Optimizing Protrusion Position on a Flat Fin Transferring Heat

S. K. Dhiman*
(Department of Mechanical Engineering
Birla Institute of Technology, Mesra, Ranchi, India

Abstract
Computations were conducted in FLUENT over squared protrusions on a flat fin transferring heat under constant-heat flux condition for 100≤Re≤30000 using conjugate heat transfer scheme. Quantative and qualitative comparison of time averaged Nusselt number were conducted for a flat plate without protrusion, with single protrusion and with two protrusions having gaps of 2L, 3L, 4L and 5L. It was observed that 4L is the optimum gap between the two protrusions for the considered range of Re..

Keywords-*squared peortrusion, computations, heat transfer, constant-heat-flux, optimising gap.*

I. INTRODUCTION

The cross-flow and heat transfer from the surfaces of the bluff bodies represents numerous applications in case of heat exchangers. Such bodies have various shapes, most common are: cylindrical or square etc., over which flows structures are complexed and associated with various possibilities of augmentation of heat transfer. In recent years, considerable interest has been shown in studying the flow past a circular and square sections oriented normal to the direction of flow. Investigation of flow and heat transfer parameters over square and rectangular sections have been recently and over the years, reported. D.D. Luo et al. [1] has reported two general turbulence models, the standard k– ε model and the RSM, to predict the forced convection heat transfer phenomenon of a fully developed turbulent flow, over square ribs at 22,000<Re<94,000, uniformly spaced with the pitch-to-height ratio of p/e=4, a height-to-hydraulic-diameter ratio of e/D= 0.25, and a width-to-height ratio of w/e= 2. The work of Davalath et al. [2] was related to a two-dimensional, conjugate heat transfer problem for laminar flow over an array of three heated rectangular blocks in a duct. The spacing between two adjacent obstacles was varied from one to two times the obstacle length. Patankar et al. [3] and Kim et al. [4] conducted extensive numerical analyses of the heat transfer in the fully developed region of a duct containing uniformly spaced heated blocks under laminar flow conditions. Experimental studies have also been carried out on forced convection in channels with surface-mounted rectangular obstacles [5-9]. Chatterjee and Amiroudine [10] has reported a two-dimensional numerical simulation carried out to understand the effects of thermal buoyancy and Prandtl number on flow characteristics and mixed convection heat transfer over two equal isothermal square cylinders placed in a tandem arrangement within a channel at low Reynolds numbers. The unsteady numerical simulations were performed with a finite volume code based on the PISO algorithm in a collocated grid system. The streamlines, vortex structures and isotherm patterns were presented and discussed. In addition, the overall drag and lift coefficients, recirculation length and average Nusselt numbers were determined to elucidate the role of Reynolds, Prandtl and Richardson numbers on flow and heat transfer.

Fins are generally protruded from the surface of heated body to dissipate heat from it when air is used as the cooling fluid. Putting the fins increases the surface area to be exposed to the cooling fluid. In case of heat exchangers the cooling fluid is generally oil or water or any liquid which has good heat transfer properties. But with the vision that the availability of these liquid is not economic always, the present study deals with the cooling to be done using air as the cooling fluid. Due to poor heat transfer properties of air the surface area required to be enhanced in order to dissipate equivalent amount of heat compared to the exchangers in which liquid is used as the cooling fluid. In addition to this the flow of air must also be at least moderately high for the equivalent dissipation of heat. The present work deals with some two dimensional computational study of flow field and heat transfer characteristics over protrusions of square shape if it is provided on the fins of inner tube of a shell and tube heat exchanger. The study has been done considering constant heat flux (CHF) condition at the surface of protrusions subjected to flow with Reynolds number ranging from 100 to 30,000. The present work also deal with the optimization of positions of squared protrusions under constant heat flux condition on the fins that have been put on the outer surface of the inner tube of shell and tube heat exchangers, Fig. 1a & 1b.

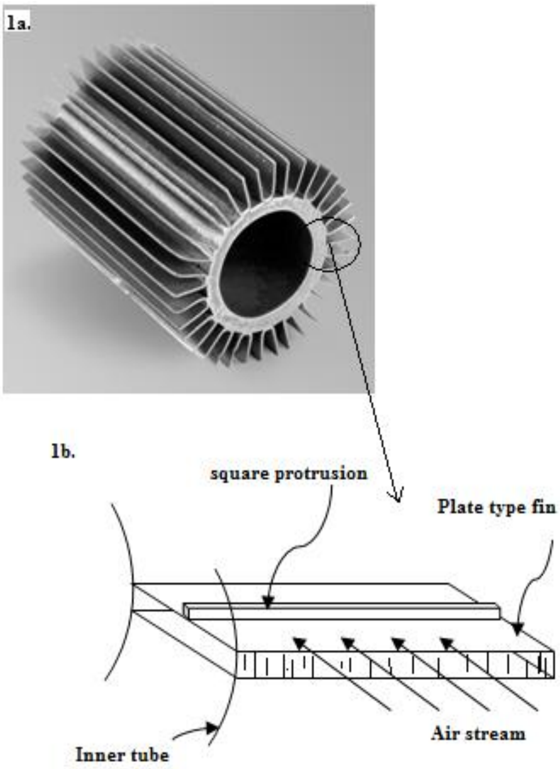

Fig.1: Element of inner tube with fin and protrusion

II. METHOD AND APPROACH

Local Nusselt number distribution along the heated squared fin surface was computationally investigated, which was validated corresponding to numerical results conducted by D.D.Luo et al. [1] at Re = 94000. Comparison of time averaged Nusselt number for a flat plate; single protrusion; two protrusions with the gaps of 2L; 3L; 4L and 5L, based on all Reynolds number under consideration, have been presented. Fig. 2 shows closed view of structured grid at two squared protrusions placed at a gap of 2L. In the similar fashion gaps of 3L; 4L and 5L were also investigated. The constant heat flux of 1000W/m2 has been taken in all the case considered. The protrusions are made of copper material and its 'width to height ratio' has been taken as 1 (given as width=1mm and height=1mm). The flow field as well as heat transfer have 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. For low Reynolds number (Re<300), laminar condition is considered while for higher Reynolds number 0.5% turbulence intensity have been taken and have been solved by k-ω turbulence with unsteady-state solver. The numerical approaches were based on the finite-volume technique. A second-order upwind scheme was applied in the calculation and a very fine mesh density was arranged in the regions near the wall boundaries, shown in Fig.2. The SIMPLEC algorithm was adopted to handle the pressure–velocity coupling in the calculation. 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.

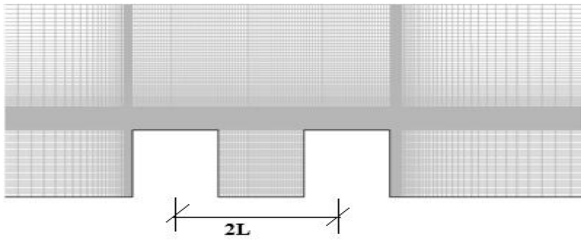

Fig.2: Closed view of structured grid at two squared protrusions placed at a gap of 2L

The governing equations were the unsteady - state continuity, the time-averaged Navier-Stokes, and the energy equations for turbulent flow, i.e.,

$$\frac{\partial}{\partial x_j}(\rho U_j) = 0$$

$$\frac{\partial U_j}{\partial \tau} + \frac{\partial}{\partial x_j}(\rho U_j U_i) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}(-\rho\overline{u_i u_j})$$

$$\frac{\partial T}{\partial \tau} + \frac{\partial}{\partial x_j}(\rho U_j T) = \frac{\partial}{\partial x_j}\left(\frac{\mu}{Pr}\frac{\partial T}{\partial x_j} - \rho\overline{\mu_j t}\right)$$

where,

$(-\rho\overline{u_i u_j})$ is turbulent shear stress

and $(-\rho\overline{u_j t})$ is turbulent heat flux

In the above governing equations, the two unknown variables, the turbulent shear stresses the turbulent heat fluxes, are required to be modeled. As the walls were the main source of mean vorticity and turbulence, and the solution variables would have large gradients at the near-wall region, where the momentum and other scalar transports occurred most vigorously, the near-wall modeling could affect significantly the accuracy of numerical solutions. Accurate representation of the flow at the near-wall region determined successful predictions of the wall-bounded turbulent flows. In the present study, k-ω turbulence model is used in conjunction with the wall functions in dealing with the wall boundary layer for the predictions.

Grid Structure

The grid structure used in the present work is shown in the Fig. 3a&b. It shows the non-uniform grid structure for the whole of the computational domain (Fig. 3a) and the expanded view near the obstacle is shown in Fig. 3b. It consists of five separate zones with uniform and non-uniform grid distribution having a close clustering of grid points in the regions of large gradients and coarser grid in the region of low gradients. Overall, the grid distribution is uniform with a constant cell size, Δ=0.25b, in an outer region that extends beyond 4 units upstream and downstream of the cylinder in the x-direction. A much smaller grid size δ is clustered in an inner region around the cylinder over a distance of 1.5 units to adequately capture wake-wall interactions in both directions. A fine grid of size δ is also clustered near the upper and lower walls of the domain to capture the wake-wall interactions.

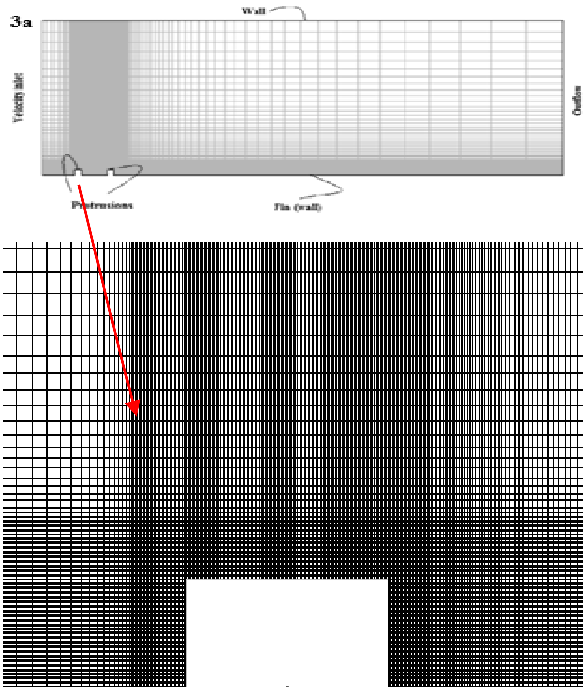

Fig.3b: Closed view of structured grid at single squared protrusion for validation with Davalath et al.[2]

III. RESULTS AND DISCUSSION

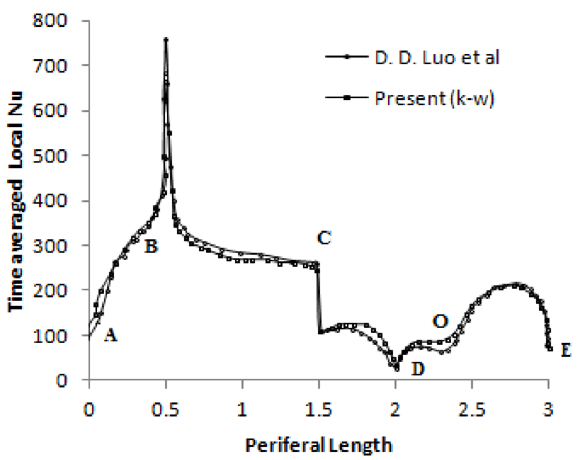

Fig. 5: Validation of present numerical scheme with D.D.Luo et al. [1] at Re=94000 for single protrusion

Referring to Fig. 5, a very close predictions of Nu distributions along the protrusion’s left face (A to B) states that the existence of protrusion reduced the cross- sectioned area of the flow, which led to a higher-pressure region on the upstream side and an acceleration of the core flow toward the rear. The upstream flow impinged on the protrusion’s left surface and formed a clockwise recirculating flow there. As the flow moved down along the rear side, the thermal boundary layer became thicker and thus Nu increased almost linearly at the region (A to B). In addition, a sharp increase of Nu around point B is clearly predicted. Farther downstream, a thermal boundary layer developed on the top face of the protrusion (B to C), and the Nu decreased almost linearly along the top surface. On the right side of the protrusion, due to the sudden expansion of the cross-sectional area, a low -pressure region formed behind the protrusion. The mainstream flow separated at the protrusion’s right upper corner (C). As there is less fluid heated by the protrusion, a sharp decrease of Nu was observed at the separating point (C). An S-shaped distribution of local Nu at the protrusion’s right face (C to D), is predicted by the standard k – ω model, which was rather similar to results of D. D. Luo et al. [1]. In the region near the protrusion’s right lower corner (D), a significant drop of Nu to a very low value is detected. On the smooth surface between the two adjacent protrusions (D to E), because of the effects of vortex and recirculating flow in this region, the distribution of local Nu became rather interesting. At the inter-spacing between the adjacent protrusions (D to E), due to the influence of the interaction of the vortex behind the protrusion with the recirculating flow, a transition process (D to O) occurred before the Nu increased nearly linearly along the bottom surface. It has been found that the transition point (O) coincided very well with the detachment points of the vortex and the recirculating flow.

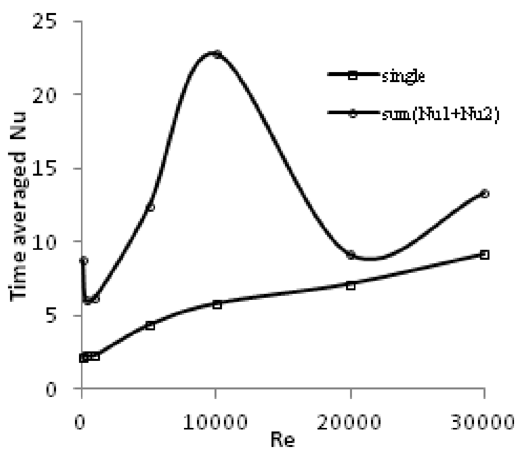

Fig. 6: Graph of Nu and Re at single protrusion and double protrusion at 4L

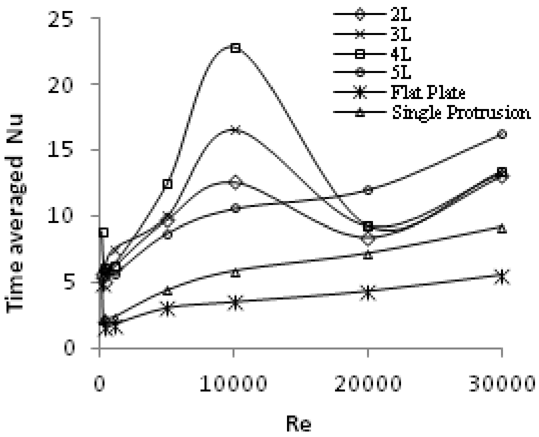

Fig.7: Comparison of Time Averaged total Nu vs Re for all bodies under consideration

Fig. 6 shows the comparison of variation Nusselt number for two protrusions with gap of 4L and single protrusion with Reynolds number. It is observed from the plots that heat transfer is more from both of protrusions together in comparison to that if single protrusion is taken. It has also been observed that Nusselt number increases with Reynolds number almost linearly in subcritical range (300≤Re≤30000). This is due to the fact that with increase in Reynolds numbers the region of free shear layer enhances because of increased length of transformation region as well as length of transition. Fig. 7 compares the time averaged Nu for a flat plate, single protrusion, two protrusion with gaps of 2L, 3L, 4L and 5L based on all 100<Re<30000. For all the cases considered, it has been observed that heat transfer increases with increase in the Re. The plots also confirms that heat transfer capability for fin with two or more protrusions is more in comparison to that of a flat plate or single protrusion of same planform area. It also confirms that providing protrusions over fins enhances the heat transfer and it is maximum when the gaps is 4L. So, putting protrusions over fins is a fruitful technique for augmentation of heat transfer in heat exchangers.

Fig. 8 shows the variation of Nu with the Re for two protrusions at gaps of 2L, 3L, 4L and 5L. Plots apparently leads to conclude that with increase in gap Nu of both the protrusions comes closure to each other and at a gap of 4L, Nu of both the protrusion exhibits same value and plots of both superimposes. Further increase of gaps from 5L or more again enhances space between the plots. This leads to conclude that a gap of 4L is the optimum space between two protrusions at all ranges of Reynolds number.

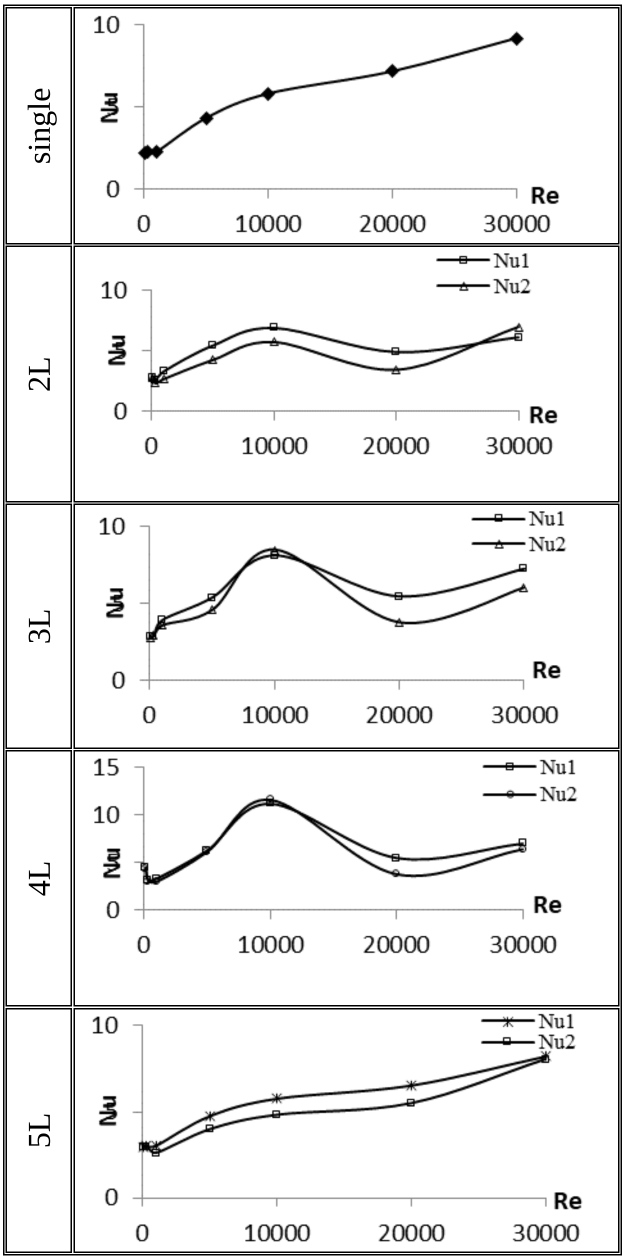

Fig. 8: Nu vs Re for different gaps

IV. CONCLUSION

Computational analysis of flow field and heat transfer over squared protrusions on fins of heat exchanger under Constant Heat Flux (CHF) condition for 100 ≤Re≤ 30000 using conjugate heat transfer scheme, validated at Re = 94000, have been performed with laminar condition for low Re (≤300) and with 0.5% turbulence intensity for higher Re, solved by k-ω turbulence model with unsteady-state solver. Comparison of time averaged Nusselt number for a flat plate without protrusion, with single protrusion and with two protrusions having gaps of 2L, 3L, 4L and 5L based on all considered Re, leads to the conclusion that heat transfer increases with increase in the Re and at a gap of 4L, Nusselt number of both the protrusion exhibits the same value and the plots of both superimposes thus leads to the conclude that a gap of 4L is the optimum space between the two protrusions at all Re under consideration.

REFERENCES

[1] D. D. Luo, C. W. Leung, T. L. Chan and W. O. Wong, "Flow and Forced-Convection Characteristics of Turbulent Flow through Parallel Plates with Periodic Transverse Ribs", Numerical Heat Transfer, PartA, vol. 48, pp.43-58, 2005.

[2] J. Davalath and Y. Bayazitoglu, "Forced Convection Cooling across Rectangular Blocks", J. Heat Transfer, vol.109, pp.321–328, 1987.

[3] S. V. Patankar and R. C. Schmidt, "A Numerical Study of Laminar Forced-Convection across Heated Blocks in Two-Dimensional Ducts", ASME Paper 86-WA/HT-88, 1986.

[4] S. H. Kim and N. K. Anand, "Laminar Heat-Transfer between a Series of Parallel Plates with Surface-Mounted Discrete Heat Sources", J. Electron. Pkging., vol. 117, pp. 52–62, 1995.

[5] J. C. Han and J. S. Park, "Developing Heat Transfer in Rectangular Channels with Rib Turbulators, Int. J. Heat Mass Transfer", vol.31, pp.183–195, 1988.

[6] E. M. Sparrow, J. E. Niethammer and A. Chaboki, "Heat Transfer and Pressure Drop Characteristics of Arrays of Rectangular Modules Encountered in Electronic Equipment", Int. J. Heat Mass Transfer, vol. 25, pp.961–973, 1982.

[7] S. Sridhar, M. Faghri, R. C. Lessmann, and R. Schmidt, "Heat Transfer Behavior including Thermal Wakes in Forced Air Cooling of Arrays of Rectangular Blocks", ASME HTD, vol. 153, pp. 15–25, 1990.

[8] G. L. Lehmann and R. A. Wirtz, "Convection from Surface-Mounted Repeated Ribs in a Channel Flow", ASME Paper 84-WA/HT-88, 1984.

[9] G. L. Lehmann and R. A. Wirtz, "The Effect of Variations in Streamwise Spacing and Length on Convection from Surface-Mounted Rectangular Components", Heat Transfer Electron. Equip., ASME HTD, vol. 48, pp. 39–47, 1985.

[10] D. Chatterjee and S. Amiroudine, "Two-dimensional mixed convection heat transfer from confined tandem square cylinders in cross-flow at low Reynolds numbers", International Communications in Heat and Mass Transfer, vol.37, pp.7-16, 2010.