ISSN (Online) : 2455 - 3662
SJIF Impact Factor :3.967

EPRA International Journal of Multidisciplinary Research

Monthly Peer Reviewed & Indexed
International Online Journal

Volume: 3 Issue: 1 January 2017

Published By :
EPRA Journals

SJIF Impact Factor: 3.967 Volume: 3 | Issue: 1 | January 2017

EPRA International Journal of Multidisciplinary Research (IJMR) ISSN (Online): 2455-3662

LARGE AND DETACHED EDDY SIMULATION OF FLOW AROUND A CIRCULAR CYLINDER

Arjun K.S[1]
[1]Department of Mechanical Engineering,
SunRise University,
Alwar – 301030,
Rajasthan, India

Anil Mahwar[2]
[2]Department of Mechanical Engineering,
SunRise University, Alwar – 301030,
Rajasthan, India;

ABSTRACT

This study focuses on the simulation of flow around a simple circular cylinder with various aspect ratios (1:2 to 1:8), in which unsteady nature and vortex shedding of flow are commonly found, using Computational Fluid Dynamics. Various turbulence models have been tested to develop understanding and proper modelling techniques for the flow around such bodies. The Detached Eddy Simulation is found to be a suitable turbulence model for the simulation of the flow. All simulations, have provided the necessary skills and knowledge for the investigation using Computational Fluid Dynamics. Satisfactory results have been observed for the simulation of the flow around a circular cylinder using Large Eddy Simulation (space filtering model) and Detached Eddy Simulation models (SST-DES model). Vortex shedding phenomenon has been successfully captured by Large Eddy Simulation and Detached Eddy Simulation models for the flow around a circular cylinder. Through comparative study with experimental work, the turbulence models, Large Eddy Simulation and Detached Eddy Simulation have been successfully validated at Reynolds number 3900. Detached Eddy Simulation saved about 1.5 to 2.5 times of computational time than Large Eddy Simulation in the simulations of the flow around circular cylinder with lower Reynolds number.

KEYWORDS: *Circular Cylinder; Large Eddy Simulation; Detached Eddy Simulation; Aspect Ratio; Computational Fluid Dynamics*

1. INTRODUCTION

Bluff bodies such as circular cylinders are structures with shapes that significantly disturb the flow around them, as opposed to flow around a streamlined body. These flows are characterized by flow separation and vortex shedding phenomena. LES is an approach that computes the more dynamic eddies and models the smaller isotropic scale motion and known to maintain and predict the vortex structure very well.

Bloor [1] focused on the flow in the near wake region of the cylinder. Experimental work on the pressure and velocity distribution on circular cylinder flow between Reynolds number of 10 to 80 was carried out [2]. Very good agreement with numerical results was observed at this Reynolds number due to the laminar and two-dimensional nature of the flow. Satisfactory results have been obtained between Re of 100 to 300 from CFD simulation [3, 4].

Spalart [5] employs a one-equation turbulence model, the Spalart-Allmaras model [6] in the DES while Menter and Grotjans [7] uses the SST model [8] in the formulation of the DES approach in Central Florida Expressway Authority. The one-equation model is computationally undemanding compared to other more complex RANS models.

Simulations performed on the flow past a blunt trailing edge body and test on shock-induced separation using the one-equation model gives satisfactory results [6]. But the shortcoming of the model is the prediction of the reattachment of flow near adverse pressure gradient region. On the other hand, the SST model performs well in the prediction of flows with adverse pressure gradient and pressure-induced boundary layer separation [9]. In the studies of the applications of DES based on the Spalart-Allmaras model, test cases on the flows around a rounded-corner square, circular cylinder and landing gear have been done and good agreement has been observed compared to experimental data [10]. It is obvious that the SST-DES offers great potential in the applications of simulations for a large class of flows. This refers to flows at high Re with chaotic and highly three-dimensional nature in wake regions. LES predicted vortices and the three-dimensional nature of the flow in the wake region similar to experimental data. Comparing to pure LES, DES is more practical considering the available computer resources at this time.

Bloor [1] investigated the flow around a circular cylinder between Re of 200 to 400 when turbulent motion starts to develop in the wake region of the flow. He observed that the transition of flow in the wake region is triggered by large-scale three-dimensional structures. Tritton [11] suggested that three-dimensional effects started to kick in at Re of 150 in the flow around a circular cylinder. Bearman [12] observed oblique shedding and vortex dislocation in the wake region of the flow around a circular cylinder. Similar experiment conducted by Williamson [13] documented the development of three-dimensional flow structures.

Philips [9] conducted experiments and concluded that between Re of 40 to 80, the laminar vortex shedding is truly two-dimensional. Beyond this Re, the existence of three-dimensionality of flow in the wake region has been reported. Gerrard [14] pointed out that end effects have a significant influence on the two and three-dimensional nature of the flow.

At Reynolds number of 120, Honji and Ishii [10] observed the existence of spanwise structures in the wake region of the flow. This is further confirmed by Bloor [1] on the observation of three-dimensional flow related to random low frequency irregularities detected on a hot wire experiment between Re of 200 to 400.

Kalro and Tezduyar [15] employed a three-dimensional finite element formulation on the flow around a circular cylinder at low Re when vortex shedding occurs. He emphasized the influence of the three- dimensional effect on the calculation of drag coefficient and Strouhal number, which in his computation agreed very well with experimental result compared to the two- dimensional analysis.

Hence, main objectives have been set in this study to investigate the flow around a circular cylinder, which is sensitive to changes of Re. Various Re (250-10,000) have been tested using steady and unsteady turbulence models. The study includes the simulation of vortex shedding phenomenon, force coefficient and pressure distribution of the flow. For the validation of the advanced modelling approaches (LES and DES) employed in the simulation, comparison with experimental data has been done at Re of 3,900. Advantages and disadvantages of each turbulence model are identified based on comparative studies with experimental results. Flow around a circular cylinder is very sensitive to the changes of Re. This paper aims to validate and identify suitable turbulence models in the application of the flow around a circular cylinder. Flow around a circular cylinder has been chosen as pilot study for the investigation on the effect of vortex shedding on such structures. The first stage of the validation process involves the simulation of the flow around a static cylinder using various turbulence models at low Re (250<Re < 10,000) to simulate the basic flow parameters and to capture the vortex shedding phenomenon in the wake region of the flow.

2. MATERIALS AND METHODS

Initially, modelling of the flow around a circular cylinder within Re 250-10,000 is done by using the unsteady and more advanced LES model on the flow around a circular cylinder to study the vortex shedding phenomenon in the wake region of the flow. This acts as a first step towards the investigation of the effect of vortices on bluff body flow. Following this, the validation of the turbulence models (LES and DES) employed in the simulation are performed at Re of 3,900 through comparative study with experimental results [16, 17].

Assuming that the flow is incompressible, the following equations can be used to describe the fluid flow,

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial u_i}{\partial x_j}\right) \qquad (1)$$

$$\frac{\partial u_i}{\partial x_j} = 0 \qquad (2)$$

where u is the velocity in the stream wise direction, p is the pressure, ρ is the fluid density and ν is the kinematic viscosity of the flow.

In LES, the flow velocity U is separated into a filtered, resolved part $\overline{U}$ and a sub-filter, unresolved part, u',

$$U = \overline{U} + u' \qquad (3)$$

The filter discretizes the flow spatially.
Applying the filter function to Eq. 3, we have

$$\bar{U}(x) = \int G(x,x')\, U(x')dx' \quad (4)$$

The filter function dictates the large and small eddies in the flow, done by the localized function G (x,x'), which determines the size of the small scales [8]. The top hat filter is common in LES.

$$G = \begin{cases} 1/\Delta, & if|x - x'| \leq \Delta/2 \\ 0, & otherwise \end{cases} \quad (5)$$

Solution is sought to achieve both computational efficiency and the capability of predicting the chaotic nature of flow such as vortex shedding. DES employs the RANS models near to the wall and LES in the wake region of a flow where unsteady and chaotic motion of flow is usually found.

DES uses a turbulent length scale, L_t which is replaced by a local grid spacing, Δ. If L_t greater than Δ, LES is activated in the DES formulation. The activation of LES or the switching to SST model in DES is controlled by a blending factor, which takes the form

$$F = \frac{L_t}{C_{DES}\Delta} \quad (6)$$

where C_{DES} is a constant.
Reynolds number of the flow around a circular cylinder,

$$Re = \frac{uD}{\nu} \quad (7)$$

where D is the diameter of the cylinder, u is the inlet velocity of the flow, and ν is the kinematic viscosity of the flow.

Then detailed comparative study of the flow around a circular cylinder with experimental results at Re of 3,900 is conducted. Strouhal number,

$$St = \frac{f_s D}{u} \quad (8)$$

where D is the diameter of the cylinder, f_s is the shedding frequency of vortices equal to 1/T and u is the incident velocity. Drag coefficient,

$$C_d = \frac{F}{1/_2\, \rho u^2 A} \quad (9)$$

where A is the projected area in the flow direction and F is the sum of the pressure force and the viscous force components on the cylinder surface acting in the along-wind direction. Lift coefficient is calculated similarly but vertical force is considered rather than along-wind force.

Non-dimensional form of the computational time is written as,

$$\Delta t^* = \frac{\Delta t U}{D} \quad (10)$$

where Δt is the timestep, U is the streamwise velocity and D is the diameter of the cylinder.

The domain and the boundary conditions for the simulation of the flow at Reynolds number are depicted in Figure 1. The cylinder is simulated with a diameter (D) of 0.1m and a depth of 2D to incorporate the spanwise effects, this has been chosen based on a similar case study for initial comparison. Distances of 10D and 30D to the side wall and to the downstream boundaries are allowed respectively to prevent blockage ratio and end effects on the flow.

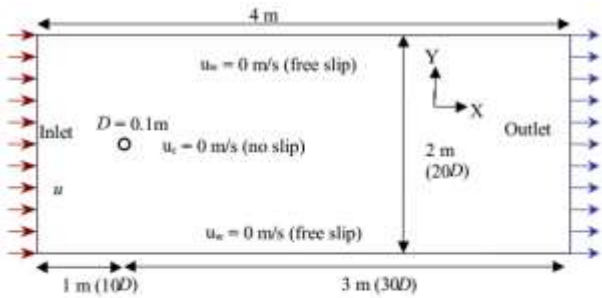

Figure 1. Computational geometry and boundary conditions (not to scale).

Longitudinal uniform velocities of 0.135m/s, 0.338m/s and 1.35m/s are introduced at the inlet correspond to the Reynolds number of 1,000, 2,500 and 10,000 respectively. The outlet boundary is defined with an average static reference pressure of 0Pa. The rest of the boundaries (side and bottom wall) are free slip walls, in which velocity near the wall is not retarded by frictional effects. The cylinder wall has a normal no slip boundary condition where velocity increases from zero at the wall surface to the free stream velocity away from the surface.

An unstructured tetrahedral mesh is employed in this simulation (Figure 2). Near to the cylinder wall, a very fine mesh is set to resolve the flow parameters in the boundary layer and set increasingly coarse in the radial direction to maintain computational efficiency.

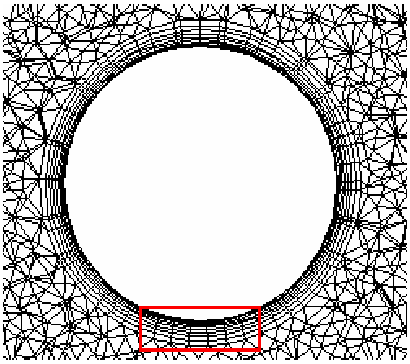

Figure 2. An unstructured tetrahedral mesh around a circular cylinder.

Flow around a circular cylinder at Re of 250 and 3,900 have been chosen for the LES. Laminar vortex shedding or von Karman vortex street (Figure 3) has been observed in the wake region of the flow around a circular cylinder at low Re between 40 to 250. Similar observations of von Karman vortex street at low Re has already been reported [18]. Within these regions of flow, the Strouhal number of the flow has a value of around 0.21. For Re that is greater than 250, the laminar periodic wake becomes unstable and the eddies start to become turbulent. Further increase of Re turns the wake region into turbulent flow.

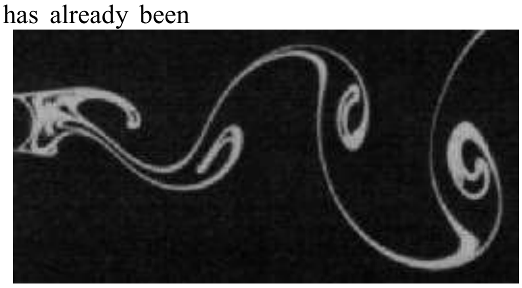

Figure 3. Von Karman vortex street in the wake of a circular cylinder.

In the LES model, simulation is done with unsteady inlet velocity of 0.034m/s by specifying the turbulence intensity at the inlet of 3%. The y+ value around the surface of the cylinder ranges between 1 and 3. The Navier–Stokes equations discretizes using a second order central difference scheme. The time discretization in the current LES is carried out by using a second order backward Euler scheme (space filtering model). The residual of the simulation has been set to the fourth order of the actual values for convergence consideration.

In the DES case, though both DES I and DES II mesh types are different simulations, they have almost similar number of elements. Meshing in the DES focuses more on local refinement of mesh rather than global refinement as in the LES. An ideal LES means the turbulence resolution length scale used needs to be fine enough to resolve all the motion in the flow [19], which suggests that a very refined mesh is required. On the other hand, the local refinement controls the contribution of the RANS model and the LES in the flow domain to get an optimal DES (SST-DES model).

3. RESULTS AND DISCUSSION

3.1. LES at Re 250

The Strouhal number computed is 0.20, agreeing well with the experimental value of 0.21 [18]. For the drag coefficient, experimental work gave a value of 1.20 [18] at this Re and LES predicted a drag coefficient of 1.24 from the time average computation. The current study demonstrates the ability of the LES model to predict the unsteady motion in the wake region of the flow. LES gives very close pressure distribution prediction with experimental data, showing the capability of LES in capturing the complex vortical type of flow, which is crucial in the simulation of flow around bluff bodies. LES shows good agreement in region where vortices are shed. This suggests that a hybrid RANS/LES model such as the DES model could be beneficial in the applications of the flow around bluff bodies.

3.2. LES at Re 3900

This flow is characterized by laminar separation region in which transition to turbulence happens in the shear layer, producing large-scale vortices and complex flow in the near wake region.

Counter rotating stream wise vortices which are highly three-dimensional have been observed in the wake region of flow at this Re [20].

Lourenco and Shih [21] used the particle image velocimetry technique to measure the velocity profiles within three diameters downstream of the cylinder while Ong and Wallace [16] documented the distribution further downstream. Breuer [22] investigated the influence of numerical aspects on the LES of flow around a circular cylinder at Re of 3900. He concluded that LES with a second order or higher central difference advection scheme yields the best solution and upwind discretization scheme is not recommended in LES, as it introduces extra dissipation. Hansen and Long [23] used LES with the finite volume method on an unstructured mesh at the same Re but has over- predicted the drag coefficient and the base pressure. Both works [22, 23] reported shorter recirculation length of flow compared to experimental result.

When simulated the flow past a circular cylinder (Re 3,900) using a high order upwind-biased finite difference method, the dynamic SGS model give better prediction of the Reynolds stresses in the vortex formation region [20] than a fixed coefficient Smagorinsky SGS model. Tremblay [24] concluded otherwise. He investigated the influences of the SGS model (fixed coefficient Smagorinky and dynamic Germano [25]) and grid resolution on the flow around a circular cylinder at Re of 3,900 and concluded that the grid resolution has greater effects on the solution of the flow than the SGS model. This could be due to the reason that very different mesh resolution has been used in the simulations.

Three-dimensional simulation is crucial in the flow at Re of 3,900 past a circular cylinder. Two-dimensional simulation overestimated the drag coefficient by 5-10% [26] due to the simulation of perfect spanwise correlation of the flow. Two-dimensional calculation also omitted the effects of spanwise wake turbulence. This agreed with the finding of Kalro and Tezduyar [15], emphasizing the importance of three dimensional calculation in sub-critical region of flow to accurately resolve the flow parameters such as the drag coefficient and the Strouhal number. Also, the base pressure is more negative in two-dimensional simulation and this affects the wake region prediction of the flow.

In unsteady flow simulation such as LES, certain numbers of vortex shedding cycles need to be simulated in order to accurately predict the near wake structure of the mean flow velocities and Reynolds stress profiles. Franke and Frank [27] pointed out that simulation of the LES on the flow around a circular cylinder at Re of 3,900 that is less than 42 averaging cycles might not yield a statistically converged solution of the flow. Comparisons show that time average symmetry flow is achieved for simulations with more than 42 averaging cycles.

Apart from the LES, DNS of the flow past a circular cylinder has been carried out by Tremblay [4] at a Re of 3900. The DNS result has been included for comparison with the LES prediction in the current work, which proves very useful in clarifying the discrepancies between experimental data and results from the LES.

Simulation of the LES on the flow around a circular cylinder at Re of 3900 in the current work employs an inlet velocity of 0.6m/s. A structured hexahedral mesh is used in the current simulation rather than the unstructured tetrahedral mesh as discussed before. Biswas and Strawn [28] documented the applications of tetrahedral mesh and hexahedral mesh on CFD problems, and concluded that a tetrahedral mesh is more suitable on flow over complex bodies due to its flexibility. Insufficient representation of separated shear layer and wake region from a tetrahedral mesh due to distortion of the mesh but a hexahedral mesh preserved the mesh quality better at Re 3900 performing LES [29]. Current LES employs a block structured hexahedral mesh, which provides more flexibility in simulating flow around bluff bodies.

3.3. DES at Re 3900

LES gives details on the unsteady motion of flow compared to RANS models. But the price is the increase in computational requirements. DES employs the RANS model at the boundary layer of the flow to save computational power and switches to the LES model in the detached flow region to capture the unsteady scales of the separated shear layer. Various works [30, 10] have been carried out using the DES at high Re (800,000), with satisfactory result obtained.

Travin et al. [31] simulated the flow around a circular cylinder at a Re of 50,000 using a Spalart-Allmaras DES model. Current work on the flow around a circular cylinder employs a SST-DES model, which is similar to the SST-DES model of Strelets but with some modification to reduce the grid induced separation problems [32]. Also, a blending of second order upwind scheme and a second order central difference scheme is used rather than the fifth order upwind as in the SST-DES model used by Strelets.

3.4. Comparison of LES and DES at Re 3900

Simulation of the flow around a circular cylinder at Re of 3,900 is done on a parallel machine consisting of 24 Pentium IV CPU each with 2.5GHz processor and 1GB RAM. This is necessary for the capturing of the unsteady flow details in the wake region of the flow and to observe the time history of the flow parameters such as the drag and the lift coefficients - simulation on a single Personal Computer is too impractical and time consuming.

A mesh sensitivity analysis has been conducted for the LES of flow around a circular cylinder at Re of 3,900. Two mesh types, LES I with 0.24 million elements and LES II with 0.55 million elements have been compared. Unlike the RANS cases, both LES has similar amount of nodes and elements in the same simulation. Similar case applied to DES. To resolve the flow profiles near the cylinder wall surface, average y+ values for LES and DES of 3 and 4 have been used respectively. Figure 4 defines the centerline and three vertical profile locations in the wake region where flow is analyzed for comparison with the experimental data.

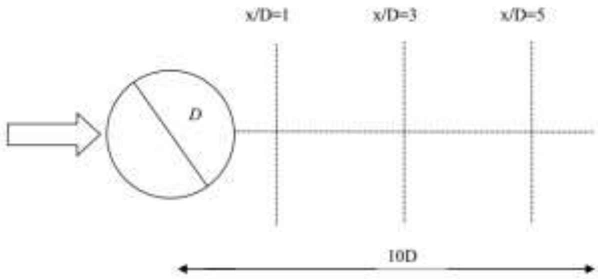

Figure 4. Centerline and vertical profiles of flow in the wake region of circular cylinder.

3.5. Comparison of simulation results with the experimental data

The average drag coefficient from the LES is 0.99 and experiment data gives a value of 0.98±0.05 [31]. The predicted flow parameters are summarized in Table 1 together with the results from the DES. Lr in the table represents the recirculation length of the flow in the wake region of the cylinder. The angle of separation of the flow (θ_s) from LES is 88.2°, close to the experimental value of 85±2° [33].

Table 1. Simulated flow parameters (LES and DES) Vs experimental results.

Simulation/ Experiment	Strouhal No.	Cd	Lr/D	θ_s
Exp. [44]	0.215±0.005	0.98±0.005	1.33±0.2	-
Exp. [45]	-	-	-	85±2
LES	0.210	0.99	1.14	88
DES	0.250	0.88	1.04	91

As mentioned, DES employs the SST model in the vicinity of the cylinder. The SST model in the current DES employs a second order upwind advection scheme for space discretization. Strelets [34] used a fifth order upwind scheme SST (DES) model on the flow around a circular cylinder at higher Re and still a lower drag coefficient has been observed. When he switched to a hybrid upwind/central advection scheme for the same simulation, great improvement is shown on the prediction of the drag coefficient. Another factor that influences the prediction of the drag coefficient in SST is the refinement of the mesh near the wall of the cylinder. Very refined mesh is needed to accurately capture the viscous drag since the flow is resolved directly without any wall function method employed. Consequently, the under-prediction of the drag coefficient resulted in a higher Strouhal number from the DES compared to the experimental value (Table 1). This agreed with the experimental finding of Roshko [35].

In the recirculation region, LES displays close agreement with experimental result and DNS. In the recovery region (x/D=2 to x/D=4), a similar trend of flow has been predicted by LES compared to DNS but not the experimental data. However, the experimental result is questionable at x/D=3 where a sudden change of the curve is observed. DES computed the shortest recirculation length among the results, leading to the discrepancy in the recovery region. Consequently, the centerline velocity does not recover to the value of the experimental and DNS result further down the wake region.

In the recirculation region, both LES and DES predict a narrower wake compared to the measured data and DNS, with the narrowest from DES. This is a consequence of a later separation angle (91°) (Table 1) of DES prediction. For x/D>1, less agreement between velocity components of DES have been observed and fluctuation of the result is obvious, indicating that longer simulation time is necessary. However, the distribution of the profiles from experiment shows a rather strange behaviour, where the spanwise velocity component at the centerline is not zero.

Further down the flow at x/D=3 and x/D=5, the spanwise velocity components exhibit fluctuations due to the coarsening of the mesh, with the DES computation having a coarser mesh and thus greater discrepancy at x/D=5. The spanwise mesh resolution also contributes to the asymmetry of the curves. Fluctuations are observed in the experimental results to some extent as well.

All the Reynolds stress components are normalized by the square of the inlet velocity, U_{in}^2. For the streamwise Reynolds stress component, prediction from both the LES and DES show lower peaks compared to both the experimental result and the DNS. The work [20] on the LES of flow around a circular cylinder reported a similar situation from a fixed Smagorinsky SGS model compared to the dynamic SGS model. In the Smagorinsky SGS model, filtering introduces a wave number cut-off where small eddies are modelled using an eddy viscosity model. The eddy viscosity model predicted different viscosity intensity compared to the dynamic model, causing the discrepancy observed in the Reynolds stress components. Hansen and Long [23] observed similar low streamwise Reynolds stress components when applying LES with the fixed Smagorinky SGS model on the flow around a circular cylinder at the same Re. The vortical structures can be visualized at the near wake with spanwise changes but the effect reduces further downstream. Better agreement of the spanwise Reynolds stress with experimental results is observed from the LES and DES rather than DNS beyond the recirculation region. Notice the discrepancy of the DNS result [18] with the measured experiment and simulated data. Similar situation has been observed in the DNS of flow around a circular cylinder conducted by Ma et al. [36].

Other researchers [22, 37] reported that a dynamic subgrid scale model shows better prediction of the stress components compared to the fixed Smagorinsky model employed in the current simulation. Despite this discrepancy, the peak locations and the size of the wake were well predicted from both of the simulation except for the case of DES at x/D=5, which shows almost zero value of vertical shear stress profiles.

3.6. Computational time

It is claimed that the advantage of the DES over LES is the shorter computational time required for simulation. Meshing is different in LES and DES. Refinement of mesh in LES needs to be done evenly throughout the computational domain while DES requires a more local mesh refinement. In other words, meshing in DES is more flexible and users have more control on the number of elements used in the simulation to be computationally efficient. Usually, regions where RANS model is active can be simulated with a much coarser mesh.

Figure 5 shows the CPU time (in minutes) for the LES and DES cases with different mesh size, corresponding to the meshes used in the mesh independence test at the beginning of this chapter. The CPU time here refers to the time in minutes required for the computer to calculate one time step on a single CPU. To provide an idea of the CPU time compared to normal wall clock time, the 5.8 CPU time of LES took 4 days of computational time running on 12 CPU with a total time step of 12000 (in order to get sufficient vortex shedding cycles).

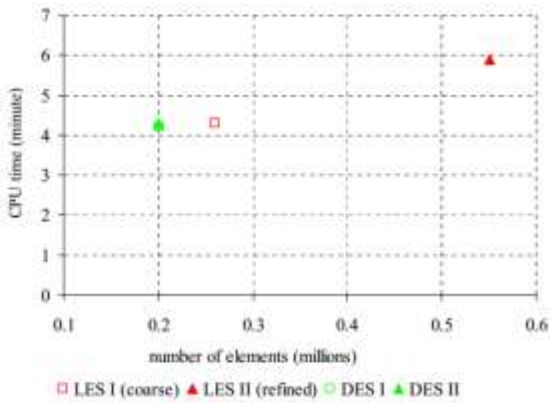

Figure 5. CPU time for LES and DES for flow around circular cylinder (Re 3900).

The non-dimensional time for the current simulation has a value of 0.04 based on Eq. 10. From Figure 5, it is noticed that similar amount of time is required for the LES and DES for a 0.2 million mesh. This mesh is considered a coarse mesh in LES but a medium or fine mesh in the DES. For a satisfactory mesh in the LES (0.55 million), simulation time is about 1.4 times longer than the DES case (Figure 5), which only takes about 2.9 days.

Although it is not at order of magnitude faster in this case (Re 3,900), the situation is expected to be different at higher Re. It is reported that the number of elements in a mesh LES required near the viscous sub layer is proportional to Re 1.8 [17], but not in the case with DES where a RANS model helps to save the computational power. So a much longer computational time is expected for LES at higher Re and the application of DES will be an advantage.

4. CONCLUSIONS

In the LES and DES computations, the unsteady scales of the flow and the vortex structure have been well predicted. LES correctly computed the recirculation length, separation point, drag coefficient and the Strouhal number of the flow while DES gave slightly different values from the measured experimental data. For the analysis of the velocity profiles distribution in the wake region of the flow, both LES and DES captured good streamwise and spanwise velocity profiles. The Reynolds stresses components were generally under predicted from both cases due to the deficiency of the fixed Smagorinsky SGS model on the assumption of the eddy viscosity distribution. DES proves to be computationally more efficient than LES and requires fewer elements for simulations at similar Re.

Vortex shedding phenomenon has been successfully captured by LES and DES models for the flow around a circular cylinder. From the comparative study with experimental work, LES and DES have been successfully validated at Reynolds number of 3,900.

The profiles of the streamwise and spanwise velocity components in the wake regions of the flow have been well predicted from both LES and DES for the flow around a circular cylinder. This resulted in good agreement of the recirculation bubbles and the size of the wake compared to experimental results, with LES predicting closer agreement than DES.

LES predicted a more complex flow and vortex structure in the wake region of the flow compared to DES. This is attributed to the fact that the SST model in DES averages out part of the unsteadiness of the flow, with LES representing a more realistic vortex shedding in complex flow situation. Generally, DES saved about 1.5 to 2.5 times of computational time compared to LES in the simulations of the flow around bluff bodies within lower Re<22,000.

Abbreviations Used

CFD	Computational Fluid Dynamics
CPU	Central Processing Unit
DES	Detached Eddy Simulation
DNS	Direct Numerical Simulation
LES	Large Eddy Simulation
RANS	Reynolds Averaged Navier-Stokes
Re	Reynolds number
SGS	Sub Grid Scale
SST	Shear Stress Transport

REFERENCES

1. *Bloor, M.S. The transition to turbulence in the wake of a circular cylinder. Journal of Fluid Mechanics* ***1964****, 19, 290-304.*
2. *Nishioka, M.; Hiroshi, S. Measurement of velocity distributions in the wake of a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics* ***1974****, 65, 91-112.*
3. *Braza, M.; Chassaing, P.; Minh, H.H. Prediction of large scales transition features in the wake of a circular cylinder. Phys. Fluids* ***1990****, A2, 1461-1471.*
4. *Franke, R.; Rodi, W.; Schonung, B. Numerical calculation of laminar vortex-shedding flow past cylinders. Journal of Wind Engineering and Industrial Aerodynamics* ***1990****, 35, 237-257.*
5. *Spalart, P.R. Strategies for turbulence modelling and simulations. International Journal of Heat and Fluid Flow* ***2000****, 21, 252-263.*
6. *Spalart, P.R.; Allmaras, S.R. A one-equation turbulence model for aerodynamic flows. La Recherche Aerospatiale* ***1994****, 1, 5-21.*
7. *Menter, F.R.; Grotjans, H. Application of Advanced Turbulence Models to Complex Industrial Flows. AEA Technology GmbH, Germany, 1999.*
8. *Menter, F.R. Two-Equation Eddy-viscosity Turbulence Models for Engineering Applications. AIAA Journal* ***1994****, 32, 1598-1605.*
9. *Philips, A. The intensity of Aeolian tones. Journal of Fluid Mechanics* ***1956****, 41, 137-145.*
10. *Honji, H.; Ishii, K. Wake shedding from a distorted cylinder. Journal of Physical Society Japan* ***1976****, 41, 1089-1090.*
11. *Tritton, D. Experiments on the flow past a circular cylinder at low Reynolds numbers. Journal of Fluid Mechanics* ***1959****, 6, 547-567.*
12. *Bearman, P. Near wake flows behind two and three-dimensional bluff bodies. Journal of Wind Engineering and Industrial Aerodynamics* ***1997****, 69-71, 33-54.*
13. *Williamson, C. Advances in our understanding of vortex dynamics in bluff body wakes. Journal of Wind Engineering and Industrial Aerodynamics* ***1997****, 69-71, 1-32.*
14. *Gerrard, J. The wakes of cylindrical bluff bodies at low Reynolds number. Philosophical Transactions Royal Society* ***1978****, 288, 351-382.*
15. *Kalro, V.; Tezduyar, T. Parallel 3D computation of unsteady flows around circular cylinders. Parallel Computing* ***1997****, 23, 1235-1248.*
16. *Ong, L.; Wallace, J. The velocity field of the turbulent very near wake of a circular cylinder. Experiments in Fluids* ***1996****, 20, 441-453.*
17. *Blazek, J. Computational Fluid Dynamics: Principles and Applications. Elsevier Science Ltd, Oxford, England, 2001.*
18. *Tremblay, F. Direct and large-eddy simulation of flow around a circular cylinder at subcritical Reynolds numbers. Ph.D. Dissertation, Technical university of Munich, 2001.*
19. *Singh, S.P.; Mittal, S. Flow past a cylinder: shear layer instability and drag crisis. Int. J. Numer. Meth. Fluids* ***2005****, 47, 75-98.*
20. *Menter, F.R.; Kuntz, M. Development and application of a zonal DES turbulence model for CFX-5. ANSYS CFX Validation Report, CFX-VAL17/0703, 2001, 1-34.*

21. *Lourenco, L.; Shih, C. Characteristics of the plane turbulent near wake of a circular cylinder, a particle image velocimetry study. In: Beaudan, P.; Moin, P. Numerical experiments on the flow past a circular cylinder at a subcritical Reynolds number. Technical report TF-62, Thermosciences Division, Department of Mechanical Engineering, Stanford University, California, USA, 1994.*
22. *Breuer, M. Large eddy simulation of the subcritical flow past a circular cylinder: Numerical and modelling aspects. International Journal for Numerical Methods in Fluids **1998**, 28, 1281-1302.*
23. *Hansen, R.; Long, L. (2002). Large eddy simulation of a circular cylinder on unstructured grids. AIAA Aerospace Science Meeting and Exhibit, Reno, Nevada, USA, 2002.*
24. *Tremblay, F.; Manhart, M.; Friedrich, R. (2002). LES of flow around a circular cylinder at a subcritical Reynolds number with cartesian grids. Proceedings of the EUROMECH Colloquim 412 on LES of complex transitional and turbulent flows, NTNU, Trondheim, Norway, 2000.*
25. *Germano, M.; Piomelli, U.; Moin, P.; Cabot, W.H. A dynamic subgrid scale eddy viscosity model. Phys. Fluids **1991**, A3, 1760-1765.*
26. *Al-Jamal, H.; Dalton, C. Vortex induced vibrations using Large Eddy Simulation at a moderate Reynolds number. Journal of Fluids and Structures **2004**, 19, 73-92.*
27. *Franke, J.; Frank, W. Large eddy simulation of the flow past a circular cylinder at Re=3900. Journal of Wind Engineering and Industrial Aerodynamics **2002**, 90, 1191-1206.*
28. *Biswas, R.; Strawn, R. Tetrahedral and hexahedral mesh adaption for CFD problems. Applied Numerical Mathematical Journal **1998**, 26, 135-151.*
29. *Frohlich, J.; Rodi, W.; Bertoglio, J.P.; Bieder, U.; Touil, H. Large Eddy Simulation of flow around circular cylinders on structured and unstructured grids. Notes on Numerical Fluid Mechanics **1998**, 66, 319-338.*
30. *Hedges, L.; Travin, A.K.; Spalart, P.R. Detached eddy simulations over a simplified landing gear. Journal of Fluid Engineering **2002**, 124, 413-423.*
31. *Travin, A.; Shur, M.; Strelets, M.; Spalart, P.R. Detached-eddy simulations past a circular cylinder Flow. Turbulence and Combustion **2000**, 63, 293-313.*
32. *Strelets, M. Detached eddy simulation of massively separated flow. Paper 2001-0879, Am. Inst. Aeronaut. Astronaut., Washington, USA, 2001.*
33. *Pope, S. Ten questions concerning the large-eddy simulation of turbulent flows. New Journal of Physics **2004**, 6, 1-24.*
34. *Norberg, C. Effects of Reynolds number and a low-intensity free-stream turbulence on the flow around a circualt cylinder. Publication No. 87/2, Department of Applied Thermodynamics and Fluid Mechanics, Chalmer University of Technology, Goteborg, Sweden, 1987.*
35. *Roshko, A. On the development of turbulent wakes from vortex streets. NACA Report 1191, Washington, USA, 1954.*
36. *Ma, X.; Karamanos, G.S.; Karniadakis, G.E. Dynamics and low-dimensionality of the turbulent near-wake. Journal of Fluid Mechanics **1999**, 410, 29-65.*
37. *Beaudan, P. Numerical experiments on the flow past a circular cylinder at subcritical Reynolds number. Technical Report TF-62, Dept. of Mechanical Engineering, Stanford University, California, USA, 1994.*