Proceedings of the 6th International and 43rd National Conference on Fluid Mechanics and Fluid Power
December 15-17, 2016, MNNITA, Allahabad, U.P., India

FMFP2016-Paper No. TH-02

Forced Convection from Heated Rectangular Cylinder in Turbulent Regime using PANS Method

Anupam Dewan
Department of Applied Mechanics
Indian Institute of Technology Delhi
Hauz Khas, New Delhi - 110016
E-mail: adewan@am.iitd.ac.in

Pritanshu Ranjan
Department of Applied Mechanics
Indian Institute of Technology Delhi
Hauz Khas, New Delhi - 110016
E-mail: pritanshu.ranjan@iitd.ac.in

Abstract

A computational study of heat transfer from rectangular cylinders is carried out. Rectangular cylinders are distinguished based on the ratio of the length of streamwise face to the height of the cross-stream face (side ratio, *R*). The simulations were performed to understand the heat transfer in a flow field comprising separation, reattachment, vortex shedding and stagnation regions. The applicability of different wall modeling approach is assessed first and then the effect of the side ratio on the heat transfer is studied in detail. The Partially-Averaged Navier-Stokes (PANS) approach is used for modeling turbulence. The Reynolds number is maintained at 22000 for the entire study. It was observed that, only the wall resolve approach was able to predict the heat transfer accurately. The maximum heat transfer rate was observed for rectangular cylinder with R = 0.62 and the strength of vortex shedding decreased continuously with increasing side ratio.

Keywords: PANS, Wall-modelling, forced convection, rectangular cylinder

I. INTRODUCTION

Forced convection from a bluff body located in a cross flow is a major issue encountered in many practical situations, such as, cooling towers, heat exchangers, turbo-engines, electronic equipment cooling, etc. Heat transfer in these cases is generally due to turbulence and predicting such kind of flows is difficult on account of various complex characteristics, such as, stagnation, vortex shedding, separation, and interaction of separated shear layers, associated with it. Understanding the influence of these flow characteristics on heat transfer from a body is indispensable for efficient and economical construction of heat exchanging devices. In this context, the modeling technique, to be used, should be able to accurately capture all the aforementioned flow physics. Conventionally, industries have been using the Reynolds-Averaged Navier-Stokes (RANS) methodology, based on the statistical approach. This methodology fails to predict 3D fluctuating components of a complex flow field. On the other hand, Large Eddy Simulation (LES) can be an option but, it requires large computational resources and it is very sensitive to grid type used, thus making it difficult for the industries to use it. Therefore, in the recent times, hybrid modeling or variable resolution (VR) modeling is being used to study these type of problems. In hybrid/VR modeling approach the extent of resolution required can be decided based on the computational resources available and thus gives an advantage over both RANS and LES. The aim of the present lecture is to show the use of a variable resolution model, with any grid type and less computational effort, for problems concerning convective heat transfer from a bluff body in a cross-flow.

Many variable resolution modeling approaches have been proposed in the literature. Recently Girimaji and his associates [1,2] proposed Partially-Averaged Navier-Stokes (PANS) approach, based on the RANS paradigm to partially resolve large eddies and consider smaller eddies using two equation models. They showed that the PANS approach can commute from RANS approach to DNS depending on the filter width used. The filter width is specified by two parameters, the ratio of the unresolved-to-total kinetic energy (f_k) and the ratio of the unresolved-to-total dissipation (f_ε), and by defining these parameters a desired level of physical resolution can be achieved between RANS approach and DNS. Many researchers [3-5] have used the PANS approach, based on various RANS models, to predict different flow fields with desired variable resolution and showed the applicability of the PANS approach to a wide range of applications. Therefore, the PANS modeling approach is used in the present study and a desired physical resolution is attempted to be achieved.

The geometry considered to study the heat transfer characteristics from a bluff body is a rectangular cylinder. The cylinder is maintained at a higher constant temperature than the incoming air flow in the cross stream direction. There are two important reasons for choosing this flow configuration. First, its similarity to typical flow over bluff body configuration encountered in many practical situations. It is considered most challenging as it presents all the above-mentioned complex

flow phenomena. Second, a large number of experimental studies have been reported in the literature to study its flow dynamics [6–8]. The key parameter which affects the fluid flow and heat transfer characteristics of flow over a rectangular cylinder, is the aspect ratio of the cross-section of the cylinder. Computationally to accurately predict the heat transfer, effect of wall modeling approach is also quite important, as the flow gets separated due to the sharp corners of the rectangular cylinder.

Most previous papers related to heat transfer from rectangular cylinders dealt with experimental studies. Experiments related to the square cross-section were conducted by [9,10] while [11,12,13] performed experiments for cylinders with various cross-sectional aspect ratios. All the experimental studies mainly investigated variation of the average Nusselt number and did not discuss unsteady characteristics associated with the heat transfer. Computationally the study of heat transfer distribution has been performed for square cross-section rectangular cylinder [14,15] and LES was used in both these studies for turbulence modeling. Therefore, it can be concluded from the above-mentioned discussion that the heat transfer mechanism associated with a heated rectangular cylinder kept in a cold cross air stream needs further study.

The present study has three objectives: (a) to gain a deeper understanding of the physics of heat transfer characteristics from a rectangular cylinder; (b) to assess various wall modelling approaches to accurately predict thermal characteristics in separated regions; and (c) to study the effect of cross-sectional aspect ratio on heat transfer from the cylinder.

II. METHODOLOGY

All the computations for a heated rectangular cylinder were performed on an open source platform OpenFOAM [16], which is a set of object oriented finite volume based codes written in C++. The PANS model based on Menter's SST k-ω [17] was derived and implemented in OpenFOAM and tested for an isothermal lid driven cavity case at Re = 2000. The computed results were compared with the reported experimental data. The implemented model was then used for further computations. The study is divided into two parts: (i) PANS computations of square cross section cylinder with different wall functions and unsteady grid and (ii) dynamic PANS computations of rectangular cylinders with different cross-sectional aspect ratio. The schematic of the computational domain for all the cases is shown in Fig. 1. As the cylinder is free standing with $H = 14D$ it was located in the middle. Top and bottom walls were considered to be the zero shear stress and uniform velocity (U_∞) and temperature (T_∞) were specified at the inlet. The atmospheric pressure condition with the zero gradient for all other flow variables was set at the outlet and periodic conditions were set at the lateral boundaries.

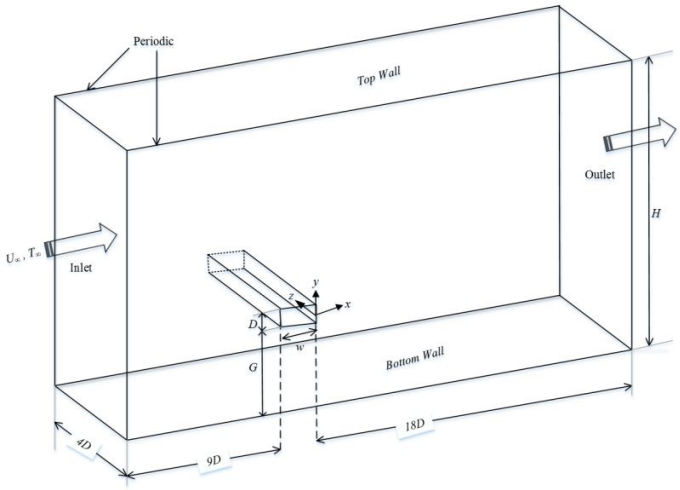


Fig. 1 Computational domain used in the present study.

III. RESULTS AND DISCUSSION

This section discusses the computational results. An attempt is made to reason out the physical explanations of various results obtained.

A. Effect of Meshing Strategy and Wall Modelling Approach on Fluid Flow and Heat Transfer from Square Cylinder

Two types of meshing strategies were used to carry out the computations, first the structured mesh with hexahedral elements and second the unstructured grid with tetrahedral elements. The reason for using tetrahedral unstructured grid was its competence to be able to fit in any arbitrary shaped volume with small effort. The following three different combinations were used.

a. *Structured mesh with wall function approach (SM-WF1)*:

Hexahedral meshing was done in the entire domain and the wall y+ values around the cylinder were maintained close to 30.

b. *Unstructured mesh with wall function approach (UM-WF2)*:

The whole domain was meshed using tetrahedral mesh elements and y+ values around the cylinder were maintained close to 30.

c. *Unstructured mesh with wall resolve approach (UM-WR)*:

Near the cylinder hexahedral prism elements were used and away from the cylinder tetrahedral elements were used and y+ values around the cylinder were maintained close to 1.

In the wall function approach, for flow variables the standard wall functions were used. For heat transfer, two different wall functions were used for the temperature profile. One, the standard wall function approach corresponding to the Reynolds analogy and, second given by Kader [18]. For the wall resolve approach (WR) the grid was refined enough to capture the viscous sub-layer, so as to obtain accurate predictions of the wall shear stress and heat flux. The PANS SST k-ω model uses

the blending functions to provide damping scaling for the eddy viscosity in boundary-layer, which makes it suitable for this approach. Thus flow field and heat transfer are solved inside the turbulent boundary-layer in this approach. The equation for the both the wall function approaches are shown here:

Wall function approach 1 (WF1):

$$T^+ = \Pr.y^+ \qquad \text{if} \qquad y^+ \le 11$$

$$T^+ = 2.08\ln y^+ + 3.9 \qquad \text{if} \qquad y^+ > 11$$

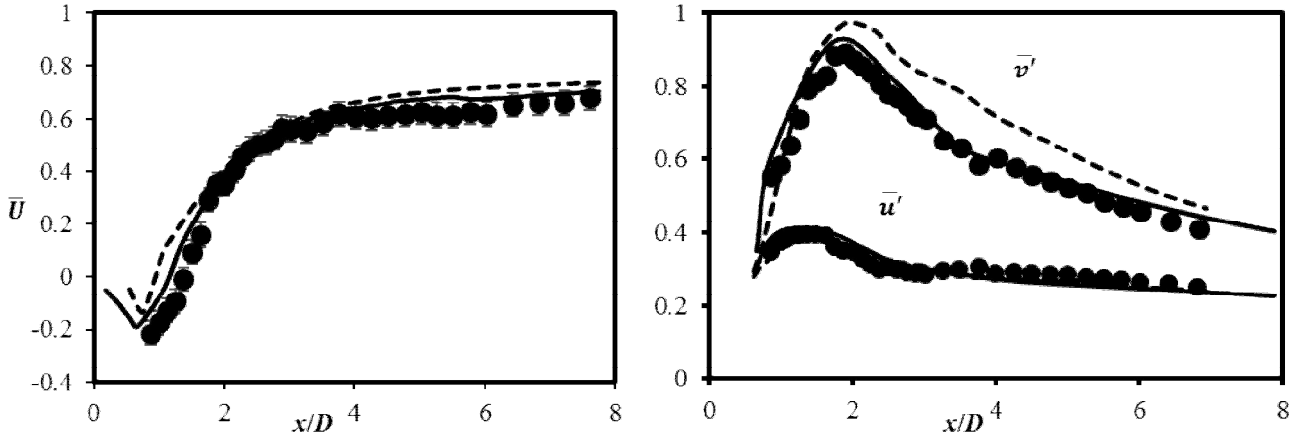
Wall function approach 2 (WF2):

$$T^+ = Pr.y^+.e^{-\Gamma} + \left[2.12\ln\left(1+y^+\right) + \beta\left(\Pr\right)\right]e^{-\Gamma}$$

$$\beta\left(\Pr\right) = \left(3.85\,\Pr^{1/3} - 1.3\right)^2 + 2.12\ln\left(\Pr\right), \Gamma = \frac{0.001\left(Pr.y^+\right)^4}{1+5Pr.y^+},$$

where, $T^+ = \dfrac{\rho C_p u_\tau \left(T_{wall} - T\right)}{q_{wall}}$.

Predictions of the flow variables were similar for both approaches, the wall function and wall resolve approach. The mean and fluctuating velocities predicted by both the wall modelling approaches were accurate when compared to the experimental data (Fig. 2). From Fig. 2 it can be observed that the mean and turbulent quantities of all flow variables are predicted well in accordance with the various experimental data using both the wall modeling approaches. But, the heat transfer predictions showed some deviations. Fig. 3 shows variation of Nusselt number (*Nu*) around the cylinder. It can be observed that, both the wall function approaches (WF1 and WF2) under-predict *Nu* around the cylinder, suggesting the incapability of the wall function approach to accurately model thermal characteristics.



(a) (b)

Fig. 2(a) Mean and (b) fluctuating velocity profiles in streamwise direction along centerline with zero at the center of back face. Solid circles: experimental [7], solid line: wall resolve approach, dotted line: wall function.

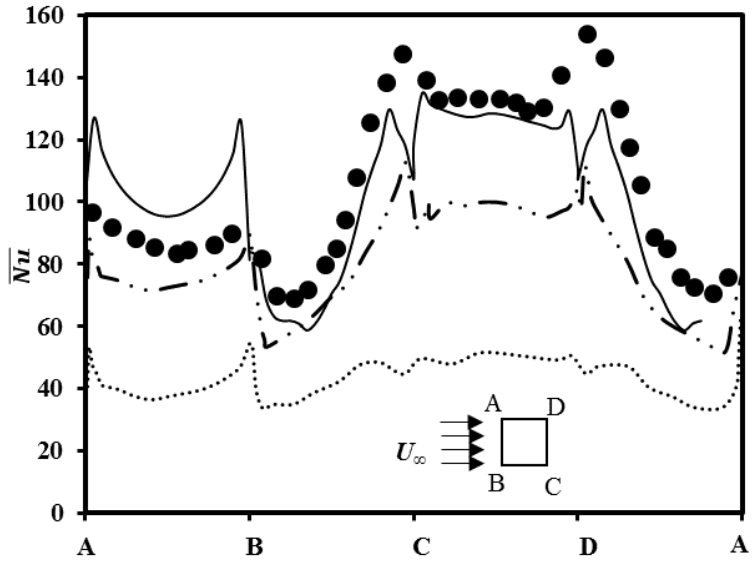
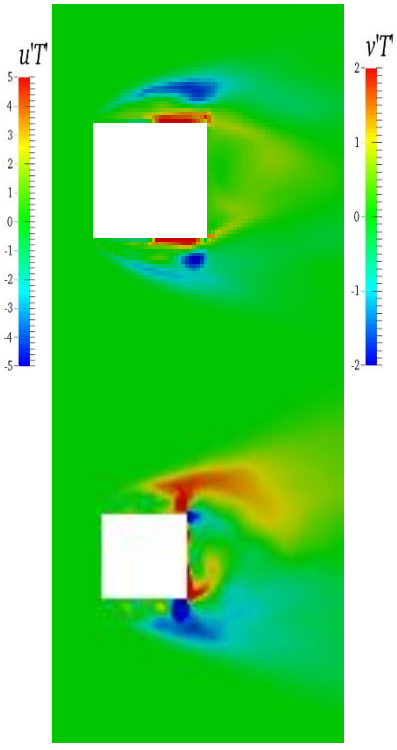


Fig. 3 Time-averaged Nusselt number profile around the cylinder. Solid circles: experimental [19], solid line: wall resolve approach, dotted line: WF1, dash double dot: WF2

Incompetency of the wall functions approach to predict the thermal behavior is due to the conditions in which any wall function is valid, namely, a constant pressure gradient or an attached shear flow. None of these conditions are satisfied in the present case of square cylinder which involves separation at the top and bottom faces. But, the flow variables are predicted quite in accordance with the experimental results and there are two reasons for this behavior: (a) the wall functions used for the flow variables are applicable for the given y^+ values and a blending function is used to make them robust and (b) the vortex shedding phenomenon and flow variables in the separated region do not depend on the wall friction, but are rather governed by large eddy motions, which are well outside the wall functions region. But, the wall functions used for the energy equation are derived using the Reynolds analogy, which is not fully verified, to be valid in the said y^+ region. Apart from it heat transfer near the wall is mainly governed by small scales turbulence in the wall boundary-layer, which heavily depends on the accurate prediction of wall friction.



(a) *u'T'* (b) *v'T'*

Fig. 4 Turbulent heat flux contours.

It is also observed from Fig. 3 that the value of Nusselt number for the rear face is more than that of top and bottom faces. This behavior can be explained by examining turbulent heat fluxes. The contours levels of the time-averaged turbulent heat flux in the mid-plane (z = 0) is shown in Fig. 4. The magnitude of the cross-stream turbulent heat flux is large at the rear corners, due to which fresh fluid is always induced towards the center region of the rear face thus increasing the convective heat transfer. On the side faces and front upper corners, the values of both the axial and transverse turbulent heat fluxes are smaller, which can be due to a backflow in the separation region. Hence a low value of heat transfer from the side walls than that from the rear face is expected. Phase averaging was also performed and it was observed that the variation of Nusselt number for top and bottom faces of the cylinder are always out of phase. For front and rear faces there was no significant change in the Nusselt number during the vortex shedding cycle.

B. Effect of Cross-sectional Aspect/Side Ratio on Fluid Flow and Heat Transfer from Rectangular Cylinder

In this section a computational study of heat transfer from rectangular cylinders is carried out. Rectangular cylinders are distinguished based on the ratio of the length of the streamwise face to the height of the cross-stream face (aspect/side ratio, R). The PANS model used is dynamic in nature, i.e., the resolution parameter is updated on the fly during computations. The application of dynamic PANS model allows more accurate prediction for heat and fluid flow across the rectangular cylinder. The aspect/side ratio (w/D) of the cross-section of the rectangular cylinder was varied from 0.62 to 4 at a Re of 22000. Two critical side ratios were obtained, R = 0.62 and 3.0. At R = 0.62, the maximum value of the drag coefficient (C_d) = 2.681 was observed which gradually reduced by 54% at R = 4.0. The base pressure coefficient (C_{pb}) and global Nusselt number also attained the maximum value at R = 0.62. But compared to C_d and C_{pb}, which decreased continuously with increasing R, the value of St decreased from R = 0.62 to 2.5 and then increased discontinuously from R = 2.5 to 3.0 by 140%. The reason for this behavior can be better explained by plotting the pressure coefficient along top/bottom faces of the cylinder in the streamwise direction.

An increase in C_d up to the critical side ratio (0.62) is due to the progressive interaction between the separated shear layers which reduces the base cavity and hence decreasing the centerline base pressure. Beyond the value of this critical side ratio, vortices are forced to form further downstream because of rear edge-separated shear layers interaction, which increases the base pressure and eventually reduces the values of C_d and C_{pb}. The interaction between the rear edge and shear layer end up reattaching the flow on the side face, which causes a jump in St, as explained earlier in this section, followed by a strong recovery of the pressure towards the base area. Fig. 5 shows the pressure distribution on the side faces, and a sharp recovery in the pressure towards the trailing edge of the side face near the base area is clearly visible for longer cylinder.

The heat transfer characteristics were studied using the Nusselt number distribution. A variation of the global Nusselt

number $\left(\overline{Nu_g}\right)$ with the side ratio is shown in Fig. 6. It can be seen that $\overline{Nu_g}$ has a maxima at R = 0.62 and then steadily decreases up to 2.5 before taking a sudden jump of 12% at R = 3.0. The reason for this behaviour can be explained by plotting $\overline{Nu}$ along the top/bottom face of the cylinder cross-section (Fig. 7).

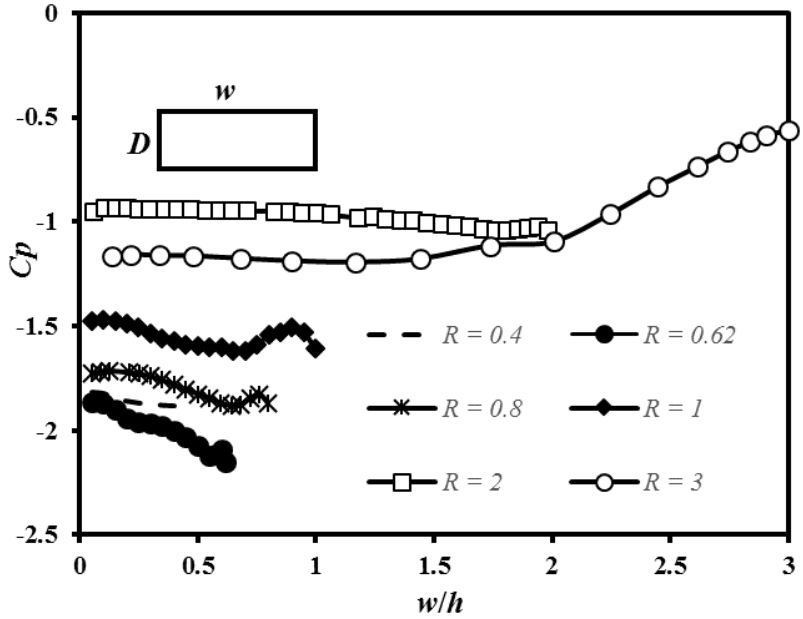


Fig. 5 Pressure coefficient along the top/bottom face of the cylinder.

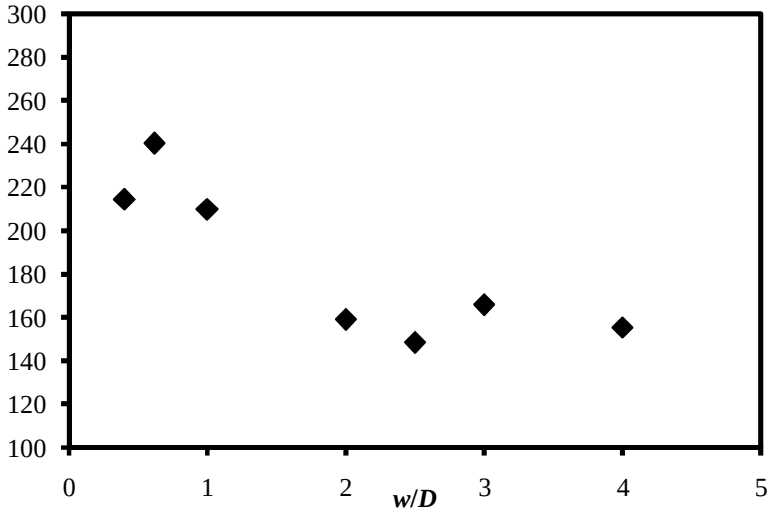


Fig. 6 Variation of time averaged global Nusselt number with side ratio.

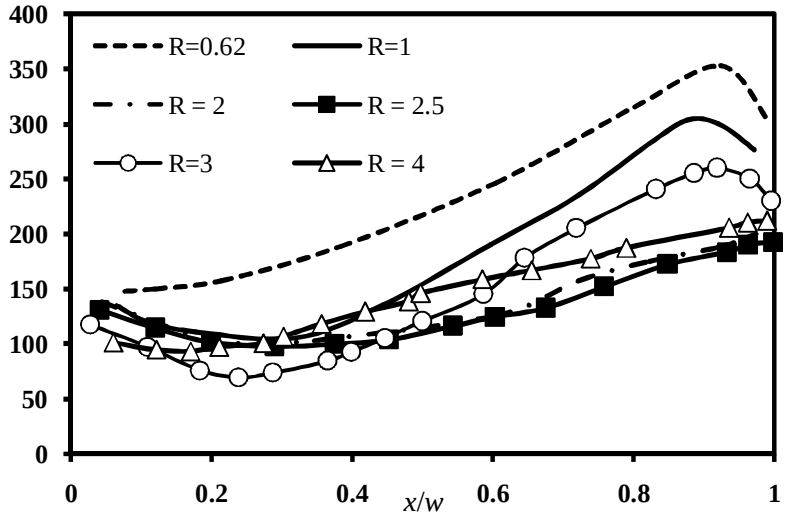


Fig. 7 Variation of $\overline{Nu}$ along the length of the side face.

It can be observed that, the Nusselt number recovery for R = 0.62 is the largest. It is because the vortex formed goes over the side face and rolls back to reattach at the trailing edge, due to which high turbulent fluctuations are produced near the trailing edge. This behavior increases the heat transfer and thus the local Nusselt number on the side face towards the rear face. As the value of R is increased above 0.62, the vortex formation gets shifted towards the trailing edge and its strength decreases due to which the Nusselt number does not recover well compared with the case with R = 0.62. But at R = 3, there is a small jump in the global Nusselt number (Fig. 6) and an increment in the recovery of the local Nusselt number as compared to the cases with R = 2.5 and R = 4 (Fig. 7). This behavior is because the flow gets attached on the side face and then the vortex is formed after the attachment which has the same behavior as of the vortex formation at R = 1 but with lower strength. Thus, the Nusselt number recovery profile on the side face for R = 3 is very similar to that of R = 1, but the overall recovery is small.

IV. CONCLUSIONS

In this study PANS, a variable resolution turbulence modeling approach was used to predict the fluid flow and heat transfer from rectangular cylinders. Detailed investigations were performed to understand the heat transfer distribution in the separated turbulent regions. The major conclusions of the present study are:

- The prediction of flow variable was not dependent on the type of wall modelling approach used. Both wall function and wall resolve approach accurately predicted the mean and turbulent flow variables.
- The heat transfer results showed large dependence on the wall modeling approach used. The wall functions approach largely under-predicted Nusselt number compared to reported experimental data. The wall resolve technique showed a good agreement with the experimental results and was further used to study the unsteady thermal characteristics.
- As the aspect ratio of the cross-section was changed the flow and thermal behavior changed drastically. Cases with R = 0.62 and 3.0 come out to be the critical side ratios. At R = 0.62, the maximum values of St, mean C_D and C_L and base pressure coefficient were achieved. But St values showed a discontinuous jump at R = 3.0 by 140% as compared to the case with R = 2.5.
- The total heat transfer from the cylinder followed the same profile as that of St w.r.t. R. It also had a maxima at R = 0.62, decreased as R was increased, increased suddenly at R = 3.0 and then decreased again.

NOTE: The contents of the this thematic lecture are taken from the published papers [20,21,22] of the presenting authors.

REFERENCES

[1] S.S. Girimaji, Partially-Averaged Navier-Stokes Model for Turbulence: A Reynolds-Averaged Navier-Stokes to Direct Numerical Simulation Bridging Method, J. Appl. Mech. 73 (2006) 413. doi:10.1115/1.2151207.

[2] S.S. Girimaji, E. Jeong, R. Srinivasan, Partially Averaged Navier-Stokes Method for Turbulence: Fixed Point Analysis and Comparison With Unsteady Partially Averaged Navier-Stokes, J. Appl. Mech. 73 (2006) 422. doi:10.1115/1.2173677.

[3] C.-S. Song, S.-O. Park, Numerical simulation of flow past a square cylinder using Partially-Averaged Navier–Stokes model, J. Wind Eng. Ind. Aerodyn. 97 (2009) 37–47. doi:10.1016/j.jweia.2008.11.004.

[4] J. Ma, F. Wang, X. Yu, Z. Liu, A Partially-Averaged Navier-Stokes model for hill and curved duct flow, J. Hydrodyn. Ser. B. 23 (2011) 466–475. doi:10.1016/S1001-6058(10)60137-0.

[5] B. Basara, S. Krajnovic, Z. Pavlovic, P. Ringqvist, Performance analysis of Partially-Averaged Navier-Stokes method for complex turbulent flows, in: 6th AIAA Theor. Fluid Mech. Conf., American Institute of Aeronautics and Astronautics, Reston, Virigina, 2011: pp. 1–10. doi:10.2514/6.2011-3106.

[6] D. Durao, M. Heitor, J. Pereira, Measurements of turbulent and periodic flows around a square cross-section cylinder, Exp. Fluids. 6 (1988) 298–304. http://link.springer.com/article/10.1007/BF00538820 (accessed January 5, 2015).

[7] D.A. Lyn, S. Einav, W. Rodi, J.-H. Park, A laser-Doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder, J. Fluid Mech. 304 (1995) 285. doi:10.1017/S0022112095004435.

[8] C. Norberg, Flow around rectangular cylinders: Pressure forces and wake frequencies, J. Wind Eng. Ind. Aerodyn. 49 (1993) 187–196. doi:10.1016/0167-6105(93)90014-F.

[9] T. Igarashi, Heat transfer from a square prism to an air stream, Int. J. Heat Mass Transf. 28 (1985) 175–181. doi:10.1016/0017-9310(85)90019-5.

[10] G.R. Ahmed, M.M. Yovanovich, Experimental Study of Forced Convection From Isothermal Circular and Square Cylinders and Toroids, J. Heat Transfer. 119 (1997) 70. doi:10.1115/1.2824102.

[11] T. Igarashi, Fluid flow and heat transfer around rectangular cylinders (The case of a width/height ratio of a section of 2.0-4.0)., Trans. Japan Soc. Mech. Eng. Ser. B. 52 (1986) 3011–3016. doi:10.1299/kikaib.52.3011.

[12] T. Igarashi, Fluid flow and heat transfer around rectangular cylinders (the case of a width/height ratio of a section of 0.33 ~ 1.5), Int. J. Heat Mass Transf. 30 (1987) 893–901. doi:10.1016/0017-9310(87)90008-1.

[13] S.-Y. Yoo, J.-H. Park, C.-H. Chung, M.-K. Chung, An Experimental Study on Heat/Mass Transfer From a Rectangular Cylinder, J. Heat Transfer. 125 (2003) 1163. doi:10.1115/1.1603780.

[14] S. aus der Wiesche, Large-eddy simulation study of an air flow past a heated square cylinder, Heat Mass Transf. 43 (2006) 515–525. doi:10.1007/s00231-006-0122-x.

[15] M. Boileau, F. Duchaine, J.-C. Jouhaud, Y. Sommerer, Large-Eddy Simulation of Heat Transfer Around a Square Cylinder Using Unstructured Grids, AIAA J. 51 (2013) 372–385. doi:10.2514/1.J051800.

[16] OpenFOAM® Documentation, (n.d.). http://www.openfoam.org/docs/ (accessed January 15, 2015).

[17] F. Menter, J.C. Ferreira, T. Esch, B. Konno, The SST Turbulence Model with Improved Wall Treatment for Heat Transfer Predictions in Gas Turbines, in: Proc. Int. Gas Turbine Congr. 2003 Tokyo, 2003. https://nippon.zaidan.info/seikabutsu/2003/00916/pdf/igtc2003tokyo_ts059.pdf.

[18] B. Kader, Temperature and concentration profiles in fully turbulent boundary layers, Int. J. Heat Mass Transf. 24 (1981) 1739–42. doi:0017~9310/81/091541-04 so2.00/0.

[19] T. Igarashi, Characteristics of the Flow around Rectangular Cylinders : The Case of the Angle of Attack 0 Deg, Bull. JSME. 28 (1985) 1690–1696. doi:10.1299/jsme1958.28.1690.

[20] P. Ranjan, A. Dewan, Partially Averaged Navier Stokes simulation of turbulent heat transfer from a square cylinder, Int. J. Heat Mass Transf. 89 (2015) 251–266. doi:10.1016/j.ijheatmasstransfer.2015.05.029.

[21] P. Ranjan, A. Dewan, Effect of side ratio on fluid flow and heat transfer from rectangular cylinders using the PANS method, Int. J. Heat Fluid Flow. ** (2016) **. doi:10.1016/j.ijheatfluidflow.2016.05.004.

[22] P. Ranjan and A. Dewan, 2016, Study of Heat Transfer Over a Square Cylinder in Crossflow Using Variable Resolution Modeling, J. of Applied Fluid Mechanics, Vol. 9, No. 3, pp. 1367-1379.