ANALYSIS OF CONCENTRIC TUBE HEAT EXCHANGER BY USING DIFFERENT INSERTS AND DIFFERENT NANO FLUIDS

APPANAPALLI KARUNAKAR[1], Mr. Dr. A.N.R REDDY[2]

1M.Tech Student, Department of Thermal engineering, MALLA REDDY COLLEGE OF ENGINEERING AND TECHNOLOGY (MRCET), Hyderabad

2 Professor, Department of Thermal Engineering, MALLA REDDY COLLEGE OF ENGINEERING AND TECHNOLOGY (MRCET), Hyderabad

ABSTRACT

The vortex generators induce stream wise longitudinal vortices. These vortices disrupt the growth of the thermal boundary layer and serves to bring about heat transfer enhancement between the fluid and the fin surfaces. The geometrical configuration considered in this study is representative of a channel with winglets spread over three rows each row 13 a pair of winglets. In this study, three dimensional turbulent flow of different nanofluids flow inside a rectangular duct with the existence of vortex generator winglets at different angle are (10º, 20º and 30º) is numerically investigated. The effects of type of the nano particles, and Reynolds number on the heat transfer coefficient and pressure drop of nano fluids are examined. Reynolds numbers (10000, 20000, 30000, 40000 and 50000). A constant surface temperature is assumed to be the thermal condition for the upper and lower heated walls. In the present work, three nanofluids are examined which are Al2O3, Cu O2 and SiO2 suspended in the base fluid of water with nanoparticles concentration ranged = 4% and the nanoparticles diameter, DP is (30 nm). The validity of the code is tested by comparing the results for a three-dimensional experimental the published results with numerical results. The results are in good agreement with the published results. It is observed from the results that the heat transfer increases with the increase in the angle of attack and Reynolds number. The result reporting showed that the case of channel with winglet at angle 30˚ presented highest heat transfer rate. Where that, the case of channel with winglet at angle 20˚ presented, the Nu values are lower than the winglet with angle 30˚. Also, the case of channel with winglet at angle 10˚ and smooth channel presented, the Nu values are apparently lower than the winglet with angle 30˚

Keywords: CFD, Winglets, nano fluids, Nusselt number, Reynolds numbers.

INTRODUCTION

Forced convection heat transfer in a channel had been a subject of interest in many research studies over the past decades. The subject of heat transfer enhancement has significant interest to develop the compact heat exchangers in order to obtain a high efficiency, low cost, light weight, and size as small as possible. Therefore, energy cost and environmental considerations are going on to encourage attempts to invent better performance over the existing designs. Stream wise vortices can be generated using small flow manipulators or protrusions such as wings and winglets configurations. Single -pair, single row, or two dimensional arrays of vortex generators (VG) can be punched, mounted, attached or embedded in the boundary layer of flow channel as in figure 1.2. (VG) generate longitudinal and transverse vortices, while longitudinal vortices are more efficient for heat transfer enhancement than transverse vortices. A dramatic augmentation in thermal performance of the thermal system can be achieved, but the pressure drop penalty is existed [1,2] in this area have been carried out, but very few attempts of numerical investigation have been made so far due to complexity of flow pattern and computational limitations. In this work, an attempt is made to predict numerically the details of both the velocity and temperature fields responsible for heat transfer enhancement.

Problem Statement

Rectangle channel is widely used in such fields as heat exchangers, automobile industries, power systems, solar water heater, solar air heater, heating and air conditioning, chemical engineering, electronic chip cooling and aerospace, etc. This subject has received considerable attention to get high efficiency, low cost, small size, and light weight. For the vehicle industries, an additional benefit can be considered such as a small size of the radiator tends to get a small frontal area of the vehicle which in turn leads to diminishing the drag force and subsequently minimizes the fuel consumption. Several engineering applications had held an enormous attention to improve heat transfer performance of the heat exchangers. The traditional heat exchangers are generally improved by serving enhancement techniques with concentration on many types of surface augmentation. The augmentation of heat transfer can be done by disrupting the boundary layer growth, increasing the turbulence intensity, and generating secondary flows.

In the past years, many techniques were utilized to improve the thermo physical properties of fluids by increasing its thermal conductivity. One of these techniques include adding the high thermal conductivity metal nanoparticles (gold, copper, aluminum, silver) etc. to the base fluid. This resulting in increasing the thermal conductivity of mixtures and provided new fluids which called nanofluids, such as, alumina (Al2O3), copper oxide (CuO), silicon dioxide (SiO2), zinc oxide (ZnO) and etc. [5,6]. Nanofluids are prepared by dispersing less than 100 nanometer-sized particles in a base fluid such as water, ethylene glycol, oil and other conventional heat transfer fluids and these types of fluids have high thermal conductivity. There are many reasons lead us to use nanofluids as working fluids

1. Conventional heat transfer fluids have inherently poor thermal conductivity compared to solids.

2. Conventional fluids that contain mm- or mm-sized particles do not work with the emerging "miniaturized" technologies because they can clog the tiny channels of these devices.

3. Modern nanotechnology provides opportunities to produce nanoparticles.

4. Argonne National Lab (Dr. Choi's team) developed the novel concept of nanofluids.

5. Nanofluids are a new class of advanced heat-transfer fluids engineered by dispersing nanoparticles smaller than 100 nm (nanometer) in diameter in conventional heat transfer fluids.

6. Due to the advantages of utilization the nanofluids as working fluids, the effects of nanofluids on the heat transfer rate are investigated in the present study. The heat transfer enhancement resulting from the diffrent type of nanofluids with pure water, the nano-particles diameter, is (30 nm) and volume fractions (concentration) of nanoparticles are used in this study (4%) The Reynolds numbers used in this study are ranged from 10,000 to 50,000.

Application of the study

As evident from the diversity of application of heat exchangers areas, the study of flow and heat transfer in channel is very important for the technology of today and the future in order to develop and produce high efficiency, small size heat exchangers. Heat exchangers are mostly used devices in many areas of the industries. Hence, the using of high performance heat exchangers is very important for saving energy. But there is limited research related to the high performance thermal system of double pipe heat exchangers by using CFD models. Heat transfer enhancement has significant meanings for environmental problems. The heat transfer coefficient, friction factor and the value of performance evaluation criterion (PEC) in heat exchanger depend on different geometrical parameters of winglet which are investigated in the past studies. The winglet in channel with different geometrical configurations are widely used to enhance the heat transfer rate in many engineering applications, for example, heat recovery processes, air conditioning and refrigeration systems, internal cooling of gas turbine blades, chemical reactors, thermal regenerators, gas-cooled reactors, food and dairy processes, boilers and heat recovery systems.

The Objectives of study

This study is conducted to investigate numerically the utilization of winglet in duct under turbulent flow of nanofluids as the objectives of the present study are:

1. To study the effects of geometrical parameters of winglet on the thermal and flow Fields. .

2. To examine the effects of (SiO_2, Al_2O_3, CuO) Nanofluids on the thermal and flow fields.

3. To analyze the effect of different Reynolds numbers Re in the range of (10000, 20000, 30000, 40000 and 50000) on the thermal and flow fields.

Scope of study

Scope of this research is numerical studies of enhance heat transfer in channel with Winglets and nanofluid. This work can be summarized as follows:

1. This numerically study 3D using ANSYS software 15.0

2. The study model, rectangular duct with winglets fixed on upper and lower from surfaces Duct.

3. Use different type of nanofluid (water+SiO2), (water + Al2O3) and (water + Cu O2) as fluid the coolant, the concentration nanoparticles, (4%) and the diameter of all nanoparticles, dp is (30 nm).

4. The using difference Reynolds numbers (10000, 20000, 30000, 40000 and 50000).

LITERATURE REVIEW

Enhanced convective heat transfer in heat exchanger by passive techniques has been a subject of interest for scientists and researchers in the past decades. Numerical and experimental studies have been reported in order to increase the amount of heat transferred by these techniques. Literature reviews on passive techniques of heat transfer augmentation focuses on different types of winglet in different application. The needs to reduce the cost, dimensions and save energy for heat exchanger have stimulated the search for various techniques of heat transfer enhancement. Present work is carried out using one of the passive heat transfer augmentation techniques which is winglet. Different types of inserts have been evaluated and examined by various investigators in order to enhance the heat transfer rates in duct. Figure 2.1 display different types of longitudinal vortex generators.

The thermo hydraulic performance of an insert is determined with reference to its ability to enhance the heat transfer coefficient with a minimum increase in friction factor. Heat exchanger channel with winglets inserts have been used for many years as reliable means for heat transfer enhancement and fouling mitigation in petroleum refineries and chemical plants.

Applications of winglet vortex generator

There are different types of winglets were used in many application such as the solar heater water, heat exchanger and refrigeration system. Experiments on enhancement of heat transfer in a tube with using winglets have been widely reported.

Ahmed et al. studied the effect of vortex generator in a triangular duct on heat transfer enhancement of laminar nanofluids flow. Two dimensional laminar flow of different nanofluids flow inside a triangular duct with the existence of vortex generator was investigated numerically. The governing equations of mass, momentum and energy were solved using the finite volume method (FVM). The effects of type of the nanoparticles, particle concentrations, and Reynolds number on the heat transfer coefficient and pressure drop of nanofluids were examined.

Winglets in Circular tube

Experimentally the heat transfer and friction behaviors of turbulent tube flow through a straight tape with double-sided delta wings (T-W). They used the T-W formed on the tape as vortex generators for enhancing the heat transfer coefficient by breakdown of thermal boundary layer and by mixing of fluid flow in tubes. The T-W characteristics were (1) T-W with forward/backward-wing arrangement, T-W with alternate axis (T-WA), three wing-width ratios and wing-pitch ratios. The experimental result showed that the using of the T-W, led to increase the Nusselt number (Nu) and friction factor were, respectively, up to 165% and 14.8 times of the plain tube and the maximum thermal performance factor was 1.19. It was also obvious that the T-W with forward-wing gave higher heat transfer rate than one with backward-wing around 7%. It was also found that the heat transfer rate and friction.

Winglets in channel or duct

Numerically the heat transfer and erosion characteristics to improve heat transfer performance and reduce erosion of economizers in coal-fired power plants for the single H-type finned oval tube with enhanced heat transfer structures including bleeding dimples, longitudinal vortex generators (LVGs), and compound dimple-(LVG).

From the results, it can be noted that the oval tube with compound LVG-dimple enquired the highest heat transfer performance while the oval tube with LVG worked most efficiently in the anti-wear performance. Then based on the H-type finned oval tube, the LVG structure on the first row of tubes together with hemisphere protrusions design, while the compound LVG-dimple on the rest tubes were also simulated. The optimized H-type finned oval tube bank heat exchanger was demonstrated of high performance on both heat transfer and anti-wear.

COMPUTATIONAL FLUID DYNAMICS

In this study a single phase models are used for solving the respective category problems. This model will calculate one transport equation for the momentum and one for continuity for each phase, and then energy equations are solved to study the thermal behavior of the system. The theory for this model is taken from the ANSYS 15.0.

COMPONENTS OF A NUMERICAL SOLUTION METHOD

1. MATHEMATICAL MODEL:

The starting point of any numerical method is the mathematical model, the set of partial differential or integral-differential equation and boundary conditions. A solution method is usually designed for a particular set of equations. Trying to produce a general purpose solution method, one which is applicable to all flows, is impractical, if not impossible and, as with most general purpose tools, they are usually not optimum for any one application.

2. DISCRETIZATION METHOD:

After selecting the mathematical model, one has to choose a suitable disc recitation method a method of approximating the differential equations by a system of algebraic equations for the variables at some set of discrete locations in space and time. There are many approaches, but the most important of which

Are: finite difference (FD), finite volume (FV), and finite element (FE) methods.

3. COORDINATE AND BASIS VECTOR SYSTEM**:**

The conservation equation can be written in many different forms, depending on the coordination system and the basis vectors used. For example one can select Citation, cylindrical, spherical, curvilinear orthogonal coordinate system, which may be fixed or moving. The choice depends on the target flow, and man influence the discretization method and grid type to be used.

4. NUMERICAL GRID:

The discrete location at which the variables are to be calculated are defined by numerical grid which is essentially a discrete representation of the geometrical domain on which the problem is to be solved .It divides the solution domain into a finite number of sub domains (elements , control volumes etc). Some of the options available are following: Regular or structured grid consist of families of grid lines with the property that members of a single family do not cross each other and cross each member of the other family only once . This allows the lines of a given set to be numbered consecutively. The position of any grid point (or control volume) within the domain is uniquely identified by a set of two (in 2D) or three (in 3D) indices Structured grids may be of H-, O-, or C-type; the names are derived from the shapes of the grid lines.

RESULTS AND DISCUSSIONS

The variation of velocity of water and its nanofluid (1% alumina and 2% alumina) with axial position (x) at Re=140 is shown in Fig. 5.5. Velocity is at centerline as indicted in figure 5.5. It is observed almost at the entrance velocity of all types of fluids got fully developed. The entrance length of all the fluids also found to be same.

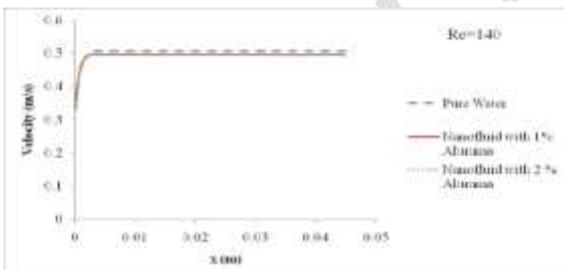

Figure 1: Velocity profile for water and its nonofluid at Re=140

Comparison of the present computed pressure drops and friction factors for water and its nano fluid at different Re 140-941 with experimental results are depicted in Table 5.2, 5.3 and 5.4 respectively. The same is also shown in Fig. 5.6 to 5.8. These show that as Reynolds number increases pressure drop increases and friction factor decreases. Here one thing is noticeable that the pressure drop increases with increasing nano particle concentration at the same Reynolds number. For example at Reynolds number 200, pure water and its nanofluid with alumina (1% and 2% volume concentration) gives pressure drop across micro channel equal to 0.38 bar, 0.46 bar and 0.61 bar respectively.

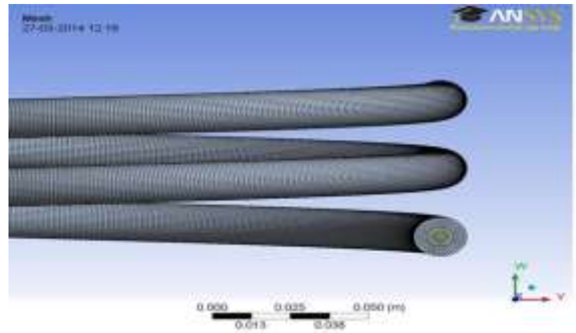

Figure 12 Ansys model of concentric tube of STHE

CONCLUSIONS AND FUTURE SCOPE

In this chapter the salient accomplishments and major conclusions of this work are summarized and recommendations for the future are made.

CONCLUSIONS

In this work the hydrodynamics and thermal behavior of circular nanofluids and a rectangular nanofluids present in a test rig were studied. Pure water and its nanofluids (Al2O3) were used as the coolant in the channel. A steady state computational fluid dynamics (CFD) models was simulated by ANSYS Fluent 15.0 here. The effect of Reynolds number and Peclet number on the flow behaviour of the nanofluids s was found in both cases.

Based on the analysis of the circular nanofluids behaviour the following conclusions can be drawn

- Computed temperatures and heat transfer coefficients were found in close agreement with the analytical values.
- The use of nanofluids as the heat transport medium in the channel were found useful both in laminar and in turbulent flow conditions.
- The change of temperature from inlet to outlet was found increasing with decreasing Reynolds number.
- Temperature distribution was found independent of radial position even at very low value of Peclet number.

- •Pressure drop increases with increase in Reynolds number.
- •The entrance length for fully developed flow depends on Nanoparticle concentrations.
- •Wall temperature has negligible variation for higher Reynolds due to greater value of Peclet no in circular micro channel.

Based on the analysis rectangular nanofluids study, the following conclusions can be Made

- •The computational temperature variations, pressure drop and friction factor values can predict the experimental data.
- •As the concentration of nanoparticle increases heat transfer coefficient also increases Greater heat transfer coefficient is obtained at rectangular micro channel entrance. Heat transfer coefficient deceases from entry to exit region in rectangular micro channel whereas it is constant in circular micro channel due to fully developed conditions
- •Wall temperature increases from entry to exit region of rectangular micro channel.
- •Pressure and temperature contours represent successfully the hydrodynamic and thermal behavior of the system

FUTURE SCOPE OF THE WORK

- •Modeling and Simulation of two phase flow in micro channel.
- •Analysis of the boiling characteristics of nanofluids using CFD models.

REFERENCES

- Allen, P. W. 2007. Experimental and Numerical Investigation of Fluid Flow and Heat Transfer in Microchannels. Msc Thesis, Mechanical Engineering Department, Louisiana State University.
- Al-Nimr, M.A., Maqableh, M., Khadrawi, A.F. and. Ammourah, S.A. 2009. Fully developed thermal behaviors for parallel flow microchannel heat exchanger, International Communications in Heat and Mass Transfer. 36, 385-390.
- ANSYS Fluent 12.0 Theory Guide, April, (2009).
- Bachok, N., Ishak, A. and Pop, I. 2011. Flow and heat transfer over a rotating porous disk in a nanofluid. Physica B. 406, 1767 – 1772.
- Bayraktar, T. and Pidugu, S.B. 2006. Characterization of Liquid Flows in Microfluidic Systems. International Journal of Heat and Mass Transfer. 49, 815-824.
- 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.
- 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.
- 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.