INTERNATIONAL JOURNAL OF SCIENTIFIC PROGRESS AND RESEARCH (IJSPR) ISSN: 2349-4689
Issue 156, Volume 56, Number 01, February 2019

# Thermal & CFD Analysis of Induced Draught Cross Flow Cooling Tower

Bablu Singh[1], Praveen Mishra[2]

[1]MTech Scholar, [2]Assistant Professor

*Mechanical Engineering Department,Oriental College of technology,Bhopal*

***Abstract-- A cooling tower is an enclosed device for the evaporative cooling of water by contact with the air. Cooling tower is a heat rejection device. Common applications include cooling the circulating water used in oil refineries, petrochemical and other chemical plants, thermal power stations and HVAC systems for cooling buildings. The efficiency and the effectiveness of cooling tower is depend on number of parameter like inlet air angle, inlet and outlet temperature of air and water, fill materials, fan speed etc. In current work the air inlet angle is optimize by selecting three different inlet angles. For this CFD analysis of induced draught cross flow cooling tower is done in ANSYS workbench.***

## I. INTRODUCTION

A cooling tower is an enclosed device for the evaporative cooling of water by contact with the air. This is achieved partly by an exchange of latent heat resulting from the evaporation of some of the circulating water, and partly by a transfer of sensible heat. Cooling tower is a heat rejection device. Its main function is to extract waste heat from warm water to the atmosphere. Heat rejection in cooling tower is specified as convection between the fine droplets of water and the surrounding air, and also as evaporation which allows a small portion of water to evaporate into moving air. Therefore, the process involves both heat and mass transfer.

Fig. 1.1 principle of cooling tower

The principle of cooling the water in cooling tower is similar to the evaporative condenser. In evaporative cooling three processes go on simultaneously, viz,

I. The transfer by convection of sensible heat from warm water to colder air,

II. If the main body of the air is at a lower vapor pressure than that at the water surface, matter is transferred in the form of water molecules, and

III. There is a movement of heat from the bulk of the liquid to the surfaces. This last effect is generally ignored in cooling calculations since the thermal resistance to such the thermal resistance to such direct internal conduction is slight.

The rate of evaporation of water in cooling tower and reduction of water temperature depends upon following factors:

i. The time of exposure

ii. Amount of water surface area exposed

iii. The direction of air flow relative to water

iv. The relative velocity of air passing over the water droplets formed in cooling tower

## II. OBJECTIVE

From the problem statement we will increase the efficiency and effectiveness of induced draught cross flow cooling tower. For this we are taking into consideration of following parameter for optimization air inlet angles. For this we perform following steps for achieving objective

- Heat transfer calculation
- Modeling of induced cross flow cooling tower
- CFD analysis in ANSYS
- Experimental reading for validation of ANSYS result
- Optimization of air inlet angle

## III. CFD ANALYSIS OF COOLING TOWER

*CFD Methodology*

CFD may be used to determine the performance of a component at the design stage, or it can be used to analyses difficulties with an existing component and lead to its improved design. For example, the pressure drop through a component may be considered excessive: The first step is to identify the region of interest: The geometry of the region of interest is then defined. If the geometry already exists in CAD, it can be imported directly. The mesh is then created. After importing the mesh into the pre-processor, other elements of the simulation including the boundary conditions (inlets, outlets, etc.) and fluid properties are defined. The flow solver is run to produce a file of results which contain the variation of velocity, pressure and any other variables throughout the region of interest. The results can be visualized and can provide the engineer an understanding of the behavior of the fluid throughout the region of interest. This can lead to design modifications which can be tested by changing the geometry of the CFD model and seeing the effect.

The process of performing a single CFD simulation is split into four components:

1. Geometry/Mesh

2. Physics Definition

3. Solver

4. Post-processor

*Geometry*

The first task to accomplish in a numerical flow simulation is the definition of the geometry followed by the grid generation. This step is the most important step for the study of isolated impeller assuming an axis symmetric flow simplifies the domain to a single blade passage.

The geometry of the design needs to be created from the initial design. Any modeling software can be used for modeling and shifted to other simulation software for analysis purpose.

*Meshing*

Mesh generation (gridding) is the process of subdividing a region to be modeled in to a set of small control volumes. Associated with each control volume there will be one or more values of dependent flow variable (e.g. velocity, pressure, temperature etc.). Usually these represent some type of locally averaged values. Numerical algorithms representing approximation to the conservation law of mass, momentum and energy are then used to compute these variables in each control volume.

I. Create Cooling Tower Cavity Model of Different Geometry in Solid works 2009.

30 Degree Original Geometry

3D Model of 30 Degree Air Angle Original Geometry

Meshing is the method to define and brake up the model into small elements. In general a finite element model is defined by a mesh network, which is made up of the geometric arrangement of elements and nodes. Nodes represent points at which features such as displacements are calculated. Elements are bounded by set of nodes, and define localized mass and stiffness properties of the model. Elements are also defined by the number of mesh, which allowed reference to be made to corresponding deflections, stresses, pressures, temperatures at specific model location. The traditional method of mesh generation is block structure (multi-block) mesh generation. The block structure approach is simple and efficient technique of mesh generation.

15 Degree Modified Geometry 1

3D Model of 15 Degree Air Angle Modified Geometry 45 Degree Modified Geometry 2

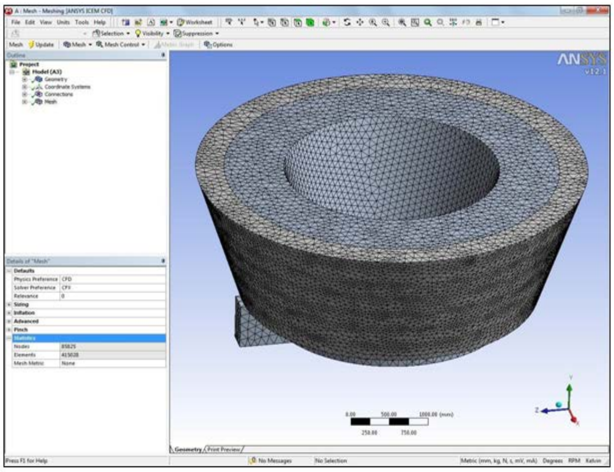
II. Create Mesh. Import Cavity 3D Model Cooling Tower in ANSYS Workbench Mesh Module.

Type of Mesh: - 3D

Type of Element: -

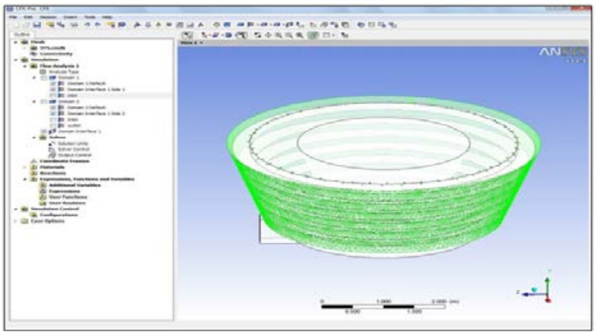
Tetrahedral No. of Nodes: - 85825

No. of Elements: - 415025



Meshed Model of Cooling Tower

III. Import meshed cavity model in ANSYS CFX for Preprocessing.

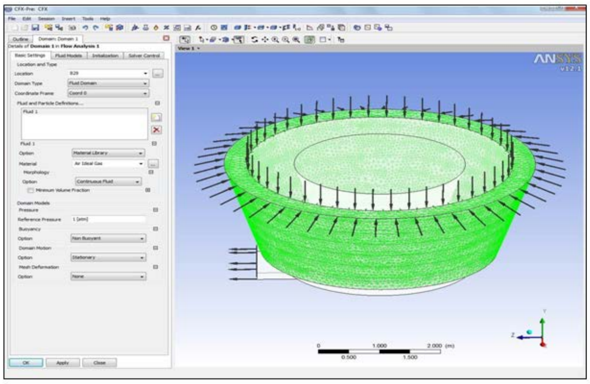


IV. Define Domain for

Air. Domain Type: -

Fluid Domain motion: -

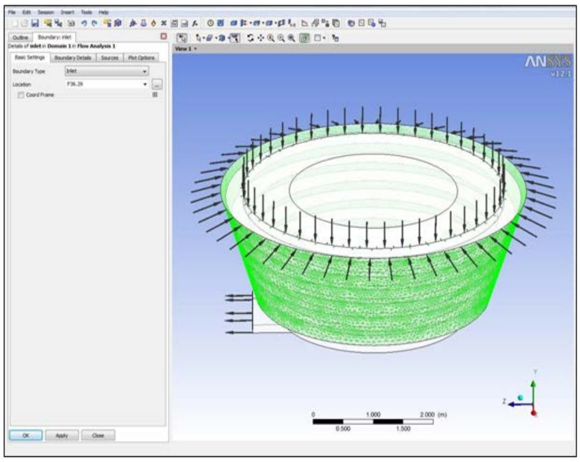
Stationary Domain Fluid: - Air



Air Domain

Define Air Domain Model for CFD Analysis Heat Transfer Model: - Total Energy Turbulence Model: - K-Ɛ

V. Define Inlet for Air Domain.



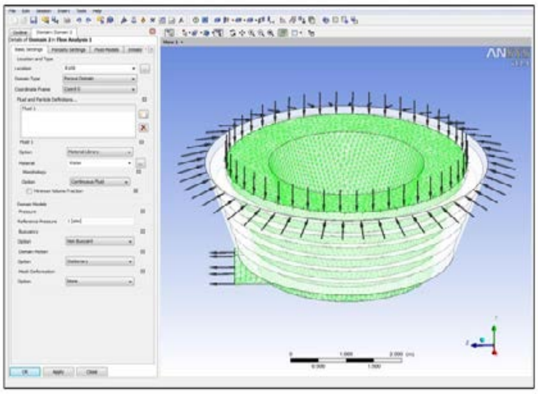
Define Inlet for Air Domain

VI. Define Water Domain.

Domain Type: -

Porous Domain motion: -

Stationary Domain Fluid: - Water



Water Domain

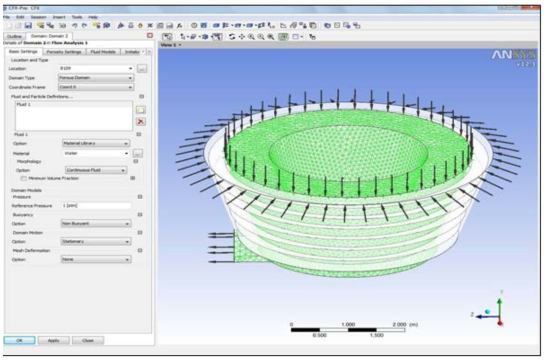
Define Water Domain Model for CFD Analysis

Heat Transfer Model: - Total Energy

Turbulence Model: - K-Ɛ

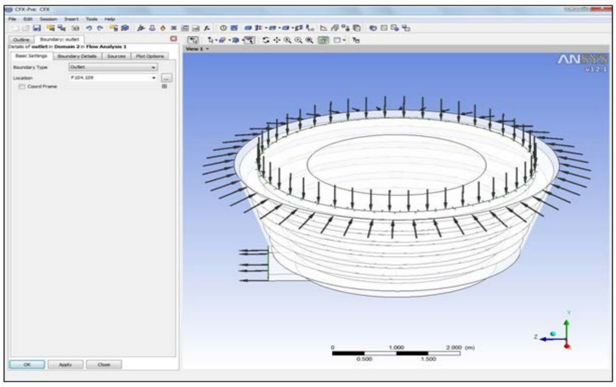
Define Volume Porosity: - 0.8

VII. Define inlet for Water Domain.



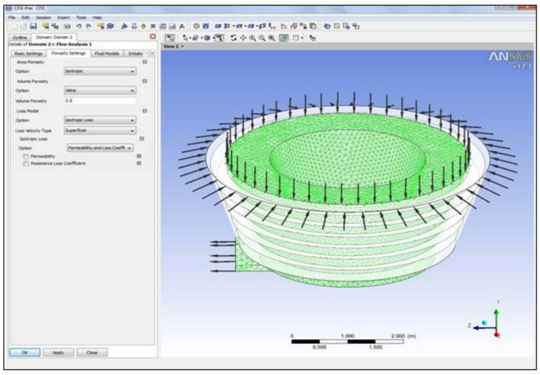
Define inlet for Water Domain

VIII. Define Outlet for Water Domain.



Outlet for Water Domain

IX. Define Fluid Porous interface.



Define Fluid Porous interface

10) Define Number of iteration and Residual target for CFD Analysis.

11) Run the Analysis.

## IV. CONCLUSION

The productivity plays a significant role in modern manufacturing competitive markets for industrial growth. The productivity is directly related to the profit level and also goodwill of the organization. Every manufacturing industry has aim to producing the large number of products with relatively lesser time. For increase the productivity they are going to use new manufacturing process with new material like as Injection molding with new plastic and rubber materials. The injection molding is one of the manufacturing process which produce Rubber parts uses plastic injection and Liquid Silicon Rubber (LSR) for the molten metal into the predefine dies. The mould for thermoplastic receives the molten plastic in its cavity and cool it’s to solidity to the point of ejection. The most is provided with cooling channels, the mould temperature is controlled by regulating the temperature of the cooling fluid and its rate of flow through the channels. Proper cooling or coolant circulation is essential for uniform repetitive mould cycling. Continuous overheating of the die is reducing the efficiency of the injection molding machine. So for increase the efficiency of the injection molding machine the heat produce during the manufacturing process will transfer most effectively from injection molding machine to the atmosphere.

A cooling tower is equipment used to reduce the temperature of a water stream by extracting heat from water and emitting it to the atmosphere. If the rate of heat transfer is reduced drastically by the interface of two metal pieces, so the efficiency and the effectiveness of cooling tower are decreases and the mould temperature is not maintained through the cooling tower then various defects are produced. Cooled water is needed for, for example, air conditioners, manufacturing processes or power generation. There are various parameters like inlet air angle, inlet and outlet temperature of air and water, porosity in fill material, fan speed that affect on temperature of the cooling tower. Current cooling tower is not providing uniform rate of heat transfer. Here to improve its performance, optimization of cooling tower parameters done successfully using the trial and error method. Air inlet angle is optimizing by selecting three different inlet angles. For this CFD analysis of induced draught cross flow cooling tower is done in ANSYS for practical reading. Trial and error method has proven to be a major tool in discovering, which parameters and interactions are significant to improve rubber injection molding process.

Based on the Trial and error method, it is found that the different air inlet angles are the significant parameter that has an effect on rate of heat transfer in cooling tower followed by cooling fluid and its rate of flow through the channels. Also from results of Trial and error method it can be concluded that optimum parameter to maintain the uniform temperature of cooling tower.

Out of this parameter inlet air angle is very effective parameter on the performance of the cooling tower. So we optimize this angle by CFD analysis of cooling tower. From the Water outlet temperature of cooling tower decreases as the air inlet angle decreases. Hence the cooling efficiency and effectiveness of cooling tower increases. Out of selected three air inlet angles, angle of 250 leads to maximum efficiency and effectiveness for selected cooling tower. On the other side, as the air inlet angle increases the water outlet temperature also increases and cooling efficiency and effectiveness of cooling tower decreases.

ANSYS and experimental result are compared and found in good agreement, thus proving that the increase heat transfer rate. After completing CFD Analysis Results, we can say that CFD Analysis is a good tool to avoid costly and time consuming Experimental Work. It also reduces the lead time of New Product Development Chain.

## FUTURE SCOPE

The work done is extended in following direction,

- Effect of Fill on the performance of cooling tower using CFD.
- Effect of Induced Draft Fan Capacity on the performance of cooling tower using CFD.
- Structural Analysis of Cooling Tower.
- Transient Analysis of Cooling Tower for finding how much time required to reach steady state condition.

## REFERENCES

[1] Numerical simulation of a wet cooling tower J. van der Merwel C.G. du Toit2 (First received October 2001; Final version May 2002)

[2] CFD Prediction of Forced Draft Counter-Flow Cooling Tower Performance Dr. Jalal M. Jalil*, Dr.Talib K.Murtadha** & Dr. Qasim S. Mehdi*** Received on: 16/12/2009 Accepted on: 16/2/2010

[3] PERFORMANCE ANALYSIS OF MECHANICAL DRAFT COOLING TOWER Si Y. Lee Savannah River National Laboratory si.lee@srnl.doe.gov 803-725-8462 Alfred J. Garrett Savannah River National Laboratory alfred.garrett@srnl.doe.gov803-725-4870 James S. BollingerSavannah River National Laboratory james02.bollinger@srnl.doe.gov 803-725-1417 Larry D. Koffman Savannah River National Laboratory larry.koffman@srnl.doe.gov 803-725-1038

[4] Numerical Simulation of the Performance Characteristics of the Hybrid Closed Circuit Cooling Tower M.M.A. Sarker1, E. Kim2, C.G. Moon2, J. I. Yoon2 1Department of Mathematics Bangladesh University of Engineering and Technology Dhaka-1000, Bangladesh masarker@math.buet.ac.bd 2Department of Refrigeration and Airconditioning Engineering Pukyong National University Namgu, Pusan 608-739, Korea Received:15.05.2007 Revised: 01.12.2007 Published online: 06.03.2008

[5] Heat and mass transfer in an indirect contact cooling tower: cfd simulation and experiment Jorge Faca˜o and Armando C. Oliveira Department of Mechanical Engineering, Faculty of Engineering, University of Porto, Portugal

[6] Performance evaluation of wet-cooling tower fills with computational fluid dynamics by yngvi gudmundsson