CFD ANALYSIS OF FLOW THROUGH COOLING TOWER

A MAJOR PROJECT -II REPORT

SUBMITTED IN PARTIAL FULFILMENT OF THE REQUIREMENT
FOR THE AWARD OF THE DEGREE OF

MASTER OF TECHNOLOGY

IN

THERMAL ENGINEERING

By

SONU SHARMA
(2K13/THE/22)

UNDER THE SUPERVISION OF

MD. ZUNAID
ASSISTANT PROFESSOR

MECHANICAL ENGINEERING DEPARTMENT

DELHI TECHNOLOGICAL UNIVERSITY

SHAHABAD DAULATPUR
BAWANA ROAD, DELHI-110042,
INDIA

SESSION 2013-2015

DECLARATION

I, hereby declare that the work embodied in the dissertation entitled "**CFD ANALYSIS OF FLOW THROUGH COOLING TOWER"** in partial fulfilment for the award of degree of MASTER OF TECHNOLOGY with specialization in "THERMAL ENGINEERING", submitted to Delhi Technological University, is an original piece of work carried out by me under the supervision of Mr. MD. ZUNAID, Mechanical Engineering Department, Delhi Technological University. The matter of this work either full or in part have not been submitted to any other institution or University for the award of any other Diploma or Degree or any other purpose what so ever.

SONU SHARMA

Roll No. 2K13/THE/22

CERTIFICATE

Date: ___________

This is to certify that the work embodied in the dissertation entitled **"CFD ANALYSIS OF FLOW THROUGH COOLING TOWER"** by **SONU SHARMA, (**Roll No.-**2K13/THE/22)** in partial fulfilment of requirements for the award of Degree of **Master of Technology** in **Thermal Engineering** of **Delhi Technological University, Delhi** is an authentic record of student's own work carried by him under my supervision.

This is also certified that this dissertation has not been submitted to any other Institute or University for the award of any other diploma or degree.

MD. ZUNAID
ASSISTANT PROFESSOR

Mechanical Engineering Department

Delhi Technological University

Delhi- 110042

ACKNOWLEDGEMENT

It is a great pleasure to acknowledge our gratitude to all the people involved, directly or indirectly in the completion of this project.

It is distinct pleasure to express my deep sense of gratitude and indebtedness to my learned supervisor **MD. ZUNAID** who devoted valuable hours for this assignment and providing the motivational guidance during the entire preparation of this project, answering the number of technical queries despite his busy schedule. His valuable suggestions, constructive criticism and timely help proved extremely fruitful.

I am also grateful to **Prof. (Dr.) R. S. Mishra**, Head, Department of Mechanical Engineering, Delhi Technological University for providing the experimental facilities. His constant support, co-operation and encouragement for successful completion of this work is indebt-able.

I would also like to take this opportunity to present my sincere regards to my teachers for their kind support and encouragement.

This research work would not have become possible without strong cooperation, immense support and keen involvement of my friends and colleagues specially **Mr. Pawan kumar yadav and mayank kumar.**

My sincere thanks to entire dear and near ones who contributed directly or indirectly for accomplishing this arduous task.

SONU SHARMA

Roll No. 2K13/THE/22

ABSTRACT

A cooling tower is equipment which is used for rejecting the waste heat from hot water to environment. When hot water comes in contact with the air then sensible heat transfer and vaporization of some water droplet takes place due to which temperature of water decreases. Cooling tower is an important element in power plant and Heating ventilation air conditioning.

The present work involves the CFD analysis of flow through the cooling tower. This thesis work is mainly on rain zone. The mass flow rate of water, air inlet temperature and water inlet temperature in rain zone is kept constant as 15000kg/s, 295K and 303K respectively throughout the study. Three geometries of different rain zone height 8.577m, 6.777m and 4.977m is made for analysis. The diameter of water droplet is varied by using Rosin-Rammler distribution and temperature drop for different heights of rain zone is found and rate of drop in temperature relatively with varying diameter for different rain zone heights is also compared.

Results are analyzed and it reveals that with decrease in droplet diameter for any height of rain zone, temperature drop is increases and the rate of drop in temperature for 8.577m height is more than other two rain zones.

CONTENTS

CHAPTER 4

CHAPTER 5

CHAPTER 6

LIST OF FIGURES

FIGURE NO. **TITLE**

LIST OF TABLES

NOMENCLATURE

ρ	Density
$\bar{\bar{\tau}}$	Stress tensor
p	static pressure
F	gravitational force
k_{eff}	effectively conductivity
t	time
m	meter
mm	milimeter

CHAPTER 1
INTRODUCTION

1.1 Background

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

1.2 Types of cooling tower

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

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

A characteristic of natural draft wet cooling tower (NDWCT) is the spine of the cooling framework being used in expansive present day thermal power plants. In NDWCT, a blend of heat and mass exchange impacts are utilized to cool the water coming from the turbine's condenser. The boiling hot water, originating from the condenser, is showered on top of sprinkle bars or film fills keeping in mind the end goal to uncover an expansive bit of water surface to the cooling encompassing air. The humidity of the cooling air is less than the humidity of saturated air at the boiling hot water temperature, which brings about evaporation an measure of water. The energy needed for evaporation is extracted from the remaining water, henceforth diminishing its temperature. The cooled water is then gathered at the bowl of the NDWCT and pumped once again into the condenser. It is shown in fig: 1.1 Ref [2]

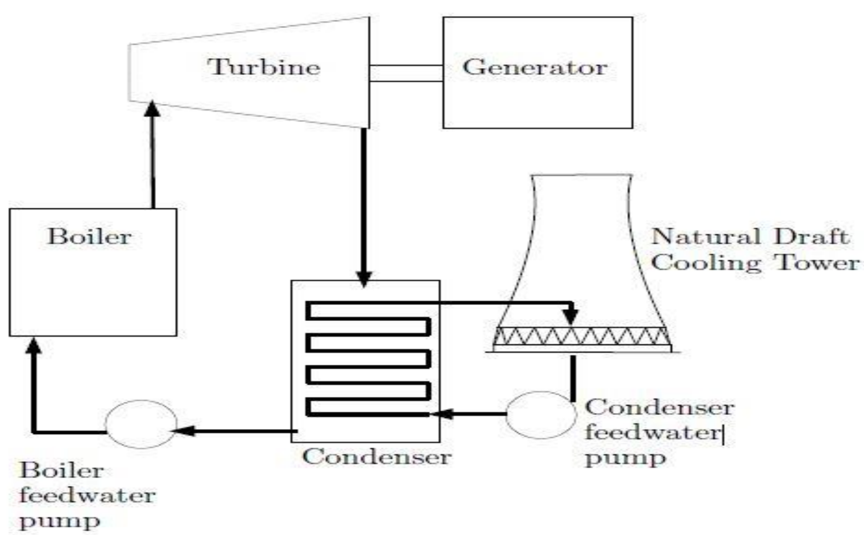


Figure 1.1: power station cycle with cooling tower.

1.3 Zones of cooling tower

Cooling tower consists of three zones.

1.3.1 Spray zone: this is the first zone where water is sprayed by means of nozzle and air comes in contact with water. Temperature drop is not much high.

1.3.2 Fill zone: this zone is honey comb like structure, it is porous zone so it increases time of contact between water droplets and air, which increase the evaporation rate and temperature drop is very high in this zone.

1.3.3 Rain zone: this zone is the last zone. Water entered into this zone after fill zone so temperature drop is less than the fill zone.

zones of cooling tower has been shown in fig: 1.2 Ref [22]

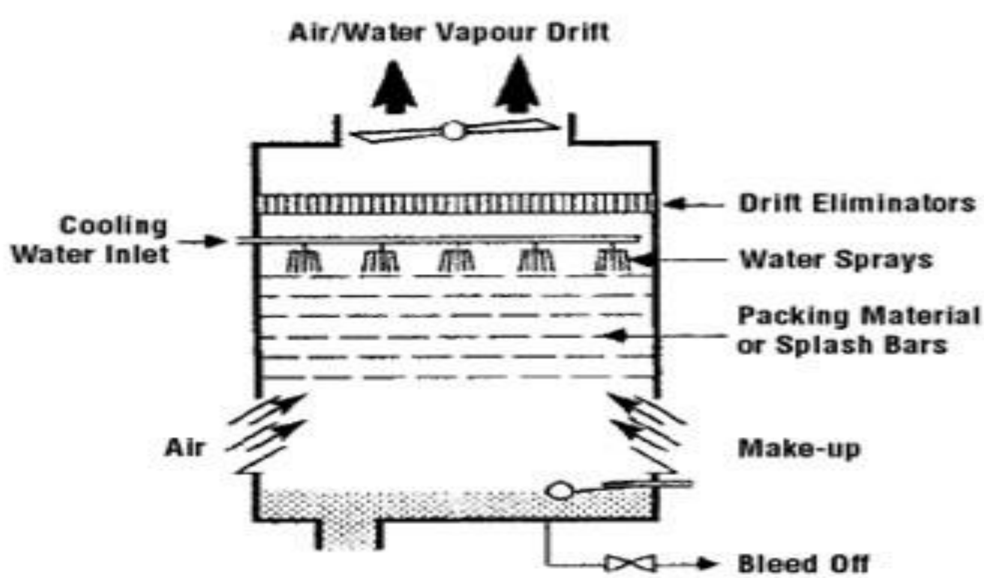


Fig:1.2 zones of cooling towers.

1.4 Applications

- **Heating ventilation air conditioning (HVAC):** cooling tower is used to cool the hot water. Some places are shown below where cooling tower is used
  - ✓ Hotel
  - ✓ Hospitals
  - ✓ Public utilities
  - ✓ Factories
- **Power plants:** In power stations, electricity is generated when steam drives a turbine. This steam must be condensed before it can be returned to the boiler to continue the cycle of steam and electricity generation. The condensation process happens in a heat exchanger. Cooling water is needed in the heat exchanger and it is this cooling water that is cycled through the cooling tower. Some power plant has been shown below where cooling tower is used
  - ✓ Gas power plant
  - ✓ Nuclear power plant
  - ✓ Coal based power plant
  - ✓ Combined power plants
- **Heat removal process**
  - ✓ Food processing
  - ✓ Natural gas processing

1.5 Terminology

Make-up: Some amount of water must be added to the circulating water system to compensate water losses such as evaporation, drift loss, blow-out, blow-down, etc.

Approach: The approach is the temperature difference between the wet bulb temperature of the air at entry and cooled water temperature. Since the cooling towers are based on the principles of evaporative cooling, the maximum efficiency of cooling tower depends on the wet bulb temperature of the air. The wet-bulb temperature (WBT) is a type of temperature measurement that reflects the physical properties of a system with a mixture of a gas and a vapor, usually air and water vapor.

Range: The range is the temperature difference between the inlet hot water and outlet cooled water. Range should be high; it shows the performance of cooling tower.

CHAPTER 2
LITERATURE REVIEW

Kaiser et al. [1] have developed numerical model to analyse their performance is based on computational flow dynamics for the two-phase flow of humid air and water droplets. The Eulerian approach is used for the gas flow phase and the Lagrangian approach for the water droplet flow phase, with two-way coupling between both phases. They have shown that strong influence of the average water drop size on efficiency of the system and expose the effect of other variables like wet bulb temperature, water mass flow to air mass flow ratio and temperature gap between water inlet temperature and wet bulb temperature.

Alok Singh et al. [2] have developed a model in commercial code FLUENT. They have done the analysis of a natural draft wet cooling tower using a two dimensional CFD approach. This model aims at determining the heat and mass transfer from water to air. They have used the Eularian multiphase model and RNG K-ε model to simulate the flow and turbulence modeling of multiphase flow respectively. They have reported a low value of density along the axis of cooling tower. Its higher value is found near the wall. Highest value of thermal conductivity is found near the axis. They have found the stream function to be linearly constant for axis.

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

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

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

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

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

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

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

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

Naphon [11] investigated both experimental and theoretical results of the heat transfer characteristics of the cooling tower and found that there is a reasonable agreement from the comparison between the measured data and predicted results in his model

Williamson [12] has focused his work on the modeling, design and optimization of natural draft wet cooling tower. He conducted a detailed analysis of heat and mass transfer. He observed that water mass flow rate and droplet size has major impact on heat transfer among all the parameters considered. He used the observations for optimization of the tower using the developed modeling.

Viljoen [13] studied the effect of water distribution, drop size distribution and heat transfer on the thermal performance of cooling tower using FLUENT. He also developed a model to predict the path of droplets. He tested two medium pressure nozzle and two low pressure nozzles. He reported the collision of water droplets has not major impact on water distribution pattern.

Kroger [14] has performed the CFD analysis for rain zone and spray zone .He showed the relative contribution of cooling zones. He concluded that rain zone can provide up to 30% cooling and spray zone can provide up to 5-10% of overall cooling. He developed one dimensional, two dimensional and three dimensional cooling towers and has found the results of all models in close proximity. He has also developed empirical correlations for flow/viscous loses.

Radosavljevic [15] has developed an axisymmetric and three dimensional CFD modal of a natural draft wet cooling tower. He has used the turbulence model and Lagrangian model for discreet phase tracking and found his results in closed approximation to that of experimental observations .He also measured the heat and mass transfer coefficients in the developed CFD model. He analyzed the effect of cross wind conditions on the cooling tower performance.

Lowe et al. [16] have worked on heat transfer and pressure drop in cooling tower. They found transfer characteristics for a fine spray system in terms of spray pressure and height through which the droplet falls. He varied the spray head from 1.2 - 3.6 meter and took the sauter mean diameter in the range of .9 – 1.28 millimeter. He showed that mean diameter of droplet increases as the droplets increases as the droplet falls through spray zone.

Hawlader et al. [17] has done the numerical study of the thermal hydraulic performance of evaporative natural draft cooling tower. He used one dimensional Lagrange particle tracking involving the coupling of heat, mass and momentum. He showed that the rain zone pressure distribution in the tower is nearly uniform.

Suresh Kumar et al. [18] showed the variance in droplet size distribution with respect to different projection angles. He also observed the variation of projection angle as a function of air inlet velocity.

Viljoen [19] in his M.Sc. thesis studied the heat transfer and drop size distribution in the spray zone theoretically as well as experimentally. He compares the experimental data with Rosin-Rammler distribution curve and found satisfactory results.

Terblanche et al. [20] measured the drop size distribution photographically below three different counter flow fills (cross fluted film, trickle and fiber cement).The data for each test case is presented as a cumulative mass distribution curve, Sauter mean diameter, Rosin – Rammler curve and Rosin- Rammler function.

RESEARCH GAP:

In cooling tower, CFD analysis has been done by varying mass flow rate of water, by varying heights of fills and rain zones, by varying air inlet temperature, by varying shapes of cooling tower. I found that there is no work by changing diameters of water droplet by rosin – rammler distribution. In this project this work has been done.

OBJECTIVE OF STUDIES:

The objective of this project is to find out the temperature drop of water by varying the diameters of water droplet at different heights of rain zone.

CHAPTER 3

BASIC GOVERNING EQUATIONS AND TURBULENT MODELING

3.1 Governing equations

3.1.1 Continuity equation

Continuity equation shows conservation of mass.

$$\frac{\partial \rho}{\partial t} + \boldsymbol{\nabla}.(\overrightarrow{\boldsymbol{\rho v}}) = S_m \qquad \text{.............................. (1)}$$

Mass conservation equation in general form and valid for both compressible and incompressible flows. The source Sm is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

3.1.2 Momentum equation

This equation conserves momentum.

$$\frac{\partial(\rho\vec{v})}{\partial t} + \boldsymbol{\nabla}.(\boldsymbol{\rho}\vec{\boldsymbol{v}}\vec{\boldsymbol{v}}) = -\boldsymbol{\nabla p} + \boldsymbol{\nabla}.(\bar{\bar{\boldsymbol{\tau}}}) + \boldsymbol{\rho g} \dot{+} \vec{\boldsymbol{F}} \qquad \text{................ (2)}$$

Where p is the static pressure, τ is the stress tensor and ρg and F are the gravitational body force and external body forces (e.g., that arise from interaction with the dispersed phase), respectively.

3.1.3 Energy equation

Energy equation represent as:

$$\frac{\partial(\rho E)}{\partial t} + \boldsymbol{\Delta}.(\vec{\boldsymbol{v}}(\boldsymbol{\rho E} + \boldsymbol{p})) = \boldsymbol{\nabla}.(\boldsymbol{k_{eff}\nabla T} - \textstyle\sum_j \boldsymbol{h_j \vec{J}_j} + (\bar{\bar{\boldsymbol{\tau}}}_{\boldsymbol{eff}}.\vec{\boldsymbol{v}})) + \boldsymbol{S_h} \quad \ldots\ldots\ldots\ldots\ldots.. (3)$$

Where k_{eff} represents effective conductivity and j_r represents diffusion flux of species j. The first three terms on the right-hand side of Equation shows energy transfer due to conduction, species diffusion, and viscous dissipation, respectively S_h Includes the heat of chemical reaction, and any other volumetric heat sources is defined.

3.2 TURBULENT MODEL

Turbulent Flows are characterized by fluctuating velocity fields. These fluctuations mix transported quantities such as momentum, energy, and species concentration, and cause the transported quantities to fluctuate as well. Since these fluctuations can be of small scale and high frequency, they are too computationally expensive to simulate directly in practical engineering calculations. Instead, the instantaneous (exact) governing equations can be time-averaged, ensemble-averaged, or otherwise manipulated to remove the small scales, resulting in a modified set of equations that are computationally less expensive to solve. However, the modified equations contain additional unknown variables, and turbulence models are needed to determine these variables in terms of known quantities.

3.3 Assumptions

- ✓ Water droplet diameter at the inlet of the rain zone is taken as same as the droplet diameter at inlet of cooling tower.
- ✓ Water droplets are spherical in shape.
- ✓ Air is uniformly distributed.

- ✓ Gravity value is taken as 9.81m/s^2.
- ✓ There is no heat generation inside the wall and walls are insulated.

CHAPTER 4

CFD MODELING

4.1 Introduction

The design of high speed digital computers, joint with the accurate numerical methods for solving physical problems, has revolutionized the technique we study and perform fluid dynamics and heat transfer. This approach is called Computational Fluid Dynamics (CFD). It has made it possible to investigate complex flow geometries with the ease way as that faced while solving idealized problems using conventional methods. CFD is the zone of study which combines fluid dynamics and numerical analysis. Historically, CFD was developed in 1960s and 1970s was driven by the need of the aerospace industries. CFD is used in all branches – civil, mechanical, electrical, electronics, chemical, aerospace, ocean, and biomedical engineering. CFD can also be used in testing and experimentation and it also decrease the total time of testing and designing. Fig. 4.1 represents the processes of CFD.

4.2 CFD Programs

The development of inexpensive high performance computing hardware and the accessibility of user-friendly interfaces have led to the development of commercial CFD packages. Before these CFD packages came into the ordinary use, one had to write his own code to carry out a CFD analysis. For different problems, the programs were usually different, although some part of the code of one program could be used in another. The programs were improperly tested and reliability of the results was often questioned. Today, well tested commercial CFD packages not only have made CFD analysis a routine design tool in industry, but are also helping the research engineer in effectively focusing on the physical system.

4.2.1 The Pre-Processor

This is the first step of CFD analysis

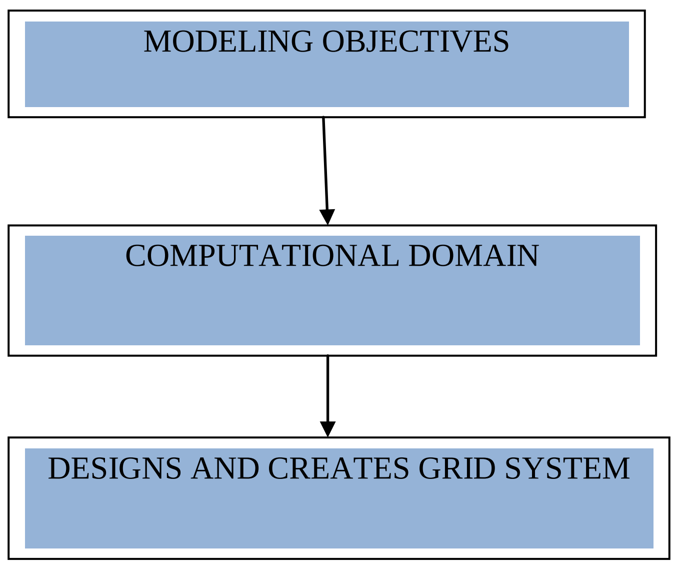
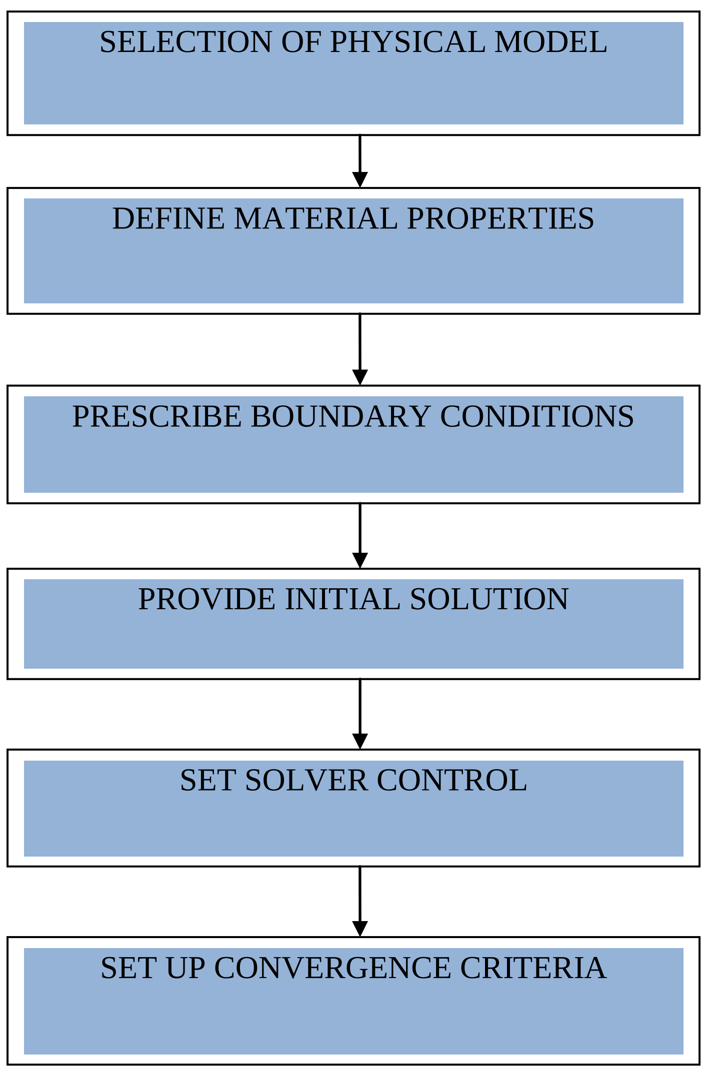


Fig: 4.1 steps of pre-processing

CFD modeling process starts with a consideration of the actual problem and identification of the computational domain. This is followed by the mesh structure generations, which is the most necessary part of the pre-processing activity. It is understood that more than half of the time taken by a CFD analyst goes towards mesh generation. Both computation time and correctness of solution depend on the mesh structure. Optimal grids are generally non-uniform – finer in areas where large variation of variables is predicted and coarser in regions where relatively little changes is predicted. In order to decrease the difficulties of engineers and maximize productivity, all the major CFD programs include provision for importing shape and geometry information from CAD packages like AutoCAD and I-DEAS, and mesh information from other packages like GAMBIT.

4.2.2 Solver

The solver is the heart of CFD software. The equations are set up which are chosen according to the options preferred by the analyst and grid points generated by the pre-processor, and solves them to calculate the flow field.



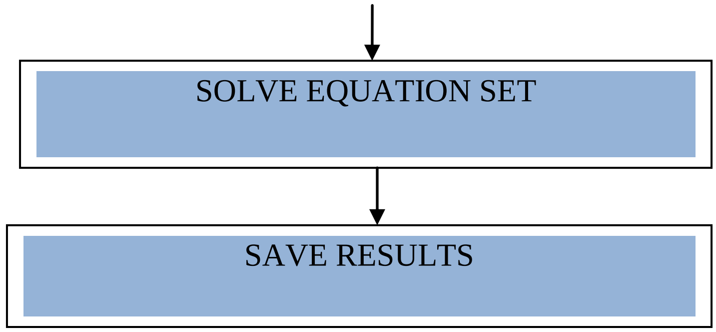


Fig: 4.2 flow chart of solver processes

When the model is fully set, the solution is initialized and calculation is start and result can be monitored at every step of time from iteration to iteration.

4.2.3 The Post-processor

This is the last part of CFD. Result is analyzed by the user and collects the data shown by the CFD software. The results may be displayed as vector plots of vector quantities like velocity, contour plots of scalar variables, for example pressure and temperature, streamlines and animation in case of unsteady simulation. Global parameters like skin friction coefficient, lift coefficient, Nusselt number and Colburn factor etc. may be computed through appropriate formulas. These data from a CFD post-processor can also be exported to visualization software for better display and to software for better graph plotting.

Many CFD packages have been developed in the past decade. Important among them are: PHOENICS, FLUENT, STAR-CD, CFX, CFD-ACE, ANSWER, CFD++, FLOW-3D and COMPACT.

4.3 Overview of fluent package

C computer language is used to written FLUENT and makes full use of the flexibility and power offered by the language. As a result, true dynamic memory allocation, efficient data structures, and flexible solver control (user defined functions) are all made possible. In addition, FLUENT uses a client/server architecture, which allows it to run separate simultaneous, processes on client desktop workstations and powerful computer servers, for efficient execution, interactive control, and complete flexibility of machine or operating system type.

4.4 CFD Procedure

Stages involve in CFD are shown in fig 4.4.

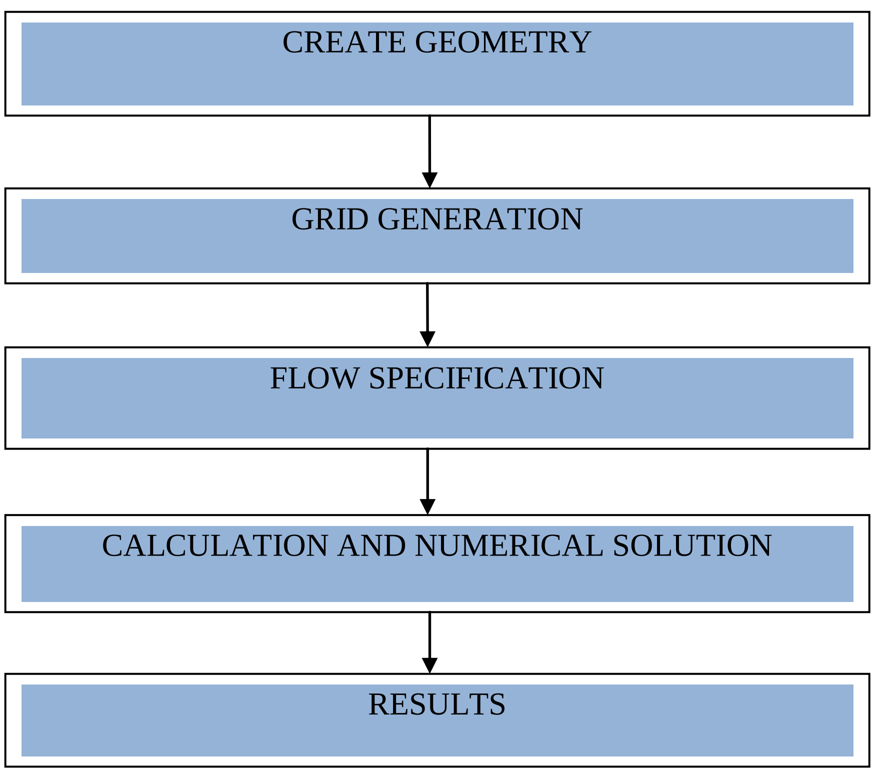


Fig: 4.3 stages involve in CFD

4.4.1 Geometry Creation

A 2-d model of rain zone has been created using design modeler GAMBIT 2.4.6. a length of 98m line created by joining two point (-49, 0) and (49, 0) on x-axis, so axis lies on y-axis. For 6.777m height rain zone other two points (-49, 6.77) and (49, 6.77) are selected. For 8.577m heights other two points are (-49, 8.577) and (49, 8.577). For 4.977m height rain zone other two points are (-49, 4.977) and (49, 4.977) then facing is done on all geometries.

4.4.2 Mesh Generation

After facing meshing is required, in meshing whole face is divided into equal size domain. The mesh of the model is shown in figs.4.5. It depicts that the domain was meshed with rectangular cells. Structural mesh is used because it gives high accuracy.

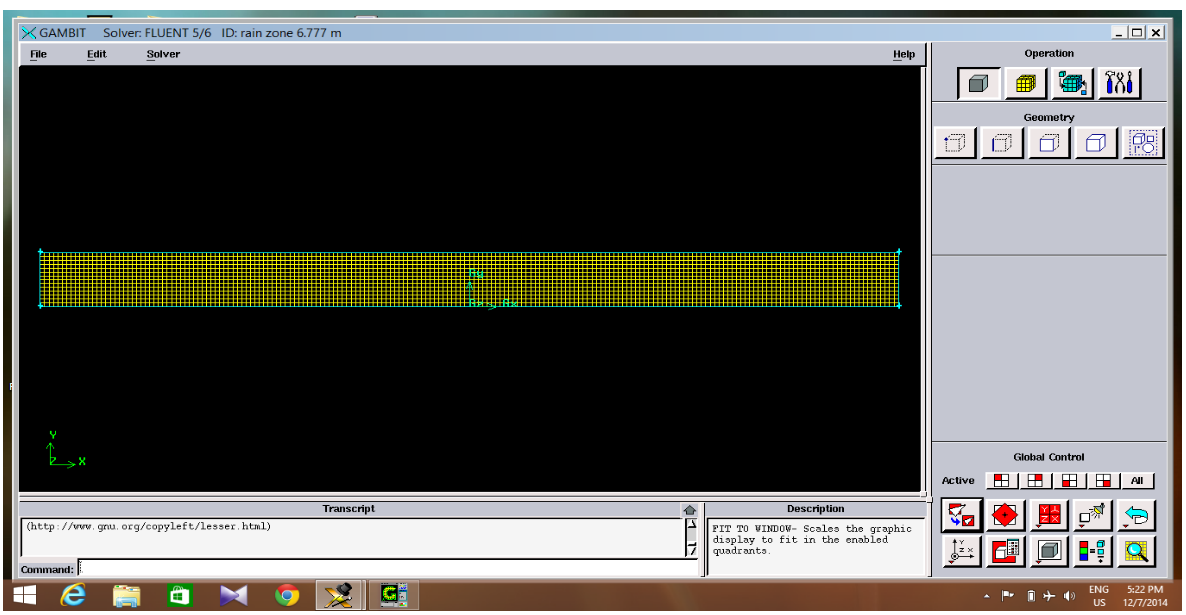


Fig: 4.4 mesh

CHAPTER 5

TWO DIMENSIONAL NDWCT MODEL

5.1 Reference for Validation

In the work presented I have taken a reference from the work carried out by N. Williamson et al. [7] titled "Comparison of a 2D axisymmetric CFD model of a natural draft wet cooling tower and a 1D model".

Wet cooling tower is used for study and same parameters and geometry has been taken. The work was verified and extended to include the effect of variable diameters in different height of rain zones. The droplets were injected using discrete phase model. The air is taken as continuous phase and water droplets as secondary phase.

For validation three different heights of rain zones has been taken which is operated at the same condition at 2.8mm diameter of water droplet and found the same result. 4.977m, 6.777m and 8.577m heights of rain zones has been taken and temperature drop has shown below:

Rain zone height(m)	**Actual temperature drop(k)**	**Predicted temperature drop(k)**	**% Error**
8.577	**3.4**	**3.54**	**4.1**
6.777	**2.8**	**2.857**	**2.0**
4.977	**2.5**	**2.3**	**8.0**

Table: 5.1 represent the percentage error with actual results.

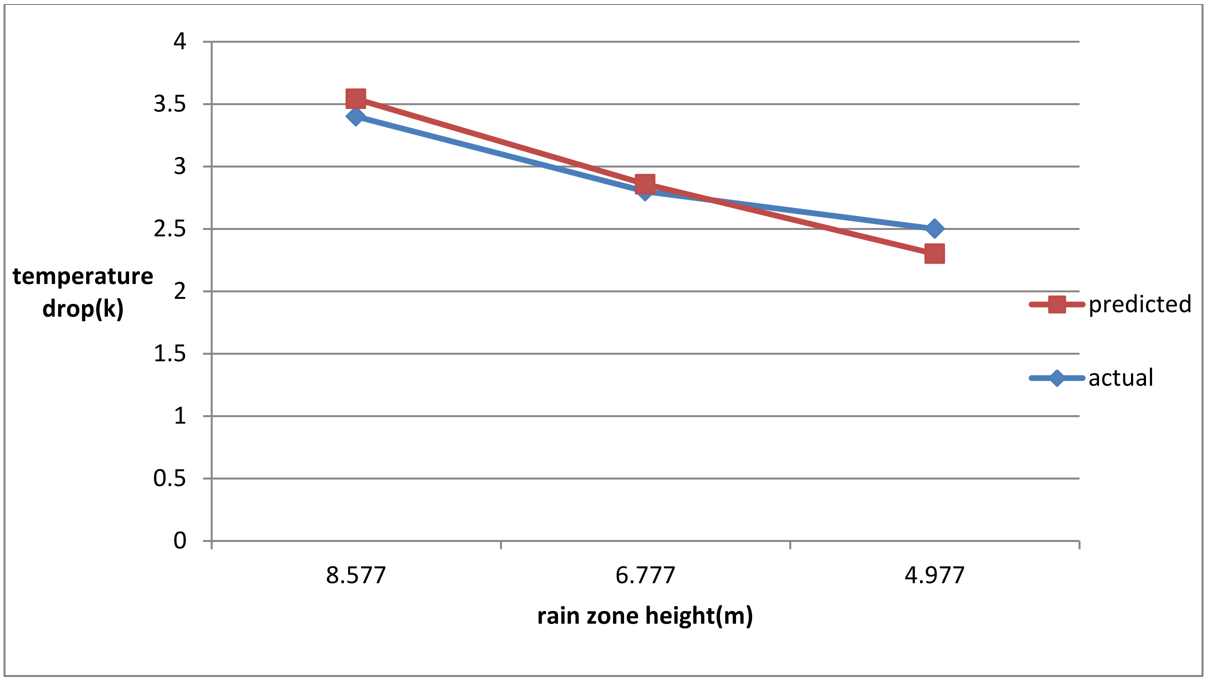


Fig: 5.1 comparison between actual temperature drop with predicted temperature drop

5.2 Reference Parameters

Tower Height - 131 m.

Base diameter - 98 m.

Flow rate - 15000 kg/sec

Water inlet temperature at spray zone - 313 K

Water inlet temperature at rain zone-303K

Rain zone heights - 8.577 m, 6.777m and 4.977m.

Ambient inlet Temperature - 295 K

The ambient air humidity was taken to be 55%.The inlet turbulence intensity was assumed 10%.

5.3 Solution Procedure

1. The ambient air is made to flow in the solver FLUENT 6 at its ambient temperature and pressure which are assumed to be 295K and 1.01325 bar respectively. The turbulence intensity in the air is assumed 10%. The gravitational acceleration is taken as 9.81m/s^2.standard k-epsilon (2-eqn) viscous model is used.

2. The species transport function is enabled in Fluent 6 and the species transport is specified for the air and water.

3. The water is injected using the custom laws. Two custom laws used are inert heating and vaporization. Now the temperature, mass flow rate and other variables are specified. The Lagrange particle tracking is used to track the droplets .The discrete random walk model is used under stochastic tracking.

4. Now the water interaction with air is enabled and unsteady particle tracking is used. Two ways turbulence coupling and coupled heat and mass transfer are used in the analysis .Automated tracking scheme is also enabled .To enhance the accuracy of the solution the droplet collision and droplet breakup is used.

5.4 Activated fluent solve control input data

Controls/ Solution/ Under-relaxation factors - Default

Controls/ Solution/ Pressure-velocity coupling - Simple

Controls/ Solution/ Discretization / Pressure - Body force weighted

Controls/ Solution/ Discretization / Density - Second order upwind

Controls/ Solution/ Discretization / Momentum - Second order upwind

Controls/ Solution/ Discretization / Turbulence - Second order upwind

Controls/ Solution/ Discretization / Turbulence - Second order upwind

Dissipation rate

Controls/ Solution/ Discretization / H_2O - Second order upwind

Controls/ Solution/ Discretization / Energy - Second order upwind

5.5 Activated fluent DPM injection input data

Injection type	Surface
Release from surface	Rain zone inlet
Particle type	Droplet
Material	water-liquid
Diameter distribution	Rosin – Rammler with diameter spread is 3 and number of diameters is 5.
Evaporating species	H_20

5.6 Activated fluent model input data

Description	Setting	Input value
Solver	Pressure based	-------------
Formulation	Implicit	-------------
Space	Axisymmetric	-------------
Multiphase model	OFF	-------------
Energy	Activated	-------------
Viscous/ model	k- epsilon	-------------

Viscous/ k-epsilon model	RNG	-------------
Viscous/ near wall treatment	Standard wall functions	-------------
Gravity	ON	- $9.81 m/s^2$
Species/ model	Species transport	-------------
Species/ mixture material	Air and H_2O	-------------
Species/ option	Inlet diffusion	-------------
Species/ option	Diffusion energy source	-------------
Discrete/ phase model	Interaction with continuous phase	-------------
Discrete/ phase model	Update DPM sources	
	No of continuous phase per DPM	2
Discrete/ phase model	unsteady particle tracking	-------------
Discrete/ phase model	Particle time step size	0 .01s
Discrete/ tracking	Maximum no of steps	50000
Discrete/ physical model	Two way turbulence coupling	-------------
Discrete/ spray model	Droplet collision	-------------
Discrete/ spray model	Droplet breakup	-------------
Initialization	all zones	

CHAPTER 6

RESULTS AND DISCUSSION

The analysis has been done for the variable diameters of the water droplets in the rain zone of the cooling tower and the cooling range has been determined for the design variables i.e. rain zone height. The efficiency of the cooling tower depends on the sensible heat transfer and vaporization which in turns depends on the saturation pressure, vapor pressure and the time available for this intimate contact. These factors are strongly depends on rain zone height, droplet diameters, base diameter and the type of flow. I have observed the variation in cooling effect for the different height of rain zones by varying water droplet diameter with Rosin-Rammler distribution and also compare the relative cooling rate among different heights of rain zones. Work presented in this project, the conditions are same unless otherwise specified.

6.1 Effect of variable droplet diameters for rain zone of 8.577m height

The following results have been obtained for the same mass flow rate and at the same reference conditions as specified before. The mass flow rate is 15000 kg/s and the turbulence intensity is 10%. The droplet diameter in the present study is varying with Rosin-Rammler distribution.

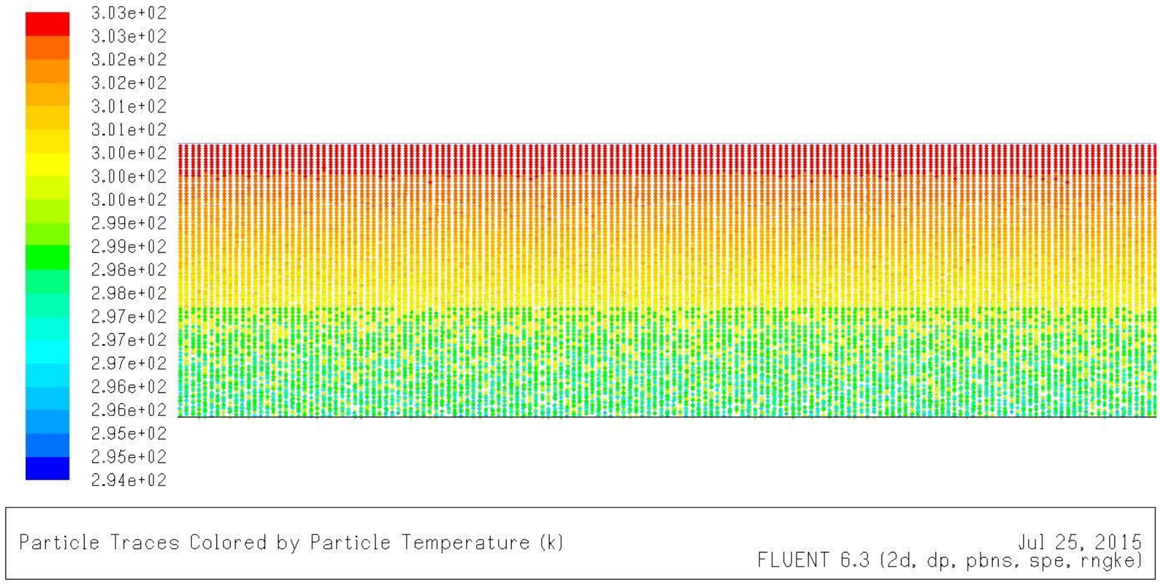


Fig: 6.1a Water temperature variation contours for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig: 6.1a shows the temperature variation in the rain zone of the cooling tower for maximum diameter 2.6mm and minimum diameter 1.8mm. In this picture the water temperature is decreasing from 303 K to 296.373K.

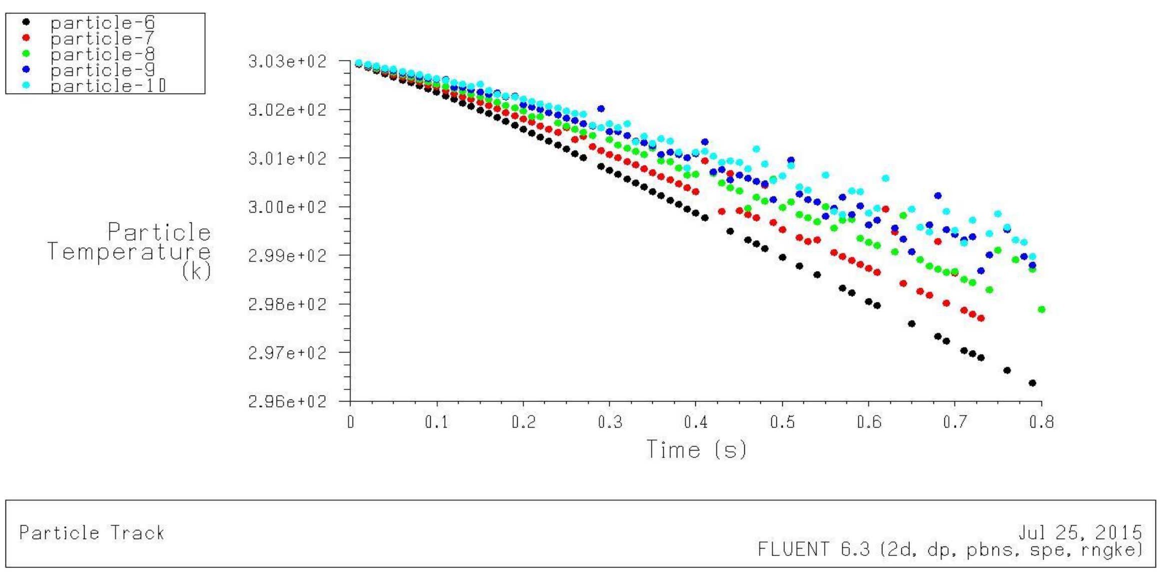


Fig: 6.1b Water temperature variation plot for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig: 6.1b represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 296.373 K.

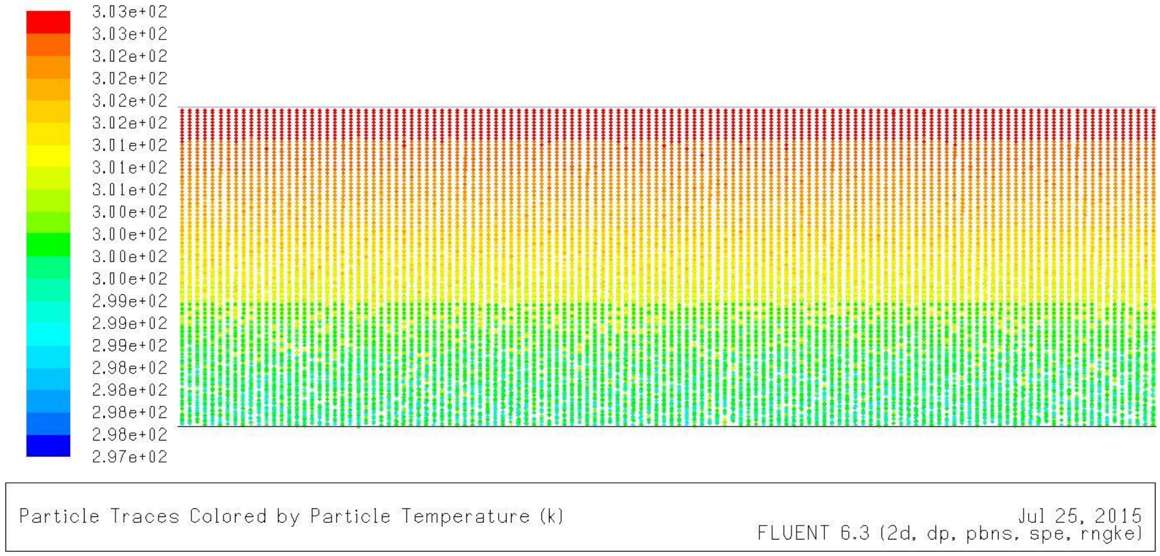


Fig: 6.1c Water temperature variation contours for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig: 6.1c shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.2mm and minimum diameter 2.4mm. In this picture the water temperature is decreasing from 303 K to 298.66K.

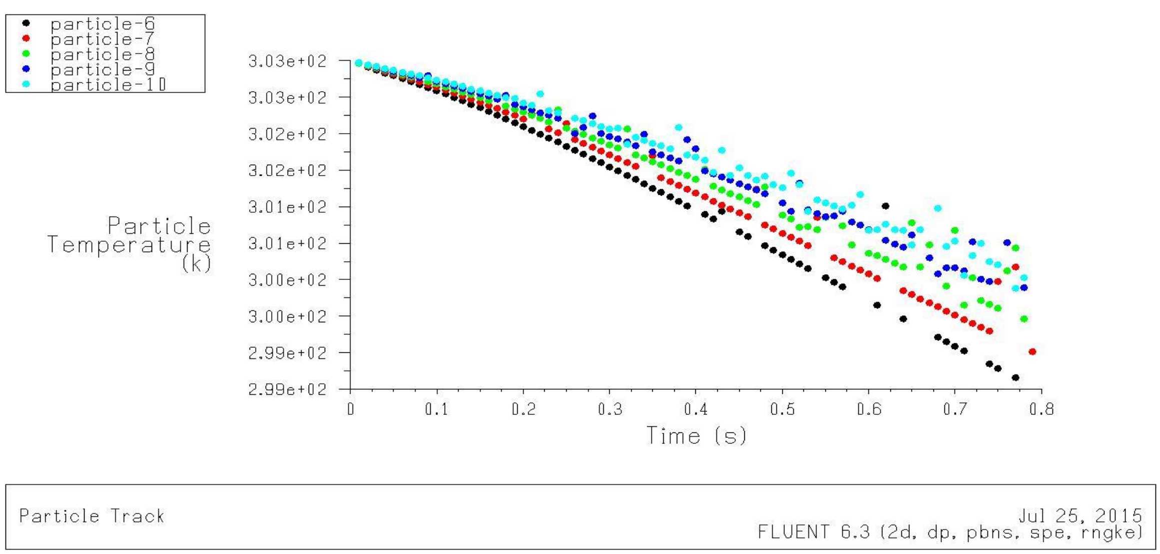


Fig: 6.1d Water temperature variation plot for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig: 6.1d represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 298.66K.

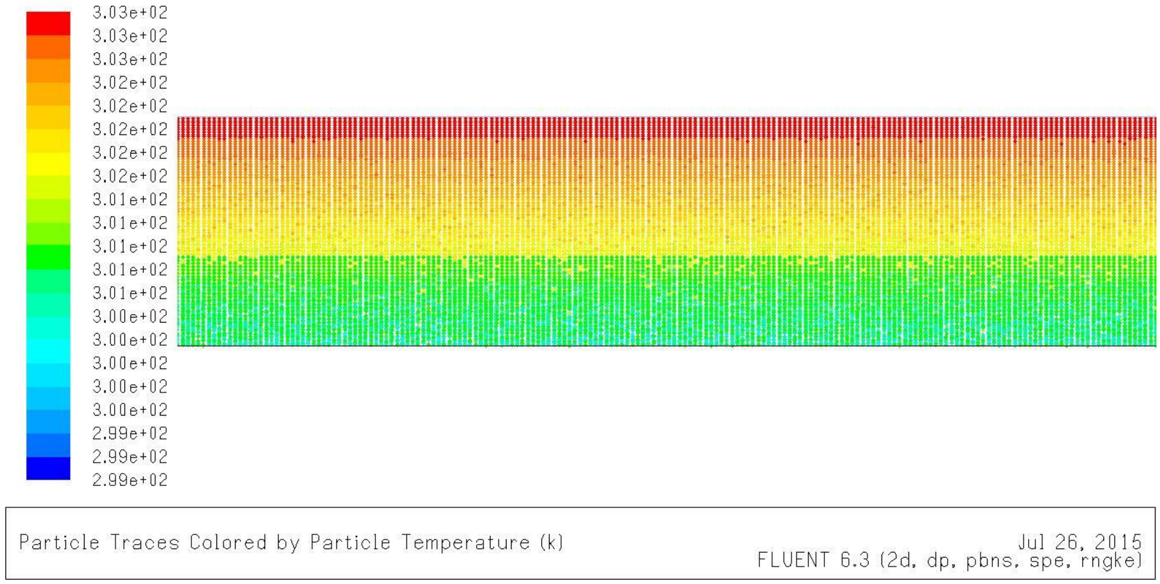


Fig: 6.1e Water temperature variation contours for maximum diameter 3.8mm and minimum diameter 3mm.

Fig: 6.1e shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.8mm and minimum diameter 3mm. In this picture the water temperature is decreasing from 303 K to 299.79K.

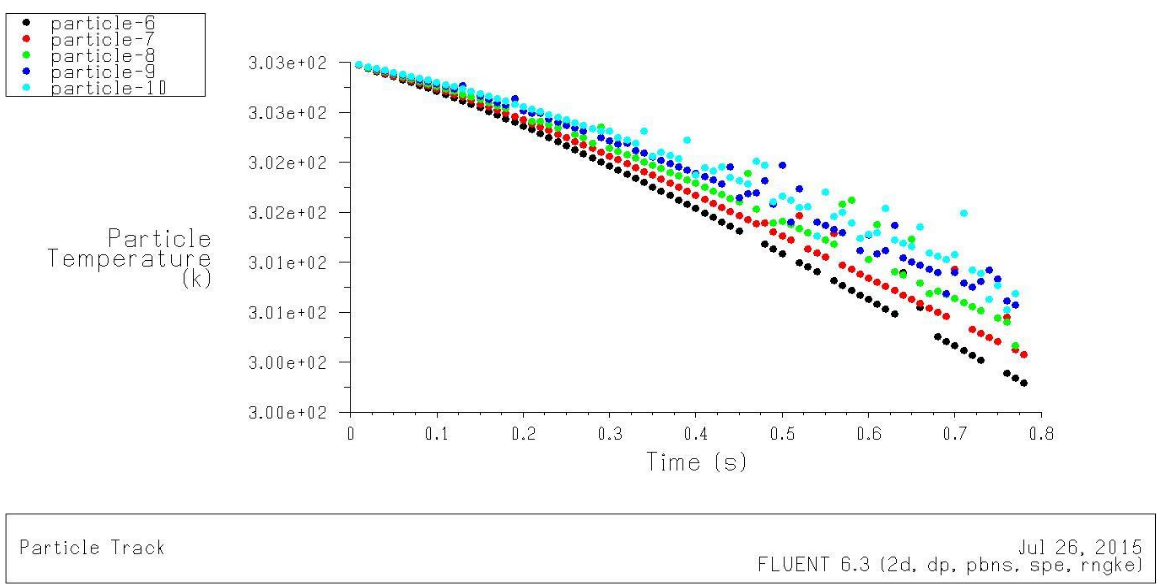


Fig: 6.1f Water temperature variation plot for maximum diameter 3.8mm and minimum diameter 3mm.

Fig: 6.1f represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 299.79K.

6.2 Effect of variable droplet diameters for rain zone of 6.777m height

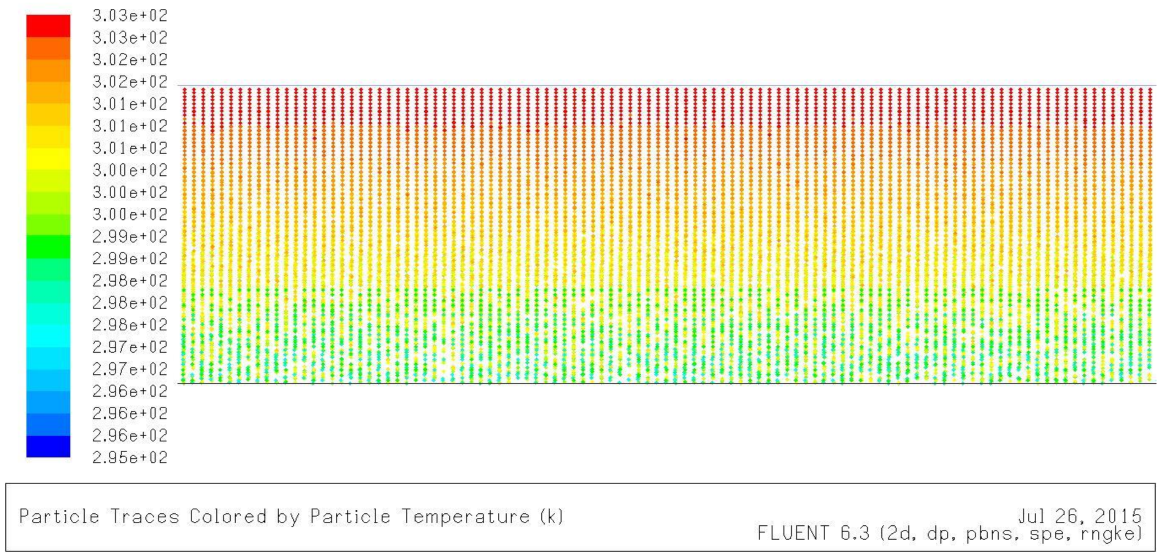


Fig: 6.2a Water temperature variation contours for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig: 6.2a shows the temperature variation in the rain zone of the cooling tower for maximum diameter 2.6mm and minimum diameter 1.8mm. In this picture the water temperature is decreasing from 303 K to 297.79K.

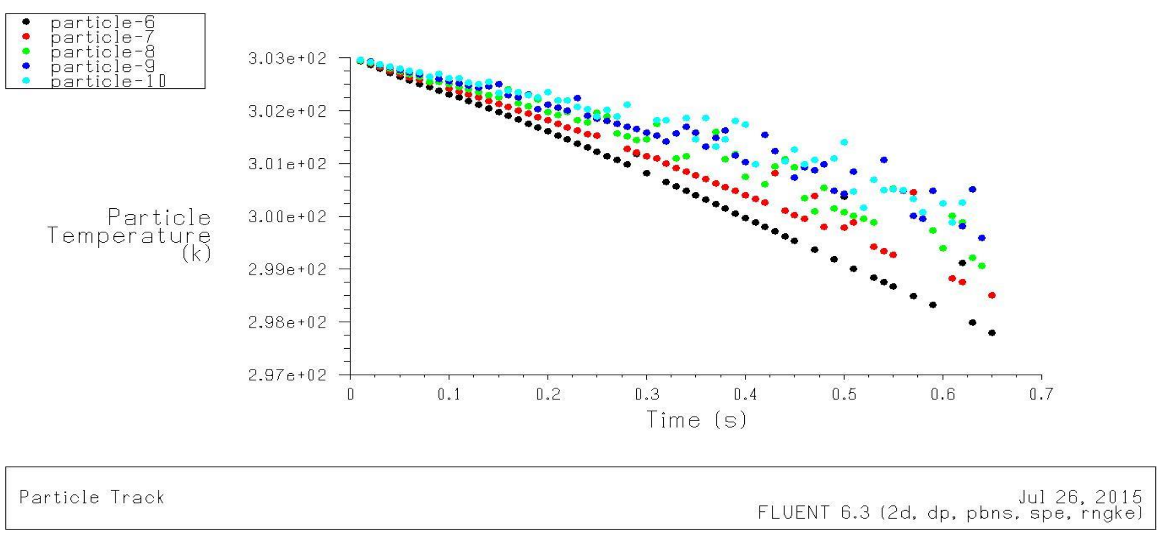


Fig: 6.2b Water temperature variation plot for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig: 6.2b represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 297.79 K.

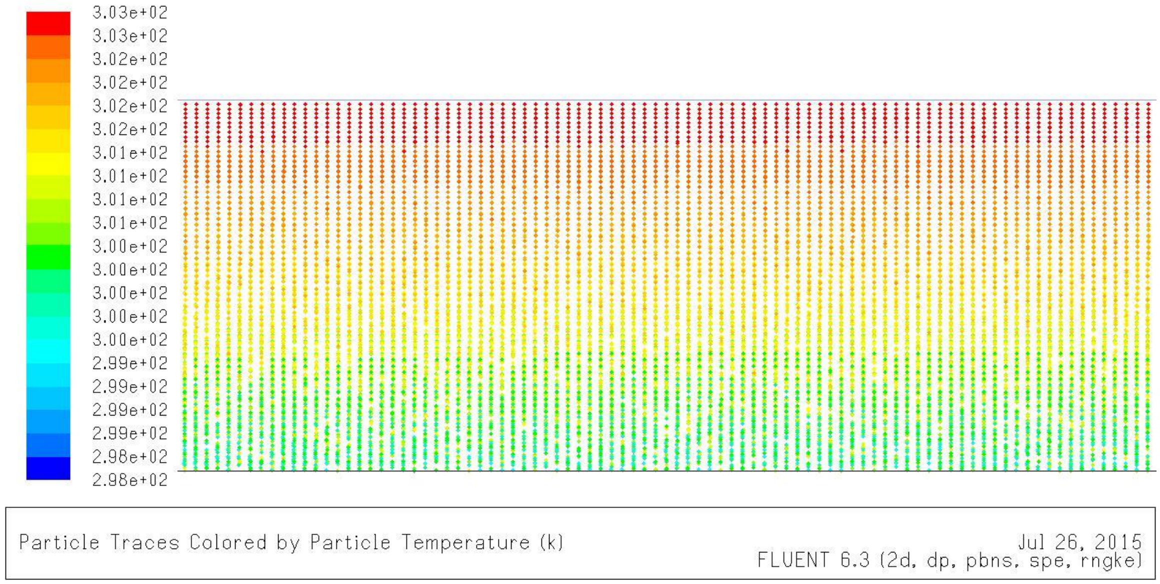


Fig: 6.2c Water temperature variation contours for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig: 6.2c shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.2mm and minimum diameter 2.4mm. In this picture the water temperature is decreasing from 303 K to 299.5921K.

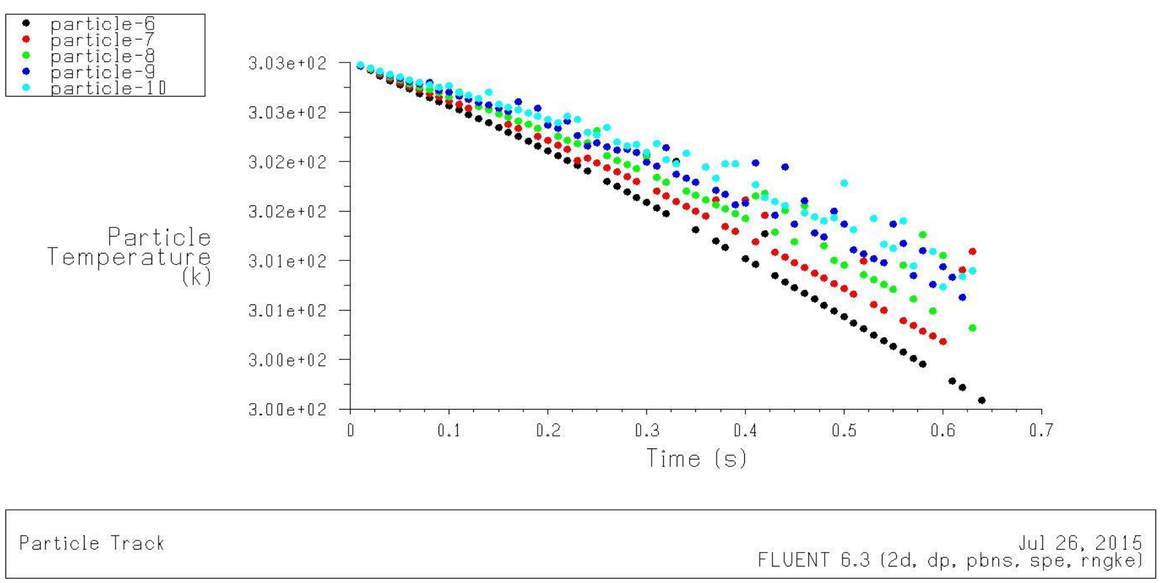


Fig: 6.2d Water temperature variation plot for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig: 6.2d represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 299.59K.

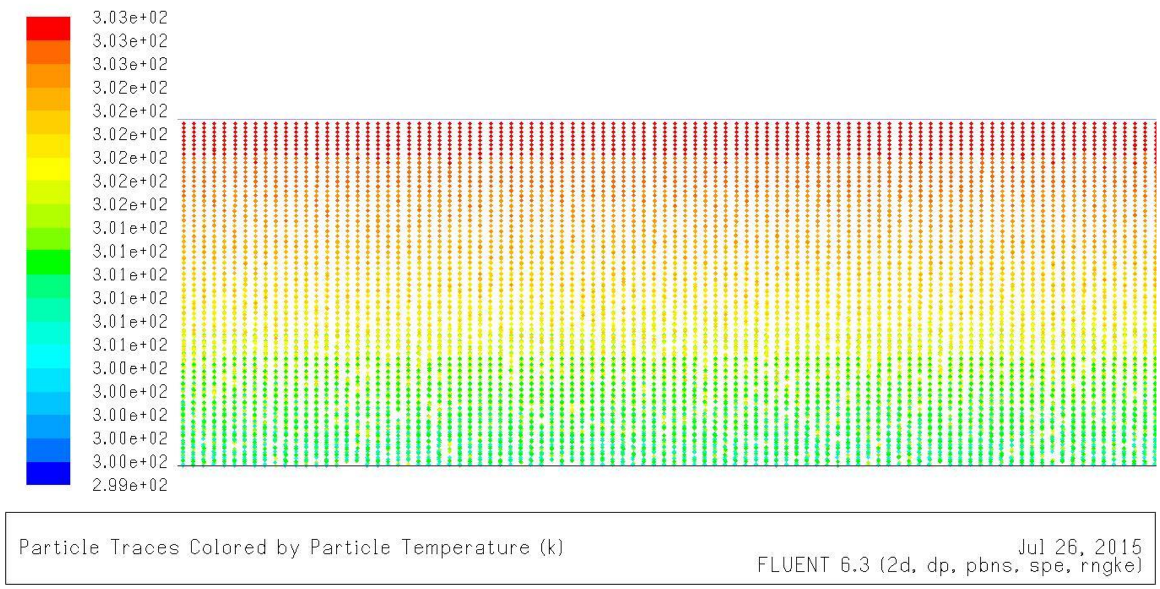


Fig: 6.2e Water temperature variation contours for maximum diameter 3.8mm and minimum diameter 3mm.

Fig: 6.2e shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.8mm and minimum diameter 3mm. In this picture the water temperature is decreasing from 303 K to 300.57K.

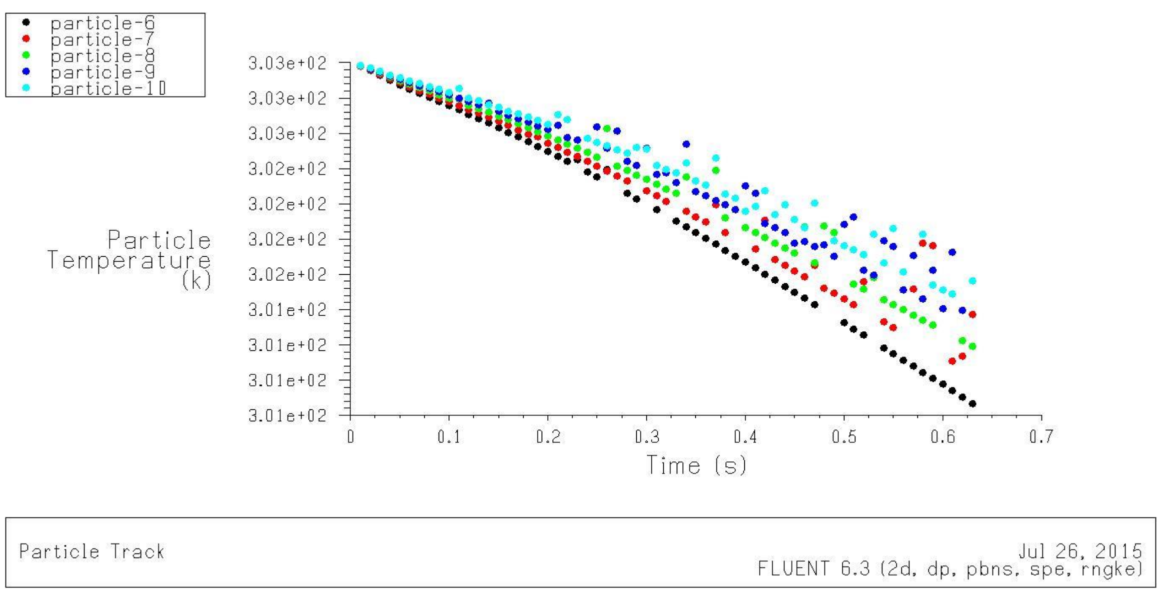


Fig: 6.2f Water temperature variation plot for maximum diameter 3.8mm and minimum diameter 3mm.

Fig: 6.2f represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 300.57K.

6.3 Effect of variable droplet diameters for rain zone of 4.977m height

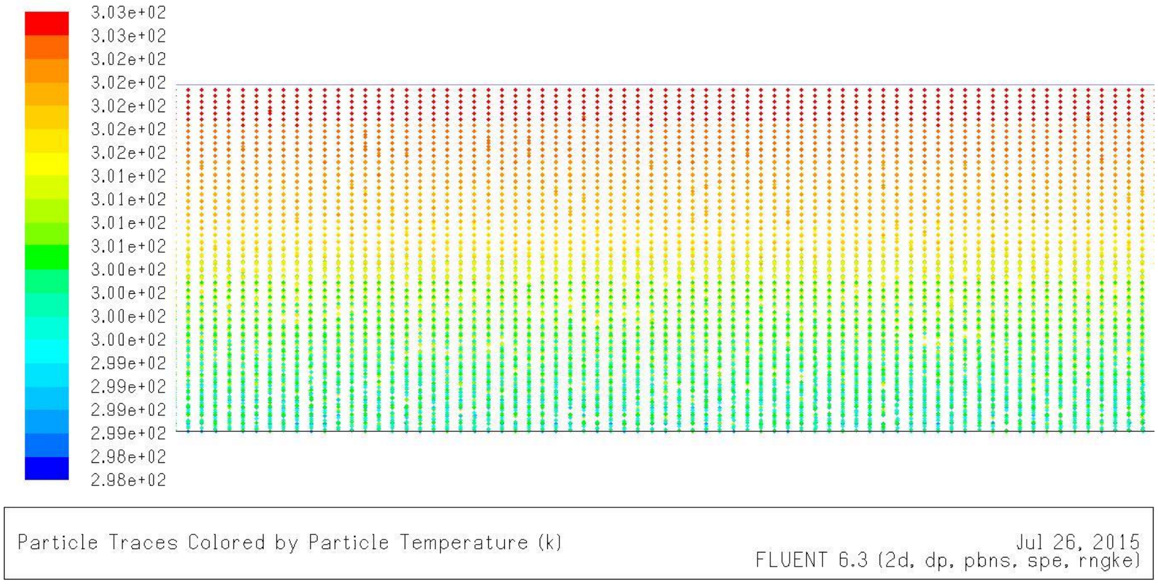


Fig: 6.3a Water temperature variation contours for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig: 6.3a shows the temperature variation in the rain zone of the cooling tower for maximum diameter 2.6mm and minimum diameter 1.8mm. In this picture the water temperature is decreasing from 303 K to 298.924K.

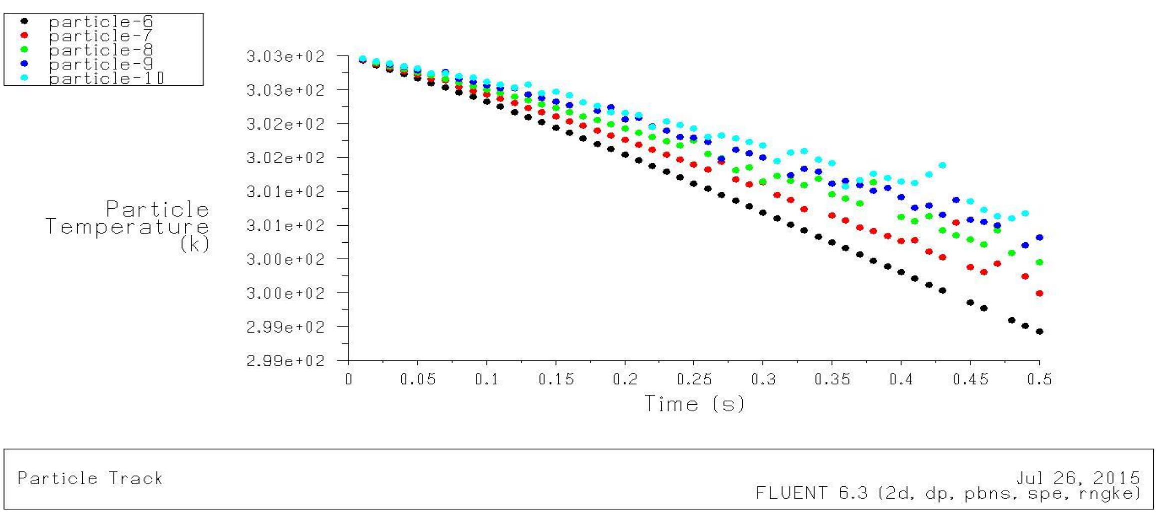


Fig: 6.3b Water temperature variation plot for maximum diameter 2.6mm and minimum diameter 1.8mm.

Fig: 6.3b represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 298.924 K.

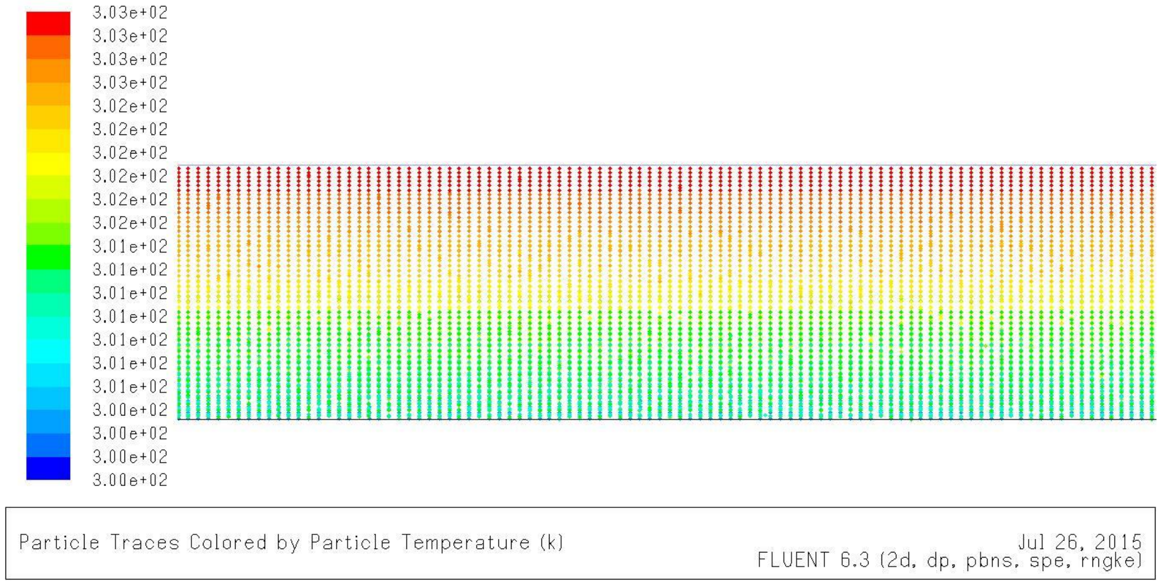


Fig: 6.3c Water temperature variation contours for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig: 6.3c shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.2mm and minimum diameter 2.4mm. In this picture the water temperature is decreasing from 303 K to 300.315K.

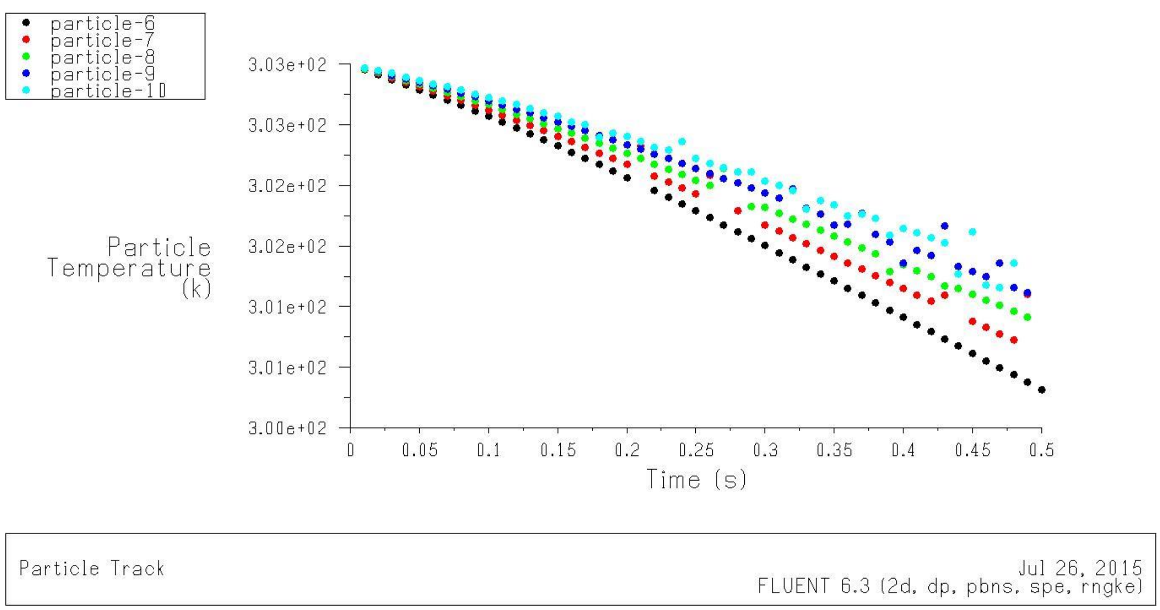


Fig: 6.3d Water temperature variation plot for maximum diameter 3.2mm and minimum diameter 2.4mm.

Fig: 6.2d represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 300.315K.

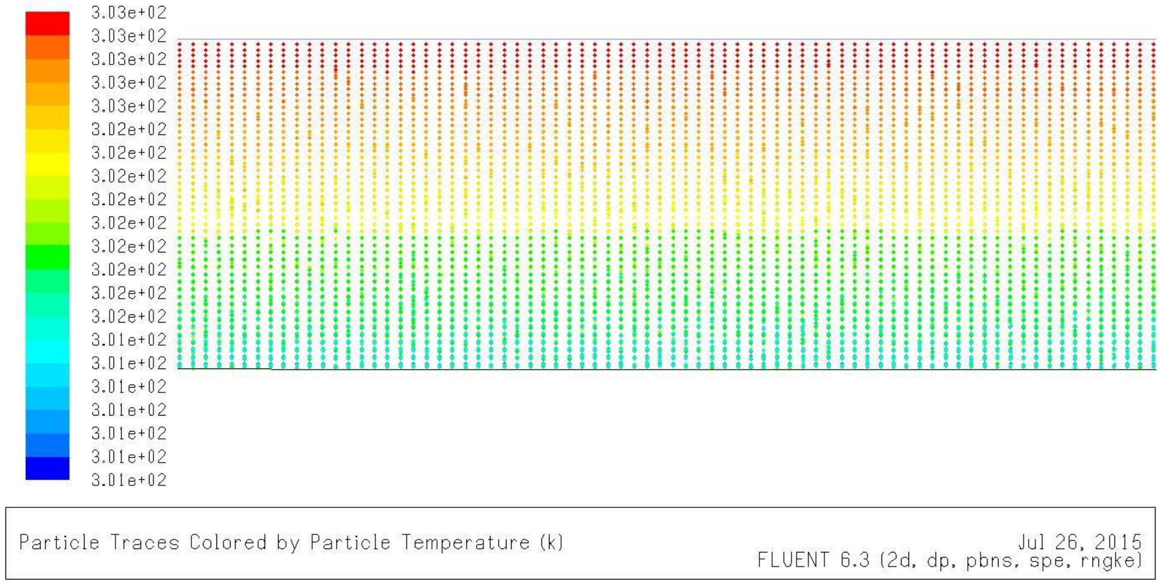


Fig: 6.3e Water temperature variation contours for maximum diameter 3.8mm and minimum diameter 3mm.

Fig: 6.3e shows the temperature variation in the rain zone of the cooling tower for maximum diameter 3.8mm and minimum diameter 3mm. In this picture the water temperature is decreasing from 303 K to 301.116K.

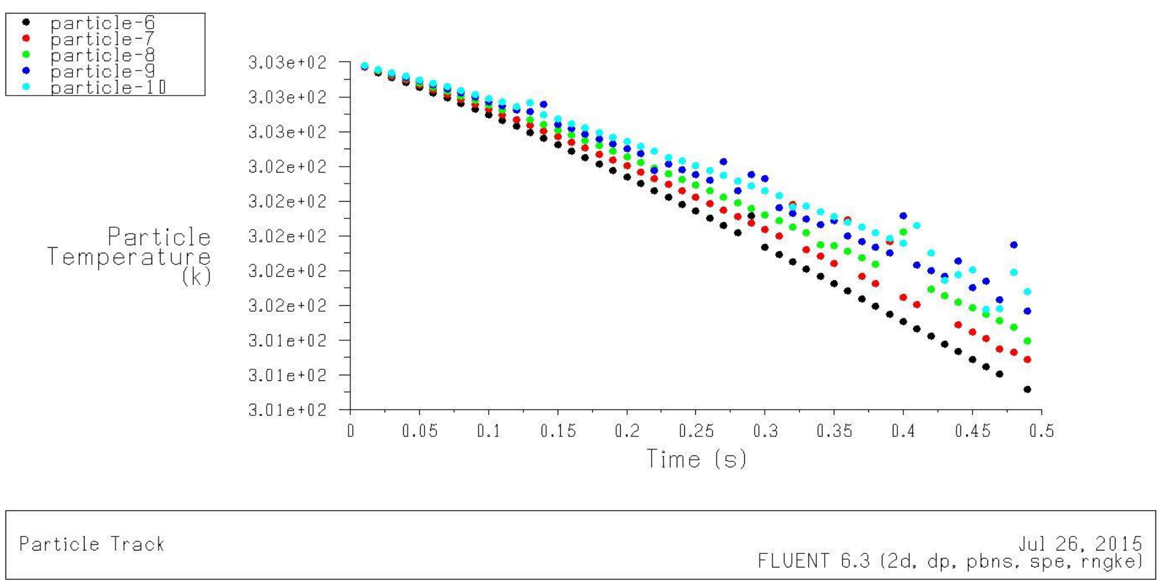


Fig: 6.3f Water temperature variation plot for maximum diameter 3.8mm and minimum diameter 3mm.

Fig: 6.3f represents temperature vs. time variation. It shows that average temperature of water is decreasing up to 301.1163K.

DISCUSSION

For 8.577 m rain zone height

Droplet diameter	Water inlet temperature	Water outlet temperature	Temperature drop
1.8mm – 2.6mm	303	296.373	6.627
2.4mm – 3.2mm	303	298.66	4.34
3.0mm – 3.8mm	303	299.79	3.21

Table: 6.1 represent temperature drop for 8.577m height rain zone.

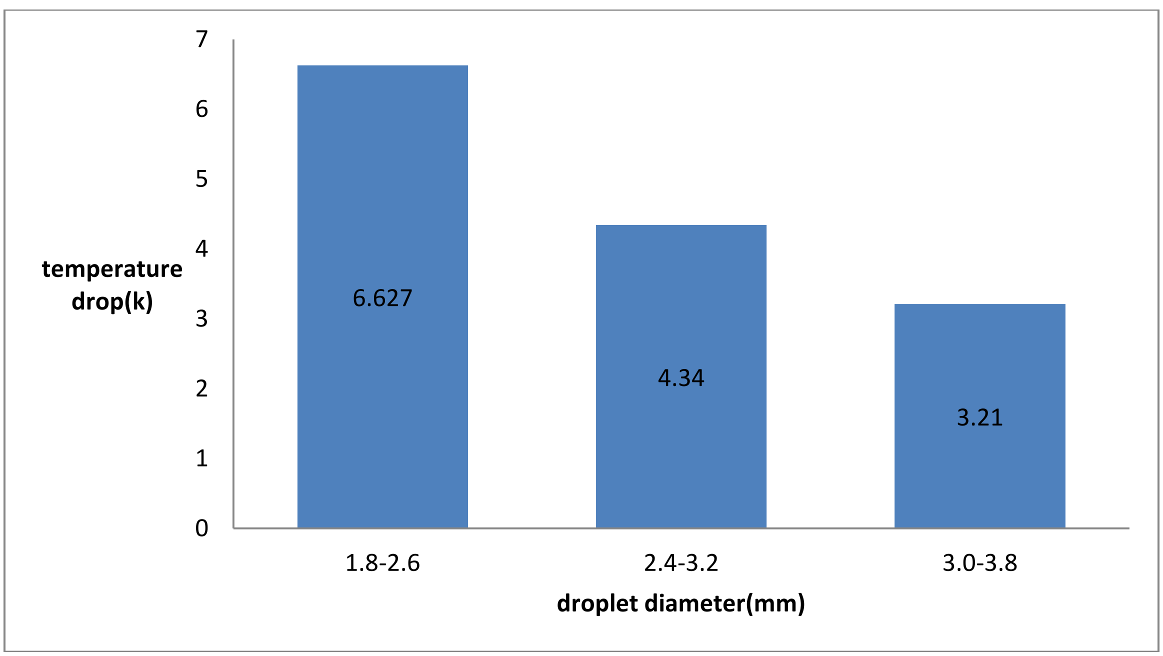


Fig: 6.4 represent temperature drop for 8.577m height rain zone.

For 6.777 m rain zone height

Droplet diameter	Water inlet temperature	Water outlet temperature	Temperature drop
1.8mm – 2.6mm	303	297.79	5.21
2.4mm – 3.2mm	303	299.592	3.408
3.0mm – 3.8mm	303	300.57	2.43

Table: 6.2 represent temperature drop for 6.777m height rain zone.

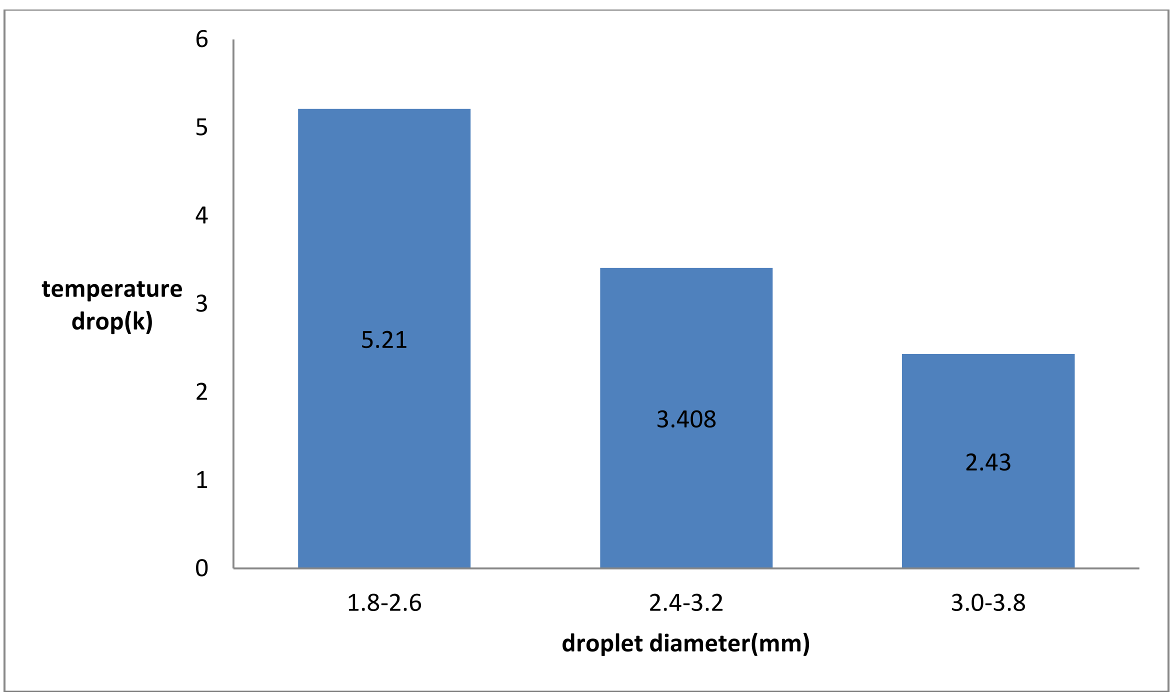


Fig: 6.5 represent temperature drop for 6.777m height rain zone.

For 4.977 m rain zone height

Droplet diameter	Water inlet temperature	Water outlet temperature	Temperature drop
1.8mm – 2.6mm	**303**	**298.924**	**4.076**
2.4mm – 3.2mm	**303**	**300.315**	**2.685**
3.0mm – 3.8mm	**303**	**301.116**	**1.884**

Table: 6.3 represent temperature drop for 4.977m height rain zone.

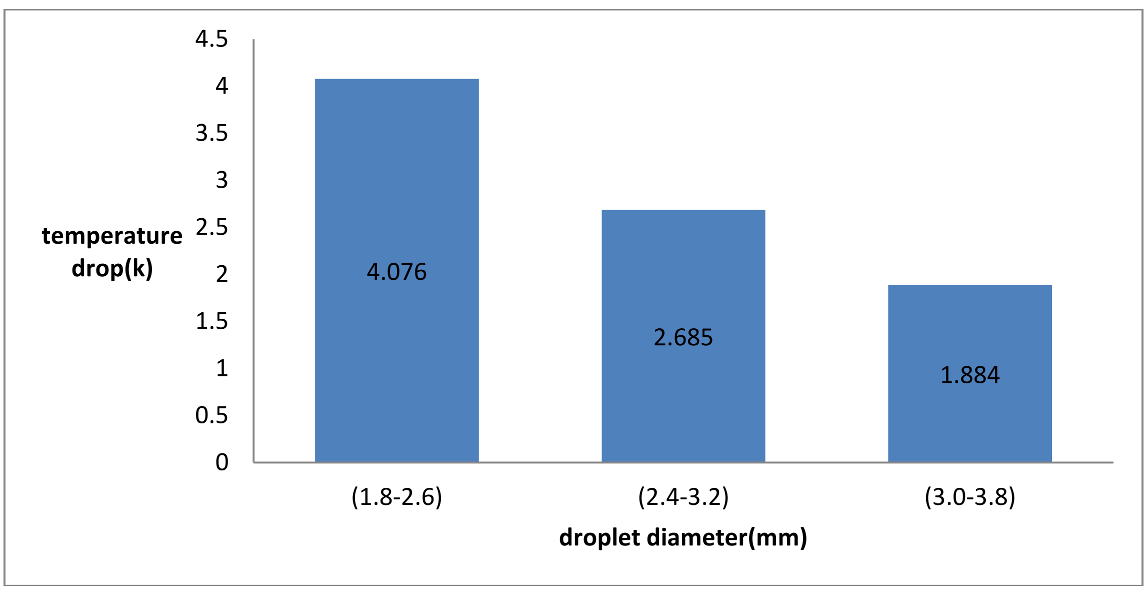


Fig: 6.6 represent temperature drop for 4.977m height rain zone.

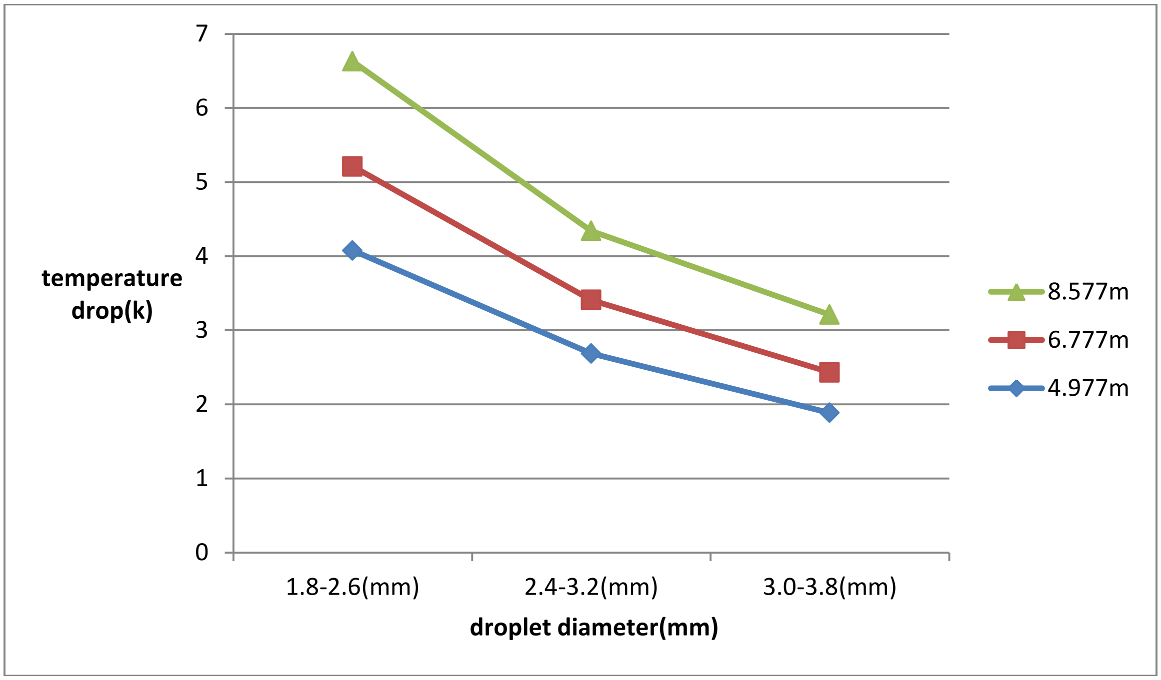


Fig: 6.7 comparison of temperature drop between different heights and different diameters.

CONCLUSION

On the basis of analysis done in this thesis, following are the conclusions:

1. Cooling range increases with decreases the diameter of the water droplet for any height of the rain zone, TABLE 6.1, TABLE 6.2 and TABLE 6.3 has shown this.
2. Cooling rate also increase with increase the height of rain zone.
3. By varying the diameter of water droplet with same value, rate of temperature drop is increases with increase of rain zone height, TABLE 6.4, TABLE 6.5 and TABLE 6.6 has shown this.
4. Fig: 6.7 shows that for larger height of rain zone, temperature drop rate increases with the decrease in droplet diameter.

FUTURE SCOPE

Rosin-rammler distribution can be used for different heights of fill zone and spray zone by varying the diameters. Combine analysis of cooling tower by rosin-rammler can also do in future.

REFERENCES

[1] A.S. kaiser, M. Lucas, A. Viedma, B. Zamora, "Numerical model of evaporative cooling processes in a new type of cooling tower", International Journal of Heat and Mass Transfer 48 (2005) 986–999.

[2] Alok Singh, S P S Rajput, "Application of CFD In Natural Draft WetCooling Tower", International Journal of Engineering Research And Applications 2 (2012) 1050-1056.

[3] G. Gan, S.B. Riffat, L. Shao, P. Doherty, "Application of CFD To Closed Wet Cooling Towers", Applied Thermal Engineering 21 (2001) 79-92.

[4] H.C.R. Reuter, D.G. Kroger "Computational Fluid Dynamics Analysis Of Cooling Tower Inlets", Journal of Fluid Engineering 133 (8) 081104-081104-12.

[5] Abdullah alkhedhair, Hal Gurgenci, Ingo Jahn, Zhiqiang Guan, Suoying He, "Numerical simulation of water spray for pre-cooling of inlet air in natural draft dry cooling towers", Applied Thermal Engineering 61 (2013) 416-424.

[6] Rafat Al-Waked, Masud Behnia, "CFD Simulation of Wet Cooling Towers", Applied Thermal Engineering 26 (2006) 382-395.

[7] N. Williamson ,M. Behnia, S. Armfield, "Comparison of A 2D Axisymmetric CFD Model of A Natural Draft Wet Cooling Tower And 1D Model", Elsevier, International Journal of Heat And Mass Transfer 51 (2008) 2227-2236.

[8] Jorge Facao, Armando C. Oliveira, "Heat And Mass Transfer In An Indirect Contact Cooling Tower", Taylor And Francis, University of Porto, Portugal, 11 July 2008.

[9] Hamid Saffari , S.M. Hosseinnia, "Two-phase Euler-Lagrange CFD simulation of evaporative cooling in a Wind Tower", Energy and Buildings 41 (2009) 991–1000.

[10] Bilal A Qureshi, Syed M Zubair "A Complete Model of Wet Cooling Towers With Fouling In Fills", Elsvier, Applied Thermal 26 (2006) 1982-1989.

[11] Naphon P., "Study of heat transfer characteristics of an evaporative cooling tower", International Communications in Heat and Mass Transfer, 32, 1066-74, 2005.

[12] N.J.Willamson "Numerical Modeling of Heat And Mass Transfer And Optimisation of a Natural Draft Wet Cooling Tower" PhD Thesis, The School of Aerospace, Mechanical and Mechatronics Engineering, The University of Sydney, 2008.

[13] D.J. Viljoen "Evaluation And Performance Prediction of Cooling Tower Spray Nozzles" PhD Thesis, University of Stellenbosch.

[14] D.G. Kroger "Air Cooled Heat Exchangers And Cooling Towers" Pennwela Corp. Tulsa, USA, 2004.

[15] D. Radosavljevic "The Numerical Simulation of Direct Contact Natural Draught Cooling Tower Performance Under The Effect of Cross Wind", PhD Thesis, University of London, 1990.

[16] H. Lowe, D.G. Christie "Heat Transfer And Pressure Drop Data On Cooling tower Packings And Model Studies of The Resistance of Natural Draught Cooling Towers To Airflow", International Heat Transfer Conference, Colorado.

[17] M.N.A Hawlader, B.M. Liu "Numerical study of The Thermal-Hydraulic Performance of Evaporative Natural Draft Cooling Towers", Applied Thermal Engineering 22 (2002) 41-59.

[18] R. Suresh Kumar, S.R. Kale, P.L. Dhar, "Heat And Mass Transfer Process Between Water Spray And Ambient Air Experimental data" Applied Thermal Engineering 28 (2008) 349-360.

[19] Viljoen D. J, “Evaluation and performance prediction of Cooling Tower spray zones”, M.Sc.(Engineering) thesis, University of Stellenbosch, South Africa, December 2006.

[20] Terblanche R, Reuter H.C, Kroger D.G, “Drop size distribution below different cooling tower fills”, Applied Thermal Engineering 29 (2009) 1552-1560.

[21] Nenad Milosavljevic, Perti Hejkkila “A Comprehensive Approach To Cooling Tower Design”, Pergamon, Applied Thermal Engineering 21 (2001) 899-915.

[22]http://www.cs.montana.edu/webworks/projects/stevesbook/contents/chapters/chapte001/section005/blue/page001.html

[23] Herbert W. Stanford III (2003). *HVAC Water Chillers and Cooling Towers: Fundamentals, Application, and Operation*. CRC Press,

[24] William Stanford, Gerald Bowen Hill (1972). *Cooling Towers: Principles and Practice: A Practical Guide to Cooling Tower Selection and Operation.* 2nd Edition, Carter Industrial Products,