

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Special Issue 3, March 2018

## Review Paper on Thermal Analysis of Water Pipe Using CFD Technique

Rushikesh N. Sonune

Department of Mechanical Engineering, Sanmati College of Engineering, Washim, India

**ABSTRACT:** The present review is aimed at studying the effect of different system parameters on the heat transfer and buoyancy induced flow inside circular tubes using CFD. They include tube length, tube diameter, tube inclination and heat supplied. Constant heat flux boundary condition is created on the tube surface. The heat transfer coefficient and the induced flow rate both were found increasing with increase in the heat supplied and the angle of inclination with the horizontal. The flow rate of water is found to be independent of the tube length if the tube length is more than 1m. The Experimental results are Compared with the CFD results. Boundary condition, Geometry is created in GAMBIT and Solver, Boundary conditions are provided to the pipe in fluent software.

**KEYWORDS:**CFD, Fluent, Gambit, Buoyancy, Inclined tube, Laminar flow.

### I.INTRODUCTION

Convection is a process which involves mass movement of fluids. Natural convection occurs due to temperature difference which produces the density difference which results in mass movement. For example, assume a plate which is maintained isothermal at temperature  $T_w$  and the surrounding temperature is  $T_\infty$ . On getting heated, the fluid near the wall moves up due to the effect of buoyancy and this hot fluid is replaced by cold fluid moving towards the wall. Hence a circular current is set up due to density difference. There is a boundary layer adjacent to the plate where the velocity and temperature vary from plate to free stream. Initially the velocity increase with increasing distance from the surface and reaches a maximum and then decrease to approach zero value. This is because of action of viscosity diminishes rapidly with distance from plate, while density difference decreases more slowly. The causes of convection can be described as either natural (free) or forced convection. In this work the water will be the working fluid while the heating for the tube by using electrical Current. The temperature difference and the velocity in the outlet tube will be determined. This work is on the Circular Copper Tube having varying length and diameter. Computational fluid dynamics (CFD) techniques can be used to perform analyses to identify the relative performance of Natural Convection concepts. This study would provide us the first step to solve Natural Convection problem in an Inclined Pipe. The CFD simulation analysis will be carried out on an inclined pipe to determine the fluid flow Velocity, Temperature across the tube in the different inclination.

### II.EXPERIMENTAL SET-UP

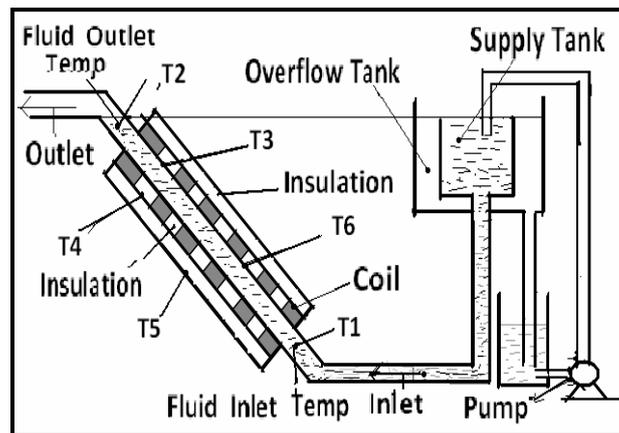
A schematic of the Experimental set-up is shown in Figure 1. It consists of a long copper tube called test section, provided with a heating coil, to create uniform heat flux boundary condition on the tube surface. The power supplied is measured directly with the help of pre-calibrated wattmeter (accuracy  $\pm 1\%$ ). Copper Constantan Thermocouples (accuracy  $\pm 0.2^\circ\text{C}$ ) were used to measure all the temperatures. The flow rates were measured using calibrated measuring jar ( $\pm 1\%$ ) and stop watch. The inlet end of the tube is connected to a constant head water tank. The exit end of the tube is maintained at the same elevation as that of the water in the tank so that, under isothermal conditions, no water flows through the tube. When heat supply is given the water in the tube gets heated and flow is established because of buoyancy effect. The flow is measured with the help of a calibrated measuring jar. The constant head tank permits measurement of flow induced due to buoyancy alone.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Special Issue 3, March 2018



**Figure 1: Schematic of Experimental Set-Up**

The experimental set-up was calibrated for the heat loss estimation and is also validated for the inclined position of the tube (12 mm tube diameter and 1 m length at 30 degrees inclination) for which results are available in the literature [13]. The comparison of the results (refer Table 1) show that the two values are in good agreement with each other.

**Table 1: Validation of the Experimental Set-up**

Sr No	Reference	Gr	Nu	Flow ml/min
1	Prayagi [13]	1397.1	3.1	21
2	Present study	1462.7	3.4	22

### III. TUBE HEAT-LOSS MEASUREMENT

The measurement of the tube heat-loss was undertaken in a temperature-controlled room without direct solar radiation. Figure 3 shows the experimental set-up. The tube is held vertical, its open end insulated with a 100-mm thick polystyrene cover. The getter (sealed end of the tube) has a higher heat loss value, and is also insulated with a polystyrene cover. Five T-type thermocouples were placed at different axial positions along the tube. The tube was filled with hot water, and was monitored as it cooled down. The starting hot water temperature is 85-90 °C (corresponds to a temperature difference with the ambient of ~ 65 °C). The experiments were done on 8 tubes from two production batches, and two of the tubes were tested twice to ensure repeatability of the measurement. The heat loss coefficient of the absorber tube was calculated based on the following correlation.

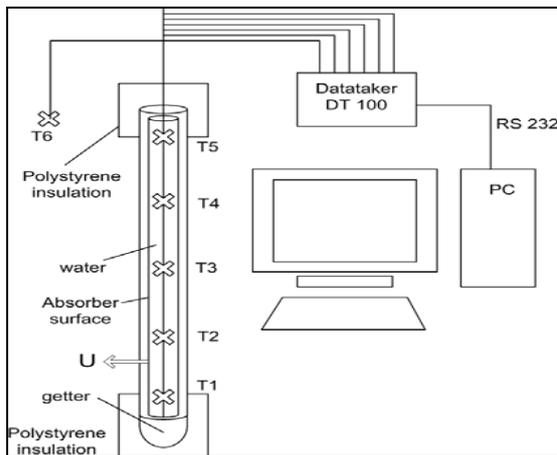
$$U_t = \frac{m c_p (t_i - t_f)}{A_a (t_m - t_a)_{av}}$$

## International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

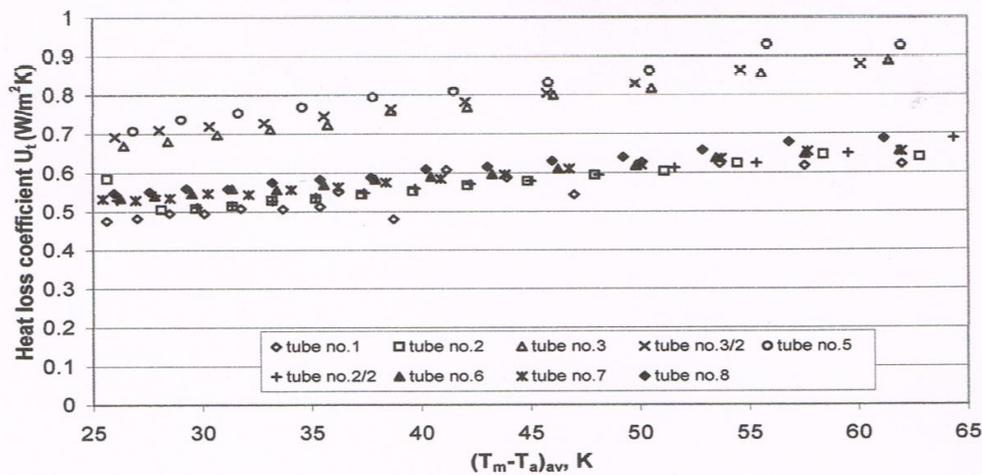
Vol. 7, Special Issue 3, March 2018



**Figure : Indoor experimental setup: Tube heat measurement**

**Figure : Experimental setup: Optical efficiency loss measurement**

Performance of a Water-in-Glass Evacuated Tube Solar Water Heater Budihardjo et al. Proceedings of Solar 2002 – Australian and New Zealand Solar Energy Society 3 where  $m$  is the mass of water inside the tube (1.28 kg) divided by the period of averaging (3600s),  $c_p$  is the specific heat of water (J / kg K),  $t_i$  and  $t_f$  are the average water temperature inside the tube, obtained by averaging 5 thermocouple readings, at the start and at the end of the test,  $A_a$  is the surface area of the absorber and  $(t_m - t_a)_{av}$  is the average difference between the mean water temperature and the ambient temperature over the one hour period. The test results were plotted in Figure 5. The coefficient varies slightly with temperature difference. It also shows that there are variations of tube quality. For good quality tubes, the heat loss coefficient varies from 0.5 W/m<sup>2</sup>K (at  $T_m - T_a = 25$ K) to 0.65 W/m<sup>2</sup>K (at  $T_m - T_a = 65$ K). For lower quality tubes coefficient varies from 0.7 W/m<sup>2</sup>K to 0.9 W/m<sup>2</sup>K for the same temperature range.



# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Special Issue 3, March 2018

## IV. CFD ANALYSIS

The working fluids considered in the present case are refrigerants as in tube condensation of refrigerants has lot of applications in refrigeration and air conditioning industry. The refrigerants are selected such that they cover the wide pressure range and variety. R407C is selected as it is high pressure mixture with large temperature glide. To calculate the average properties, the condensation temperatures considered are 40°C, 50°C and 60°C.

### A. Modeling of the Tube

3-D tube is modeled in GAMBIT and exported to FLUENT. Analysis is done for grid independent study and the optimum mesh size is selected as 24X80. The meshed model of pipe is shown in Figs. 1 and 2.

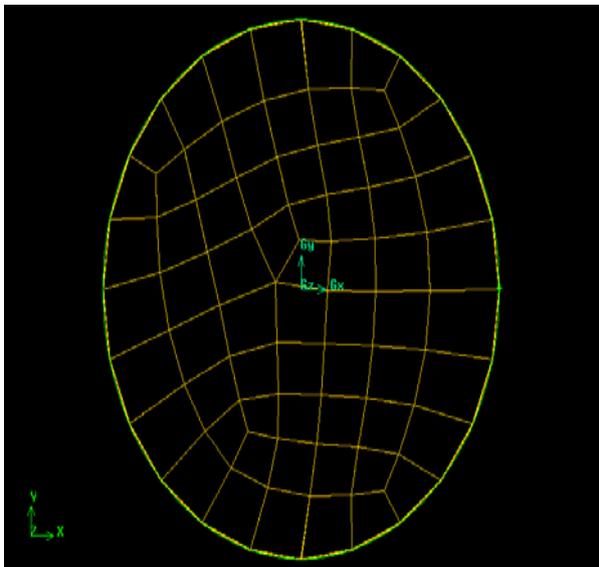


Fig. 1 Front view of the 3-D mesh model

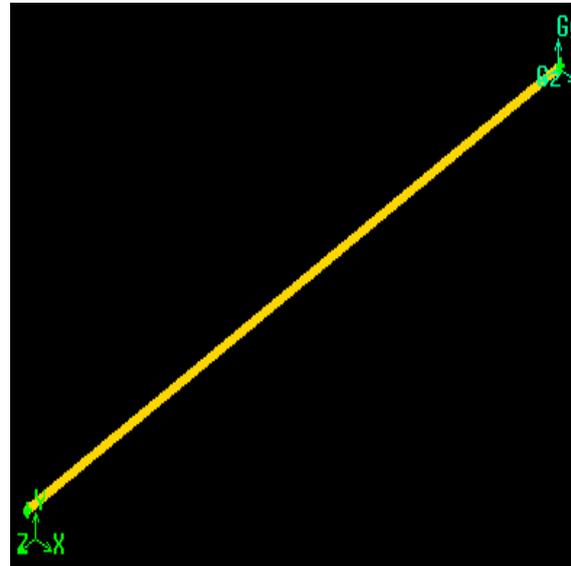


Fig. 2 Isometric view of the meshed 3-D pipe

### B. Selection of Solver

3-D Segregated model with implicit formulation is selected. The solver conditions are set for steady flow with enhanced wall treatment. Adiabatic conditions are considered at the wall.

### C. Selection of Viscous Model

Since for the entire range of flow rates, the Reynolds Number based on the average properties exceeds 2300,  $k-\epsilon$  model is considered.

### D. Material Properties

With these solver conditions set, for each quality  $x$  average properties are calculated for each refrigerant and material properties are given. Simulations are performed for different average kinematic viscosity values calculated using equations, (9) – (11).

### E. Setting of Operating Conditions

The operating pressure is set as per the saturation pressure considered for each refrigerant. Since the flow rates considered fall under turbulent – turbulent case, the gravity is not considered in the flow analysis.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Special Issue 3, March 2018

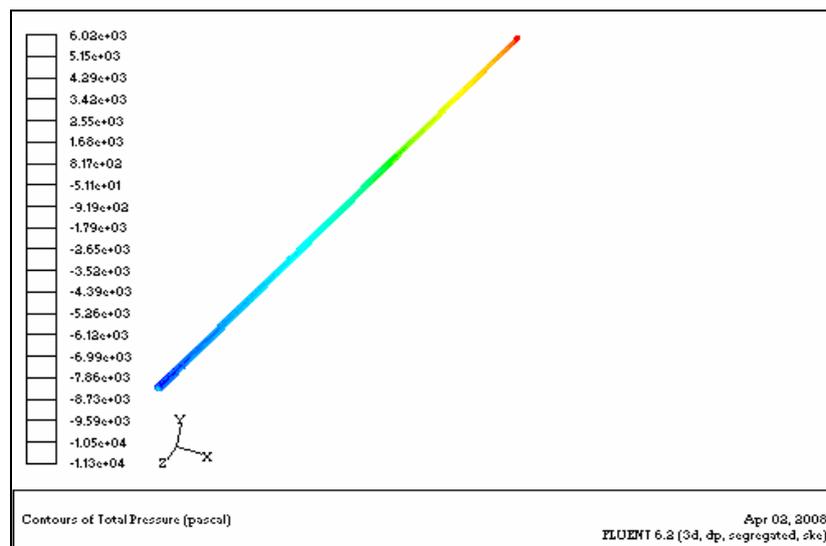
## F. Applying the Boundary Conditions

- Mass flow inlet: Mass flow rates of 0.01 kg/s to 0.06kg/s is considered. The flow rates selected are to cover the entire flow regimes for a horizontal tube.
- Outflow condition: At the outlet, outflow condition is given as fluid is not left to atmosphere.
- Wall: No slip condition is given at the wall.

## G. Solution Controls

SIMPLE algorithm is selected for Pressure – Velocity coupling. Second order upwind method is selected for discretization of momentum, turbulence K.E and turbulence dissipation rate. For pressure, standard discretization scheme is selected.

## H. Convergence



## V. DISCUSSION

The results of the present work lead to the following conclusions:

First, a validation exercise was performed by comparing the computed convective heat transfer coefficients ( $h_c$ ) for laminar air flow over flat plate by Computational Fluid Dynamics to analytical solutions. The CFD simulations were performed for constant wall temperature conditions. The CFD results showed a 5% error with the analytical solutions, indicating a performance of the CFD code, at least for the cases studied. A grid sensitivity analysis was performed on the mesh for laminar air flows. The convective heat Transfer coefficient ( $h_c$ ) was calculated over a flat plate. The boundary-layer thickness decreases with distance from the leading edge of the plate and increases with Reynolds number. It is observed that the variation of nusselt number is linear till the Reynolds number increases to 5876. After that the variation of nusselt number with increasing Reynolds number does not remain linear. The effect of increasing Reynolds number and Prandtl number on flow tends to increase average and local nusselt number. It is observed that the variation of nusselt number is linear till the Reynolds number increases to 5876. After that the variation of nusselt number with increasing Reynolds number does not remain linear. The effect of increasing Reynolds number and Prandtl number on flow tends to increase average and local nusselt number. Computational Fluid Dynamics can be used to determine the convective heat transfer coefficients.

# International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: [www.ijirset.com](http://www.ijirset.com)

Vol. 7, Special Issue 3, March 2018

## VI. CONCLUSION

In this Review, a two-dimensional forced convection laminar air flow over flat plate is analyzed using the finite element method through solving partial differential equations of the fluid flow. . The fluid flow is expressed by partial differential equation (Poisson's equation). While, heat transfer is analyzed using the energy equation. This paper demonstrated the use of CFD for heat flow in forced laminar air flow over flat plate. This paper also gives a few guidelines on the use of CFD to perform such calculations. As a result, the CFD models can be used with confidence for cases similar to the ones described here.

## REFERENCES

- [1] S. V. Prayagi, "Parametric Studies on Buoyancy Induced Flow through Circular Pipes in Solar water heating system" International Journal of Engineering Science and Technology (IJEST).
- [2] Henderson D. "Experimental and CFD investigation of an ICSSWH at various inclinations", Renewable and Sustainable Energy Review 11 (2007) 1087-1116.
- [3] Senthilkumar R. "Performance Analysis of Heat Pipe using Copper Nanofluid with Aqueous Solution of n-Butanol" International Journal of Mechanical and Materials engineering 2010.
- [4] Bhramara P. "CFD Analysis of Two Phase Flow in a Horizontal Pipe Prediction of Pressure Drop" International Journal of Aerospace and Mechanical Engineering 3:2 2009.
- [5] Hyo Min Jeong, "Natural convection heat transfer estimation from a longitudinally finned vertical pipe using CFD", Journal of Mechanical Science and Technology 23 (2009) 1517-1527.
- [6] Arun K Sukumaran, "Fluid Flow Simulations within Rotating Annulus", Proceedings of the 37th International & 4th National Conference on Fluid Mechanics and Fluid Power", December 16-18, 2010, IIT Madras, Chennai, India.
- [7] R.H.Thombre, S.B.Thombre, G.K.Awari, "Parametric studies on the transfer and BUOYANCY induced flow inside inclined tubes."
- [8] I.Budihardjo, G.L.Morrison, M.Behnia, "Performance of a water in glass evacuated tube solar water heater."
- [9] P.Bhramara, V.D.Rao, K.V.Sharma, T.K.K.Reddy, "CFD analysis of two phase flow in a horizontal pipe-prediction of pressure drop"
- [10] Rajesh Khatri, Pankaj Agrawal, Mohan Gupta, Jitendra Verma, " Laminar flow analysis over a flat plate by computational fluid dynamics.