Isı Bilimi ve Tekniği Dergisi, 29, 1, 91-98, 2009
J. of Thermal Science and Technology

ISSN 1300-3615

UNSTEADY TURBULENT VORTEX STRUCTURE DOWNSTREAM OF A THREE DIMENSIONAL CYLINDER

Selin ARADAG*
*TOBB University of Economics and Technology , Department of Mechanical Engineering
06560, Ankara, saradag@etu.edu.tr

(Geliş Tarihi: 15. 12. 2008, Kabul Tarihi: 13. 02. 2009)

Abstract: This study examines the near wake behavior of the turbulent, unsteady flow over a three dimensional circular cylinder at Re=20,000, based on cylinder diameter, computationally. From a CFD perspective, this work assesses the capability of unstructured grids and large eddy simulations without a subgrid scale model, for the prediction of flow behavior at transitional Reynolds numbers for the time-dependent flow over a circular cylinder. Proper orthogonal decomposition is also applied to the computational results in order to investigate the flow structure further and examine the energy content of the turbulent flow at hand. The results of the computations are in close agreement with the experimental results in literature. The energy content of the POD modes reveals that only four spatial modes have 98% of the total energy in the flow and are enough to represent the flow.
Keywords: Computational fluid dynamics, Proper orthogonal decomposition, Three dimensional cylinder, Turbulence, Von Karman vortices, Vortex shedding.

ÜÇ BOYUTLU SİLİNDİR ARKASINDAKİ ZAMANLA DEĞİŞEN TÜRBÜLANSLI GİRDAP YAPISI

Özet: Bu çalışmada üç boyutlu silindir çapına gore hesaplanan Reynolds sayısı 20,000 olan silindir üzerindeki zamana bağlı değişen türbülanslı akış sayısal olarak incelenmiştir. Sonuçlar litern atürdeki deneysel çalışmalarla kıyaslanmıştır. Hesaplamalı akışkanlar dinamiği (HAD) açısından bakıldığında çalışma, düzensiz ağların ve Large Eddy Simulation yönteminin alt ağ yöntemi kullanmadan laminerden türbülanslı akışa geçiş Reynolds sayılarına denk gelen bir akış rejimi için kullanılabilirliğini test etmektedir. Sayısal sonuçlara dikgen ayrıştırma yöntemi uygulanarak eldeki türbülanslı akışın enerji dağılımı ve akış özellikleri ayrıntılı olarak incelenmiştir. Sayısal analiz sonuçları literatürdeki deneysel çalışmaların sonuçlarıyla uyum içerisindedir. POD kip enerjilerini gösteren grafikler, sadece dört mekansal kipin toplam enerjinin %98'ini oluşturduğunu ve akışı modellemek için yeterli olduğunu göstermektedir.
Anahtar Kelimler: Dikgen ayrıştırma, Girdap oluşumu, Hesaplamalı akışkanlar dinamiği, Türbülans, Üç boyutlu silindir, Von Karman girdapları.

NOMENCLATURE

D:	Diameter of the cylinder
L:	Length of the cylinder
M:	Mach number
Re:	Reynolds number
St:	Strouhal number for the vortex shedding
U:	Free-stream velocity
f:	Shedding frequency of the flow
$y^+_{average}$	Average non-dimensional first cell height
Δt:	Time step

INTRODUCTION

Vortex shedding behind bluff bodies has been a subject of extensive research. Many flows of engineering interest produce this phenomenon Applications include aircraft and missile aerodynamics, marine structures, underwater acoustics, and civil and wind engineering (Cohen et al, 2005). The control of the wake behind bluff bodies has been very important in several engineering applications. In order to be able to control the vortex shedding behind bluff bodies, it is first important to predict the flow structure accurately.

A circular cylinder is a well documented benchmark for a bluff body wake problem. The near wake for the flow past a circular cylinder determines the dominant instability in the flow which leads to the vortex street formation (Unal and Rockwell, 1988). There is no established way to define the near wake; however it was defined as the region up to ten diameters downstream from the cylinder by Ma et al (2000).

There are several experimental and computational studies in the literature dealing with the instability and transition in the near wake of a circular cylinder. Norberg (1994), Zdravkovich (1990), Lin et al (1995) and Dong et al (2006) are some of the theoretical and experimental studies that give insight to the instability and transition in the near wake. Dong et al (2006) also involves a detailed direct numerical simulation (DNS) analysis at Reynolds numbers of 3900 and 10,000 that compares well with Particle Image Velocimetry (PIV) measurements.

Most of the current CFD approaches use Reynolds-Averaged Navier-Stokes equations (RANS) for the prediction of turbulent flows. Although RANS models are sufficient for predicting time averaged flow quantities, they are not adequate in predicting flows with large separation and the resulting unsteadiness. These massively separated flows include geometry dependent and three dimensional turbulent eddies which can not be simulated by RANS turbulence models (Hansen and Forsythe, 2003). Direct Numerical Simulations (DNS), on the other hand, makes no modeling assumption but is the most expensive approach since all turbulent motions must be resolved by the grid. Since the smallest scales of turbulence (the Kolmogorov length scale) decrease rapidly with increasing Reynolds number, this approach is limited to relatively low Reynolds number flows. Large Eddy Simulation (LES) is less expensive than DNS since it models only the small subgrid scales of motion and resolves the rest of the turbulent motions. However, since the "large" scales in the boundary layer are on the order of the boundary layer thickness (which is quite thin for high Reynolds number flows), this method is cost prohibitive at high Reynolds numbers for wall bounded flows.

The flow at a Reynolds number of Re=3900 has been most extensively studied computationally in the transition range since there are several experimental studies in literature for comparison purposes at this Reynolds number. Beaudan and Moin (1994) performed LES of flow at this Reynolds number and they assessed the performance of the dynamic subgrid-scale eddy viscosity model for the turbulent wake behind a circular cylinder. They also showed that upwinding schemes are very dissipative for this type of flow. Mittal and Moin (1997) and Kravchenko and Moin (2000) also numerically studied the flow over a circular cylinder at Re=3900 using LES. Kravchenko and Moin (2000) used a high-order accurate numerical method based on B-splines and compared their results with previous upwind-biased and central finite difference simulations. They found out that the results show similar trend for the very near wake. Franke and Frank (2002) investigated the effect of averaging times on mean values. They showed that the short averaging times used in previous LES computations do not lead to converged mean values. Direct numerical simulations (DNS) of flow at Re= 3900 were performed by Ma et al (2000) and Dong et al (2006). Ma et al performed the simulations using a spectral element scheme on unstructured grids. The mean velocity profiles they obtained agree well with the experiments in the vicinity of the cylinder. Jordan (2002) studied a higher Reynolds number of Re=8000 in the transitional range using LES. However, he collected a short time history which may affect the results. Although shear layer frequency and a number of other quantities were captured well, the lift coefficient he obtained is different than experiments (Norberg, 2003). There are also two-dimensional simulations in the literature (Braza et al, 1990). However, Beaudan and Moin (1994) performed both two and three dimensional computations and showed that the near wake is highly three dimensional at transitional Reynolds numbers and it contains pairs of counter-rotating streamwise vortices, the effect of which cannot be produced in two-dimensional computations. Three-dimensional computations are essential for predicting flow statistics of engineering interest. It was found that LES with subgrid scale eddy viscosity model was very dissipative (Beaudan and Moin, 1994).

Simulations of flow around complex configurations offer a cost-effective tool for analysis. Larger, more complex problems can now be handled by computational fluids dynamics; however, more complex geometries have created a larger burden on grid generation. Unstructured grids which were also used in the computations presented in this study can alleviate this burden by reducing the complexity of grid generation and they also offer excellent parallel performance (Hansen and Forsythe, 2003).

RESEARCH OBJECTIVES

There are two fundamental objectives of this study. The first aim is to perform the CFD analysis of the flow over a three dimensional circular cylinder at a Reynolds number of 20,000 based on the cylinder diameter to access the capability of unstructured grids and large eddy simulations without a subgrid scale model for the prediction of turbulent flow over a circular cylinder. The next objective is to apply Proper orthogonal decomposition (POD) to the computational results in order to further reveal the flow structure and examine the energy content of the flow and to access the capability of POD to analyze the structure of this type of flow.

COMPUTATIONAL METHODOLOGY

For the computations, the solver Cobalt from Cobalt Solutions, LLC, was used (Strang et al, 1999). It is a commercial code which solves the compressible Navier-Stokes equations using a cell-centered finite volume approach applicable to arbitrary cell topologies. (e.g. prisms, tetrahedra) In order to provide second order accuracy in space, the spatial operator utilizes an exact Riemann solver (Gottlieb and Groth, 1988) and, least squares gradient calculations use QR factorization. It also employs TVD flux limiters to limit extremes at cell faces. A point implicit method using analytic first-order inviscid and viscous Jacobians is used for advancement of the discretized system. A Newton sub-iteration scheme is employed to achieve second order accuracy in time.

The flow was simulated using Large Eddy Simulations with no explicit subgrid scale model, along with unstructured grids. The numerical dissipation of the code was relied upon to remove the energy from the resolved scales, mimicking the effect of turbulence at the subgrid level, an approach also used by Hansen and Forsythe (2003). At the Reynolds number of 20,000, the attached boundary layer on the cylinder surface is laminar but the wake is fully turbulent.

The flow was simulated at a Mach number of 0.1. The length to diameter ratio of the cylinder was 4. It was shown by Lim and Lee (2002) that if the cylinder cannot be considered as infinitely long, where the length to diameter ratio is around 3.8, the three dimensionality of the cylinder plays role in the analysis. The diameter of the cylinder was 2m. A periodic boundary condition was used on the computational surfaces at the cylinder ends, modified Riemann invariants were used as a farfield boundary condition, whereas a no slip, adiabatic wall was employed for the cylinder surface. A time step of $1.152 \cdot 10^{-3}$ seconds corresponding to 0.02 non-dimensional time steps was used in the original computations; however a time step study was also performed and a time step of $0.576 \cdot 10^{-3}$ seconds was also used to validate the results. Time was non-dimensionalized by dividing by D/U where, the D is the cylinder diameter and U is the free-stream velocity. The total simulation time was 10,000 time steps, corresponding to 200 non-dimensional time steps and 40 cycles of vortex shedding for the computations with the larger time step. The static pressure and temperature are 457.79 Pa and 300 K, respectively. In order to increase the stability, an advection damping coefficient of 0.01 was used in the computations. No damping was used for diffusion. To make the change of the variables at every time step converge, 3 Newton sub-iterations were used. As an initial perturbation to trigger the unsteadiness in the flow simulations, the incoming flow was skewed by an angle of attack of 1 degree.

The two grids employed in the computations consist of clustering of prismatic cells in the boundary layer and tetrahedral cells outside the boundary layer. These were generated by Gridtool/VGRID. The minimum cell height is 1.10^{-3} m for the coarse grid, which corresponds to 5×10^{-4} when non-dimensionalized by the cylinder diameter. $y^+_{average}$ (average non-dimensional first cell height for the boundaries) is around 0.3. The grid has 879,603 cells. The farfield boundaries are 20 diameters away from the cylinder surface. The fine grid consists of 2,311,000 cells with a minimum cell height of $0.5\ 10^{-3}$. The coordinate system used in the computations and the grid around the cylinder surface are shown in Figure 1. x is the streamwise direction and y axis aligns with the cylinder axis. The origin is on the center of the circular face at the back of the cylinder. Figure 2 shows the surface grid at the cylinder ends.

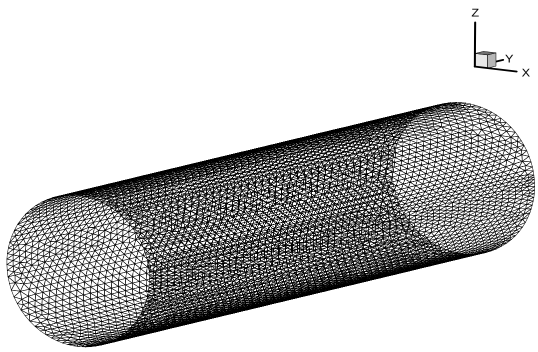

Figure1. Surface grid of cylinder.

For proper orthogonal decomposition of the wake behavior, the data from the simulations with the smaller time step ($0.576 \cdot 10^{-3}$ seconds) was used. 20 cycles of vortex shedding were utilized with a temporal resolution of 500 time steps per von Karman shedding cycle.

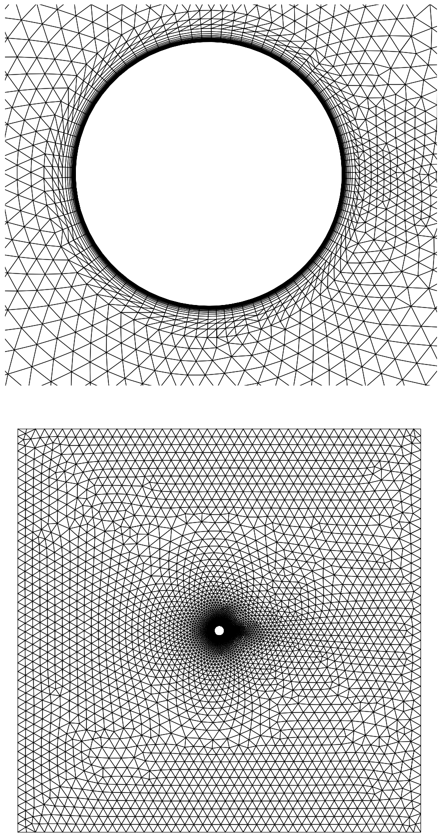

Figure 2. Surface mesh around the periodic surfaces.

All the computations were performed on a parallel processing environment where 80 processors in parallel were used for the computation of the complex 3D, turbulent flow over a circular cylinder.

PROPER ORTHOGONAL DECOMPOSITION

The Proper Orthogonal Decomposition (POD) also known as Karhunen-Loeve Expansion, is an optimal decomposition since it sorts the spatial modes with respect to their energy content. The method of snapshots introduced by Sirovich (1987) is used here for reducing the order of the eigenvalue problem. The details of POD can be found in literature (Holmes et al, 1996; Newman, 1996). In this study, proper orthogonal decomposition is applied to U-velocity.

RESULTS AND DISCUSSION

Time averaged drag coefficient is equal to 1.20, which is within the range of the experimental result provided by Lim and Lee (2002) and Anderson (1991) seperately. The drag coefficient is 1.20 according to Anderson (1991) and 1.16 according to Lim and Lee (2002). Time averaging is started at t = 60, corresponding to twelve vortex shedding cycles. Figure 3 shows the time history of the drag coefficient with two different grid resolutions and Figure 4 shows the drag coefficient computed using two different time steps. For these plots, the drag coefficient was calculated based on the drag force:

$$C_D = \frac{F_x}{\frac{1}{2}\rho U^2 A} \tag{1}$$

where U is the free-stream velocity, ρ is the free-stream density and A is the frontal area. Figure 3 shows that the results can be considered grid-independent, as far as a flow field with an absolute instability such as the cylinder wake, where there are small scale fluctuations because of turbulence, is concerned.

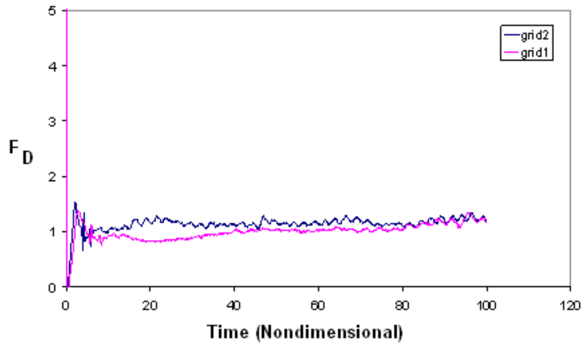

Figure 3. Grid Refinement Study (Drag Coefficient versus time – time is nondimensional).

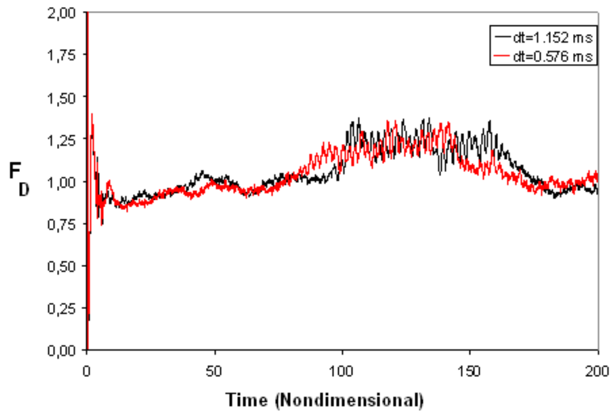

Figure 4. Time step study (Drag Coefficient versus time – time is nondimensional).

The turbulence intensity fluctuations at the centerplane of the cylinder (z/D=0, y/D=2) is shown in Figure 5. The turbulence intensity is given by $\sqrt{u'u'}\,/Ux100$, where $u'u'$ is the streamwise Reynolds stress component.

Lim and Lee (2002) defined the end of the length of the vortex formation region as the peak in the turbulent intensity distribution. The x-location of the peak value of the turbulent intensity and the magnitude of the turbulent intensity are measured experimentally at

Reynolds numbers of 16,000 and 24,000 by Lim and Lee (2002) as shown in Table 1. The computed value for the length of the vortex formation region is within 1% and the peak magnitude of turbulent intensity is within 3% of the average of the measured results by Lim and Lee (2002) at Reynolds numbers of 16,000 and 24,000 based on the cylinder diameter.

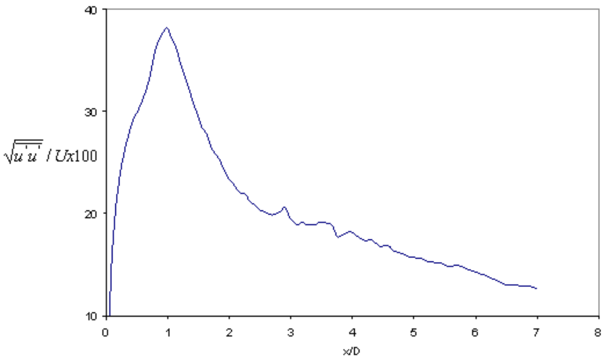

Figure 5. Turbulence Intensity Distribution Along the Wake Centerline.

Table 1. Comparison of Time-averaged Quantities with Experimental Results.

	Length of vortex formation region (x/D)	Peak magnitude of turbulent intensity ($\sqrt{u'u'}/Ux100$)
Lim and Lee (2002), Re=16,000	1.0	37.0
Lim and Lee (2002), Re=24,000	1.0	37.0
Present study Re=20,000	0.99	38.1

Time-averaged values for the iso-surfaces of y-vorticity are shown in Figure 6. Time-averaged contours of y-vorticity at several planes in the flow direction are shown in Figure 7. Both figures show two counter rotating vortices. In figure 6, the vortex on the upper part turns in the counterclockwise direction, whereas the other vortex is in the clockwise direction. In figure 7, positive numbers show the counterclockwise vortices, whereas the negative numbers show clockwise vortices. As seen in Figures 6 and 7, both large scale von Karman vortices and small scale turbulent fluctuations can be well captured in the computations. Also, in Figure 7, the three-dimensional nature of the flow field can be observed.

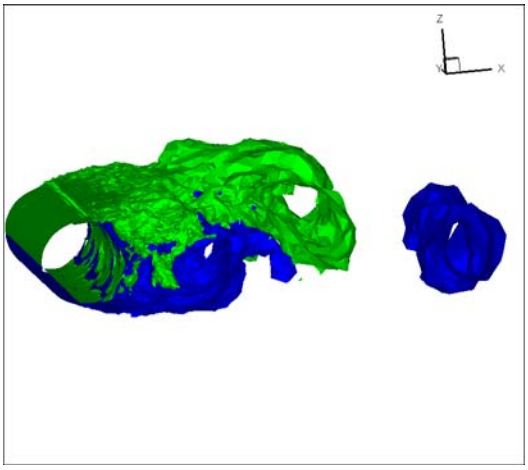

Figure 6. Time-averaged iso-surfaces of y-vorticity, green=20, blue=-20.

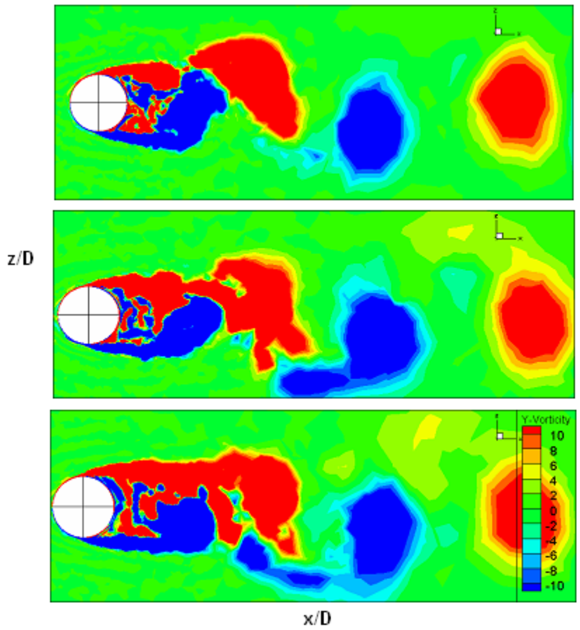

Figure 7. Time-averaged y-vorticity Contours on planes a) y/D=1, b) y/D=2 (centerplane), c) y/D=3.

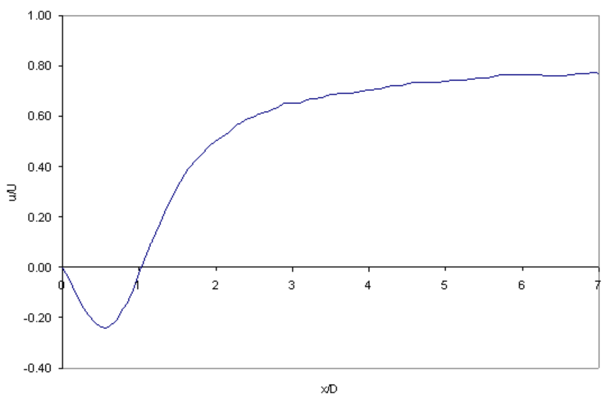

Figure 8. Streamwise velocity profile on centerplane of the cylinder.

The non-dimensional streamwise velocity profile on the centerplane (y/D=2, z/D=0) is shown in Figure 8. The velocities are negative in the near wake, and they change sign as we go further downstream.

The time-averaged pressure coefficient on the centerplane (y/D=2) computed using two different grids and from the experimental study of Lim and Lee (2002) is shown in Figure 9. Cp is the surface pressure coefficient obtained from surface pressure values around the circumference of the cylinder, using:

$$C_P = \frac{P - P_\infty}{\frac{1}{2}\rho U^2} \tag{2}$$

where P is the static pressure, and P_∞ is the freestream static pressure.

Pressure coefficient decreases at the point where the separation occurs, as expected. The results are in good agreement with the experimental results of Lim and Lee (2002), where the average value of the deviation from experimental results is less than 1% up to the separation point for both the coarse and fine grids and on the overall, it is 3.8% for the fine grid and 4.5 % for the coarse grid computations. The horizontal axis, Angle, is defined from the x axis, increasing in the counter clockwise direction.

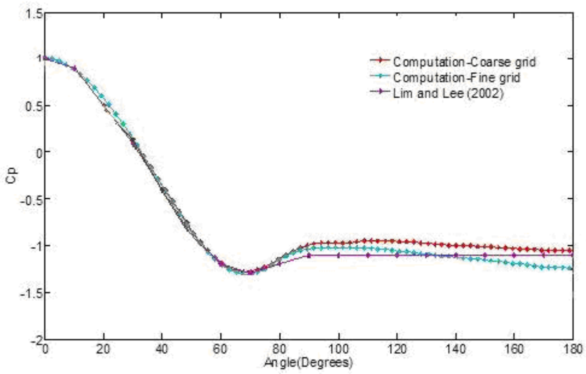

Figure 9. Time-averaged Surface Pressure Distribution on the centerplane of the cylinder.

The first four three dimensional POD spatial modes obtained from the CFD data are shown in Figure 10, where all the grid points in the solution were used for proper orthogonal decomposition. As seen in Figure 10, the second and third modes are showing the von Karman vortex street for the cylinder wake superimposed with small scale turbulent fluctuations present in the flow. The POD spatial modes show that both small scale fluctuations and the von Karman vortex street present in the flow are well captured with the computational procedure.

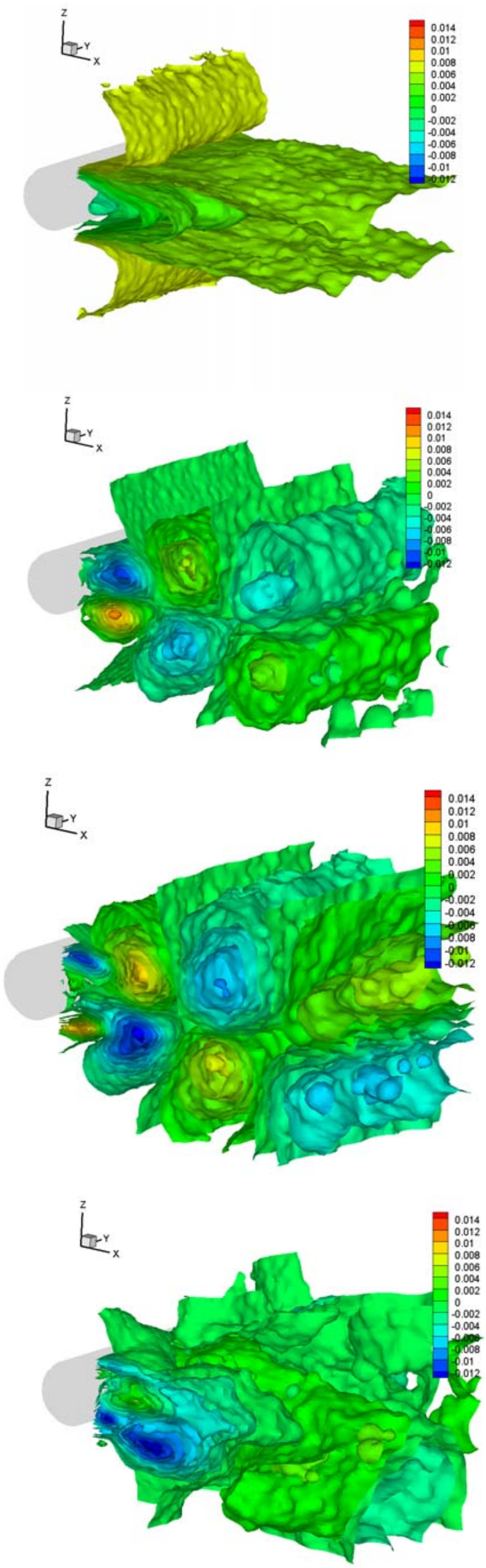

Figure 10. Spatial POD modes U modes 1, 2, 3, 4 (m/s) from 3D Proper Orthogonal Decomposition of the CFD data.

POD is a procedure which distinguishes between several structures of the flow based on the energy content. (Sirovich, 1987). Therefore, one of the aims of POD is to reveal the energy content of structures in the flow. Figure 11 shows the non-dimensional energy content of the first nine modes in the proper orthogonal decomposition. Most of the energy is accumulated in the first four modes as shown in Figure 11. Mode 4 contains 1% of the total energy of the flow, whereas the modes after mode 4 contain less energy. 98% of all the energy for the flow is accumulated in the first four modes only, which shows that using only 4 modes, the velocity field can be reconstructed and still 98% of all the flow features can be represented, which is usually sufficient for especially flow control applications.

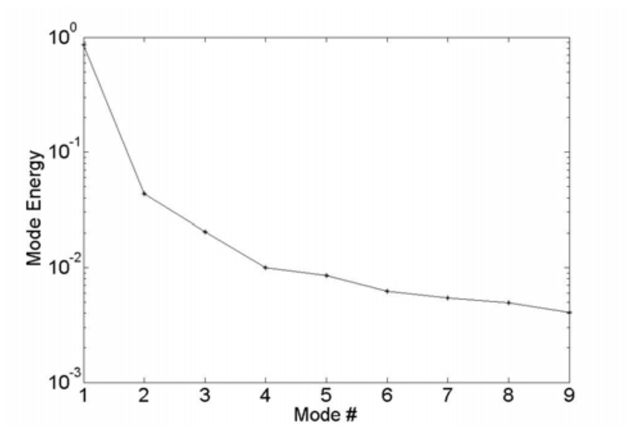

Figure 11. Mode Energies.

CONCLUSION

This study was performed to access the capability of unstructured grids and large eddy simulations with no subgrid scale model for the three dimensional flow over a circular cylinder at a Reynolds number based on cylinder diameter, where the incoming flow and the flow at the cylinder surface is fully laminar, however the flow at the wake is turbulent. Unsteady, three dimensional computations of flow over a circular cylinder were performed in a parallel processing environment using unstructured grids and large eddy simulations without a subgrid scale model, for the prediction of flow behavior. The results of the computations are in close agreement with the experimental results in literature in terms of several steady and time-dependent quantities such as drag coefficient, the structure of the von Karman vortices, pressure coefficient on the cylinder surface and turbulence intensity, demonstrating that LES with no subgrid scale model can be used in predicting the flow where the Reynolds number is a transitional Reynolds number, where the flow is not fully turbulent; however it is mixed, such that the incoming flow and the boundary layer are laminar and the cylinder wake is fully turbulent. Utilizing unstructured grids is also an important tool to gain knowledge about the flow structure in such flows.

Proper orthogonal decomposition was also applied to the computational results in order to investigate the flow structure further and examine the energy content of the turbulent flow. The results show that both the large scale von Karman vortices and small scale turbulence fluctuations can be well captured with the CFD method used. The energy content of the POD modes reveals that only four spatial modes have 98% of the total energy in the flow and are enough to represent the flow.

ACKNOWLEDGEMENTS

All the computations related to this work were performed at US Air Force Academy Blackbird computer cluster. The author would like to thank to the Director of the Aeronautics Research Center Dr. Tom McLaughlin and the director of the Modeling and Simulation Research Center Dr. Keith Bergeron, as well as the Flow Control team including Dr's Stefan Siegel, Jurgen Seidel and Kelly Cohen for their help, support and helpful comments. This work was sponsored as a part of Air Force Office of Scientific Research Project Grant no: FA9550-05-C-0048.

REFERENCES

Anderson, J. D., *Fundamentals of Aerodynamics*, Second Edition, McGraw-Hill, New York, 1991.

Beaudan, P. and Moin, P., Numerical Experiments on the Flow past a Circular Cylinder at Subcritical Reynolds Number, Report TF-62, Department of Mechanical Engineering, Stanford University, 1994.

Braza, M., Chaissaing, P. and Ha Minch, H., Prediction of Large Scale Transition Features in the Wake of a Circular Cylinder, *Physics of Fluids*, Vol A2, pp.1461-1521, 1990.

Cohen, K., Siegel, S., McLaughlin, T., Gillies, E. and Myatt, J., Closed-loop approaches to control of a wake flow modeled by the Ginzburg–Landau equation, *Computers and Fluids*, Vol. 34, pp. 927-949, 2005.

Dong, S., Karniadakis, G. E., Ekmekci, A. and Rockwell, D., A Combined Direct Numerical Simulation-Particle Image Velocimetry Study of the Turbulent Near Wake, *Journal of Fluid Mechanics,* Vol 569, pp. 185-207, 2006.

Franke, J. and Frank, W., Large Eddy Simulation of the Flow past a Circular Cylinder at Re_D=3900, *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 90, Issue 10, pp. 1191-1206, 2002.

Gottlieb, J. J. and Growth, C. P. T., Assessment of Riemann Solvers for Unsteady One-Dimensional Inviscid Flows of Perfect Gases, *Journal of Computational Physics*, pp. 437-458, 1988.

Hansen, R., Forsythe, J., Large and Detached Eddy Simulations of a Circular Cylinder Using Unstructured Grids, *41st Aerospace Sciences Meeting and Exhibit*, Reno, Nevada, AIAA-2003-775, January 2003.

Holmes, P., Lumley, J. L., Berkooz, G., Turbulence, Coherent Structures, Dynamical Systems and Symmetry, *Journal of Fluid Mechanics*, Vol. 322, 215-241, 1996.

Jordan, S. A., Investigation of the Cylinder Separated Shear-Layer Physics by Large Eddy Simulation, *International J. Heat Fluid Flow*, Vol. 23, pp 1-12, 2002.

Lim, H. and Lee, S., Flow Control of Circular Cylinders with Longitudinal Grooved Surfaces, *AIAA Journal*, Vol. 40, No:10, October 2002.

Lin, J. C., Towfighi, J. and Rockwell, D., Instantaneous Structure of a Near Wake of a Cylinder: On the Effect of Reynolds Number, *J. Fluids Struct.,* Vol. 9, pp. 409-418, 1995.

Kravchenko, A. G. and Moin, P., Numerical Studies of Flow over a Circular Cylinder at Re_D=3900, *Physics of Fluids*, Vol. 12, Issue 2, pp. 403-417, 2000.

Ma, X., Karamanos, G. S. and Karniadakis, G. E., Dynamics and Low-Dimensionality of a Turbulent Near Wake, *Journal of Fluid Mechanics*, Vol. 410, pp. 29-65, 2000.

Mittal, R. and Moin, P., Stability of Upwind-Biased Finite Difference Schemes for Large- Eddy Simulation of Turbulent Flows, *AIAA Journal*, Vol. 35, No.8, pp. 1415-1417, 1997.

Newman, A. J., Model Reduction via the Karhunen Loeve Expansion Part 1: An Exposition, Technical Research Report, 96-32, ISR, 1996.

Norberg, C., An Experimental Investigation of the Flow around a Circular Cylinder: Influence of Aspect Ratio, *Journal of Fluid Mechanics*, Vol. 258, pp. 287-316, 1994.

Norberg, C., Fluctuating Lift on a Circular Cylinder: Review and New Measurements, *J. Fluids Struct.,* Vol. 17, pp.57-96, 2003.

Strang, W. Z., Tomaro, R. F., Grismer, M. J., The Defining Methods of Cobalt60: A Parallel, Implicit, Unstructured Euler/Navier Stokes Flow Solver, *AIAA 99-0786*, January 1999.

Sirovich, L., Turbulence and the dynamics of coherent structures part I: Coherent structures, *Quarterly of Applied Mathematics*, Vol. 45, No:3, 561-571, 1987.

Unal, M. and Rockwell, D., On Vortex Shedding from a Cylinder, Part 1: The Initial Instability, *Journal of Fluid Mechanics*, Vol. 190, pp. 491-512, 1988.

Zdravkovich, M. M., Conceptual Overview of Laminar and Turbulent Flows Past Smooth and Rough Circular Cylinders, *J. Wind Engineering Indust. Aero*, Vol. 33, pp. 53-62, 1990.

Selin ARADAĞ

Born in Ankara, Turkey in 1979, Selin Aradağ received BS and MS degrees from Middle East Technical University in 2000 and 2002 respectively and a Ph.D. from Rutgers University, New Jersey, USA in Mechanical an Aerospace Engineering in 2006. Before joining the faculty of TOBB University of Economics and Technology as an Assistant Professor in 2008, she worked for the US Air Force Academy for two years. Her areas of expertise are fluid mechanics, computational fluid dynamics, parallel computing, high speed flows and flow control. She is currently the vice chair of the Mechanical Engineering Department at TOBB University of Economics and Technology. She is married and has a son.