Evaluation of Partially Averaged Navier-Stokes Method in Simulating Flow Past a Sphere

S. Saroha\textsuperscript{1†}, S. S. Sinha\textsuperscript{1}, and S. Lakshmipathy\textsuperscript{2}

\textsuperscript{1} Applied Mechanics Department, IIT Delhi, 110016, India
\textsuperscript{2} Gexcon AS, Bergen, 5072, Norway

\textsuperscript{†} Corresponding Author Email: sagarsaroha18@gmail.com
(Received January 4, 2018; accepted April 15, 2018)

ABSTRACT

In recent past partially averaged Navier-Stokes equation (PANS) has been proposed as a scale-resolving bridging method for turbulence computations. Despite the geometric simplicity of the involved boundary conditions, the flow past a sphere is ripe with various complex flow phenomena, which make it an excellent test bed to evaluate various computational fluid dynamics modelling methodologies – both in terms of numerical schemes as well as turbulence models. Specifically, in this work we evaluate PANS in conjunction with the standard $k$-$\varepsilon$ model in terms of (i) influence of filter parameters, (ii) sensitivity to free stream viscosity ratio and (iii) choice of numerical schemes at supercritical Reynolds number of $1.14 \times 10^{6}$. Careful evaluations are made by comparing PANS results against available experimental data as well available detached eddy simulation (DES) and large eddy simulation (LES) results. Our study finds that indeed – as purported by the PANS theory – a reduction in the value of the first filter parameter ($f_k$) successfully captures the complex vortical structures that exist past a sphere, shows far superior performance than unsteady Reynolds-averaged Navier-Stokes (URANS) simulations and somewhat improved performance even over some of the LES studies reported in literature. Our study shows that in terms of most of the quantities of interest, PANS performance is almost at par with that of DES.

Keywords: Computational Study; Scale-resolving methods; High Reynolds number; Flow past a sphere.

NOMENCLATURE

- $C_D$: drag coefficient
- $C_p$: pressure coefficient
- $C_f$: skin friction coefficient
- $F_x$: force on sphere along streamwise direction
- $\tau_w$: wall shear stress
- $\Omega_{ij}$: rotation-rate tensor
- $S_{mn}$: strain-rate tensor
- $\rho$: density of fluid
- $V_\infty$: freestream velocity of fluid
- $V$: velocity of fluid
- $A$: projected area of sphere
- $D$: sphere diameter
- $P$: pressure
- $p_\infty$: freestream pressure

1. INTRODUCTION

Despite the simple geometry of a sphere, flow past it at high Reynolds number (Re) is quite complex, involving the concurrence of several complicated flow phenomena: large-scale unsteadiness and Reynolds number-dependent three-dimensional vortex shedding patterns multiple instability modes, laminar to turbulent transition and massive separation [Achenbach (1974), Drikakis (1995), Jindal et al. (2004)]. Co-existence of these multiple flow phenomena has not only motivated many experimental studies [Achenbach (1972), Achenbach (1974), Taneda (1978)], it has also made the flow past a sphere a unique and one of the most demanding benchmark cases for testing computational fluid dynamic methods - in terms of both the discretization schemes as well as turbulence models [Constantinescu and Squires (2000),...
Constantinescu and Squires (2004). The focus of this work is to evaluate the so-called partially averaged Navier-Stokes (PANS) method of turbulence in simulating flow past a sphere at adequately high Reynolds number \((1.14 \times 10^9)\) when all the aforementioned flow features are known to exist.

Application of direct numerical simulation (DNS) for flow past a sphere has been restricted to sub-critical Reynolds number only. Shirayama and Kuwahara (1992), Gebing (1992), Aliabadi and Tezduyar (1995), Shen and Loc (1997), Kalro and Tezduyar (1998), Mansoorzadeh et al. (1998) and Johnson and Patel (1999) have performed DNS studies with Reynolds number up to \(10^3\). Seidl et al. (1997) performed DNS studies at \(Re = 5000\); Kim and Choi (2001) and Bazilevs et al. (2014) has performed their study at \(Re = 10^4\).

Despite its reasonable acceptability in several other flows of practical interest, Reynolds-averaged Navier-Stokes (RANS) and even its unsteady counterpart - Unsteady RANS (URANS) have been found to be severely inadequate for simulating flow past a sphere. Drikakis (1995) performed simulation of flow past a sphere at \(Re_0 = 10^2\) and 106 using URANS in conjunction with the k-\(\varepsilon\) model. While the pressure distribution upstream of the separation point was found to be adequate, the same in the post-separation regime shows large errors. Further, the URANS method could not capture any unsteadiness at all. Based on their studies, Tomboulides and Orszag (2000) concluded that due to the reduced coherence of shed structures in flow past a sphere (as compared to a cylinder), the resolution of finer flow features are essential to correctly capture the vortex shedding and three dimensionality which dominate the flow behaviour past a sphere. Since it indiscriminately suppresses all fine scale unsteadiness, irrespective of the choice of turbulence model, URANS has been judged to be inherently unsuitable for simulating various features of the flow past a sphere (Constantinescu and Squires 2003b).

In recent years, researchers have attempted to address the gap between DNS and URANS by using scale-resolving methods or filter-based methods for simulating the flow past a sphere. Two such methods - large eddy simulation (LES) and detached eddy simulation (DES) - have been employed by several workers to simulate the flow past a sphere at high Reynolds numbers. LES uses a filter cutoff: all scales below that cutoff scale are modelled, whereas scales larger than that are solved directly (resolved). The cutoff length scale in LES is typically chosen near the dissipation range of the spectrum so that the burden and the uncertainty incurred while modeling the unresolved motion is meagre. This allows for the application of simple Samargorinsky-like algebraic (zero-equation) closures. However, given that the cutoff length scale in LES is close to the typical dissipation range, the reduction in computational burden in LES as compared to DNS is limited. A part of this problem is addressed by the so-called detached eddy simulation method wherein the formulation behaves like LES in the bulk of the flow and switches to RANS in the near-wall zones. DES uses the Spalart-Allmaras (SA) model in the near-wall region in conjunction with the RANS equations. By switching to RANS variables in the near-wall region DES avoids the burden of resolving small scale motion in the near-wall region (thus reducing the burden of having a very fine mesh there) and at the same time, uses the capability of LES to reliably resolve wider scales of motion in regions away from walls. Tomboulides et al. (1993) have performed LES simulations of flow past a sphere at \(Re_0 = 2 \times 10^4\). They used a subgrid model based on the renormalization group theory. Several other LES-based studies have been performed by Kim and Durbin (1988), Kim and Choi (2001), Schmidt (2002), Jindal et al. (2004), Constantinescu and Squires (2003a), Constantinescu and Squires (2003b) have employed and validated the DES method in flow past the sphere over a range of Reynolds number \((10^5-10^6)\). Based on our literature review, we deem the DES results of Constantinescu and Squires (2004) to be the state-of-the-art numerical results for the flow past a sphere at Reynolds number of \(1.14 \times 10^9\).

Partially averaged Navier-Stokes (PANS) method has emerged as a new promising paradigm of turbulence computation. While PANS, like LES and DES, is a filter-based or a scale-resolving method, it is purported to be more versatile than both LES and DES. Girimaji (2006) proposed PANS as a bridging method which uses two implicit filtering parameters: \(f_I\) and \(f_\varepsilon\). The parameter \(f_\varepsilon\) is ratio of the unresolved kinetic energy in PANS to the un-resolved kinetic energy in a corresponding URANS simulation, whereas \(f_I\) is the ratio of unresolved dissipation-rate in PANS to that in URANS simulation (more details in Section 2). Unlike the LES or DES in which the filter cutoff is typically placed near the dissipation range, PANS allows the user to change the filter parameter seamlessly, encompassing large to inertial to dissipative range of motion. Setting \(f_\varepsilon = 1\) and \(f_I = 1\) makes PANS equivalent to RANS, whereas \(f_\varepsilon = 0\) and \(f_I = 1\) makes PANS equivalent to DNS. With this flexibility, depending on the size of the grid that the user can afford to have, the filtering parameter can be chosen anywhere between the two extremes of RANS and DNS. Further, it is argued that unlike DES and other hybrid RANS/LES methods, PANS is a superior scale-
resolving method because its formulation is completely consistent with the averaging invariance property of the Navier-Stokes equation [Germano (1992), Girimaji (2006), Suman and Girimaji (2010)].

In recent years, PANS method has been evaluated in a variety of both canonical and practical flows of interest and the performance has been found to be quite encouraging. To further evaluate the performance of PANS method, in this work we subject PANS to its (perhaps) the most severe test: massively separated three-dimensional flow past a sphere at high Reynolds number. The authors believe that such an evaluation is essential to extensively benchmark the PANS method in extremely challenging flow environment (co-existence of large scale unsteadiness, massive three-dimensional flow separation, presence of several instability modes) and identify its shortcomings, if any. Naturally, it is expected that this critical examination may lead to further improvements of PANS. With this motivation, we examine the performance of PANS model in flow past a sphere specifically in incompressible flow,

\[ \frac{\partial V_i}{\partial t} + V_j \frac{\partial V_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 V_i}{\partial x_i^2} + \frac{1}{\rho} \frac{\partial^2 P}{\partial x_j \partial x_i}, \quad \text{(1)} \]

where \( V_i \), \( p \) and \( \nu \) represent instantaneous velocity, pressure and density. The symbol \( \nu \) is kinematic viscosity and the symbol \( t \) & \( x_i \) represent time & spatial coordinates. Equation 2 is the Poisson’s equation of pressure. This equation is obtained by subjecting the momentum equation to the divergence operator, and subsequently using the continuity equation for incompressible flow,

\[ \frac{\partial V_i}{\partial x_i} = 0. \]

PANS equations are obtained by applying a partial-averaging filter to the instantaneous incompressible flow equations. Correspondingly the filtered fields are \( < V_i > \) (= \( U_i \)) and \( < p > \) (= \( p_u \)), where \( < > \) represents the partial-averaging filter. Assuming that the filter is constant preserving, and that it commutes with spatial and temporal derivative operators [Girimaji (2006)], the governing equations of \( U_i \) and \( p_u \) obtained are:

\[ \frac{\partial U_i}{\partial t} + U_j \frac{\partial U_i}{\partial x_j} + \frac{\partial \tau(V_i,V_j)}{\partial x_j} = \]

\[ \frac{1}{\rho} \frac{\partial p_u}{\partial x_i} + \nu \frac{\partial^2 U_i}{\partial x_i^2}; \quad \text{(3)} \]

\[ 1 \frac{\partial^2 p_u}{\partial x_j \partial x_i} = - \frac{\partial U_i}{\partial x_j} \frac{\partial U_j}{\partial x_i} - \frac{\partial^2 \tau(V_i,V_j)}{\partial x_j \partial x_i}. \quad \text{(4)} \]

The quantity \( \tau(V_i,V_j) \) is called the generalized central moment of \( V_i \) and \( V_j \) in accordance with the definition of Germano (1992):

\[ \tau(V_i,V_j) = < V_i V_j > - < V_i > < V_j >. \quad \text{(5)} \]

In Eqs. (3) and (4), \( \tau(V_i,V_j) \) is the unclosed quantity. In this work, we employ the PANS version of Boussinesq eddy viscosity assumption [Girimaji (2006)] to tackle this:

\[ \tau(V_i,V_j) = \frac{2}{3} \kappa \delta_{ij} - 2 \nu_s S_{ij}; \quad \text{(6)} \]

where \( \nu_s \) is unresolved eddy viscosity and \( S_{ij} \) is the filtered strain-rate: \( S_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \).

The unresolved eddy viscosity \( (\nu_s) \) is related to unresolved kinetic energy \( (k_u) \) and unresolved

---

**2. PARTIALLY AVERAGED NAVIER-STOKES**

In this section, we provide a brief overview of PANS methodology and its purported superiority over URANS method. For full details of the PANS methodology, the reader is referred to Girimaji (2006).

We begin with the instantaneous Navier-Stokes equations for an incompressible flow:

---

This paper is organized into six sections. In section 2 we provide a brief review of PANS methodology and list the governing equations. In section 3 we present details of our computational set-up. In section 4 we present the plan of our study. In section 5 we present our results and pertinent discussion. Section 6 concludes the paper with a summary.

---

dissipation-rate ($\epsilon_u$) as: $\epsilon_u = C_v \frac{k^2}{\varepsilon_u}$, where $C_v$ is a model coefficient. These quantities $k_u$ and $\varepsilon_u$ can be expressed as generalised central moments:

$$k_u = \frac{1}{2} \tau(V_i, V'_i), \quad \varepsilon_u = \nu \left( \frac{\partial V_i}{\partial x_j} \frac{\partial V'_i}{\partial x_j} \right).$$

Finally, the closure is achieved by including the evolution equations of $k_u$ and $\varepsilon_u$. Using $f_k$ and $f_\varepsilon$ (the filter control parameters of PANS), the modeled evolution equation of $k_u$ and $\varepsilon_u$ are:

$$\frac{\partial k_u}{\partial t} + U_j \frac{\partial k_u}{\partial x_j} = P_u - \varepsilon_u + \frac{\partial}{\partial x_j} \left( \frac{\nu_u + \nu}{\sigma_{uu}} \frac{\partial k_u}{\partial x_j} \right),$$

$$\frac{\partial \varepsilon_u}{\partial t} + U_j \frac{\partial \varepsilon_u}{\partial x_j} = \int \left( C_{\varepsilon 2} \frac{\rho u}{k_u} - C_{\varepsilon 2} \frac{\nu'_u}{k_u} \right) + \frac{\partial}{\partial x_j} \left( \frac{\nu_u + \nu}{\sigma_{\varepsilon u}} \frac{\partial \varepsilon_u}{\partial x_j} \right),$$

where, $f_k = k/k$ and $f_\varepsilon = \varepsilon_u/\varepsilon$; $k$ and $\varepsilon$ are turbulence kinetic energy and its dissipation rate. The quantity $P_u$ is production of unresolved kinetic energy: $P_u = \tau(V_i, V'_i) \frac{\partial U_j}{\partial x_j}$; the symbols $\sigma_{uu}$ and $\sigma_{\varepsilon u}$ are turbulent prandtl numbers for unresolved kinetic energy and dissipation rate.

The modified model coefficients appearing in Eqs. (7) and (8) are (Girimaji and Abdol-Hamid 2005):

$$C_{\varepsilon 2}^* = C_{\varepsilon 1} + \frac{f_k}{f_\varepsilon} (C_{\varepsilon 2} - C_{\varepsilon 1});$$

$$\sigma_{k_u} = \sigma_k \frac{f_k^2}{f_\varepsilon}, \quad \sigma_{\varepsilon u} = \sigma_\varepsilon \frac{f_k^2}{f_\varepsilon}.$$  

Other model coefficients hold the same values as in standard $k$-$\varepsilon$ model. $C_{\varepsilon 1} = 1.44, C_{\varepsilon 2} = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.3$. Equations (3), (4), (6), (7) and (8) form a closed set of PANS equations employing standard $k$-$\varepsilon$ model and the Boussinesq eddy-viscosity approximation.

The value of $\nu_u$ can be controlled by choosing apt values of filter-width control parameters. Choosing a sub-unity value of $f_k$ reduces the production of turbulent kinetic energy, which in turn can reduce $\nu_u$, allowing for more scales to be released and resolved. On the other hand, reducing the value of $f_\varepsilon$ can reduce the dissipation-rate of turbulence kinetic energy, which in turn has the opposite effect of leading to larger accumulation of kinetic energy in the flow domain and consequently suppressing unsteadiness in the flow field. These two control parameters can be used to adjust the overall level of $\nu_u$ so that a PANS simulation can generate and resolve a range of scales which is commensurate with the fineness of the grid specified by the user.

### 3. COMPUTATIONAL SET-UP

The computational domain employed in this study is a cylindrical volume with diameter being 10 times and the length being 15 times the sphere diameter (D). The centre of the sphere is located at 5D from the upstream boundary. A schematic of the domain is shown in Fig. 1.

**Fig. 1. Schematic of Computational Domain**

For our simulations we employ an unstructured mesh consisting of prism layers and tetra elements similar to that used by Bazilevs et al. (2014). The height of the prism layer is chosen such that the boundary layer is resolved within prism elements. The $y^+$ value of the first node in near wall region is ensured to be sub-unity. Density boxes are used to control element sizing around the sphere and in the wake region. The total mesh element count is 1.8 million. The count of prism cells is 0.5 million and the rest are tetra elements. Figure 2 shows screenshots of the mesh.

**Fig. 2. 2D slice (Z=0 plane) of the mesh used for computations**

The solver used in this study is ANSYS FLUENT, which is industry-standard for internal/external fluid flows and flows involving heat transfer. FLUENT facilitates appropriate...
modification of turbulence model coefficients in a relatively simpler way, thus enabling implementation of PANS methodology [Eqs. (3), (4), (6), (7) and (8)].

For near-wall treatment, we employ the two-layer approach developed by Chen and Patel (1988). The near-surface region is divided into a viscosity-affected layer and a fully-turbulent region. Demarcation of the two regions and the length scale in the inner layer is determined using turbulent Reynolds number. In the viscosity affected inner layer, one equation model is solved retaining the momentum and kinetic energy Eq. (7). In fully turbulent/outer region, complete two-equation model is employed with active dissipation rate Eq. (8). Since instead of using the wall function approximations, one equation model is solved in inner layer, the value of first near-wall node is ensured to be near unity (Chen and Patel 1988).

In this study, results from several simulations are presented. In all these simulations, no-slip boundary and no-penetration conditions are specified at the sphere surface and the free-slip boundary condition is specified at lateral walls with zero shear stress. At the exit/outlet of the computational domain outflow boundary condition is used. Inlet boundary conditions are specified in terms of the velocity and turbulence variables. The free stream turbulence condition at the inlet of the domain is parameterized in terms of the viscosity ratio ($\eta$) using following relations [Constantinescu and Squires (2003b), Han et al. (2012)]:

$$\eta = \frac{v_{\infty} \nu}{\nu u} = \frac{C_{\mu} k^2}{\nu u} = \frac{3}{2} \left( \nu' I \right)^2;$$

$$\epsilon_u = \frac{C_{\mu} k^2}{\nu u} \frac{1}{\eta}.$$ (11)

Table 3 Computational Settings

<table>
<thead>
<tr>
<th>Solver</th>
<th>Pressure-based</th>
</tr>
</thead>
<tbody>
<tr>
<td>turbulence model</td>
<td>standard $k-e$ model</td>
</tr>
<tr>
<td>wall treatment</td>
<td>two layer treatment</td>
</tr>
<tr>
<td>Pressure-Velocity coupling</td>
<td>SIMPLE</td>
</tr>
<tr>
<td>gradient</td>
<td>least square cell based</td>
</tr>
<tr>
<td>pressure</td>
<td>second order</td>
</tr>
<tr>
<td>momentum, TKE, TDR</td>
<td>BCD/ SOU/ MUSCL</td>
</tr>
</tbody>
</table>

where $V_{\infty}$ is free stream velocity, $l$ is turbulent intensity, $C_{\mu}$ is model coefficient from standard $k-e$ model (0.09), $\nu$ is laminar viscosity and $v_u$ is unresolved turbulent viscosity. Interior of the domain is initialized with the value of turbulence kinetic energy and dissipation rate picked from the inlet.

In Table 3, we present a summary of the solver settings employed in all our simulations. We have used a pressure-based solver. Pressure-velocity coupling is addressed employing the SIMPLE algorithm. For convective terms in the momentum equation one of the following schemes - third order MUSCL, bounded central difference (BCD) or second order upwind (SOU) - is employed. Other gradient terms are discretized using the least square cell based method. For temporal discretization bounded second order fully implicit method is used.

We perform our simulations at $Re = 1.14 \times 10^6$, which is in the supercritical regime of flow past a sphere. In supercritical regime the flow is already turbulent in boundary layer before separation happens. Since the two-layer $k-e$ model estimates length scale in near-wall region using turbulent Reynolds number, it is inappropriate for prediction of flow involving laminar separation. Thus, we assume the boundary layer to be turbulent throughout. Simulations with this assumption have been performed by other workers as well [Constantinescu and Squires (2003b), Jones and Clarke (2008)]. For sensible comparisons, we employ relevant experimental data with fully turbulent boundary layer, which is available at Reynolds number of $1.14 \times 10^6$ (Achenbach 1974).

Table 4 Simulation Cases

<table>
<thead>
<tr>
<th>Case</th>
<th>Scheme</th>
<th>$f_i$</th>
<th>$f_e$</th>
<th>$v_u/v_{\infty}$</th>
<th>CFL</th>
</tr>
</thead>
<tbody>
<tr>
<td>X01</td>
<td>MUSCL</td>
<td>0.5</td>
<td>1.0</td>
<td>10</td>
<td>4</td>
</tr>
<tr>
<td>X02</td>
<td>MUSCL</td>
<td>0.5</td>
<td>1.0</td>
<td>10</td>
<td>2</td>
</tr>
<tr>
<td>X03</td>
<td>MUSCL</td>
<td>0.5</td>
<td>1.0</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>X04</td>
<td>SOU</td>
<td>0.5</td>
<td>1.0</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>X05</td>
<td>BCD</td>
<td>0.5</td>
<td>1.0</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>X06</td>
<td>MUSCL</td>
<td>0.5</td>
<td>1.0</td>
<td>100</td>
<td>1</td>
</tr>
<tr>
<td>X07</td>
<td>MUSCL</td>
<td>0.5</td>
<td>1.0</td>
<td>100</td>
<td>1</td>
</tr>
<tr>
<td>X08</td>
<td>MUSCL</td>
<td>0.7</td>
<td>1.0</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>X09</td>
<td>MUSCL</td>
<td>1.0</td>
<td>1.0</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>X10</td>
<td>MUSCL</td>
<td>0.5</td>
<td>0.5</td>
<td>10</td>
<td>1</td>
</tr>
<tr>
<td>X11</td>
<td>MUSCL</td>
<td>0.7</td>
<td>0.7</td>
<td>10</td>
<td>1</td>
</tr>
</tbody>
</table>

4. PLAN OF STUDY & QUANTITIES OF INTEREST

For this study, in total 11 different simulations (X01-X11) have been performed. Table 4
presents an overview of the variations in the simulation settings. All these simulations have been performed on the same grid which is described in Section 3.

As outlined in the introduction (Section 1), we intend to comprehensively evaluate the performance of PANS method in simulating the flow past a sphere in terms of (i) Choice of discretization scheme, (ii) Influence of \( f_k \), (iii) Influence of \( f_\epsilon \) and (iv) Influence of free stream viscosity ratio. Evaluation of the PANS method in terms of each of these parameters is important in its own way.

Scale-resolving methods are well-known to show sensitivity to the discretization schemes. Constantinescu and Squires (2003a) showed that the results of both LES and DES, despite having adequately refined grid and time steps, change considerably with change of discretization scheme.

Ascertaining the extent of influence of the discretization scheme on PANS simulation of the flow past a sphere is thus important. Toward this objective, we evaluate the performance of the three advanced discretization schemes available in ANSYS FLUENT: MUSCL, second order upwind (SOU) and bounded central difference (BCD) scheme. In Section 5b, we present a comparative study of the results from simulations X03, X04 and X05 to evaluate the influence of the discretization scheme on PANS results. While simulation X03 uses MUSCL scheme; X04 and X05 employ SOU and BCD scheme respectively. Except for the discretization scheme, other parameters of these three simulations are identical.

Turbulent features of incoming freestream flows are often not completely characterized in many engineering flow scenarios. CFD simulations of such flow fields are often done with a user-assumed turbulent viscosity ratio. This could prove to be a major source of uncertainty in the final CFD results, if the turbulence model/computational methodology has significant sensitivity to the inlet viscosity ratio. This problem has been observed especially in unsteady RANS simulations of massively separated flows (Han et al. 2012). Thus, in order to minimize the overall uncertainty of CFD simulations, it is desirable to have a certain level of robustness in computational methodology and an ensuing low sensitivity to the chosen inlet viscosity ratio and indeed PANS is also an unsteady simulation paradigm, its closure models are extracted from existing RANS models. Thus, it is quite appropriate that PANS methodology is also carefully evaluated in terms of sensitivity to the freestream turbulent viscosity ratio. Flow past a sphere, with its inherently unsteady and massively separated flow field, makes an excellent test bed for such a numerical experiment. With this motivation, in Section 5c we compare the results from simulations X03, X06 and X07. These simulations are identical except for the initial chosen value of the turbulent to laminar viscosity ratio (Eq. 11). To the best of authors’ knowledge, no such sensitivity study for PANS has been performed in context of any flow field so far.

The PANS method is supposed to have the ability of releasing more scales of motion as the value of \( f_\epsilon \) is reduced. We test whether this expected behaviour is indeed demonstrated by PANS while simulating a complex field flow such as that past a sphere. In Section 5d, we perform a systematic study to evaluate the influence of \( f_\epsilon \) on PANS simulation of flow past a sphere. For this study, results from simulations X03, X08 and X09 have been compared. These simulations differ only in terms of the value of filter parameter \( f_\epsilon \).

At high enough Reynolds number, wherein the turbulence kinetic energy spectrum is broad-based with a well-defined inertial range, dissipation rate can be expected to be predominately concentrated at the smallest scale of motion. For such a flow field, it is reasonable to set \( f_\epsilon = 1 \) in a PANS simulation. However, if the Reynolds number is not high enough, significant dissipation may occur in inertial range as well. This would necessitate a sub-unity value of \( f_\epsilon \) (Girimaji and Abdol-Hamid 2005). Since at the onset we don’t know if the value of \( Re = 1.14 \times 10^5 \) is high enough to set \( f_\epsilon = 1 \), in Section 5e we examine the influence of second filter parameter \( f_\epsilon \) on our PANS simulation. We perform this examination at two different values of \( f_\epsilon \). Simulations X03 and X10 examine the influence of \( f_\epsilon \) with \( f_\epsilon \) fixed at 0.5, whereas as simulations X08 and X11 examine the influence of \( f_\epsilon \) with \( f_\epsilon \) fixed at 0.7.

### 4.1 Quantities of Interest

In this paper, extensive evaluation of the results is performed in terms of: (i) time-averaged value of drag coefficient, (ii) mean pressure coefficient distribution, (iii) mean skin friction coefficient distribution, (iv) contours of viscosity ratio and (v) visualization of dominant flow structures in instantaneous flow fields. Each simulation is performed over an extended duration of time to ensure that the influence of initial condition on the flow field is removed and reliable flow statistics can be extracted by performing time-averaging. Each simulation is run for 60 \( D/V_\infty \) time units. Results of the first 30 \( D/V_\infty \) time units are discarded. Subsequently, time averaging is performed using the results of next 30 \( D/V_\infty \) time units.

The mean flow fields is evaluated in terms of parameters like mean drag coefficient, mean coefficient of pressure and mean skin friction coefficient:

\[
C_D = \frac{\langle F_x \rangle}{0.5 \rho V_\infty^2 A}, \quad C_p = \frac{\langle p \rangle - p_\infty}{0.5 \rho V_\infty^2}, \quad C_f = \frac{\langle \tau_w \rangle}{0.5 \rho V_\infty^2}
\]
where the overhead represents the time-averaged value of a quantity and $A$ represents reference area ($= \pi D^2/4$). The symbol $P_{\infty}$ represents freestream pressure.

The instantaneous flow field is studied using viscosity ratio contours and iso-surfaces of vorticity. The iso-surfaces are generated using so-called Q-criterion (Hunt et al. 1988), where:

\[ Q = \frac{\Omega^2_{ij} \Omega_{ij} - [S_{mn} S_{mn}]}{2} \]

$\Omega_{ij}$ is filtered rotation-rate tensor and $S_{mn}$ is filtered strain-rate tensor:

\[ \Omega_{ij} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} - \frac{\partial U_j}{\partial x_i} \right) ;
S_{mn} = \frac{1}{2} \left( \frac{\partial U_m}{\partial x_n} - \frac{\partial U_n}{\partial x_m} \right) \]

5. RESULTS AND DISCUSSION

This section is organized into five subsections. In subsection 5a we first present our time-step study. Subsequently, in subsections 5b, 5c, 5d and 5e we present our studies on the influence of discretization schemes, influence of freestream viscosity ratio, influence of filter parameter $f_k$ and influence of filter parameter $f_\varepsilon$, respectively.

5.1 Time-Step Convergence

Simulations X01, X02 and X03 are employed for time-convergence study. Corresponding CFL number of these simulations are 4, 2 and 1 respectively (based on the smallest grid dimension).

In Fig. 3, we show the mean pressure distribution, and in Fig. 4, we show the mean skin friction coefficient distribution obtained from simulations X01, X02 and X03. Further, results from experiments (Achenbach 1974) and DES results (Constantinescu and Squires 2004) are also presented. Over the range $0^\circ < \theta < 80^\circ$, there is no discernible difference between the results from these simulations -- neither in mean $C_D$ nor in mean $C_f$. In the range $100^\circ < \theta < 180^\circ$, some differences between the results of these simulations are observed, especially in the separated region ($120^\circ < \theta < 180^\circ$). However, the differences are not very significant. In fact, the difference between simulations X02 [CFL=2] and X03 [CFL=1] is smaller than that between X01 [CFL=4] and X02 [CFL=2].

In Table 5a, we present the value of overall drag coefficient. The experimentally measured value of $C_D$ at the chosen Reynolds number is 0.120. The difference in value of $C_D$ between simulation X01 and simulation X03 is 2.5% of experimental value, whereas the difference in CD values between simulation X02 and X03 is 0.83% of experimental value. Clearly, reduction in time-step reduces the error. Thus, it can be concluded that results have ceased to differ significantly with reduction in the time-step.

Based on these observations we deem the non-dimensionalized time-step of 0.0009 (CFL = 1) to be adequate for performing a PANS simulation with $f_k = 0.5$ on our grid. All other simulations in the paper employ the same time-step. To present our results in perspective, we have included experimental results (Achenbach 1974) and DES results (Constantinescu and Squires 2004) as well. Clearly, PANS results show gross disagreement with experimental case especially in terms of $C_f$ distribution. Indeed, our results show behaviour similar to what is observed in DES (Constantinescu and Squires 2004). Accurately capturing of variation of $C_f$ is a challenge for scale-resolving methods in general. The goal behind the comparisons presented in this subsection (Figs. 3, 4 and Table 5a) is to confirm the adequacy of time-step convergence only. Agreement/disagreement with experimental
and DES data is the subject of discussion for later subsections.

5.2 Influence of Discretization Schemes

To examine the influence of discretization schemes, we present results from simulations X03, X04 and X05. The third-order MUSCL, second order Upwind and second-order BCD schemes have been employed in simulations X03, X04 and X05 respectively. In each of these simulations, the filter-control parameters \( f_k \) and \( f_\varepsilon \) have been chosen as 0.5 and 1 respectively.

In Table 5b, we present the time-averaged value of overall drag coefficient from these simulations. Among the three discretization schemes, performance of BCD scheme shows the largest deviation from experimental results with its \( C_D \) value = 0.187, which is 55.83% overestimated relative to the experimental value of 0.120. With SOU scheme the performance significantly improves, however the best performance is observed in simulation X03 which employs MUSCL scheme. The \( C_D \) value computed using MUSCL scheme is 0.138 which is 15% different from that of experimental observation. Clearly, the performance of MUSCL scheme is much superior (\( C_D = 0.138 \)) to BCD (\( C_D = 0.187 \)) and marginally better than SOU (\( C_D = 0.141 \)) scheme.

Table 5b Discretization Scheme Study

<table>
<thead>
<tr>
<th>Case</th>
<th>( f_k )</th>
<th>( f_\varepsilon )</th>
<th>Scheme</th>
<th>( C_D )</th>
</tr>
</thead>
<tbody>
<tr>
<td>X03</td>
<td>0.5</td>
<td>1</td>
<td>MUSCL</td>
<td>0.138</td>
</tr>
<tr>
<td>X04</td>
<td>0.5</td>
<td>1</td>
<td>SOU</td>
<td>0.141</td>
</tr>
<tr>
<td>X05</td>
<td>0.5</td>
<td>1</td>
<td>BCD</td>
<td>0.187</td>
</tr>
<tr>
<td>LES (Kim and Choi 2001)</td>
<td></td>
<td></td>
<td></td>
<td>0.139</td>
</tr>
<tr>
<td>EXP (Achenbach 1974)</td>
<td></td>
<td></td>
<td></td>
<td>0.120</td>
</tr>
<tr>
<td>DES (Constantinescu and Squires 2004)</td>
<td></td>
<td></td>
<td></td>
<td>0.104</td>
</tr>
</tbody>
</table>

In the Table 5b, we have presented results from LES simulation (Kim and Choi 2001) and DES (Constantinescu and Squires 2004) as well. The LES results (Kim and Choi 2001), like PANS simulation, overestimate the \( C_D \) value. DES results, on the other hand underestimate mean \( C_D \). The absolute percentage of error of DES and LES results when compared to the experimental observations are 13% and 15% respectively. Clearly, PANS (simulation X03) performance is quite comparable to that of the LES and DES results.

In Fig. 5, we present the variation of \( C_f \) along the mid-plane of the sphere as a function of \( \theta \). Over the region of 0° < \( \theta < 60° \), results from simulations X03, X04 and X05 are indistinguishable, and these in turn are in excellent agreement with both DES as well as the experimental results. Over the region of 60° < \( \theta < 100° \), MUSCL results show better agreement with experimental and DES data compared to SOU and BCD scheme. Over the region of 100° < \( \theta < 180° \), all three schemes show departure from the experimental data. However, over this region MUSCL scheme still performs better than SOU and BCD, because the former shows good agreement with at least the DES results.

In Fig. 6, we present mean skin friction coefficient distribution. All numerical results (PANS as well as DES) show gross disagreement with experimental data of Achenbach (1974) (except over 0° < \( \theta < 20° \) & 100° < \( \theta < 120° \)). Prediction of \( C_f \) is closely linked to the wall treatment strategy adopted in a CFD solver. Accurate prediction of skin friction coefficient in flows with massive three-dimensional separation has not been conclusively addressed yet by the CFD community at large -whether in the paradigm of RANS or in the paradigm of scale resolving methods like LES, DES (Constantinescu and Squires 2003a). PANS, being a relatively newer method, thus, also suffers from such a shortcoming- as revealed in our simulations. We have used the two-layer k-epsilon strategy (Chen and Patel 1988) for our calculations, and our results show that this approach is inadequate in capturing the skin friction coefficient in three-dimensionally separated flow field past a sphere. A better wall treatment strategy-specifically designed for three dimensional separated regions is certainly required to improve prediction of \( C_f \) in a flow such as that past a sphere. Nonetheless, among the three PANS simulations, clearly MUSCL scheme seems to agree with the DES results closer than SOU and BCD scheme. Among the three schemes, performance of the BCD scheme is the worst, especially in the aft region where separated flow exists.

Fig. 5. \( C_f \) distribution [X03, X04 and X05]
Further evaluation of the schemes, is performed in terms of the vortical structures around the sphere. In Figs. 7a, 7b and 7c we present the iso-surfaces of $Q$ colored with contours of instantaneous axial velocity. At the Reynolds number of $1.14 \times 10^6$, Constantinescu and Squires (2004) report identification of horseshoe vortices. A comparison of the results from our three simulations clearly shows that the MUSCL scheme is the most successful in capturing well-defined horseshoe vortices in the wake. While simulation X04 (SOU) shows more diffused vortical structures in comparison to MUSCL scheme (simulation X03); BCD scheme (simulation X05) shows two vortex streets, which is grossly inconsistent with results observed in DES or in experiments [Constantinescu and Squires (2004), Achenbach (1974)].

In Figs. 8a, 8b and 8c we present the contours of viscosity ratio $[\nu/\nu]$. Note that, like Constantinescu and Squires (2004), each of these figures is shown in the plane of the resultant lateral force. All three figures have been plotted using the instantaneous flow field at the same chosen time instant ($t = 60 \ D/\nu_x$) and the contour levels are generated automatically based on the range of data available in the flow field around the sphere region.

Among the three schemes, again the MUSCL scheme seems to be producing results which are in better agreement with DES results (Constantinescu and Squires 2004) rather than SOU or BCD scheme. At the Reynolds number of $1.14 \times 10^6$, Constantinescu and Squires (2004) present that the wake structure show a prominent asymmetry: the wake is misaligned with the flow direction. Clearly, MUSCL scheme captures this effect more predominantly than either BCD or SOU. The contour maps displayed in Figs. 8a, 8b and 8c also indicate that the viscosity ratio is, in general, lower with MUSCL scheme as compared to simulation with BCD or SOU scheme. In a scale-resolving method, it is indeed desirable to have a lower viscosity ratio in the wake region to
successfully release more scales and features of fluid motion. Our results demonstrate that the MUSCL scheme is achieving this goal better than the other two schemes.

Summarizing this subsection, we conclude that the performance of the third-order MUSCL scheme is superior to both the BCD scheme and the SOU scheme in simulating flow past a sphere using PANS methodology. Among these three discretization schemes, performance of the second order BCD scheme is found to be the most dissipative and the least accurate. Based on these findings, all the remaining PANS studies have been performed employing the third-order MUSCL scheme.

5.3 Influence of Freestream Viscosity ratio

In this subsection we examine the sensitivity of PANS results on the freestream viscosity ratio. This examination is performed by comparing the results from simulations X03, X06 and X07. These simulations differ only in terms of the freestream viscosity ratio \( \eta_r \). Simulation X03 has the viscosity ratio as 10, whereas simulations X06 and X07 have the ratio as 1 and 100 respectively. The \( f_t \) value in all these simulations is 0.5 and the value of \( f_z \) has been set to unity. The chosen discretization scheme is the third-order MUSCL. We study the influence of \( \eta_r \) on both the mean quantities and the instantaneous flow features.

Table 5c Effect of \( \eta_r \) Study

<table>
<thead>
<tr>
<th>Case</th>
<th>( f_t )</th>
<th>( f_z )</th>
<th>( \eta_r )</th>
<th>( C_D )</th>
</tr>
</thead>
<tbody>
<tr>
<td>X06</td>
<td>0.5</td>
<td>1</td>
<td>1</td>
<td>0.140</td>
</tr>
<tr>
<td>X03</td>
<td>0.5</td>
<td>1</td>
<td>10</td>
<td>0.138</td>
</tr>
<tr>
<td>X07</td>
<td>0.5</td>
<td>1</td>
<td>100</td>
<td>0.141</td>
</tr>
<tr>
<td>Experimental (Achenbach 1974)</td>
<td></td>
<td></td>
<td></td>
<td>0.120</td>
</tr>
<tr>
<td>DES (Constantinescu and Squires 2004)</td>
<td></td>
<td></td>
<td></td>
<td>0.104</td>
</tr>
</tbody>
</table>

In Table 5c, we present mean value of \( C_D \) calculated from the results of simulations X03, X06 and X07. With \( \eta_r = 1 \), we get mean \( C_D \) as 0.140, whereas with \( \eta_r = 100 \) we get mean value of \( C_D \) as 0.141. Simulation X03, where we use \( \eta_r = 10 \), shows \( C_D = 0.138 \). Clearly, the influence of \( \eta_r \) is not very significant on the mean \( C_D \) value.

Fig. 9 distribution of \( C_p \) in simulations with \( \eta_r = 1 \) and \( \eta_r = 10 \) are indistinguishable, where as the mean \( C_v \) variation of \( \eta_r = 100 \) simulation shows some deterioration in performance especially in the range of 60° < \( \theta < 120° \). In Fig. 10, where mean \( C_f \) is plotted, the results from simulation with \( \eta_r = 10 \) and \( \eta_r = 100 \) are indistinguishable. However, the simulation with \( \eta_r = 1 \) shows some deterioration of performance in the separated region 90° < \( \theta < 180° \).

Comparisons of mean pressure coefficient and skin friction coefficient are presented in Figs. 9 and 10. In

Fig. 9. \( C_p \) distribution [X03, X06 and X07]

In Figs. 11a, 11b and 11c, we present iso-surfaces of Q from instantaneous flow fields. The figures show very similar results for simulations X03 and X06 (with \( \eta_r \) as 1 and 10 respectively). However, the iso-surfaces obtained from simulation X06 (\( \nu/n = 1 \) ) show less sharp vortical structures especially in the near-wake region. The high value of this viscosity ratio adds dissipation to the flow field, thus leading to suppression of turbulent structures.

We conclude that the PANS paradigm, in conjugation with the two-layer \( k-\varepsilon \) model, shows reasonable stability of mean quantity (\( C_D, C_p, C_f \)) in response to the level of viscosity ratio in the free stream conditions. However, the instantaneous results show same mild deterioration in performance (as compared to the experimental results) at very high value of the viscosity ratio (\( \eta_r = 100 \)). This is an important demonstration highlighting the robustness of the PANS paradigm in conjunction with the two-layer \( k-\varepsilon \) model in a highly complex, three-dimensional, separated flow field. Indeed Han et al. (2012) in their study of RANS models, find standard \( k-\varepsilon \) to demonstrate undesirable sensitivity to freestream viscosity ratio. However, our PANS results seem to suggest that despite the parents model being \( k-\varepsilon \), its PANS version shows acceptable immunity to freestream
turbulence. Overall, we judge $\eta = 10$ to be an optimum choice for initialization of the flow field at Reynolds number under consideration and the remaining studies of this paper performed keeping $\eta = 10$.

5.4 Influence of Filter Parameter $f_k$

PANS has been purposed as a seamless method with the inherent potential to release increasingly more scales as $f_k$ is reduced. Such a trend has indeed been demonstrated in several flow fields [Lakshmipathy and Girimaji (2006)].

In Table 5d we present mean $C_D$ values from the three simulations. The experimental mean $C_D$ value reported by Achenbach (1974) is 0.120. As the $f_k$ value is reduced from 1 to 0.5, the mean $C_D$ value monotonically reduces and moves closer to the experimental results of Achenbach (1974). In the table we have also presented the LES results of Kim and Choi (2001) ($C_D = 0.139$), Jindal, Long, Plassmann, and Sezer-Uzol (2004) [$0.141$] and DES results of Constantinescu and Squires (2004) [$0.102$]. Our PANS results, like the LES simulations of Kim and Choi (2001) and Jindal et al. (2004), overestimate the value of $C_D$. The DES simulation of Constantinescu and Squires (2004), on the other hand, underestimates the value of $C_D$. Examining the percentage difference of the $C_D$ values (with the experimental value as the datum) we find that our simulation with $f_k = 0.5$ has error of around 15% which is equal to that of DES result (Constantinescu and Squires 2004). The LES results of Kim and Choi (2001) and that of Jindal et al. (2004) show error of 15.8 % and 19.1 % respectively. Clearly, the PANS simulation with $f_k = 0.5$ shows improved prediction of mean drag coefficient over the reported LES results.

In Fig. 12, we present the mean $C_p$ distribution from various simulations and experiments. Like mean $C_D$, mean $C_p$ also shows maximum departure from the experimental observations in the case of URANS simulations. However, as the value of $f_k$ is reduced, we observe an improvement in the results. Indeed, PANS result with $f_k = 0.5$ is almost indistinguishable from DES results of Constantinescu and Squires (2004). In the range $0 \leq \theta < 100^\circ$, PANS with $f_k = 0.5$ shows excellent agreement with the experimental data (Achenbach 1974) as well. In the range $100^\circ < \theta < 180^\circ$, like DES (Constantinescu and Squires 2004), $f_k = 0.5$
PANS results show some departure from experimental data.

Fig. 12. $C_p$ distribution [X03, X08 and X09]

In Fig. 13, we present the surface distribution of the mean value of $C_f$. As expected, the URANS results are the most inaccurate, especially in the wake region ($\theta > 120^\circ$). Again, reducing the value of $f_k$ improves the performance of PANS results. In the region with $80^\circ < \theta < 120^\circ$, even though all numerical simulations show gross departure from the experimental observations, $f_k = 0.5$ results do show some improvement over even the DES results. At other locations, PANS result with $f_k = 0.5$ are very similar to those in DES simulation.

Further examination of the influence of $f_k$ is performed in terms of the instantaneous flow field and turbulence structures. In Figs. 14a, 14b and 14c we present contours of viscosity ratio in the plane of the net instantaneous lateral force. As the value of $f_k$ is reduced in the PANS paradigm, it is expected that the levels of viscosity ratio in the flow field would monotonically decrease − thus allowing smaller scales of motion to be released (resolved) in the flow field.

In Figs. 14a, 14b and 14c contours are plotted with same range of color map. Clearly, the levels of viscosity ratio are the highest in URANS simulation with $f_k = 1$. Scale levels monotonically decrease as we go to simulation X08 ($f_k = 0.7$) and then to simulation X03 ($f_k = 0.5$). This is again along the expected lines.

Fig. 13. $C_f$ distribution [X03, X08 and X09]

Fig. 14. Viscosity ratio contours
Further evidence of the release of more scales of motion in simulations X03 and X08, as compared to URANS simulation, can be observed in Figs. 15a, 15b and 15c, wherein we show iso-surfaces of Q-criterion from three simulations.

Fig. 14a Simulation X09 [$f_k = 1$]

Fig. 14b Simulation X08 [$f_k = 0.7$]

Fig. 14c Simulation X03 [$f_k = 0.5$]

Fig. 15. Instantaneous vortical structures
With $f_k = 0.5$, simulation X03 clearly shows several horseshoe vortices being shed in the wake of the cylinder. This behaviour is in line with the results of DES (Constantinescu and Squires 2004) as well as experimental observations. In fact, based on the comparison of our results and the results of Constantinescu and Squires (2004), it seems that the PANS Simulation X03 is resolving the horseshoe vortices perhaps better than even those in the DES simulations (compare our Fig. 14c with Fig. 9d of (Constantinescu and Squires 2004)). In simulation X08 with $f_k = 0.7$, the size of the horseshoe vortices decreases and they look underdeveloped. However, their presence is still very much evident. In contrast, the vortex structure captured by the URANS (simulation X09) is quite different. We do not observe any evidence of horse-shoe vortices. Instead, two counter-clockwise extended vortex tubes are seen. This is in gross disagreement with the behaviour expected at supercritical Reynolds number of $1.14 \times 10^6$. Thus, based on our observations, we conclude that decreasing the value of $f_k$ for flow past a sphere, PANS indeed shows a clear capability to capture the complex vortex shedding patterns expected at high Reynolds number. Further, at $f_k = 0.5$, the vortex structure seem to be better resolved than even DES results of Constantinescu and Squires (2004).

5.5 Influence of filter parameter $f_k$

We study the influence of $f_k$ by comparing the results from simulations X03, X08, X10 and X11. Simulations X03 ($f_k = 0.5$, $f_k = 1$), and X08 ($f_k = 0.7$, $f_k = 1$) have been earlier employed in Section 5d to study the influence of $f_k$. These simulations use $f_k = 1$. To evaluate the influence of $f_k$, we compare results of simulation X03 and X10, which have identical $f_k (= 0.5)$ but different $f_k$ (1 and 0.5). Since all other parameters of the two simulations are identical, differences observed in these two flow fields can be attributed to the difference in the value of $f_k$. To further understand the trend, if any, we further compare results from simulation X08 ($f_k = 0.7$, $f_k = 1$) and X11 ($f_k = 0.7$, $f_k = 0.7$).

In Fig. 16, we present the distribution of mean $C_p$ obtained from simulations X03, X08, X10 and X11. Results from simulation X03 show significant differences as compared to that from X10. Similarly, simulation X08 and X11 show differences. In the aft region of the sphere, the pressure coefficient predicted by simulations X10 and X11 (with a lower value of $f_k$) shows significantly more error when compared to the experimental data or the DES results.

In Fig. 17, we show the variation in the mean skin friction coefficient value. Similar deterioration in performance is observed when $f_k$ is reduced. The performance of simulation X11 with $f_k = 0.7$ and $f_k = 0.7$ is much inferior to the performance shown by simulation X08 with $f_k = 0.7$ and $f_k = 1$. Similarly, some deterioration in performance is seen at $f_k = 0.5$ when $f_k$ is changed from 1 to 0.5.

![Fig. 16. $C_p$ distribution [X03, X08, X10 and X11]](image)

![Fig. 17. $C_f$ distribution [X03, X08, X10 and X11]](image)
has counteracting effects on the unresolved kinetic energy and in turn, on the eddy viscosity.

At very high Reynolds number, it is indeed prudent to choose the maximum possible value of \( f_\varepsilon \) (=1) because it is reasonable to expect that the filter cut-off lies far right to the dissipation scales of the turbulence kinetic energy spectrum. However, if the Reynolds number is not ‘adequately high’, the separation between the cut-off scale of turbulence kinetic energy and dissipation scales may not be large. In such a case, setting \( f_\varepsilon = 1 \) may not be the correct choice, and choosing a subunity value of \( f_\varepsilon \) may

balances the reduction in production achieved by a lower value of \( f_\varepsilon \). In extreme cases, the dissipation rate may be reduced so much that large values of turbulence kinetic energy accumulate in the flow field (despite a subunity value of \( f_\varepsilon \)) causing the turbulence viscosity to become comparable with the URANS levels. Such a situation will lead to suppression of scales and the simulated flow field may resemble more like a URANS simulation. Thus, it is important to choose the correct balance between \( f_k \) and \( f_\varepsilon \) values.

Indeed, Lakshmipathy and Girimaji (2010b) in their study of the performance of PANS method, demonstrate that while at Reynolds number = 1.4x10^5 the performance of PANS simulation is superior with \( f_\varepsilon = 1 \), the performance is superior at Reynolds number = 3900 with \( f_\varepsilon = 0.7 \) (at the same \( f_k \) value). Since it is not at all straightforward to conjecture the optimum value of \( f_\varepsilon \) at a given Reynolds number, a careful study of the influence of \( f_\varepsilon \) is thus required.

Results of our simulations suggest that the optimum value of \( f_\varepsilon \) for flow past a sphere, at Reynolds number = 1.14x10^6, is indeed 1. When a lower value of \( f_\varepsilon \) is chosen, dissipation reduces – thus counteracting the reduction in production achieved by sub-unity \( f_\varepsilon \) value. This is confirmed
in Figs. 19a, 19b, 19c and 19d wherein we show the scales for the contour levels of dissipation from simulations X03, X10, X08 and X11. Clearly, simulation X03 shows more dissipation scales as compared to simulation X10. Similarly, Figs. 19c and 19d show more dissipation scales prevailing in the flow field in simulation X08 as compared to simulation X11.

6 SUMMARY

We study the performance of partially averaged Navier-Stokes (PANS) method in simulating massively separated flow field around a sphere at supercritical Reynolds number $= 1.14 \times 10^6$. Despite its geometric simplicity, flow past a sphere is acknowledged to be quite complicated due to the coexistence of numerous complex phenomena. This complexity makes this flow an excellent test bed for turbulence models and numerical schemes alike. In this work, we evaluate PANS specifically in terms of four aspects: (i) influence of filter parameter $f_k$, (ii) influence of filter parameter $f_\varepsilon$, (iii) influence of discretization scheme and (iv) influence of freestream turbulence viscosity ratio.

As purported, a monotonic reduction in the value of $f_k$ systematically releases more scales of motion and takes the results closer to experiments/DES simulation. Indeed, mean quantities calculated using PANS simulations with $f_k = 0.5$ are found to be fairly close to the DES results of Constantinescu and Squires (2004).

Further, our study reveals that, like other scale resolving methods [DES (Constantinescu and Squires 2004), LES (Kim and Choi (2001), Jindal et al. (2004))], PANS shows significant sensitivity to the numerical schemes. The third-order MUSCL scheme is found to be superior to the SOU and BCD method – especially, in terms of resolving the horseshoe vortex system, which has been observed in experiments as well the state-of-the-art DES simulations.

Further, we find that the influence of freestream viscosity ratio does not affect the mean quantities much. However, it does influence the instantaneous flow features. Freestream viscosity ratio is indeed known to influence the preference of RANS models like $k-\varepsilon$. Our investigations reveal that in comparison to the parent RANS model (in our case the standard $k-\varepsilon$ model), the PANS paradigm tends to mitigate the sensitivity towards freestream viscosity ratio in the flow field.

Our study on the influence of $f_\varepsilon$ on the results of PANS simulation ascertains that the Reynolds number of $1.14 \times 10^6$ is high enough to choose the value of the second filter parameter to be unity.

We demonstrate that at several $f_k$ values, PANS performance is optimal with $f_\varepsilon = 1$. At lower values of $f_\varepsilon$, kinetic energy dissipation reduces – thus annuling the advantage of reduced production achieved by lowering values of $f_k$.

Based on our findings, we conclude that the performance of PANS in simulating flow past a sphere is quite satisfactory. However, since some flow quantities, especially skin friction coefficient, is not captured adequately. Further improvements in PANS modelling, especially in terms of wall treatment, are required for reliably simulating three dimensional separated flows.

ACKNOWLEDGMENTS

We thank the IIT Delhi HPC facility for computational resources.

REFERENCES


Constantinescu, G. and K. Squires (2004). Numerical investigations of flow over a sphere in the sub-critical and supercritical


