International Journal For Technological Research In Engineering
Volume 6, Issue 5, January-2019 *ISSN (Online): 2347 - 4718*

WORKING AND PERFORMANCE OF NATURAL DRAFT WET COOLING TOWER WITH AIR INLET HEIGHT AS A VARIABLE PARAMETER

Alok Singh[1], Sohail Bux[2]
[1]Agnos College of Technology (RKDF UNIVERSITY)
[2]Mechanical Engineering Department RKDF UNIVERSITY Bhopal

Abstract: Current paper deals with performance of natural draft wet cooling tower with air inlet height and reduce the evaporation losses in natural draft wet cooling tower by using 2D CFD model. A software model of CFD k-Ɛ is used by this model we are analyzed how the large scale difference occurs in 2D CFD model of NDWCT. By varying the inlet height of tower. This study is based on the 118 m total tower height with varying inlet height of 6, 6.2, and 6.4 meters. By these models we have check the outlet let temperature and relative humidity and in which model outlet temperature is high and relative humidity is low that model is called optimized model.

I. INTRODUCTION

Cooling towers are a very important part of many chemical plants. The primary task of a cooling tower is to reject heat into the atmosphere. They represent a relatively inexpensive and dependable means of removing low-grade heat from cooling water. The make-up water source is used to replenish water lost to evaporation. Hot water from heat exchangers is sent to the cooling tower. The water exits the cooling tower and is sent back to the exchangers or to other. A cooling tower is a heat rejection device, which extracts waste heat to the atmosphere through the cooling of water stream to a lower temperature. The type of heat rejection in a cooling tower is termed "evaporative" in that it allows a small portion of water being cooled to evaporative into a moving air stream to provide significant cooling to the rest of that water stream. Evaporative heat rejection devices such as cooling towers are commonly used to provide significantly lower water temperature than achievable with "air cooled" or "dry" heat rejection devices. the design of current model is prepared with the help of fluent code" Ansys 13".by this model we are checking different operational parameter of natural draft wet cooling tower. Current design procedure is based on the two dimensional model of heat and mass transfer, the equation used in this model is ordinary differential equation and basic navior-stoke equation is used. Air flow is solved as continuous phase using Eulerian approach whereas droplet trajectories are solved as dispersed phase using the Lagrangian approach. The influence of crosswind condition to the thermal performance of natural draft wet cooling tower was also investigated in this work.

II. NATURAL DRAFT WET COOLING TOWER

Natural draft wet cooling tower (NDWCT) are based on the Merkel's theory of heat and mass transfer. According to Markel’s theory heat and mass transfer between fluids and air depends upon the enthalpy gradient between the state of saturation of air in the boundary layer on the surface of the hot water and the state of air in the colder core stream.There are three different heat and mass transfer zone:
1) spray zone 2) fill zone 3) rain zone

1	Overall height of cooling tower	118 m
2	Air inlet height of cooling tower	6.2 m
3	Water inlet height	8.7 m
4	Depth of cooling tower basin	2.35m
5	Diameter of cooling tower basin	82 m
6	Diameter of cooling tower at fill material	78.744 m
7	Diameter of cooling tower outlet	47.744 m
8	Diameter of cooling tower at throat	49.648 m

Table 1: Physical Parametre Of Cooling Tower Model

The water is brought into the tower form the condenser through a pump and injected into the cooling tower approximately 8-10 m above the basin. This water is injected through nozzle. The function of spray zone is splash the water into smaller droplets before collides with the fin. There is a range of fill type 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 when water strikes with the fin there is a very high humidity inside the cooling tower.

Steps Used In Preparing Geometry of Cooling Tower

In the first step geometry is created in 2D using reference data of different part of cooling tower. The structure of whole model imagined in advance, because the possibilities in the subsequent steps depended on the composition of different geometrical shapes. The various assumptions were made during preparing the model.

- 2-D symmetric model is developed; fix the fill corresponding to the real arrangement.
- Inlet and outlet space is created in bottom and top of the tower.
- Cooling tower shall 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.

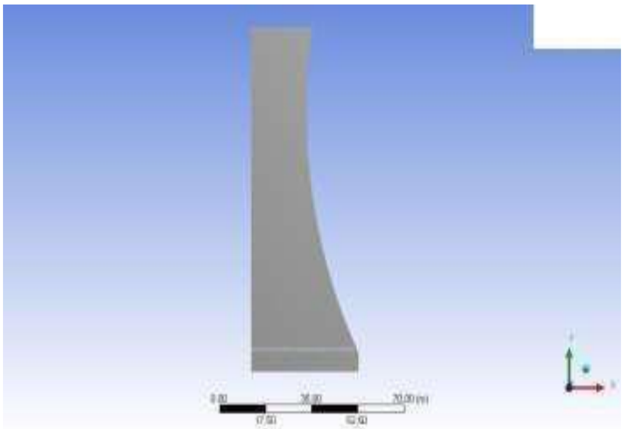

Figure 1: Geometry of cooling tower model

Meshing Of the Model

Appropriate resolution of the flow field inside the computational fluid dynamics after geometry mesh is generated. During mesh generation so much care is to be taken because the end results very much depend on quality of mesh. In order to have an domain is define into a large number of finite volume cells.

Various steps are used in generating mesh:

Different part is meshed with different element sizing.

- Fill zone must be fine meshed.
- By used mapped face mesh the model with appropriate element sizing
- The inner and outer surface of the wall inside the model has identical shapes but is disconnected so the mesh size on the two walls can be differ

Thermal parameters used in fluent analysis:

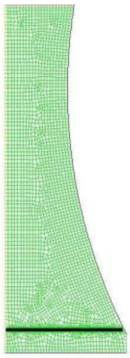

Figure 2: Meshed model of cooling tower.

Cell Zone Condition:

In cell zone surface body is considered as fluid. The operating pressure is atmospheric in upstream from the centre line of cooling tower. The gravitational acceleration is 9.81 m/s2. Operating temperature is 298 K and operating density is 1.22 kg/m3.

Boundary Condition:

Velocity inlet boundary condition is used to define the inlet velocity and other property of air takes normal to the boundary inlet. Turbulence is taken as intensity and length scale. Pressure outlet of air is defined at outlet of air.

1	Hot water temperature	317 K
2	Cold water temperature	306 K
3	Dry bulb temperature of ambient air	307.96 K
4	Wet bulb temperature of ambient air	300 K
5	Relative humidity of ambient air	48.98 %
6	Cooling range	12 K
7	Approach	6 K
8	Water flow rate	6949 lit/sec
9	Water flow rate through per nozzle	239 lit/sec
10	Number of nozzle	2899
11	Makeup water	140 lit/sec

Table 2: About the Fluent Model: Model Used In Analysis Standard k-ε Model

The Standard k-ε (Launder and Spalding, 1974) model is the most widely used complete RANS model and it is incorporated in most commercial CFD codes. In this model, the model transport equations are solved for two turbulence quantities i.e., k and ε. The Standard k-ε model turbulence model solves the flow based on the assumption that the rate of production and dissipation of turbulent flows are in near-balance in energy transfer We use k and ε to define velocity scale ϑ and length scale L representative of the large scale turbulence as follow

III. RESULTS AND CONCLUSIONS

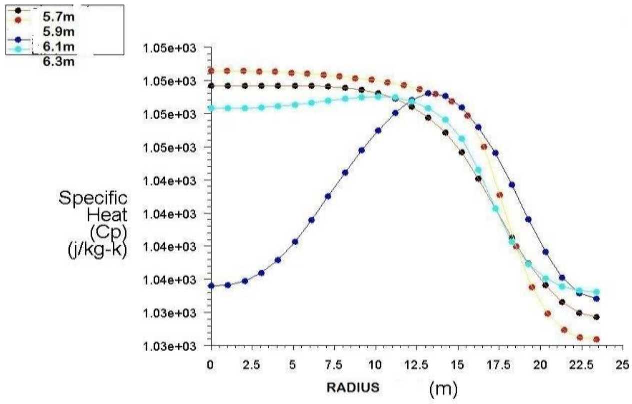

Figure 3 : Comparison between specific heat and outlet radius at different air inlet height

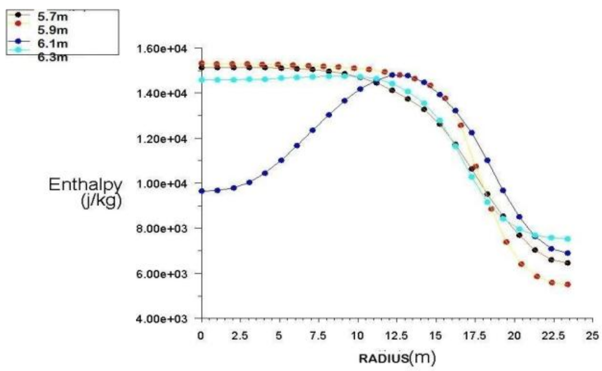

Figure 4 : Comparison between enthalpy and outlet radius at different air inlet height

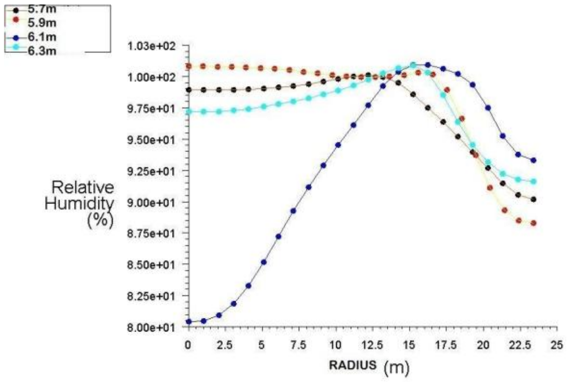

Figure 5: Comparison between relative humidity and outlet radius at different air inlet height

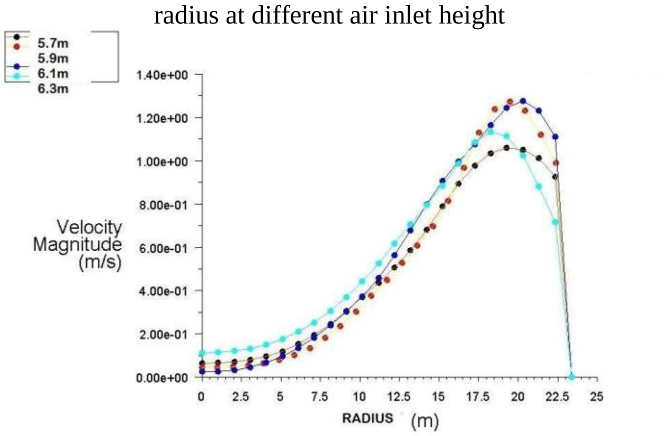

Figure 6: velocity and outlet radius at different air inlet height

IV. CONCLUSIONS

A two dimensional CFD model of natural draft wet cooling tower has been developed and validated along with actual data of NTPC dadri power plant and it is found that the result was very near and accurate to the actual data. By modeling and simulation of various air inlet heights it was found that the result with 6.3 m air inlet height is optimum and best. Simulation results with 6.3 m air inlet height reduce significantly the amount of makeup water and increase the effectiveness of cooling tower. Amount of makeup water saved with 6.3 m air inlet height is 7.15*107 liters in a year. The effectiveness of cooling tower with actual air inlet height is 68.75% and the effectiveness of cooling tower with optimum air inlet height is 73.29%. Means with the optimum air inlet height the effectiveness of cooling tower is increased by 6.194%. Also the power which is required to mix the makeup water with the cooled water is saved. So the efficiency of power plant is also increased by significantly.

REFERENCE

[1] D.G. Kro¨ger, Air-cooled Heat Exchangers and Cooling Towers, Pennwalt Corp., Tulsa, USA, 2004.
[2] ASHRAE, Cooling towers, in: ASHRAE Systems and Equipment Handbook, 2000, pp. 36.1–36.19.
[3] H. Jaber, R.L. Webb, Design of cooling towers by the effectiveness-NTU method, J. Heat Transfer 111 (1989) 837–843.
[4] J.C. Kloppers, A critical evaluation and refinement of the perfor-mance of wet-cooling towers, PhD Thesis, University of Stellenbosch, Stellenbosch, South Africa, 2003.
[5] J.C. Kloppers, D.G. Kro¨ger, A critical investigation into the heat and mass transfer analysis of counterflow wet-cooling towers, Int. J. Heat Mass Transfer 48 (3–4) (2005) 765–777.
[6] R.L. Webb, A critical evaluation of cooling tower design methodol-ogy, in: R.K. Shah, E.C. Subbarao, R.A. Mashelkar (Eds.), Heat Transfer Equipment Design, Hemisphere, New York, 1988, pp. 547–558.
[7] D.R. Baker, H.A. Shryock, A comprehensive approach to the analysis of cooling tower performance, J. Heat Transfer 83 (1961) 339–350.
[8] A.E. Feltzin, D. Benton, A more nearly exact representation of cooling tower theory, Cool. Tower Inst. J. 12 (2) (1991) 8–25.
[9] J. Khan, S. Zubair, An improved design and rating analysis of counter flow wet cooling towers, J. Heat Transfer 123 (2001) 770–778.
[10] H.T.A. El-Dessouky, A. Al-Haddad, F. Al-Juwayhel, A modified analysis of counter flow wet cooling towers, J. Heat Transfer 119 (1997) 617–626.

