Int. J. Mech. Eng. & Rob. Res. 2013

International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 2, No. 3, July 2013

Research Paper

CFD ANALYSIS OF NATURAL CONVECTION FLOW THROUGH VERTICAL PIPE

Prashant M Khanorkar[1] and R E Thombre[1]

Corresponding Author:* ***R E Thombre,* ✉ *Rethombre1@rediffmail.com*

In this study, CFD analysis of the vertical tube is conducted. A vertical copper tube having constant cross section area is used for representing the medium through which natural convection of water takes place. In this present work the study and analysis of natural convection flow of water through vertical pipe is done. In this study includes what is the effect of the physical parameters of tube like diameter, length and heat flux on the outlet flow parameters like velocity and temperature. Constant heat flux is boundary condition is provided on the entire tube surface. In this study we found that outlet temperature outlet velocity that is going to be increased as tube length is increased but as diameter of pipe is increased outlet temperature is increased but velocity is decreased. In this thesis GAMBIT is used for creating geometry and specifying boundary conditions as well as FLUENT software is used as solver and giving boundary condition. Finally CFD results and experimental results are compared which validate the software results.

Keywords: CFD, FLUEN , GAMBIT, VERTICAL PIPE, Natural convection flow

INTRODUCTION

Natural convection flow of liquid is process of heat transfer mostly occurs due to density difference caused by temperature gradient. The process of natural convection through vertical tube has many applications like nuclear reactor, water tube boiler, solar water heating systems, HVAC applications, etc. The problem regarding in this there is no proper design criteria about diameter, length, heat flux and material of the tube. This present work mostly useful for the design the water tube boilers and solar water heating systems. In these applications vertical copper tubes are used for the generation the steam where the flow of water takes place only due to natural convection flow. In these applications constant heat flux are provided for heating of copper tube. The copper are mostly used for the maximum heat transfer to water. As the due to

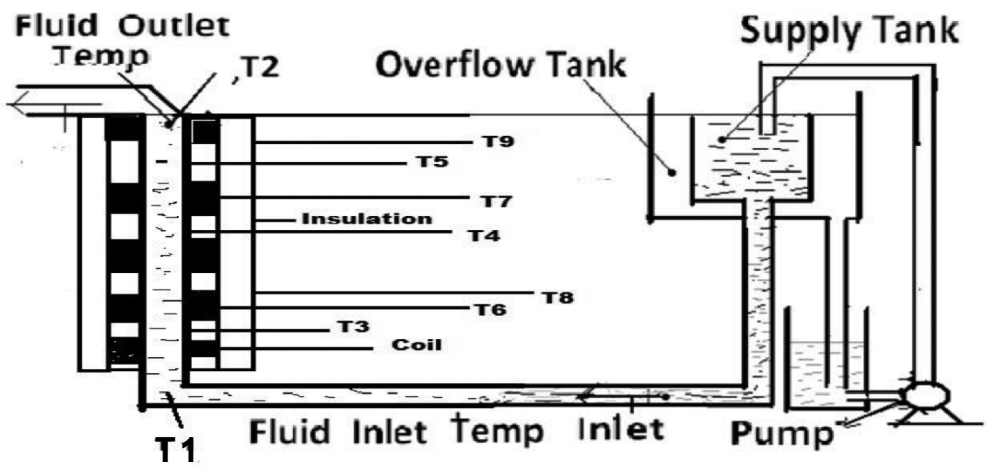
[1] Department of Mechanical Engineering, Rajiv Gandhi College of Engineering Research and Technology Chandrapur.

the constant heat flux provided on the vertical tube temperature of water inside the tube increases, density of water decrease and it becomes lighter and flows in the upward direction at the same time cold liquid replaces these hot water. Due to that there is creating setup of natural convection flow in the upward direction. From this after steady state we have certain mass flow rate, velocity and temperature at outlet condition. But calculating numerically these outlet conditions is very difficult. But this difficulty is solved CFD analysis. Many literatures on this topic are related with analytical and experimental results. CFD results are mainly concerned with compressible fluids. But this paper is related with incompressible fluids and CFD results are compared with experimental results. For the CFD analysis we are using the two software FLUENT and GAMBIT, where GAMBIT is a preprocessor and FLUENT is for postprocessor. In this paper the effect tube length, tube diameter and heat flux on the heat transfer and fluid flow is studied. Computational fluid dynamics techniques are can be used to perform the analysis and identify the relative performance of natural convection concept. This study would provide us the steps followed in natural convection problems in vertical pipes. This CFD analysis can be used to perform the analysis of natural convection flow characteristics like velocity, temperature and heat transfer coefficients.

EXPERIMENTAL SET UP

Experimental set up shows in a schematic diagram (Figure 1). It consists of long copper tube called as test section having uniform cross section and uniform thickness wounded with heating coil to provide constant heat flux boundary condition through the total outer surface area of the pipe. The pre calibrated Wattmeter (accuracy +1%) connected at input power supply can be used to measure the power supply. The copper constant calibrated thermocouples (accuracy $\pm$ 0.20 ^{0}C) were used to measure the temperature at various points. The flow rates were measured with

Figure 1: Schematic Diagram of Experimental Set Up



measuring jar (accuracy +1%) and stop watch. The bottom of vertical tube is considered as a inlet connected with a water tank where constant head is provided. The arrangement is so provided that constant head of water is supplied to inlet of pipe. The topmost end of a pipe considered as outlet is maintained at the same level as that of the level of the water tank so that there is no any flow will take under zero heat flux condition. This whole test section arrangement along with thermocouples is wounded by asbestos rope as insulator so that the heat with the surrounding can be minimized. The wooden stand is provided to hold the test section in exactly vertical manner. When 230V ac supply is switch on the copper tube get heated due to heating of a coil and then flow is set up due to buoyancy effect only. The constant head tank permits measurement of flow induced due to buoyancy alone. The flow is measured with help of measuring jar and temperature is recorded by digital temperature indicator.

EXPERIMENTAL PROCEDURE

Initially the level of the supply tank adjusted in a way that the water level at outlet section of test pipe exactly at the same level as that of level of water in supply tank as shown in figure so that there is no flow of water through the test section even if there is overflow in supply tank. Switch on the heating coil. Adjust the wattmeter so that constant heat flux is supplied to the test section. Due to that temperature of the water in the pipe gradually increases and starts the flow of water in the up word direction because of buoyancy driven force. The hot water replaced by cold water overflowing supply tank ensures that the flow set up in the test section is only due to buoyancy effect. Record the temperature at every five minutes. Once the steady state is reached, i.e., there is no further increase in temperature, measure the flow rate at outlet section and record all the temperatures. The energy balanced is checked between the heat gained by flowing water and heat supplied minus heat loss. The various parameters and their specifications are shown in Table 1.

Table 1: Parameters and Specification During Experimentation

S. No.	Parameters	Specification
1.	Tube Length	1 m
2.	Tube Diameter	16 mm
3.	Inner insulation Thickness	8 mm ceramics Bolls
4.	Outer Insulation Thickness	8 mm Asbestos Rope
5.	Location of Thermocouples	1. At inlet and outlet of the sections 2. At different location on the surface of the tube. 3. On the inner and outer location of the insulation 4. For measuring ambient and heater temperature
6.	Types of thermocouples used	T- Type (copper and constant)
7.	Watt meter readings	20, 40 and 60 watts

COMPUTATIONAL FLUID DYNAMICS (CFD) ANALYSIS

CFD is considered a powerful and an almost essential tool for the design and development and optimization for the many engineering applications. CFD becoming a critical tool to solving the many complicated fluid flow problems. It helps to find out various fluid flow characteristics like temperature, pressure, velocity and other species concentration throughout a solution domain, allowing the design to be optimized prior to the prototype phase. For the CFD analysis there are many software were developed like CFX, ANSYS, and Fluent etc. In our research project we have used Fluent and Gambit software. Gambit is single integrated Pre-processor for CFD analysis which used for creating and meshing geometry of complicated structures. It constructs the geometry and import using STEP, Para solid, and IGES import. Fluent can read this geometry and mesh after that it analyzes the model. The Fluent is a CFD solver has undergone extensive development to extend its robustness and accuracy for wide range of flow regimes. Fluent is very leading engineering CFD software provides the 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 ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/ hexahedral/ pyramid/ wedge, and mixed (hybrid) meshes. Our research includes constructing the geometry and meshing the geometry in Gambit followed by reading the geometry and mesh in Fluent. Once the grid has been read into Fluent, all refining operations are performed within the solver. These includes the setting the model for natural convection, defining the fluid properties, setting the boundary conditions, executing the solution, refining the grid viewing and post processing the result.

In our research we have solved total forty cases by varying the different diameters, different lengths and different heat flux but keeping constant inlet temperatures. Table 2 shows various parameters and specification for which cases are solved.

RESULTS AND DISCUSSION

By solving the various cases the data has been generated for different values of tube diameters, tube length and heat flux. The correlation and flow characteristics described below. The comparative results have been shown in following Figures 2, 3, 4 and 5.

Table 2: Shows Various Parameters and Specification Considered for CFD Analysis

S. No.	Parameters	Specification
1.	Diameter of pipe	12, 16, 20, 24 mm
2.	Length of pipe	0.5,0.75,1,1.25,1.5 M
3.	Heat flux	1061 W/m^2 and 2122 W/m^2
4.	Inlet temperature	20^{0}C
5.	Solver	Pressure based

Table 3: Shows Comparision Between CFD Results and Experimental Results

Parameters	Heat Flux (W/m2)	CFD Results	Experimental Results	% Error
Outlet Velocity m/s	796	0.00322	0.00315	2.17
	1194	0.00396	0.0039	1.5151
	1592	0.00434	0.0043	0.921
Outlet Temperatures ^{0}C	796	46.06	41	8.81
	1194	49.939	47	5.885
	1592	54.54	55	0.8434
Mass Flow Rate kg/s	796	0.000644	0.000638	0.931
	1194	0.000792	0.00079	0.2525
	1592	0.000865	0.00087	0.578

Figure 2: Comparative Results Between L/D Vs. Outlet Velocity and Temperature for Heat Flux 1061 W/m²

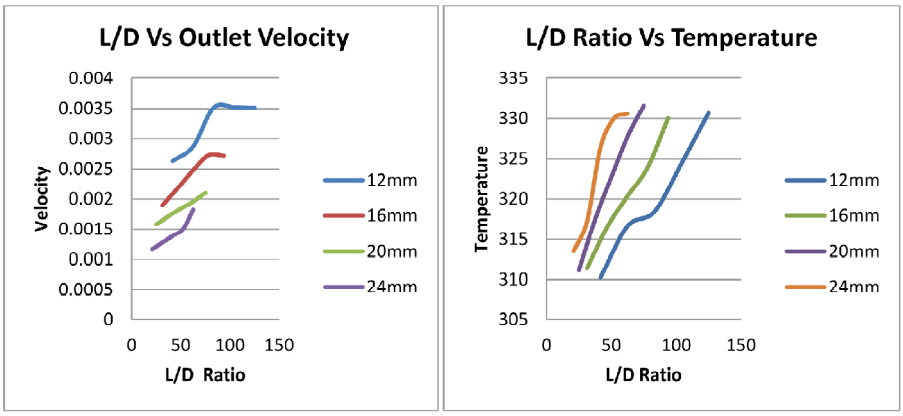


Figure 3: Comparative Results Between L/D Vs. Outlet Velocity and Temperature for Heat Flux 2122 W/m²

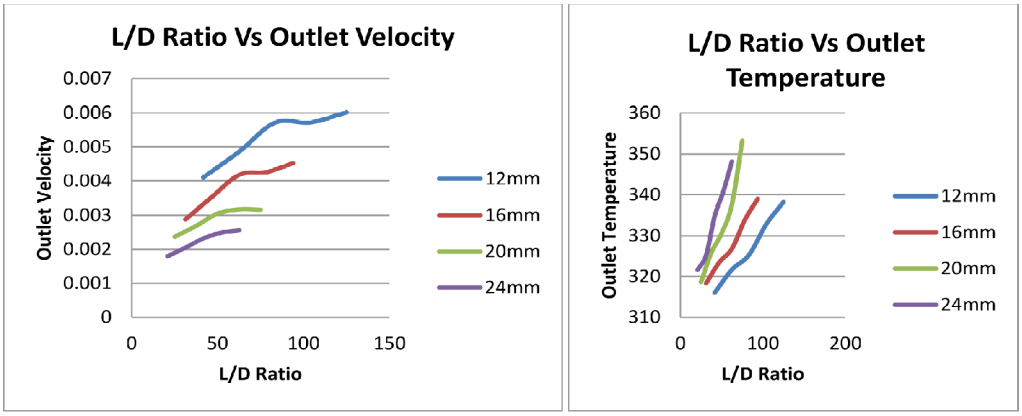


Figure 4: Comparative Results Between Length vs. Outlet Velocity and Temperature for Heat Flux 1061 W/m²

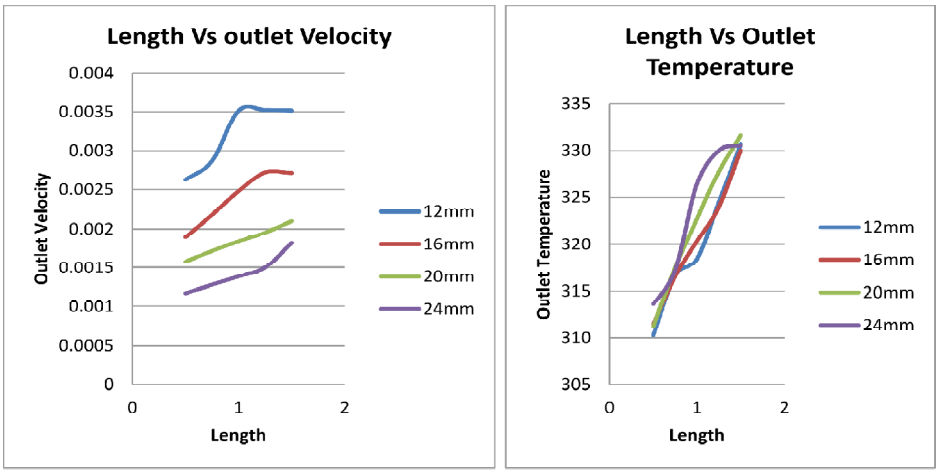


Figure 5: Comparative Results Between Length vs. Outlet Velocity and Temperature for Heat Flux 2122 W/m²

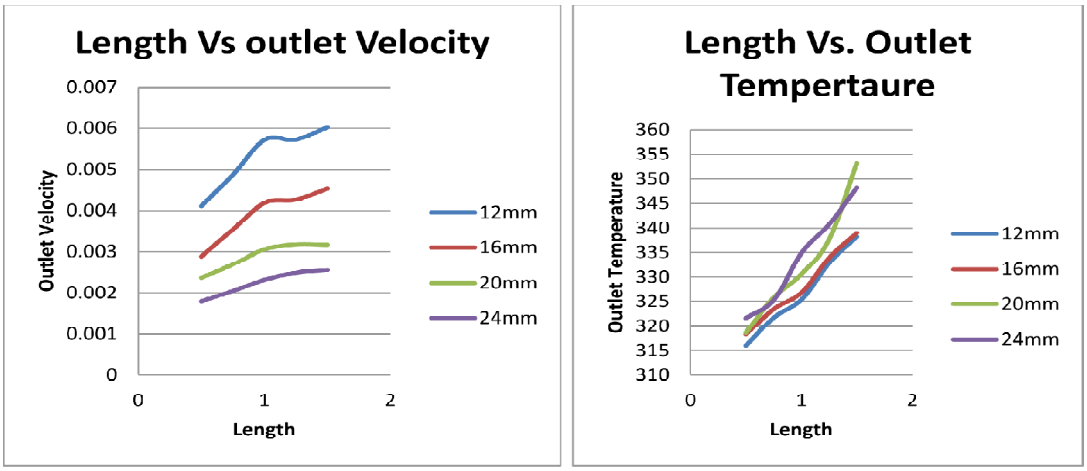
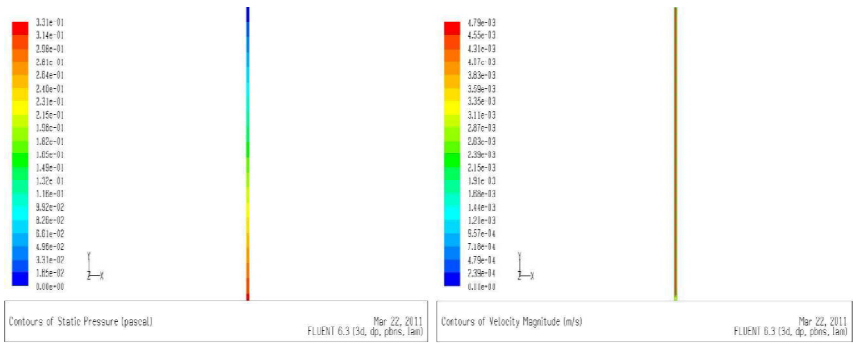
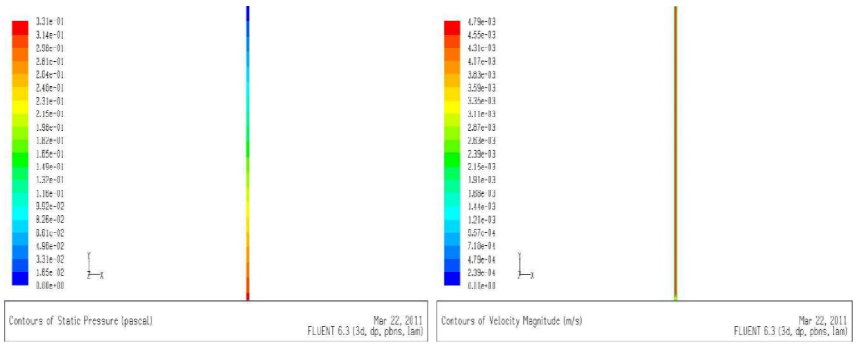


Figure 6: Shows Contours of Pressure and Contours of Velocity



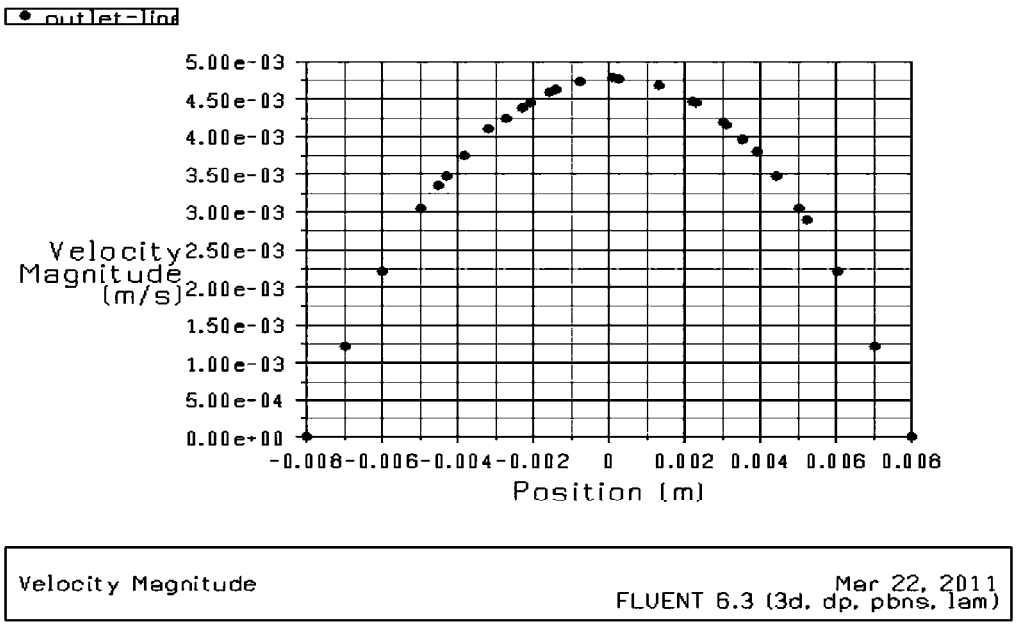
a) Contours of Static Pressure b) Contours of velocity

Figure 7: Shows Contours of Temperature and Contours of Velocity



a) Contours of Static Pressure b) Contours of velocity

Figure 8: Shows Plot of Velocity Vs. Position Along Diameter at Outlet Positon



Finally we have compare the CFD results with Experimental results. These are shown in Table 3.

CONCLUSION

From this work it is observed that

1. From Table 3 it is observed that CFD results and Experimental results both very similar to each other and we found very much less error percentage of error.
2. From above we found that CFD is better tool to solve the fluid flow and heat transfer problem.
3. From the Figures 2, 3, 4 and 5 it is found that as the diameter increases outlet velocity decreases but outlet temperature increases.
4. From the Figures 2, 3, 4 and 5 it is found that as length is increases then both outlet

velocity and outlet temperatures are increases.

5. From the Figure 6 we can see that contours of pressure and contours of velocity. Static pressure is decreased from bottom to top. Maximum velocity found at centre of the pipe.
6. From the figure 7 we can see the contours of temperature and density. In this figure we found that temperature along the surface is maximum and it goes on decreasing towards the centre. In density contours we found that density goes on decreasing from bottom to top.
7. From the Figure 8 we found outlet velocity profile along diameter is parabolic because of fully developed flow at outlet boundry condition.
8. As there is increase in the heat flux we observed that both outlet velocity and outlet temperature going to be increased.
9. From the contours of velocity, temperature, pressure we observed that these characteristics are gradually increases from bottom to top outlet.

FUTURE SCOPE

1. Above observations can be applicable for many engineering applications like solar water heater, HVAC, water tube boilers, etc.
2. Above CFD analysis can be useful for the CFD analysis of natural convection flow for vertical and inclined pipe by varying different material of pipe, type of liquid, thickness of pipe, etc.

REFERENCES

1. Bhargava R and Agrawal R S (1979), "Fully Developed Free Convection Flow In A Circular Pipe", *Indian journal pure appl. Math,* Vol. 10, No. 3, pp. 357-365.
2. Brindley J (1963), "An Approximation Techniques for Natural Convection in a Boundary Layer", *International Journal of Heat and Mass Transfer,* Vol. 6 pp. 1035-1048.
3. *Fluent User's Guide* by Fluent Inc.
4. Heat and Mass Transfer, by D S Kumar.
5. Heat and Mass Transfer, by Holman Mc Graw Hill Publication
6. Hussain A. Mohmmad (2007), "Heat Transfer by natural convection from uniformly Heated vertical circular pipe with different entry restriction configuration", *Energy conservation and management, Vol.* 48, pp. 2244-2253.
7. Hyo Min Jeong (2009), "Natural convection heat transfer estimation from a longitudinally finned vertical pipe using CFD", *Journal of mechanical science and Technology,* Vol. 23, pp. 1517-1527.
8. Prayagi S (2011), "Parametric Studies on Buoyancy Induced Flow through Circular Pipes in Solar water heating system", *International Journal of Engineering Science and Technology (IJEST).*
9. Wright J L (2006), "Flow Visualization of Natural Convection in a Tall Air Filled Vertical Cavity", *International Journal of Heat and Mass Transfer*, Vol. 49, pp. 889-904.