Study of Heat Transfer over a Square Cylinder in Cross Flow using Variable Resolution Modeling

P. Ranjan and A. Dewan†

Department of Applied Mechanics, Indian Institute of Technology Delhi, Hauz Khas, New Delhi - 110016, India

†Corresponding Author Email: dewan_anupam@yahoo.com

(Received December 11, 2014; accepted April 29, 2015)

ABSTRACT

In the present study, a method of Partial-Averaged Navier-Stokes (PANS) equations, purported to perform variable resolution modeling, is used to predict the heat transfer over a square cylinder in a cross-flow. The PANS closure is based on the RANS SST $k$-$\omega$ model paradigm. The simulations were carried out using an open source software, namely, Open FOAM, at Reynolds number = 22000. The open source code and the PANS model are validated against the experimental work reported in the literature and it was observed that both the mean flow properties and turbulent statistics were in good agreement with the experimental results. Further the capability of the PANS approach in predicting heat transfer at turbulent flow is also studied. An algebraic wall function is used for the near wall treatment of the energy equation. The computed, average and local Nusselt numbers are compared with the experimental and LES results reported in the literature. The phase-averaged analysis of the shedding phenomenon is studied to understand the heat transfer phenomenon at different faces of the cylinder and turbulent heat fluxes are also considered to understand the effect of turbulence on convection.

Keywords: Turbulent heat transfer; Open FOAM; Partially-averaged navier-stokes (PANS); SST $k$-$\omega$ turbulence model.

1. INTRODUCTION

The need for an accurate prediction of turbulent heat transfer through a bluff body is encountered in many industrial applications, which include, composite materials that are less impervious to heat used in aeronautics industry, cooling towers, turbomachinery, cooling of electronic equipment and various heat exchange devices, etc. The complexity involved in these kinds of flow and heat transfer characteristics is due to separation of shear layers from the body and interaction of these shear layers in the near wake region formed by the separation behind the body. Therefore an accurate modeling technique is required to handle the complexity involved with a bluff body in a turbulent flow field. Thus the computational method which is to be used should be able to accurately capture the flow physics namely: recirculation, vortex shedding, wake region, and shear layer interaction, etc.

The most widely used models in industrial CFD tools for the aforementioned types of problems are those based on the RANS paradigm, which captures the mean flow properties but fails to predict turbulent scales. On the other hand, the LES approach is very promising for capturing flow fluctuations and turbulent scales. But, LES approach is computationally quite expensive when it comes to industrial applications because of complex geometries and large values of Reynolds number involved. Therefore a model is required which is computationally less expensive and also captures the turbulence phenomenon. Girimaji et al. (2006, a) and Girimaji (2006, b) proposed the Partially-Averaged Navier-Stokes equations (PANS) approach to meet the said requirements.

The PANS approach is a variable resolution method in which the extent of resolution is based on the turbulence kinetic energy distribution between eddies which are to be resolved and which are to be modeled. The details of the PANS approach can be found in Girimaji et al. (2006, a) and Girimaji (2006, b). Further Girimaji and Khaled (2005), Lakshimipathy and Girimaji (2006, 2007), Jeong and Girimaji (2010), Murthi et al. (2010), Basara et al. (2011), and Girimaji and Wallin (2013) have also provided the theoretical foundation. They have subsequently assessed the application of PANS approach to many isothermal flows associated with various geometries, such as, backward facing step, circular and square cylinders.

However, before PANS approach can be applied to practical problems concerned with heat transfer, it has to be tested with one of the benchmark
problems. Flow over a heated square cylinder, maintained at a constant temperature, is considered in the present paper to test the effectiveness of the PANS approach. The reason for choosing this flow configuration is, one, a large number of experimental studies have been reported in the literature to study its flow dynamics and, second, its similarity to typical flow over bluff body configuration encountered in many practical situations.

We first discuss experimental studies reported in the literature that are relevant to flow past a square cylinder. Durao et al. (1988) used laser-Doppler velocimetry (LDV) to measure the turbulent flow properties of a square cylinder with Reynolds number based on the cylinder height of $1.4 \times 10^5$. They performed experiments in a water channel with blockage ratio of 0.14. Similarly Lyn and Rodi (1994) and Lyn et al. (1995) also performed experiments with LDV at Reynolds number of $2.2 \times 10^5$. The former focused their studies on shear-layer and recirculation regions and the later focused on near wake flow around the cylinder. Only few experimental studies have been reported in the literature on heat transfer from a square cylinder in cross flow. Igarashi (1985), Ahmed and Yovanovic (1997) and Yoo et al. (2003) provided the mean values of the Nusselt number and derived empirical correlation for the global Nusselt number with respect to Reynolds number.

Further a large number of studies concerning computational analysis of flow field calculations for a square cylinder using various modeling methodologies have been reported in the literature but to the best of author’s knowledge, only a couple of studies have been reported concerning forced convective heat transfer in a flow past a heated square cylinder. Both Wiesche (2007) and Boileau et al. (2013) used LES approach to predict heat transfer around a square cylinder. Boileau et al. (2013) showed the effectiveness of LES with unstructured grid. However, as already stated, LES is still very expensive in terms of the computational resources required and is not feasible for industries to afford such computational demand.

In the present work, flow dynamics and heat transfer from a square cylinder kept in cross flow are investigated using the PANS approach, which is a variable resolution model, capable of predicting scales of turbulence depending on the need. Therefore unsteady flow past a heated square cylinder, with diameter $d$, at fixed surface temperature ($T_s$), at a Reynolds number of $Re_d = 2.2 \times 10^5$ for an incompressible fluid with $Pr = 0.7$ (air). The results obtained are compared with the experimental data of Lyn and Rodi (1994) and Lyn et al. (1995) and with the LES predictions of Boileau et al. (2013). Therefore the current study presents the capability of PANS approach to provide insight into the heat transfer in the wake region of a bluff body.

The paper is organized as follows. In section 2 the equations governing the fluid flow and heat transfer are presented along with the detailed formulation of the PANS approach used in the present work. Section 3 provides the details about the geometry, physical conditions and numerical schemes used in the present study. Finally section 4 presents all results and their comparison with the experimental data. At last the unsteady flow field with heat transfer over a square cylinder is investigated and credibility of PANS approach is concluded.

\section{GOVERNING EQUATIONS}

\subsection{Basis of PANS Approach}

In this section, the Partially-Averaged Navier-Stokes (PANS) equations are briefly summarized as given by Girimaji et al. (2006, a) and Girimaji (2006, b). Starting from the instantaneous incompressible flow equations

\begin{equation}
\frac{\partial V_i}{\partial t} + V_j \frac{\partial V_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 V_i}{\partial x_j \partial x_j} \quad (1)
\end{equation}

\begin{equation}
\frac{\partial V_i}{\partial x_j} = 0 \quad (2)
\end{equation}

We define \( \chi \) as an arbitrary filter which commutes with both spatial and temporal differentiations. Thus by applying this filter to the instantaneous velocity field it can be decomposed as

\begin{equation}
V_i = \bar{U}_i + u_i
\end{equation}

where \( \bar{U}_i = \langle V_i \rangle \) is the filtered/resolved field, \( u_i \) the residual/ fluctuation field that needs to be modeled. Each of the filtered velocity field satisfies the continuity equations separately, such that

\begin{equation}
\frac{\partial \bar{U}_i}{\partial x_i} = \frac{\partial u_i}{\partial x_j} = 0 \quad (4)
\end{equation}

It is well understood that for this type of arbitrary filtering process, the average of the unresolved velocity and the correlation between resolved and unresolved velocities are non-zero, i.e., \( \langle u_i \rangle \neq 0, \langle U_i u_j \rangle \neq 0 \).

Now applying this arbitrary filter to the equation of motion we get

\begin{equation}
\frac{\partial \bar{U}_i}{\partial t} + U_j \frac{\partial \bar{U}_i}{\partial x_j} = -\frac{\partial \langle p \rangle}{\partial x_i} + \nu \frac{\partial^2 \bar{U}_i}{\partial x_j \partial x_j} - \frac{\partial \tau(V, V_i)}{\partial x_j} \quad (5)
\end{equation}

\begin{equation}
\nu \frac{\partial^2 \langle p \rangle}{\partial x_j \partial x_j} = \frac{\partial \bar{U}_i}{\partial x_j} \frac{\partial \bar{U}_j}{\partial x_j} + \frac{\partial \tau(V, V_j)}{\partial x_j} \quad (6)
\end{equation}

In equations (5) and (6), \( \tau(V, V_i) \) denotes the generalized central second moment tensor and is defined as the sub-filtered shear (SFS) stress. It is given by the expression

\begin{equation}
\tau(f, g) = \{fg\} - \{f\} \{g\}
\end{equation}

\begin{equation}
\tau(f, g, h) = \{fg\} - \{f\} \tau(g, h) - \{g\} \tau(h, f) - \{h\} \tau(f, g) - \{f\} \{g\} \{h\} \quad (7)
\end{equation}
Based on equation (7) the sub-filter kinetic energy and dissipation are given as

$$k_u = \frac{1}{2} \tau (V_j V'_j); \epsilon_u = \nu + \nu_t \left( \frac{\partial V_j}{\partial x_j} \right)$$

(8)

All properties with the subscript $u$ indicate the PANS statistics. The RANS kinetic energy and dissipation are represented by $k$ and $\epsilon$, respectively. Equations (7) and (8) reduce to their RANS counterpart, when averaging is performed over all the scales of motion (denoted by an over bar). Further according to Germano (1992) the RANS statistics are related to its PANS counterpart as

$$\bar{V}_j = \left\{ \bar{V}_j \right\} = U_j;$$

$$R(\bar{V}_j) = \bar{V}_j - \bar{V}_j = \tau (\bar{V}_j \bar{V}_j') + R(U_j')$$

(9)

It can be observed from the above set of equations (1) to (9) that the governing equations obtained after filtering are invariant to the filter used and consequently the modeling of SFS stress term must be invariant to the type of averaging. Hence to model the SFS term any modeling approach based on the RANS paradigm can be used.

The arbitrary filter used in equation (3) was defined by Girimaji et al. (2006, a) and Girimaji (2006, b) in terms of the fraction of kinetic energy and dissipation associated with the scales to be modeled rather than on basis of the cut-off wave number, as in LES. They quantified it as the ratios of the unresolved to total kinetic energy and dissipation and it is given by

$$f_k = \frac{k_u}{k}; f_\epsilon = \frac{\epsilon_u}{\epsilon}$$

(10)

Where $f_k$ and $f_\epsilon$ are termed as the resolution control parameters. It is well known that much of the kinetic energy is contained in large scales and most of the dissipation occurs in the smallest scales due to which $0 \leq f_k \leq 1$. Further they showed that, mathematically, when $f_k$ tends to zero, the model approaches DNS behavior as it resolves more scales of motion.

### 2.2 RANS k-\omega Type SFS Stress Closure

For the SFS shear stress the Boussinesq assumption is invoked in combination with the averaging-invariance property for arbitrary filters,

$$\tau (V_j V'_j) = \nu_u \left( \frac{\partial U_j}{\partial x_j} + \frac{U'_j}{\partial x_j} \right) - \frac{2}{3} k_u \delta_{ij}$$

(11)

where, $\nu_u = C_n \frac{k^\frac{3}{2}}{\epsilon}$ and it denotes the unresolved eddy viscosity. Therefore to obtain a closure for the SFS stress term, $k_u$ and $\epsilon_u$ have to be given or modeled. This can be achieved by using any of the previously proposed models of RANS paradigm. In the present work the SST k-\omega model, given by Menter (1993, 1994), is used to close the set of equations (5) to (11). According to Menter (1994) equations for $k$ and $\omega$ are

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho U_j k)}{\partial x_j} = \nu + \nu_t \left( \frac{\partial k}{\partial x_j} \right)$$

(12)

$$\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho U_j \omega)}{\partial x_j} = \frac{\nu k}{\epsilon}$$

(13)

With, $\nu_j = \frac{a_k k}{a_0 \sqrt{2S_f}}$

To derive the PANS SST k-\omega model from the above closure, steps given by Girimaji et al. (2006, a) and Girimaji (2006, b) are followed and the final equations are

$$a_k = \frac{e_u}{\beta k_u}; a_\omega = \frac{\epsilon_u}{\omega}; f_k = \frac{\epsilon}{\beta k}$$

(15)

$$\frac{\partial (\rho k_u)}{\partial t} + \frac{\partial (\rho U'_j k_u)}{\partial x_j} = \frac{\nu k_u}{\epsilon_u} + \frac{\epsilon u}{\epsilon u}$$

(16)

$$\frac{\partial (\rho \omega_u)}{\partial t} + \frac{\partial (\rho U'_j \omega_u)}{\partial x_j} = \frac{\nu \omega u}{\epsilon u} + \frac{\epsilon u}{\epsilon u}$$

(17)

$$\Gamma_{k_u} = \mu \frac{U'_j \epsilon_u}{\sigma_k \epsilon u} \frac{\epsilon_u}{\epsilon u}$$

(18)

$$P_{k_u} = \tau_{\epsilon u} \frac{\partial U'_j}{\partial x_j}$$

(19)

$$CD_{k_u \epsilon_u} = \max \left( \frac{2 \rho \sigma_{\omega_k} \frac{1}{\epsilon_u} \frac{\epsilon u}{\epsilon u}}{CD_{k_u \epsilon_u}} \times 10^{-10} \right)$$

(20)
$F_{2u} = \tanh \left( \arg_2 \right)$,

$\arg_2 = \max \left( \frac{2\sqrt{k_u}}{\beta \theta_n y^2 - 500}, 0 \right)$ \hspace{1cm} (21)

With $\mu_s = \frac{f_{sr}}{f_{sr}} \mu_s$ and the values of the model coefficients are the same as those used by Menter (1994): $\alpha_1 = 2, \alpha_2 = 2, \beta_1 = 0.075, \beta_2 = 0.09, \gamma_1 = 10, \sigma_{a2} = 1.168, \gamma_2 = 0.4403, \beta_2 = 0.0828, \kappa = 0.41$.

Near the wall, automatic wall treatment given by Menter and Esch (2001) is used, which is insensitive to wall grid density and takes advantage of the fact that analytical solutions for both the sub layer and the logarithmic region are known:

$\alpha_{vis} = \frac{6\nu}{0.075}, \alpha_{log} = \frac{1}{0.3e} \frac{u^+}{y^+}$ \hspace{1cm} (22)

and can be reformulated in terms of $y^+$ and a smooth blending as

$\sigma_{p1}(y^+) = \left[ \alpha_{vis}^2 (y^+) + \alpha_{log}^2 (y^+) \right]^{0.5}$ \hspace{1cm} (23)

Similarly velocity profile near the wall is given as:

$u_{vis}^{+} = \frac{U}{y^+}, u_{log}^{+} = \frac{U}{\kappa \ln (y^+)}$ \hspace{1cm} (24)

For $\kappa$ a zero flux boundary condition is applied.

Analogous to the modeling of the SFS stress, the turbulent heat flux vector is also modeled with the help of the turbulent diffusivity:

$\left\langle u_i T_j \right\rangle = -\alpha_t \frac{\partial T}{\partial x_i} = \nu_T \frac{\partial T}{\partial x_i}$ \hspace{1cm} with \hspace{1cm} $Pr_T = \frac{\nu_T}{\alpha_T}$ \hspace{1cm} (25)

where, $\alpha_t$ and $Pr_T$ are turbulent diffusivity and turbulent Prandtl number, respectively.

Therefore with no source terms and constant property assumptions the temperature equation can be written as

$\frac{\partial T}{\partial t} + \frac{\partial u_i T_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \mu + \mu_s \right] \frac{\partial T}{\partial x_j}$ \hspace{1cm} (26)

In Equation (26) $Pr$ is the fluid property and the turbulent Prandtl number, $Pr_T$, is set to a constant value of 0.9 (Boileau et al., 2013).

For the treatment of the temperature equation near the wall, an algebraic formulation given by Kader (1981) is used here:

$\Theta^+ = \frac{Pr y^+ u^+ e^{-T}}{\left( 1 + 5 Pr y^+ \right)^{5/3}} + \left[ \frac{2.12 \ln (1 + y^+)}{1 + 5 Pr y^+} \right] e^{-T}$

$\beta(Pr) = \left( 3.85 Pr \frac{Pr y^+}{2} \right)^{2} + 2.12 \ln (Pr)$ \hspace{1cm} (27)

with

$\Gamma = \frac{0.01 \left( Pr y^+ \right)^4}{1 + 5 Pr y^+}$ and $\Theta^+ = \frac{T_{f} - T}{T_T}$

$T_T = \frac{q_f}{\rho c_p u_T}$ where $u_T$ is the friction velocity.

3 FLOW CONFIGURATION

3.1 Geometry and Physical Conditions

The computational domain used in the present case is shown in Figure 1. All dimensions are considered as a function of the cylinder’s side length $d = 10$ mm. The grid (Figure 2) used in the current study is based on that used by Jeong and Gilrimaji (2010), Barone and Roy (2006) and Lubecke et al. (2001) and these details are summarized in Table 1.

Fig. 1. Computational domain used in present study.

Fig. 2. Side view of computational grid.

The following boundary conditions are prescribed:

1. The no-slip condition for the velocity and a fixed temperature ($T_a$) on the square cylinder all along the spanwise direction are applied.

2. The periodic conditions are prescribed on the lateral sides, so as to compensate for the large
span length used in the experiment of Lyn and Rodi (1994).

3. At the inlet, a uniform flow ($u_x = U$, $u_y = u_z = 0$) based on $Re = 2.2 \times 10^4$ and a uniform inflow temperature ($T_{\text{in}}$) are prescribed. At the outlet atmospheric pressure condition is applied.

4. For the top and bottom planes, a slip condition is applied, considering a thin boundary on the wind tunnel walls of the experiment and are maintained at free stream temperature.

### 3.2 Numerical Treatment

Simulations were performed using a finite-volume based open source code OpenFOAM. The existing SST $k-\omega$ model code was modified according to the PANS equations. The spatial discretization was carried out using the standard Gaussian finite volume integration method with different interpolation schemes. For the gradient terms a linear interpolation of the second order accuracy was implemented. The second order linear interpolation scheme was used for the Laplacian terms while second order upwind scheme was used for the divergence terms. The temporal discretization was performed using the second order linear interpolation schemes. For the gradient terms a diagonal based incomplete Cholesky preconditioner conjugate gradient (PCG) iterative method with a diagonal based incomplete LU (DILU) decomposition preconditioner was applied.

### 3.3 Averaging Procedure

The data obtained was averaged in the spanwise direction, considering it to be statistically homogeneous, by taking values at eight different locations in the $z$-direction. Two types of averaging were performed as suggested by the experimental procedure of Lyn and Rodi (1994):

1. Time-averaging was done for 400D/$U_0$ seconds (approximately 40 periodic cycles) once the flow reached the statistically stationary state (after approximately 400 cycles).

2. Phase averaging was done by averaging any flow property over a constant phase angle for every vortex shedding cycle. As given by Lyn and Rodi (1994), $\phi$ is obtained from a pressure signal $p(t)$ measured by the spanwise averaging at center of the top face of cylinder, as shown in Figure 3a. The filtering of the instantaneous pressure signal $p(t)$ was done through a low pass second-order Butterworth filter with cut-off frequency equal to the shedding frequency, as shown in Figure 3a. Time was non-dimensionalized as $t'$, where $t_\sigma$ is the time pair of each peak and valley is defined as the half of the vortex shedding cycle. The shedding frequency was obtained by the fast Fourier transform of instantaneous pressure signal as shown in Figure 3b. The reference phase angle ($\phi = 0$) was taken in accordance with the heat transfer study of Boileau et al. (2013) as shown in Figure 3a. For the flow properties, as the results are compared with those of Lyn et al. (1995), $\phi = 0$ was taken at one of the peak of the periodic signal.

### 4 RESULTS AND DISCUSSIONS

Firstly the flow dynamics characteristics of flow over square cylinder are compared with the experimental results reported in the literature to evaluate the accuracy of the open source code and the PANS strategy used. Further onwards in all the results, velocity normalization is done by $U_\infty$ (free stream velocity) and length is normalized by $d$ (cylinder diameter), unless specified otherwise. The sections at which various results are plotted are shown in Figure 4.

Initially a 2D steady RANS simulation using the SST $k-\omega$ model was carried out to determine the values of parameters $k$ and $\omega$. Using the values of $k$ and $\omega$, the Taylor scale of turbulence ($\Lambda = k^{1/2}/\omega$) was calculated, which was further used to determine the value of $f_k$ based on the expression given by the Girimaji and Khaled (2005)
$f_k(x) = \frac{1}{\sqrt{C_\mu}} \left( \frac{\Delta}{\lambda} \right)^2 \approx 2 \left( \frac{\Delta}{\lambda} \right)^{1.3}$  (28)

$f_k$ comes out to be 0.6, and $f_i$ is taken as unity considering all the dissipation occurs at the smallest scales of motion which are to be modeled. But a posteriori validation is necessary to check whether the considered value of $f_k$ will be able to produce the necessary physical system based on the imposed theoretical filter. As suggested by Lakshmipathy and Girimaji (2007) the recovery of turbulent viscosity ratio by PANS from the parent RANS model is a good parameter to check the amount of physical filtration achieved. We know that for RANS modeling $v_r = C_p \frac{k^2}{\varepsilon}$ and similarly for PANS approach $v_r = C_p \frac{k^2}{\varepsilon_s}$. Therefore the viscosity recovery is given by the expression $v_r = \frac{f_k}{f_i}$.

If $f_i$ is considered to be unity. Thus to conduct this posteriori validation, three values of $f_k$ (0.8, 0.6 and 0.4) were considered for the PANS simulations. The probability density function (PDF) of the viscosity recovery ratio were then calculated for all three PANS simulations based on the 2D RANS simulation carried out initially. Figure 5 shows the PDF plots of the viscosity ratio for different $f_k$ values. It can be observed that for $f_k = 0.6$ the plot shows peak nearly at 0.37 which is quiet close to $f^*_k$. For $f_k = 0.8$ and 0.4 the peaks achieved are at approximately 0.72 and 0.24 respectively. These values do not correspond to the desired value of the viscosity recovery ($f^*_k$). From this discussion it can be said that for the present grid system the value of $f_k = 0.6$ is able to achieve the required physical behavior for the flow field concerned.

Now the validation of the flow behavior is to be done to test the current modeling strategy and the open source code used to carry out the simulations. The first major check for any computational study on bluff bodies is an assessment of the integral properties, such as, Strouhal number ($St$), mean drag coefficient ($C_D$), and root-mean-square (r.m.s.) lift coefficient ($C_L$). Many experimental studies are available for the similar $Re$ range and these are tabulated in Table 2. It can be observed from Table 2 that the values of $St$ for square cylinder in a cross flow at $Re = 22000$ ranges from 0.12 to 0.138. Figure 3a shows a recorded pressure signal. Subsequently a Fourier-transform of this instantaneous pressure signal is performed and the frequency is normalized by $D/U_o$ according to the definition of $St$. Figure 3b shows the spectra of $St$, which shows that there is only one dominant peak which corresponds to the value of 0.129. This value is well within the range of that observed experimentally. Similarly the values of ($C_D$)$_{max}$ and ($C_L$)$_{max}$ are also in the same range as given by the Lyn et al. (1995) and LES predictions of Boileau et al. (2013).

Further the present results of the flow dynamics quantities are also compared with the experimental data of Lyn et al. (1995). In Figure 6, the streamwise variation of the time-averaged and
fluctuating velocities are plotted at the center-line ($y = 0$). The experimental results of Lyn et al. (1995), Durao et al. (1988) and LES results of Boileau et al. (2013) are also included in Figure 6 for comparisons. From Figure 6a, a strong variation in the mean flow field is observed near the cylinder base region, which is mainly because of strong interaction between the separated shear layers from the top and bottom walls of the cylinder. In the near wake region further downstream ($x > 3$), the variation in the mean flow is quite small. Similar to LES results the current model also over predicts the velocity for most of the downstream regions, showing faster recovery of the streamwise velocity after the separation. At regions close to the cylinder ($x = 2$), as the streamwise velocity is overpredicted, a shorter recirculation length compared to the experimental values is observed (Table 2).

It can be observed from Figure 6a that in the present results and those by Lyn et al. (1995), the recovery of the mean velocity at the centerline is slow as compared to that observed by Durao et al. (1988). This deviation can be attributed to a larger wake region formed in the present case at high $Re$ (22000) than that observed by Durao et al. (1988) in their experiments conducted at $Re = 14000$.

Figure 6b shows a variation of the time-averaged velocity fluctuations in the streamwise and transverse directions at the centerline. It can be seen that both streamwise and transverse velocity fluctuations are very well within the experimental range of Lyn et al. (1995). Figure 6b shows higher values of $\bar{v}$ than those of $\bar{u}$ in the wake region, at all $x$ values, with a maximum value of approximately 0.9, which is due to the strong recirculation in the separation region. The current PANS results show that the transverse fluctuations match with the experimental data only in the near wall region ($x < 2$) and are generally overestimated farther downstream in the wake region. Based on Figures 6a and 6b and above discussions it can be concluded that the current PANS modeling strategy accurately predicts the mean flow properties well in accordance with the experimental and LES results. This shows the capability of the current PANS strategy to predict accurate results for the present flow configuration at a low computational cost.

To observe further dynamic features of the flow field iso-surface of the Q-criterion is plotted, as it represents the Kelvin-Helmholtz ($K-H$) instabilities. From Figure 7 it is clear that the PANS SST $k-\omega$ model is able to capture these $K-H$ instabilities originating from the leading edge of the square cylinder, but it fails to exhibit a fully 3D turbulent wake in the downstream, as these instabilities are damped and the large flow structures become two-dimensional. A similar trend can also be observed with the calculated dynamic viscosity ratio. As can be seen from Figure 8, in accordance with the

<table>
<thead>
<tr>
<th>Case</th>
<th>Re</th>
<th>St</th>
<th>$\langle \omega \rangle_{\text{mean}}$</th>
<th>$\langle \omega \rangle_{\text{rms}}$</th>
<th>$\bar{u}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Durao et al. (1988), EXP</td>
<td>22000</td>
<td>0.133</td>
<td>-</td>
<td>-</td>
<td>1.33</td>
</tr>
<tr>
<td>Lyn et al. (1995), EXP</td>
<td>14000</td>
<td>0.126-0.132</td>
<td>2.1</td>
<td>-</td>
<td>1.38</td>
</tr>
<tr>
<td>Boileau et al. (2013), LES</td>
<td>22000</td>
<td>0.13</td>
<td>2.11</td>
<td>1.37</td>
<td>1.25</td>
</tr>
<tr>
<td>Wiesche (2007), LES</td>
<td>22000</td>
<td>0.19</td>
<td>-</td>
<td>-</td>
<td>1.15</td>
</tr>
<tr>
<td>Present, PANS</td>
<td>22000</td>
<td>0.129</td>
<td>1.98</td>
<td>1.35</td>
<td>1.31</td>
</tr>
</tbody>
</table>

**Table 2 Global integrated flow parameters.**

![Fig. 7. Iso-surface of the Q-criterion ($Q=\text{100}x^2$) colored by dynamic viscosity ratio.](image1)

![Fig. 8. Dynamic viscosity ratio distribution.](image2)
above-mentioned Q-criterion iso-surface, the vortex shedding is very regular and the spatial stretching of these vortices leads to large coherent quasi 2D vortices. The reason for the damping of 3D turbulent wake can be attributed to large grid size used for the simulations.

Similarly results are also compared with the experimental data in the base region \((x = 1)\) in the cross-stream direction. Figure 9a(e) shows that a good agreement with the experimental results is achieved by the present PANS SST \(k-\omega\) model for the time-averaged values of both mean and fluctuating velocities in the transverse direction. The pronounced blockage effect caused by the square cylinder is clearly visible in Figure 9a showing the mean streamwise velocity which attains a maximum value close to 130\% of the free stream velocity. Further the plot of \(\bar{u}v\) at \(x = 1\) (Figure 9e) shows that the current modeling strategy captures both the peaks, in the opposite directions, one in the shear layer of the wall \((y \approx 0.72)\) and other near the center region of recirculating region \((y \approx 0.15)\) as also observed by the experimental results of Lyn et al. (1995).

So far the time-averaged flow properties were considered and the PANS modeling provided good predictions for both mean and fluctuating flow in both the streamwise and transverse directions. The PANS method also needs to be assessed for the energy equation in the present study by using the formulation given by Boileau et al. (2013). A more detailed comparison of the time-averaged local Nusselt number profile around the square cylinder with experiment data is shown in Figure 11. LES results of both the strategies (wall resolved and standard wall functions, termed as LES-WR and LES-WF, respectively) used by Boileau et al. (2013) are also considered to show the effectiveness of the current modeling strategy. From Figure 11 it can be observed that LES-WR method provides better results as compared to its LES-WF and the present PANS method. As already discussed in Section 1, the near wall treatment for the energy equation is done in the present study by using the formulation given by Kader (1981) which is also applicable in the viscous shear layer.

It can be observed from Figure 11 that the time-averaged values of both mean and fluctuating velocities in the transverse direction. The pronounced blockage effect caused by the square cylinder is clearly visible in Figure 9a showing the mean streamwise velocity which attains a maximum value close to 130\% of the free stream velocity. Further the plot of \(\bar{u}v\) at \(x = 1\) (Figure 9e) shows that the current modeling strategy captures both the peaks, in the opposite directions, one in the shear layer of the wall \((y \approx 0.72)\) and other near the center region of recirculating region \((y \approx 0.15)\) as also observed by the experimental results of Lyn et al. (1995).

4.1 Heat Transfer Characteristics

As already stated in the Introduction, only experimental results are available for the heat transfer from a square cylinder in a cross flow. Igarashi (1985) provided the correlation for global Nusselt number:

\[
\overline{Nu}_g = 0.14 \left( \frac{\frac{\mu_s}{\mu_{wall}}} {\frac{Re}{Re_{exp}}} \right)^{0.14} \quad \text{Re}^{0.66}
\]

(29)

This correlation is known to be an accurate representation of the average Nusselt number results for square cylinder in a cross-flow (Sparrow et al., 2004). Igarashi (1985) also measured local Nusselt number profiles around the cylinder for different Reynolds number, but as Boileau et al. (2013) pointed out that none of these Reynolds number corresponds to the current case \((Re = 22000)\). Therefore Boileau et al. (2013) scaled two Nusselt number profiles, corresponding to the closest Reynolds number \((Re = 18500\) and 29600\), using the correlation given by Equation (29) as

\[
\overline{Nu}_g = \overline{Nu}_{exp} \left( \frac{\frac{\mu_s}{\mu_{wall}}} {\frac{Re}{Re_{exp}}} \right)^{0.14} \quad \text{Re}^{0.66}
\]

(30)

and the local Nusselt number profile around the cylinder is calculated at \(Re = 22000\).

Based on the effectiveness of the current modeling strategy for the flow dynamics shown in the preceding section, its capacity to predict convective heat transfer is also evaluated here. Table 3 shows the values of the global Nusselt number \(\overline{Nu}_g\) obtained by the space averaging the local Nusselt number profile around the square cylinder. It is observed that the current PANS model is more close to the experimental result than the LES with wall functions used by Boileau et al. (2013).
Fig. 9. Comparison of transverse profile \((x = 1)\) of time–averaged streamwise and transverse velocity, streamwise and transverse velocity fluctuations and velocity correlations respectively. \(\bullet\) Lyn et al. (1995) experimental, \(\text{Present PANS})\.

Even though the wall function approach associated with PANS model is not able to predict the thermal behavior quantitatively, it can still be used to get an idea of the unsteady heat transfer mechanism from the square cylinder in the cross flow. The unsteady characteristics are studied by phase-averaging the Nusselt number profile and then superimposing it with streamlines (Figure 12).

Figure 13 shows the phase-averaged quantities for different phases of the periodic flow. The \(\overline{Nu}\) is higher at the front face than that at the top and bottom faces, because of the regular sweeping of near wall flow by the incoming cold flow, irrespective of the phase. At the rear face, due to continuous oscillation, the cold fluid is entrapped from the outer shear layer, thus the Nusselt number...
Fig. 10. Variation of \( k \) (● experimental, —— computational), \( \langle u'v' \rangle \) (● Experimental, —— computational), at two different streamwise directions, (a) \( x = 2 \) and (b) \( x = 6 \).

Fig. 11. Time-averaged local Nusselt number around the cylinder.
is higher than that at the front face and it shows a small variation with the phase. Apart from it, the flow is highly turbulent at the rear face, which further increases the Nusselt number. In the case of top and bottom faces, the variations of Nusselt number are out of phase, which is because of the transient flow of the shear layer. The value of the Nusselt number at these faces are smaller compared to those at both front and rear faces as during most of the phases the flow is entrapped in the recirculating shear regions on the faces. These recirculating regions do not interact much with the outer layer for most of the flow cycle.

Figure 14 shows the contours of the time-averaged turbulent heat flux, which can be another way to study the heat transfer. From Figure 14 it is clear that the value of the heat flux on both top and bottom faces in both directions is low due to which the heat transfer from these surfaces is small as compared to that at the rear face, which is also shown in Figures 10 and 11. At the rear corners of cylinder, strong heat flux $\langle \overrightarrow{q} \rangle$ value is observed due to which fresh fluid is induced towards the core region of the rear face which as a result increases the heat transfer. This observation also supports the discussions made above based on the streamlines.

Thus, the present study shows that the wall function used for temperature equation provide a fair idea of the unsteady heat transfer but fail to quantify it compared to the experimental observations. This is because of the complexity involved in the flow physics, which is not considered in the wall function approach.
5 CONCLUSIONS

An unsteady computational analysis of flow and heat transfer around a square cylinder at Re = 22000 is carried out using a variable resolution model strategy, PANS. The PANS SST $k$-$\omega$ model is derived and implemented in an open source code Open FOAM to carry out the simulations. The flow dynamics and heat transfer results have been compared with the experimental data reported in the literature. The heat transfer results are also compared with LES predictions reported in the literature. The computational results predicted by the present model concerning flow properties are well in accordance with the experimental data. To simulate thermal behavior, energy equation is solved with the PANS SST $k$-$\omega$ model. For the near wall treatment of the heat flux, a wall function is used which is valid for all values of $y^+$. The results show that the wall function approach is not able to accurately predict the time-averaged value of Nusselt number. But because of accurate predictions of flow dynamics, the blend of wall function for heat flux and PANS SST $k$-$\omega$ model, is able to predict the main features associated with the unsteady heat transfer. Therefore it can be concluded that the present PANS approach is very promising for flow dynamic predictions associated with complex industrial flows, as it is less expensive in terms of computational power. However for the prediction of heat transfer in such complex situation there is need for an improved scalar modeling and blend it with PANS method to obtain more accurate predictions.

REFERENCES


