Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

Thermal and Numerical Analysis of Wet Cooling Tower for Shore based Naval Marine Boiler

Davinder Singh Grover[1]
[1] PG Student,
Department of Mechanical Engineering,
Defence Institute of Advanced Technology, Girinagar,
Pune, Maharashtra- 411025

Pravin Hadgekar[2]
[2] Assistant Professor,
Department of Mechanical Engineering,
Defence Institute of Advanced Technology,
Girinagar, Pune, Maharashtra- 411025

***Abstract*:- A cooling tower is a direct contact atmospheric cooling heat exchanger that is widely used in power plants, HVAC applications and other process industries. It is mainly used for cooling of fluid streams using dry atmospheric air. It works on the principle of evaporative cooling, wherein the latent heat of evaporation is supplied by the fluid stream itself, thereby resulting in cooling.**

In this case of a shore based marine boiler, there is a requirement to cool the water being utilised to condense the steam in the steam condenser and recirculate it back into the system due to limited supply of water. The cooling tower is the most efficient and economical heat exchanger for this purpose. This research mainly focuses on the thermal design of a wet induced draught cooling tower for the given heat load using Merkel's Cooling Tower theory.

Thereafter, the flow analysis of air through a tower column has been undertaken using CFD. The main objective of using CFD is to justify the use of the analytical relationships in Merkel's theory by studying uniformity in air flow pattern. Also, the tower model used is validated by comparing the pressure drop in the tower column obtained through CFD against the experimental data provided by M/s Advanced Cooling Towers.

LIST OF ACRONYMS

CFCs - Chloro Fluoro Carbons
CFD - Computational Fluid Dynamics

NDWCT - Natural Draught Wet Cooling Tower
TR - Tons of Refrigeration
NTU - Number of Transfer Units
TPH - Tons per hour
RH - Relative Humidity
Deg C - Degree Celcius
HVAC - Heating, ventilation and air conditioning

LIST OF SYMBOLS

Λ - Specific latent heat of evaporation of water
α - Film heat transfer coefficient
A - Superficial surface area of drop
t - Bulk air dry bulb temperature
t - Temperature
K_g - Diffusion coefficient of the film
x - Absolute humidity
Q - Total heat transfer (KW)
K - Overall enthalpy transfer coefficient
S - Heat transfer surface area
h - Enthalpy
C_w - Specific heat of water
L - Water mass flow rate
G - Air mass flow rate
D_i - Viscous resistance coefficient
C_i - Inertial Resistance coefficient

1 INTRODUCTION

A shore based training boiler similar to boilers used on naval ships is to be installed for training. Since the boiler is a shore based one and is likely to be operated at low loads due to the absence of steam auxiliaries as well as the steam driven main propulsion system. In ship based boilers, the exhaust steam from steam driven auxiliaries and main engines is rejected to a steam condenser, where the steam is condensed and then re-circulated back into the system as feed water. The cooling fluid used to condense the steam is sea water. Since, sea water is abundantly available at sea, the sea water is continuously discharged back into the sea at the condenser exit and fresh sea water is pumped into the condenser inlet. However, at a shore based facility, due to limited availability of cooling water, it is not possible to continuously discharge water from the condenser outlet. This cooling water needs to be recycled back into the system. Here arises the requirement of a cooling tower. The cooling tower will cool the cooling water from the steam condenser outlet to a suitable lower temperature and then it will be pumped back again into the condenser inlet.

A cooling tower is a special type of heat exchanger in which the water to be cooled and atmospheric air are in direct contact with each other. The mode of cooling is '**evaporative cooling'**. This method of cooling ensures that the contact area between the warm water and air is very large. Further, it also ensures a high mass transfer co-efficient of water vapour while providing minimum resistance to the flow of air, thereby resulting in low pressure drop. The other advantage is that this mode of cooling results in water temperatures below the ambient air dry bulb temperature and these are highly beneficial in

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

industry and air conditioning systems. No other cooling method, apart from refrigeration, can achieve these temperature levels. However, the cost of refrigeration systems is exorbitant as compared to cooling towers and they consume large amounts of electric power for the desired cooling effect. Further, their use of CFCs has an adverse effect on the ozone layer and contributes greatly towards global warming. Therefore, cooling towers are the heat exchangers of choice in power plants, Air conditioning and ventilation applications and process industries to name a few.

The design and manufacture of cooling towers is based on the following principal criteria:-

(i) To maximise the area of contact between the water spray and the cooling air by optimal design of the packing fills and the water distribution system.

(ii) To provide forced draft of air by use of fans.

(iii) To minimise water losses due to water spray escaping the body of the tower by use of drift eliminators. Controlling the water spray loss also reduces the risk of transmission of bacterial diseases by the warm moist air.

(iv) To relate the cooling tower design to the critical flow parameters, i.e. the volume flow rate of the water to be cooled and to the three important temperatures, i.e. ambient air wet bulb, warm water inlet and desired cooled water outlet.

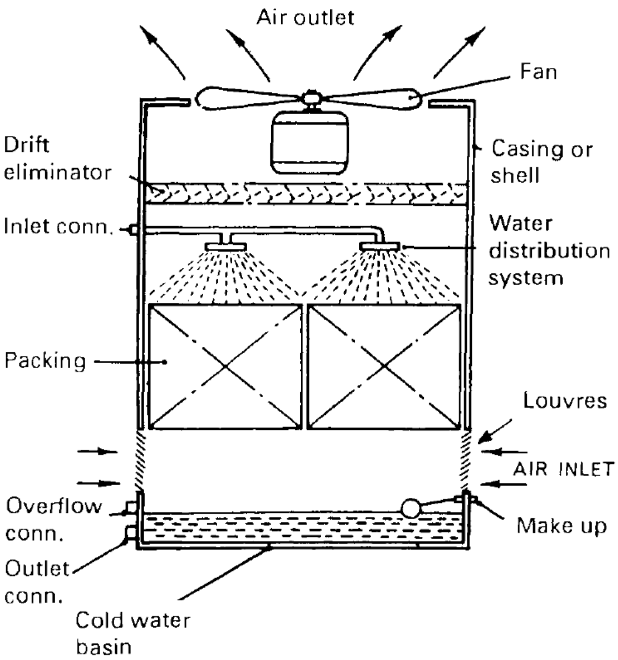

Figure 1 - Basic Cooling Tower Configuration [11]

In this research, it is proposed to carry out thermal design of a cooling tower analytically arrive at the tower demand(NTU), for known mass flow rates of hot water as well as desired outlet temperatures at tower exit. Further, it is proposed to study the air flow through the fills region which will be modelled as a porous zone in CFD and validate results against experimental data.

1.1 Problem Statement

To design a cooling tower for 1000 TPH static training boiler. The design hot water inlet mass flow rate has been estimated at 1000TPH, with an inlet temperature of 50 deg C. The ambient air conditions are 28 deg C and RH of 75%. The water is desired to be cooled to an outlet temperature of 35 deg C.

1.2 Determination of type of cooling tower

Determination of the type of cooling tower to be designed has been carried out based on a study of different types of cooling towers and their advantages and disadvantages vis-a-vis each other. Also, since a cooling tower is a widely used, commercial product, due consideration was given to types of cooling towers used widely for this type of application and heat load. The classification of cooling towers based and their merits/de-merits are as follows:-

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

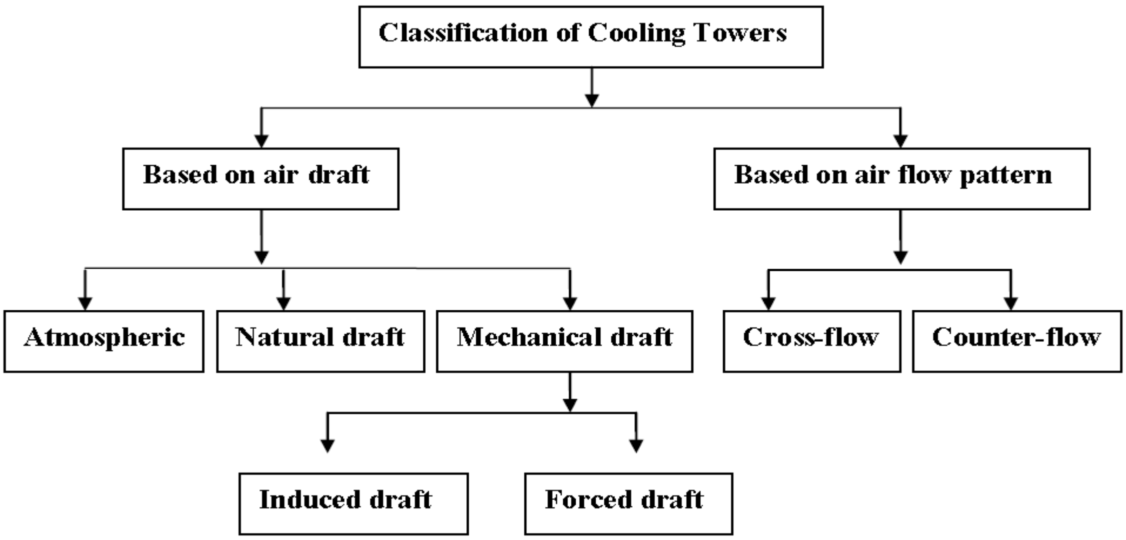

Fig 2 - Classification of cooling towers [11]

(i) Natural Draught Cooling Towers

Natural flow of air occurs through the tower due to natural convection currents; hence it is called natural draft. They are generally hyperbolic in shape and constructed of concrete material. Due to the natural flow of air, these towers are required to be very tall in order to achieve the desired outlet temperature and are generally used in thermal and nuclear power plants. Generally not considered for industrial and HVAC applications.

Factors responsible for creating natural draft:-

(a) Increase in temperature and humidity of air in the tower reduces its density and causes it to rise upwards in the column.

(b) Wind velocity at the tower bottom.

(ii) Mechanical Draft Towers: Forced Draught Towers and Induced Draught towers

Fans are used to circulate air through the tower in mechanical draft cooling towers. Two types of mechanical draft towers are there, namely, forced draught towers and induced draft towers.

(a) Forced Draught Towers

In these types of towers, a fan/blower is installed at the lower end of the tower and air which forces air through the tower. This results high velocities at the bottom and low velocity at the top, which increases the likelihood of air re-circulating back into the tower.

(b) Induced draught Towers

In these towers, a fan is installed at the top which draws in air from the bottom.

1.3 Counterflow Induced Draught Cooling Tower

In this research, a counterflow induced draught cooling tower has been selected due to the following reasons:-

(i) In an induced draught tower, the exit velocity of air stream is high and therefore, there is less risk of recirculation in which the exit air flows back into the air inlet.

(ii) Relatively dry air comes in contact with the cold water at the bottom of the cooling tower.

(iii) Moist air is in contact with the warm water at the tower top and hence the average driving force for both heat and mass transfer is maximum. This results in higher tower efficiency.

2 EVAPORATIVE COOLING FUNDAMENTALS

A finite amount of energy needs to be supplied to water in order to change its state from liquid to vapour or steam. This energy is called its latent heat of vaporisation. This energy can be supplied in the form of heat by combustion of fuel or it can also be extracted from the surroundings. In a cooling tower also, a change of state takes place, wherein liquid water evaporates in a stream of moving air. The latent heat of evaporation required is extracted from the hot water itself, resulting in the cooling of the water stream and heating of the air stream. This process is known as evaporative cooling. The principle itself is very simple; however, the heat transfer mechanisms involved in the process are very complex.

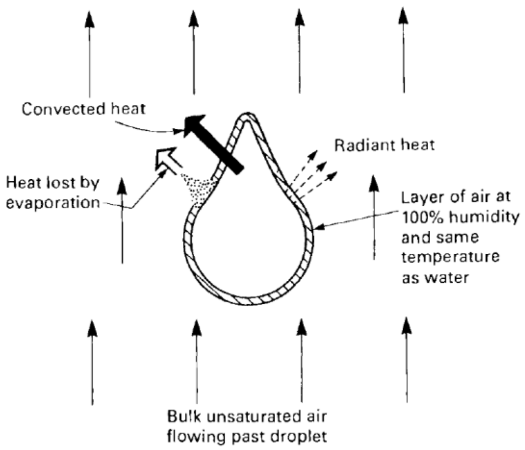

Figure 3 - Mode of Heat Transfer in a Cooling Tower [11]

Fig 3 above shows a single droplet of water in the tower. This droplet is surrounded by a thin film of saturated air and is not affected at all by the passing air stream. Heat transfer occurs through this static film of saturated air by the following three mechanisms:-

(i) By radiation from the surface of the droplet. This forms a very small percentage of the total heat transfer and can be neglected.

(ii) By conduction and convection between water and air. The amount of heat transfer depends on the air and water temperature. This mechanism of heat transfer accounts for almost one third of the total heat transfer and therefore forms a significant percentage.

(iii) By evaporation. This mechanism accounts for almost two thirds of the entire heat transferred. Thus this process is called evaporative cooling.

When water and air in contact with each other, evaporation occurs. This evaporation is due to the difference in vapour pressures at the water surface and in the air. The vapour pressure is the pressure exerted by the water vapour and it depends on water temperature and saturation of air. In a cooling tower, the air and water streams are moving in opposite directions, such that the cool water at the tower bottom is in contact with unsaturated air and warm water at the top is in contact with warm, moist air. However, evaporation is taking place continuously throughout the tower column. At the top of the tower, the air is warm and moist. This is compensated by the high temperature of water which results in a higher vapour pressure at the water surface, thereby providing sufficient gradient for evaporation. The amount of evaporation depends on a number of factors. The design of the fills region is an important aspect and it decides the area of contact between the water droplets and air. Further, increase in the air mass flow rate also improves cooling achieved, because greater the air flow rate, lesser will be the effect of water on its temperature and humidity and partial pressure differences throughout the pack will increase. Also a lower wet bulb temperature at tower inlet results in lower outlet water temperature. The factors which influence a cooling tower performance are as follows:-

(i) The cooling range - The difference between the water inlet and outlet temperature.

(ii) Tower approach - The difference water outlet temperature and inlet wet bulb temperature.

(iii) Ambient wet bulb temperature.

(iv) The flow rate of water to be cooled.

(v) The air flow rate.

(vi) The performance coefficients of packing used.

(vii) The volume of the packing (i.e. the packing height multiplied by its cross sectional area).

2.1 Heat transfer theory applied to cooling towers

There are two fluids involved in the energy transfer process in cooling towers, viz water which enters at high temperature and exits at a lower temperature and air which exits the tower in an almost saturated condition. The main source of heat transfer is through evaporation of water vapour from the water stream into the air stream, i.e. the latent heat of evaporation and this is extracted from the warm water stream as it flows through the tower. Almost two thirds of the total energy transferred is by the diffusion of water vapour from the water into the air through the interface between them. This is then distributed into the air mass by further convection. The remaining energy transfer occurs by convection and conduction between the air and water. Thus energy transfer takes place through two ways, i.e. mass transfer (75%) and heat transfer (25%). Consider the water droplet in Fig 3. This droplet is surrounded by a thin film of static and saturated air. Water vapour diffuses through this layer at a rate of W kG/s. Consider that the specific latent heat of evaporation of water is λ KJ/Kg.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

Then the rate of diffusion is given by W x λ kW. Due to the cooling of the water stream, sensible heat will flow from the air stream into the water stream at a rate Q kW. The wet bulb temperature is achieved when there is equilibrium between the heat of evaporation and sensible heat transfer, i.e.

$$\mathbf{Q = W \times \lambda} \qquad (2.1)$$

From heat transfer theory, we know that

$$\mathbf{Q = \alpha \times A \times (t - t_i)} \qquad (2.2)$$

Q = sensible heat flow rate (kW)
α = film heat transfer coefficient (kW/m^{2}K)
A = superficial area of drop (m^2)
t = bulk air dry bulb temperature
t_i = temperature at the interface between water and air

The rate at which the water vapour is diffusing into the air stream can be shown in terms of the absolute humidities

$$\mathbf{W = K_g \times A \times (x_i - x_g)} \qquad (2.3)$$

K_g = diffusion coefficient of the film(kg/m^{2}s)
x_i = absolute humidity at the interface (kg/kg)
x_g = absolute humidity in the bulk air (kg/kg)

2.2 The Approach to Practical Cooling Tower Calculations

In order to understand the mechanism of heat transfer in a cooling tower, let us consider a counter flow tower in which warm water enters at the top and ambient air is either induced or forced through the tower in opposite direction to the water flow.

At the top of the tower, the temperature of water is above the dry bulb temperature of bulk air stream. However, as the water travels down through the column, its temperature will fall below the dry bulb temperature of the air, but still remains above the wet bulb temperature. These two sets of conditions are elaborated further with the help of Figs 4 and 5 below. As the water falls further, it will approach the wet bulb temperature, but will not equal it. For this to happen, the tower must have infinite height.

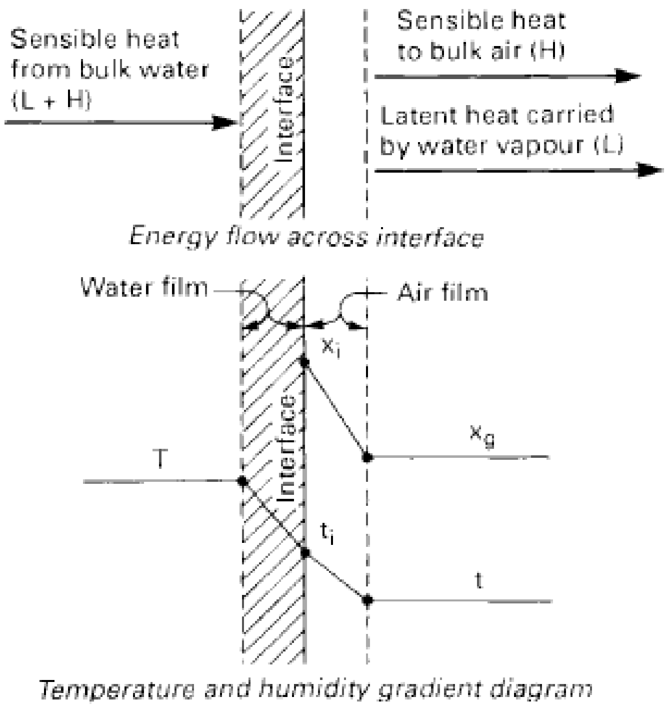

Fig 4 – Temperature Humidity Gradient (T > DBT) [11]

Fig 4 above shows the temperature and humidity gradients across the interface of water film and air film for the condition where the water temperature is higher than the dry bulb temperature of the air stream. The interface temperature (t_i) is lower than the temperature (T) of the bulk of the water, but is still higher than the temperature of bulk of air(t). Thus there is sensible heat transfer (H) from the water to the air due to temperature difference (T-t) and latent heat transfer (L) due to diffusion of water vapour across the interface.

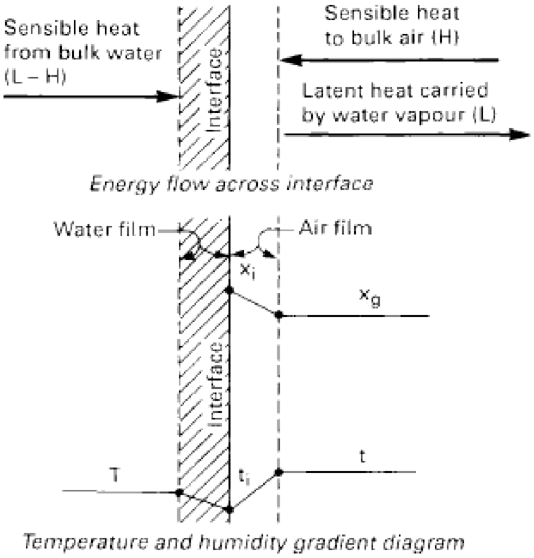

Fig 5– Temperature Humidity Gradient (T< DBT) [11]

Fig 5 above shows the temperature and humidity gradients across the interface of water film and air film for the condition where the water temperature is lower than the dry bulb temperature of the air stream. In this case, there are two sources of heat at the interface, due to temperature difference (t-t_i) denoted by (L-H) and one due to temperature difference (t-t_i) denoted by H. These two sources are together equal to the latent heat L due to diffusion of water vapour from the interface into the air stream. In both of the above cases, the absolute humidity at the air-water film interface (x_i) is always higher than the absolute humidity of the air stream (x_g) and this difference in absolute humidities is the driving potential for diffusion of water vapour and consequent transfer of latent heat to the air. The net transfer of heat into the air will gradually decrease as the water falls further through the tower. At the tower bottom, the driving potential will be minimum and sensible heat(H) will almost be equal to latent heat(L) so that total energy transfer is minimum and sensible heat *(H)* will almost equal latent heat (L) so that total energy transfer is also at a minimum.

2.3 Overall heat transfer and volumetric coefficients

Sensible heat transfer and latent heat transfer in the form of diffusion from the water air depend on the air film coefficients α and K_g (see equations 2.2 and 2.3) and also on the heat transfer coefficient of the water film. Since it is not possible to determine the actual interface temperature as well as the contact area between water and air film experimentally, it is not practical to assign any values to both these coefficients. As a result, overall coefficients are used, i.e. an overall heat transfer coefficient based on the temperature difference between water and air (T - t) and an overall diffusion coefficient based on the humidity difference (x_w - x_g), where x_w is the saturation humidity corresponding to the water temperature T, and x_g is the absolute humidity of the bulk air. These two coefficients are further modified such that they refer to a unit volume of the cooling tower packing. These coefficients are then known volumetric coefficients.

2.4 Merkel's Theory

Many theories have been developed in order to describe the heat and mass transfer which takes place in several types of atmospheric water cooling devices. The cooling tower is considered as a heat exchanger in which water and air are in direct contact with one another. As stated above, it is not experimentally possible to calculate the area of contact between the air and water films accurately. Therefore, a definite value cannot be assigned to a heat transfer coefficient. The process is further complicated by mass transfer, due to which we cannot accurately assign a value to diffusion coefficient.

The dual problem of heat and mass transfer has been overcome by the Merkel theory by combining the two into a single process based on enthalpy potential. This theory states that mass and energy flow from the bulk of water to an interface and from the interface to the surrounding bulk of air. Each boundary offers resistance to the flow of mass and heat which results in a gradient in temperature, enthalpy, and absolute humidity. Merkel demonstrated that the total heat transfer is directly proportional to the difference between the enthalpy of saturated air at the bulk water temperature and the enthalpy of air at the point of contact with water. Therefore,

$$\mathbf{Q = K \times S \times (h_w - h_a)} \quad (2.4)$$

where,

Q = Total heat transfer (KW)

K = Overall enthalpy transfer coefficient (Kg/s.m^2)

S = Heat transfer surface (m^2). S equals to a x V, which "a" means area of transfer surface per unit of tower volume. (m^2/m^3), and V means an effective tower volume m^3).

$\mathbf{h_w}$ = enthalpy of air-water vapor mixture at the bulk water temperature, KJ/Kg dry air

$\mathbf{h_a}$ = enthalpy of air-water vapor mixture at the wet bulb temperature, KJ/Kg dry air

The water temperature and air enthalpy are continuously changing along the tower column and Merkel's relation can only be applied to a small element of heat transfer surface *dS*.

$$dQ = d[K \times S \times (h_w - h_a)] = K \times (h_w - h_a) \times dS \quad (2.5)$$

The heat transfer rate from water side is

$$Q = C_w \times L \times \text{Cooling Range} \quad (2.6)$$

where,
C_w = specific heat of water
L = water mass flow rate

Therefore,

$$dQ = d[Cw \times L \times (tw_2 - tw_1)] = Cw \times L \times dtw \quad (2.7)$$

The heat transfer rate from air side is

$$Q = G \times (ha_2 - ha_1) \quad (2.8)$$

Where,
G = air mass flow rate.

Therefore,

$$dQ = d[G \times (ha_2 - ha_1)] = G \times dh_a \quad (2.9)$$

$$\Rightarrow \quad K \times (h_w - h_a) \times dS = G \times dh_a$$

$$\Rightarrow \quad K \times (h_w - h_a) \times dS = C_w \times L \times dt_w$$

$$\Rightarrow \quad K \times dS = G / (h_w - h_a) \times dh_a$$

$$\Rightarrow \quad K \times dS / L = Cw /(h_w - h_a) \times dt_w$$

By Integration,

$$KxS/L = KaV/L = G/L \int dh /h_w\text{-}h_a = Cw \int dt_w/ h_w\text{-}h_a \quad (2.10)$$

This basic heat transfer equation is integrated using the four point Tchebycheff method.

For the evaluation of KaV/L,

$$KaV/L = C_w \int dtw/ h_w\text{-}h_a$$

$$= C_w(tw_2\text{-}tw_1) \times [(1/Dh_1)+(1/Dh_2)+(1/Dh_3)+(1/Dh_4)] / 4 \quad (2.11)$$

Where,
Dh_1 = value of (h_w-h_a) at a temperature of CWT+ (0.1 x Range)
Dh_2 = value of (h_w-h_a) at a temperature of CWT+ (0.4 x Range)
Dh_3 = value of (h_w-h_a) at a temperature of CWT+ (0.6 x Range)
Dh_4 = value of (h_w-h_a) at a temperature of CWT+ (0.9 x Range)

2.5 NTU Calculations

From equation 2.10 above, we have

$$NTU = KaV/L = G/L \int dh /h_w\text{-}h_a = Cw \int dtw/ h_w\text{-}h_a$$

The right side of the above equation is a dimensionless quantity. This quantity can be calculated by simply knowing the temperature of the flows entering the cooling tower. It is totally independent of tower size and fills configuration and is often called NTU. By plotting several values of NTU as a function of L/G gives what is known as the "Demand" curve. So, NTU is called Tower Demand too.

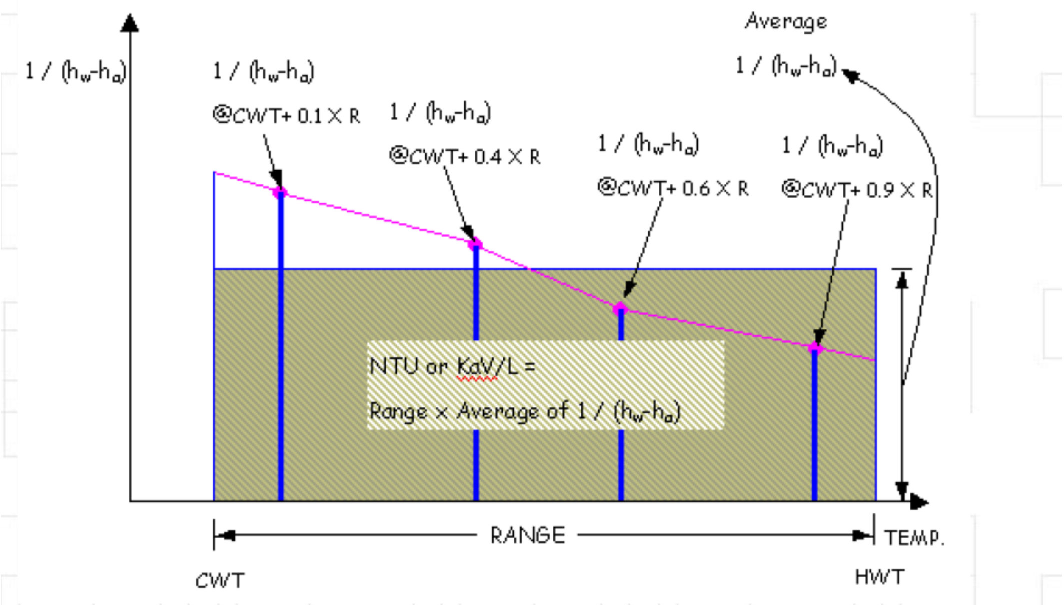

Fig 6 - Calculation of Tower Demand Graphically[12]

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

As shown in Fig 6 above, NTU is an area obtained by multiplying the cooling range by the average of $1/(h_w - h_a)$ at four different points(temperature) along the X- axis.

$$\textbf{NTU or KaV/L = Cooling Range x [Sum of } 1 / (h_w - h_a)] / 4 \qquad (2.12)$$

2.6 Heat Balance

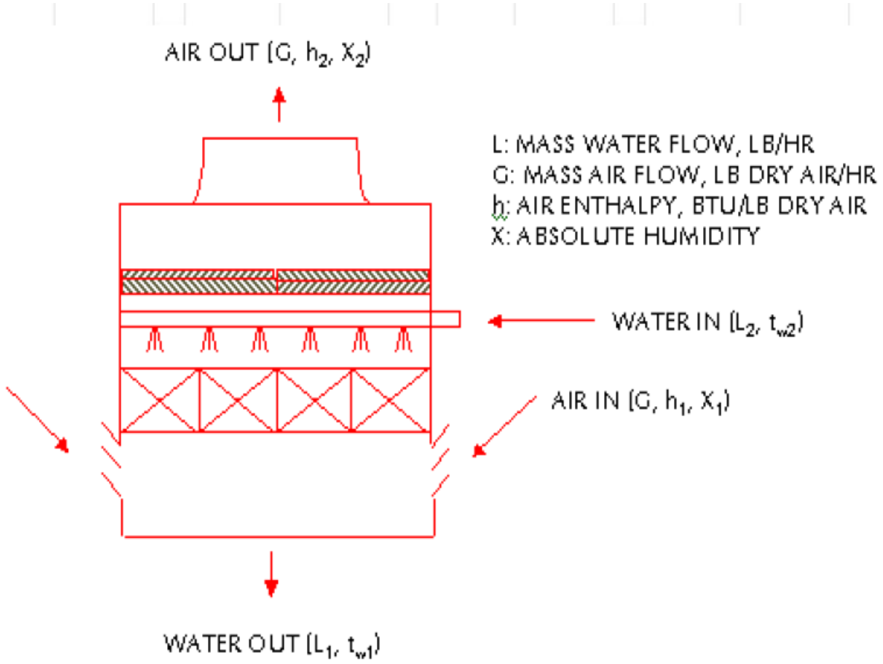

Figure 7 - Heat Balance [12]

$$HEAT_{in} = HEAT_{out}$$

$$WATER\ HEAT_{in} + AIR\ HEAT_{in} = WATER\ HEAT_{out} + AIR\ HEAT_{out}$$

$$C_w L_2 tw_2 + G\ ha_1 = C_w L_1 tw_1 + G\ ha_2 \qquad (2.13)$$

The difference between L2 (entering water flow rate) and L1 (leaving water flow rate) is the amount of water lost due to evaporation by diffusion of water vapour into the air stream. This evaporation loss is a result of difference in the water vapor content between the inlet air and exit air of cooling tower. Evaporation loss is expressed as:-

=> $$L1 = L2 - G \times (w_2 - w_1) \qquad (2.14)$$

Substituting value of L1 in equation 2.12, we get,

=> $$C_w L2\ tw_2 + G\ ha_1 = C_w [L2 - G \times (w_2 - w_1)] \times tw_1 + G\ ha_2$$

=> $$C_w \times L2 \times (tw_2 - tw_1) = G \times (ha_2 - ha_1) - C_w \times tw_1 \times G \times (w_2 - w_1)$$

The 2nd term of right side is ignored in order to simplify the calculation under the assumption of G x (w2 - w1) = 0(negligible evaporation loss).

=> $$Cw \times L \times (tw_2 - tw_1) = G \times (ha_2 - ha_1) \qquad (2.15)$$

Therefore, the enthalpy of exit air is

$$ha_2 = ha_1 + Cw \times L / G \times (tw_2 - tw_1)$$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

=> $$ha_2 = ha_1 + Cw \times L/G \times \text{ Cooling Range} \quad (2.16)$$

Consequently, the enthalpy of exit air is a summation of the enthalpy of entering air and the addition of enthalpy from water to air (this is a value of *Cw x* L/G x Range).

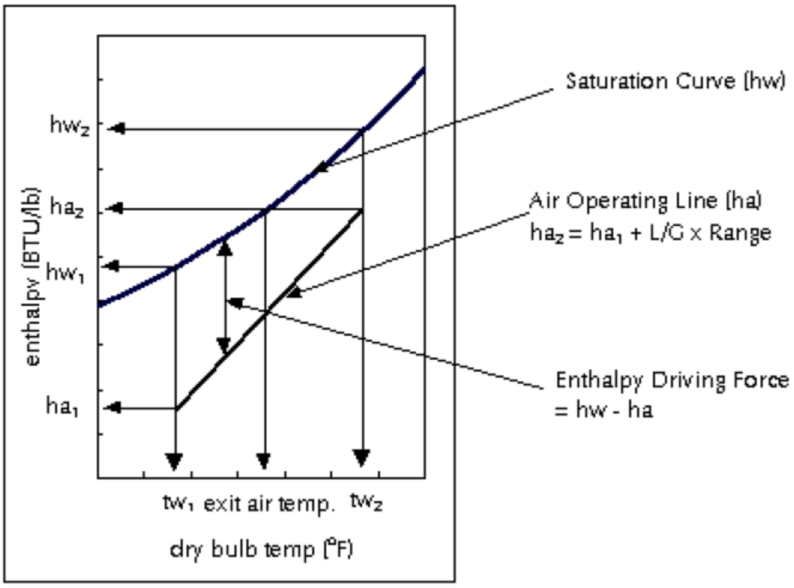

Figure 8 - Operating Air Curve and Saturated Curve [12]

In Fig 8 above, the operating air curve (h_a) is exactly same as a linear function of y = a + b x. The ha1 corresponds to "a", C_w X L/G corresponds to "b", and the cooling range corresponds to "x". So, C_w X L/G is a slope of the operating air curve (h_a).

2.7 Mass Balance

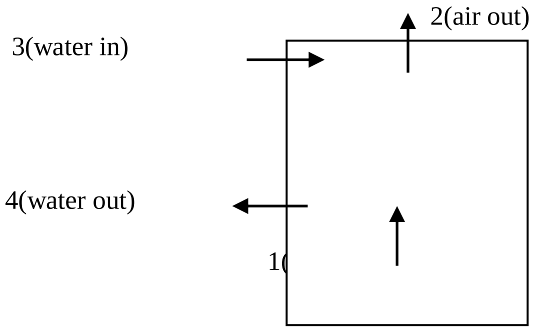

Fig 9 - Mass Balance

Dry air mass balance

$$m_{a1} = m_{a2} = m_a \quad (2.17)$$

Water mass balance

$$m_3 + m_{v1} = m_4 + m_{v2}$$

=> $$m_3 + w_1 m_{a1} = m_4 + w_2 m_{a2}$$

=> $$m3 - m4 = ma\{w2 - w1\} = m_{\text{make-up}} \quad (2.18)$$

Energy Balance

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

$$\Sigma(mh)_{in} = \Sigma(mh)_{out} \tag{2.19}$$

=> $$m_{a1}h_1 + m_3h_3 = m_{a2}h_2 + m_4h_4$$

=> $$m_3h_3 = m_a[h_2 - h_1] + m_4h_4$$

=> $$m_3 - m_4 = m_{make\text{-}up}$$

=> $$m_4 = m_3 - m_{make\text{-}up}$$

=> $$m_3h_3 = \frac{m_a[h_2\text{-}h_1] + m_{make\text{-}up} \times h_4}{h_2 - h_1}$$

=> $$m_a = \frac{m_3[h_3 - h_4]}{(h_2\text{-}h_1) - (w_2 - w_1)h_4} \tag{2.20}$$

2.8 Calculations

A few assumptions must be made whilst carrying out thermal calculations for a cooling tower. These include the desired outlet temperature as well as the ambient wet-bulb temperature. The following factors were used as guidelines while assuming the value of these parameters.

(i) Selection of desired outlet temperature

For a given ambient wet bulb temperature, the water outlet temperature has a significant influence on the size of the cooling tower. If we desire to cool the water to the wet bulb temperature, we will require a tower of infinite size. It is generally practical to have an approach of 4°C. More importantly, it is necessary to select a temperature that is suitable to the cooling operation for which the water is required. Selection of a temperature below that limit will only lead to an oversized and more expensive tower than what is actually required. Keeping this in mind, the required water outlet temperature was selected to be 35°C.

(ii) Choice of air wet bulb temperature

The choice of the design wet bulb temperature usually depends on the available meteorological data available for that region. However the meteorological data records the peak temperatures observed during the day or temperatures recorded at the same hour during the day. Further, the temporarily warm water at the tower exit is quickly cooled after mixing with the water in the static wet basin. Hence it is normal to design for a wet bulb temperature 3-5°C below the peak recorded temperatures.

(iii) The outlet stream of air is considered to be fully saturated at 36 deg C and is fully saturated(RH=100%).

2.8.1 Problem parameters

The following values/ parameters have been used for calculations:-

(i)	Hot water Inlet temp(T_3)	= 50 deg C
(ii)	Desired Hot Water Outlet temp(T_4)	= 35 deg C.
(iii)	Mass flow rate of water(m_w)	= 1000TPH=280Kg/s
(iii)	Enthalpy of inlet water (h_3)	= 209.34 KJ/Kg
(v)	Enthalpy of outlet water (h_4)	= 146.64 KJ/Kg
(vi)	Inlet air dry bulb temperature (T_1)	= 28 deg C
(vii)	Relative Humidity of inlet air	= 75%
(viii)	Specific humidity (w_1)	= 0.0174
(ix)	Inlet air enthalpy (h_1)	= 73.89 KJ/Kg
(x)	Specific volume (v_1)	= 0.876 m3/Kg
(xi)	Outlet air temperature (T_2)	= 36 deg C
(xii)	Outlet air relative humidity	= 100%
(xiii)	Outlet air specific humidity (w_2)	= 0.0388

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

2.8.2 Enthalpy of Outlet air

The enthalpy of outlet air stream is calculated using equation 2.16.

$$\mathbf{ha_2 = ha_1 + Cw \times L/G \times Range}$$

$$\mathbf{ha_2 = 134.64 KJ/Kg}$$

2.8.3 Mass Flow of air (M_a) required

The mass flow of air required is calculated using equation 2.20.

$$\mathbf{M_a = \frac{m_w(h_3-h_4)}{(h_2-h_1) - h_4(w_2-w_1)}}$$

$$\mathbf{M_a = 290\ Kg/s}$$

2.8.4 Calculating Tower Demand

The tower demand(NTU) is calculated using equation 2.11.

$NTU = KaV/L = G/L \int dh / h_w - h_a = Cw \int dtw/ h_w - h_a$

NTU or $KaV/L = C_w$ X Cooling Range x [Sum of $1 / (h_w - h_a)$] / 4

Cooling Range = (t_3-t_4) =15 degC

Table1- Calculation for $1/(hw-ha)_{avg}$

Temperature(degC)	Hw(KJ/Kg)	Ha(KJ/Kg)[ha2=ha1+L/G(Range)]	1/hw-ha
CWT = 35	129	ha_1 = 73.89	0.018
CWT+0.1(Range)=36.5	139.91	$ha=ha_1+0.1C_wL/G(Range)$=79.96	0.016
CWT+0.4(Range)=41	174.57	$ha= ha_1+0.4C_wL/G(Range)$=98.20	0.013
CWT+0.6(Range)=44	202.83	$ha= ha_1+0.6C_wL/G(Range)$=110.36	0.010
CWT+0.9(Range)=48.5	254.10	$ha= ha_1+0.9C_wL/G(Range)$=128.60	0.00796
CWT + RANGE = 50	274.01	ha_2 = 134.64	0.00717

$1/(hw-ha)_{avg} = 0.0120$

$KaV/L= Cw(Range)/(hw-ha)_{avg}$

KaV/L = 15 x 4.2 x 0.0120

= 0.756 Kg air/Kg water

2.8.5 Rating of a cooling tower

For a given mass flow rate of water through the tower, the amount of heat to be dissipated or the rating of a tower can be expressed by simply using the relationship:-

Tower Rating = (Mass flow) x (Sp heat of water) x (cooling range) (2.21)

Tower Rating = 280 x 4.18 x 15

Tower Rating = 17556 kW

Tower Rating = 4987 TR

2.8.6 Specifying the Tower Capacity

Generally, a cooling tower capacity can be expressed in terms kW for a given cooling range. Alternatively, it may also be specified in terms of three temperatures, i.e. the cooling range and ambient wet bulb temperature. Information about these factors helps in approximating the physical size of the tower. The main determinants of the cooling tower size are its cross sectional area and its height. The nomogram shown in Fig 10 below is often used in order to estimate the liquid loading factor for a medium sized counter flow cooling tower by using the three variables, i.e. water inlet temperature, cooling range and wet bulb temperature. The plan area of the fills/ packing is determined by dividing the water mass flow rate by the loading factor obtained from the nomogram.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

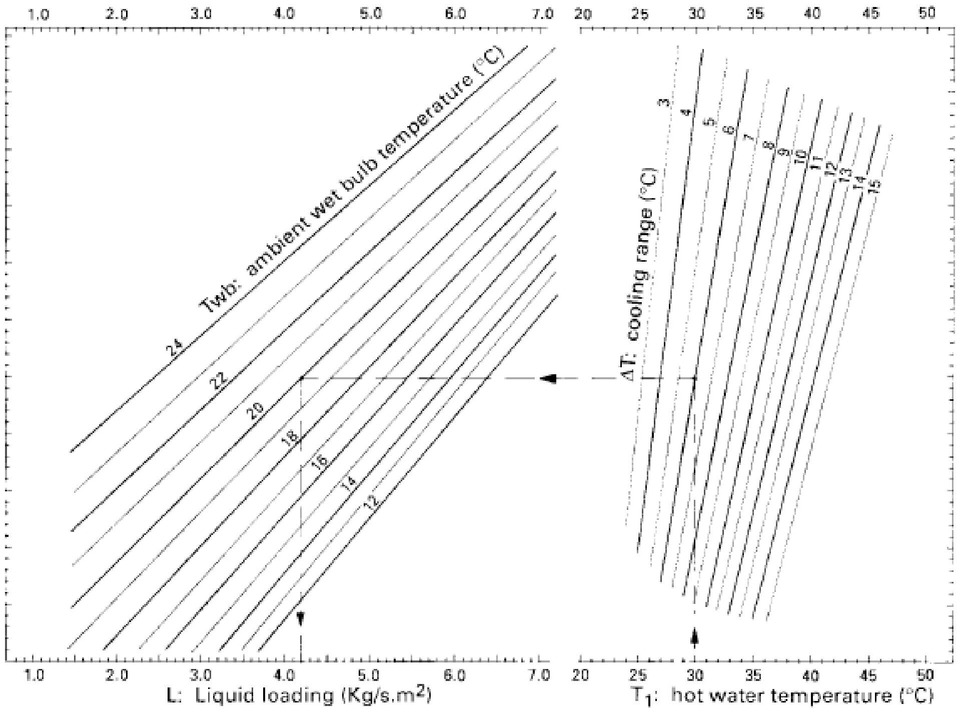

Fig 10 – Nomogram used for calculating Liquid Loading Factor [11]

Liquid Loading factor($\bar{L}$) = 7.2Kg/m^2.s{calculated from above graphically}

Plan area of fills/pack = L/$\bar{L}$ (2.22)
= 280/7.2
= 40.27m^2

In actual practice, pack/fills of a pre-determined height and plan area are then selected from available pack modules and then stacked in order to achieve the required plan area and height.

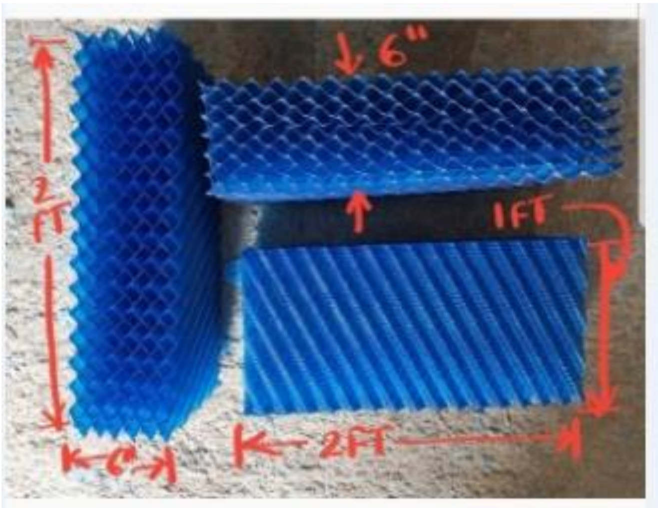

Fig 11 - Fills Module

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

Fig 11 above shows a typical pack module of height 2 feet and a plan area of 1feet x 6”. Many such modules are stacked together in order to achieve the desired plan are and height.

2.8.7 Calculating tower height

The height of the tower depends on the height of the pack/fills. The pack height can be calculated by using the following relation:-

$$\mathbf{Z(tower\ height) = (Tower\ Demand)_{calc}\ x\ \bar{L}/Ka} \qquad (2.23)$$

The only unknown in the above relationship is the value of enthalpy transfer coefficient, Ka. The value of this coefficient varies with the kind of pack/ fills used in the tower and is proprietary data that is determined experimentally. Thus, calculation of height of the cooling tower is not included in the scope of this study.

3 NUMERICAL ANALYSIS

In order to justify the use of Merkel’s theory for thermal analysis of the cooling tower in the preceding paragraphs, air flow through the fills region of the tower should be uniform. The study of airflow through the tower column has been undertaken by using the numerical simulation process. The fills region in the tower has been simulated as a porous region. In order to model a porous region in CFD, it is essential that the pressure drop in the porous zone for various flow rates of the fluid are known. The pressure drop across the fills region of a cooling tower depend upon the nature, size and fitment of the fills and are determined experimentally by cooling tower manufacturers and are proprietary data. Therefore, values of pressure drop at various air flow rates were not available. However, the value of pressure drop in a commercially available 5TR cooling tower for the design air flow rate was obtained and has been used for modelling the porous zone. The goal of the analysis is to simulate the porous region using this value and calculate the pressure drop using CFD and then validate it against the commercially available data. This would then validate the design of the porous model. Thereafter, air flow through the porous medium will be analysed for uniformity in flow.

The entire CFD simulation was undertaken in ANSYS Workbench. The advantage of this tool is that all simulation operations like creating of geometry, meshing and setting up of flow can be done in a single module. There is no requirement of using different modules for geometry creation, mesh generation and setting up of flow. Since, there is no requirement to import files from one module to the other, any variation in geometry is automatically updated in the mesh.

3.1 Geometric model

Numerical simulation of a problem requires computer simulation of the problem domain. In this case, the domain is a 5TR commercial cooling tower, since, the pressure drop data was available for the same. The dimensions and flow parameters used for modelling the domain are as follows:-

(i)	Tower height	:	2.1 m
(ii)	Tower diameter	:	750 mm
(iii)	Air flow rate	:	1260 CFM
(iv)	Fills height (porous zone)	:	2 ft(0.6 m)
(v)	Water flow rate	:	60LPM

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

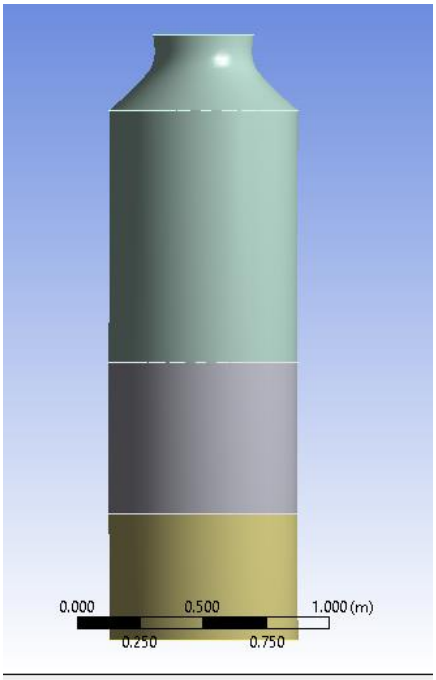

Fig 12 - 5 TR Cooling Tower Geometry

3.2 Grid Generation

The problem domain is required to be discretised into a grid. This is done in the mesh feature of Workbench. The type of grid depends on the geometry and the degree of accuracy required. A grid may be structured or unstructured based on the discretisation method used. The grid elements may be triangular or rectangular in 2-D and quadrilateral or hexahedral in 3-D.

In this problem, the mesh was automatically generated by Workbench based on geometry. The size of the element was however specified based on Y+ value.

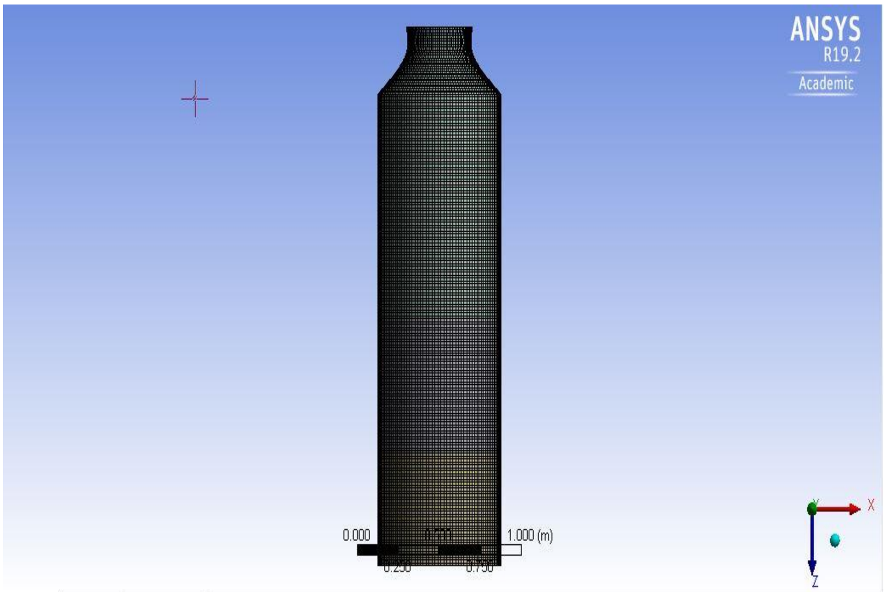

Fig 13 - Mesh

The details of mesh are as follows:-

(i)	Type of Mesh	:	Automatic/ Sweepable
(ii)	Type of element	:	Hexahedral/ Wedge
(iii)	Number of nodes	:	443321
(iv)	Number of elements	:	429312
(iv)	Size of element	:	1.3 e-002 m

3.3 Mesh Quality

The accuracy of the results obtained post CFD simuation of the problem depend on the quality of the mesh. The quality of the mesh can be evaluated from the following parameters:-

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

(i) Element Quality

After generation of mesh, the quality of the elements was checked in the Mesh Metric option. The quality of the mesh was found to be satisfactory at this stage and there was no requirement for refinement. The element quality of majority of the elements is above 0.9 as seen in Fig 14 below.

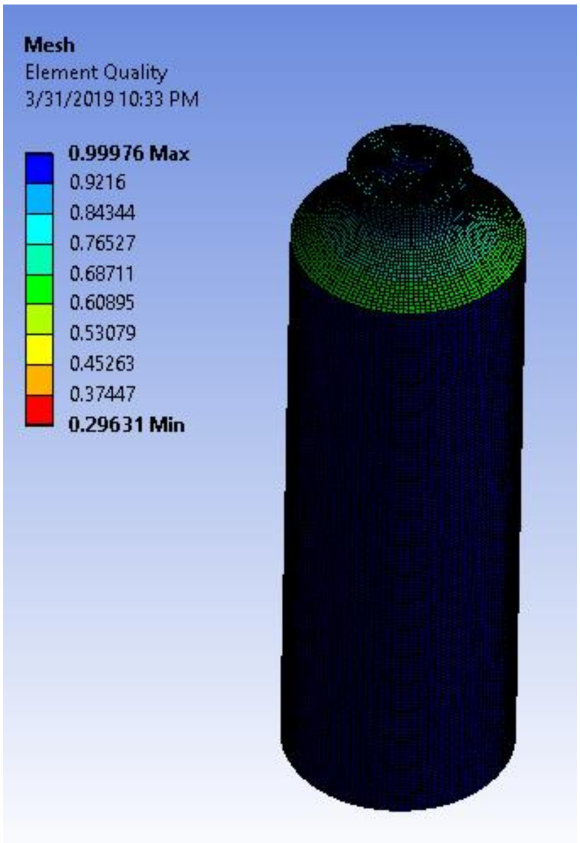

Fig 14 – Mesh element quality

(ii) Orthogonal Quality

The value of cell orthogonality ranges between 0 and 1, with the minimum value a cell should have being 0.01. The minimum orthogonal quality in this case is 0.58775 as can be seen in Fig 4.15 below. Majority of the elements have an orthogonal quality in excess of 0.9 as shown in Fig 15 below.

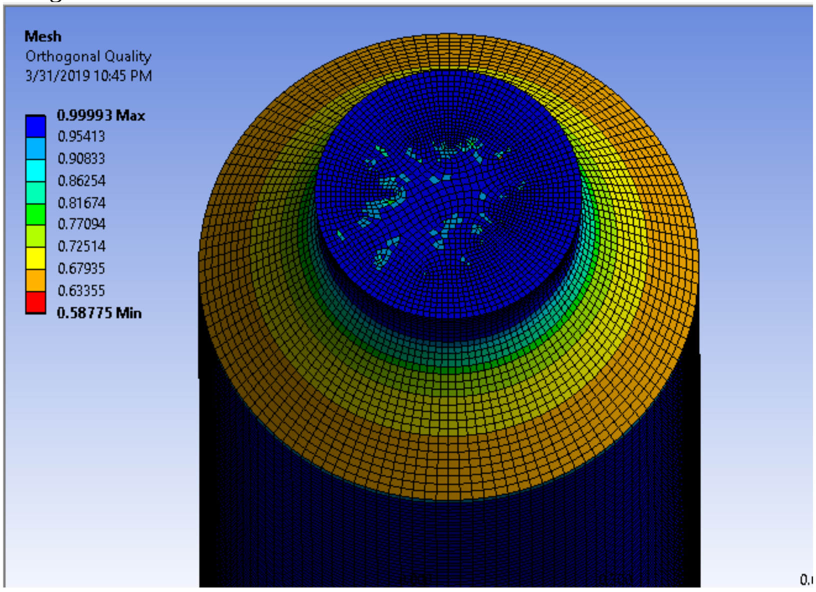

Fig 15 – Orthogonal Quality

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

(iii) Skewness

Skewness of hexahedral and quadrilateral cells shoud be below 0.85 for obtaining accurate solutions. Skewness is defined as the difference between the shape of a cell and the shape of an equilateral cell of the same volume. The maximum skewness value is 0.59054 as shown in Fig 4.17. Majority of cells have skewness below 0.1 as shown in Fig 16.

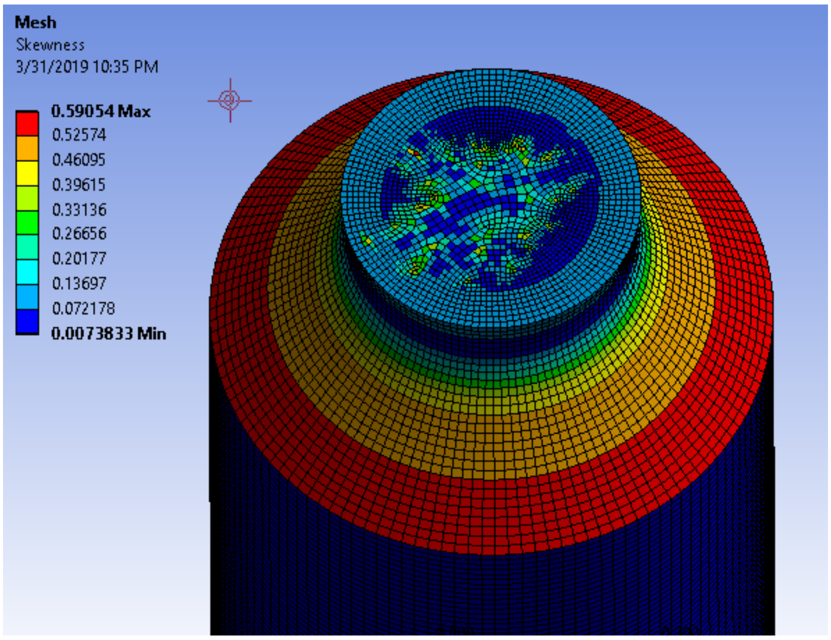

Fig 16- Skewness of Cells

(iv) Aspect Ratio

It is defined as the ratio of the longest edge length to the shortest edge length. An ideal aspect ratio is 1. It is the measure of stretching of cell. Majority of the cells have an aspect ratio close to 1 as shown in Fig 17.

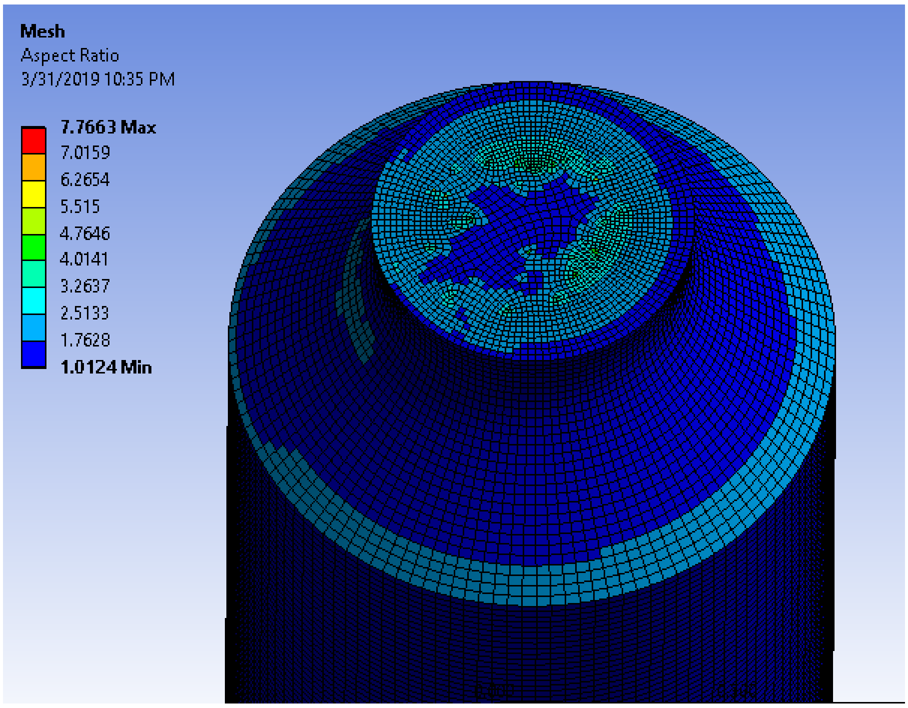

Fig 17 – Aspect Ratio

3.4 *Setting up of Flow*

The flow physics has been set up in two parts. The simulation of airflow has been done by specifying the air mass flow rate and pressure at the boundaries. The water flow has been simulated by using the discrete phase model(DPM) by introducing an injection. The standard K-Epsilon turbulent model with standard wall functions has been selected.

3.5 *Modelling of Porous Zone*

The fills region in the cooling tower has been modelled as a porous region. A cell zone is defined as porous and the pressure drop across this zone is then calculated. ANSYS Fluent uses the superficial velocity formulation as default while resolving for a porous medium, i.e. the software uses superficial velocity inside the porous medium based on the volumetric flow rate of the fluid, in order to ensure that there is continuity in the velocity vectors across the porous medium. ANSYS assumes that the porous medium is isotropic in nature. The porous zone is modelled by addition of an additional momentum source term to the fluid flow equations. This source term is given as:-

$$S_i = -[D_i \mu V_i + 0.5 C_i \rho V_i^2] \quad (3.1)$$

Here, the first term on the right hand side is the viscous loss term and the second term is the inertial loss term. The coefficients Di and C_i are the viscous and inertial resistance coefficients respectively and V_i is the velocity in the i^{th} direction. This momentum sink (due to negative sign) results in a pressure gradient in the porous zone, leading to a pressure drop that is proportional to the fluid velocity (or velocity squared) in the zone. This pressure drop is given by,

$$(dP/dz)_{porous} = -[D_i \mu V_i + 0.5 C_i \rho V_i^2] \quad (3.2)$$

The above equation is of the form,

$$\Delta P = aV_i + b V_i^2 \quad (3.3)$$

where a and b are constants. Comparing the above two equations, we get the values of the viscous and inertial resistance as follows:-

$$\Rightarrow \quad D_i = a/L\mu \quad (3.4)$$

$$\Rightarrow \quad C_i = 2b/L\rho \quad (3.5)$$

where, L is the length of the porous zone in the direction of flow(2 ft in this case).

In order to calculate the values of the viscous and inertial resistance coefficients, we require the pressure drop v/s velocity data for the given porous medium. In this case, the value of pressure drop across the fills region of a 5TR cooling tower was available for a given air flow rate. The same has been extrapolated and tabulated below:-

Table 2 – Pressure Drop v/s Flow rate

Ser No	1	2	3	4	5
Pressure(Pa)	95	105	115	125	135
Velocity(m/s)	1.2	1.25	1.3	1.35	1.4

The above values were then plotted to obtain a pressure v/s velocity curve, which is a polynomial of second order.

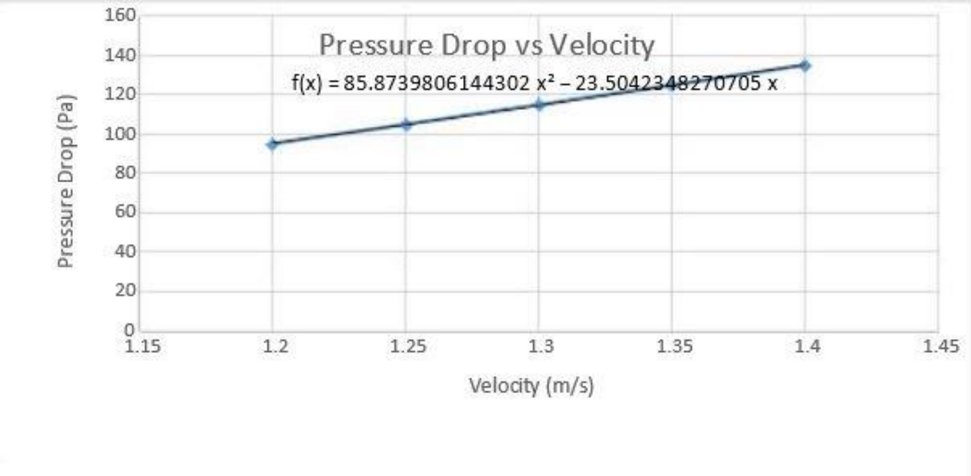

Fig 18 – Pressure drop v/s Flow rate Plot

The equation of the curve is given by,

$$85.87\, x^2 – 23.50x$$

On comparison with equation 3.3, the values of viscous and inertial resistance are then deduced by using standard values of air density and viscosity at 25 deg C.

$$\Rightarrow \quad a= 23.50; \; b= 85.87$$

$$\Rightarrow \quad D_i = a/L\mu \; = 2.24 \times e6 \; m^{-2}$$

$$\Rightarrow \quad C_i = 2b/L\rho = 246.55 \; m^{-1}$$

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

3.6 Defining the porous zone

The porous zone is a special type of fluid zone. In cell zone conditions option, porous zone has been selected. Then in the fluid dialog box, check the porous zone option and select the porous zone tab. The relative velocity resistance formulation is selected by default. Designate the two direction vectors normal to each other as shown in the figure 4.23. Now the values of the viscous and inertial resistance calculated above were entered. Since, the flow is predominantly along the Z axis and we wanted to eliminate flow in the X and Y directions, the value of viscous and inertial resistance in X and Y directions have been increased by a factor of 100.

3.7 Water Spray

The water spray has been simulated using the discrete phase model. An injection has been defined at 1.5m from the bottom of the cooling tower. The injection details are as follows:-

(i) Injection location : 1.5 m
(ii) Type of injection : Pressure Swirl Atomiser
(iii) Particle type : Droplet
(iv) Temperature : 340 K
(v) Injector diameter : 0.001m
(vi) Spray half angle : 30 deg
(vii) Upstream pressure : 2 bar

3.8 Boundary Conditions

The following boundary conditions have been defined for air flow:-

(i) **Inlet Velocity**

Based on the mass flow rate of air in a 5TR cooling tower, the inlet velocity has been calculated at 1.33m/s.

(b) **Outlet Pressure**

The outlet pressure is atmospheric pressure and guage pressure has been set to zero.

3.9 Convergence

Convergence of a solution is generally determined by the value of the residuals. For steady state problems as in this case, the satisfactory value for residual is typically between e-05 and e-04. The residual value signifies the error in the solution. The value of the residual is never zero for a numerical solution. However, for lower values of residual, the solution is more accurate. In the case of parameters that are to be monitored (total pressure this case), the values are required to reach a steady state for it to be deemed as converged. The continuity equation has converged after the residual is in the range of e-04, whereas the residuals in X, Y velocity are in the range of e-08, the Z velocity residual is of the order e-05 and the residuals in K and Epsilon are also of the order e-05. Further, the total pressure values being monitored at inlet and outlet have also attained steady state values. The residual plot and total pressure plot at convergence are shown in the figures below:-

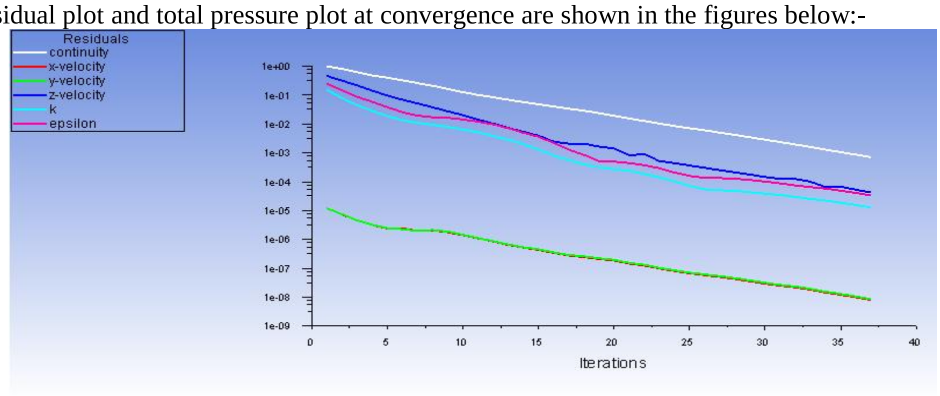

Fig 19 – Scaled Residuals Plot at Convergence

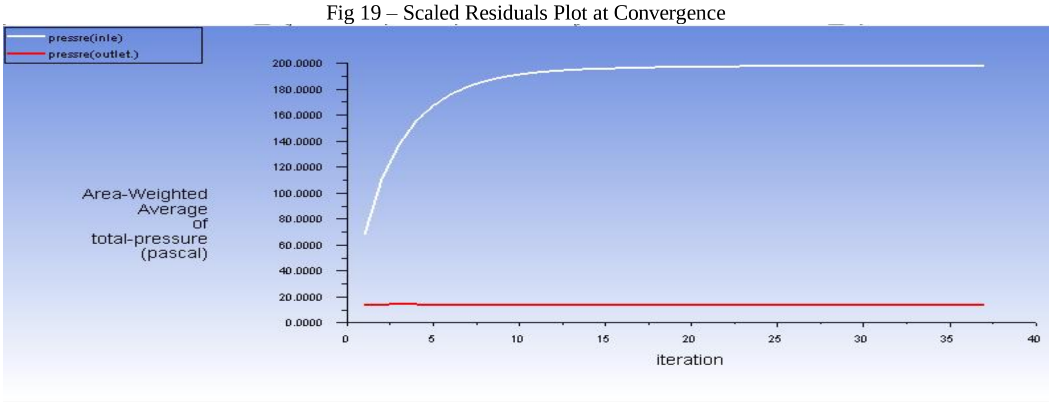

Fig 20 – Total Pressure Plot at Convergence

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

4 POST PROCESSING

Fluent post processor has been used in order to carry out analysis of pressure and velocity of air flow in the tower column. A variety of plots that include pressure contours, velocity contours, velocity vectors have been generated and are discussed in the subsequent paragraphs.

4.1 Pressure Contours

The total pressure contour has been generated by creating a plane parallel to the Z axis as shown in figure below highlights a significant pressure drop across the porous region as expected. The pressure drop across the porous region contributes significantly towards the pressure drop in the tower column. The values are as follows:-

Table 3 – Total Pressure at Inlet and Outlet

Ser	**Zone/Boundary**	**Pressure(Pa)**
(a)	Inlet (maximum)	192.61
(b)	Outlet(minimum)	12.28

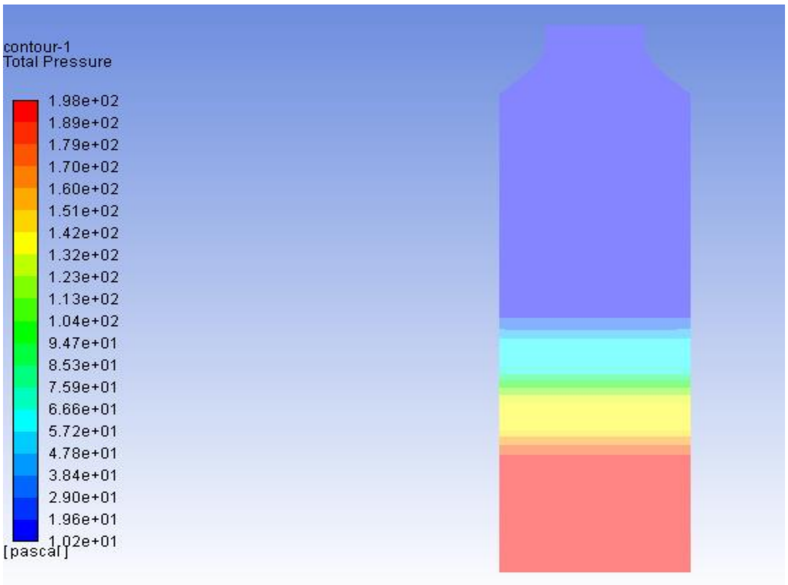

Fig 21- Pressure Contour Plot

4.2 Velocity Contours

The velocity contour plot has also been similarly obtained at the plane and is shown below. There is no significant change in the velocity in the tower column along the direction of flow. However, there is a slight velocity gradient due to the boundary layer, which is even more prominent in the porous region. As the air exits the tower at the top, there is a velocity recovery due to the involute profile. The inlet velocity is 1.33 m/s and the maximum velocity at outlet is recorded as 5.05 m/s.

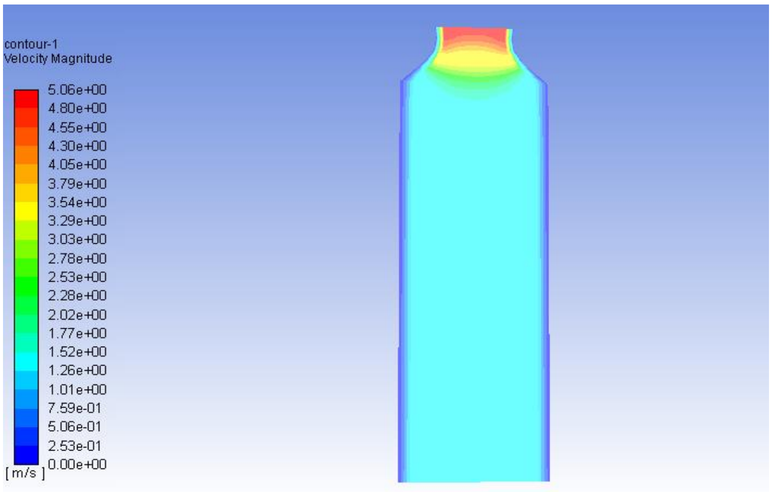

Fig 22 – Velocity Contour Plot

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 04, April-2019

4.3 Velocity Vectors

The plot for velocity vectors is shown below. The arrows from inlet to outlet have good directional properties and follow a set pattern. The increase in velocity at the tower outlet can be seen. Further, it can be seen that there is no recirculation of warm and moist air back into the tower.

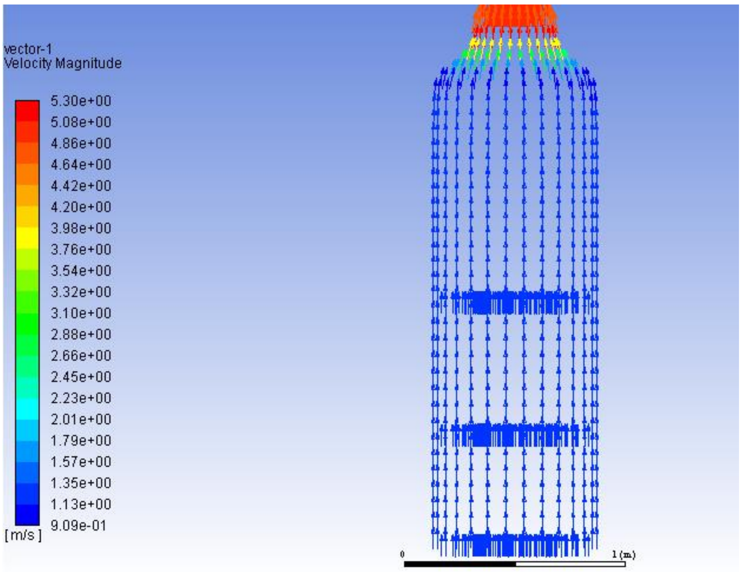

Fig 23 – Velocity Vector Plot

4.4 Uniformity in flow

In order to study uniformity in flow, two planes aligned with the inlet have been created at the inlet and outlet of the porous region and named plane 12 and 13. Therefter, in the report option, we will calculate the maximum and minimum velocities on the two planes. Similarly, we will also calculate the value of maximum and minimum total pressure at the two planes. As can be seen in the figure below, the difference between the maximum and minimum velocities and pressures at the planes is very insignificant. The same is also highlighted in the table below:-

Table 4 – Pressure and Velocity Facet Values

Ser	**Plane**	**Velocity(m/s)**			**Total Pressure(Pa)**		
		Max	**Min**	**Difference**	**Max**	**Min**	**Difference**
(a)	12	1.33	1.31	0.02	197.98	197.94	0.04
(b)	13	1.33	1.32	0.01	15.56	15.55	0.01

```
             Maximum of Facet Values
                  Velocity Magnitude                (m/s)
-------------------------------- --------------------
                            plane-12            1.3360882
                            plane-13            1.3319862
                ---------------- --------------------
                                 Net            1.3360882

             Minimum of Facet Values
                  Velocity Magnitude                (m/s)
-------------------------------- --------------------
                            plane-12            1.3178512
                            plane-13            1.3266422
                ---------------- --------------------
                                 Net            1.3178512

             Maximum of Facet Values
                      Total Pressure             (pascal)
-------------------------------- --------------------
                            plane-13            15.561943

             Maximum of Facet Values
                      Total Pressure             (pascal)
-------------------------------- --------------------
                            plane-12            197.98119
                            plane-13            15.561943
                ---------------- --------------------
                                 Net            197.98119

             Minimum of Facet Values
                      Total Pressure             (pascal)
-------------------------------- --------------------
                            plane-12            197.94792
                            plane-13            15.554684
                ---------------- --------------------
                                 Net            15.554684
```

Fig 24 – Maximum and Minimum Facet values of Velocity

Since the variation in maximum and minimum total pressure and velocity at both planes is minimal, the flow is uniform in nature.

5 Result Data

The results obtained in the research undertaken on cooling towers are summarised as below:-

5.1 Thermal Analysis

(i) Mass flow of air required: 290 Kg/s.
(ii) Tower demand(NTU): 0.756 Kg air/Kg water
(iii) Tower Rating: 4987TR
(iv) Liquid Loading: 7.2 Kg/ m^2s
(v) Tower Cross Section: 40 m^2

5.2 CFD Analysis

(i) Drop in pressure across tower column : 180 Pa = 18mm WC
(ii) Drop in velocity across tower column : 3.72 m/s
(iii) Difference between maximum and minimum velocity in a plane : 0.01 m/s

5.3 Validation of CFD

As stated earlier, the objective of the CFD simulation was to justify the use of the Merkel's cooling tower theory and the use of analytical formulae for calculation of the cooling tower NTU. Thus a model of a commercial 5TR cooling tower was chosen. The fills region in the tower have been modelled as porous region and air flow characteristics through the porous region have been studied. The pressure drop in the actual 5TR cooling tower as stated by its manufacturer was experimentally determined and was in the range of 15- 20mm of water column. This is equivalent to 147 – 196 Pa. This simulation was mainly focused on validating the experimental pressure drop data with results obtained through numerical analysis. As per table 5.1, the pressure drop obtained through CFD simulation is 180 Pa. This validation of pressure drop shows that the modelling of the porous region, which is the core heat transfer region in the cooling tower has been modelled fairly accurately. Further, as brought out in pargraph 5.1.4 and table 5.2 above, there is uniformity of air flow through the tower column, which in turn justifies the use of analytical formulae used in chapter 4 for calculation of tower NTU.

6 CONCLUSION

Based on the requirements on a 1000TPH training boiler, a thermal design for an induced draught wet cooling tower has been carried out using the Merkel's theory of cooling towers. The corresponding mass flow of air required, tower rating, tower plan area and most importantly, the tower demand have been calculated using analytical formulae. Since the proposed cooling tower is yet to be fitted, it is likely that the tower will be procured commercially for installation. Further, since the boiler load is not yet known exactly, the calculations have been carried out assuming a baseline load as already mentioned earlier. However, the methodology for calculating the tower demand remains unchanged irrespective of cooling load. The NTU/ tower demand is an important characteristic of a cooling tower and it provides sufficient information about the tower dimensions that need to be installed.

Further, CFD analysis of a 5TR commercial tower was undertaken and the pressure drop observed in the CFD model has been validated against experimental data for the 5TR commercial tower. Thus, the porous zone in the tower has been modelled fairly accurately and this model can now be used over a range of different dimensions and fluid flow rates to analyse fluid flow, pressure drops and thereby estimate the fan motor rating and fan characteristics for different tower dimensions.

REFERENCES

[1] Yogesh Parkhi, Dilip Vaghela, Jitendra Prajapati, "CFD Analysis of induced draught cross flow cooling tower", International Journal of Emerging Technology and Advanced Engineering, Volume 3, Issue 3, March 2013.
[2] Adam Klimanek, Michał Cedzich, Ryszard Białecki, "3D CFD modelling of natural draft wet-cooling tower with flue gas injection", Applied Thermal Engineering Journal, Sep 2015.
[3] Dudley J. Benton, Charles F. Bowman, Mark Hydeman Paul Miller, "An Improved Cooling Tower Algorithm for the CoolToolsTM Simulation Model", ASHRAE_Symposium_AC-02-9-4.
[4] Reuter H.C.R., Kroger D.G, "Computational Fluid Dynamics Analysis Of Cooling Tower Inlets", Stellenbosch University Library-Sunscholar Research Repository-2011.
[5] Rafat Al-Waked, Masud Behnia, "CFD simulation of wet cooling towers", Applied Thermal Engineering Journal, Vol 26, Issue 4,Mar 2006.International Journal of Advanced Research Trends in Engineering and Technology (IJARTET), Vol. 4, Special Issue 9, April 2017.
[6] M.Ravindra, Md Faiz Akram, "Modal analysis of cooling tower", International Journal of Advanced Research Trends in Engineering and Technology, Vol. 4, Special Issue 9, April 2017.
[7] MK Chopra, HK Patel, "CFD analysis of counter flow cooling tower used in blow moulding process", International Journal of Advanced Research Trends in Engineering and Technology (IJARTET), Vol. 2, Issue 2, Feb 2013.
[8] S. Parimala Murugaveni, P. Mohamed Shameer, "Analysis of forced draft cooling tower performance", International Journal of Research in Engineering and Technology, Volume: 04 Issue: 04, Apr-2015.
[9] Vali Kalantar, "Numerical simulation of cooling performance of wind tower in hot and arid region", Renewable Energy Journal, 30 Jun 2008.
[10] J. van der Merwel, C.G. du Toit, "Numerical simulation of a wet cooling tower", R and D journal. Vol. 18, No.2, July 2002.
[11] Cooling Towers Principles and Practice,Third edition by G. B. Hill, J. Pring, Peter D. Osborn.

[12] Cooling Tower Thermal Design Manual, Dael Aqua Pvt Ltd.
[13] Lecture Series on Thermal Analysis of Cooling Towers, Prof PMV Subbarao, IIT Delhi.
[14] Lecture Series on Psychometry, Chemical and Mass Transfer Operations, NPTEL.
[15] ANSYS Fluent User Guide.

BIOGRAPHY

Davinder Singh Grover received his B.Tech in Mechanical Engineering from JNU, New Delhi in Jun 2009. He is employed with the Indian Navy since 2003 and has served on steam propelled warships and aircraft carriers. He is presently pursuing his M. Tech in Marine engineering from Defence Institute of Advanced Technology, Pune.