

Membrane Cooling Technique for Cooling of Air

Mithil Ram¹, Vivek V Kumar², NidheeshP³

P.G. Student, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India¹

Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India²

Assistant Professor, Dept. of Mechanical Engineering, Toc H Institute of Science and Technology, Ernakulam, Kerala, India³

ABSTRACT: Cooling the inlet air of the compressor of a gas turbine is a known technique to increase the power output of the turbine. Various cooling methods have been currently used to cool the inlet air, such as evaporative cooling, chiller cooling and absorption cooling. This paper is about a new method for cooling of air, known as membrane cooling. Micro porous membranes are used, in order to reduce the quantity of water required for humidifying the air. Porous ceramic tubes are used as membranes to humidify air with water. This paper involves the optimization of the above cooling system for maximum efficiency by varying the number of tubes, and changing the pattern of ceramic tubes using numerical simulation. The optimized model shows a further drop of about 25% in air temperature, when compared with the existing model. The simulation is done using C.F.D software package ANSYS FLUENT 6.0.

KEYWORDS: Gas Turbine, Porous Membrane, Ceramic tubes, Numerical Simulation

I. INTRODUCTION

Combustion turbines have ambient temperature sensitivity, both the capacity and efficiency decreases as the ambient temperature increases. The mass flow rate is proportional to the density of the inlet air, which is inversely proportional to the absolute temperature. Thus when the temperature of air is decreased the mass flow rate of the air increases thus increasing the power output of the turbine. Various methods are being used to reduce the temperature of the inlet air in which, evaporative cooling and mechanical refrigeration have been extensively applied, and are well developed and documented. This paper explains about a new method for cooling of air in which micro-porous membrane are used to cool the air with a low quantity of water. The micro-porous membrane used here are porous ceramic tubes which are having a porosity of 40%. The tubes are made of alumina. The existing experimental data obtained from the literature is validated and optimization of the model is done with the help of Computational Fluid Dynamics using ANSYS Fluent. This paper mainly focuses on the simulation of the existing experimental procedure with the help of the computer and to optimize the experimental test section for more effectiveness with the help of the computer.

II. PROBLEM DEFINITION

Various methods of cooling have been already discussed in the previous sections. This paper is about the study and optimization of membrane cooling method, which uses porous ceramic tubes to humidify the air with water. As mentioned by O. Zeitoun, et.al [4], the current effectiveness of the cooling system is about 40%, which can be further improved by changing the diameter of the tube or by increasing the number of tubes. Hence, a study is done to optimize the effectiveness of the existing model by increasing the number of tubes or decreasing the tube diameters.

III. COMPUTATIONAL FLUID DYNAMICS

Fluid (gas and liquid) flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy. Computational Fluid Dynamics (CFD) is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers. CFD gives an insight into flow patterns that are

difficult, expensive or impossible to study using traditional (experimental) techniques. As a rule, CFD does not replace the measurements completely but the amount of experimentation and the overall cost can be significantly reduced. Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows by means of Mathematical modelling (partial differential equations), Numerical methods (discretization and solution techniques) and Software tools (solvers, pre- and post-processing utility). The 3 steps of C.F.D simulation with the help of the software are discussed in the following sections.

1. Pre-Processing

The computational domain that has to be analysed is designed in this step. It is done with the aid of modelling software, ANSYS GAMBIT. The dimensions to design the domain are obtained from O. Zeitoun, et.al [4]. The complete test section has been adopted for simulation.

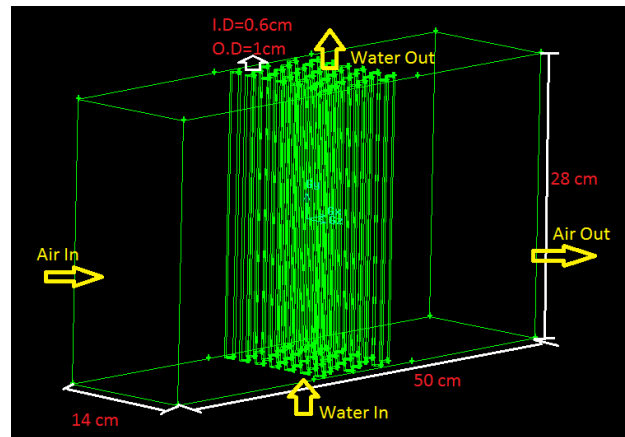


Fig.2. Isometric view of the Computational Domain.

The next step in pre-processing is the discretization of the domain, which is decomposing the domain into cells or volumes. This phenomenon is called meshing. Meshing of the domain produces nodes on which the iteration is done for a converging value. The domain of this work consists of two mediums, air and water. The mesh type used in this paper is Tetra Hybrid, T-Grid. The total number of nodes was about 1,58,913.

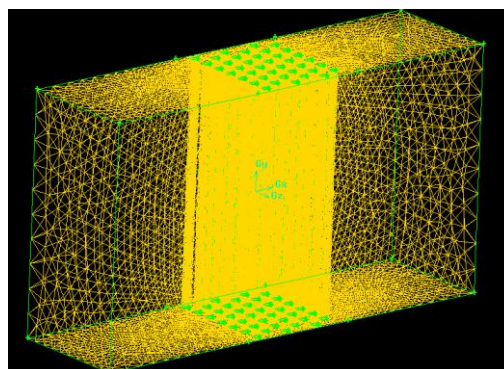


Fig.3. 3-D mesh of the computational domain.

2. Processing

This is the second step in C.F.D. In this step the governing equations for the iterations, Numerical algorithm Turbulence models, boundary and continuum types, boundary conditions for porous media(ceramic tubes), air and water are discussed. Grid Independent study and the code validation are also done in this step.

a)Governing equations

The Governing equations used for the computational simulation are :

Continuity Equation

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{1}$$

Where u, v, and w are the velocity components in x, y, and z directions.

Momentum Equation

(2)

$$\left[\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right] \times \rho = - \frac{\partial p}{\partial x} + \rho g_x + \mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right]$$

$$\left[\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right] \times \rho = - \frac{\partial p}{\partial y} + \rho g_y + \mu \left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right] \tag{3}$$

(4)

$$\left[\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right] \times \rho = - \frac{\partial p}{\partial z} + \rho g_z + \mu \left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right]$$

Along x, y and z directions respectively.

Energy Equation

$$\rho C_p \left[\left(\frac{\partial T}{\partial t} \right) + u \left(\frac{\partial T}{\partial x} \right) + v \left(\frac{\partial T}{\partial y} \right) + w \left(\frac{\partial T}{\partial z} \right) \right] = k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + S \tag{5}$$

b) Boundary and Continuum Types.

- Air and water inlet-Velocity Inlet
- Air and water outlet –Pressure outlet
- Walls of duct and tubes-Wall

c)Boundary Conditions

- Air Inlet Velocity: 3.6 m/s, Water Inlet Velocity: 0.0167m/s
- Air Inlet Temperature: 312K, Water Inlet Temperature: 303 K
- Wall (Duct and Porous): no slip, u=v=w=0
- Porous Media-Porosity: 40%, Inertial Resistance: $9.72 \times 10^7 \text{ m}^{-1}$, Viscous Resistance: $1.9 \times 10^{16} \text{ m}^{-2}$

d) Numerical Algorithm and Turbulence Models

The numerical algorithm used for the simulation is a pressure based algorithm and is **SIMPLE**. The viscous model used for the simulation is k-ε Standard. “k” is the turbulence kinetic energy and “ε” is its dissipation rate.

e) Grid Independent Study

Grid Independent study is the process in which the optimum mesh size for the meshing of fluid domain is selected. In the above process four mesh sizes were selected for doing the Grid Independent Study. They were 0.0026 mm, 0.0028 mm 0.0030 mm and 0.0032 mm. The first 2 mesh sizes were coarse and the next two were fine mesh sizes. In the above study, it was found that the mesh size of 0.0028 had less deviation in velocity magnitude at two points (error of 3.45%).The maximum allowable error percentage is 10%. Hence it was selected as the optimum mesh size for simulation.

f) Code Validation

Water Velocity(m/s)	Air Inlet Temp.(⁰ C)	Air Outlet Temp.(Experiment) ⁰ C	Air Outlet Temp.(C.F.D) ⁰ C
0.0167	38.64	29.14	29
0.0229	39.21	32.33	31

Table.1 Code Validation

From the table it can be summarized that, the temperature drop was found to be 9.64⁰c in simulation when compared to 9.5⁰c of experimental results.

IV. OPTIMIZATION

O. Zeitoun, et.al [4] commented that the efficiency of the cooling system can be improved if the numbers of tubes were increased or the diameters were reduced. As commented in the literature, the optimization of the same was done in simulation by varying different parameters and it was found that only increasing the number of tubes will result in improvement of temperature drop. All other parameters were rejected because of the high pressure drop at the inlet and outlet of air, when compared to the existing model.

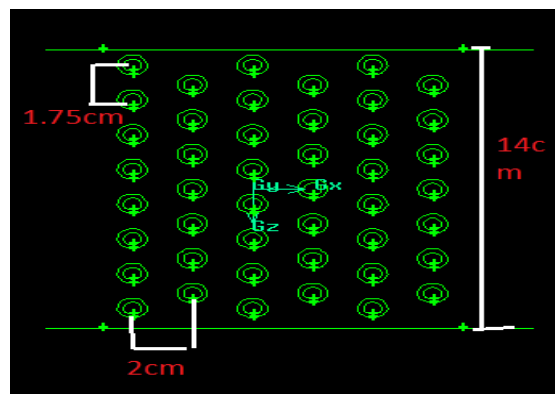


Fig .4. Number of tubes increased to 45 tubes (Top View).

The numbers of tubes were added to the same column of tubes, in the existing model by reducing the distance between adjacent tubes. Now the number has been increased to 45 tubes.

V. RESULTS

No. of Tubes	Air Inlet Temp.(⁰ C)	Air outlet Temp.(⁰ C)	Temperature Drop(⁰ C)	Pressure Drop (Pascal)
39	39	29.36	9.64	130
45	39	26.61	12.21	331

Table 2. Comparison of results

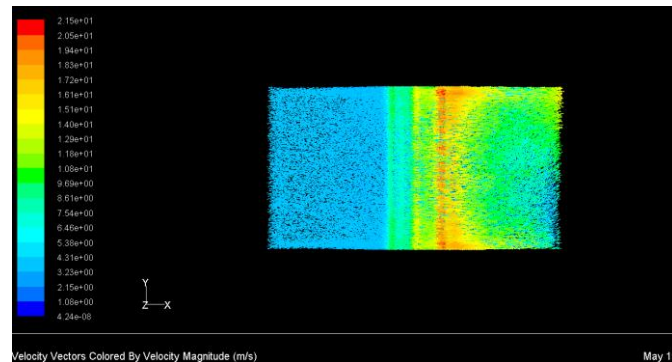


Fig.5. Velocity Vectors of air and water of the optimized model

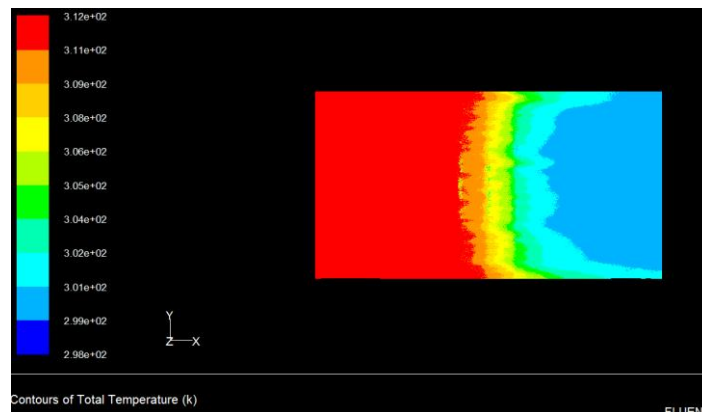


Fig.6. Temperature Contour of air of optimized model

VI. CONCLUSION

The study about the new method for cooling of air was done. The micro-porous membranes can be used to cool the air with a low quantity of water. Various properties of ceramic tubes were also studied. Zeitoun, et.al [4] used a matrix of 39 micro-porous tubes with a number and was able to achieve a temperature drop of 9.5⁰C from the inlet air temperature. In this work, the model which Zeitoun, et.al [4] did, was optimized using C.F.D, by increasing the number of tubes to 45, to the existing design. The optimized model achieved a temperature drop of about 12.2⁰ C in simulation, which was 25% more than the temperature drop achieved when 39 tubes were used.

REFERENCES

- [1] Ferziger, J. H.; Peric, M. (2001), "Computational Methods for Fluid Dynamics", Springer-Verlag.
- [2] Mohamed Ali, Obida Zeitoun, Hany Al-Ansary, and Abdullah Nuhait, "Humidification Technique Using New Modified Mini-Module Membrane Contactors for Air Cooling". Research Article, Advances in Mechanical Engineering, Article ID 174016, 2013.
- [3] Anders Berg, "Numerical and experimental study of the fluid flow in porous medium in charging process of stratified thermal storage tank "2013.
- [4] O. Zeitoun, M. Ali, H. Al-Ansary, A. Nuhait, "Ceramic tubes membrane technology as a new humidification technique for gas turbine inlet air cooling", International Journal of Thermal Sciences 1-10, 2014.