LAWRENCE
LIVERMORE
NATIONAL
LABORATORY

UCRL-TR-219145

A preliminary investigation of Large Eddy Simulation (LES) of the flow around a cylinder at ReD = 3900 using a commercial CFD code

J. S. Paschkewitz

February 21, 2006

Disclaimer

A preliminary investigation of Large Eddy Simulation (LES) of the flow around a cylinder at $Re_D = 3900$ using a commercial CFD code

By JOHN S. PASCHKEWITZ

Energy and Environment Directorate, Lawrence Livermore National Laboratory, Livermore, CA, 94551

21 February 2006

Engineering fluid mechanics simulations at high Reynolds numbers have traditionally been performed using the Reynolds-Averaged Navier Stokes (RANS) equations and a turbulence model. The RANS methodology has well-documented shortcomings in the modeling of separated or bluff body wake flows that are characterized by unsteady vortex shedding. The resulting turbulence statistics are strongly influenced by the detailed structure and dynamics of the large eddies, which are poorly captured using RANS models (Rodi 1997; Krishnan *et al.* 2004). The Large Eddy Simulation (LES) methodology offers the potential to more accurately simulate these flows as it resolves the large-scale unsteady motions and entails modeling of only the smallest-scale turbulence structures. Commercial computational fluid dynamics products are beginning to offer LES capability, allowing practicing engineers an opportunity to apply this turbulence modeling technique to much wider array of problems than in dedicated research codes. Here, we present a preliminary evaluation of the LES capability in the commercial CFD solver StarCD by simulating the flow around a cylinder at a Reynolds number based on the cylinder diameter, D, of 3900 using the constant coefficient Smagorinsky LES model. The results are compared to both the experimental and computational results provided in Kravchenko & Moin (2000). We find that StarCD provides predictions of lift and drag coefficients that are within 15% of the experimental values. Reasonable agreement is obtained between the time-averaged velocity statistics and the published data. The differences in these metrics may be due to the use of a truncated domain in the spanwise direction and the short time-averaging period used for the statistics presented here. The instantaneous flow field visualizations show a coarser, larger-scale structure than the study of Kravchenko & Moin (2000), which may be a product of the LES implementation or of the domain and resolution used. Based on this preliminary study, we conclude that StarCD's LES implementation may useful for low Reynolds number LES computations if proper care is used in the problem and mesh definition.

---

1. Introduction

One of the central goals of the LLNL Heavy Vehicle Aerodynamics project is to apply computational fluid dynamics tools to the evaluation of drag-reducing concepts for heavy vehicles. The simulation of time-varying separated flows around bluff bodies like heavy vehicles at high Reynolds numbers is extremely challenging since these flows exhibit strong unsteadiness and complex vortex structures. The Reynolds-averaged Navier Stokes

(RANS) modeling methodology predominant in engineering calculations has well-known shortcomings in accurately simulating this class of flows. For example, steady RANS models have been shown to poorly predict the flow features in a variety of "canonical" problems such as the surface-mounted cube (Rodi 1997), the surface hump (Krishnan *et al.* 2004), and the asymmetric two-dimensional diffuser (Iaccarino 2001), particularly when the popular $k-\epsilon$ turbulence model is employed. More accurate solution methodologies, such as unsteady RANS (Iaccarino *et al.* 2003) as well as more complex turbulence models such as v^2-f (Iaccarino 2001) and large eddy simulation (LES) (Rodi 1997) have been shown to improve the agreement between the experimental and computational data.

Large eddy simulation (LES) has been primarily confined to the realm of research CFD codes until recently; in the last two years, the major commercial CFD code vendors have begun offering LES capability in their products (cd adapco 2005; Kim & Cokljat 2005; cfx 2005). This development offers the exciting possibility of applying the more sophisticated LES model to a wider array of problems than is typically possible in problem-specific research codes. In this report, we present the preliminary results of an investigation of the LES capability in StarCD using the simulation of flow around a cylinder at a Reynolds number of 3900. The $Re_D = 3900$ cylinder flow problem was chosen since it is a well-accepted "canonical" flow that has an extensive database of both computational and experimental results with which to compare; for example see the work of Kravchenko & Moin (2000); Franke & Frank (1998); Breuer (1998) and the references within. This test case is especially relevant since it is an unsteady, bluff-body wake problem that shares many of the characteristics of the truck problems, albeit at a much lower Reynolds number than typically of interest. We compare StarCD's predictions of the mean and fluctuating quantities in the wake as well as integrated measures of lift and drag with this database.

We must emphasize that this report is simply meant to be a "first look" at the LES capability of a commercial CFD solver, and is not to be viewed as either an official endorsement or criticism of the capabilities of StarCD by this project, Lawrence Livermore National Laboratory, the University of California or the Department of Energy. The StarCD CFD code has been used extensively for modeling efforts on the Heavy Vehicle Aerodynamics project at LLNL and at Argonne National Laboratory, and was therefore the logical choice for evaluating a commercial CFD code LES implementation. Based on the results of Iaccarino (2001), we anticipate that similar results would be obtained using any of the other commercial CFD packages offering an LES option such as Fluent or CFX.

This report is organized as follows. In Section 2, we describe the Large Eddy Simulation model. In Section 3, the simulation parameters and the computational domain are described. In Section 4 we present comparisons of StarCD's predictions with those in the literature. Finally, in Section 5 we provide conclusions and recommendations for future work using the LES model in StarCD.

2. Problem Formulation

2.1. *Governing equations for fluid flow*

The equations for the carrier fluid flow are solved using the Large Eddy Simulation (LES) methodology. For the LES simulations we used StarCD's large eddy simulation (LES) implementation. The LES model is based on a spatial filter of the velocity:

$$\overline{U_i}(\mathbf{x},t) = \int G(\mathbf{r},\mathbf{x})U_i(\mathbf{x}-\mathbf{r},t)d\mathbf{r}. \tag{2.1}$$

The exact form of the filter kernel $G(\mathbf{r}, \mathbf{x})$ is not important within the context of the constant coefficient LES models used in StarCD as the application of the filter is never computed. The velocity can then be decomposed into resolved and sub-grid scale components:

$$u_i(x_k, t) = \overline{U_i}(x_k) + u'(x_k, t). \tag{2.2}$$

Inserting this definition into the Navier-Stokes equations, the filtered momentum equations are obtained:

$$\frac{\partial \overline{U_j}}{\partial t} + \frac{\partial \overline{U_i}(\overline{U_j})}{\partial x_i} = -\frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i} + \nu \frac{\partial^2 \overline{U_j}}{\partial x_i \partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j}. \tag{2.3}$$

There remains a closure problem for the sub-grid scale Reynolds stresses. The eddy viscosity hypothesis can be used to relate the Reynolds stresses to the resolved rate of strain, $\overline{S_{ij}}$:

$$\tau^r_{ij} = -2\nu_t \overline{S_{ij}} + \frac{2}{3} k \delta_{ij}. \tag{2.4}$$

To close the problem, an implementation of the constant coefficient Smagorinsky model is used:

$$k = C_k \Delta^2 (S_{ij} S_{ij}); C_k = 0.202, \tag{2.5}$$

$$\nu_t = C_s \Delta^2 (S_{ij} S_{ij})^{1/2}; C_s = 0.02. \tag{2.6}$$

Details of the Smagorinsky model, as well as more sophisticated LES models, can be found in Pope (2000).

Finally, StarCD uses an *ad hoc* implementation of the filter width, Δ, that incorporates a simple wall damping function:

$$\Delta = V^{1/3}; \Delta_c = min(0.042y, \Delta), \tag{2.7}$$

where Δ_c is the filter width used in the solver and y is the distance to the nearest wall. It can be seen that at the wall ($y = 0$) this relationship gives a zero eddy viscosity. Although this is a crude wall function, the StarCD documentation recommends that the near-wall region be fully resolved, with the first cell having a distance less than $y^+ = 1$ and at least 30 cells in the boundary layer, which severely restricts the range of Reynolds numbers that can be accurately modeled using the LES approach (cd adapco 2005).

3. Problem Definition

3.1. *Problem geometry*

The mesh used in this study is shown in Figure 1. This mesh was provided by the Center for Turbulence Research (CTR) at Stanford University for benchmarking studies using their LES research code, CDP. We have used this same mesh here to allow for future comparisons of LES predictions obtained using StarCD with those from the CDP research code. The mesh origin is at the cylinder center. All distances are given in terms of the cylinder diameter, which is equal to one. In the flow direction, the domain extends from -23.5 to +17, for a total length of 40.5 cylinder diameters. The mesh is "D-shaped", with a curved inflow boundary that is parallel to the cylinder surface then extends horizontally in the streamwise direction. At the widest point the mesh has a vertical extent of $y = \pm 29.7D$. Finally, the mesh has a width of $1.0D$ with 32 grid points in the spanwise direction. This domain is smaller than the spanwise domain of πD typically used in numerical studies of the $Re_D = 3900$ cylinder; however in the reducing the computational cost for this preliminary study, the smaller domain was used. We discuss the impact of

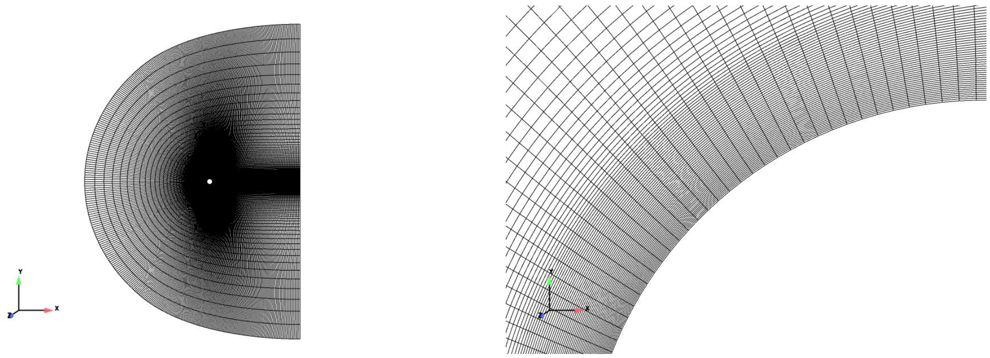

FIGURE 1. Mesh and domain used in simulation. Left: entire domain; Right: detail of near-cylinder mesh

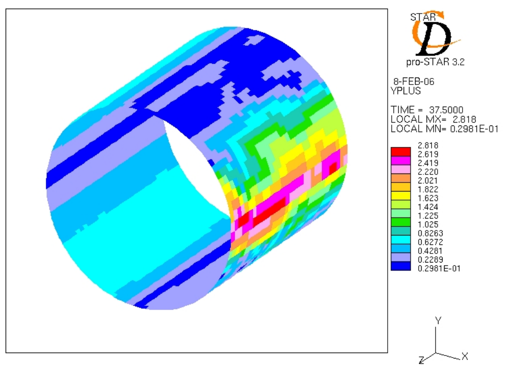

FIGURE 2. (Color) Contour plot from StarCD of local value of y^+ on the cylinder surface.

the smaller domain of the computational results in Section 4. The mesh has a total of 1.53 million cells. Only one mesh was used for this preliminary evaluation of StarCD's LES capabilities.

Mesh points are non-uniformly distributed near the cylinder wall as shown in Figure 1. For the flow of interest here, the y^+ values of the first cell off the cylinder surface range from a minimum of 0.03 to a maximum of 2.81. A contour plot of the distribution of y^+ is provided in Figure 2 and shows that the largest y^+ values are located on the back side of the cylinder. These results suggest that the mesh may be underresolved on the back of the cylinder. We have not investigated this aspect of the problem definition further in this preliminary study.

3.2. *Simulation Parameters*

The problem was defined as a "non-dimensional" problem using a geometric scaling of 1.0, an inlet velocity of 1.0, working fluid density of 1.0, and a viscosity of $1/Re = 2.56\times10^{-4}$. Cyclic boundary conditions were used in the spanwise directions, while a fixed flowrate inlet velocity condition was used for the curved inflow plane. Spurious pressure fluctuations were observed throughout the domain when an outflow condition was used so a constant pressure outlet with $P/(\rho U^2) = 0.0$, which eliminated the fluctuations.

The spatial discretization scheme used in the simulations was StarCD's proprietary Monotone Advection and Reconstruction Scheme (MARS) with a blending factor of 0.75 for both the velocity components and the pressure. The MARS scheme is less dissipative

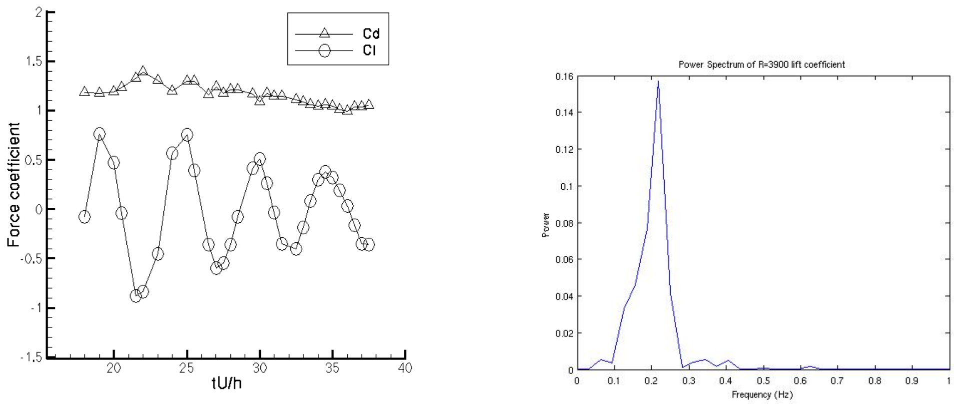

FIGURE 3. Left: Time history of lift and drag coefficients; Right: FFT of lift coefficient to extract mean Strouhal number of 0.21

with higher values of the blending factor (cd adapco 2005). All other numerical settings were kept at the default values in StarCD. Note that StarCD's tutorial on LES simulation of channel flow actually recommends the use of central differencing and tighter convergence criteria; the MARS scheme with the settings employed here was chosen since it is more commonly used in heavy vehicle simulations. The "Pressure Implicit with Splitting Operators" (PISO) scheme (Issa 1986), which is a time-implicit numerical integration scheme, was used with a pressure underrelaxation factor of 1.0.

The simulation flowfield was initialized using a steady RANS solution obtained with StarCD. The solution was then run in a time-dependent manner with a non-dimensional timestep (time scale of D/U) of 0.005, giving a maximum CFL number of 1.23 and mean value of 0.02. The simulation was run for total of 2 flowthrough times (80 time units in total) with averaging over the last 37.5 time units, which was the longest time allowable under project constraints. Time-dependent force data for lift and drag coefficients were obtained on the last 19.5 time units of the simulation. The computations were performed on the MCR Pentium 4 cluster at LLNL using 60 processors. A total of 2500 timesteps, for a total of 2.5 time units, were completed during each run. These jobs required between 1-2 hours of wall time so that the simulations required approximately 2400 hours of CPU time in total.

4. Results

4.1. *Integrated quantities and instantaneous visualizations*

Integrated quantities such as lift, drag and separation angle are the simplest metrics with which to assess the quality of the LES solution obtained using the commercial CFD solver StarCD. We begin with the time-varying lift (C_L) and drag (C_d) coefficients, shown in Figure 3. The drag coefficient shows considerable noise, while the lift coefficient has a much cleaner variation with time. An estimate of the Strouhal number was obtained by performing a Fast Fourier Transform (FFT) on the C_L versus time data; the resulting power distribution shows a peak value at a non-dimensional frequency (or St) of ≈ 0.21.

In Table 1 we present the predicted results for the mean drag coefficient C_D, Strouhal number (St), separation angle (θ_{sep}), length of the recirculation region in the wake $\overline{L_{rec}}/D$, and the minimum value of the mean streamwise velocty ($\overline{U_{min}}$) as compared

Data from	$\overline{C_D}$	St	θ_{sep}	$\overline{L_{rec}}/D$	$\overline{U_{min}}$
Experiment	0.99 ± 0.05	0.215 ± 0.005	86.0°	1.4 ± 0.1	-0.24 ± 0.1
Kravchenko	1.04	0.210	88.0°	1.35	-0.37
2-D (Breuer)	1.625	n/a	100.7°	n/a	n/a
StarCD	1.15	≈ 0.21	90.2°	1.39	-0.33

TABLE 1. Flow parameters for cylinder flow computations at $Re_D = 3900$. Experimental data is a combination of data from the unpublished PIV study of Lourenco and Shih and other sources provided in Kravchenko & Moin (2000). Two-dimensional results are from Breuer (1998).

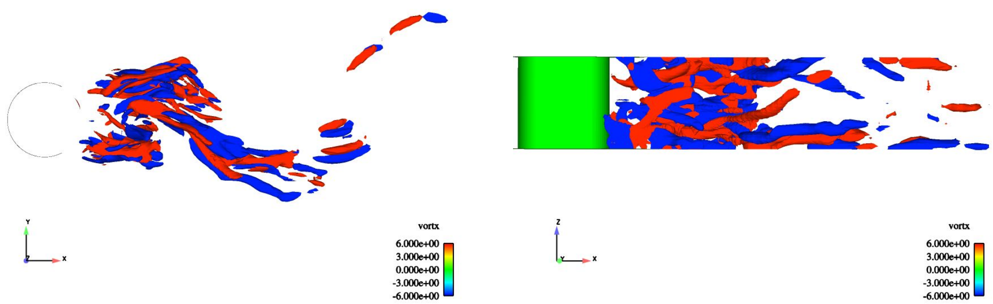

FIGURE 4. (Color) Isosurfaces of instantaneous streamwise vorticity in the flow over a circular cylinder at $Re_D = 3900$ using same visualization criteria as Kravchenko & Moin (2000). Left: x-y plane view. Right: x-z plane view. The positive positive vorticity threshold is $\omega_x D/U = 5.5$ while the negative vorticity threshold is $\omega_x D/U = -5.5$

to selected experimental and computational studies. Overall, the values of these metrics predicted by StarCD show generally good agreement with the study of Kravchenko & Moin (2000) with behavior tending toward the two-dimensional simulation of Breuer (1998). These trends are expected with the short spanwise domain used here; as discussed in Kravchenko & Moin (2000), the wavelength of streamwise structures in the spanwise direction varies from $0.4D$ to greater than $1.0D$. The larger wavelength structures are important in the the downstream wake evolution and mean quantities; these structures are not captured using a span of $1.0D$. This lack of spanwise resolution is likely responsible for the discrepancies in the statistics.

In Figure 4 we present instantaneous visualizations of isosurfaces of the streamwise vorticity, using the same thresholds as Kravchenko & Moin (2000). Compared with Figure 7 in this reference, we see that the instantaneous structures predicted using StarCD have a smoother, larger-scale appearance while showing the same qualitative features. This result may be an artifact of the lower resolution employed here compared to the study of Kravchenko & Moin (2000) or may be a product of the LES implementation; a grid resolution study is necessary to isolate the effects. The spanwise length scales of the structures appears to be roughly one-fourth to one-third of the span (or cylinder diameter) in agreement with the results of Kravchenko & Moin (2000).

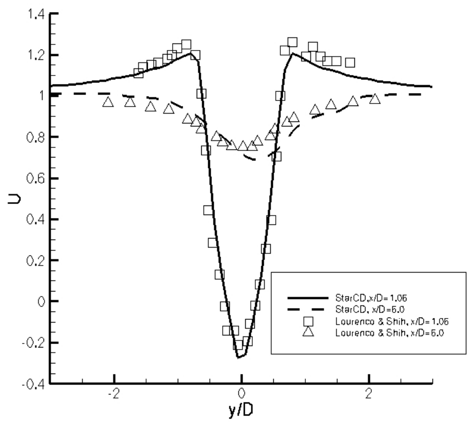

FIGURE 5. Comparison of time-averaged mean streamwise velocity profiles to the experimental data of Lourenco and Shih provided in Kravchenko & Moin (2000)

4.2. *Time-averaged quantities*

More quantitative comparisons can be made using the time-averaged mean and fluctuating velocity data. StarCD automatically calculates running time averages of these quantites when the LES model is enabled; this data was then used to obtain the profiles presented here. In all plots, we compare to the unpublished PIV-based data of Lourenco and Shih presented in Kravchenko & Moin (2000).

In Figure 5, we present profiles of the mean streamwise velocity at two downstream positions. Agreement at both positions is excellent, with both the qualitative shape and quantitative values captured accurately by StarCD.

The streamwise velocity fluctuations are presented in Figure 6 at a single downstream station. We observe similar behavior at other measurement points. In all cases, the qualitative profiles show generally good agreement, but the quantitative values differ slightly from the benchmark data. Fluctuation profiles at all locations are also quite noisy, suggesting that a much longer averaging time is necessary to obtain clean statistics for these metrics. The results of Franke & Frank (1998) suggest that much longer averaging times, on the order of 100-200 time units, are required to obtain truly converged statistics.

Finally, in Figure 7 we present the profile of the Reynolds shear stress ($\overline{u'v'}$) at a single station. Again we see qualitative correspondence with the experimental result, but considerable scatter in the data and substantial quantitative disagreement. The large difference in the peak value suggests that the flowfield has much larger correlations between streamwise and normal velocity fluctuations, which can be related to the vortical structures in the wake. The instantaneous visualizations (Fig. 4) show a larger-scale and more coherent structure than would be expected and this non-physical vortex structure would be reflected in larger Reynolds shear stresses. There is also a noticeable asymmetry in the profile, which may be an artifact of the short averaging interval.

5. Conclusions and Recommendations

We have performed a preliminary study of the $Re_D = 3900$ flow around a cylinder using an implementation of the Large Eddy Simulation (LES) turbulence model in the commercial CFD program StarCD. The results show reasonable agreement with the

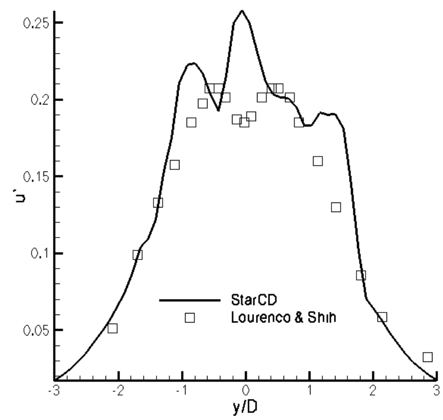

FIGURE 6. Comparison of time-averaged r.m.s. streamwise velocity fluctuation profile at $x/D = 6.0$ to the experimental data of Lourenco and Shih provided in Kravchenko & Moin (2000)

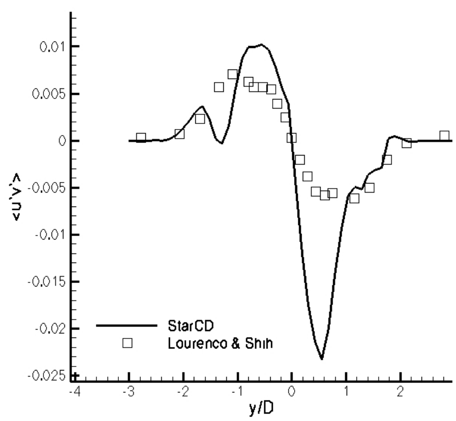

FIGURE 7. Comparison of time-averaged Reynolds stress $(\overline{u'v'})$ profile at $x/D = 6.0$ to the experimental data of Lourenco and Shih provided in Kravchenko & Moin (2000)

published data, especially given the truncated domain in the spanwise direction, the short time period used for averaging and the dissipative spatial discretization scheme (MARS) used. These preliminary results suggest that with a carefully prepared mesh and care in the definition of the boundary conditions - specifically, use of the outflow boundary condition is not recommended for wake problems in StarCD - a commercial code with an LES implementation may be a useful tool for studies of unsteady, separated turbulent flows such as those of interest on the Heavy Vehicle Aerodynamics project at LLNL. Future studies should more carefully explore this test case or other canonical problems with a greater range of mesh resolutions and longer averaging times for a more thorough comparison of flow statistics with published data. Additionally, a fuller exploration of the impact of the spatial differencing scheme on the quality of the results should be completed.

6. Acknowledgments

This work was performed under the auspices of the U.S. Department of Energy by the University of California, Lawrence Livermore National Laboratory under Contract No. W-7405-ENG-48. Helpful discussions with Rose McCallen, Kambiz Salari, Jason Ortega, Paul Castellucci and Craig Eastwood are gratefully acknowledged. UCRL-TR-219145

REFERENCES

CD ADAPCO 2005 StarCD v.3.24, http://www.cd-adapco.com/products/STARCD/index.html.

BREUER, M. 1998 Numerical and modeling influences on large eddy simulations of the flow past a circular cylinder. *Intl. J. Heat and Fluid Flow* **19**, 512–521.

CFX 2005 Http://www.ansys.com/products/cfx_features.asp.

FRANKE, J. & FRANK, W. 1998 Large eddy simulation of the flow past a circular cylinder at $re_d = 3900$. *Intl. J. Heat and Fluid Flow* **19**, 512–521.

IACCARINO, G. 2001 Predictions of a turbulent separated flow using commercial cfd codes. *ASME J. Fluids Eng.* **123**, 819–828.

IACCARINO, G., OOI, A., DURBIN, P. & BEHNIA, M. 2003 Reynolds averaged simulation of unsteady separated flow. *Intl. J. of Heat and Fluid Flow* pp. 147–156.

ISSA, R. 1986 Solution of implicitly discretized fluid flow equations by operator splitting. *J. Comp. Phys.* **62**, 40–65.

KIM, S. & COKLJAT, D. 2005 LES is more. *Fluent News* Http://www.fluent.com/about/news/newsletters/05v14i1/a1.htm.

KRAVCHENKO, A. & MOIN, P. 2000 Numerical studies of flow over a circular cylinder at $re_d = 3900$. *Phys. Fluids* **12** (2), 403–417.

KRISHNAN, V., SQUIRES, K. & FORSYTHE, J. 2004 Prediction of separated flow characteristics over a hump using RANS and DES. *Tech. Rep.*. AIAA-2004-2224.

POPE, S. 2000 *Turbulent Flows*. Cambridge Univ. Press.

RODI, W. 1997 Comparison of LES and RANS calculations of the flow around bluff bodies. *J. Wind Eng. and Ind. Aero.* **69–71**, 55–75.