International Journal of Modern Studies in Mechanical Engineering (IJMSME)
Volume 1, Issue 1, June 2015, PP 8-18
www.arcjournals.org

Single-Phase Convective Heat Transfer Enhancement in A Two Dimensional Semi-Circular Protrusion on Fin Surface at A Constant Heat Flux (CHF) Condition

Piyush Kumar Kashyap[1], Priyanka Jhavar[2]

Sri Satya Sai, College of Engineering & Science Sheore
[1]*Kashyap.piyush1989@gmail.com*, [2]*Priyanka.jhavar10@gmail.com*

Abstract: *The enhancement to the flow and heat transfer in a semi-circular protrusion fitted with fins. Understanding these kinds of pin fins which are used widely in industry are believed give a rich contribution to elucidation of the phenomena in normal and compact heat exchangers used in different application. It was, therefore, chosen to investigate the heat transfer and pressure drops characteristics in a channel containing semi-circular fins at both low and high Reynolds numbers. the present report deals with the numerical investigation of various aspects of single-phase convective heat transfer enhancement in a two dimensional semi-circular protrusion on fin surface at a constant heat flux (CHF) condition. By applying the conjugate heat transfer boundary conditions, numerical simulations close to the realistic working conditions were performed. Pressure, temperature and velocity profile were drawn for different inlet velocities for a protrusion arrangement, considering the flow to be laminar and turbulence. The working fluid considered here is air. Numerical study was done using Fluent software, for the same Reynolds number and under same boundary conditions by applying single protrusion of semi-circular shape on a longitudinal fin of inner tube and the results revealed that the same Nusselt number 20, results at Reynolds number of 300 with single protrusion using air as cooling fluid. The present numerical result at Reynolds number 300 is validated with the results of Kubacki S. for the same conditions and constraints. Further numerical study has been done for the optimization of gap between the two protrusions for the maximum heat transfer and results reveal that at a gap of 4L the heat transfer is maximum and both protrusions dissipates equivalent amount of heat. The present work is undertaken to evaluate the performance of the vortex generator used to enhance the heat transfer rate in a heat exchanger. These vortex generators are mounted on the circular fins which are used as inserts between the plates. For detailed investigation a circular protrusion which produces vortex are considered. This work deals with the formulation of the present problem.*

Keywords: *Heat Exchanger, Vortex Generator, Boundary Conditions, Geomerty, Gambit Part.*

1. Introduction

1.1. Heat Exchanger

A heat exchanger is a device which is used to transfer thermal energy between two or more fluids, between a solid surface and a fluid, or between solid particulates and a fluid, at different temperatures and in thermal contact.. The heat exchangers can be classified in several ways such as, according to the transfer process, number of fluids and heat transfer mechanism [1].

To increase the heat transfer area, appendages may be intimately connected to the primary surface to provide an extended, secondary, or indirect surface. These extended surface elements are referred to as fins. Thus, heat is conducted through the fin and convected (and/or radiated) from the fin (through the surface area) to the surrounding fluid, or vice versa, depending on whether the fin is being cooled or heated. The present research involves a numerical investigation of heat transfer characteristic of different shape finned tubes. The goal of this research is to develop strategies for finding the optimal pin-fin geometry to use in external finned tubes. To reach this goal, a computational modeling of heat transfer phenomena has been done using commercial CFD software and through it the convective and conductive heat transfers in the model has been examined. As pressure loss is one of the important parameters in the systems, in addition to the thermal study, effect of air flow bypass characteristics and flow field has been investigated too.

1.1.1. Extended Surface Heat Exchanger

Heat exchangers, on the basis of constructional details, can be classified into tubular, plate-type, extended surface and regenerative type heat exchangers. The surface area density of these heat exchangers is usually less than 700m2/m3. One of the most common methods to increase the surface area and compactness is to have extended surface (fins) with an appropriate fin density (fin frequency, fins/m) as per the requirement.

1.1.2. Plate-Fin Heat Exchangers

this type of extended surface heat exchanger has corrugated fins mostly of triangular or rectangular cross-sections sandwiched between the parallel plates Plate-fins are categorized as: (1) plain i.e. uncut and straight fins, such as plain triangular and rectangular fins, (2) plain but wavy fins, and (3) interrupted fins such as offset strip, louvered fins, perforated fins etc. shows some of the most commonly used fins in parallel plate heat exchanger.

1.1.3. Tube-Fin Heat Exchangers

These heat exchangers may further be classified as (a) conventional and (b) specialized tube-fin exchangers. Tube-fin exchangers are employed when one fluid stream is at a high pressure and/or has a significantly higher heat transfer coefficient than that of the other fluid stream. In a conventional tube-fin heat exchanger, the transfer of heat takes place by conduction through the tube surface. In a specialized tube-fin exchanger

Depending on the fin type, tube-fin heat exchangers are further classified as (a) individually finned tube, (b) continuously finned tube and (c) longitudinally finned heat exchangers. Density up to 5900m2/m3.

1.2. Passive Heat Transfer Enhancement Techniques

1.2.1. Coating of the Surfaces

Condensation occurs on the surface whose temperature is less than the vapor saturation temperature. Drop wise condensation yields a high heat transfer coefficient but it cannot be sustained permanently. A porous coating on the base surface is an effective enhancement method for film condensation. Condensate drainage is assisted by capillary flow within the porous coating, resulting in a thinning of the condensate film thickness [2].

1.2.2. Rough Surfaces

Surfaces may be made rough by machining or restructuring the base surface or by placing some “roughness” adjacent to the surface e.g. a wire coil insert. For single phase flow, mixing in the boundary layer is promoted near the surface rather than to increase the heat transfer surface area.

1.2.3. Extended Surface

It is a most common approach to enhance the heat transfer by using the extended surfaces. A plain fin may increase the surface area but a special shape extended surface may increase heat transfer coefficient in addition to the area of heat exchanger. Externally finned tube and internally finned tube are the examples of extended surfaces for liquids.

1.2.4. Displaced Inserts

These are the devices inserted into the flow energy transport at the heated surface indirectly. The displaced inserts mix the main flow in addition to that in the wall region. These devices periodically mix the gross flow struck but not affecting the main flow significantly.

1.2.5. Swirl Flow Devices

These devices include a number of geometrical arrangements or tube inserts for forced flow that create rotating or secondary flow. Full length twisted tape inserts or inlet vortex generator and axial coil inserts with a screw type winding are some examples of swirl flow devices.

1.3. Vortex Generator

One of the most important passive techniques to augment the heat transfer is the use of vortex generators. Transverse vortex generators produce vortices, whose axis is transverse to the main flow direction, whereas, the longitudinal vortex generators generate vortices whose axis is parallel to the main flow direction. It has been found that longitudinal vortex generators are more suitable than the transverse vortex generators when the heat transfer augmentation with pressure drop is an important consideration. The longitudinal vortices behind a slender aerodynamic object have been investigated for many years. Longitudinal vortices are found to persist for more than 100 protrusion heights downstream [3]. A vortex generator is called a wing when its span is attached to the surface and is known as a winglet when its chord is attached to the surface. Longitudinal vortex generators may have any of the four basic shapes i.e. delta wing, rectangular wing, delta winglet and rectangular winglet. The aspect ratio 'Λ' of a longitudinal vortex generator is the ratio of the square of the span 'b' and the area of the vortex generator 's' i.e. b2/s. The aspect ratio of vortex generator is an important criterion to compare the performance of the different shapes.ss Figure 1 shows the orientation of a winglet pair both in common flow-up and common flow-down configuration.

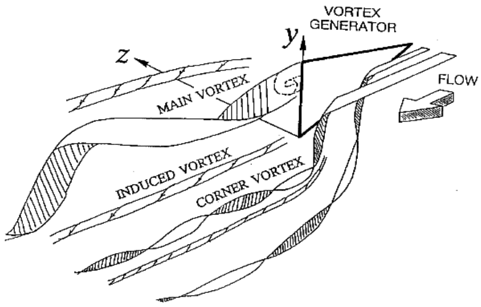


Fig1.1. *Vortex systems behind a delta winglet*

Active vortex generation techniques are associated with control over heat transfer enhancement and pressure drop. When heat transfer augmentation is required, vortices are introduced at the expense of the power to produce vortices along with the added pressure drop. During normal operation, vortex generation is stopped. A number of ways to achieve this control are available; yet very little work is directed at active vortex methods. The use of an injected transverse jet is proved to be an effective active method to produce stream wise vortices. The jets injected are typically circular and are injected with particular pitch and skew angles with respect to the main flow as shown in Figure2. Vortices can be generated for a wide range of jet skew angle and studies show that a jet injected with a pitch of 45° and zero skew angle introduces two counter-rotating vortices with common out-flow.

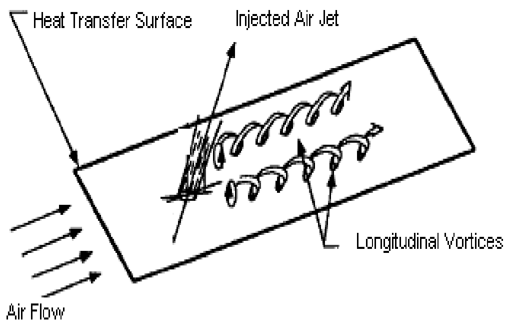


Fig1.2. *actively generated longitudinal vortices*

These normal velocities introduce stream wise vortices and the resulting secondary flow could take the form of a longitudinal vortex. Thus EHD is an active vortex induced heat transfer enhancement technique. Acoustic excitation is a somewhat different way to actively generate a secondary flow. The secondary flow may take the form of longitudinal vortices; however its manifestation is highly dependent on the geometry and flow conditions.

1.4. Available Boundary Types

The boundary types available in FLUENT are classified as follows:

- Flow inlet and exit boundaries: pressure inlet, velocity inlet, mass flow inlet, and Inlet vent, intake fan, pressure outlet, pressure far-field, outflow, outlet vent, and Exhaust fan.
- Wall, repeating, and pole boundaries: wall, symmetry, periodic, and axis.
- Internal cell zones: fluid, and solid (porous is a type of fluid zone).
- Internal face boundaries: fan, radiator, porous jump, wall, and interior.

The internal face boundary conditions are defined on cell faces, which mean that they do not have a finite thickness and they provide a means of introducing a step change in flow properties. These boundary conditions are used to implement physical models representing fans, thin porous membranes, and radiators. The interior" type of internal face zone does not require any input.

2. LITERATURE REVIEW

This section presents a brief look at the present work that has been conducted prior to the writing of this report. This literature review includes a discussion of current state-of-the-art issues and optimization techniques involved with thermal management in compact heat exchangers and e-cooling. Several researchers have considered the pin fins as heat transfer elements. A comprehensive theoretical and experimental study was carried out by Tsutsui and Igarashi [4] on the thermal performance of a pin-fin heat sink. The study concluded that for given fin spacing, thermal performance of a fin array heat sink is only a weak function of fin diameter, and it is improved when fin length is increased.

A large number of recent investigations have undertaken to study the fundamentals of micro channel flow as well as to compare the flow and heat transfer characteristics of micro channels with conventional channels. A comprehensive review of these investigations conducted over the past decade is presented in this chapter. Studies on micro channel flows in the past decade are categorized in to various topics such as temperature, heat transfer in micro channels, Nusselt number, heat flux, comparison with flow in conventional channels, investigation of single phase and two-phase flows in micro channels, mini channels and small tubes, gas flow in micro channels, analytical studies on micro channel flows and design and testing of micro channel heat sinks for electronics cooling.

Hung et al.[8] performed numerical simulations to investigate convective-conductive heat transfer due to a laminar boundary layer flow of air over a two dimensional array of rectangular chip blocks which represent the finite heat sources.

Masud et al. [9] tried to validate the CFD package FLUENT with the experimental data obtained by then earlier. Here they have taken a heated chip with temperature 353 K and the air inlet velocity at temperature 293 K. The inlet velocities were varied from 1 m/s to 7 m/s. various turbulence models have been tested, and the effect of the channel inlet flow on the heat transfer rate has been determined by considering both a uniform and fully-developed condition. Cheng et al. [11] numerically investigated the fluid flow and heat transfer characteristics of mixed convection in three-dimensional rectangular channel with four heat sources. The Simple solver was applied to deal with the coupling between pressure and velocity; and new high-order stability guaranteed second-order difference scheme was adopted to discredit the convection term. They studied the influence of four parameters: Richardson number, heat source distribution, and channel height and inclination angle. They analyzed the numerical results from the viewpoint of the field synergy principle, which says that the enhanced convective heat transfer is related not only to the velocity field and temperature field, but also to the synergy between them.

Kumara et al. [12] investigated the complex unsteady flow through and around a channel in the presence of an obstruction at the entry is studied by solving directly the unsteady Navier-Stokes equations. They considered the Reynolds number of 4000, as experimental results is available for comparison. The computed results are in close agreement with experiments. The computations help with better understanding of the phenomenon of reverse flow and fluid pumping.

One of the most referenced works is that of C. L. Chapman and Seri Lee paper [15] they carried out comparative thermal tests using aluminum heat sinks made with extruded fin, cross-cut rectangular pins and elliptical shaped pins in low air flow environments. They developed an elliptical pin fin heat sink with specific design parameters, maintaining large exposed surface area for heat transfer and minimizing vortex flow by incorporating an airfoil design. The approach taken in the paper was to compare this elliptical shaped heat sink with a conventional extruded fin heat sink of equal volume. They used thermal resistance and amount of flow bypass terms to measure the effects of different thermal conductivity, flow characteristics and pressure drop on heat sink performance.

The other work which is studied for current work is a master thesis [16]. This thesis presents the results of a combined numerical and experimental study of heat transfer and pressure drop behavior in a CHE designed with drop-shaped pin fins. A numerical study using ANSYS was first conducted to select the optimum pin shape and configuration for the CHE. This was followed by an experimental study to validate the numerical model. The results indicate that the drop shaped pin fins yield a considerable improvement in heat transfer compared to circular pin fins for the same pressure drop To compare the performance of different types of finned-tubes, (Kroger, 1986 [18] derived a method to present the heat transfer and pressure drop characteristics of finned tube bundles. Another correlation for finned-tubes was empirically derived by Idem et al. (1987) [19]. It could be used to predict the performance characteristics of untested, geometrically similar heat exchangers.

3. METHODOLOGY

A two dimensional circular bluff body is Modeling and meshes generation is being done on Gambit 2.4.6 and simulations are being done on FLUENT (6.3.26). Fluid flow and heat transfer for turbulence & laminar flow are simulated and results calculated using two turbulence models (k-Є, k-ω SST), with unsteady state solver. The bluff body was used to discritize into a number of elements defining the flow parameter and the boundary condition.2-d bluff body case is used as reference and to validate the value of coefficient of drag (c_d).For the model of 2-d semi- circle of different make is made using suitable software. A mesh generating software gambit 2.4.6 is used to discritize into a no. of element after defining the flow parameters and boundary conditions. The 2-d model is then exported to CFD software Fluent (6.3.26). The analysis of the bluff body is performed using suitable turbulence model. The results. The main difficulty encountered in the solution of incompressible flow is the non availability of explicit equation for pressure. This difficulty associated with the determinant of the pressure field can be resolved in the stream function vortices approach obtained by CFD analysis is validated by experimental studies.

3.1. Geometry

Based on control volume method, 2-D analysis of fluid flow and heat transfer for the semi-circular enclosure is done on fluent software. The geometry for the same done using preprocessor gambit.

3.2. Gambit Part

The flow inside a two-dimensional channel is characterized by a complex flow field, which is affected by the blockage and recirculation zones. The geometry and the mesh were generated using the commercial software GAMBIT and the solutions of the governing equations presented in the previous section were achieved using the commercial CFD software FLUENT 6.3.26, which uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically

3.3. The Governing Equations and the Corresponding Boundary Conditions can be Expressed as

Generally**,** when the convective heat transfer coefficient is small, (a condition commonly encountered when the surrounding fluid is a gas) the rate of heat transfer can be increased appreciably by installing extended surface or fin on the surface. The extended surfaces are classified as straight fins, annular or circumferential fins etc. Here we will focus on pin fins which provide heat transfer augmentation via the repeated growth of laminar boundary layers, followed by dissipation in the wake regions. To show the essential features of fin behavior and introducing the basic definitions, we start with the analysis of a straight fin protruding from a wall. For all flows FLUENT solves conservation equations for mass and momentum. For flows involving heat transfer or compressibility, an additional equation for energy conservation is solved. For flows involving species mixing or reactions, a species conservation equation is solved or, if the non-premixed combustion model is used, conservation equations for the mixture fraction and its variance are solved. Additional transport equations are also solved when the flow is turbulent. The fin with one dimensional characteristic. The temperature of the fin's root is kept constant at T0. The fin body is cooled or heated by ambient air, its temperature is assumed to be $T\infty$. The convective heat transfer coefficient, h is supposed to be uniform along the whole length of the fin. Through this assumption is not very accurate, we use it to formulate the fin behavior.

2-Dimensional Equations:

Continuity Equation:

$$\frac{\partial}{\partial x}(\rho u) + \frac{\partial}{\partial y}(\rho v) = 0$$

Momentum Equation in x-direction:

$$\frac{\partial}{\partial x}(\rho uu) + \frac{\partial}{\partial x}(\rho vu) = \frac{-\partial p}{\partial x} + [\frac{\partial}{\partial x}\left(\mu \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial u}{\partial y}\right)$$

Momentum Equation in y-direction:

$$\frac{\partial}{\partial x}(\rho uv) + \frac{\partial}{\partial y}(\rho vv) = \frac{-\partial p}{\partial y} + [\frac{\partial}{\partial x}\left(\mu \frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial v}{\partial y}\right)]$$

Energy Equation:

Heat Energy Convected in X-direction-

Energy influx, $(E_{conv.})x$ = Mass×Specific heat ×Temperature

$= (\rho\ udy\ ) \times c_p \times t$

Energy efflux, $(E_{conv.})_{x+dx} = \left[\rho\left(u + \frac{\partial u}{\partial x}dx\right)dy\right] c_p \left(t + \frac{\partial t}{\partial x}dx\right)]$

Neglecting product of small quantities

$(\boldsymbol{E_{conv.}})\ x{+}dx = \rho c_p$.dy [u t + u$\frac{\partial u}{\partial x}dx + t\frac{\partial u}{\partial x}dx$]

Net energy convected in X- direction,

D$(E_{conv.})$x = ($E_{conv.})x - (E_{conv.})$ x+dx

$= (\rho\ udy)c_p t - \left[\rho c_p.\text{ dy [u t } + \text{ u}\frac{\partial u}{\partial x}dx + t\frac{\partial u}{\partial x}dx\ ]\right]$

$= -\ \rho c_p \left[u\frac{\partial t}{\partial x}dx + t\frac{\partial u}{\partial x}dx\right]$ dy

d $(E_{conv.})$x $= \ \rho c_p \left[u\frac{\partial t}{\partial x}dx + t\frac{\partial u}{\partial x}\right]$ dx.dy

Heat Energy Convected in Y- direction-

Energy influx, $(E_{conv.})$**y = (**$\rho v dx) \ c_p$.t

Energy efflux $(E_{conv.})$y+dy $= \left[\rho\left(u + \frac{\partial v}{\partial y}dy\right)dx\right].c_p \left(t + \frac{\partial t}{\partial y}dy\right)$

Net energy convected in Y- direction,

d $(E_{conv.})$y = $(E_{conv.})$y – $(E_{conv.})$ y+ dy

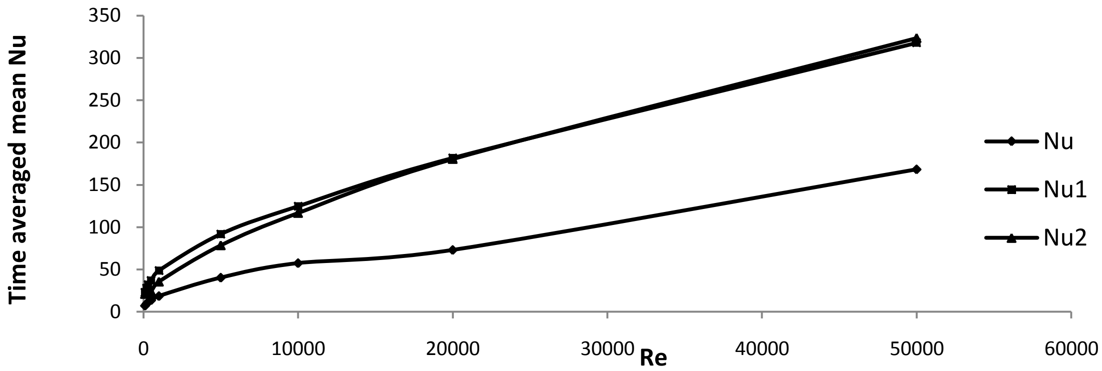
$= -\ \rho c_p \left[v\frac{\partial t}{\partial y} + \ t\frac{\partial v}{\partial y}\right]$ dx dy

3.4. Boundary Conditions

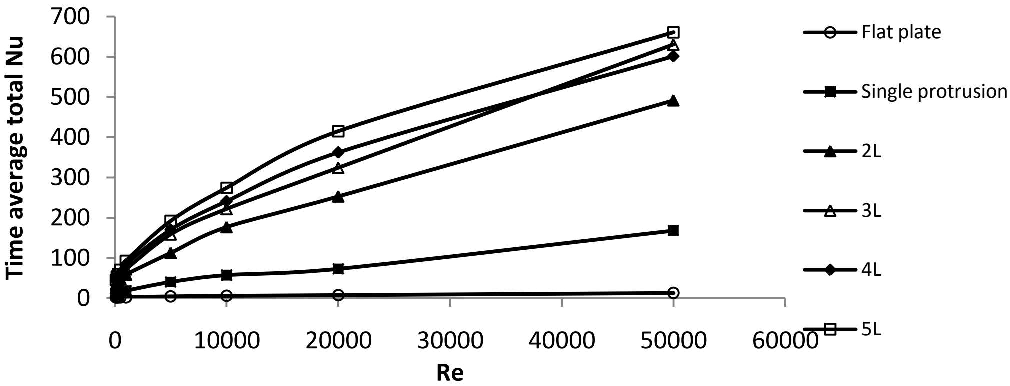
For a steady, fully developed laminar flow, $\frac{\partial u}{\partial x}$ = 0, v = 0. If the whole unit cell is taken as a unite domain. Boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your FLUENT simulations and it is important that they are specified appropriately.

4. RESULTS AND DISCUSSION

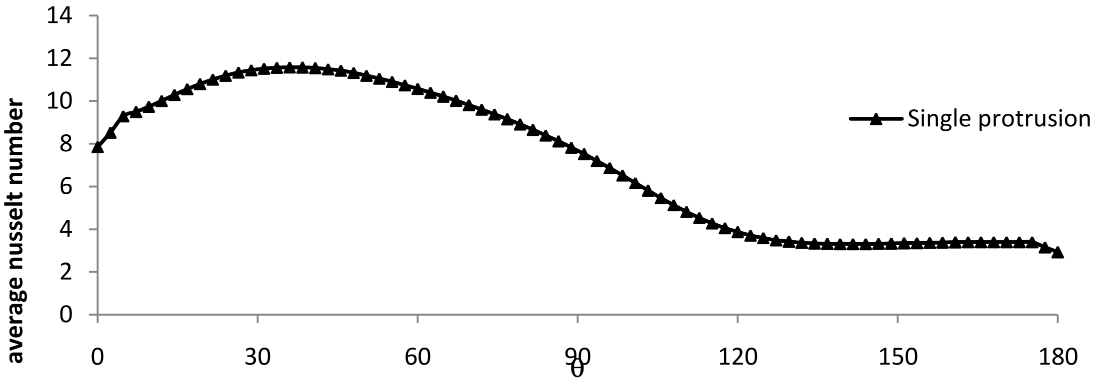
The work presented in the present report deals with the numerical investigation of various aspects of single-phase convective heat transfer enhancement in a two dimensional semi-circular protrusion on fin surface at a constant heat flux (CHF) condition. The flow field as well as heat transfer has been numerically solved using the commercial CFD package FLUENT-6.3.26. By applying the conjugate heat transfer boundary conditions, numerical simulations close to the realistic working conditions were performed. Two similar geometric comparison criteria were applied so that the conclusions derived from the numerical computations are valid for various possible geometric parameters under the constant heat flux and boundary conditions. The working fluid considered here is air. The unsteady, incompressible, viscous flow under CHF condition is studied which is governed by continuity, Navier-Stokes and energy equations. Fluid flow and heat transfer for results are presented for laminar and turbulence with 0.5% intensity. Turbulence model k-ω SST was used to study the effect of turbulence. Various contours of isotherms and vorticity are also presented for the better understanding of the flow field and temperature distribution around the protrusions.



(a)

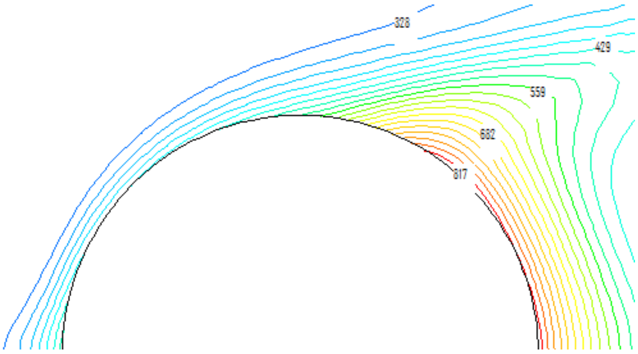


(b)

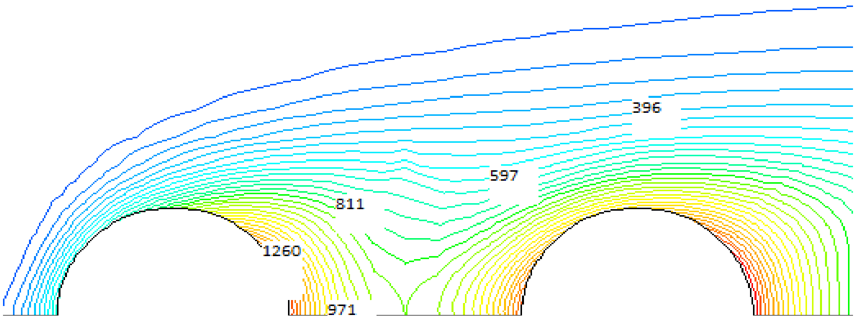


(c)

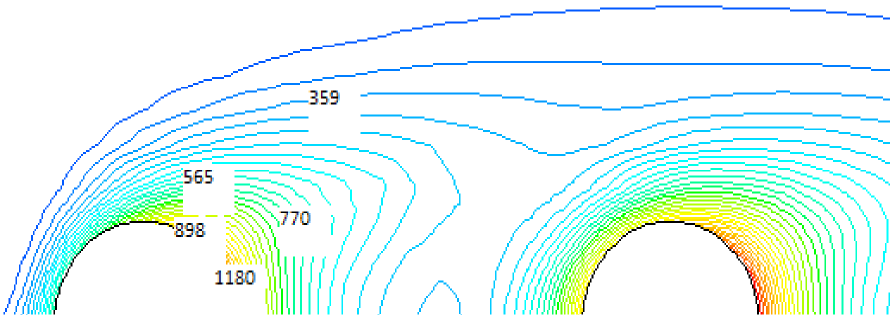
Fig4.1. *(a) time averaged Nusselt number v/s Reynolds number for 4L and single protrusion, (b) time averaged Nusselt number v/s Reynolds number for flat plate, single protrusion, and two protrusion with gaps of 2L, 3L, 4L and 5L (c): variation of Nusselt number with angular position at Re = 100*



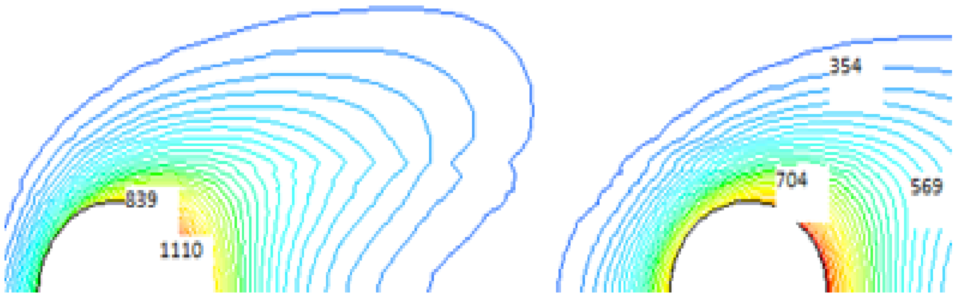
(a) Single



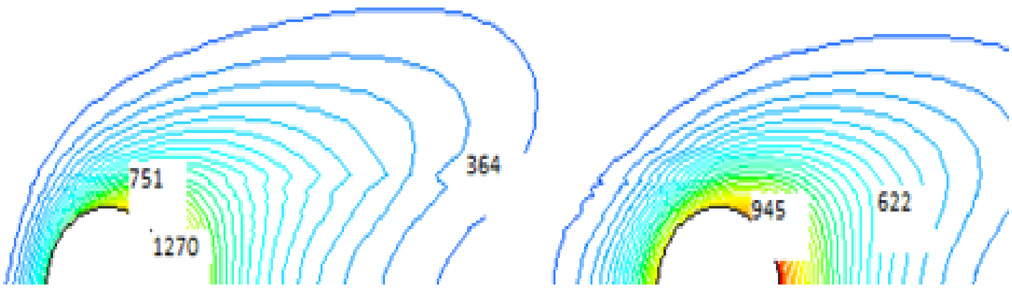
(b) 2L



(c) 3L



(d) 4L



(e) 5L

Fig4.2. *Isotherms (in K) for (a) single and two protrusions with different gap (b) 2L (c) 3L (d) 4L (e) 5L*

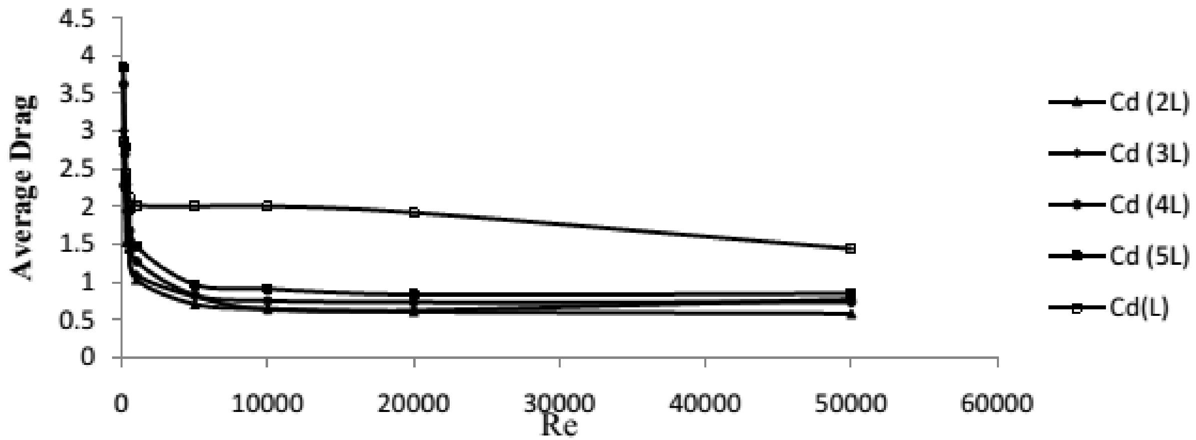


Fig4.3. *Average drag v/s Reynolds numb*

Table: *Data of Nu & Cd for all length of protrusion*

Re	**Single**		**Plate**		**2L**			**3L**			**4L**			**5L**		
	Nu	Cd	Nu	Cd	Nu1	Nu2	Cd	Nu1	Nu2	Cd	Nu1	Nu2	Cd	Nu1	Nu2	Cd
100	4.967	3.125	4.864	4.627	2.798	2.69	4.333	3.498	3.597	3.7998	4.3945	4.389	2.897	2.899	2.899	3.895
300	1.4502	2.765	2.356	1.011	2.596	2.33	3.228	2.564	2.845	2.4587	3.0091	3.0014	2.0024	2.899	2.899	3.495
1000	1.854	2.59	2.568	0.112	3.294	2.68	2.2546	3.946	3.495	1.3459	3.1958	2.942	1.0098	3.069	2.598	2.394
5000	3.045	2.354	4.359	0.029	5.354	4.12	0.9458	5.379	4.492	0.89945	6.1978	6.1338	1.314	4.698	3.8954	0.829
10000	3.038	2.158	5.859	0.015	6.799	5.64	0.7684	8.094	8.468	0.97894	11.198	11.596	1.689	5.698	4.798	0.329
20000	4.356	2.367	7.689	0.009	4.899	3.44	0.6254	5.396	3.779	0.55024	5.448	3.689	0.9489	6.49	5.478	0.619
30000	5.256	1.987	9.158	0.008	6.132	6.89	0.5218	7.197	6.019	0.434	6.934	6.329	8.190	8.19	8.039	0.289

5. CONCLUSION

The work presented in the present report deals with the numerical investigation of various aspects of single-phase convective heat transfer enhancement in a two dimensional semi-circular protrusion on fin surface at a constant heat flux (CHF) condition. By applying the conjugate heat transfer boundary conditions, numerical simulations close to the realistic working conditions were performed. Pressure, temperature and velocity profile were drawn for different inlet velocities for a protrusion arrangement, considering the flow to be laminar and turbulence. The working fluid considered here is air. The unsteady, incompressible, viscous flow under CHF condition is studied which is governed by continuity, Navier-Stokes and energy equations. Fluid flow and heat transfer for results are presented for laminar and turbulence with 0.5% intensity. Turbulence model k-ω SST was used to study the effect of turbulence

The present computations validates the results obtained by Kubacki within the root mean square error of 0.73% for pressure coefficient while for averaged Nusselt number the root mean square error is found to be 1.58%. Separation over the protrusion starts before 90^o and close to it. Up to the separation point the flow is laminar. The flow reattaches at around 120^o. The detachment occurs due to the formation of free shear layer just after the separation point and transverse vortices forms. The heat transfer is more from both of the protrusion individually in comparison to that if a single protrusion is taken. The Nusselt number increases with Reynolds number almost linearly in the subcritical range (300<Re< 50000). From the plots of figure 4.1 it has been observed that heat transfer increases with increase in the Reynolds number while the heat transfer capability for fin with two or more protrusions is more in comparison to that of a flat plate or single protrusion of the same plan form area. Discussion of figure 4.3concludes that the plan form area of a single protrusion case is less than that of all the cases of two protrusions although the frontal or the projected area of both the cases is same thus single protrusion is subjected to more drag as compared to the all cases of two protrusions. The drag is mainly due to the pressure as in these cases pressure drag dominates over the shear drag. From the contours of isotherms shows in figure 4.2.it is revealed that, qualitatively also, it could be concluded that protrusion with the gaps of 4L is the optimized position for the augmentation of heat transfer using protrusions over the fins.

REFERENCES

[1] N.Sahiti, F.Durst, A.Dewan, 2005, Heat transfer enhancement by pin elements, Int. J. of Heat Mass Transfer, Vol. 48, pp.4738-4747.

[2] N. Sahiti, F. Durst, and A.Dewan,2006, Strategy for Selecting of Elements of Heat Transfer Enhancement, Int. J. of Heat and Mass Transfer,Vol.49,pp.3392-3400.

[3] F.P.Incropera, 1999, Liquid cooling electronic devices by single-phase convection, John Wiley & Sons, New York.

[4] T. Tsutsui and T. Igarashi, 2006, The heat transfer enhancement of a circular cylinder, J. Heat of Transfer Volume128, pp. 226-233.

[5] V.P.Isachenko, V. Osipova, A. Sukomel, 1969, Heat Transfer, Mir, Moscow.

[6] N. Sahiti, A. Lemouedda, D. Stojkovic, F. Durst and E. Franz, 2007, Performance comparison of pin fin in-duct flow arrays with various pin cross-sections, Int. J. of Applied Thermal Engineering, Vol. 27, pp. 111-119.

[7] Megerlin et al., 1974. F.E. Megerlin, R.W. Murphy and A.E.Bergles Augmentation of heat transfer in tubes by use of mesh and brush inserts. Heat Transfer, 145-151 (1974).

[8] Hung, T.C., Wangi S.K. and Tsai, F.P., (1997), “Simulation of passively

[9] Enhanced conjugate heat transfer across an array of volumetric heat sources”, International Journal for Numerical Methods in Engineering, Vol. 13, pp. 855-866.

[10] Masud, Bebnia, Wataru, Nakayama, and Jeffrey, Wan. (1998), “CFD simulations of heat transfer from a heated module in an air Stream: comparison with experiments and parametric study”, IEEE Intersociety Conference on Thermal Phenomena, Vol. 54, pp.539-544

[11] Dhiman, A.K., Chhabra, R.P. and Eswaran, V., (2005), “Flow and heat transfer across a confined square cylinder in the steady flow regime: effect of peclet number”, International Journal of Heat and Mass Transfer, Vol. 48, pp. 4598-4614.

[12] Cheng, Y.P., Lee, T.S. and Low, H.T., (2006), “Numerical analysis of mixed convection in three-dimensional rectangular channel with flush mounted heat sources based on field synergy principle”, International Journal for Numerical Methods in Fluids, Vol. 52, pp.987-1003.

[13] Kumara, K.S., Tulapurkaraa, E.G., Biswasb, G. and Gowdac, B.H.L., (2005), “Reverse flow in channel with obstruction at entry”, Fluid Dynamic Research, Vol. 37, pp. 387-398.

[14] Roy, A. and Bandyopadhyay, G., (2004), “Numerical investigation of confined flow pasta square cylinder placed in a channel”, Journal of Institution of Engineers, Vol. 85, pp.60-63.

[15] P. fund, D., Rector, D., Shekarriz, A., Popescu, A. and Welty, J., (2000), “Pressure drop measurements in a micro channel”, International Journal of Heat and Mass transfer, Vol.46, pp. 1496-1507.

[16] Christopher L. Chapman, Seri Lee, Bill L. Schmidt, Thermal Performance of an Elliptical Pin Fin Heat Sink, Tenth IEEE Semi-TherMP , Feb. 2000.

[17] Jihed Boulares , Numerical and Experimental Study of The Performance of a Drop-Shaped Pin Fin Heat Exchanger, Master Thesis, NAVAL POSTGRADUATESCHOOL Monterey, California, 2003.

[18] B. Moshfegh, R. Nyiredy, Comparing RANS Models for Flow and Thermal Analysis of Pin Fin Heat Sinks, 15th Australasian Fluid Mechanics Conference, the University of Sydney, Sydney, Australia 13-17 December 2004.

[19] Kröger,D.G, Performance characteristics of industrial finned tubes presented in dimensional form, Int. Journal of Mass and Heat Transfer, vol. 29, no. 8, pp. 1119-1125, 1986.

[20] Idem, S.A., Jung, C., Gonzalez, G.J. and Goldschmidt, V.W., Performance of air-to water copper finned-tube heat exchangers at moderately low air-side Reynolds Numbers, including effects of baffles, Int. Journal of Mass and Heat Transfer, vol. 30, no.8,pp.1733-1741, 1987.

[21] VanFossen, G.J., Heat- Transfer Coefficient for staggered Arrays of Short Pin Fins, Journal of Engineering for Power, vol. 104, pp. 268-274, 1982.

[22] Metzger, D. E., Berry, R. A., Bronson, J. P., Developing Heat Transfer in Rectangular Ducts with Staggered Arrays of Short Pin Fins, Journal of Heat Transfer, vol. 104, pp.700-706, 1982.

[23] Chyu,M. K., Goldstein, R. J., Influence of an Array of Wall-Mounted Cylinders on the Mass Transfer from a Flat Surface., International Journal of Heat and Mass Transfer, vol. 34, pp.2175-2186, 1991.

[24] Lee, P. and Garimella, S.V., (2003), "Experimental investigation of heat transfer in micro channels", Paper No.HT2003-47293, *ASME Proceedings of HT2003 Summer Heat Transfer Conference*, Vol. 37.

[25] Kubacki.S Augmentation of horizontal in-tube condensation by means of twisted-type inserts and internally finned tubes, Journal of Heat Transfer, vol.100,February, pp.17-24, 1978.