(IJAER) 2016, Vol. No. 11, Issue No. VI, June **e-ISSN: 2231-5152, p-ISSN: 2454-1796**

DECREASE THE OUTLET WATER TEMPERATURE OF CROSS FLOW COOLING TOWER

***R.Immanual, **S.Rajakumar, ***B.Ragupathi, ***S.P.Ramkumar,**

**PG student, ** Assistant Professor*

Department of Mechanical Engineering,

Regional campus, Anna University Tirunelveli Region, Tirunelveli, India

****UG student, Department of Mechanical Engineering,*

Government college of Engineering, Tirunelveli

Tirunelveli, India

ABSTRACT

In this paper two methods developed to reduce the outlet water temperature of the cross flow cooling tower in DCW limited. Due to reduction of the outlet water temperature the cooling range will be increased and it also leads to the increase the efficiency of the overall power plant. The first method deals with the water path cooling. The decrease in the outlet water temperature by using water path cooling is proved by the conventional heat transfer formulae. The second method deals with the decrease the liquid to gas flow ratio(L/G) by increasing the air velocity with the help of cylinder blocks near the louvers of cooling tower. Decrease in the outlet water temperature by using decrease the L/G ratio proved by the combination of literature review and CFD analysis of air flow.

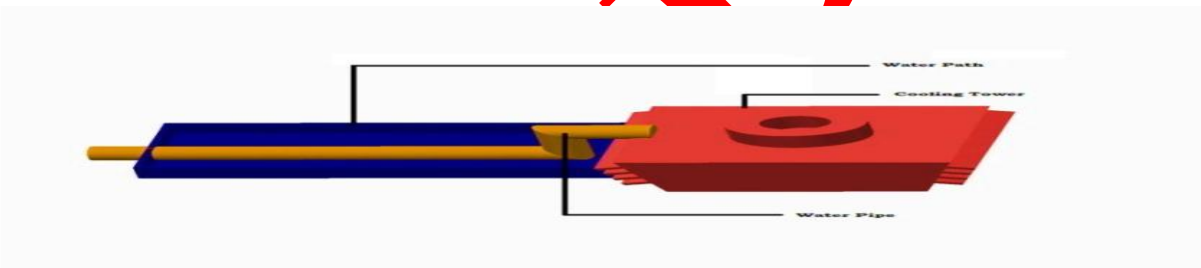
Keywords: cross flow cooling tower, L/G ratio, CFD analysis, cooling range, water path cooling

INTRODUCTION

In a diesel power plant cooling water must be need to cool's the part's of the diesel generator. The cooling water do not directly cool's the diesel generator part's. In real case the diesel generator parts first cooled by the oil circulating system that oil cools by the cooling water by means of a Heat exchanger. If the cooling water temperature is very low means it can extract more heat from the diesel power plant circulating oil. So the diesel power plant efficiency will be more. Here in this paper proposed two methods to reduce the outlet temperature of water from cooling tower. Because the cooling tower outlet water go to the inlet of Heat exchanger to extract the heat from the oils. This will be practically implemented in the DCW cooling tower for reducing the outlet temperature of cooling tower. From the literature, optimization of air inlet angle for increase the efficiency of Induced draught cross flow cooling tower. Ravindrakirar et.al [1] optimized the louver's inlet angle for increase the effectiveness of Induced draught cross flow cooling tower using CFD Analysis in ANSYS work bench. According to his conclusion the water outlet temperature of cooling tower decreases as the air inlet angle decreases. Hence the cooling efficiency and effectiveness of cooling tower increases out of selected three air inlet angles, angle of 15° leads to maximum efficiency and effectiveness for selected cooling tower. On the otherside, as the air inlet angle increases the water outlet temperature also increases and cooling efficiency and effectiveness of cooling tower decreases.Ebrahim Hajda Vallo reference [2] predicts the thermal behavior of an existing cross flow tower under variable wet bulb temperature by a conventional mathematical model and the available fill characteristics curve of the tower. According to his prediction, the wetbulb temperature increases means they came near range of evaporation loss are decreases at constant dry temperature. Tawsif Mahmud [3] studied experimentally the heat transfer phenomena between water and air by direct contact in a cross flow cooling tower model. From his observation the range of the cooling tower increases with the increase of air velocity approach decreases with the raise in air velocity. Evaporation loss increases with the increase of air velocity. Cooling capacity increased slowly with the increase in air velocity. Pushpa B [4] evaluates the performance of thermal power station cooling tower. According to this if the inlet air temperature is higher means the evaporation rate will also increases and inlet water temperature increases the rate of heat loss will also increases. Kirankumar et.al [5], studied the flow field in the hyperbolic natural draft wet-cooling tower, which has great effects on the economy and security of power plant through numerical simulation according to his study the thermal conductivity of the fluid is more in the vertical axis of cooling tower. The density of fluid is having reciprocal values of input temperature. The enthalpy of fluid is low at the wall of cooling tower. Turbulent intensity is increases up to rain zone and then decreases, turbulent viscosity decreases up to fill zone and then increases. Stream function is linearly constant for axis and decreasing according to height for middle line and line near wall. Enthalpy decreases after rain zone and its value is low near the wall area. Irtaza [6] studied the effective and reliable approach for evaluation of wind force on cooling towers using the standard K-ε, RNG K-ε, Realizable K-ε turbulence model from his study he realize that the pressure co-efficient obtained using RNG K-ε turbulence model is in good agreement on the windward face of the cooling tower and has advantage of providing rapid solution.By conducting 2 dimensional CFD simulations for investigating the thermal performance of wet cooling tower the overall changes take place in three zones. 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 7 pa to zero at fill fluent software. S.ParimalaMurugaveni [7] compares the effectiveness of 8 cooling tower models, the cooling zone area than increase slightly according to height. Highest value of thermal conductivity is near axis. Density will be the temperature reciprocal values. Enthalpy is low at near wall. 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. Enthalpy decreases after rain zone and its value is very low near wall area. In CFD analysis of Induced Draught cross flow cooling tower, Prof. Yogeshparki [8] concluded that ANSYS is use for analysis Induced draught cross flow cooling tower. Because of his analyzed reading will nearly match with the practical reading. Analysis of forced Draft cooling tower performance using Ansys tower with air inlet pipe at 0° and the cooling tower with air inlet pipe inclined at 30° about both horizontal and vertical axis have nearly same effectiveness. She also predicts that nozzle assembled air inlet pipe cooling tower models have not obtained any performance enhancement. From the literature survey RNG K-ε model is suitable for turbulence analysis. ANSYS commercial package give good results on finest element Engineering application.

WATER PATH COOLING

To reduce the water outlet temperature in the DCW cooling tower the one of the practically possible method is make a water path cooling in the inlet pipe of cooling tower it's like a lake through which the inlet pipe of cooling tower passed so the hot water from heat exchanger first cools from the artificial water path by nearly 3°celcius. After that it will go to the cooling tower so the range of the cooling tower increase more if it's calculated from the water path cooling.the structure of water path cooling will be showed in the figure 1.



THEORETICAL ANALYSIS OF WATER PATH COOLING

Inlet Water Temperature	T_i	=	40°C
Properties of Water at 40 °C			
Density		=	995Kg/m^3
Thermal Conductivity	k	=	0.628 W/mK
Kinematic Viscosity	v	=	0.000000657m^2/s
Specific Heat Capacity	C_p	=	4178 J/kg k
Prandtl Number	Pr	=	4.34
Diameter of the Pipe	D_h	=	0.2032 m
Pipe Length	L	=	10 m
Lake Water Temperature	T_s	=	30°C
Area of Pipe	A	=	0.032412838m^2
Mass Flow Rate of Water	m	=	86.68 kg/s
Velocity of Water	v_m	=	2.687687416 m/s

$$Re = \frac{V_m D_h}{\nu}$$

Reynolds Number	Re	=	831260.4002

Condition of Flow IS TURBULENT

$$L_h \approx L_t \approx 10D$$

Entry Length = 2.032 m
Fully Developed Flow
Nusselt Number Nu = 2251.714899

$$\mathrm{Nu} = \frac{hD_h}{k} = 0.023\ \mathrm{Re}^{0.8}\ \mathrm{Pr}^{0.3}$$

Heat Transfer Coefficient h = 6959.04014 W/m^2K
Surface Area of Pipe A_s = $3.19024m^2$

$$T_e = T_s - (T_s - T_i)\exp(-hA_s/\dot{m}C_p)$$

Exit Water Temperature T_e = 37.8°C
Due to this, the hot water enters to the cooling tower will reduced to low temperature. So this will be satisfies the main object to reduce the inlet water temperature. According to theoretical analysis nearly 3°C will be reduced at the cooling tower inlet water.
For L/G ratio = 1

INCREASE THE AIR VELOCITY BY SPERICAL BLOCKS

Another practical approach to reduce the outlet water temperature is to increase the air velocity at inlet this will leads to increase the L/G ratio if the L/G ratio increases the outlet water temperature decreases. To increase the air velocity place some spherical blocks inline near the cooling tower louvers this will increase the velocity may be increase the turbulence. The increase in the velocity will be measured from outlet because in DCW cooling tower it have induced fan to sucks the air. So at first using the CFD analyses to find the outlet velocity present case. Then check whether the outlet velocity increases or not for the presence of spherical blocks.

CFD ANALYSIS

Here ANSYS workbench is used for CFD analysis of cooling tower. For CFD analysis following step are perform. In step 1 cooling tower model make in CREO Software are converted in to IGES file and this IGES file are imported in ANSYS.In step 2 the meshing of this cooling tower model isdone. In meshing CFD mesh type is selected and finemeshing is done by using ten node tetrahedral elements.The reason for selecting this element is that is gives thegood meshing on curvature parts here the ANSYS isautomatically select the element.After meshing the boundary condition is shown in fig 2.In step 3 various domains is define. Here there are two domain are define. Domain 1is for wall. The domain 2 is airs domain. In boundary condition the inlet air velocity3m/s, outlet air massflowrate50kg/sec, Post processor result for normal wind flow through cooling tower of ANSYS is shown in Fig 3below.

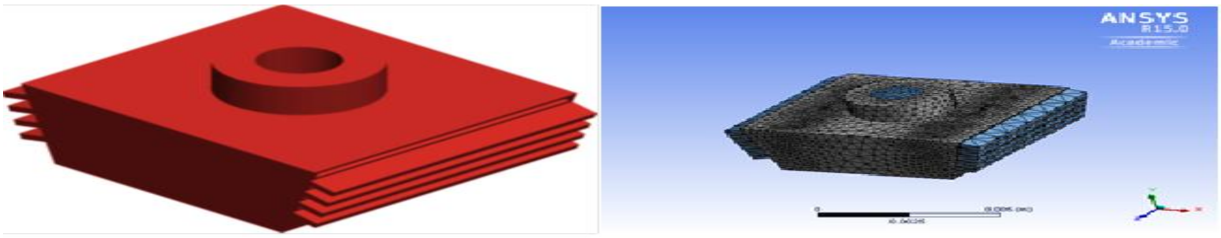


Fig 1 Cooling tower model in Pro-e and Meshing detail of cooling tower

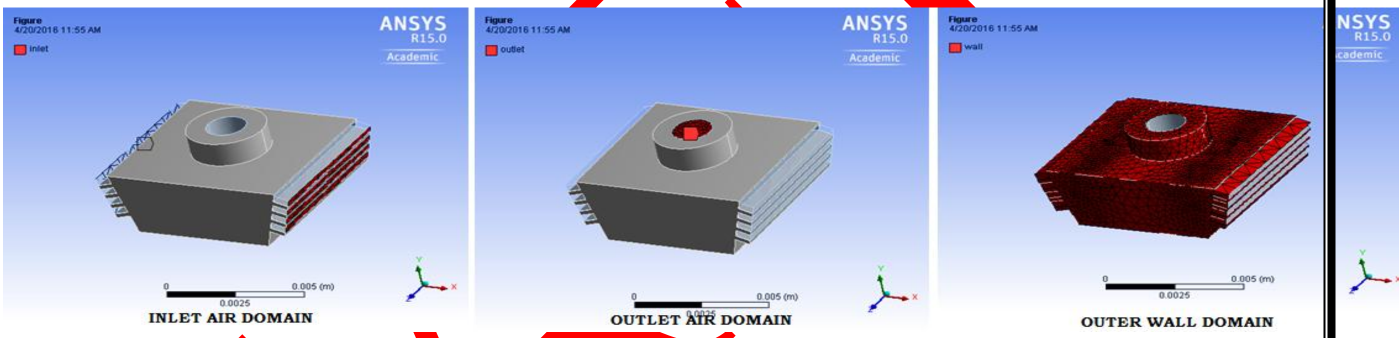


Fig 2Inlet Air domain, Outlet Air domain, Outer wall domain

From the result contour the velocity increases due to the induced draught fan. In ANSYS the outlet condition was given as induced draught fan and its mass flow rate was fixed as 50 Kg/sec. This value was selected from the site measured value. The air flow converged to that center of the fan so the hot water near the sides of the wall not get completely contact with the air due to this convergence. So Design the cooling tower with evenly distributed air system was necessary it's beyond the scope of this paper. Here only discuss about the increase of inlet air velocity by spherical shapes near the louvers. So next find the increase in velocity when the air passes through the spherical surface. This result is shown in the Figure 4.From the analysis of flow over the spherical surface the velocity of the air increases double when the air come out from the spherical surface. If the spherical surface place near the louvers of the cooling tower the velocity increases in some times the air come after the spherical sphere surface will interact with the louvers due to this interaction some turbulence create near the inner side of the louvers so that also one advantage because if turbulence creates it will enhance the heat transfer.

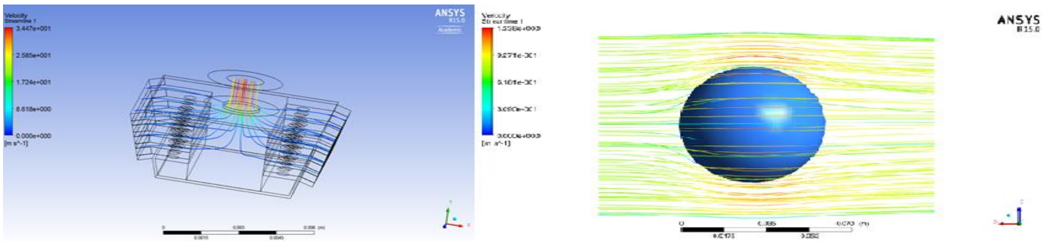


Figure 3 Normal wind flows through cooling tower, wind flow over the spherical surface

RESULTS

The results of the Theoretical analysis of water path cooling are shown in figure 4 & 5.Therefore using water path the approach will reduce and the effectiveness also reduced. At the same time, percentage of loss and outlet water temperature will reduce. If we take the range from water path means Range and effectiveness also increases.

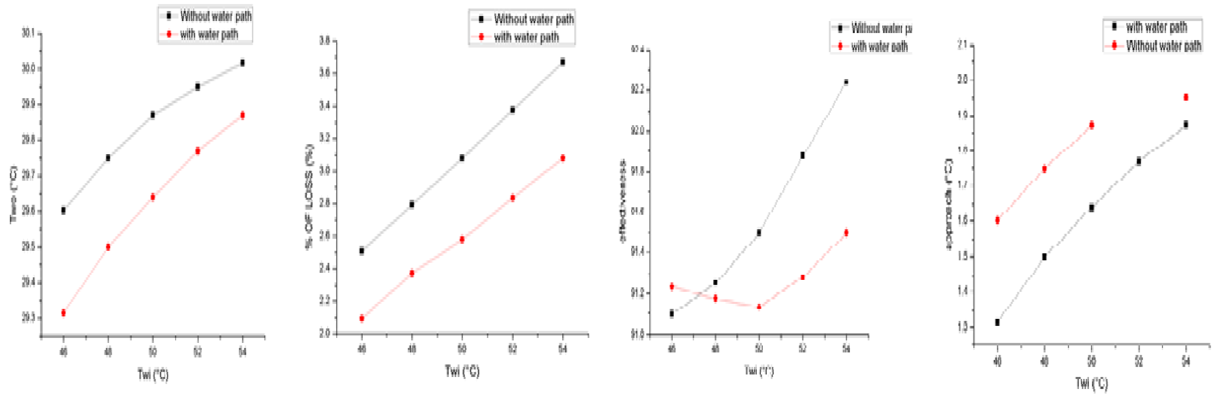


Figure 4 Inlet water temperature Vs Outlet water temperatures, % of loss,Effectiveness, Approach

CONCLUSION

Therefore using water path the approach will reduce and the effectiveness also reduced. At the same time, percentage of loss and outlet water temperature will reduce.If we take the range from water path means Range and effectiveness also increases. Placing the spherical surface objects near the cooling tower louvers will increase the inlet air velocity due to increase in air velocity L/G ratio will be reduced due to this the temperature of water from the cross flow cooling tower will be reduced.

REFERENCES

[01] R. Kirar, D. Vaghela, H. Patel, “Optimization of Air Inlet For Increase The Effectiveness Of Induced Draught Cross Flow Cooling tower”, International Journal of Engineering Research and Technology 2013;vol 2:1-4.

[02] R. Shakeri, E. Hajidavalloo, M. A. Mehrabian,“Thermal performance of cross flow cooling towers in variable wet bulb temperature”, *International Journal of Energy Conversion and Management*,vol. 51, no. 6, pp. 1298-1303, 2010.J. Clerk Maxwell, A Treatise on Electricity and Magnetism, 3rd ed., vol. 2. Oxford: Clarendon, 1892, pp.68–73.

[03] T. Mahmud, Md. K. Islam,B.Salam, “Experimental study of forced draft cross flow wet cooling tower using splash type fill”, International conference on MechEng and Renewable Energy (ICMERE)2013.

[04] Pushpa B. S, VasantVaze, P. T. Nimbalkar, “*Performance Evaluation Of Cooling Tower In Thermal Power Plant - A Case Study Of Rtps,Karnataka*”, International Journal Of Engineering And Advanced Technology , vol.4, no.2, pp.110-114,December 2014.

[05] Kanteyya a and Kiran Kumar Rokhade, “ Performance Analysis for Natural Draught cooling tower and Chimney through Numerical Simulation” Karnataka, vol.4, Issue 3, pp.1040-1045, March 2015.

[06] H.Irtaza, S. Ahmadand T.pandey,”2D study of wind forces around multiple cooling towers using computational fluid dynamics”Aligarh , Vol.3,No.6, pp 116-134,2011.

[07] Alok sing and SPS Rajput “Application of CFD in natural draft wet cooling tower flow”Bhopal,Vol.2,Issue1,pp.1050-1056 Jan-Feb 2012.

[08] Prof.Yogeshparkthi,DilipVaghela and Jitendraprajapati”CFD analysis of Induced draught flow cooling tower “ Volume 3,Issue3,Marchpp 791-793,2013.

[09] S.ParimalaMurugaveni and P. Mohamed shameer,”Analysis of forced draught cooling tower performance using ANSYS FLUENT software “,Coimbatore , Vol.4,Issue4,pp 217-229 Apr-2015.

[10] AnkitRameshraoThakre , AkashDipakKewate and ponraj K.Bhoyar,”Thermal and Computational fluid flow analysis of natural draught cooling tower using ANSYS 11.0”,pp-228-233.