International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-8, Issue-1, May 2019

Computational Fluid Dynamics Simulation Study on Isothermally Heated Surface Applied Air and Water as a Collant Medium

Abhinav, Indhudhar T, Jest Arun R S

***Abstract*: Numerical investigations (Computational Fluid Dynamics) have been carried out on the two-dimensional isothermal heated surface using Ansys software. A comparative analysis of boundary layer development viz. velocity, thermal boundary layer, etc. was examined under the free stream velocity of air and water. Results revealed that water exhibit enhanced heat transfer against air over the isothermally heated surface. Heat transfer is mainly governed by initial conduction between the fluid particles and change in surface heat transfer and pressure drop is the direct consequence of velocity gradient.**

***Index Terms*: CFD, Boundary layer, Convective diffusion.**

I. INTRODUCTION

The use of Computational Fluid Dynamics (CFD) within the gas turbine engine and design of the thermal system (not limited to) has become a common practice, where airflow, thermal sluggishness, velocity, and pressure contours are easily visualized, monitored and optimized to maximize thermodynamic efficiency [1]. The addition of coolant air is essentially a common engineering practice where the air is extracted from the compressor as a result decrease in thermodynamic efficiency is realized [2]. It is a good exercise if the optimal amount of air is allowed to bleed without trading off the thermodynamic efficiency. Series of an experiment carried out on overall cooling effectiveness on a flat plate and both film cooling and impingement cooling in hot gas conditions and found that overall cooling effectiveness found to be maximum when the blowing ratio was around 1.0 and the overall cooling effectiveness decreases as the coolant continue to increase [3].

Revised Manuscript Received on 22 May 2019.

***** Correspondence Author

Abhinav*, Assistant Professor, Department of Mechanical Engineering, M.Tech. (Ph.D.) Affiliation: Alliance College of Engineering and Design, Alliance University, Bangalore, India.

Indhudhar T, Pursuing M.Tech. in Thermal Power Engineering , Affiliation : Visvesvaraya Technological University ,Center for P.G. Studies, Mysuru, India.

Jest Arun R S, B.Tech in Mechanical Engineering, Affiliation: Alliance College of Engineering and Design, Alliance University, Bangalore,India.

Jesuraj Felix [4] has studied the cooling performance of combustor liner of a gas turbine, using a flat plate model using Matlab and Ansys software (conjugate heat transfer analysis) and found that increase in effectiveness with the increase in blowing ratios (0.5 -2.5) and also, claim that an increase in density ratio leads to a decrease in the overall cooling effectiveness marginally. In another research work heat transfer the analysis is carried out on a porous plate using air like a hot gas stream and water as coolant medium. In the study, it was found that the increase in Reynolds number causes an increase in surface temperature and a decrease in the cooling efficiency of the porous plate [5]. Ehab Hussein Bani-Hani [6] has done investigations on the boundary-layer theory of fluid flow past a flat-plate using Matlab and compared the Matlab codes with the solutions published in the literature. In another work, CFD analysis has been carried out on the effect of Reynolds number on local heat transfer distribution for jet impingement on a smooth plate by incompressible chevron jet and found that an increase in Reynolds number the Nusselt number increases at any given nozzle to plate distance (z/d) [7]. In another paper, investigations were carried out on rectangular plate slamming, both the theoretical and experimental works were carried out focused on the hydrodynamic response on the two flat plates structural arrangement and found that a good agreement between numerical investigation with experimental data for the force components, local peak pressure, and peak pressure propagation velocity [8]. Apart from the above works, many research reported and extended to flat plate solar collector where numerical techniques and analytical techniques are used to study the thermodynamic performance characteristics [9, 10]. The objective of the present analysis is to investigate the change in the effect of dynamic characteristics (velocity, pressure and boundary layer development) when air and water used as a coolant and applied on isothermally heated plate. The work carried out in CFD simulation (academic version) software and also, a great deal of scientific discussion made on the boundary layer signatures for the above said coolant medium. The results obtained has great significance may help the scientific community and design engineers in designing of heat exchangers, solar panels cooling system, turbine blades and can be extended to the design of any thermal systems.

II. METHODOLOGY

A. Analysis Details

Comparative analysis is carried out in pursuit of boundary layer development viz. velocity and thermal boundary layer, convective heat transfer coefficient, Nusselt number, heat flux and skin friction over an isothermally heated surface. Air and water were used as a coolant medium and free stream constant velocity maintained and applied from the leading edge.

B. Physical model and boundary

A two-dimensional geometry is created to investigate the fluid flow over the isothermally heated surface. The schematic of the model and applied boundary conditions are shown in Fig.1 & 2 respectively.

The surface length assumed to be 2mts. The fluid (air and water) made to enters at the constant speed of 1.5m/sec, at 300K (Pr=0.6). from the extreme left side and leaves from the right side to the atmosphere. Impermeable boundary and no-slip wall conditions have been implemented over the surface. The temperature of the plate/surface maintained at 450 Kelvin. The expected boundary layer thickness is calculated for the physical domain using the formula as shown in equation 1. The height of the domain has been set ten times the boundary layer thickness. In the present analysis 0.2 meters assumed.

$$\frac{\delta_{99}}{x} = \frac{5}{\sqrt{Re_x}} \qquad \text{Eq.1}$$

Where δ_{99} is boundary layer thickness,

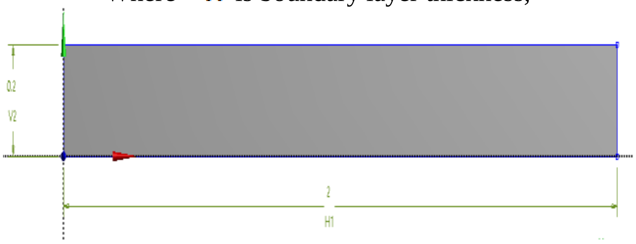

Fig.1. Physical model

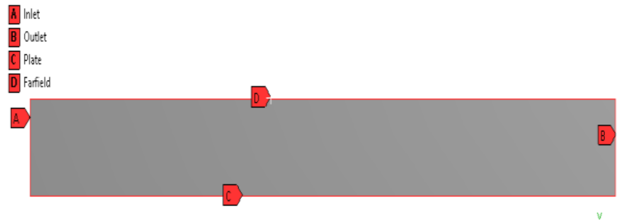

Fig.2. Boundary Conditions

C. Numerical method

The numerical simulations were carried out using FLUENT CFD Software that uses the finite-volume method to solve the governing equations. Meshing has been created in the CFD model with quad shapes refer Fig.3.

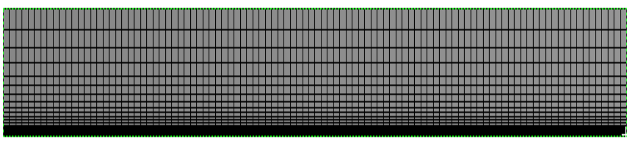

Fig.3.Meshed Model

The following assumptions were made to investigate the fluid flow and heat transfer characteristics over isothermal surface and are as follows:

Assumptions:

1. Steady two-dimensional fluid flow and heat transfer.
2. The flow is laminar and incompressible.
3. Constant fluid properties.
4. Body forces and viscous dissipation are ignored.
5. Negligible radiation heat transfer.

Based on the above assumptions, the fluid flow over the surface is governed by the continuity, the Navier- Stokes equations and the energy equation. In the Cartesian tensor system of the equations are shown in Equation 2, 3 & 4.

Continuity Equation:

$$\frac{\partial(\rho u_i)}{\partial x_i} = 0 \qquad \text{Eq.2}$$

Momentum Equation:

$$\frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]}{\partial x_j} \qquad \text{Eq.3}$$

Energy Equation:

$$\frac{\partial(\rho u_i T)}{\partial x_i} = \frac{\partial\left(\tau \frac{\partial T}{\partial x_j}\right)}{\partial x_j} \qquad \text{Eq.4}$$

Where τ is thermal diffusivity and is given by $\tau = \frac{\mu}{P_r}$

The governing equations were discretized by the second-order the upwind scheme, decoupling with the SIMPLE algorithm and solved using a finite volume approach. The solutions were considered to be converged when the normalized residual values were less than 10^{-6} for energy and 10^{-3} for momentum variables. The physical properties of the air and water have been assumed to remain constant at average bulk temperature. Table 1 show the air and water properties respectively.

Table: 1 Fluid properties

Air at 300 K	
Density	1.225
Specific heat C_p(J/Kg-K)	1006.43
Thermal conductivity(W/m-K)	0.0242
Viscosity (Kg/m-s)	1.7894e^{-05}
Water at 300 K	
Density	998.2
Specific heat C_p(J/Kg-K)	4182
Thermal conductivity(W/m-K)	0.6
Viscosity (Kg/m-s)	0.001003

III. RESULTS AND DISCUSSIONS

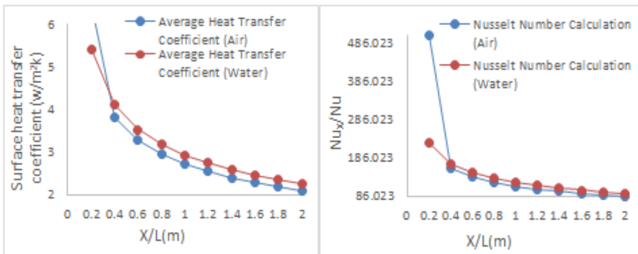

Fig.4. Average Heat Transfer Coefficient and Nusselt Number Comparison Air and Water

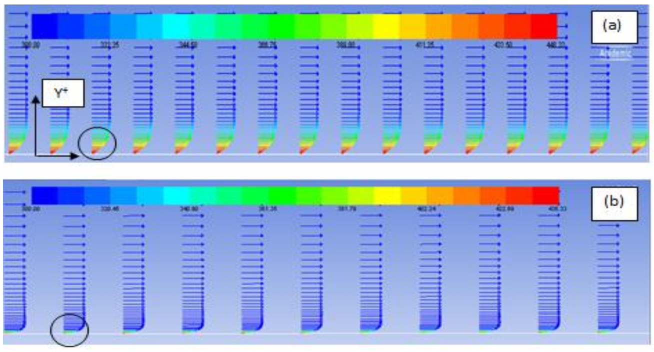

Fig.5.Vector representation of thermal boundary layer (a) Air (b) Water

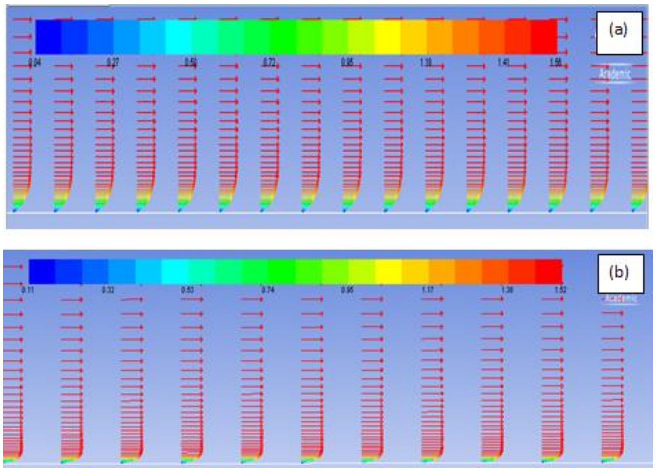

Fig.6. Vector representation of velocity boundary layer (a) Air (b) Water

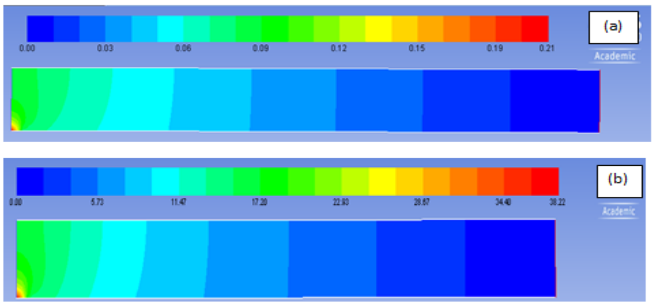

Fig.7.Pressure drop (a) Air (b) Water

A sudden rise in Nusselt number, 486.023 and 223.519 at x=0.2 m acknowledge in the present analysis for both air and water respectively. The similar initial effect observed in case of heat transfer coefficient. An improved heat transfer observed in the case of water, used as a coolant medium. An abrupt escalation in surface heat transfer coefficient and Nusselt number at x=0.2 m acknowledge and attributed to the initial thermal instability, (initial fluid particle contact with isothermally heated plate) as the fluid flow further the fluid particles exchange energy by the adjoining fluid layer and gradually establishing thermal equilibrium. The thermal equilibrium can be seen by the gradually descending nature of the curve, refer Fig 4.The phenomenon can be also understood as follows: As the fluid enters at the leading edge where the temperature of the surface is more compared to fluid no slip condition happens due to which the fluid ceases for a moment and when fluid flows further the fluid particles tries to achieve thermal equilibrium with the adjoining layers.

Due to conduction heat transfer (between the first fluid layers to the heated surface) at the outset sudden rise in "h" and "Nu" followed by gradual decreasing pattern attributed to the thermal equilibrium. In the study, it has been understood that rapid convection is possible by adopting water instead of air near the surface. It has been found that at the beginning, the free stream of air and water remains the same. However, as soon as air enters to the leading edge of isothermally heated surface significant thermal gradient developed in y^+ direction. The temperature drops from 448.33 to 300 K and 436.33 to 300 K observed in case of air and water respectively. The vector representation of concentrated thermal boundary layer for both the fluid medium air & water is shown in Fig.5 (a) & (b) respectively. The slope of the tangent line both in case of the thermal boundary layer and the velocity boundary layer is more compared to water. The slope gradually increases from y+ equals to zero to a definite value refer to Fig.5 (a). Due to the significant magnitude of the thermal boundary layer lesser heat transfer takes place in case of water. When water was used as coolant a decrease in thermal boundary layer noticed refer Fig.5 (a) & (b) thus enhancing the heat transfer rate. The vector representation of the thermal boundary layer and velocity boundary layer found to be different for both the fluid medium refer Fig 5 & 6. A very similar kind of effect noticed in the velocity boundary layer with a significant change in slope in case of air but not in water. Velocity found to be zero at the surface and gradually increases in y+ direction in the hydrodynamic layer. In case of water fluid medium viscous sublayer found to be extended more compared to air reason attributed to relatively less shear stress in the case of water where particles of water found to cease at the points on the surface and stay for a little longer duration of time compared to air refer Fig.6 (b). The development of the velocity boundary layer is only possible only when there exist in thermally excited fluid flow. The thickness of the velocity boundary layer and thermal boundary layer mainly depends on the Prandtl number. It has been noticed that there exist a significant change in pressure drop at the leading edge.
However, a gradual pressure drop acknowledges in both cases. Pressure drop found relatively more in case of water reason attributed and understood that change in pressure and heat transfer are consequences of a velocity gradient. A velocity gradient yields a shear force which is overcome by a sacrifice of fluid pressure.
On the other hand, a velocity gradient gives rise to convective diffusion.

IV. CONCLUSIONS

The present numerical investigations carried out in pursuit of enhance heat transfer over isothermally heated surface by utilizing two fluid medium air & water at a fixed free stream velocity of 1.5 m/s. The following conclusions can be made from the above studies

1. Heat transfer is mainly governed by initial conduction between the fluid particles and the isothermally surface followed by convective heat transfer among the adjoining layers in hydrodynamic boundary layers.

2. At a given free stream velocity 1.5 m/s water exhibit enhanced heat transfer against air over isothermally heated surface.
3. The change in surface heat transfer and pressure drop is the direct consequence of velocity gradient also, velocity gradient give rise to convective diffusion.

REFERENCES

1. Juan Manuel García- Guendulain et al., "Numerical Study on the Effect of Distribution Plates in the Manifolds on the Flow Distribution and Thermal Performance of a Flat Plate Solar Collector", *Energies*, vol.11, Issue 5, 2018.
2. Ou, S. et al., "Shaped-Hole Film Cooling With Pulsed Secondary Flow," *ASME Paper* No. GT 2006-90272, 2006.
3. Bo Shi et al., "Overall Cooling Effectiveness on a Flat Plate with Both Film Cooling and Impingement Cooling in Hot Gas Condition", *ASME Turbo Expo: Turbomachinery Technical Conference and Exposition,* vol. 5A: Heat Transfer, June 13–17, 2016.
4. Jesuraj Felix et al., "Experimental and Numerical Investigation of Effusion Cooling Performance Over Combustor Liner Flat Plate Model", *Journal of heat transfer engineering*, Taylor and Francis, 23 April, 2018.
5. M.Kilic, "Numerical Investigation of Heat Transfer From A Porous Plate With Transpiration Cooling", *Journal of Thermal Engineering*, Yildiz Technical University Press, Istanbul, Turkey vol. 4 No. 1, January, 2018, pp. 1632-1647.
6. Ehab Hussein Bani-Hani et al., "Boundary-Layer Theory of Fluid Flow past a Flat- Plate: Numerical Solution using MATLAB", *International Journal of Computer* Applications, vol.180 – No.18, February 2018. Harpreet Singh et al., "CFD Analysis of Effect of Reynolds Number on Local Heat Transfer Distribution for Jet Impingement on Smooth Plate by Incompressible Chevron Jet", *International Journal of Engineering Technology Science and Research*, vol. 5, Issue 3, March 2018, pp.807-812.
7. Jose D. Mesa et al., "Numerical Investigation of Rectangular Flat Plate Slamming", *7th European Conference on Computational Flui Dynamics* (ECFD 7), Glasgow, UK, June 2018, pp.11-15.
8. Dawit Gudeta Gunjo et al., "CFD and experimental investigation of flat plate solar water heating system under steady state condition", *Renewable Energy*, vol.106, June 2017, pp. 24-36.
9. VibhootiVaibhav Singh et al., "Computational Evaluation of Thermal and Hydraulic Characteristics of Flat-Plate Solar Collector for Different Glazing Material", *Materials today Proceedings*, vol.5, Issue 14, Part 2, 2018, pp. 28211-28220.

AUTHORS PROFILE

Abhinav, Assistant Professor, Department of Mechanical Engineering, M.Tech. (Ph.D.) Affiliation: Alliance College of Engineering and Design, Alliance University, Bangalore, India. Email: abhinavtechno5@gmail.com
.

Indhudhar T, Pursuing M.Tech. in Thermal Power Engineering , Affiliation : Visvesvaraya Technological University ,Center for P.G. Studies, Mysuru, India.

Jest Arun R S, B.Tech in Mechanical Engineering, Affiliation: Alliance College of Engineering and Design, Alliance University, Bangalore,India.