Comparative Analysis of Forced Draft Cooling Tower Using Two Design Methods

Neetish Kumar Sao*, Dr. Surendra K. Dwivedi**

* PG Student, Department of Mechanical Engineering, O. P. Jindal University, Raigarh.
Email: neetishsao@gmail.com
**Associate Professor, Department of Mechanical Engineering, O. P. Jindal University, Raigarh.
Email: surendra.dwiwedi@opju.ac.in

Abstract: The cooling towers are widely used in industrial applications which reduces the temperature of water steam. The current research investigates 2 different design of cooling tower using techniques of Computational Fluid Dynamics. The CAD model of cooling tower is developed in Creo design software and CFD analysis is conducted using ANSYS CFX software. Standard k-epsilon turbulence model is used which gave reasonably good fluid flow predictions. The temperature distribution and velocity plots of both designs are obtained and comparative analysis is performed.

Key Words: Cooling tower, CFD, thermal analysis

1. INTRODUCTION

A cooling tower is a type of heat exchanger used to reduce the temperature of a water stream by extracting heat from water and emitting it to the atmosphere. Cooling towers use the evaporation of water to remove process heat and cool the working fluid to near the wet-bulb air temperature. Cooling towers are able to lower the water temperatures more than devices that use only air to reject heat, like the radiator in a car, and are therefore more cost-effective and energy efficient. They are generally used in HVAC application. There are many types of cooling tower available. The forced draft cross flow and counter flow cooling tower are the most common ones used in HVAC application.

2. LITERATURE REVIEW

Priyanka G, M. R. Nagraj [1] has conducted CFD analysis of cooling tower using ANSYS 13 software. The analysis involved different fluid types. The comparative analysis of different fluids are conducted on the basis of temperature and pressure plots.

Mohd Amir, Fithry, Yusoff, MohdZamri [2] has conducted investigation on cross flow cooling tower having porous media in the vicinity of single-phase flow. The air flow distribution inside fill is analysed. The high-pressure drop is induced inside cooling tower due to porosity.

Ramzi R. Ibraheem , Sherzad N. Muhammed [3] has experimental investigation on forced draft cooling tower located in Erbil. The NTU units increased with increase in water to air flow ratio along with increase in temperature, humidity and vapor pressure.

Dr. Jalal M. Jalil, Dr.Talib K.Murtadha, Dr. Qasim S. Mehdi [4] has conducted investigation on open type forced draft cooling tower using numerical and experimental techniques. The findings have shown an increase in effectiveness with higher air flow rates.

Dr. D. Al. D.H. Alwan Dr. I. W. Maid A. H. Soheel [5] has investigated forced draft counter flow cooling tower having wire mesh with small square holes (WMSSHSP) and wire mesh with diamond holes configuration (EWMDHSP). The findings have shown that performance of cooling tower is enhanced using (EWMDHSP) which is due to higher pressure drop as compared to (WMSSHSP).

3. OBJECTIVES

The current research investigates the new design configuration of cooling tower using techniques of Computational Fluid Dynamics to improve the effectiveness of cooling tower. The 1st design of cooling tower has single air inlet and 2nd design of cooling tower has 3 air inlets.

4. METHODOLOGY

The CAD model of cooling tower is developed in Creo design software. The individual arts are 1st modelled which are assembled later in assembly module. The CAD model of cooling tower is shown in figure 1 below.
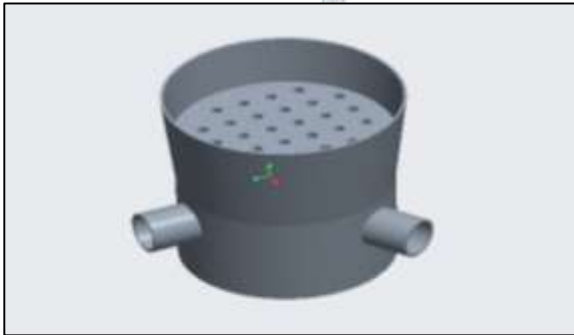
Figure 1: CAD model of cooling tower

The dimensions of cooling tower are taken from literature [6]. The CAD model is then imported in ANSYS design modeler and checked for geometric errors like hard edges, curvatures etc as shown in figure 2 below.

Figure 2: Imported CAD model in ANSYS

Figure 3: Meshed model in ANSYS

The transition ratio is set to 0.77, maximum layer set to 5 and growth rate set to 1.2. The number of nodes generated is 294035 and number of elements generated is 465601.
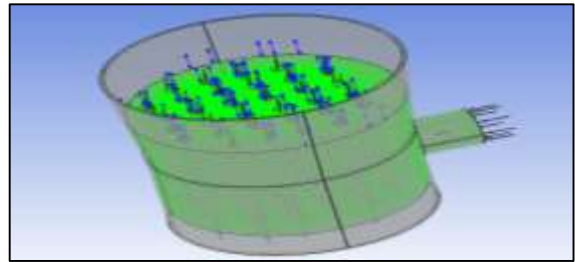

Figure 4: Boundary condition

The reference pressure is set to 1atm and domain type is set to fluid type. The energy model is set to thermal energy and heat transfer set to total energy. The turbulence model is set to 2 variable k-epsilon

5. RESULTS AND DISCUSSION

The CFD simulation is conducted on cooling tower to determine temperature plot, velocity plot. The temperature plot of cooling tower shows high temperature on zones near inlet as shown in red color. The temperature at the inlet is 329K which reduces as water moves towards exit.
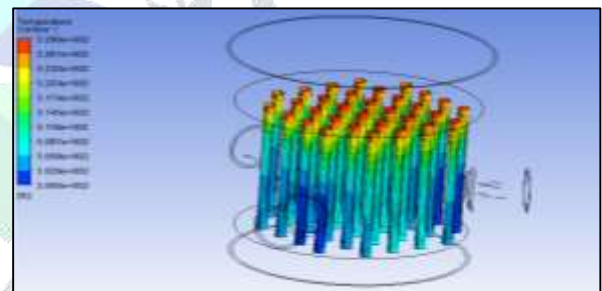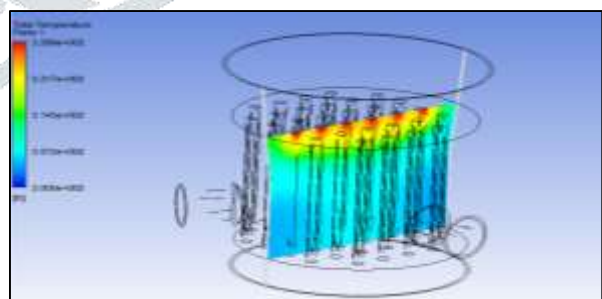

Figure 5: Temperature plot of water domain

Figure 6: Temperature plot of air along plane

The temperature distribution across plane shows the zones near inlet with temperature of 314.5K which further reduces downwards and reaches to 307.2K. The corner regions has lowest temperature of nearly 301 K.
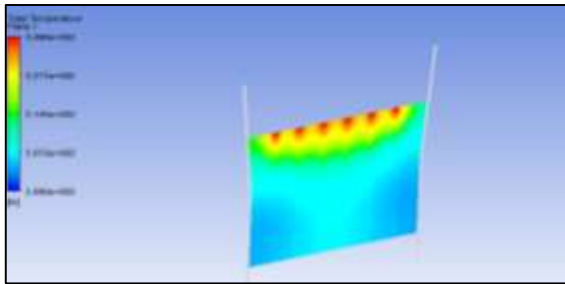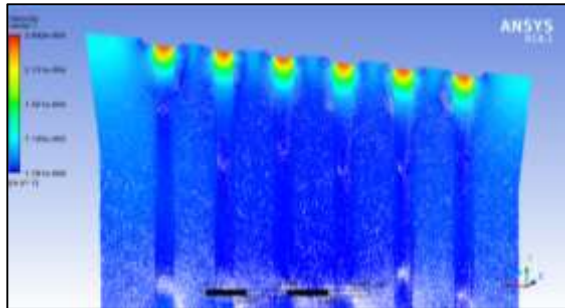
Figure 7: Temperature plot of air along plane

Figure 8: Velocity vector plot of longitudinal plane

The velocity vector plot is obtained across plane as shown in figure 8 above. The velocity near the inlet is maximum with the magnitude of .000284m/s which decreases gradually downwards as shown in dark blue color.
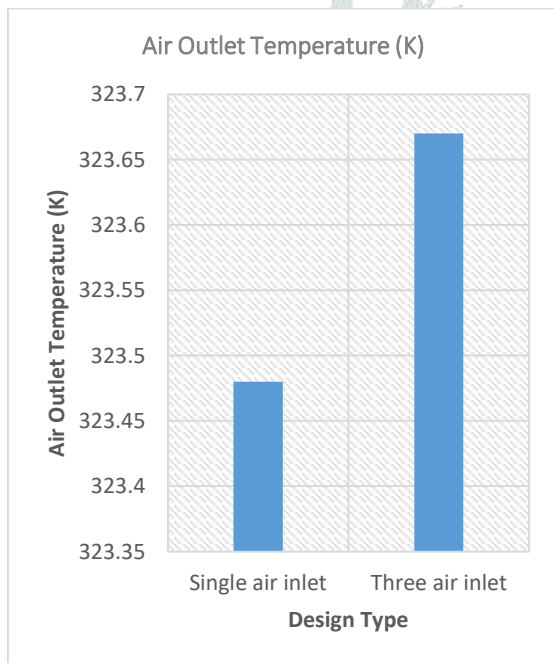

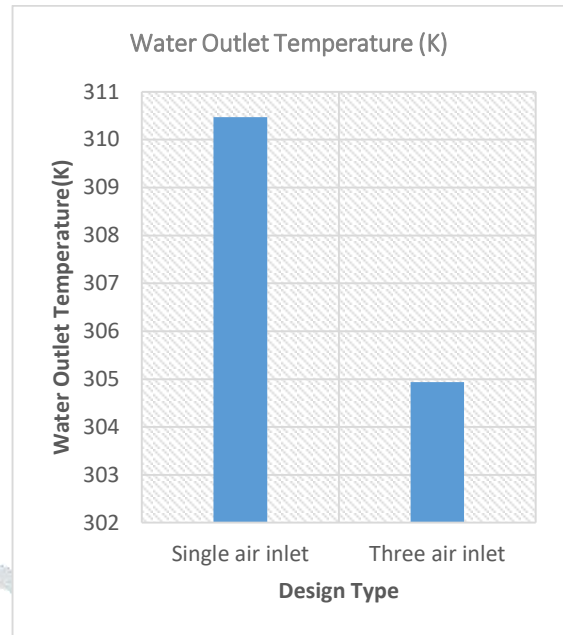Figure 9: Air outlet temperature comparison

Figure 10: Water outlet temperature comparison

The temperature comparison of air outlet in figure 9 shows high outlet temperature with 3 air inlets. The water outlet temperature for 3 air inlet has lower temperature as compared to single air inlet design.

6. CONCLUSION

The CFD simulation is viable tool in determining thermal characteristics of cooling tower. The CFD results have shown that three air inlet design configurations have better results as compared to single air inlet design configuration. The three air inlet design configuration has reduced water temperature by nearly 25^0 C while single fuel inlet design reduced water temperature by nearly 19^0 C.

REFERENCES

[1] Priyanka G, M. R. Nagraj, "CFD Analysis of Shell and Tube Heat Exchanger With and Without Fins for Waste Heat Recovery Applications", International Journal of Science and Research (IJSR) ISSN: 23197064 (Online).

[2] Mohd Amir, Fithry, Yusoff, MohdZamri, "Numerical Investigation of a Single Phase Air Flow inside Porous Media of a Cross Flow Cooling Tower", International Journal of Science and Research (IJSR), ISSN: 2319-7064 (Online).

[3] Ramzi R. Ibraheem , Sherzad N. Muhammed, "Study Performance of Forced Draft Counter Flow Cooling Tower", International Journal of Advances in Engineering Science and Technology ISSN: 23191120 (Journal).

[4] Dr. Jalal M. Jalil, Dr.Talib K.Murtadha, Dr. Qasim S. Mehdi, "CFD Prediction of Forced Draft CounterFlow Cooling Tower Performance", Eng. &Tech. Journal, Vol.28, No.11, 2010. (Journal)

[5] Dr. D. Al. D.H. Alwan Dr. I. W. Maid A. H. Soheel, "Numerical and Experimental Study of Counter Flow Cooling Tower Performance with Difference Packs Porosity and Configuration", Alwan Technical Report at Baghdad. (Technical Report)

[6] S. Parimala Murugaveni, ANALYSIS OF FORCED DRAFT COOLING TOWER PERFORMANCE USING ANSYS FLUENT SOFTWARE, IJRET: International Journal of Research in Engineering and Technology, eISSN: 2319-1163 | pISSN: 2321-7308