THOMSON REUTERS ENDNOTE

[Azam*, 5(3): March, 2016] ISSN: 2277-9655

(I2OR), Publication Impact Factor: 3.785

IJESRT

INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

EXPERIMENTAL VALIDATION AND CFD PREDICTION OF NATURAL CONVECTION IN A VERTICAL PIPE USING FLUENT 14.5 SOLVER

Farooque Azam*, DR. Jitendra Kumar, Arham Javed

*PG (Post Graduate) Student, Department of Mechanical Engineering, Bansal Institute of Research and Technology, Bhopal (M.P.), India

Professor and Head of Department Mechanical Engineering, Bansal Institute of Research and Technology,Bhopal (M.P), India

Assistent Professor, Department of Mechanical Engineering, Bansal Institute of Research and Technology, Bhopal (M.P.) India

DOI: 10.5281/zenodo.47632

ABSTRACT

In this study, an experimental and CFD prediction of natural convection heat transfer in a vertical cylinder (pipe) is done by FLUENT 14.5 solver. Rosseland radiation model available in ANASYS CFD FLUENT 14.5 software will be investigate to find their usefulness for different parameters and cases as well as their limitation. Simulations will be performed with systematic parameterization to investigate the effect of changing models and parameters. Governing equations are solved using a finite volume method. The set up consist of brass cylinder with air as a working fluid. CFD results and experimental results are compared which validate the software result. The present analytical studies help to give best suitable material for free convection in a vertical cylinder.

KEYWORDS: Natural Convection, Vertical Cylinder, Heat Transfer, CFD, FLUENT 14.5

INTRODUCTION

A considerable portion of heat loss from typical residence occurs through the windows. The problem is finding an insulating material that is transparent. An examination of thermal conductivities of insulating materials reveals that is a better insulator than most common insulating material. Besides, it is transparent. Therefore, it makes sense to insulate the window with the layer of air. Of course we need to use another sheet of glass to trap the air. The result is an enclosure. Other examples of enclosures include wall cavities, solar collectors and cryogenic chambers involving concentric cylinders or spheres. Enclosures are frequently encountered in practice, and the heat transfer through them is of practical interest. The heat transfer in enclosed spaces is complicated by the fact that fluid in enclosure, in general, does not remain stationary, the fluid adjacent to the hotter surface rises and the fluid adjacent to the cooler fluid one falls, setting a rotationary motion within the enclosure that enhances the heat transfer through the enclosure. Most fluids, however, have temperature dependent properties, and under circumstances where large temperature gradients exist across the fluid medium, fluid properties often vary significantly. Under many conditions, ignoring such variations may cause inaccuracies in estimating heat transfer rates. The thermophysical properties that appears in the governing equations includes thermodynamic and transport properties that appears in the governing equations. Thermodynamic properties define the equilibrium state of the system. Temperature, density and specific heat are such properties. The transport properties include the diffusion rate coefficient such as the thermal conductivity and viscosity. Larrode et al. [1] and Yu and Ameel [2] considered the temperature jump condition and found that the effect of fluid- wall interaction is also important. Natural convection of an enclosed fluid has received considerable attention in recent year due to its wide application in engineering problems. Chen and Weng [3] recently present analytical investigations on fully developed natural convection in open-ended vertical parallel plate microchannels, by taking the velocity slip and temperature jump effects into account. And important conclusion from their study in that for fully developed free convection flows with symmetrically heated walls, the Nusselt number turn out to be zero. Arkilic et al. [4] and Liu et al. [5] found that the theoretical results for some microflows would fit the experimental data as the slip-flow condition induced by rarefaction effect is considered. A classification of different flow regimes with a rarefied gas considered near the continuum region in the range 10-2 < kn > 10-1, the so-called slip flow. Using the Navier- stokes equation they gave their results in microflows. I. P. Jones [6] worked on natural convection problem, involving buoyancy driven flow in a cavity, was first suggested as a suitable validation test case for CFD codes the boundary condition for problem involved two vertical walls which differ in temperature leading to a thermal gradient across the solution domain. This thermal gradient leads to the buoyancy forces varying between the walls giving rise to flows. Adiabatic conditions are assumed on remain boundary. Biswal et al. [7] have recently executed a comprehensive computational study on free convection heat transfer in the entrance region followed, by that in the fully developed region, in long vertical microchannels for different value of Knudsen number and Rayleigh number. In their study, special implications of accommodating the effects of the developing region in the heat transfer analysis were discussed in details and some important conclusions based on the same were pinpointed. Cuce et al. [8] present a detail experimental study of the effects of passive cooling on performance parameters of photovoltaic cells. In the study, an aluminium heat sink was used in order to dissipate waste heat from photovoltaic cell. The dimensions of heat sink were determined considering the results of a steady state heat transfer analysis. The experiments were carried out for different ambient temperatures and various illumination intensities up to one sun under a solar simulator.

EXPERIMENTAL SET UP

The apparatus consist of a vertical brass cylinder fitted in an enclosure. The enclosure is open at top and bottom to ensure that undistributed natural convection condition. One side of enclosure is made up of Perspex for visual display. An electrical heating element is kept in brass vertical cylinder which heats the cylinder surface. The heat loss by cylinder to ambient air is by natural convection. The temperature of vertical cylinder is measured by thermocouples which are fixed on the cylinder by drilling holes along the cylinder wall. The heat input is measured by an ammeter and voltmeter and is varied with the help of a dimmerstat. The cylinder surface is polished to minimize the heat loss. Adjust heat input by using dimmerstat to desired value (50). Wait for sometime 45-60 minutes to achieve steady state condition. Record all temperatures voltmeter readings, ammeter readings. Make dimmerstat to zero position and then put "OFF" main switch. Repeat the procedure for different heat inputs.

Figure: experimental setup of verticval cylinder

COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics or CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena such as chemical reactions by means of computer based simulation. The technique is very powerful and spans a wide range of industrial and non-industrial application areas. CFD codes are structured around the numerical algorithms that can tackle fluid problems. In order to provide easy access to their solving power all commercial CFD package include sophisticated user interfaces to input problem parameters and to examine the results. All CFD codes contain main three elements: (1) pre processor (2) solver (3) post-processor. We briefly examine the function of each of these elements within the context of CFD code. In solving fluid flow problems we need to be aware that the underlying physics is complex and the results generated by a CFD code are at best as good as the physics (and chemistry) embedded in it and at worst as good as its operator. Elaborating on the latter issue first, the user of a code must have skills in a number of areas. Prior to setting up and running a CFD simulation there is a stage of identification and formulation of the flow problem in terms of the physical and chemical phenomena that need to be considered.

FLUENT

FLUENT is very powerful tool to investigate the multi-dimensional flow and heat transfer phenomena in engineering applications. FLUENT numerically solves the governing equations for fluid flows using finite volume methods. Mass transport equation and three-dimensional momentum transport equations are the fundamental governing equations solved in the CFD code. Energy equation is included for problems involving heat transfer or compressible flow. Turbulence models require the transport equations for the turbulence flow variables in addition to the Navier-Stokes equations. The number of extra governing equations for the other phases depends on which multiphase model is employed.

CFD PREDICTION

CFD validation and prediction is done using same boundary conditions and dimensions which was kept in experimental setup but material of vertical cylinder is used in CFD FLUENT 14.5 is aluminium in place of brass. Geometry is done in CFD fluent 14.5 Solver.

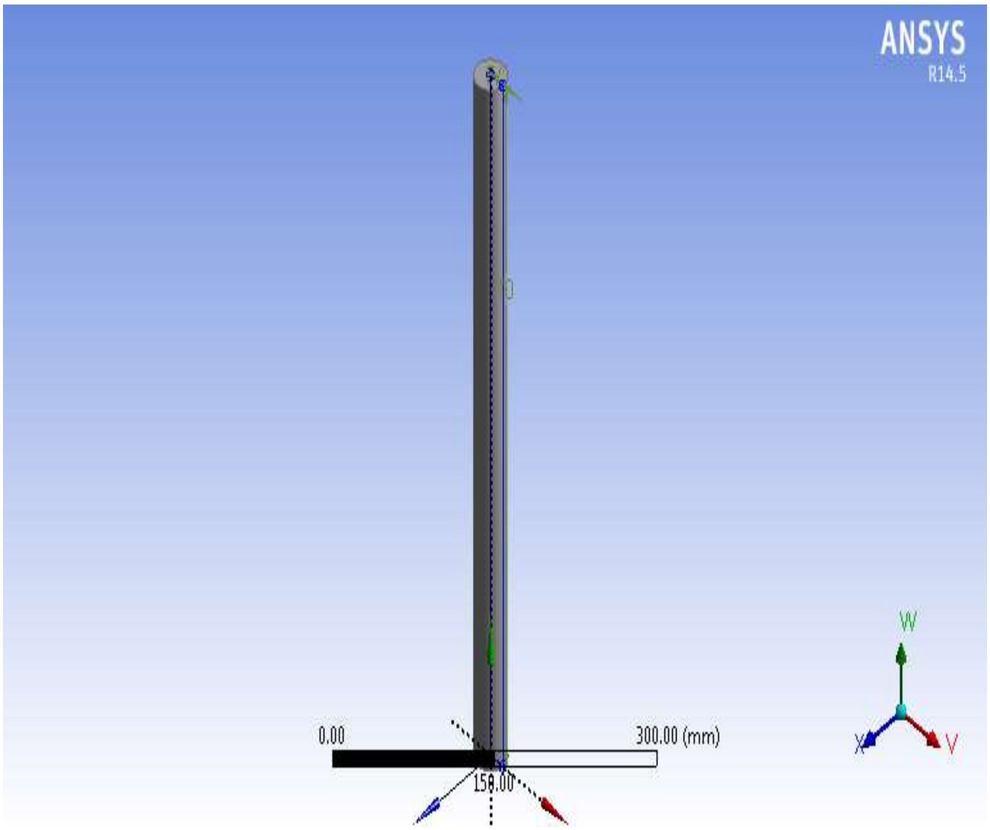


Figure: modeling of vertical cylinder in pre processor

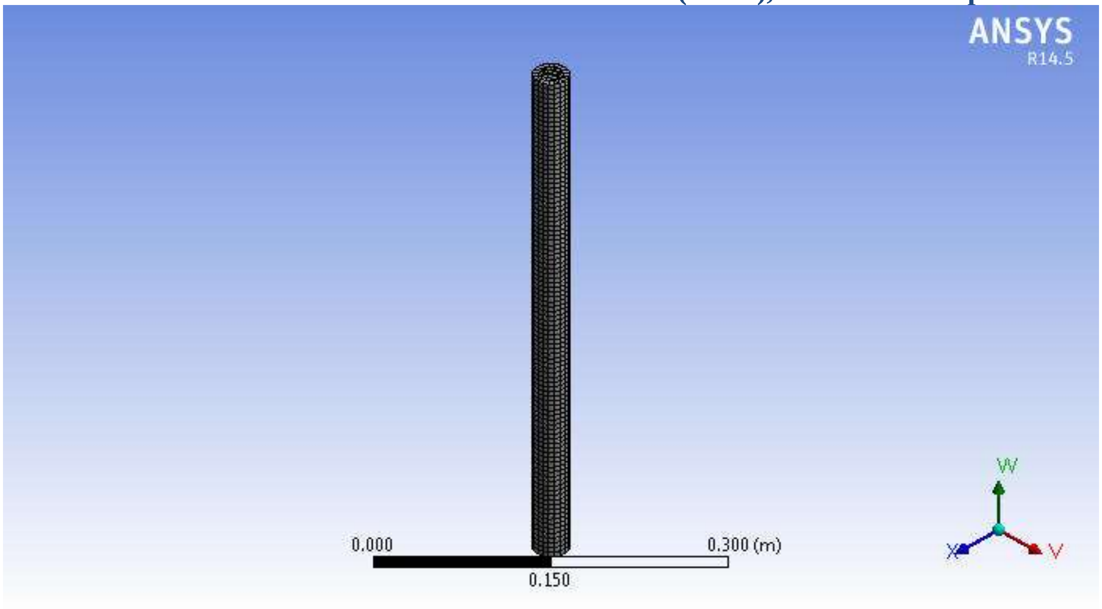


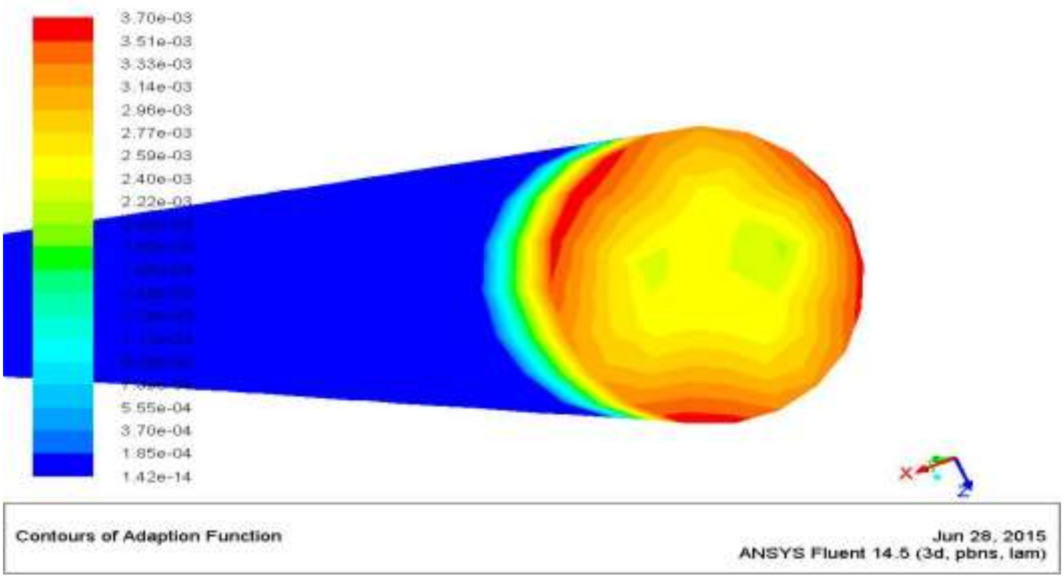
Figure: meshing structure of vertical cylinder

MESHING DETAILS

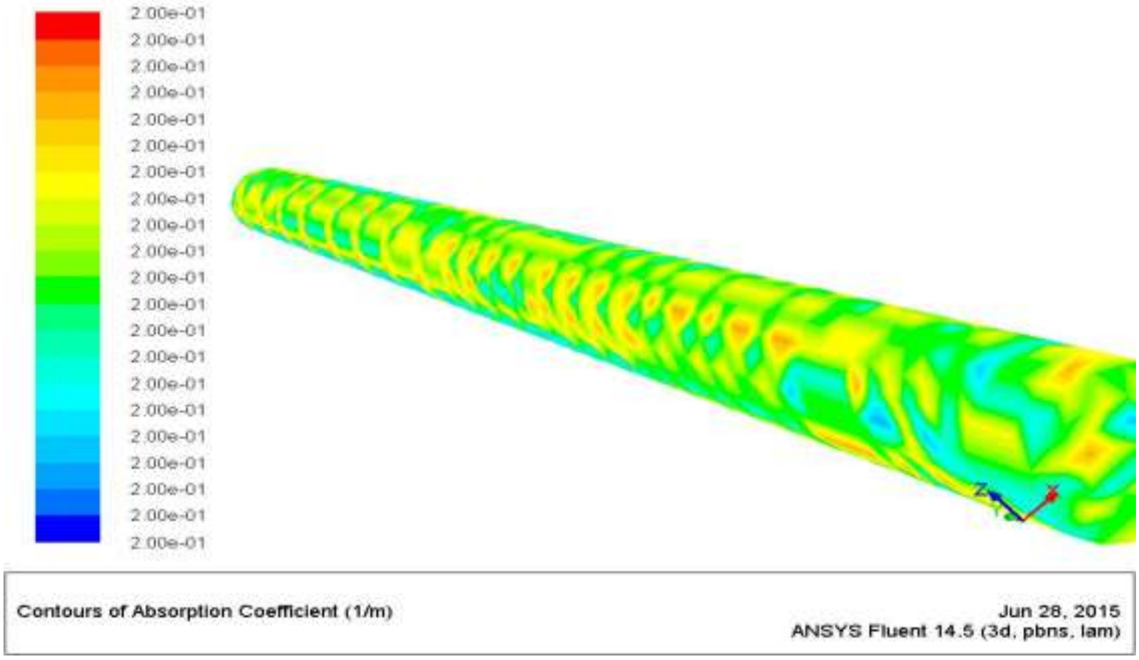
Use advanced size function	on: curvature
Relevance center	Fine
Initial size seed	active assembly
Smoothing	medium
Transition	slow
Span angle centre	fine
Curvature normal angle	18.0°
Min. size	$703291e^{-005m}$
Max. face size	$7.3291e^{-003m}$
Max. size	$1.4658e^{-002m}$
Growth rate	1.20
Nodes	5959
Element	4800

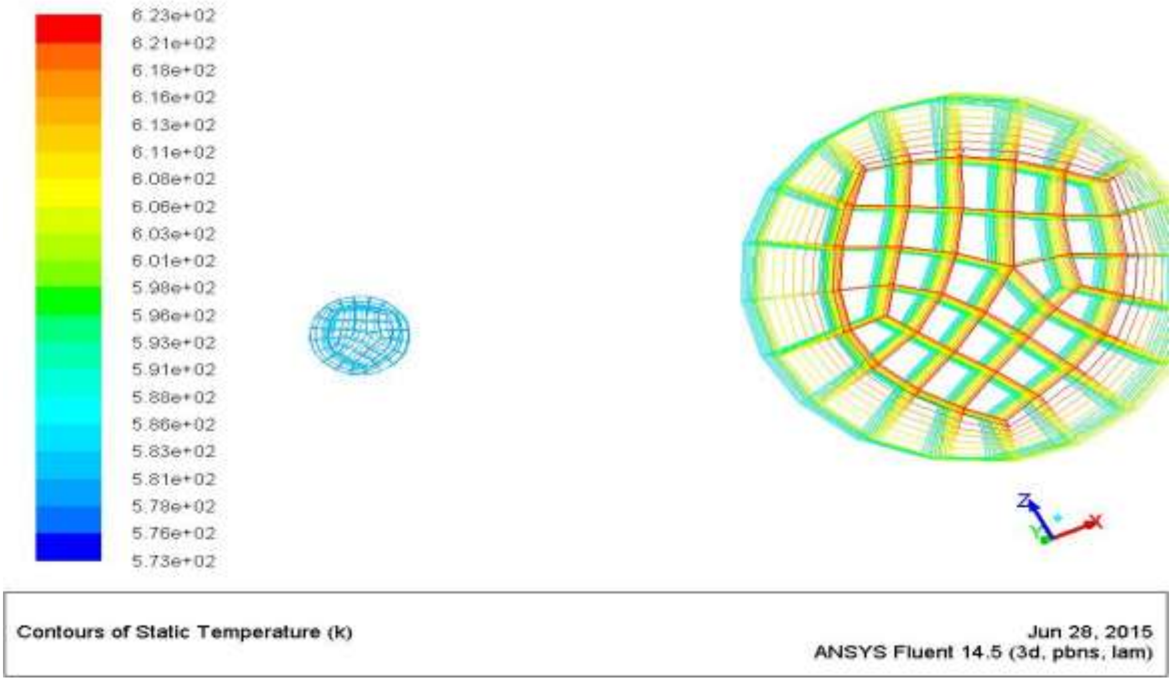
RESULTS & DISCUSSION

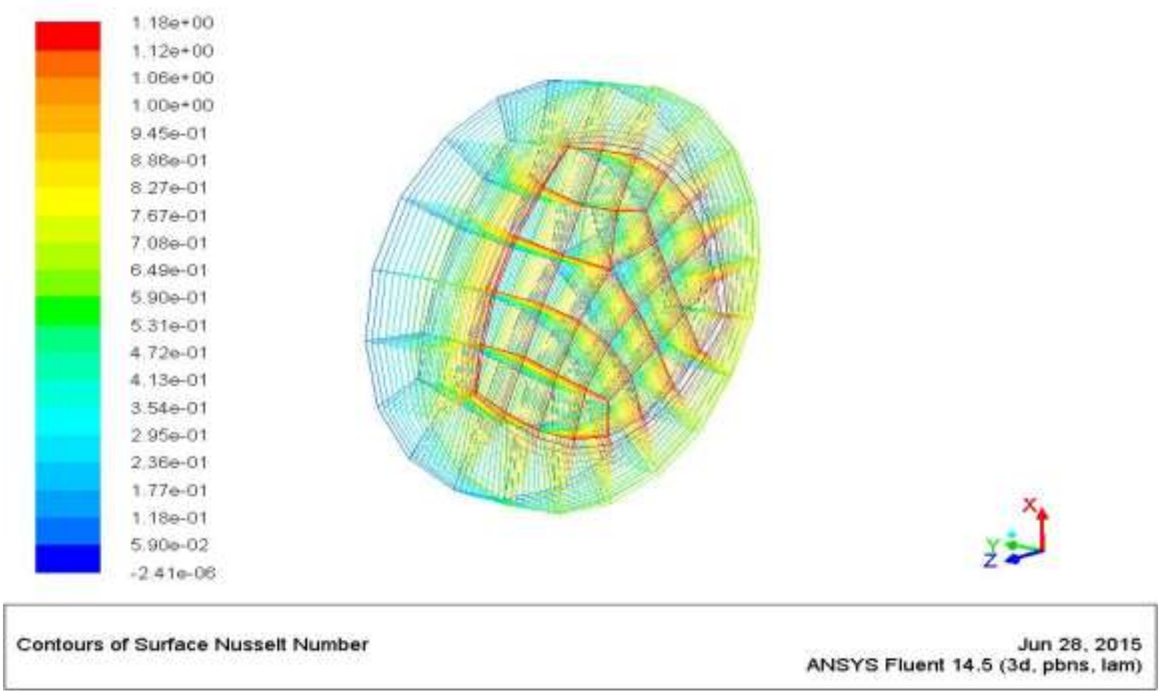
From above analysis various contours and vectors plots are obtained.

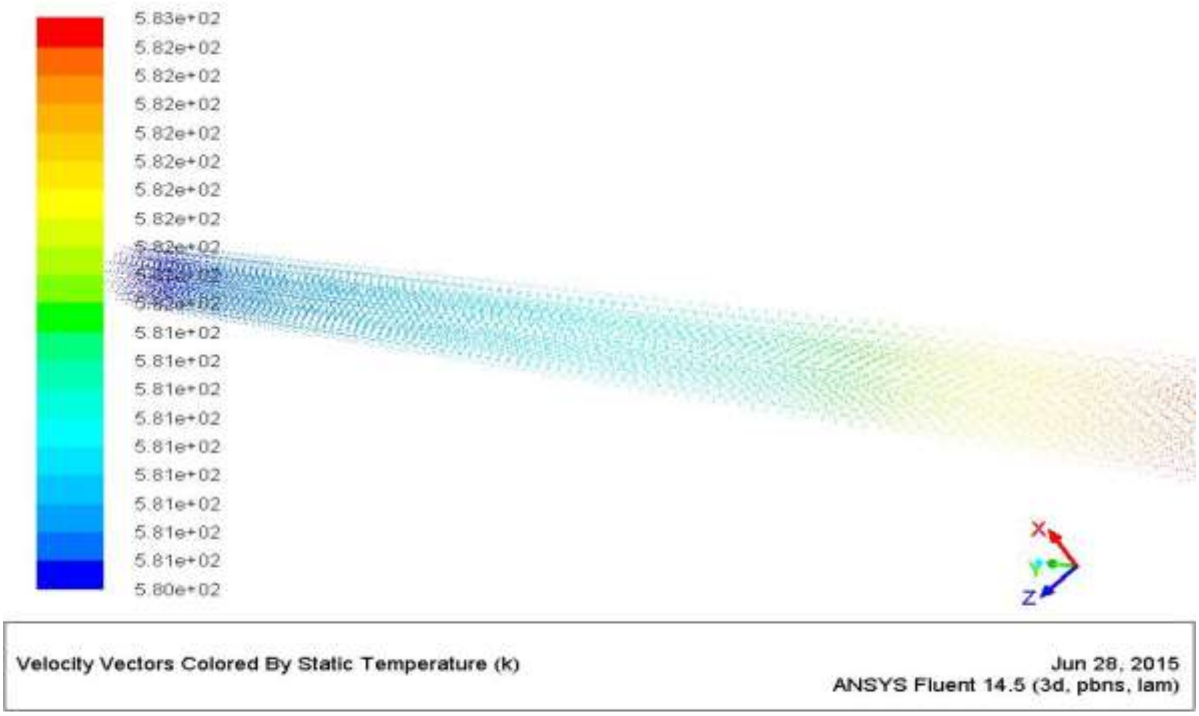


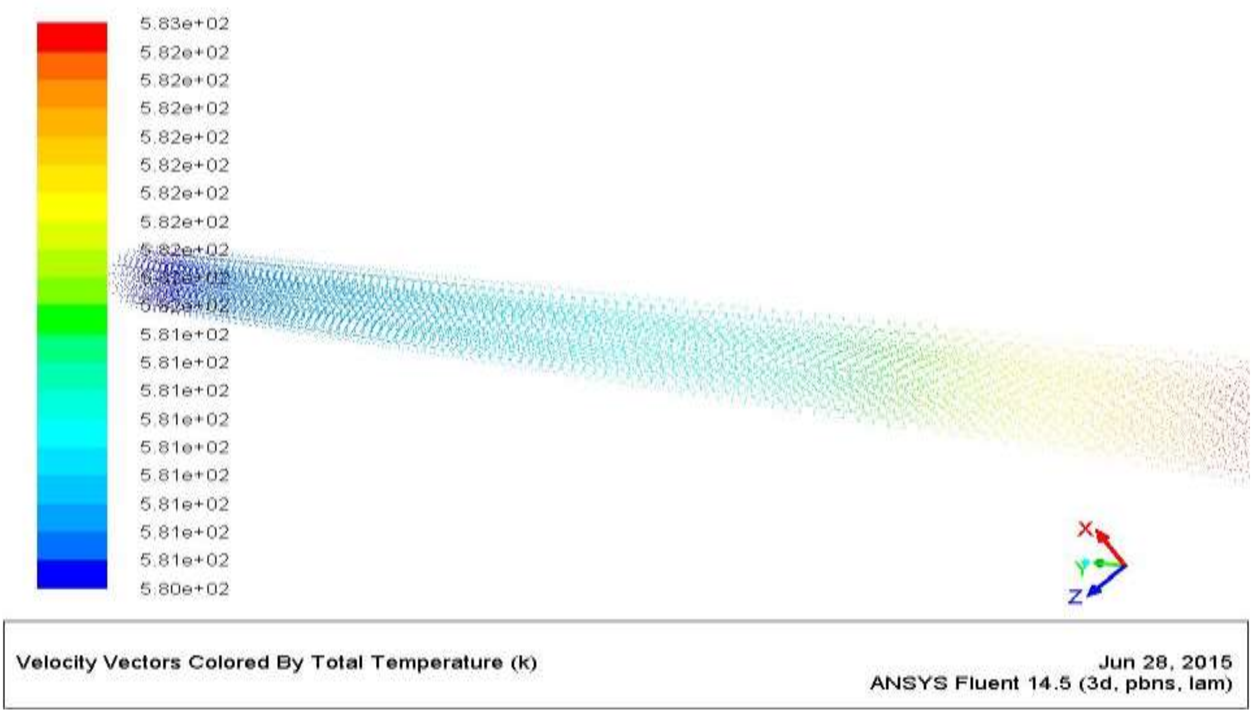
ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785

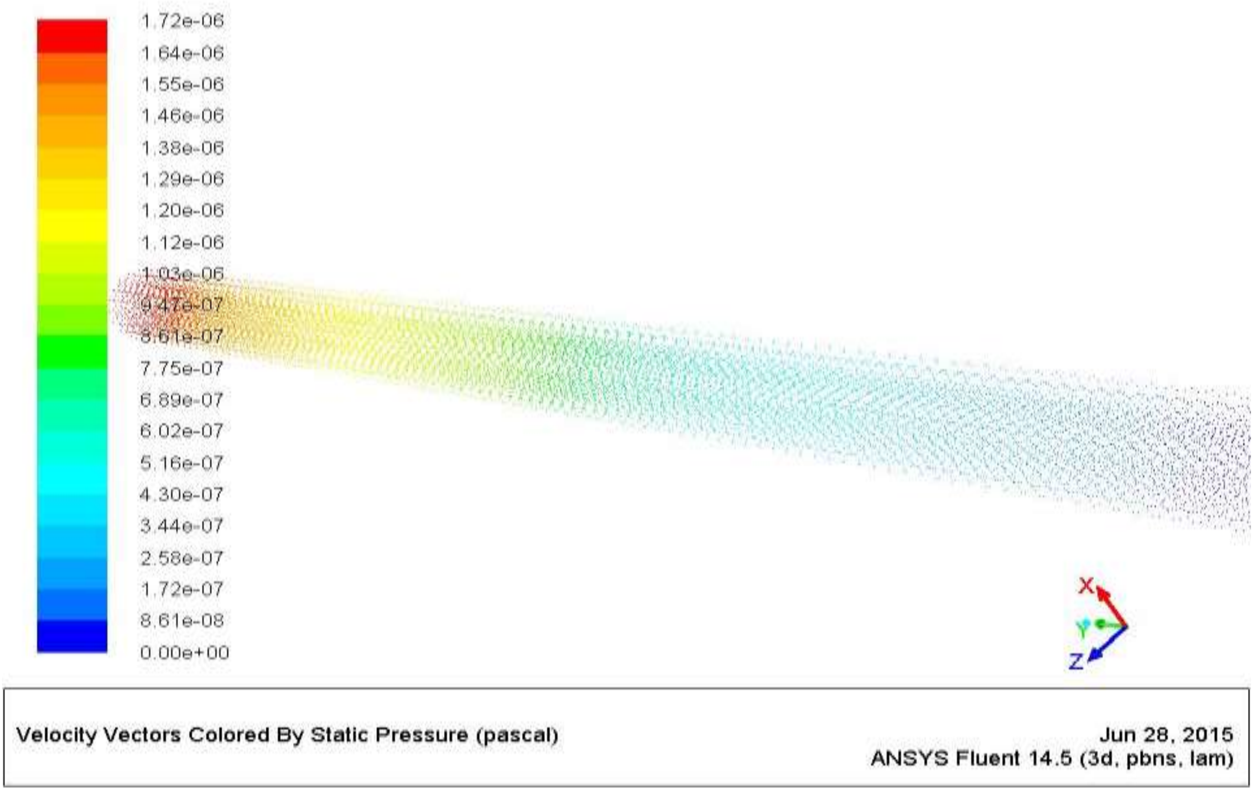




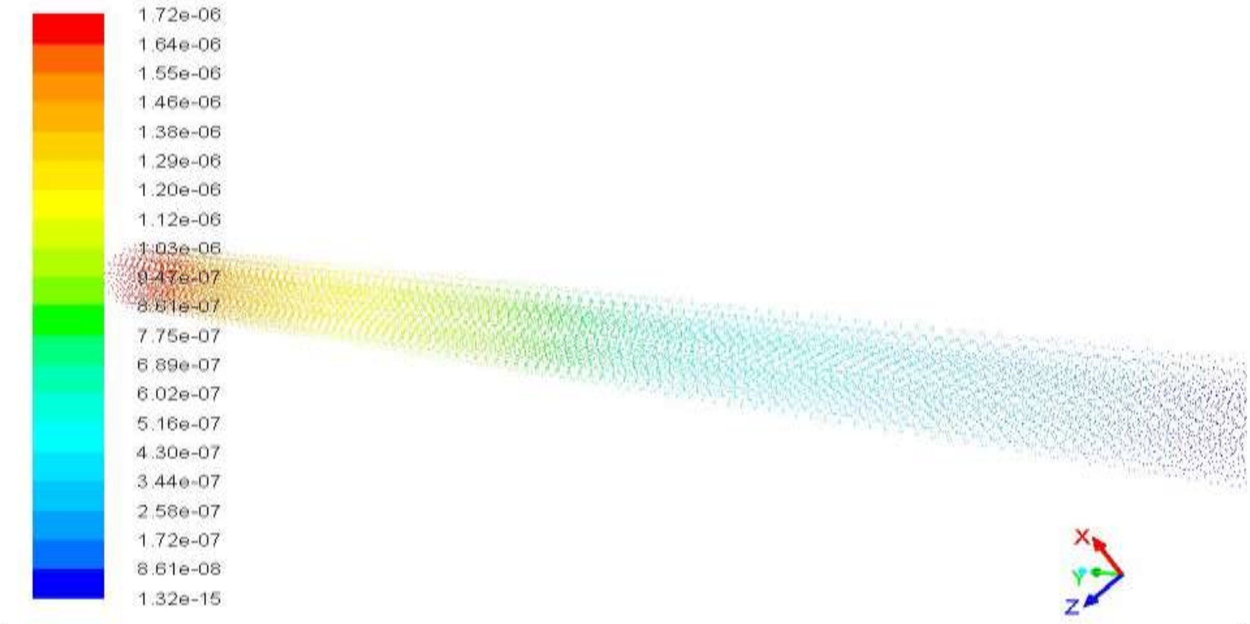

1.18e+00
1.12e+00
1.06e+00
1.00e+00
9.45e-01
8.86e-01
8.27e-01
7.67e-01
7.08e-01
6.49e-01
5.90e-01
5.31e-01
4.72e-01
4.13e-01
3.54e-01
2.95e-01
2.36e-01
1.77e-01
1.18e-01
5.90e-02
-2.41e-06
Contours of Surface Nusselt Number
Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)


5.83e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.80e+02
Velocity Vectors Colored By Static Temperature (k)
Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)


5.83e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.82e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.81e+02
5.80e+02
X
Y
Z
Velocity Vectors Colored By Total Temperature (k)
Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)


1.72e-06
1.64e-06
1.55e-06
1.46e-06
1.38e-06
1.29e-06
1.20e-06
1.12e-06
1.03e-06
9.47e-07
8.61e-07
7.75e-07
6.89e-07
6.02e-07
5.16e-07
4.30e-07
3.44e-07
2.58e-07
1.72e-07
8.61e-08
0.00e+00
X
Y
Z
Velocity Vectors Colored By Static Pressure (pascal)
Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)

ISSN: 2277-9655
(I2OR), Publication Impact Factor: 3.785



Velocity Vectors Colored By Total Pressure (pascal) Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)

CONCLUSIONS

It is concluded that from above figure, procedures and results, for natural convection in a vertical cylinder which is inclosed in a box, aluminium is the best suitable material for cylinder as compared to brass. We done experiment as kept brass material cylinder and done CFD prediction using aluminium as a cylinder material. From the contours of velocity, temperature, pressure we observed that these characteristic are gradually increases from bottom to top outlet. In many application of natural convection now we can use aluminium as a pipe material on the basis of above performed worked and its beneficial results.

FUTURE SCOPE

CFD analysis can be useful for vertical pipe by other different material, pipe dimensions and different atmospheric and boundary conditions.

REFERENCES

1. Larrode, F. E., Housiadas, C., and Drossinos, Y., 2000, Slip-Flow Heat Transfer in Circular Tubes, Int. J. Heat Mass Transfer, **43**, pp. 2669-2680. [Inspec] [ISI] first citation in article.
2. Yu. S. and Ameel, T. A., 2001, "Slip-Flow Heat Transfer in Rectangular Microchannels," Int. J. Heat Mass Transfer, 44, pp.4225- 4234. [Inspec] [ISI] first citation in article.
3. C. K. Chen, H. C. Weng, Natural convection in a vertical microchannel, J. Heat Transfer 127 (2005) 1053-1056.
4. Arkilic, E, B, Breuer, K. S., and Schmidt, M. A., 1944, "Gaseous Flow in Microchannels, Application of Micro- Fabrication to Fluid Mechanics," ASME FED, **197**, pp. 57-66. first citation in article.
5. I. P. Jones, A Comparison Problem for Numerical Methods in Fluid Dynamics, the 'Double Glazing' Problem in Numerical Methods in Thermal Problems, Proceeding of the First International Conference, 338-348 (1979).
6. Liu, J.Q., Tai, Y.C., and Ho, C.M., 1955, MEMS for Pressure Distribution Studies of Gaseous Flow in Microchannels, "*Proceedindings, IEEE Micro Electro Mechanical System,* pp.209-215. first citation in article in article.
7. L. Biswal, S. K. Som, S. Chakraborty, Effects of entrance region transport processes on free convection slip flow in vertical microchannels with isothermally heated walls, Int. J. Heat Mass Transfer 50 (2007) 1248-1254.
8. Cuce E, Bali T, Sekucoglu SA. Effects of passive cooling on performance of silicon photovoltaic cells. Int J Low Carbon Technol 2011; 6:299-308.

9. Hussain A. Mohammad (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.
10. 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.
11. 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).