ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013

CFD Analysis of Natural Convection Flow through Inclined Pipe

Rupesh G.Telrandhe, R.E.Thombre
Department of Mechanical Engineering Rajiv Gandhi College of Engineering Research and Technology, Chandrapur, India

Abstract- The present work 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 is to be Compare 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. Heat transfer by natural convection inside a circular tube has large number of applications in industries. Solar water heaters, cooling of gas turbine blades, heat exchangers, cooling of electric transformers etc are a few examples to cite. The present study is carried out for the specific application, i.e. domestic solar water heating system. In domestic solar water heating system, copper tubes; called risers are kept inclined to horizontal depending on the latitude of the place. Solar radiation falls on the collector surface and heats up the water inside by natural convection. The change in the density, due to change in the temperature causes water to flow through the risers. Presently all the dimensions are standardized but it is shown that these dimensions do not yield the best results. The literature is reviewed for analytical and/ or experimental studies on the heat transfer and flow characteristics inside a tube, of different cross-sections and subjected to different boundary conditions on the heat transfer surfaces. In this research the experimental results is compare with CFD results. For the CFD, Fluent and Gambit tool is used. As the GAMBIT is a Pre processing and FLUENT is Post-Processing Software. The Inclined pipe is mesh using Quadratic Elements. In this paper, effect of tube length, tube diameter and tube inclination with horizontal, on the heat transfer and fluid flow is studied. 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 ±1%). Copper Constantan Thermocouples (accuracy ±0.20C) were used to measure all the temperatures. The flow rates were measured using calibrated measuring jar (±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.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013

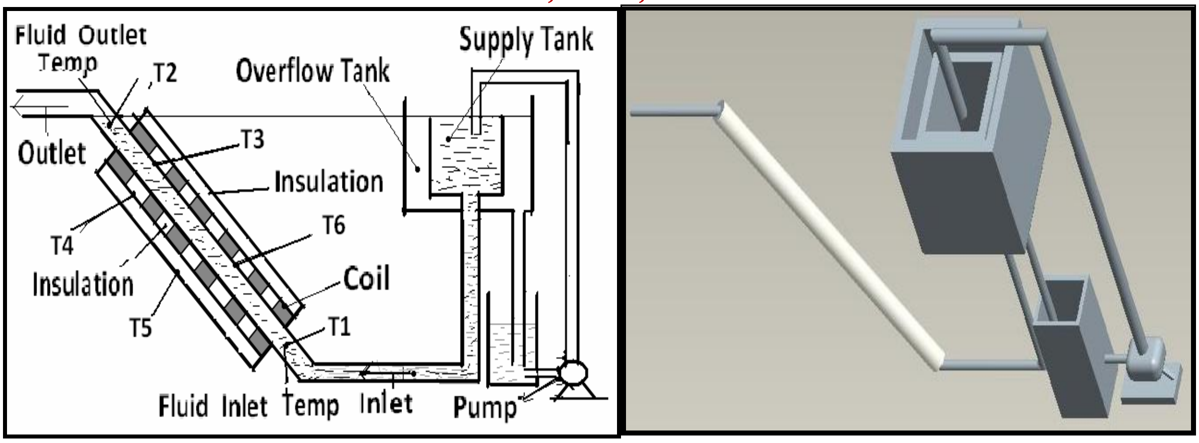


Fig 1: Schematic of Experimental Set-Up

III. EXPERIMENTAL PROCEDURE

The water level in the supply tank is maintained initially with the outlet of the test section as shown in figure 1, so that no water flows through the test section even when the supply tank is overflowing. The constant heat is supplied to the coil. The temperature of fluid in the pipe increases and causes fluid to flow in the upward direction due to buoyancy. The hot water is replaced by cold water overflowing supply tank ensures that the flow set-up in the test section is because of buoyancy effect only. Once the steady state is reached, the flow and temperature became stable and readings are taken. The energy balance is checked between the heat gained by the flowing fluid and the heat supplied minus the heat loss.

Table 1: Parameters during the Experimentation

Sr. No	Parameter	Specifications
1	Tube length	1 m
2	Tube diameter	16 mm
3	Tube inclinations	45^{o}
4	Inner Insulation thickness	8 mm (Asbestos Rope)
5	Outer Insulation thickness	8 mm (Asbestos Rope)
6	Locations of thermo-couples	1. At the inlet and outlet of test section 2. At different locations on the surface of tube 3.On the inner and outer surfaces of outer insulation 4.for measuring ambient and heater temperature
7	Type of thermocouples used	T - type (Copper and constantan)
8	Heat supplied	20 Watts to 240 Watts in step of 20 Watts

IV. COMPUTATIONAL FLUID DYNAMICS (CFD)

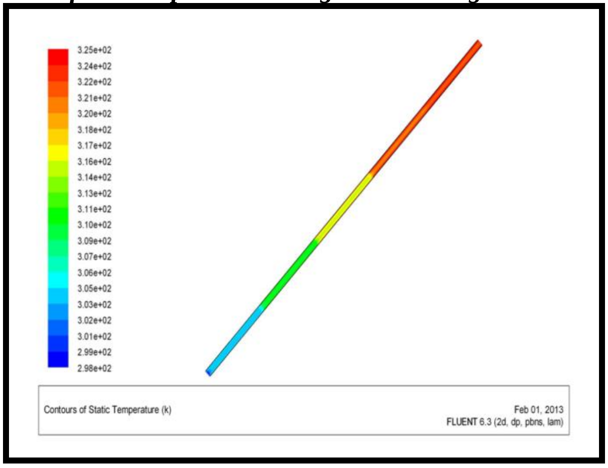
Computational Fluid Dynamics (CFD) is becoming a critical part of the design process for more and more companies. CFD makes it possible to evaluate velocity, pressure, temperature, and species concentration of fluid flow throughout a solution domain, allowing the design to be optimized prior to the prototype phase. This research contains Gambit and Fluent. GAMBIT means Geometry and Mesh Building Intelligent Toolkit. It is a single integrated Pre-processor for CFD analysis. It constructs the geometry and import using STEP, Para solid, and IGES etc import. It generates mesh for all fluent solvers. Gambit is modeling software that is capable of creating meshed geometries that can be read into FLUENT and other analysis software. The FLUENT computational fluid dynamics (CFD) solver has undergone extensive development to extend its robustness and accuracy for a wide range of flow regimes. FLUENT is a state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT is very leading engineering software provides a state of the art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving the flow problems with unstructured meshes that can be generated about complex geometries with relative

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 2, Issue 2, March 2013
ease. Supported mesh types include 2D triangular/ quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge, and mixed (hybrid) meshes. Once a grid has been read into FLUENT, all refining operations are performed within the solver. These include the setting boundary conditions, defining fluid properties, executing the solution, refining the grid viewing and post processing the results.

V. RESULT AND DISCUSSION

The steady state data generated on the experimental set-up for different values of tube inclination, tube length and heat supplied is correlated for heat transfer and flow characteristic as described below.

A. Surface Temperature along the tube length



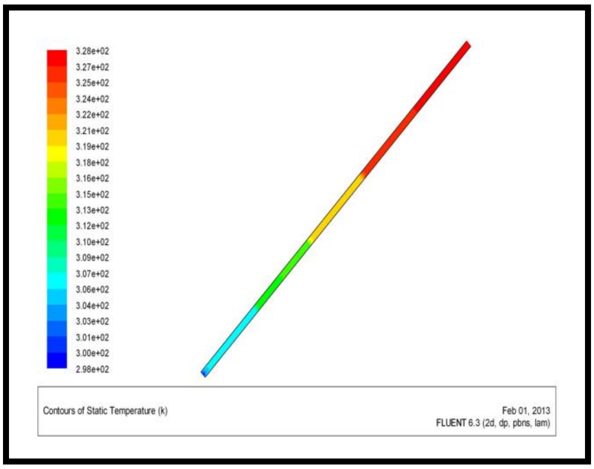
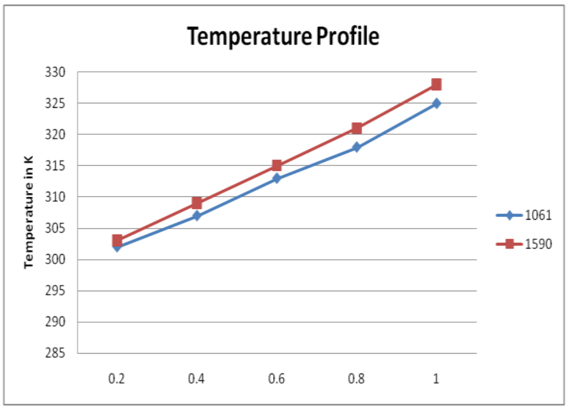


Fig.2 Temperature contour of 1 m length & Heat flux 1061w/m2

Fig.3 Temperature contour of 1 m length & Heat flux 1590w/m2



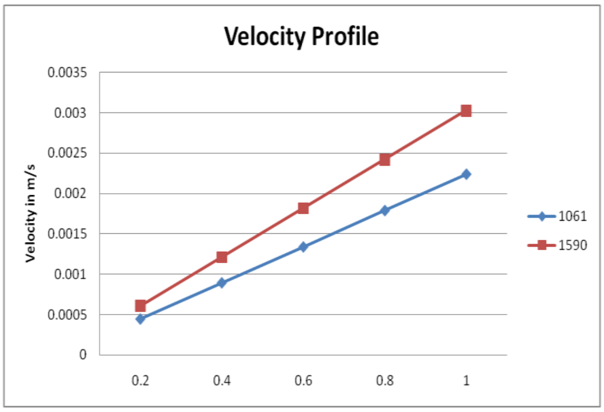


Fig.4 Temperature Profile of 1 m length

Fig.5 Temperature Profile of 1 m length

Table 2: Comparative Results of Temperature

Sr.No.	Heat Flux (W/m^2)	Inclination	Length & Diameter	CFD Results of Temperature	Experimental Results of Temperature
1	1061	45 ^{o}	1 M,16mm	52 ^oC	50.4 ^oC
2	1590	45 ^{o}	1 M,16mm	55 ^oC	54.5 ^oC

VI. CONCLUSION

From this work it is observed that

- The Temperature and Velocity both increases with increases in the heat supplied and length of the tube.
- The Software and Experimental results are comparatively matched to each other.
- The degree of accuracy of the CFD software solution will check with the experimental Results.

REFERENCES

[1] Henderson D.," Experimental and CFD investigation of an ICSSWH at various inclinations", Renewable and Sustainable Energy Review 11 (2007) 1087–1116.

[2] Rajesh Khatri," Laminar flow analysis over a flat plate by Computational fluid dynamics", International Journal of Advances in Engineering & Technology, May 2012, ISSN: 2231-1963.

[3] Sakr R.Y."Experimental and Numerical Investigation of Natural Convection Heat Transfer in horizontal Elliptic Annuli", Journal of Applied Sciences Research, 4(2): 138-155, 2008 INSInet Publication.

[4] Edlabadkar R. L." Computational Analysis of Natural Convection with Single V-Type Partition Plate" 5th European Thermal-Sciences Conference, The Netherlands, 2008.

[5] Senthilkumar R." Performance Analysis of Heat Pipe Using Copper Nanofluid with Aqueous Solution of n-Butanol" International Journal of Mechanical and Materials Engineering 1:4 2010.

[6] 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.

[7] Perumal Kumar," A CFD Study of Turbulent Convective Heat Transfer Enhancement in Circular Pipe flow" World Academy of Science, Engineering and Technology 68 2012.

[8] Sahu M.," Developed Laminar Flow In Pipe Using Computational Fluid Dynamics",7th International R&D Conference on Development and Management of water and Energy Resources,4-6 Feb.2009,Bhubaneshwar,India.

[9] Bhore V.R (November 5 -11, 2005) "Performance Comparison of Flat Plate Collectors Fitted with Non Circular Risers with Integral Fins", ASME International Mechanical Engineering Congress And Exposition, Orlando, Florida.

AUTHORS PROFILE

1. R. G. Telrandhe

Student of M.Tech IV Semester at Rajiv Gandhi College of Engineering Research and Technology Chandrapur, India.

2. Prof.R.E.Thombre

Associate Professor at Rajiv Gandhi College of Engineering Research and Technology Chandrapur, India