermore, they have found that the stream function remains linearly constant along the axis.

In [3], have applied CFD to determine the performance of closed wet cooling towers according to their cooling capacity and pressure loss. They have compared their results with experimental observations for a large industrial as well as prototype cooling tower. They have shown that CFD can be used for determining the performance, pressure loss and optimum design of cooling tower. They have also suggested that for good results and simplifications in CFD the varying heat flux should be used instead of constant heat flux.

In [4], have investigated the effects of cooling tower inlet design on inlet viscous flow losses using ANSYS-FLUENT. They have compared the axial velocity profile data, tower inlet losses with the experimental results. Further they have used the same CFD model for determining the effect of the inertia forces and viscous forces, shell wall thickness and shell wall inclination on the air flow pattern. Finally they have developed simple correlations for determining the cooling tower inlet losses. They have reached at a conclusion that the inlet diameter to height ratio has a major impact on inlet losses.

In [5], have done a numerical investigation of inlet air pre-cooling with water sprays to enhance the performance in Natural Draft Dry Cooling Towers (NDDCT). A 3-D numerical model of a test channel was generated and the evaporation from a single spray nozzle was analyzed. Their results showed that up to 81% evaporation can be achieved for water droplets of 20 mm at a velocity of 1 m/s and they reveal droplet transport and evaporation strongly depend on droplet size and air velocity.

In [6], have investigated the effect of operating conditions and crosswind conditions for a 3 dimensional natural draft wet cooling tower using FLUENT and have utilized the standard K-ε turbulence model. They considered Eularian approach for air phase and Lagrangian approach for water phase. They have also investigated the effect of droplet diameter, no. of nozzles and no. of tracks per nozzle. They have reported that droplet diameter has the most significant improvement on the thermal performance of the cooling tower.

In [7], have developed one dimensional and two dimensional CFD models and compared them under the design variables. They have reported a difference of less than 2% between the results of one dimensional model and two dimensional model. The difference between the tower range is found to be less than .4% in most cases.

In [8], have focused on the understanding on heat and mass transfer mechanism involved and to check the possibility of using the CFD code FLUENT for simulating mass and heat transfer phenomena in an indirect cooling tower. They found that the mass transfer coefficient obtained through CFD code FLUENT was close to experimental correlation, especially at higher flow rate.

In [9], have investigated the effects of water droplet diameter and water droplet temperature on the thermal performance of the Wind Tower at specific inlet air velocity and relative humidity and height of wetted columns. Also studied the effects of wind velocity, temperature, and relative humidity inlet to Wind Tower. Changing the height of the wetted columns and its effect on the evaporative cooling in other specific parameters is studied. They reveal the height of 10 m of wetted columns decreases 12 K of the ambient air temperature and increases 22% of its relative humidity

In [10], have solved the cooling zone using engineering equation solver software. They compared the results with experimental data. They focused on the fouling of fills and presented his fouling model in terms of normalized fill performance index. They found fouling to be a major source of degradation of cooling tower performance.

IV. CFD MODELLING

The technique of studying and performing fluid dynamics and heat transfer has been revolutionized by the design of high-speed digital computers and accurate numerical methods for solving physical problems. This revolutionary approach, known as Computational Fluid Dynamics (CFD), enables the investigation of complex flow geometries with the same ease as solving idealized problems using conventional methods. CFD combines the fields of fluid dynamics and numerical analysis, and its development in the 1960s and 1970s was primarily driven by the aerospace industries' needs. CFD finds applications in various branches of engineering, including civil, mechanical, electrical, electronics, chemical, aerospace, ocean, and biomedical engineering. Additionally, CFD can be utilized in testing and experimentation, leading to a reduction in the overall time required for testing and designing. The processes of CFD are depicted in Fig. 2.
The availability of affordable high-performance computing hardware and user-friendly interfaces has resulted in the emergence of commercial CFD packages.

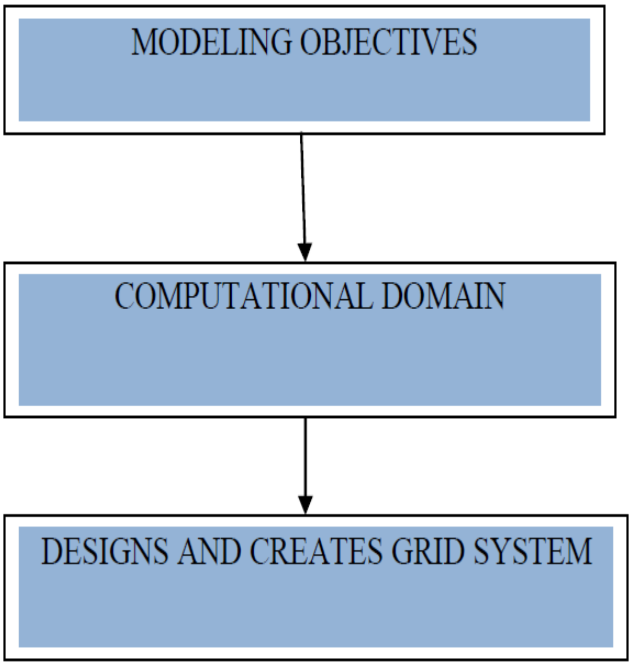

Figure 2: Steps of pre-processing

Prior to the widespread use of these packages, individuals had to develop their own code to conduct CFD analysis. Each program varied depending on the specific problem, although certain portions of code could be reused. These programs were not adequately tested, leading to doubts about the reliability of the results. Nowadays, thoroughly tested commercial CFD packages have not only made CFD analysis a standard design tool in the industry but also assist research engineers in effectively studying the physical system.
CFD modeling process starts with a consideration of the actual problem and identification of the computational domain. This is followed by the mesh structure generations, which is the most necessary part of the pre-processing activity. It is understood that more than half of the time taken by a CFD analyst goes towards mesh generation. Both computation time and correctness of solution depend on the mesh structure. Optimal grids are generally non-uniform – finer in areas where large variation of variables is predicted and coarser in regions where relatively little changes is predicted. In order to decrease the difficulties of engineers and maximize productivity, all the major CFD programs include provision for importing shape and geometry information from CAD packages like AutoCAD and I-DEAS, and mesh information from other packages like GAMBIT.

V. CONCLUSION

Evaporative towers are the most widely used systems for cooling water in industrial processes. These systems, however, are characterized by high energy and water consumptions. Therefore, technologies and operating strategies for optimization of these devices are highly recommended. The heat exchange, which takes place inside an evaporative tower is very complex and not easy to model. In this work, an induced-draft evaporative tower is modelled by means of computational fluid dynamics (CFD), which is relatively new in this field of application.

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.