International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

Improvement of Heat Transfer Using Nanofluids

K.Srinivas
M.Tech Scholar
Department Of Mechanical Engineering
Sphoorthy Engineering College
Nadergul,Saroornagar-Hyderabad

Dr.G.Poshal
Professor & HOD
Department Of Mechanical Engineering
Sphoorthy Engineering College
Nadergul,Saroornagar-Hyderabad

P.Sudheer Rao
Associate Professor
Department Of Mechanical Engineering
Sphoorthy Engineering College
Nadergul,Saroornagar-Hyderabad

Abstract: **In this paper, thermal and flow behavior models for circular microchannel using water and its nanofluids with alumina as a coolant fluid in single phase flow have been developed. A finite volume-based CFD technique is used and models are solved by using Fluent Solver. The 2D axis symmetric geometry with structured mesh and 100 x 18 nodes are used for single phase flow with Al2O3 nanoparticles of 23 nm average diameter. Viscous laminar and standard k-εmodels are used to predict the steady temperature in laminar and turbulent zone. The heat transfer enhancement upto 83% in laminar and turbulent zones are obtained with the Reranging from 5 to 11980 and particle volume concentration from 0 to 5%. Even though the pressure drop increases with increase in Re, it is comparatively less compared to the corresponding decrease in temperature. The increase in temperature depends on Re and Pe; but the temperature distribution is found to be independent of radial position even for verylow Pe. Comparison with analytical results both in laminar and turbulent zone is provided to justify the assumptions introduced in the models and very close agreement is observed statistically. Nusselt number can well predict the analytical data.**

Keywords: **Nanofluids, Porous media, Effective thermal conductivity, Effective viscosity, porosity, Permeability, Inertia coefficient**

1. INTRODUCTION

Nanofluids are engineered colloids made up of a base fluid and the nanoparticles. The introduction of nanoparticles enhances the heat transfer performance of the base fluids significantly. The base fluids may be water, organic liquids (e.g., ethylene, tri-ethylene-glycols, refrigerants, etc.), oils and lubricants, biofluids, polymeric solutions, and other common liquids. The nanoparticle materials include chemically stable metals (e.g., gold, copper), metal oxides (e.g., alumina, silica, zirconia, and titania), oxide ceramics (e.g., Al2O3, and CuO), metal carbides (e.g., SiC), metal nitrides (e.g., AIN, SiN), carbon in various forms (e.g., diamond, graphite, carbon nanotubes, and fullerene), and functionalized nanoparticles. The benefits of using nanofluids compared to the conventional base fluids are as follows.

1. The amount of heat transfer increases as a result of increase in the heat transfer surface area between the particles and fluids.
2. The pumping power required for the equivalent heat transfer is less than that compared to pure liquids.
3. The properties like thermal conductivity, density, and so forth may be varied by varying particle concentrations to suit different applications.

The nanofluids have various applications as they provide an efficient thermal energy transfer due to higher heat transfer with comparable pumping power required. Some of these applications are stated as follows.

(1) Heating and cooling of buildings.
(2) Cooling of electronics.
(3) Cooling of welding.
(4) Nuclear systems cooling.
(5) Solar water heating.
(6) Refrigeration (domestic refrigerator, chillers).
(7) Biomedical applications.
(8) Drilling.
(9) Lubrications.
(10) Thermal storage.

The effective physical properties are required to be calculated from the properties of nanoparticles and base fluid of all the properties, the thermal conductivity is the most important one regarding many applications. Several models have been developed for predicting the thermal conductivity like byMaxwell-Garnett, Hamilton and Crosser, Buongiorno, Koo and Kleinstreuer, Keblinski et al., Azizian et al., Jang and Choi, Kumar et al., Patel et al., and several others. The classicalmodels likeMaxwell-Garnett andHamilton-Crosser models have failed to predict the significant enhancement in the thermal conductivity of the nanofluids. This shows that other mechanisms are also present which increases the thermal conductivity and in turn enhances the heat transfer in nanofluids.

2. INTRODUCTION TO FLUID MECHANICS

Fluids are very familiar to us. Our body itself is mostly water while what surrounds us is largely air, which again is a fluid. In fact, Greeks and Indians in the past worshiped earth, fire, sky, water and air three of these being fluids. Fluid Mechanics is a science that studies the behavior of fluids and its effect on other bodies. It comprises of Fluid Statics, which is a study of fluids at rest and Fluid Dynamics, which is a steady of fluid in motion.

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

There are three approaches to Fluid Mechanics – Experimental, Theoretical and Computational**.** Experimental approach is the oldest approach, perhaps also employed by Archimedes when he was to investigate a fraud. It is a very popular approach where you will make measurements using a wind tunnel or similar equipment. But this is a costly venture and is becoming costlier day by day. Then we have the theoretical approach where we employ the mathematical equations that govern the flow and try to capture the fluid behavior within a closed form solution i.e., formulas that can be readily used. This is perhaps the simplest of the approaches, but its scope is somewhat limited. Not every fluid flow renders itself to such an approach. The resulting equations may be too complicated to solve easily. Then comes the third approach- Computational. Here we try to solve the complicated governing equations by computing them using a computer. This has the advantage that a wide variety of fluid flows may be computed and that the cost of computing seems to be going down day by day. With the result the emerging discipline Computational Fluid Dynamics, CFD**,** has become a very powerful approach today in industry and research.

3. COMPRESSIBLE FLOW

Compressible flow is defined as that flow in which the density of the fluid does not remain constant during flow. This means that the density that the density changes from point to point in compressible flow.

The examples of compressible flow are as follows:

i.) flow of gases through orifices and nozzles,

ii) Flow of gases in machines such as compressors, and

iii) Projectiles and airplanes flying at high altitude with high velocities, entire the density of the fluid changes during the flow. The change in density of a fluid is accompanied by the changes in pressure and temperature and hence the thermodynamic behavior of the fluids, will have to be taken into account.

THERMODYNMIC RELATIONS:

The thermodynamic relations have been discussed as follows:

Equation of State: Equation of state is defined as the equations which give the relationship between the pressure, temperature and specific volume of gas. For a perfect gas the equation of state is

$$Pv = RT \quad \text{...... (4.1)}$$

Where P= Absolute pressure in N/ m^2

v= Specific volume or volume per unit mass

T= Absolute temperature=273+ t^0 (centigrade)

R= Gas constant in (J/kg K)

287 J/kg for air.

In equation (2.1), v is the specific volume which is the reciprocal of density

$$V = \frac{1}{\rho} .$$

Substituting this value v in equation (4.1), we get

$$\frac{p}{\rho} \quad RT \quad \text{..... (4.2)}$$

4. COMPUTATIONAL FLUID DYNAMICS

Over the past half-century, we have witnessed the rise in the new methodology for attacking complex problem in fluid mechanics, heat transfer and combustion. It has come to the state that wherever there is a flow, computer can help to understand and analyze the same. This new methodology of solving a flow problem using a computer is given the name 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 numerical approach, In this numerical approach, the equations (usually in partial differential form) that govern a process of interest are solved numerically. The technique is very powerful and spans a wide range of industrial and non-industrial application areas.

4.1 Analysation of a Fluid Flow Problem

There are three methods to analyze a fluid flow problem.

1. Experimental
2. Theoretical
3. Computational (CFD)

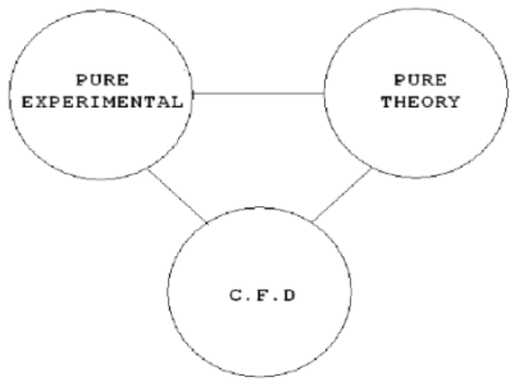


Fig.3.1: The "three dimensions" of fluid dynamics

CFD synergistically complements the other approaches but will never replace either of them. The future advancement of fluid dynamics will rest upon a proper balance of all three approaches, with CFD to interpret and understand theory and experiment and vice versa.

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

Experimental approach

- Most reliable information
- Full scale tests are prohibitively expensive and often impossible
- The general rules for modeling and extrapolation to full scale are often unavailable
- Simulation of all the features such as combustion or boiling are often omitted from models tests
- Serious difficulties of measurement in many situations
- Measuring instruments have uncertainly errors

Theoretical or Analytical approach

- Solve mathematical models rather than physical models
- Analytical methods cannot predict many phenomena
- Analytical or exact solutions are possible only for very simple and ideal situations with many assumptions Examples: ideal flows (potential flows), Couette flow, Blassins flow etc.

4.2 Computational Fluid Dynamics:

Advantages

- Low cost, high speed
- Complete information at any inaccessible point
- Ability to simulate realistic conditions and also ideal conditions
- Can handle any complex geometry

Disadvantages

- Proper mathematical model may not be available
- Validation of computer results needs experimental data

Pre-Requisites for CFD

- Fluid mechanics
- Heat transfer
- Partial differential equations
- Numerical methods
- Any programming language with graphical tools.

4.3 Mathematical Behaviour of Governing Equations In Computational Fluid Dynamics

The development of the high speed digital computer combined with the development of a accurate numerical algorithms for solving problems on these computers has had a great impact on the way principles from the science of Fluid Mechanics are applied to problems of design in modern engineering practice

The physical aspects of any fluid flow are governed by three fundamental principles: conservation of mass, conservation of momentum, conservation of energy and these can be expressed in terms of basic mathematical equations which in their more general form are either integral equations or partial differential equation in computational approach; these equations that govern a process are solved numerically.

These partial differential equations have certain mathematical behavior. This behavior is not fixed and varies from one circumstance to another, depending on the magnitude of the dimensionless flow parameters governing, the situation, the equations governing the flow and the steady or unsteady nature of the flow.

4.4. Discretization:

The word “discretization” requires some explanation. Obviously, it comes from “discrete,” defined in *The American Heritage Dictionary of the English Language* as “constituting a separate thing; individual; distinct; consisting of unconnected distinct parts.” However, the word “discretization” cannot be found in the same dictionary; it cannot be found in *Webster’s New World Dictionary* either. The fact that it does not appear in two of the most popular dictionaries of today implies, at the very least, that it is a rather new and esoteric word. Indeed, it seems to be unique to the literature of numerical analysis, first being introduced in the German literature in 1955 by “W.R. Wasow”, carried on by Ames in 1965 in his classic book on partial differential equations, and recently embraced by the CFD community as closed-for mathematical expression, such as a function or a differential or integral equation involving functions, all of which are viewed as having an infinite continuum of values throughout some domain, is approximated by analogous points or volumes in the domain. This may sound a bit mysterious, so let us elaborate for the sake of clarity. Also, we will single out partial differential equations for the purposes of discussion. Therefore, the remainder of this introductory section dwells on the meaning of “discretization”.

4.5 Explicit and Implicit Approaches:

4.5.1 Definitions and Contrasts: We have discussed some basic elements of the finite –difference method. We have done nothing more than just create some numerical tools for future use; we have not yet described how these tools can be put to use for the solutions of CFD problems. The *way* that these tools are put together and used for a given solution can be called a CFD *technique*, and we have not yet discussed any specific techniques. However, once you choose a specific technique to solve your given problem, you will find that the technique falls into one or the other of two different general approaches, an *explicit* approach or an *implicit* approach. It is appropriate to introduce and define these two general approaches now; they represent a fundamental distinction between various numerical techniques, a distinction for which we need to have some appreciation at this stage of our discussion.

For simplicity, let us return to the one-dimensional equation given by Eq. (5.a) repeated below.

$$\frac{\partial V}{\partial t} \quad \alpha \frac{\partial^2 V}{\partial x^2} \qquad \text{….. (5.a)}$$

We will treat Eq. (5.a) as a “model equation” for our discussion in this section; all the necessary points concerning

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

explicit and implicit approaches can be made using this model equation without going to the extra complexity of the governing flow equations. Above, we used Eq. (5.a) to illustrate what was meant by a difference equation. In particular, in that section we choose to represent $\frac{\partial V}{\partial t}$ with a forward difference and $\frac{\partial^2 V}{\partial x^2}$ with a central second difference, leading to the particular form of the difference equation given by Eq. (5.18) repeated below:

$$\frac{V_i^{n+1} - V_i^n}{\Delta t} \quad \alpha \frac{V_{i+1}^n - 2V_i^n + V_{i-1}^n}{(\Delta x)^2}$$

….. (5.18)

With some rearrangement, this equation can be written as

$$V_i^{n+1} \quad V_i^n + \alpha \Delta t \frac{V_{i+1}^n - 2V_i^n + V_{i-1}^n}{(\Delta x)^2} \quad \text{….. (5.19)}$$

4.6 Finite Volume:

Finite volume method is one of the very popular approximate methods to solve the governing equations originated from fluid dynamics. The governing equations discretized using this method may resemble similar to the equations discretized with finite difference method but the basic idea behind these two schemes is very different. In general, in finite difference method, the mathematically modeled differential or integral equations are taken as the correct and appropriate form of the conservation principles governing the physical problem and then making use of Taylor series or integral methods the differential or integral equations are converted into algebraic form. However, in finite volume method, after discretizing the domain under consideration as sub-domains called control volumes, the conservation statements are applied in each of these control volumes. That is, the conservation principles are made to satisfy in each of the control volumes. The generation of control volumes can be done in two ways

1. *Cell-Centered method:* In this method, the control volumes are identified first and then grid points will be placed at the center of each cell.
2. *Cell-Vertex method:* In cell vertex method the grid points will be identified first and then the boundaries of the control volume are fixed at half way between the grid points. If the grid points are identified non-uniformly in this scheme then these points need not be at the geometric centre of the control volumes.

4.7 MACCORMACK'S TECHNIQUE

MacCormack's technique is a variant of the lax-Wendroff approach but is much simpler in its application. Like the Lax-Wendroff method, the MacCormack method is also an explicit finite-difference technique which is second order accurate in both space and time. First introduced in 1969, it became the most popular explicit finite difference method for solving fluid flows for the next 15 years. Today, the MacCormack method has been mostly supplanted by more sophisticated approaches. However, the understand and program. Moreover, the results obtained by using MacCormack's method are perfectly satisfactory for many fluid flow applications.

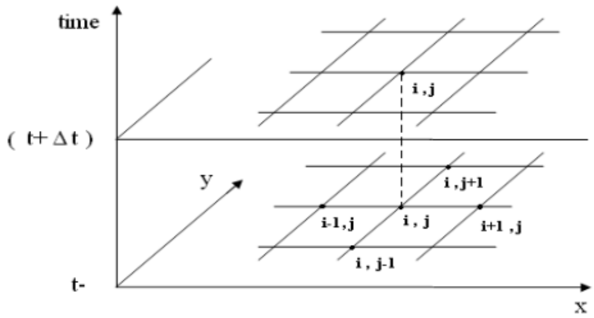


Fig. 4.1: A schematic of the grid for time marching

Consider again two dimensional grid show in fig 'a'. For purpose of illustration, let us address again the solution of the Euler equations itemized. Here we discussed a time marching solution using the Lax Wendroff technique. Here, we will address a similar time marching solution but using MacCormack's technique.

5. SIMULATION OF FLUID FLOW IN A CIRCULAR MICRO CHANNEL

It is well known that nanoparticles have very high thermal conductivity compared to commonly used coolant. Thus, the thermal conductivity and other fluid properties are changed by mixing the particle in fluid. The changed properties of the Nanofluids determine the heat transfer performance of the microchannel heat exchanger with nanofluids. This point is illustrated in this chapter by doing the computational fluid dynamics (CFD) analysis of the hydrodynamics and thermal behaviour of the single phase flow through a circular micro channel (Lee and Mudawar, 2007).

5.1 Specification Of Problem:

Consider a steady state fluid flowing through a circular micro channel of constant cross section as shown in Fig. 4.1 (Lee and Mudawar, 2007). The diameter and length of circular micro channel are 0.0005m and 0.1m respectively. The inlet velocity is u (m/s), which is constant over the inlet cross-section. The fluid exhausts into the ambient atm2osphere which is at a pressure of 1 atm.

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

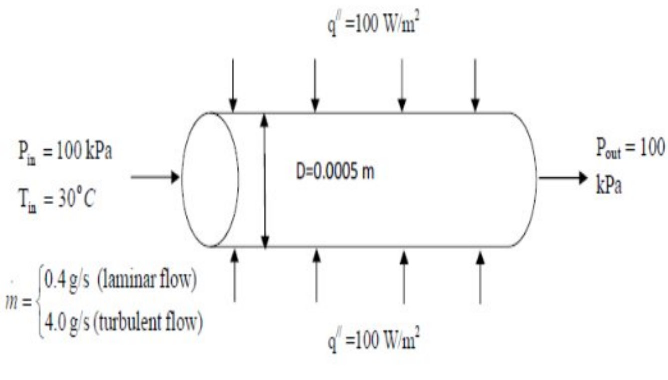


Figure 5.1: Fluid flow through a circular micro channel of constant cross-section

As fluid flows through in a pipe at both hydraulic and thermally fully developed condition, the Nusselt number is constant for laminar flow and it follows the Dittius- Boelter equation for turbulent flow.

Nu = 4.36 for laminar flow 6.1

i.e. $\frac{??}{?}$ = 4.36 ⇒ h ≈ κ 5.2

And

Nu = $0.023Re^{0.8} Pr^{0.4}$ for turbulent flow 6.3

i.e. = $\frac{??}{?}$? ??? ?? $\frac{???}{?}$?$^{??}$ $\frac{???}{?}$?$^{??}$ ⇒ $h \approx \kappa^{0.6} \upsilon^{0.8} \mu^{-0.4}$ 6.4

From Eq. 4.2 and Eq. 4.4 it is clear that thermal conductivity has greater effect on heat transfer coefficient for laminar flow as compared to turbulent flow. This implies the enhancement effect due to the increased thermal conductivity of nanofluids is significantly weaker for turbulent flow than for laminar. The enhancement in turbulent flow is also dependent on flow rate in addition to viscosity and specific heat. Since $h \approx \kappa^{0.6} \upsilon^{0.8} \mu^{-0.4}$ and because increased nanoparticle concentration enhances viscosity and degrades specific heat, the enhancement effect of nanoparticles in turbulent flow is further reduced compared to thermal conductivity alone.

6.2 GEOMETRY IN ANSYS WORKBENCH:

The Computational domain of circular micro channel is represented in two dimensional (2D) form by a rectangle and displayed in Fig. 4.2. The geometry consists of a wall, a centerline, and an inlet and outlet boundaries. The radius, R and the length, L of the pipe are specified in the figure.

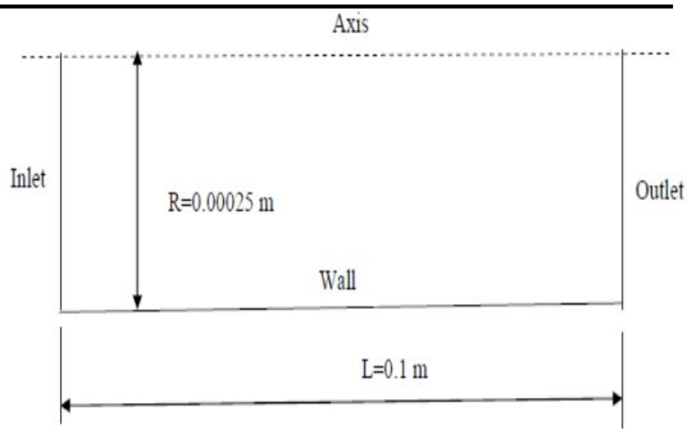


Figure 5.2: Computational Domain of Circular Micro channel

5.3 PHYSICAL MODELS:

Based on the Reynolds number,Re = Duρ/ μ , either viscous laminar model or standard κ −ε model is used for laminar and turbulent flow respectively. The choice of the model is shown in Table 4.1. D is the diameter of the microchannel, ρ and μ are the density and viscosity of the fluid.

Table.1: Choice of model based on Reynolds number

Reynolds no. (Re)	Flow (Model)
< 2000	Laminar
> 2000	κ −ε Model

5.4 MATERIAL PROPERTIES

Pure water is used as base working fluid and Alumina (Al_2O_3) is taken as nanoparticles. The density, heat capacity and thermal conductivity of alumina are 3,600 kg/m^3, 765 J/kgK and 36 W/mK respectively. The properties of nanofluids (nf) are given in Table 6.2 at 30^oC temperature and 100 kPa pressure. Table 4.2 Water base fluid properties with different concentration of alumina nanoparticles (Lee and Mudawar, 2007)

Table .2: Material properties

k (w/mK)	0.603	0.62	0.638	0.656	0.675	0.693
ρ(kg/m³)	995.7	1021.7	1047.7	1073.8	1099.8	1125.9
μ(kg/ms)	7.97E-04	8.17E-04	8.38E-04	8.57E-04	8.78E-04	8.97E-04
Cp(kJ/kg K)	4.183	4.149	4.115	4.081	4.046	4.012

5.5 Boundry Conditions:

A no slip boundary condition was assigned for the non porous wall surfaces, where both velocity components were set to zero at that boundary i.e. $\upsilon_x = \upsilon_r = 0$. A constant heat flux (100 W/m^2) is applied on the channel wall. Axis symmetry was assigned at centerline. A uniform mass flow inlet and a constant inlet temperature were assigned at the channel inlet. At the exit, pressure was specified.

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

5.6 Meshed Model:

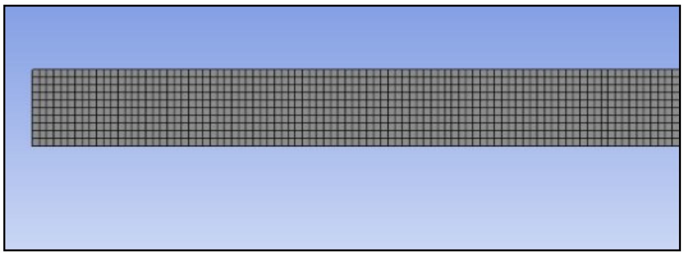


Fig 5.3 Geometry with structured mesh

6. RESULTS

The hydrodynamics behaviour of the channel can be studied in terms velocity distribution within the channel. The value of Re is 1278.0 for inlet mass flow rate 0.4 gm/s and it becomes 12780.0 for 2.0 gm/s inlet mass flow rate. Thus the flow is laminar and turbulent for 0.4 gm/s and 2.0 gm/s inlet mass flow rates respectively. The variation of centerline velocity at laminar state with axial position (X) for water and its nanofluid are displayed in Fig. 7.4. The figure shows that for all the fluids the entrance lengths i.e. the length required to reach fully developed state are the same. The density and viscosity of nanofluids increases with increase in the nanoparticles concentration in water. Thus, the velocity at any axial position decreases with increase in the nanoparticles concentration as found in Fig. 7.4. The same is shown at turbulent state in Fig. 7.5. It is also observed here that the axial velocity decreases with increase in nanofluid concentrations. Unlike laminar flow, the entrance lengths are found different in turbulent flow condition.

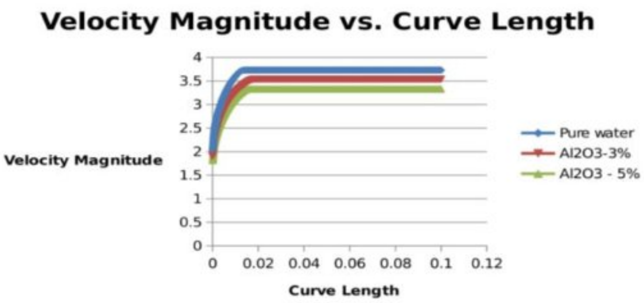


Figure 6.1: Velocity profile at centerline in the circular micro channel at Re =1278 For laminar.

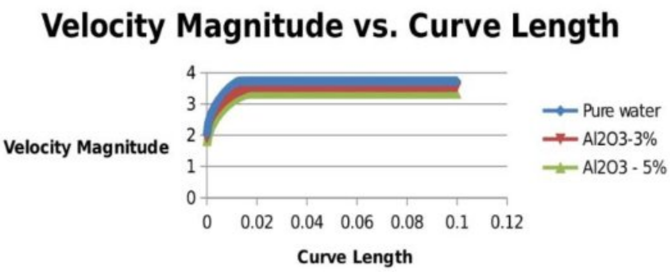


Figure 6.2: Velocity profile at centerline in the circular micro channel at Re =6390 For Turbulent.

The heat transfer data shows that inclusion of nanoparticles in water increases the heat transfer coefficient. More nanoparticle concentration more is the heat transfer coefficient. In case of laminar flow, both the analytical and CFD results show that heat transfer coefficient increases approximately by 15% magnitude from pure water to 5% nanofluid. As expected, both the tabular data and figures show that heat transfer coefficients in turbulent flow are more than laminar values. Like laminar heat transfer coefficient, the values of heat transfer coefficients in turbulent flow also increases with increase in nanoparticles concentration.

CFD values of it increase by 12% magnitude from pure water to 5% nanofluid. Thus, the increase of heat transfer coefficient is more in laminar flow than turbulent flow. But the percentage increase in heat transfer coefficient in turbulent zone is not negligible. These results contradict the results obtained by Lee and Mudawar, (2007). The use of Nanofluids favourable therefore favourable both in laminar and turbulent fluid flow regimes. The heat transfer coefficient values for both type of flow are found to be independent of axial position. It means that circular micro channel is at fully thermal developed condition in both cases.

Laminar:

Pure water (0% of Al_2O_3)

To see the entrance effect of circular micro channel on the velocity profile a velocity contours plot for 0% volume fractions of Al_2O_3 are shown in Figs. 7.3 & 7.4. The figures show that the fluid requires to travel certain distance in the flow direction called entrance length to get fully developed velocity profile. From the velocity contours it is observed that velocity shrinks with increase in nano fluid particle percentage.

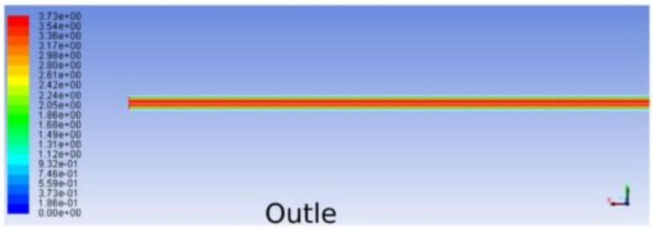


Figure 6.3 Velocity plot of Al_2O_3 – 0% outlet

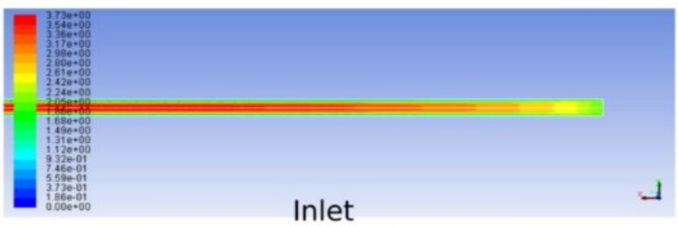


Figure 6.4 Velocity plot of Al_2O_3 – 0% inlet

The graph depicts the trend of velocity with respect to the length of the tube for pure water (0% of Al_2O3). The maximum value of heat transfer coefficient is observed as 3.73 m/s shown in Fig. 6.5.

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

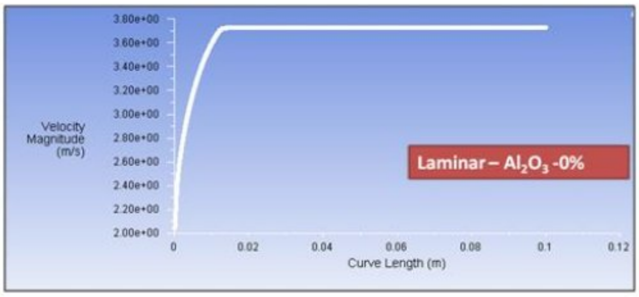


Figure 6.5: Variation of velocity of water in radial direction at different values of X for

Al_2O_3 - 0% Water is used as the fluid in the heat exchanger.

The graph depicts the trend of heat transfer coefficient with respect to the length of the tube for pure water (0% of Al_2O_3). The maximum value of heat transfer coefficient is observed as 4E+04 W/m^2-K is shown in fig 6.6.

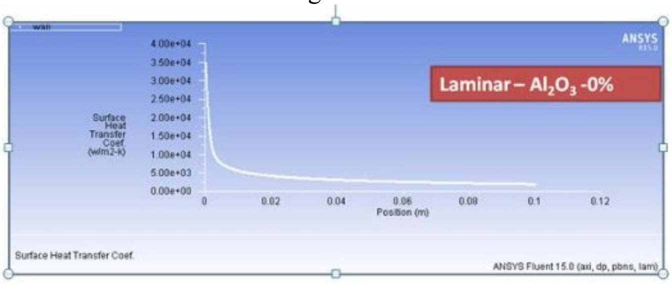


Figure 6.6: Variation of heat transfer coefficient of water in radial direction at different values of X for Al_2O_3 - 0% Water is used as the fluid in the heat exchanger.

Nano Fluid particles 5% of Al_2O_3

To see the entrance effect of circular micro channel on the velocity profile a velocity contours plot for 5% volume fractions of Al_2O_3 are shown in Figs. 6.7 & 6.8. The figures show that the fluid requires to travel certain distance in the flow direction called entrance length to get fully developed velocity profile. From the velocity contours it is observed that velocity shrinks with increase in nano fluid particle percentage.

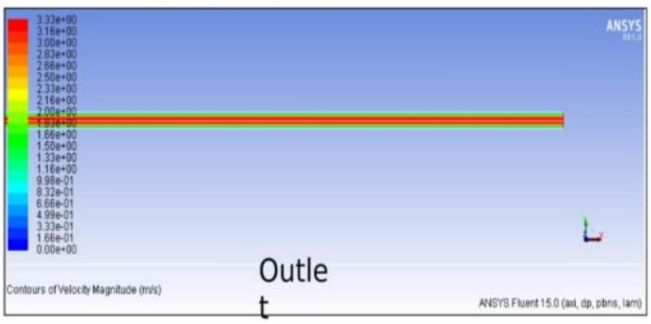


Figure 6.7: Velocity plot of Al_2O_3 – 5% outlet

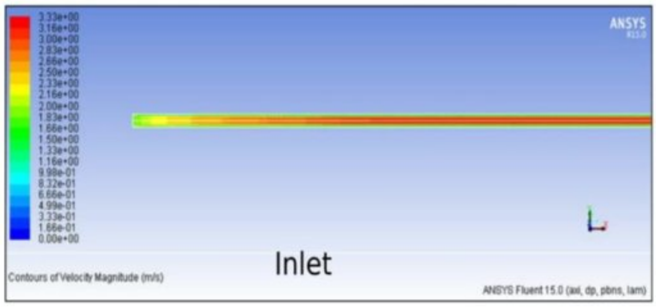


Figure 6.8: Velocity plot of Al_2O_3 – 5% inlet

The graph depicts the trend of velocity with respect to the length of the tube for pure water (5% of Al2O3). The maximum value of heat transfer coefficient is observed as 3.33 m/s shown in Fig.6.9

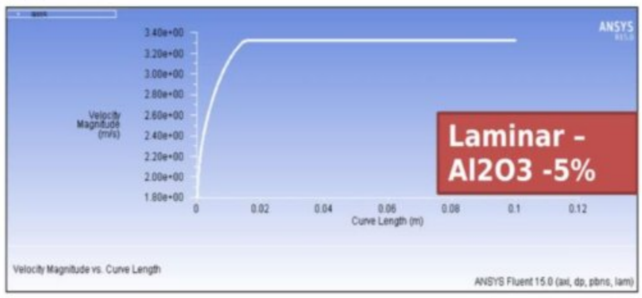


Figure 6.9: Variation of velocity of water in radial direction at different values of X for Al_2O_3 - 5% Water is used as the fluid in the heat exchanger.

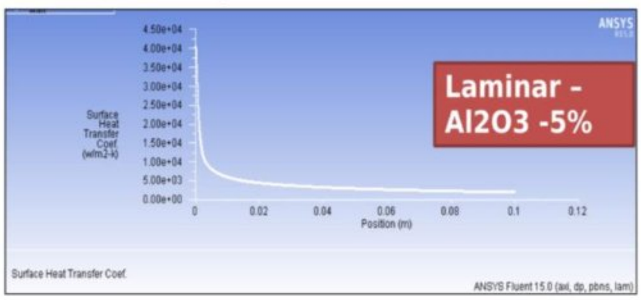


Figure 6.10: Variation of heat transfer coefficient of water in radial direction at different values of X for Al_2O_3 - 5% Water is used as the fluid in the heat exchanger.

Turbulent:

Pure water (0% of Al_2O_3)

To see the entrance effect of circular micro channel on the velocity profile a velocity contours plot for 0% volume fractions of Al_2O_3 are shown in Figs. 6.11 & 6.12. The figures show that the fluid requires to travel certain distance in the flow direction called entrance length to get fully developed velocity profile. From the velocity contours it is observed that velocity shrinks with increase in nano fluid particle percentage.

International Journal of Advanced Research Foundation
Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

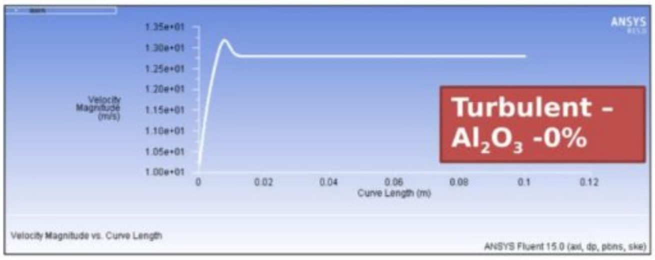


Figure 6.11 Velocity plot of Al_2O_3 – 0% outlet

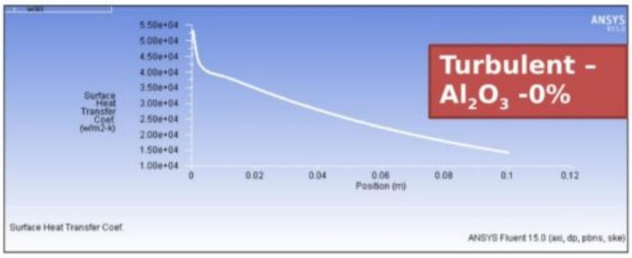


Figure 6.12 Velocity plot of Al_2O_3 – 0% inlet

The graph depicts the trend of velocity with respect to the length of the tube for pure water (0% of Al_2O3). The maximum value of heat transfer coefficient is observed as 13.2 m/s shown in Fig. 6.13.

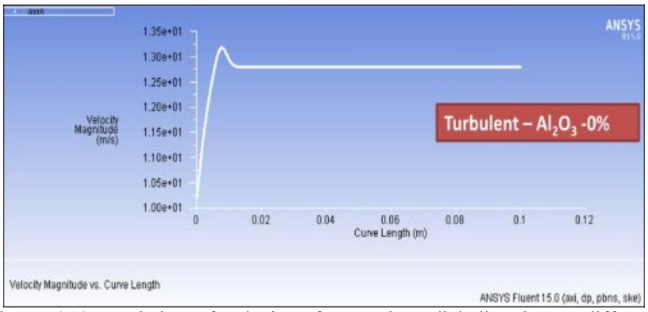


Figure 6.13: Variation of velocity of water in radial direction at different values of X for Al_2O_3 - 0% Water is used as the fluid in the heat exchanger.

The graph depicts the trend of heat transfer coefficient with respect to the length of the tube for pure water (0% of Al_2O_3). The maximum value of heat transfer coefficient is observed as 5.5E+04 W/m^2-K is shown in fig 6.14.

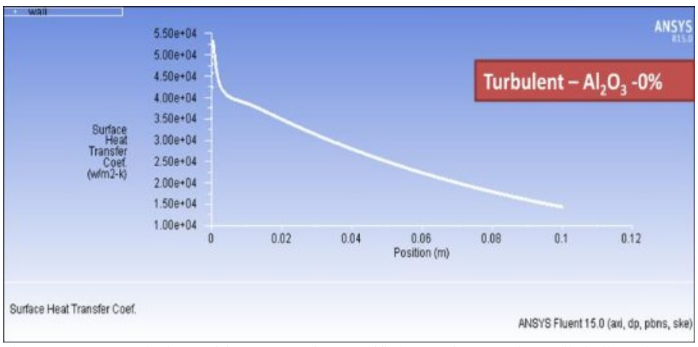


Figure 6.14: Variation of heat transfer coefficient of water in radial direction at different values of X for Al_2O_3 - 0% Water is used as the fluid in the heat exchanger.

Nano Fluid particles 3% of Al_2O_3

To see the entrance effect of circular micro channel on the velocity profile a velocity contours plot for 3% volume fractions of Al_2O_3 are shown in Figs. 7.20 & 7.21. The figures show that the fluid requires to travel certain distance in the flow direction called entrance length to get fully developed velocity profile. From the velocity contours it is observed that velocity shrinks with increase in nano fluid particle percentage.

Nano Fluid particles 5% of Al_2O_3

To see the entrance effect of circular micro channel on the velocity profile a velocity contours plot for 5% volume fractions of Al_2O_3 are shown in Figs. 6.15 & 6.16. The figures show that the fluid requires to travel certain distance in the flow direction called entrance length to get fully developed velocity profile. From the velocity contours it is observed that velocity shrinks with increase in nano fluid particle percentage.

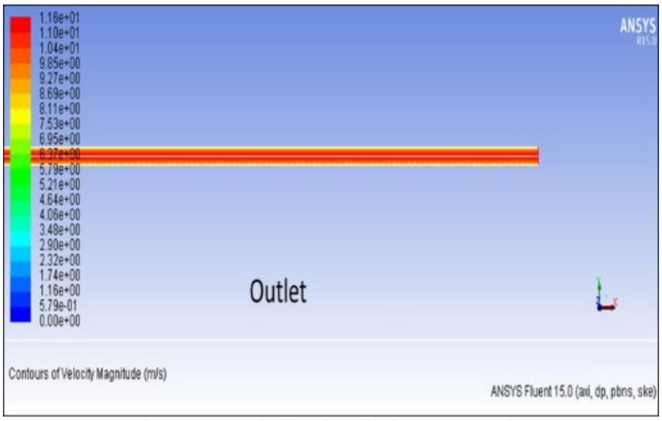


Figure 6.15 Velocity plot of Al_2O_3 – 5% outlet

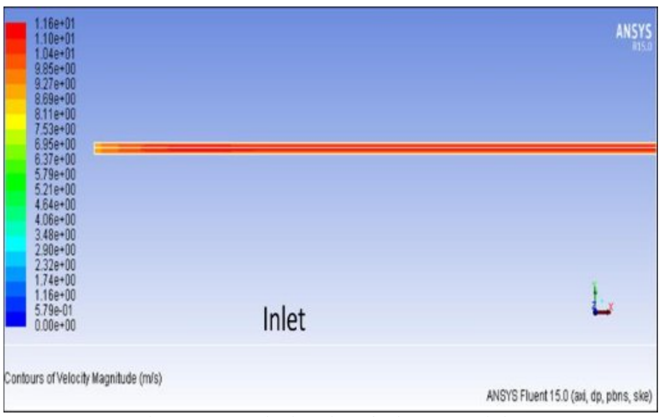


Figure 6.16 Velocity plot of Al_2O_3 – 5% inlet

To see the entrance effect of circular micro channel on the temperature profile a temperature contours plot for 5% volume fractions of Al_2O_3 are shown in Figs. 6.17 & 6.18. The figures show that the fluid requires to travel certain distance in the flow direction called entrance length to get fully developed

International Journal of Advanced Research Foundation

Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

temperature profile. From the velocity contours it is observed that temperature increases with increase in nano fluid particle percentage.

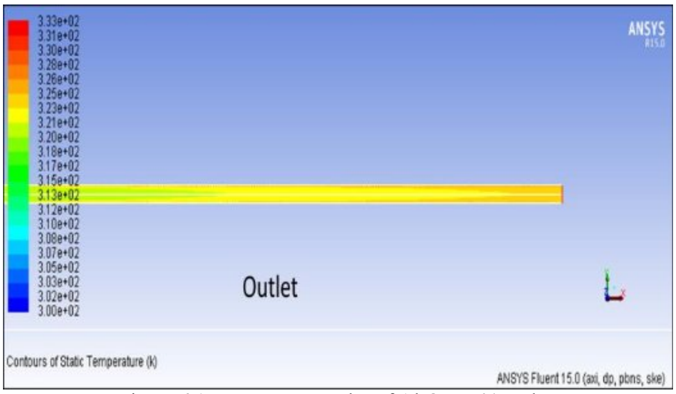


Figure 6.17 Temperature plot of Al_2O_3 – 5% outlet

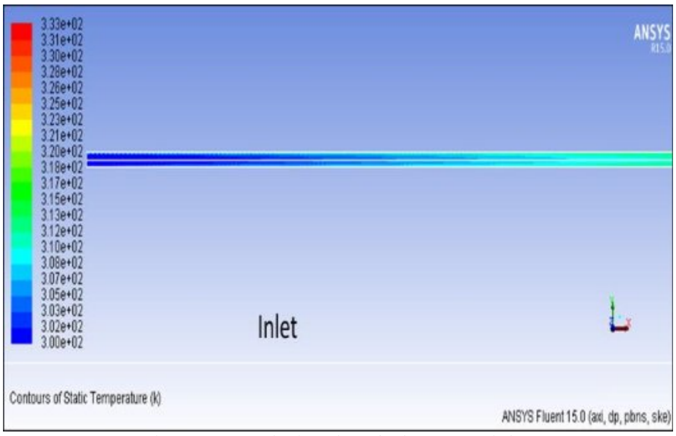


Figure 6.18 Velocity plot of Al_2O_3 – 5% inlet

The graph depicts the trend of velocity with respect to the length of the tube for pure water (5% of Al_2O3). The maximum value of heat transfer coefficient is observed as 11.6 m/s shown in Fig. 6.19.

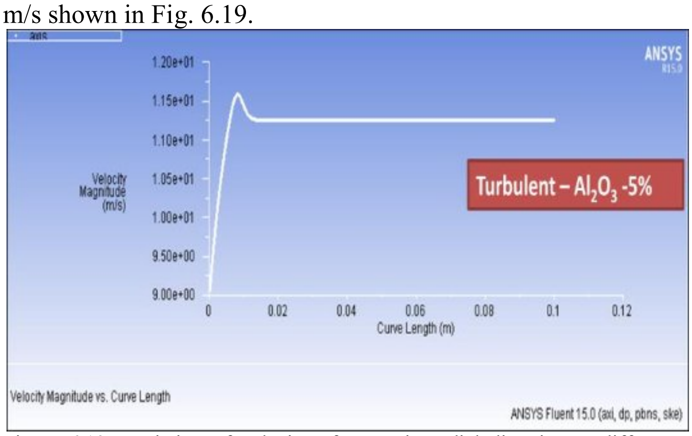
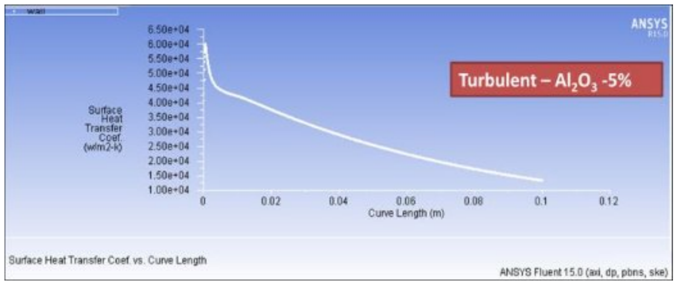


Figure 6.19: Variation of velocity of water in radial direction at different values of X for Al_2O_3 - 5% Water is used as the fluid in the heat exchanger.

The graph depicts the trend of heat transfer coefficient with respect to the length of the tube for pure water (3% of Al_2O_3). The maximum value of heat transfer coefficient is observed as 6.5E+04 W/m^2-K is shown in fig 6.20



.

Figure 6.20: Variation of heat transfer coefficient of water in radial direction at different values of X for Al_2O_3 - 5% Water is used as the fluid in the heat exchanger.

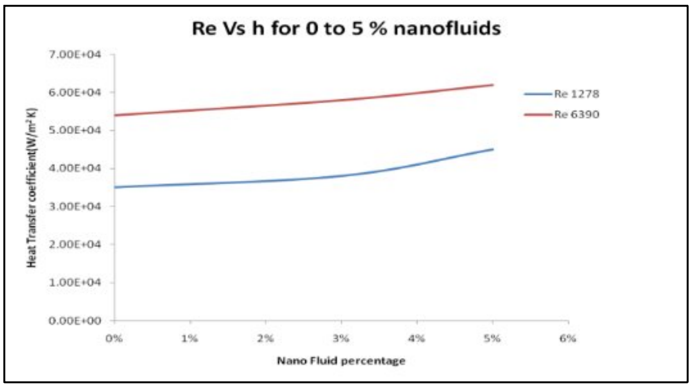


Figure 6.20: Re vs h graph

From figure 6.21 it is observed that as nano fluid percentage is increases the heat transfer coefficient is increases. And the heat transfer coefficient is more in turbulent as compare with laminar.

7. CONCLUSIONS

In this study, the thermal and flow behaviour modelling of circular microchannel has been performed. Velocity, Temperature and Heat transfer coefficient have been formulated. The heat transfer during laminar and turbulent regime has been solved using the viscous laminar and standard k-ε methods. The results show that:

- Thermally developing conditions for a particular Re and nanofluid concentration with higher heat transfer coefficient mostly in entrances region of micro channels. As the concentration of nanoparticle increases, heat transfer coefficient also increases. With increase in Re, heat transfer coefficient also increases.
- The enhancement of heat transfer in Turbulent nanofluid flow is greater as compared to laminar nanofluid flow with respect to its base fluid. Velocity and temperature contours represent successfully the hydrodynamic and thermal behaviour of the microchannel system.

International Journal of Advanced Research Foundation

Website: www.ijarf.com (ISSN: 2394-3394, Volume 3, Issue 10, October 2016)

- Even though axial velocity decrease with increase in nanofluid concentration for laminar and turbulent zones, no variation is found at a particular concentration except for the entrance length. Velocity profile is flat at very low Re and parabolic at higher Re. Wall temperature at an axial position decrease with respect to increase in nanofluid concentration. But there is no nanofluid temperature variation with radial position

REFERENCES

[1]. Bahrami, M and Jovanovich, M. M. 2006. Pressure Drop of Fully Developed Laminar Flow in Microchannels of Arbitrary Cross-Section. Journal of Fluids Engineering. 128, 1036-1044.

[2]. Bahrami, M., Jovanovich, M. M. and Culham, J.R. 2006. Pressure Drop of Fully Developed, Laminar Flow in Rough Microtubes, Journal of Fluids Engineering, 128, 632-637.

[3]. Bianco, V., Chiacchio, F., Manca, O. and Nardini, S. 2009. Numerical investigation of nanofluids forced convection in circular tubes. Applied Thermal Engineering, 29 (17-18), 3632–3642.

[4]. Chein, R. and Chuang, J. 2007. Experimental microchannel heat sink performance studies using nanofluids. International Journal of Thermal Sciences. 46, 57-66.

[5]. Choi, S.U.S., 1995. Enhancing thermal conductivity of fluids with nanoparticles. In: Proceedings of the 1995 ASME International Mechanical Engineering Congress and Exposition, San Francisco, CA, USA. 66, 99-105.

[6]. Das, S.K., Putra, N., Roetzel, W., 2003a. Pool boiling characteristics of nano-fluids. International Journal of Heat and Mass Transfer 46, 851–862.

[7]. Das, S.K., Putra, N., Roetzel, W., 2003b. Pool boiling of nano-fluids on horizontal narrow tubes. International Journal of Multiphase Flow 29, 1237–1247.

[8]. Daungthongsuk, W. and Wongwises, S. 2007. A critical review of convective heat transfer of nanofluids. Renewable and Sustainable Energy Reviews. , 11, 797–817.

[9]. Foli, K., Okabe, T., Olhofer, M., Jin, Y. and Sendhoff, B. 2006. Optimization of micro heat exchanger-CFD, analytical approach and multi-objective evolutionary algorithms, International Journal of Heat and Mass Transfer, 49, 1090-1099.

[10]. Gabriel, G., Favre-Marinet, M. and Asendrych, D. 2005. Conduction and entrance effects on laminar liquid flow and heat transfer in rectangular micro channels. International Journal of Heat and Mass Transfer. 48, 2943–2954.

[11]. Gherasim, I., Roy, G., Nguyen, C. T. and Vo-Ngoc, D. 2009. Experimental investigation of nanofluids in confined laminar radial flows. International Journal of Thermal Sciences. 48, 1486 – 1493.

[12]. Hasan, M. I., Rageb, A. A., Yaghoubi, M. and Homayoni, H. 2009. Influence of channel geometry on the performance of a counter flow microchannel heat exchanger, International Journal of Thermal Sciences, 48, 1607-1618.

[13]. Tu, J.P., Dinh, N., Theofanous, T., 2004. An experimental study of nanofluid boiling heat transfer. In: Proceedings of 6th International Symposium on Heat Transfer, Beijing, China.

[14]. Vassallo, P., Kumar, R., Damico, S., 2004. Pool boiling heat transfer experiments in silica-water nano-fluids. International Journal of Heat and Mass Transfer, 47, 407–411.

[15]. Versteeg, H. and Malalasekra, W. 2007. An Introduction to Fluids dynamics: The Finite Volume Method. 2nd Edition, Pearson.

[16]. Wang, B.X., Zhou, L.P., Peng, X.F., 2003. A fractal model for predicting the effective thermal conductivity of liquid with suspension of nanoparticles. International Journal of Heat and Mass Transfer 46, 2665–2672.

[17]. Wang, Xiang-Qi, Arun, S. and Mujumdar, 2007. Heat transfer characteristics of nanofluids: a review. International Journal of Thermal Sciences. 46, 119.

[18]. Wang, X., Xu, X. and Choi, S.U.S. 1999. Thermal conductivity of nanoparticle–fluid mixture. Journal of Thermophysics and Heat Transfer. 13 (4), 474–480.

[19]. Wen, D.S., Ding, Y.L., 2004a. Effective thermal conductivity of aqueous suspensions of carbon nanotubes (Nanofluids). Journal of Thermophysics and Heat Transfer 18 (4), 481– 485.

[20]. Wen, D.S., Ding, Y.L., 2004b. Experimental investigation into convective heat transfer of nanofluids at entrance area under laminar flow region. International Journal of Heat and Mass Transfer 47 (24), 5181–5188