Abstract

The Detached Eddy Simulation (DES) and steady-state Reynolds-Averaged Navier-Stokes (RANS) turbulence modeling approaches are examined for the incompressible flow over a square cross-section cylinder at a Reynolds number of 21,400. A compressible flow code is used which employs a second-order Roe upwind spatial discretization. Efforts are made to assess the numerical accuracy of the DES predictions with regards to statistical convergence, iterative convergence, and temporal and spatial discretization error. Three-dimensional DES simulations compared well with two-dimensional DES simulations, suggesting that the dominant vortex shedding mechanism is effectively two-dimensional. The two-dimensional simulations are validated via comparison to experimental data for mean and RMS velocities as well as Reynolds stress in the cylinder wake. The steady-state RANS models significantly overpredict the size of the recirculation zone, thus underpredicting the drag coefficient relative to the experimental value. The DES model is found to give good agreement with the experimental velocity data in the wake, drag coefficient, and recirculation zone length.

Introduction

The flow around bodies at high Reynolds number, especially buffet bodies, can be unsteady with turbulent eddies shed and detached from the boundary layer flow. The usual steady-state Reynolds-Averaged Navier-Stokes (RANS) equations become inappropriate for these flows. The Large Eddy Simulation (LES) approach is becoming a popular technique to model these flows. In LES, the larger structures (eddies) in the turbulent spectrum are resolved, and the smaller structures are modeled. This approach is computationally expensive and there are still modeling issues that are being investigated. A subgrid-scale model must be used. The classic subgrid-scale model was introduced by Smagorinsky, while the dynamic model approach of Germano et al. has become very popular. This paper is concerned with wall bounded flows where additional modeling is required near the surface. The unsteady turbulence modeling technique investigated herein is the hybrid RANS/LES model Detached Eddy Simulation (DES).

There have been a number of reviews of the LES approach. In the reviews by Rodi and Piomelli, the modeling problem in the near wall region has been discussed. There have been various approaches proposed to model the near wall flow with the LES approach. Some of the earliest work on this problem was done by Piomelli et al. One approach uses boundary conditions at the first mesh point away from the wall that approximate the flow behavior near the wall. This approach is similar to the wall function method used with the RANS equations and is computationally very efficient. The more accurate approach is to refine the mesh near the wall with a significant increase in computational resources.

Recently there have been several papers that suggest a blending of RANS and LES. This approach has been discussed by Speziale, Germano, Arunajatesan et al., and Spalart et al. From a practical point of view, the blending of RANS and LES approaches is attractive because the majority of industrial computational fluid dynamics codes already employ RANS-type turbulence models. Furthermore, steady-state RANS models can provide good predictions for a wide variety of flows, while LES can provide additional accuracy, at increased cost, in regions where the steady-state RANS models fail (e.g., massive flow separation).
Turbulent flow past a square cross-section cylinder is a geometrically simple, bluff body flow that has been the basis of numerous computational and experimental studies. The dynamic nature of the induced, unsteady vortex shedding in the base flow makes the square cylinder problem particularly attractive as a test problem for unsteady RANS, LES, and hybrid RANS/LES models.

The square cross-section cylinder has been studied experimentally by Lyn et al.\(^\text{12}\) (Re\(_D\) = 21,400) and Durao et al.\(^\text{13}\) (Re\(_D\) = 14,000) who provide global measurements, i.e., drag and lift, as well as mean flow and turbulence quantity profiles. Lyn et al.\(^\text{12}\) provide details of instantaneous mean velocity field, vorticity and turbulence quantities profiles. Though comprehensive, both experiments have uncertainties with regards to the effects of tunnel blockage and model aspect ratio. Despite these uncertainties, both experimental studies are widely used for validating computational studies of the square cylinder flow.

Computational models for flow over a square cylinder include the LES simulations of Sohanker et al.,\(^\text{14}\) Nakayama and Vengadesan,\(^\text{15}\) along with the unsteady RANS computations of Iaccarino et al.,\(^\text{16}\) Shimada and Ishihara,\(^\text{17}\) Bosch and Rodi,\(^\text{18}\) and Lee.\(^\text{19}\) These computations are all performed within the scope of finite-volume based solvers. By contrast, Jeong et al.\(^\text{20}\) simulate vortex shedding in the base flow make the square cylinder problem particular attractive as a test problem for unsteady RANS, LES, and hybrid RANS/LES models.

The purpose of the current paper is two-fold. The first goal is to explore the numerical accuracy of time-accurate turbulence simulations with regards to temporal and spatial discretization error, iterative convergence, and convergence of the statistical properties of the flow. The second goal is to validate the Detached Eddy Simulation (DES) hybrid RANS/LES model for a simple, well-defined, moderate Reynolds number flow.

Simulation Approach

Simulation Code

The computational fluid dynamics code used herein is SACCARA, the Sandia Advanced Code for Compressible Aerothermodynamics Research and Analysis. The SACCARA code was developed from a parallel distributed memory architecture based on multi-block structured grids. SACCARA code employs a massively parallel distributed memory version\(^\text{21}\) of the INCA code,\(^\text{22}\) originally written by Amtec Engineering. The SACCARA code is used to solve the Navier-Stokes equations for conservation of mass, momentum, energy, and turbulence transport in either 2D or 3D form. Prior code verification studies with SACCARA include code-to-code comparisons with other Navier-Stokes codes\(^\text{23,24}\) and with the Direct Simulation Monte Carlo method.\(^\text{25}\) These studies provide confidence that the code is free from coding errors affecting the discretization.

Discretization

The governing equations are discretized using a cell-centered finite-volume approach. A finite-volume form of Harten and Yee’s symmetric TVD scheme\(^\text{26,27}\) is employed. This flux scheme is second-order accurate and reduces to a first-order Roe-type scheme\(^\text{28}\) in regions of large gradients based on a minmod limiter. The viscous terms are discretized using central differences. The SACCARA code employs a massively parallel distributed memory architecture based on multi-block structured grids.

The solver is a Lower-Upper Symmetric Gauss-Seidel scheme based on the works of Yoon et al.\(^\text{29}\) and Peery and Inlay,\(^\text{30}\) which provides for excellent scal-
ability up to thousands of processors. Second-order accuracy is obtained in the temporal discretization via a sub-iterative procedure. In this approach, the time derivative in the governing equations is discretized with a second-order backward difference. The three-point backward time derivative is added to the steady-state residual, and the solution at time level \( n+1 \) is iterated until the right-hand side, which includes both the steady-state residual and the time derivative, are driven below a given tolerance. The second-order temporal accuracy was verified by examining a uniform flow where the inflow velocity was varied in time according to a sinusoidal function. The level of iterative convergence was also examined for the simple flow. A plot of iterative error versus residual norm reduction is presented in Fig. 1. The iterative error shows a nearly linear reduction with residual convergence level on the log-log plot, with a residual reduction of four orders of magnitude corresponding to roughly 1\% iterative error.

Turbulence Models

For all of the simulation results presented herein, the turbulence transport equations are integrated all the way to the cylinder walls, thus no wall functions are employed. In all cases, the distance from the wall to the first cell center off the wall is less than unity in normalized turbulence distance (i.e., \( y^+ < 1 \)).

Steady-State RANS

Four turbulence models are examined in the current work. The first turbulence model is the one-equation Spalart-Allmaras eddy viscosity model\(^{32,33}\) which has a robust numerical formulation and has shown promising results for a wide variety of flows. The second model is the standard \( k-\varepsilon \) model\(^{34}\) which uses low Reynolds number damping functions\(^{35}\) near solid walls. The third model is Menter’s hybrid model\(^{36}\) which switches from a \( k-\varepsilon \) formulation in the outer flow to a \( k-\omega \) formulation near solid walls. The fourth model is Wilcox’s improved version\(^{34}\) of his earlier \( k-\omega \) turbulence model. Additional details on the form and implementation of these models can be found in Ref. 37.

Detached Eddy Simulation (DES)

The hybrid RANS/LES method developed by Spalart and co-workers (Refs. 3, 38) has been developed the furthest and has been named the Detached Eddy Simulation (DES) approach. In their early work, Spalart and co-workers\(^{38}\) used grid generated isotropic turbulence to calibrate the DES model, and the model constant was determined as \( C_{DES} = 0.65 \). The DES approach was then applied to the flow over an airfoil at angle of attack. The spanwise extent was one chord length and employed periodic boundary conditions applied at these locations. Constantinou and Squires\(^{39}\) have used the DES approach to predict the 3D flow over a sphere at a Reynolds number of 10,000.

The DES approach uses the unsteady form of the Spalart-Allmaras one-equation eddy viscosity model\(^{32}\) to provide the eddy viscosity \( \nu_T = \mu_T/\rho \) for use in SGS stress model. The Spalart and Allmaras one equation eddy viscosity model provides the usual RANS-based eddy viscosity in the boundary layer, but must be modified to the appropriate eddy viscosity for LES outside of the boundary layer. This modification is performed by changing the definition of the distance to the wall \( d \) in the destruction term. The distance \( d \) is replaced with \( \tilde{d} \), where this new term is defined as

\[
\tilde{d} = \min(d, C_{DES} \Delta)
\]

Far from the wall, the value of \( \tilde{d} \) thus becomes

\[
\tilde{d} = C_{DES} \Delta.
\]

The local grid spacing is defined as \( \Delta \) and is equal to the maximum mesh spacing in the three coordinate directions.

\[
\Delta = \max(\delta_x, \delta_y, \delta_z)
\]

As discussed by Spalart et al.,\(^3\) when the production term is balanced with the destruction term (at equilibrium), the following is obtained

\[
\mu_T = \tilde{C}_S \Delta^2 \Delta \tilde{S}, \quad \tilde{C}_S = C_{DES} \frac{C_{b f} \nu}{C_{w f}}
\]

where \( \tilde{S} \) is related to the magnitude of the strain rate. In the outer part of the boundary layer, \( \tilde{C}_S \) asymptotes to \( \tilde{C}_S = 0.29C_{DES} = 0.19 \). The constant \( C_{DES} \) is given by
Problem Description

The three dimensional flow over a square cross-section cylinder in cross flow has been investigated experimentally at a Reynolds number $Re_D$ of 21,400 by Lyn et al.\textsuperscript{12} This case is considered somewhat simpler than a circular cross-section cylinder since the separation points are clearly defined and occur at the cylinder corners. A schematic of the domain used in the current computations is presented in Fig. 2. The physical dimensions correspond to the water tunnel used in the experimental investigation of Lyn et al.\textsuperscript{12} Since a compressible CFD code is employed with air as the fluid, the freestream velocity was increased to approximately 32.5 m/s (compared to 0.535 m/s in the experiment) to achieve a freestream Mach number of 0.1, thus avoiding the convergence and accuracy issues associated with compressible flow algorithms at low Mach numbers. For a freestream Mach number of 0.1, compressibility effects are not expected to affect the results. The fluid was air with a ratio of specific heats of 1.4 and a specific gas constant of 287.0 J/(kg K), and had a freestream density of approximately 1.162 kg/m$^3$. The fixed absolute viscosity was therefore increased to $7.5374 \times 10^{-5}$ Ns/m$^2$ to match the Reynolds number based on diameter of 21,400.

Boundary Conditions

The inflow boundary employs stagnation values for pressure (100,701.75 N/m$^2$) and temperature (300.6 K) and enforces inflow normal to the boundary. The outflow boundary employs a fixed static pressure of 100,000 N/m$^2$. The choice of stagnation and back pressures yield a nominal freestream velocity of 32.5 m/s. Slip conditions on velocity are employed on the top and bottom walls, while the cylinder surface employs no-slip velocity conditions and assumes an adiabatic wall. The three-dimensional simulations employ a span-wise extent of 4D, with periodic boundary conditions applied.

The freestream RMS axial velocity, normalized by the freestream velocity, is given as 0.02 in Lyn et al.\textsuperscript{12} Based on the freestream velocity of 32.5 m/s used in the current simulations, this 2% turbulence intensity corresponds to a turbulent kinetic energy of 0.72 m$^2$/s$^2$. The freestream turbulent viscosity was chosen as 0.13 times the molecular viscosity, or $1 \times 10^{-5}$ Ns/m$^2$. These freestream turbulence values result in a freestream dissipation rate and frequency of $\varepsilon = 7255.3$ m$^2$/s$^3$ and $\omega = 1.1145 \times 10^5$ 1/s, respectively. Solid wall boundary conditions for the turbulence models can be found in Ref. 37.

Normalization of Flow Variables

All velocities and velocity statistics reported herein are normalized by the freestream velocity $U_{\text{inf}}$, which is found by averaging the axial velocity at the inflow boundary. This freestream velocity for the coarse, medium, and fine meshes was 32.5, 31.5, and 31.0 m/s, respectively. The wake flow is dominated by large scale vortex shedding from the cylinder. A time history of the lift coefficient on the cylinder is given in Fig. 3 using the 2D fine mesh. The time is normalized by a reference time scale defined as

$$t_{\text{ref}} = D/U_{\text{inf}} = 0.00123 \text{ s} \quad (5)$$

where $U_{\text{inf}}$ is taken as the coarse mesh value of 32.5 m/s. Unless otherwise noted, the simulations employed a dimensionless time step $\Delta t/t_{\text{ref}}$ of 0.0032, which corresponds to approximately 2500 steps per lift cycle (approximately 8.13 reference times). The lift oscillates about zero as expected, with the peaks and valleys corresponding to large-scale shedding from the top and bottom of the cylinder.

Numerical Accuracy

In order to explore the role of the numerics at a moderate cost, numerical accuracy studies are conducted using the two-dimensional DES simulations on the medium mesh (40,000 mesh cells). Velocity profiles in the wake centerline are employed in the analysis.

Statistical Convergence

It is unclear a priori how long the velocity statistics sampling should be performed before the statistical error becomes sufficiently small. The size of the time window for collecting velocity statistics was varied between 2 and 52 periods of the vortex shedding. The results for mean axial velocity $u$ and RMS vertical velocity $v_{\text{RMS}}$ are presented in Fig. 4. A time window of 2 periods is clearly too short, while increasing the time window to...
10 or 20 periods provides a decrease in the statistical error. The results for a time window of 36 and 52 periods are essentially indistinguishable. For the remainder of this paper, statistics are collected for at least 20 periods of the shedding (approximately 160 reference times).

**Iterative Convergence**

While the level of iterative convergence of the unsteady residual (steady-state residual plus time derivative), was examined for a simple time varying flow in Fig. 1, the applicability of these results to the current square cylinder case is unclear. Four different cases were computed using the DES model where the number of sub-iterations was varied from 2 to 30, giving residual reductions between 2.4 and 4.7 orders of magnitude, respectively. The centerline wake velocities (both mean axial and RMS vertical) are presented in Fig. 5. The figure shows that roughly a four order of magnitude reduction in the residual is sufficient for the iterative error to be negligible, especially when compared to the temporal and spatial discretization errors to be discussed in the next sections. For practical engineering solutions, a three order of magnitude drop in the residuals is probably sufficient.

**Temporal Discretization Error**

In order to assess the effects of the temporal discretization errors on the solution, the time step was varied between 0.0008 and 0.0064 characteristic times. The effects on the wake centerline velocity profiles are shown in Fig. 6. The time step of 0.0064 gives a significant underprediction of the minimal axial velocity in the wake, and a slight underprediction of the peak $v_{RMS}$ value. While the time step of 0.0032 also underpredicts the minimum axial velocity in the wake, this temporal discretization error is small when compared with the spatial discretization errors examined next.

**Spatial Discretization Error**

**DES**

Two dimensional DES calculations have been performed on three mesh levels: a coarse mesh (10,000 cells), a medium mesh (40,000 cells), and a fine mesh (160,000 cells). Each coarser mesh is found by eliminating every other grid line in each direction from the finer mesh (i.e., grid halving). The wake centerline profiles for mean axial velocity on all three meshes are given in Fig. 7. Also shown in the figure is a profile resulting from the Richardson extrapolation of the fine and medium mesh results, i.e.,

$$u_{RE} \approx u_{fine} + (u_{fine} - u_{medium})/3$$  

Richardson extrapolation essentially extrapolates the so-
solutions from the medium and fine meshes to obtain an estimate of the continuum solution (i.e., $\Delta x, \Delta y \to 0$) by using the assumption that the spatial discretization is second-order accurate. The coarse mesh greatly over-predicts the length of the ensemble-averaged recirculation zone relative to the extrapolated results. While the medium mesh shows significant improvement over the coarse mesh results, the error on this mesh is still large relative to the extrapolated result. The fine mesh appears to be fairly well grid converged except in the vicinity of $x/D = 2$. This mesh employs roughly homogeneous cells in the wake up to a distance of 8 diameters downstream, with approximately 40 points located transversely across the cylinder diameter. While Richardson extrapolation has been successfully applied to steady-state solutions, the extension to the ensemble-averaged statistics for unsteady problems requires further investigation.

Steady-State RANS

The same three meshes were run for the steady-state RANS solutions, however a symmetry plane was added at $y/D = 0$ in order to suppress the vortex shedding instabilities and thus obtain true steady-state solutions. The results of this grid refinement study for the Spalart-Allmaras turbulence model are presented in Fig. 8. The fine mesh shows little error relative to the extrapolated solution. Although not shown, the grid convergence studies for the other three models was similar.

Model Validation

Both two- and three-dimensional DES results were obtained on the medium mesh. The 2D mesh employed approximately 40,000 mesh cells, and the 3D mesh used the 2D mesh as the cross-section and had 80 planes in the span-wise direction, resulting in a total of roughly 3.1 million cells. A comparison of mean and RMS velocities along the wake centerline is presented in Fig. 9. The similarity of the results is due to the strong two-dimensional nature of the vortex shedding instability. Model validation results will therefore focus on the 2D DES simulations using the fine mesh (160,000 cells). These fine mesh DES simulations were run with the normalized time step of 0.0032 and statistics were sampled over a time window of approximately 12 periods (20 periods still running) of the vortex shedding. Furthermore, each time step was iteratively converged until a four order of magnitude drop in the residuals was obtained, corresponding to approximately 15 sub-iterations.

Wake Centerline Profiles

Results for the axial velocity on the wake centerline using all four steady-state RANS models are compared to the experimental data of Lyn et al.\textsuperscript{12} ($Re_D = 21,400$)
and Durao et al.\(^\text{13}\) (\(Re_D = 14,000\)) in Fig. 10. All four turbulence models greatly overpredict the length of the recirculation zone. The results for the Spalart-Allmaras one-equations model are nominally better than the others, while the Wilcox \(k-\omega\) model gives the poorest agreement with the data. These trends are consistent with those found at much higher Reynolds numbers over bluff bodies, where the large scale shedding of coherent vortices is less apparent (e.g., Roy et al.\(^\text{40}\)). The nominally better results with the Spalart-Allmaras model should not be taken to mean that this model is more accurate for bluff-body flows, since the model is known to underpredict the size of recirculation zones in other cases. Results for both the Menter \(k-\omega\) and Spalart-Allmaras models will be retained for comparison to the DES results discussed in detail next.

A comparison of the time-averaged axial velocity from the fine mesh 2D DES simulation to experimental data on the centerline of the wake is given in Fig. 11. Also shown in the figure are the results using the steady-state RANS models of Menter and Spalart-Allmaras. The DES results show marked improvement over the steady-state RANS models, with the solution generally falling within the ±0.05 error bands quoted by Lyn et al.\(^\text{12}\).

The RMS values in the wake centerline for \(u\)- and \(v\)-velocities are presented in Figs. 12 and 13 for the DES model (resolved components only) and Menter \(k-\omega\) models (modeled components only). Estimation of the normal stress components of the Reynolds stress tensor with “traceless” eddy-viscosity models (which do not explicitly transport subgrid turbulent kinetic energy) being problematic.\(^\text{41}\) The steady-state RANS model of Menter greatly underpredicts the peak RMS velocity values in the wake. The DES simulation shows good agreement with the data for RMS \(v\) velocity, but slightly overpredicts the RMS axial velocity.

**Wake Transverse Profiles**

Transverse profiles of the mean \(u\)- and \(v\)-velocities at a location one-half diameter downstream of the cylinder (\(x/D = 1\)) are given in Figs. 14 and 15, respectively. For the DES model, results in both the upper and lower quadrants are shown by plotting against the absolute value of \(y/D\). The good agreement between the upper and lower profiles provide further evidence that the statistics for the DES simulations are sufficiently converged. As expected, the steady-state RANS models overpredict the extent of the wake (see Fig. 14), while the DES model shows fair agreement with the experimental data. Results for the vertical component of the mean velocity are presented in Fig. 15. The gross overprediction of the size of the recirculation zone with the steady-state RANS models results in both the magnitude

---

**Fig. 9:** Comparison of 2D versus 3D DES simulations on the medium mesh for the mean and RMS velocities along the wake centerline.

**Fig. 10:** Wake centerline mean axial velocities for steady-state RANS: Low Re \(k-\varepsilon\), Menter \(k-\omega\), Wilcox \(k-\omega\), and Spalart-Allmaras.

**Fig. 11:** Wake centerline mean axial velocities for steady-state RANS, 2D DES, and 3D DES (preliminary results).
and the sign of the vertical velocity to be incorrect. The DES model gives results well within the experimental uncertainty bounds.

RMS axial and vertical velocities are presented in Figs. 16 and 17, respectively. The Menter k-ω model underpredicts the RMS velocities, especially near the wake centerline ($y/D = 0$), while the DES model gives good agreement with the experimental data.

The transverse profile of Reynolds stress is given in Fig. 18 for the steady-state RANS models and DES. The steady-state RANS models underpredict the positive peak of the Reynolds stress, and miss entirely the negative peak close to the centerline. The DES model matches the experimental data for the positive peak, but does not capture the negative peak. The failure of the DES model to capture the negative peak is probably due to the short time window used to collect the velocity statistics (currently only 12 periods of the vortex shedding).

Fig. 12: Wake centerline RMS axial velocities for steady-state RANS and DES.

Fig. 13: Wake centerline RMS vertical velocities for steady-state RANS and DES.

Fig. 14: Wake transverse mean axial velocities for steady-state RANS and DES.

Fig. 15: Wake centerline mean vertical velocities for steady-state RANS and DES.

Fig. 16: Wake transverse RMS axial velocities for steady-state RANS and DES.
This hypothesis is backed up by Fig. 19, which gives results for the medium mesh DES as well. The medium mesh does indeed capture the negative peak, but underpredicts the magnitude. Also shown in the figure are the subgrid contributions to the Reynolds stress on the two meshes. These subgrid contributions are negligible (i.e., near zero) in both cases.

Global Quantities

The time history of the drag coefficient is given in Fig. 20 for the fine mesh DES simulations. The mean value is approximately 2.4, which is approximately 15% higher than the experimentally reported value of 2.1. Finally, the normalized length of the recirculation zone \( l_R/D \) from the experiment was 1.38. The steady-state RANS models of Menter and Spalart-Allmaras predicted recirculation zone lengths of 4.8 and 3.7, respectively. The medium mesh DES computation gave 1.89, while the fine mesh had 1.22. These results are summarized in Table 1.

<table>
<thead>
<tr>
<th></th>
<th>( C_D )</th>
<th>Uncert. ( (C_D) )</th>
<th>( l_R/D )</th>
<th>Uncert. ( (l_R/D) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment(^2)</td>
<td>2.10</td>
<td>--</td>
<td>1.38</td>
<td>±0.07</td>
</tr>
<tr>
<td>DES (medium)</td>
<td>1.80</td>
<td>±0.8</td>
<td>1.89</td>
<td>±0.76</td>
</tr>
<tr>
<td>DES (fine)</td>
<td>2.40</td>
<td>±0.2</td>
<td>1.22</td>
<td>±0.09</td>
</tr>
<tr>
<td>Spalart-Allmaras</td>
<td>1.64</td>
<td>--</td>
<td>3.70</td>
<td>±0.12</td>
</tr>
<tr>
<td>Menter ( k-\omega )</td>
<td>1.58</td>
<td>--</td>
<td>4.79</td>
<td>±0.12</td>
</tr>
<tr>
<td>Low Re ( k-\varepsilon )</td>
<td>1.52</td>
<td>--</td>
<td>5.22</td>
<td>±0.12</td>
</tr>
<tr>
<td>Wilcox ( k-\omega )</td>
<td>1.53</td>
<td>--</td>
<td>8.29</td>
<td>±0.12</td>
</tr>
</tbody>
</table>

Conclusions

The flow over a square cross-section cylinder was examined using two different turbulence modeling approaches. For the steady-state RANS methods, a symmetry plane at \( y/D = 0 \) was employed which suppressed the vortex shedding instability, thus allowing steady-state solutions to be achieved. The second approach employed the Detached Eddy Simulation (DES) model developed by Spalart and co-workers. In this case, two-dimensional simulations were found to be sufficient to capture the predominantly two-dimensional vortex shedding events.

Since the interaction between the sub-grid scale models and the numerical scheme can play an important role for LES-type approaches, careful attention was paid to the numerical accuracy of the simulations. The numeri-
Numerical issues examined include statistical convergence, iterative convergence, temporal discretization error, and spatial discretization error. Numerically accurate results for velocities in the wake region were obtained when 1) statistics were collected for more than 20 shedding periods, 2) the unsteady residuals of the sub-iterations were reduced to approximately 0.0001, 3) the normalized time step was less than 0.0032, and 4) a fine mesh was employed with maximum cell sizes in the wake of approximately 0.025D (i.e., approximately 40 mesh cells across the cylinder diameter).

All four of the steady-state RANS models greatly overpredicted the length of the recirculation zone, thus underpredicting the drag. These models also gave poor predictions of the mean and fluctuating velocities in the wake. The DES model, on the other hand, showed good agreement for both the global quantities (drag coefficient, recirculation zone length) and the wake velocities (mean velocities, RMS velocities, and Reynolds stress).

The results presented herein provide strong evidence that LES-type turbulence models can be accurately employed in second-order accurate upwind discretizations. While higher-order methods may produce similar results at a reduced computational expense, accurate results for second-order methods can be achieved when the mesh is sufficiently refined so as to limit the adverse effects of numerical dissipation. This conclusion supports the extension of second-order computational fluid dynamics codes, which are ubiquitous in industry, to the LES regime.

Acknowledgements

We would like to thank Sharath Girimaji of Texas A&M University and Paul DesJardin of SUNY-Buffalo for their helpful discussions on unsteady turbulence modeling and Large Eddy Simulation.

References


