Flow and Heat Transfer Characteristics of Square Cylinder with Protrusions at Different Channel Confinements

Amit Varakhedkar, Rajendran Senthil kumar

Abstract: The present 2-D numerical study investigates flow and convection heat transfer characteristics of a square cylinder with protrusions by considering electronic cooling applications. The flow confinement effect is observed by varying the blockage ratio (channel height/characteristic length of component) from 2 to 4. A finite volume-based flow solver is used to solve flow and energy equations. The resulting flow and heat transfer characteristics of various geometries studied from pressure coefficient, drag coefficient, Total Pressure Drop, Local Nusselt Number and average Nusselt number for various Reynolds numbers from 75 to 200. The addition of protrusion on rare side of square cylinder did not enhance the heat transfer in a significant rate but it eliminates flow shredding and guarantees the structural stability.

Index Terms: Confinement, Drag Coefficient, Finite Volume Method, Nusselt number, Protrusions

I. INTRODUCTION

Microfluidics is basically the manipulation of fluids at the micron scale implying a profound knowledge of the behaviour of fluid at this scale and the engineering of device with micro geometry and microchannels. Microfluidics has a very vast application in the field of biomechanics and microelectronics, widely used in preparation of microfluidic chip. Microfluidic chips are used to control the flow of fluids in a confined microchannel. The Cooling of microfluidic chip is one of the biggest challenges faced by electronic industries. The need of cooling has enhanced due to increase of usage, and miniature of these devices. Incropera [1], and Yang and Fu [2] have laid focus on cooling of microelectronic chips by optimizing the geometrical properties of various electronic components for achieving effective and efficient heat dissipation. Dawalath and Bayazitoglu [3] has focussed on forced convection cooling of rectangular geometry, because it is easy for manufacturing and installation, and the geometry is widely used in manufacture of microelectronic chips. There are various high-end cooling technologies available to enhance heat transfer rate from electronic components, and to reduce the temperature at faster rate using various types of fluids and traditional cooling of components using atmospheric air.

Revised Manuscript Received on April 25, 2019.
Amit Varakhedkar, Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, India.
Rajendran Senthil kumar, Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, India.

This method is cost efficient, and easy to fabricate, also usage of air instead of fluid can prevent damages to electrical component caused by fluid contacting with circuits. Despite not being the most efficient method, its effectiveness can be improved by altering the dimensions, changing orientation and by applying fins on the surface.

II. LITERATURE REVIEW

Okajima [4] studied and reported the behaviour of bluff body when it is subjected to higher flow velocity by estimating large pressure gradient, drag coefficient and vortex shedding phenomena. This wake is responsible for inciting fluctuating forces on the structure and thus devastating the structural integrity. Sohankar et al. [5] and Subhankar et al. [6] have shown effect of geometry on flow disintegration. Sharma and Eswaran [7] have worked on formulating a correlation for Reynolds Number of flow and Surface Nusselt number of square cylinder. It has been observed that the surface averaged Nusselt number for square cylinder increases linearly by increasing Reynolds Number. It was observed the Nusselt number for the front surface is highest, followed by the bottom and top sides with intermediate effects. The rear side was the least contributing of the four sides. In order to increase heat transfer from square geometry, various methods have been applied. Although, aforementioned researches have been done on unconfined region, the effect of drag, pressure drop and heat transfer will differ for a confined domain. Through computation and experimental study, Davies et al [8] has discussed that with increase in blockage ratio, there has been increase in drag parameters and Strouhal Number. According to Sharma and Eswaran [9] Nusselt number due to channel confinement increases with increasing blockage ratio, with the greatest enhancement obtained at the lowest Reynolds number. Various methods have been applied for enhancing heat transfer effectiveness for given bluff body. Corner altercation on square cylinder results in higher heat transfer, with chamfered corner being most efficient Ambreen and Kim [10]. Other methods involve addition of fin geometry to the existing geometry for enhancing heat transfer Kahn et al [11]. Although various fin geometries have been applied for the square cylinders, its effect has not been studied for confined region. This paper focusses on the influence of a linear protrusion on the rear side of square geometry and its heat transferring behavior at different confinements.

III. PROBLEM FORMATION

The 2-D square cylinder with linear protrusion attached to the rear side is considered basically to formulate the problem for analysis (Fig. 1). Square cylinder of side D, with length of protrusion 1.5D and thickness 0.2D are in a computational domain of length 10D, and height of computational domain is 4D, is exposed to free steam of fluid velocity U0 and temperature T0. The computational Domain is made dimensionless by incorporating characteristics diameter D of geometry. The solid walls are moving at free stream velocity of the fluid (U0).

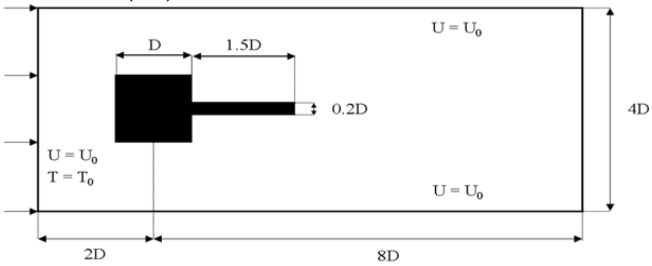


Fig. 1 Schematics of Physical Computational Domain

IV. GOVERNING EQUATIONS, BOUNDARY CONDITIONS, AND SOLUTION PROCEDURE

In this study, the flowing fluid is atmospheric air. Here the air is viscous and incompressible. It has been assumed that the temperature does not vary along Z-direction. Hence, decided to conduct 2-dimensional computational studies on hot bluff body exposed for fluid flow and heat dissipation. The suitable assumptions have been imposed on general governing Navier-Stokes and Energy equations and reduced to the following convenient form to solve.

Continuity Equations:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

X – Momentum Equation:

$$\rho\frac{\partial u}{\partial t} + \rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

Y – Momentum Equation:

$$\rho\frac{\partial v}{\partial t} + \rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

Energy Equation:

$$\frac{\partial T}{\partial t} + u\left(\frac{\partial T}{\partial x}\right) + v\left(\frac{\partial T}{\partial y}\right) = \alpha\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

Exactly at the domain inlet, uniform velocity profile has been preferred. Simultaneously, pressure outlet boundary condition has been imposed on domain exit. The top and bottom walls are moving at free stream velocity (U_0) to resemble the effect of confined fluid on bluff body with protrusion. No slip boundary condition assigned on the walls of bluff body at constant temperature (T_w). The convective terms in the governing equations are discretized using second-order upwind scheme. The SIMPLE algorithm has been used to couple pressure and velocity. Relative convergence criteria of 10^{-6} for the continuity, x- and y-components of velocity, and 10^{-12} for energy are used in this study.

V. GRID INDEPENDENCE

The ANSYS Workbench 18 was used to model the problem formulated in this study. The geometry is having protrusions and sharp corners, hence the unstructured triangular mesh elements has been preferred to mesh which is shown in Fig. 2. The grid convergence study gives appropriate number of mesh elements as 54,677 with deviation of Total Drag (*Cd*)- 1.26% and Nusselt Number (Nu) -0.2% with successive mesh sizes.

Fig. 2 Computational Domain with Unstructured Meshing

VI. RESULTS AND DISCUSSION

A. Validation of Computations:

The flow and heat transfer characteristics have been validated by considering and reproducing numerical results of unconfined square cylinders maintained at constant temperature in literature. It was observed that the present numerical estimation closely matches with literature by Cao et al.[12] and Sharma and Eshwaran [9]. Here the maximum variations are also within the acceptable range.

Table 1. Comparison of Total Drag coefficient of present work with literature.

Reynolds Number	Cao et al. (2012)	Sharma and Eshwaran (2010)	Present Simulation
50	1.81	1.72	1.81
100	1.46	1.39	1.42

Table 2. Comparison of Average Nusselt Number of present work with literature.

Reynolds Number	Sharma and Eshwaran (2010)	Present Simulation
75	3.54	3.64
100	4.02	4.03

B. Streamline, Pressure and Temperature Contours

Some contours have been shown in Fig. 3 and Fig. 4 depicting streamlines, pressure and temperature contours for different Reynolds numbers (75,100,150 and 200), and different blockage ratios (BR = Height of the channel/ Characteristics length) of 2, 3 and 4.

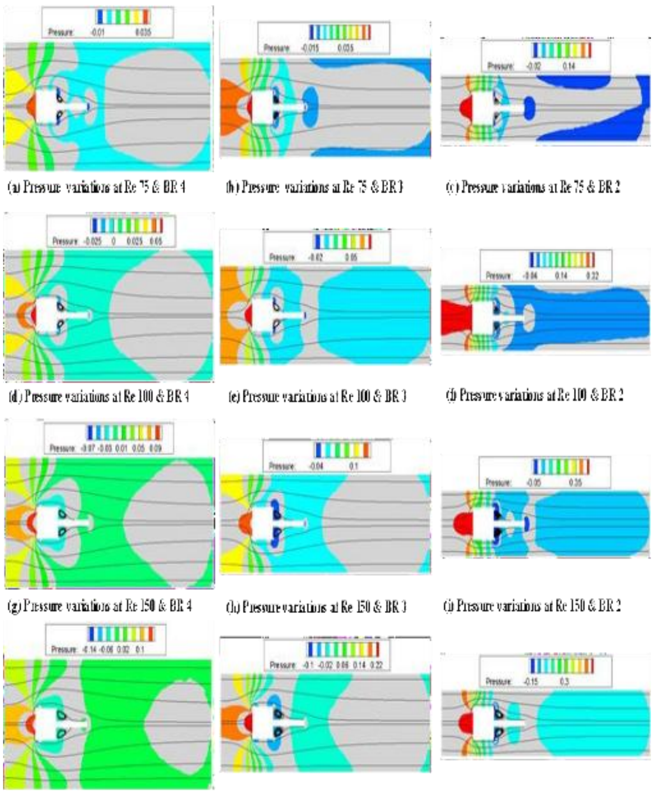


Fig. 3 Comparison of pressure (Pascal) variations at different blockage ratios (BR = 2, 3 & 4) and Reynolds number (Re = 75, 100, 150 & 200)

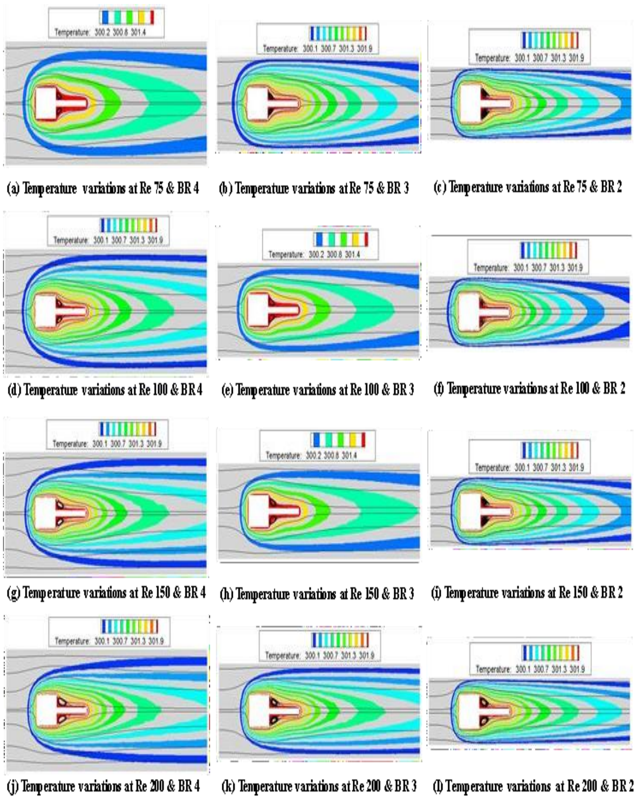


Fig. 4 Comparison of temperature (K) variations at different blockage ratios (BR = 2, 3 & 4) and Reynolds number (Re = 75, 100, 150 & 200)

Here, with the increase of Reynolds number, there is an increase in the upstream pressure which also leads to increase in the maximum pressure maintained on the stagnation point on the front surface of bluff body.The blockage ratio decreases that means the blockage effect increases the pressure drop also increases. The rear side of bluff body two identical vertices exist to see whose sizes are also increases adversely when Reynolds number increases but it could not shred sufficiently which will guarantee structural stability. Simultaneously, the decrease in blockage ratio supresses the vertices and reduces the sizes favourably. In specific, the temperature contour clearly exposes that irrespective of blockage ratio, heat dissipation increases when Reynolds number increases. The decrease in blockage ratio highly supportive in enhancing convection heat transfer due to variations in local acceleration of air nearer to the bluff body.

C. Drag Coefficient Variations

The pressure drag (Fig. 5), viscous drag (Fig. 6) and total drag coefficients (Fig. 7) have been studied for different blockage ratios at different Reynolds number.

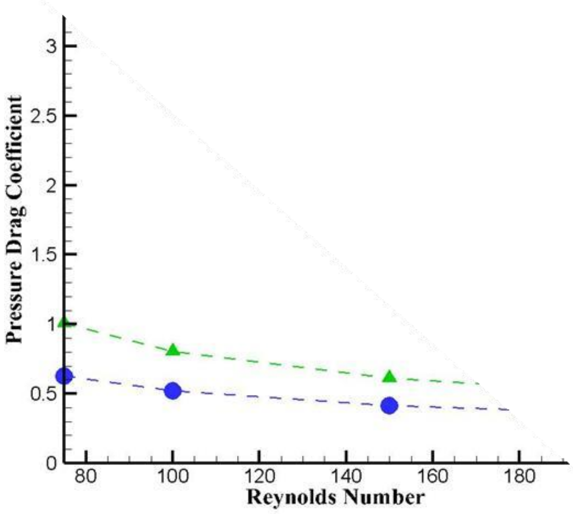


Fig. 5 Variation of Pressure Drag Coefficient with Re at different BR

Here it can be observed a slight increment of drag coefficient for blockage ratio 3 with respect to blockage ratio 4, but there is a drastic increase in drag coefficient when we further decrease the blockage ratio to 2. When the fluid hits the front side of square geometry, the flow is spited, drifting away from cylinder. For higher channel height, the walls are located further from geometry, allowing the fluid to follow a path away from the geometry. As the channel height decreases, and the walls move towards the geometry, the fluid separated has to flow through the confined path. As the contact between fluid ang geometry increases, it drags the geometry towards its flow direction, increasing Drag Coefficient.

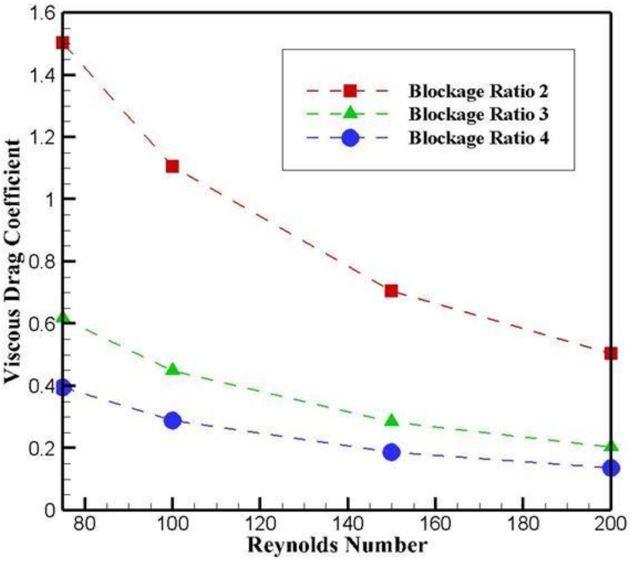


Fig. 6 Variation of Viscous Drag Coefficient with Re at different BR

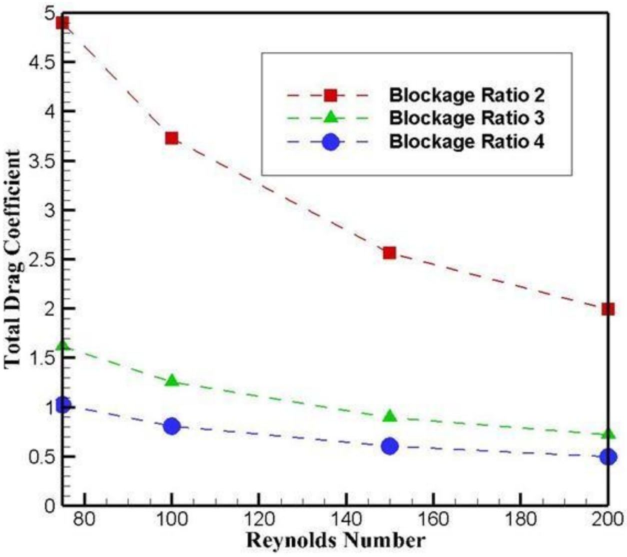


Fig. 7 Variation of Total Drag Coefficient with Re at different BR

D. Pressure Drop Variations

The total pressure drop has been studied for different blockage ratios at different Reynolds number. From the graph (Fig. 8), it can get a huge increment in pressure drop at blockage ratio 2 than the higher blockage ratios. As blockage effect is increased, the fluids are squeezed to flow through the confined region, where its flow velocity increases. When it exits the confined region, there is sudden increase in area, which leads to sudden deceleration of fluid. This leads to an increase in pressure variations at either side of the geometry. The pressure drop increases with increase in Reynolds number. The corresponding local variations in pressure coefficient (Fig. 9) on the outer vicinity of bluff body are also presented for better understanding.

E. Local Nusselt Number Variation

By evaluating Fig 10, it is observed that Nu is almost similar at top and bottom sides of geometry. Nu is optimum at front surface, followed by rear surfaces. It can be observed that on protrusions, heat transfer is not optimum. This is because of formation of eddies and bypassing of the fluid as depicted from streamlines in contour. The Nusselt number increases with an optimum range by increasing confinement for upstream side, but the enhancement is not similar on protrusions. The confinement is having a lesser influence on the resolving of eddies, and enhancing heat transfer through given protrusion.

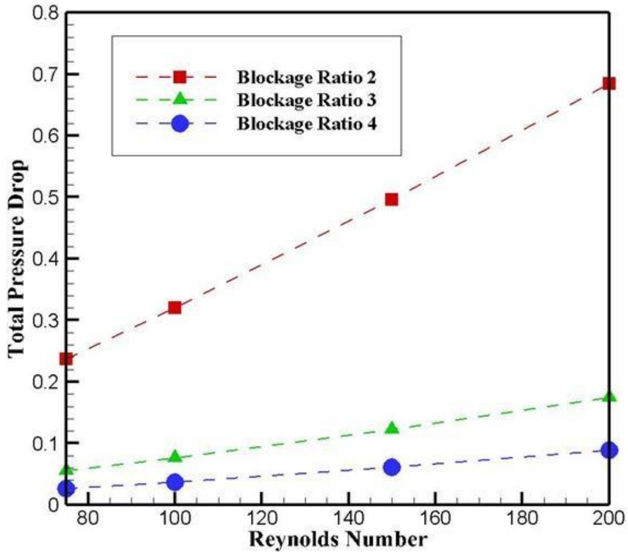
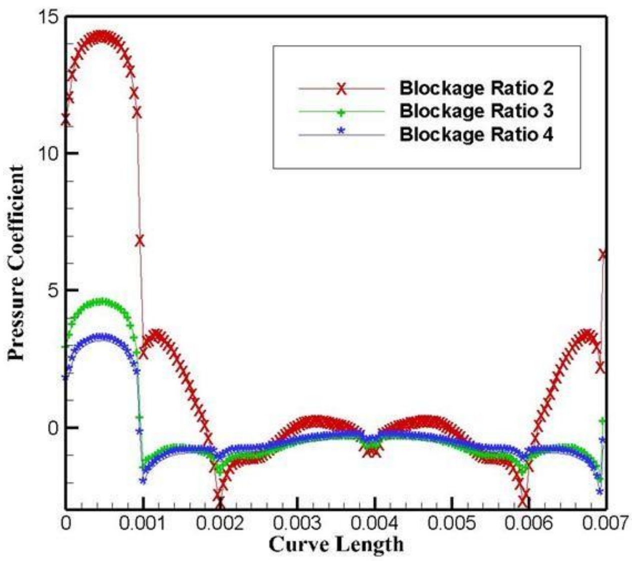


Fig. 8 Variation of Total Pressure Drop with Re at different BR

Fig. 9 Local Pressure Coefficient along surface for Re = 200 at different BR



F. Variation of Average Surface Nusselt number

For the given cases. Nusselt Number is calculated by keeping the surface temperature of geometry as constant. Here with increase in Reynolds number, there is increment in heat transfer coefficient and surface Nusselt number (Fig 11). It is observed that, by increasing confinement, there is an increase in transfer of heat from the surface. By increasing confinement, the fluid after splitting hits the walls, and is forced to strike the geometry, hence increasing its contact with geometry.

As seen in contours, the streamline tends to bulge inwards by decreasing confinement, its interaction with the protrusion region is increasing, hence resulting in increased heat transfer coefficient and surface Nusselt number.

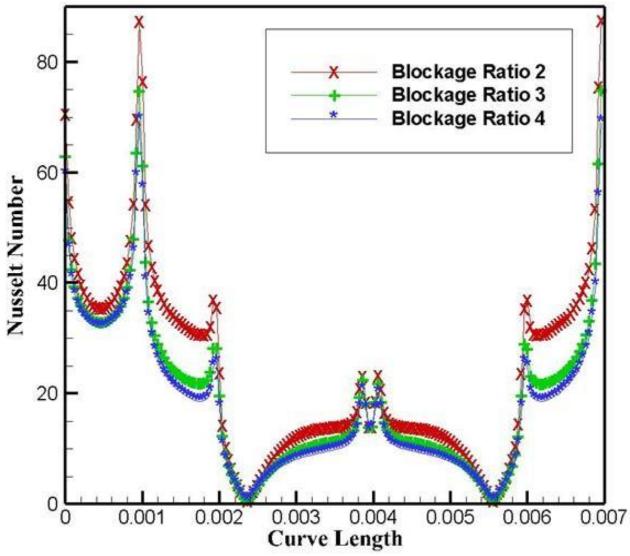


Fig. 10 Local Nusselt Number along surface for Re = 200 at different BR

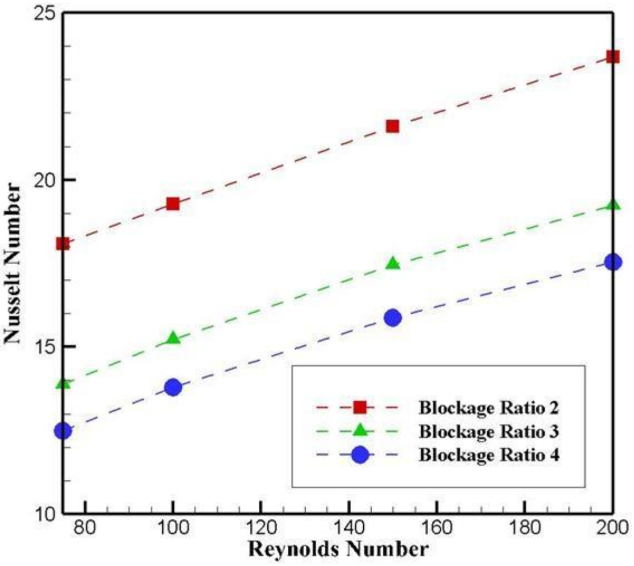


Fig. 11 Variation of Surface Nusselt Number with Re at different BR

VII. CONCLUSION

The flow past heated bluff body with protrusions along downstream has been numerically studied for different Reynolds numbers (75 to 200) and Blockage Ratios (2, 3 & 4). Better heat dissipation was observed with additional pressure drop when Reynolds number increases and also Blockage ratio decreases. Even though square bluff body surface extended towards downstream of flow does not influence the heat transfer much because it arrests flow shredding in the downstream. Hence, such kind of protrusions are more reliable when the failure of the component due to flow shredding frequency.

ACKNOWLEDGMENT

We sincerely acknowledge Product Development Lab and Computational Fluid Dynamics Lab at Department of Mechanical Engineering, SRMIST for carrying out the modelling, and fluid flow analysis.

REFERENCE

1. P. Incropera, “Convection heat transfer in electronic equipment cooling,” J. Heat Transfer, vol. 110, no. 4b, 1988, pp. 1097–1111.
2. R.-J. Yang and L.-M. Fu, “Thermal and flow analysis of a heated electronic component,” Int. J. Heat Mass Transfer, vol. 44, no. 12, 2001, pp. 2261–2275.
3. J. Davalath and Y. Bayazitoglu, “Forced convection cooling across rectangular blocks,” J. Heat Transfer, vol. 109, no. 2, 1987 pp. 321–328.
4. Okajima,”Numerical simulation of flow around rectangular cylinders”, J. Wind Eng. Ind. 33, 1990, 171-180.
5. Sohankar , C. Norberg , L. Davidson , “Numerical simulation of unsteady low-Reynolds number flow around rectangular cylinders at incidence”, Journal of Wind Engineering and Industrial Aerodynamics, 1997, 69-7, 189-201.
6. Subhanker Sen, Sanjay Mittal, “Free vibration of a square cylinder at low Reynolds number”, J. Fluids Struc. 27, 2011, 875-884.
7. Sharma and V. Eswaran, “Heat and Fluid Flow Across a Square Cylinder in the Two-Dimensional Laminar Flow Regime”, Numerical Heat Transfer, Part A, 2004, 45: 247–269.
8. R. W. Davis and E. F. Moore, “A numerical study of vortex shedding from rectangles,” J. Fluid Mech., vol. 116, 1982, pp. 475–506.
9. Sharma and V. Eswaran, “Effect of channel confinement on the two-dimensional laminar flow and heat transfer across a square cylinder,” Numer. Heat Transfer, vol. 47, no. 1, 2004, pp. 79–107.
10. T. Ambreen, M.-H. Kim, “Flow and Heat transfer characteristics over a square cylinder with corner modifications, International Journal of Heat and Mass Transfer” 117, 2018, 50–57.
11. W. A. Khan, J. R. Culham, and M. M. Yovanovich, “The role of fin geometry in heat sink performance,” ASME Transfer J. Electron. Packag., vol. 128, no. 4, 2006, pp. 324–330.
12. S. Cao, Y. Ge, Y. Tamura, “Shear Effects on Flow past a Square Cylinder at Moderate Reynolds Numbers”, J. Eng Mech. 138, 2012, 116-123.

AUTHORS PROFILE

Amit Varakhedkar, Third year B.Tech. Mechanical Engineering student of SRM Institute of Science and Technology, Kattankulathur, Chennai. His current research interests include computational fluid dynamics and heat transfer, electronic cooling, and fluid structure interaction.
Email id: amitvarakhedkar@gmail.com

Rajendran Senthil Kumar received the B.E. degree in mechanical engineering from Bharathidasan University, Tiruchirappalli, India, in 2000, the M.Tech. degree in design and production of thermal power equipment from the National Institute of Technology, Tiruchirappalli, in 2004, and the Ph.D. degree in mechanical engineering from the Indian Institute of Information Technology Design and Manufacturing Kancheepuram, Chennai, India, in 2017. He is currently a Research Assistant Professor with the SRM Institute of Science and Technology, Chennai. His current research interests include computational fluid dynamics and heat transfer, electronic cooling, heat transfer enhancement using nano materials and energy efficient buildings.
Email id: senthilkumar.r@ktr.srmuniv.ac.in