INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.3 Issue.6,

June 2015.

Pgs: 12-21

EXPERIMENTAL AND CFD PREDICTION OF FREE CONVECTION IN A VERTICAL CYLINDER INCLOSED IN A BOX BY FLUENT 14.5 SOLVER USING ROSSELAND RADIATION MODEL AND BOUSSINESQ DENSITY PARAMETER

Farooque Azam[1], Mahendra Dawande[2] , Abhishek Tripathi[3]

[1]PG (Post Graduate) Student, Mechanical Engineering Department , Bansal Institute Of Research and Technology, Bhopal (M.P.) farooque95@gmail.com, +918982358451

[2] Assistent Professor, Mechanical Engineering Department, Bansal Institute Of Research and Technology, Bhopal(M.P.)

[3]Abhishek Tripathi, abhiyoho@gmail.com, +919630444300

Abstract

In present work the experimental investigation and cfd validation of natural convection in vertical cylinder inclosed in a box is done by FLUENT 14.5 solver. For getting better effect and accurate result Rosseland Radiation Model and Boussinesq density parameter is used. The thermal control in many systems is widely accomplished applying natural convection process due to its low cost, reliability and easy maintenance. Typical applications include the heat exchangers, cooling of electronic equipment and nuclear reactors, solar chimneys and Trombe walls in building industry, etc. Laminar simulations are obtained by solving the governing equations using a Fluent 14.5. It is considered that the temperature variations are not so high and the Boussinesq approximation is applied. It is highly desirable to understand the fluid flow and the heat transfer characteristics of buoyancy-induced micropump and microheat exchanger in microfluidic and thermal systems. In this study different material of pipe (cylinder) is investigated and found that aluminium gives better results then brass material. The experiment was done using brass material and it is validated in cfd and another material i.e. aluminium which replaced brass and then predicted in cfd FLUENT 14.5 solver. Operating conditions and Boundary conditions kept same for brass material and aluminium material. Same Boundary conditions are applied in cfd FLUent, which is used in Experimental investigation.

***Keywords**:* Natural Convection, Rosseland Radiation Model, Boussinesq density parameter, FLUENT 14.5

1. Introduction

Convective heat transfer is a mechanism of heat transfer occurring because of bulk motion (observable movement) of fluids. As convection is dependent on the bulk movement of a fluid it can only occur in liquids, gases and multiphase mixtures. Convective heat transfer is split into two categories: natural (or free) convection and forced (or advective) convection, also known as heat advection. **Natural convection** is a mechanism, or type of heat transport in which the fluid motion is not generated by any external source (like a pump, fan, suction device, etc.) but only by density differences in the fluid occurring due to temperature gradients. In natural convection, fluid surrounding a heat source receives heat, becomes less dense and rises. The surrounding, cooler fluid then moves to replace it. This cooler fluid is then heated and the process continues, forming a convection current; this process transfers heat energy from the bottom of the convection cell to top. The driving force for natural convection is buoyancy, a result of differences in fluid density. Because of this, the presence of a proper acceleration such as arises from resistance to gravity, or an equivalent force (arising from acceleration, centrifugal force or Coriolis force), is essential for natural convection. **Microelectromechanical** systems (MEMS) based devices find their applications in a wide variety of emerging technologies, ranging from the microactuators, microsensors, microreactors to the microchannel heat sinks and the thermo-mechanical data storage systems, to name a few. Design and optimization of many of these microdevices involve the analysis of gas flows through microfluidic conduits, thereby emphasizing the need for reliable and efficient mathematical models to address the issues of coupled flow physics and heat transfer over the reduced length scales. Microfluidic systems typically have characteristic lengths of the order of 1–100 μm. Natural convection flows in a rectangular enclosure subject to a horizontal temperature gradient have been extensively studied by numerical means (e.g. Polezhaev, 1967, Macgregor and Emery, 1969, Rubel and Landis, 1970, Mallinson and de Vahl Davis [2], 1973 and 1977). Only three of these have treated variable fluid properties. Macgregor and Emery (1969) used the Boussinesq approximation and a variable viscosity while Rubel and Landis (1970) used a linearized approach and reported results for moderate Rayleigh numbers. Polezhaev (1967) solved the complete equations, including the continuity equation, for a square cavity and for one value of non-dimensional temperature difference between hot and cold walls. The study of fully developed free convection between parallel plates at constant temperature has been initiated by Ostrach [3]. Sinha [4] studied this problem using as working fluid water at low temperatures where the relation between density and temperature is nonlinear. However the other water properties (viscosity and thermal conductivity) have been considered constants. The first exact solutions for free convection in a vertical parallel plate channel with asymmetric heating for a fluid with constant properties was presented by Aung [5]. Vajravelu and Sastri [6] reconsidered the problem treated by Sinha using a more accurate relation between water density and temperature, ignoring again the variation of other water properties with 4 temperature. Vajravelu [7], in a subsequent paper, treated the same problem using water and air as working fluids and considering all fluid thermophysical properties (ρ, μ, k, cp)_ as linear functions of temperature. However, the results are valid for room temperatures between 10 and 25°C. Chenoweth and Paolucci [8] presented exact solutions for a perfect gas using the Sutherland law for viscosity and thermal conductivity and considering the ambient fluid temperature equal to the reference temperature (mean temperature of the two plates). Chenoweth and Paolucci [9] extended the previous work to cases where the ambient fluid temperature is different from the reference temperature. The presented results in both works are valid for the temperature range between 120 K and 480 K.

2. EXPERIMENTAL SET UP

The apparatus consists of a vertical cylinder fitted in a large enclosure, with top and bottom open to ensure undistributed Natural Convection conditions. The front side of enclosure is provided with an acrylic sheet for visual display. The cylinder is provided with heating element from inside which heats it uniformly and the heat is dissipated from outer surface by natural convection to ambient air. The temperature of the cylinder surface is measured by thermocouples and one more thermocouple records the ambient temperature in the duct.
The heater input can be varied with the help of a dimmer stat and is measured by voltmeter and an ammeter. A separate control panel is provided for housing all instruments.

2.1 SPECIFICATIONS

- Brass pipe diameter = 32mm
- Length of pipe = 500mm
- Thermocouples = 8 Nos.
- Heater coil – Nichrome heater
- Temperature Indicator 0-300^{0}C
- 2Amp. Open type Dimmer stat.
- Digital Voltmeter
- Digital Ammeter

3. COMPUTATIONAL FLUID DYNAMICS (CFD)

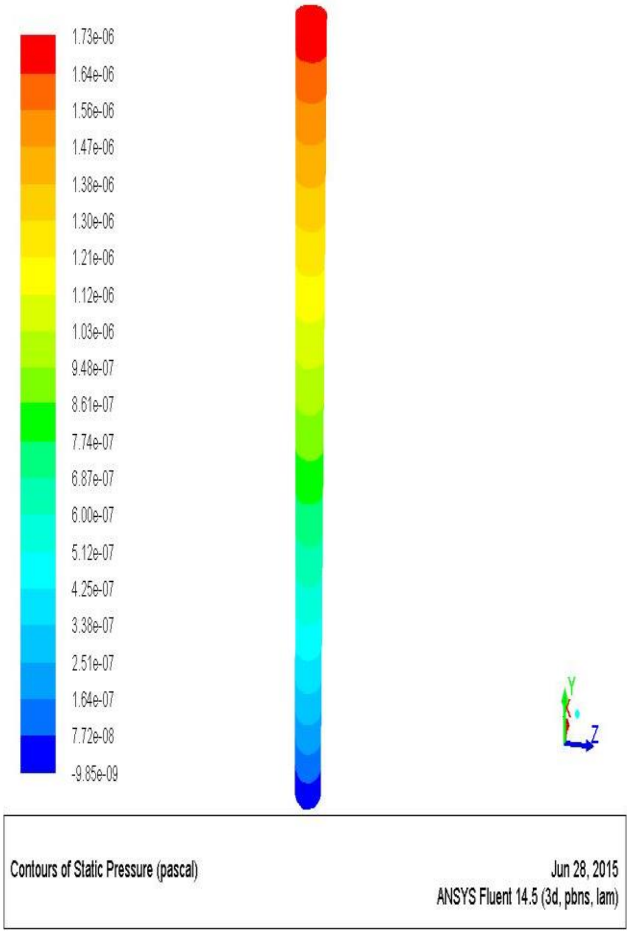
CFD is considered a powerful and an almost essential tool for the design and development and optimization for the many engineering applications. CFD becoming a critical tool to solving the many complicated fluid flow problems. It helps to find out various fluid flow characteristics like temperature, 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. The fundamental basis of any CFD problem are the Navier-Stokes equations, which define any single-phase fluid flow. These equations can be simplified by removing terms describing viscosity to yield the discretised algebraic equations. Rosseland Radiation Model and Boussinesq density parameter is used in this analysis in FLUENT 14.5 solver.

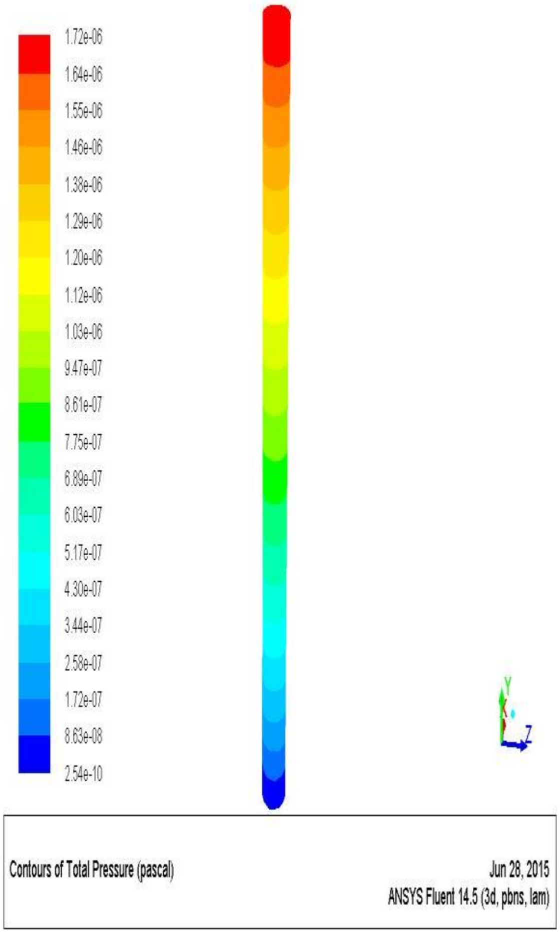
3.1 Fluent

The Fluent solver is based on the centre node FVM discretisation technique and offers both segregated and coupled solution methods. Three Euler-Euler multiphase models are available; the Eulerian model, the mixture model and the VOF model. In addition, one particle tracking model is available. Fluent offers three main approaches to model dispersed phases with a two-fluid formulation. With the default settings it is assumed that the dispersed phase has a constant diameter or a diameter defined by a user-defined function. With this setting, phenomena such as coalescence and breakage are not considered.

4. RESULTS & DISCUSSION

In this section, the results from the simulations and from the boundary conditions are presented.


1.73e-06
1.64e-06
1.56e-06
1.47e-06
1.38e-06
1.30e-06
1.21e-06
1.12e-06
1.03e-06
9.48e-07
8.61e-07
7.74e-07
6.87e-07
6.00e-07
5.12e-07
4.25e-07
3.38e-07
2.51e-07
1.64e-07
7.72e-08
-9.85e-09
Y
X
Z
Contours of Static Pressure (pascal)
Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)

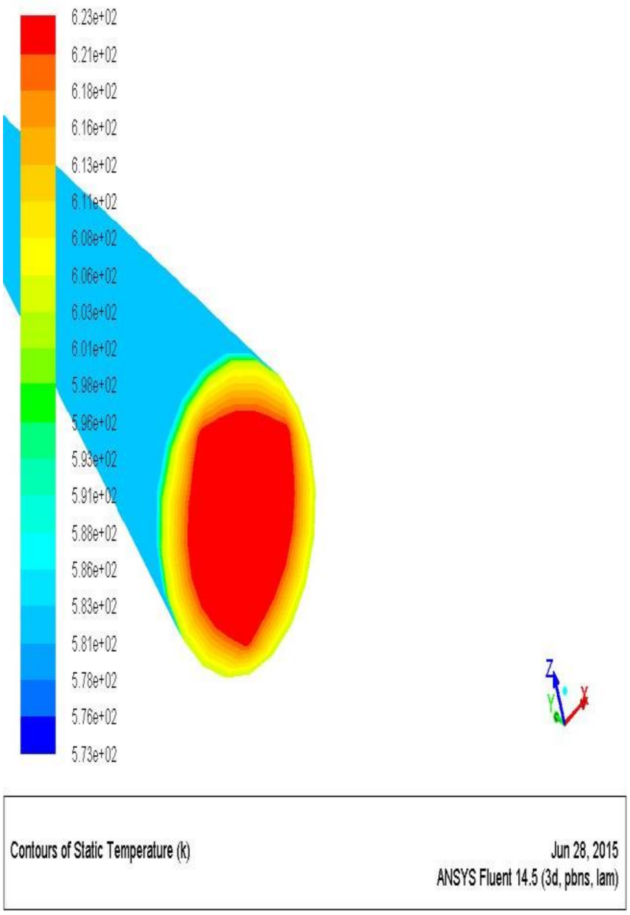

1.72e-06
1.64e-06
1.55e-06
1.46e-06
1.38e-06
1.29e-06
1.20e-06
1.12e-06
1.03e-06
9.47e-07
8.61e-07
7.75e-07
6.89e-07
6.03e-07
5.17e-07
4.30e-07
3.44e-07
2.58e-07
1.72e-07
8.63e-08
2.54e-10
Y
X
Z
Contours of Total Pressure (pascal)
Jun 28, 2015
ANSYS Fluent 14.5 (3d, pbns, lam)

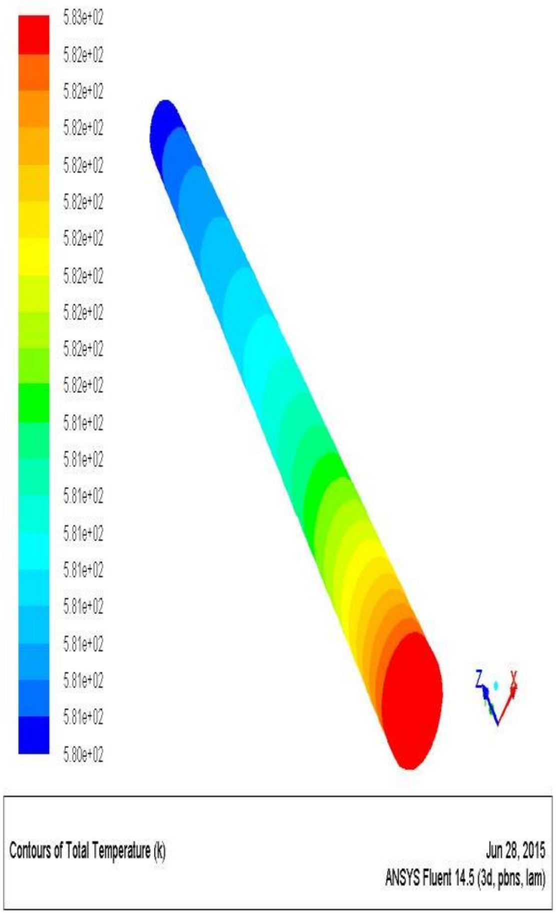
ISSN (ONLINE): 2321-3051

Vol.3 Issue.6,

June 2015.

Pgs: 12-21





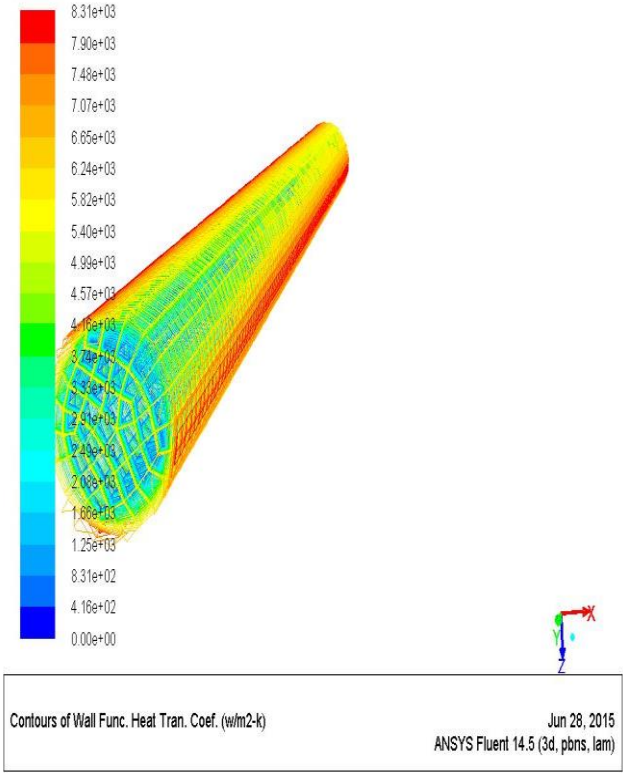
INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

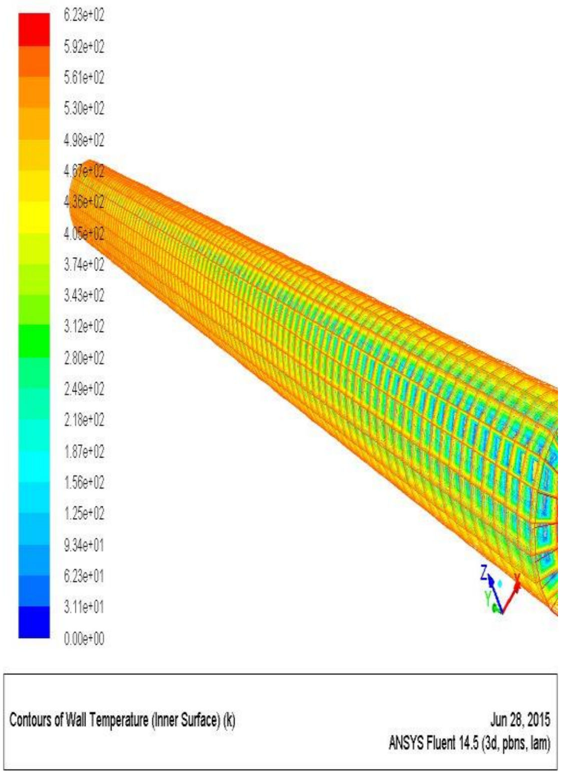
ISSN (ONLINE): 2321-3051

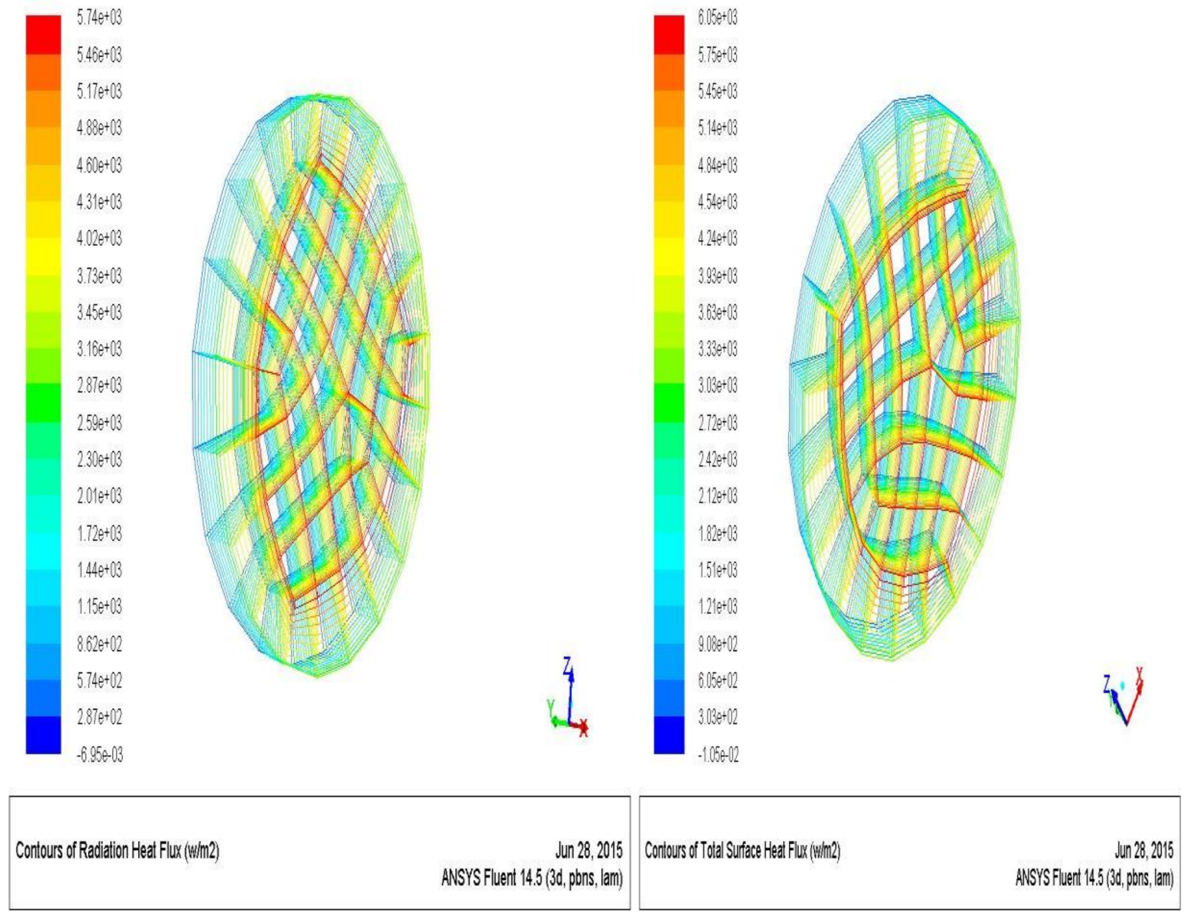
Vol.3 Issue.6,

June 2015.

Pgs: 12-21







5. CONCLUSIONS

Results of present study show that the effects of rarefaction and fluid-wall interaction are important and should be considered for micronatural convective flow and heat transfer problems. Such effects may result in the increase of the volume flow rate and the decrease of the heat transfer rate. As the wall-ambient temperature difference ratio decreases, the effects on the volume flow decreases and those on the heat transfer rate increase. The present analytical studies help the understanding of fluid transport and heat transfer behavior in microchannels and benefit the design of micropumps and microheat exchangers. Rosseland Radiation Model and Boussinesq density parameter is best suitable tool for getting accurate and desired effects and result.

References

[1] I. P. Jones, `A Comparison Problem for Numerical Methods in Fluid Dynamics, The 'Double-Glazing' Problem', in *Numerical Methods in Thermal Problems, Proceedings of the First International Conference*, 338-348 (1979).

[2] G. de Vahl Davis and I. P. Jones, `Natural Convection in a Square Cavity: A Comparison Exercise', *International Journal for Numerical Methods in Fluids*, **3**, 227-248 (1983)

[3] Ostrach, S., 1952, "Laminar Natural Convection Flow and Heat Transfer of Fluids With and Without Heat Sources in Channels with Constant Wall Temperature," Report No. NACA TN 2863. [4] Sinha, P. C., 1969, "Fully Developed Laminar Free Convection Flow between Vertical Parallel Plates," Chem. Eng. Sci., **24**, pp. 33–38.

[4] Aung, W., 1972, "Fully Developed Laminar Free Convection between Vertical Plates Heated Asymmetrically," Int. J. Heat Mass Transfer, **15**, pp. 1577–1580.

[5] Vajravelu, K., and Sastri, K., 1977, "Fully Developed Laminar Free Convection Flow Between Two Parallel Vertical Walls," Int. J. Heat Mass Transfer, **20**, pp. 655–660.

[6] Vajravelu, K., 1979, "The Effect of Variable Fluid Properties on the Free Convective Flow of Air/Water Confined Between Two Parallel Vertical Walls," Acta Mech., **31**, pp. 199– 211.

[7] Chenoweth, D. R., and Paolucci, S., 1985, "Gas Flow in Vertical Slots With Large Horizontal Temperature Differences," Phys. Fluids, **28**, pp. 2365–2374.

[8] Chenoweth, D. R., and Paolucci, S., 1986, "Gas Flow in Open Vertical Slots With Large Horizontal Temperature Differences and Arbitrary External Temperature," Phys. Fluids, **29**, pp. 3187–3198.

[9] C.-K. Chen, H.C. Weng, Natural convection in a vertical microchannel, J. Heat Transfer 127 (2005) 1053–1056

[10] 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.

[11] O.M. Haddad, M.M. Abuzaid, M.A. Al-Nimr, Developing free convection gas flow in a vertical open–ended microchannel filled with porous media, Numer. Heat Transfer A 48 (2005) 693–710. [13] C.-K. Chen, H.C. Weng, Developing natural convection with thermal creep in a vertical microchannel, J. Phys. D: Appl. Phys. 39 (2006) 3107–3118.

[12]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

[13]Kavehpour, H. P., Faghri, M., and Asako, Y., 1997, "Effects of Compressibility and Rarefaction on Gaseous Flows in Microchannels," Numer. Heat Transfer, Part A, **32**, pp. 677–696. [ISI] first citation in article