AFRREV STECH
An International Journal of Science and Technology
Bahir Dar, Ethiopia

Vol.1 (2) April-July, 2012:128-141

ISSN 2225-8612 (Print) ISSN 2227-5444 (Online)

# 3D Flow around a Rectangular Cylinder: a review

**Odesola, Isaac F.**
Department of Mechanical Engineering
Faculty of Technology, University of Ibadan, Ibadan, Nigeria
E-mail: ifodesola@mail.ui.edu.ng or ifodesola@yahoo.com

**Olawore, Ayodeji**
Department of Mechanical Engineering
Faculty of Technology, University of Ibadan, Ibadan, Nigeria

**Abstract**

*Turbulent flows around three-dimensional obstacles are common in nature and occur in many applications including flow around tall buildings, vehicles and computer chips. Understanding and predicting the properties of these flows are necessary for safe, effective and economical engineering designs. This paper presents the review of 3D flow around a rectangular cylinder using large eddy simulation as the turbulence model and the computational study is developed in the frame of the Benchmark on the Aerodynamics of a*

 

*Rectangular Cylinder (BARC). Different simulations around bluff bodies were reviewed and the results obtained through different methodologies are presented. The effect of change by vortex shedding on the magnitude of fluid forces of rectangular cylinders are examined and reported. The aerodynamic integral parameters obtained from different papers are compared.*

**Keywords:** Rectangular cylinder, large eddy simulation, BARC.

## Introduction

The study of flow around bluff bodies of rectangular shape has a deep engineering interest because many civil but also industrial structure can be assimilate to this shape. The fluid-dynamic forces acting on a rectangular cylinder have been mainly investigated for unbounded flow conditions. Even the bounding effect of a fixed wall occurs in many application of civil engineering, for example in the case of bridges and/or buildings placed in the proximity of other structures or particular land morphology, both in water or in air-flow [16]. Similarly, piers, bridge pillars, and legs of offshore platforms are continuously submitted to the load produced by maritime or fluvial streams. This kind of flow is also found in many other technical applications, especially those concerning with thermal and hydraulic devices [11]. The geometry of bluff bodies plays a pivotal role in the flow-structure interactions and the resulting flow and pressure fields around them [18].

The understanding of turbulent flow around a bluff body is far from mature, even if the flow is around simple geometry. Since the Direct Numerical Simulation (DNS) of such flow is still not feasible for a quite long foreseeable time, using wall resolved Large Eddy Simulation (LES) technique to investigate this high Reynolds number flow would be both valuable and challenging [19]. The benchmark is oriented to the flow around bluff-bodies, and they are suitable to explore the feasibility of the LES to relate engineering problems because they show complicated flow events such as massive separations, impinging, transition to turbulence and formation of vortex streets. These flow patterns are commonly found in the aerodynamic analysis of ground vehicles [10]. LES is particularly attractive for the simulation of bluff body flows, which are characterized by a complex three-dimensional and intrinsically unsteady dynamics that is difficult to be accurately simulated by the RANS approach [1]. The flow around a circular or rectangular cylinder in the uniform flow is the most basic fluid dynamic phenomenon. It is known that the flow around a rectangular cylinder exhibits an unsteady behavior such as full-separation flow, alternately reattachment flow and full-reattachment flow, accompanied by a change in fluid dynamic force according to changes in its side ratio [15].

Nowadays, the flow around bluff bodies has been widely investigated; the greater numbers of studies concern the flow past circular cylinders. Less attention has been dedicated to the flow around rectangular cylinders, although this phenomenon is of great interest for engineering and aerodynamic applications, especially for the fluid–structure interactions via experimentation and numerical approach. After the first experimental works of Okajima [12] that investigated the Strouhal number varying the cylinder width-to-height ratio (B/D) from 1 to 4 in a wide range of Reynolds numbers, some studies regarding both the laminar and turbulent flows have been performed through numerical and experimental techniques. Okajima [13] investigated the numerical simulation of flow around a rectangular cylinder at a width-to-height ratio (B/D = 1.5 to 4) on the unsteady 2D flow pattern, at zero angle of incidence at a critical range Reynolds number of flows, whereas Sarioglu et al [8] carried out experimental and numerical analysis of this phenomenon at moderate Reynolds numbers. Bruno et al. [2] studied the flow around a B/D = 5 cylinder at Re = 40,000 by means of a finite volume discretization, whereas Nigro et al. [10] applied a finite element method to the large eddy simulations of the flow past a sharp-edged surface mounted cube at Re = 40,000.

Yu and kareem [18] numerically conducted parametric study of flow around rectangular prisms of different aspect ratios at a Reynolds number of $10^5$ and the Navier Stokes equations in the large eddy simulation (LES) framework are solved using a finite volume method. Rokugou et al. [14] carried out three-dimensional numerical analysis of the flow around rectangular cylinders with various side ratios, D/H, from 0.2 to 2.0, for Reynolds number of 1000 by using a multi directional finite difference method on a regular-arranged multi grid. Nakaguchi et al [9] pointed out that pressure distribution around the cylinder surface provides basic knowledge of the force exerted by the fluid on a body, the pressure distribution varying according to the B/H ratios of the section.

Turbulence models are equations that account for turbulence of flow based on some assumptions, since it is computationally not practical to thoroughly represent all the physical characteristics of a flow using the current available computer power. The simulations start with the application of LES to flow around a rectangular cylindrical body with aspect ratio 5:1 [5]. The LES approach has emerged as a more attractive scheme which has the promise of providing improved results with reasonable computational effort [4]. Large eddy simulation (or LES) tries to simulate the largest scales of motion while treating the small scales by a model [6].

**Nomenclature**

| | |
|---|---|
| Re | Reynolds number |
| $\Phi$ | chord ratio |
| f | vortex shedding frequency (Hz) |
| $U_\infty$ | mean velocity of the flow (m/s) |
| St | Strouhal number |
| D | body characteristic length (m) |
| U | velocity in the streamwise direction (m/s) |
| P | pressure (Pa) |
| $\rho$ | fluid density (m/s) |
| $\upsilon$ | kinematic viscosity of the flow ($m^2/s$) |
| $\Delta$ | characteristic spatial length of the filter |
| $\mu$ | dynamic viscosity(kg/ms) |
| $\tau_{ij}$ | subgrid scale (SGS) Reynolds Stress |
| $\upsilon_{SGS}$ | SGS eddy viscosity |
| $\overline{S}$ | strain rate tensor |
| l | characteristic length scale of the unresolved motion |
| x | space coordinate |
| t | time coordinate |
| $\overline{u}$ | filtered velocity |
| $\overline{p}$ | filtered pressure |
| $\delta$ | filter width |

| | |
|---|---|
| $C_s$ | Smagorinsky constant |
| y+ | dimensionless distance of grid points from the wall |

**Flow around Rectangular Cylinders**

From the study of the flow around rectangular cylinders, one learns about the flow characteristics around similar shape bodies such as the deck section of long span bridges. Majority of the bridge deck sections resemble more complex rectangular shape. Flow separates from the leading edge corner and forms vortices that travel along the surface of the deck, which are eventually shed from the trailing edge resulting in the vortex shedding phenomenon. A vortex core tends to have a local minimum pressure, so the formation and progression of vortices introduce forces on the deck surface. The same phenomenon is observed on the flow around rectangular section with similar ratio. Also, the Strouhal number and drag coefficient of the flow change accordingly.

Flow can be laminar or turbulent depending on the Reynolds number. For engineering applications, most flows are turbulent in nature. Turbulence is the chaotic nature of flow in motion showing random variation in space and time. Turbulent flow is characterized by its irregularity, three-dimensionality and dissipative nature. Turbulence contains eddies with different sizes which are always rotational in motion. Different scales of eddies are found in a flow. Large scale eddies are responsible for the carrying of energy and transfer of momentum in the flow. On the other hand, the smaller scale eddies, where dissipation of energy occurs are known as the Kolmogorov scale eddies. The large eddies extract energy from the mean flow and transfer it to the smallest eddies where energy is taken out of the flow through viscosity.

**Navier-Stokes equations**

Combining these fundamental principles, the physics of fluid flow is expressed in terms of a set of partial differential equations known as the Navier-Stokes equations. By solving these equations (continuity equation, momentum equation and the energy equation), the pressure and velocity of the fluid can be predicted throughout the flow. For more explanation on the derivation of Navier-Stokes equations, see Versteeg and Malalasekera

(1995). Assuming that the flow is incompressible, the following equations can be used to describe the fluid flow,

- Navier-Stokes Equations: (conservation of momentum),

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial p}{\partial x_j}\left(\upsilon\frac{\partial u_i}{\partial x_j}\right) \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 1$$

- The continuity equation: (conservation of mass),

$$\frac{\partial u_i}{\partial x_j}=0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 2$$

**Large Eddy Simulation (LES)**

Large eddy simulation (LES) is classified as a space filtering method in CFD. LES directly computes the large-scale turbulent structures which are responsible for the transfer of energy and momentum in a flow while modelling the smaller scale of dissipative and more isotropic structures. In order to distinguish between the large scales and small scales, a filter function is used in LES. A filter function dictates which eddies are large by introducing a length scale, usually denoted as $\Delta$ in LES, the characteristic filter width of the simulation. All eddies larger than $\Delta$ are resolved directly, while those smaller than $\Delta$ are approximated.

**Filtering of Navier-Stokes equations**

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

$$U=\overline{U}+u' \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad 3$$

The filter discretises the flow spatially. Applying the filter function to Eq. 2.2, we have,

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

 

As mentioned, the filter function dictates the large and small eddies in the flow. This is done by the localized function G (x, x'). This function determines the size of the small scales,

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

Various filtering methods exist; the top hat filter is common in LES. By imposing the filter function in the continuity and the Navier-Stokes equations, one obtains the filtered equations governing the fluid flow in LES,

$$\frac{\partial \overline{u_i}}{\partial x} = 0 \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \; 6$$

$$\frac{\partial \overline{u_i}}{\partial t} + \frac{\partial \overline{u_i u_j}}{\partial x_j} = -\frac{1}{\rho}\left(\frac{\partial \overline{p}}{\partial x_i}\right) + \frac{\mu}{\rho}\left(\frac{\partial^2 \overline{u_i}}{\partial x_i \partial_j}\right) \quad \ldots\ldots\ldots\ldots\; ..7$$

The over bar denotes the space filtered quantities. In fluid flow around an immersed object, shear stress occurs because not all the fluid exerts forces tangentially to the wall of the object. This results in the appearance of the stress terms in the equations governing fluid flow. After dividing the Navier-Stokes Equation into filtered and sub-filter components, the unknown stress term $\overline{u_i u_j}$ arises due to the nonlinearity of the equations and the shear stress of the flow.

This term needs to be approximated to solve the filtered Navier-Stokes Equations.

$$\overline{u_i u_j} = \overline{\left(\overline{u_j} + u_j'\right)\left(\overline{u_i} + u_i'\right)} = \overline{\overline{u_i}\,\overline{u_j}} + \overline{\overline{u_i} u_j^{i}} + \overline{u_i' \overline{u_j}} + \overline{u_i' u_j'} \quad \ldots\ldots\ldots\; 8$$

The unknown term $\overline{u_i u_j}$ comprises the resolvable scale component $\overline{u_i}$ and the small scale component u' of the flow. Thus, a relationship based on the interaction among components of various scales in the flow has been derived to estimate the unknown. This is written as,

$$\tau_{ij} = \left(\overline{\overline{u}_i \overline{u}_j} - \overline{u}_i \overline{u}_j\right) + \left(\overline{\overline{u}_i u_j^{'}} + \overline{u_i^{'} \overline{u}_j}\right) + \overline{u_i^{'} u_j^{'}} \ldots\ldots\ldots\ldots\ldots\ldots \quad 9$$

The term is known as the subgrid scale (SGS) Reynolds Stress. Physically, the right hand side of Eq.9 represents the large scale momentum flux due to turbulence motion.

**Smagorinsky model**

To approximate the SGS Reynolds stress $\tau_{ij}$, a SGS model can be employed. The most commonly used SGS models in LES is the Smagorinsky model. In a flow, it is the shear stress and the viscosity of the flow that cause the chaotic and random nature of the fluid motion. Thus, in the Smagorinsky model, the effects of turbulence are represented by the eddy viscosity based on the well-known Boussinesq hypothesis [17]. The Boussinesq hypothesis relates the Reynolds stress to the velocity gradients and the turbulent viscosity of the flow. It is therefore assumed that the SGS Reynolds stress $\tau_{ij}$ is proportional to the modulus of the strain rate tensor of the resolve eddies [7],

$$\tau_{ij} - \frac{1}{3}\tau_{kk} = -2v_{SGS}.\overline{S}_{ij} = v_{SGS}.\left|\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right| \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 10$$

Where $\overline{S}_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right)$ ........................................................ 11

where $v_{SGS}$ is the SGS eddy viscosity and $\overline{S}$ is the strain rate tensor,

$$\left|\overline{S}\right| = \left[2\overline{S}_{ij}\overline{S}_{ij}\right]^{\frac{1}{2}} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 12$$

 

The SGS eddy viscosity $v_{SGS}$ needs to be approximated in order to solve Eq. 2.9. Based on dimensional analysis, the following relationship has been obtained,

$$v_{SGS} \; \alpha \; lq_{SGS} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 13$$

where l is the characteristic length scale of the unresolved motion that usually takes the value of the filter width $\Delta = \left(\Delta_x \Delta_y \Delta_z\right)^{\frac{1}{3}}$. $\Delta_x$, $\Delta_y$ and $\Delta_z$ are the grid spacings in the x, y and z direction respectively. By relating the velocity scale of the unresolved motion $q_{SGS}$ to the gradients of the filtered velocity based on an analogy of the mixing length model, the SGS viscosity is written as [20],

$$v_{SGS} = \left(C_s \Delta\right)^2 \left|\overline{S}\right| \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots 14$$

where $C_s$ is the Smagorinsky constant that changes depending on the type of flow. For isotropic turbulent flow, the Cs value is usually around 0.18 to 0.20.

Basically, the Smagorinsky SGS model simulates the energy transfer between the large and the subgrid-scale eddies. Energy is transferred from the large to the small scales but backscatter (reverse of cascade process) sometimes occurs where flow becomes highly anisotropic, usually near to the wall. To account for backscattering, the length scale of the flow can be modified using Van Driest damping [7]

$$\left(C_s \Delta\right) = C_s \Delta \left(1 - e^{-y^+/25}\right) \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots 15$$

where y+ is the dimensionless distance of grid points from the wall. Van Driest damping accounts for the reduced growth of the small scales near the wall which gives a smaller value of SGS viscosity in order to represent the flow more accurately.

 

## Flow modeling and computational approach

The key features in the simulation of the flow around rectangular cylinders is the ability of the turbulence models to capture the changes of the flow characteristics with respect to the increasing B/H ratio and the prediction of the reattachment of flow along the side surface of the cylinders. The 3D, turbulent, unsteady flow around the cylinder is modeled in the frame of the large eddy simulation approach to turbulence using the classical time-dependent filtered Navier Stokes equations. Only flow at zero angle of incidence has been conducted in the current study. The author is aware of the significant effect of the non-zero incident flow. Positive incident flow generates negative cross wind force as a consequence of the pressure distribution around the cylinder, which results in aeroelastic instability if the enveloped structure is free to move. This is very important in the study of the flow around a bridge deck section.

Domain size:

- The computational domain and boundary conditions used in the simulation for the flow around the rectangular cylinder is depicted in figure1. The spanwise length of the computational domain is set equal to L/B = 1 on the basis of a short review of the state of art. The breadth (B) to depth (D) ratio is set equal to 5. Dirichlet conditions on the velocity field and on the sub-grid kinetic energy are imposed at the inlet boundaries. Neumann conditions on the normal component of the stress tensor T, as well as the same Dirichlet conditions on kt, are imposed at the outlet boundaries. Periodic conditions are imposed on both the side surfaces and on the upper-lower surfaces, as depicted in Fig. 1. No-slip conditions are imposed at the section surface.

 

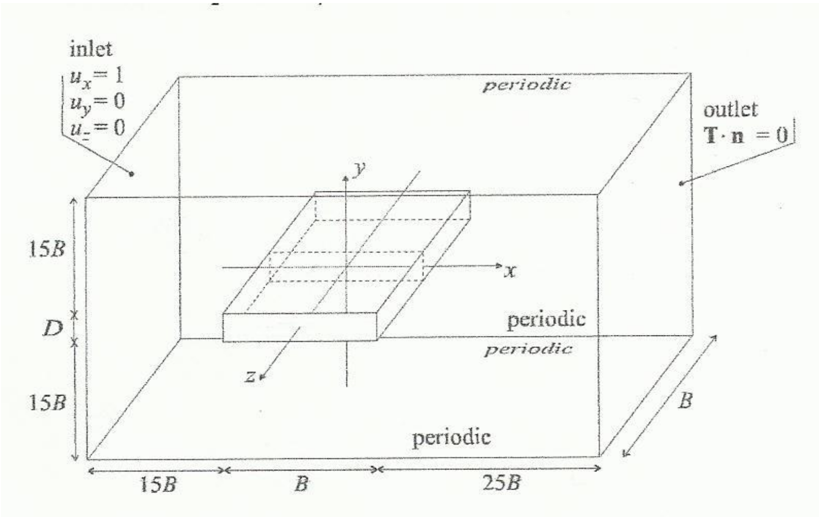


Fig. 1: Computational domain and boundary conditions (not drawn to scale)

Discretisation:

- The LES uses a second order central differencing scheme for space discretisation and a second order backward Euler scheme for the time discretisation. A hexahedral grid is adopted to discretize the spatial computational domain.

Meshing:

- LES requires more globally refined mesh in order to properly resolve the eddies in the flow (fine mesh in the mesh sensitivity analysis) to make sure that flow near to the wall is properly resolved. The rest of the regions where SST are active have a coarser mesh, which helps to save computational power.

Time step and convergence:

- The time is non-dimensionalised by U and D. The non-dimensional time-step is set equal to $\Delta t = 5 \times 10^{-3}$, which provides an accurate advancement in time and a CFL number close to unit. The simulation is extended over T = 800 non-dimensional time units in order to have a long enough statistical sample to obtain converged statistics, after having excluded the initial transient. Computations are carried out on 8 Intel Quadcore X5355 2.66GHz CPUs and require about 2.5 GB of RAM memory and 15 days of CPU time for the whole simulation [7].

- The time step used in the simulation is 0.00015, where 50 time steps are necessary to generate one vortex cycle in the flow. This is equivalent to 0.08 in term of the non-dimensional time step. The residuals of convergence of the solution are maintained at $10^{-4}$ to keep the errors at an acceptable level [5].

## Conclusion

The study of the flow around rectangular sections provides fundamental understanding of the flow characteristics around the sections and bodies with similar shapes. This also provides insight into the aerodynamic characteristics of the flow around a bridge deck section. Validation of the turbulence models (LES) on the flow around rectangular sections with aspect ratio of 5:1 has been conducted based on the comparison of fundamental flow characteristics with experimental findings. These include velocity profiles, pressure field and distribution of vortices, as well as the changes of Strouhal number and drag coefficient at the aspect ratio. From the work done on the LES, it was concluded that LES is a reliable and an accurate model for unsteady and complex flow simulation. Details of eddies and vortex structures are well captured but a more refine mesh is needed. The application of the principal component analysis (PCA) on the study of the pressure distribution around the cylinders simplified and identified the dominant pressure distribution on the surface of the cylinder and thus the aerodynamic forces induced. This approach proved fruitful and indicated the possibility of its application on the flow around a bridge deck section where complex flow features with intense vortex interaction is involved.

## References

1. A.N. Grozescu, L.Bruno, D. Fransos, M.V. Salvetti. Large-eddy simulations of a Benchmark on the Aerodynamics of a Rectangular 5:1 Cylinder. In BBAA VI international colloquium on bluff bodies aerodynamics & applications, Milano, Italy; (2008).

2. Bruno L, Fransos D, Coste N, Bosco A. 3D flow around a rectangular cylinder: a computational study. Journal of Wind Engineering and Industrial Aerodynamics, 2008.

3. C. Mannini, K. Weinman, A. Šoda, G. Schewe. Three-dimensional numerical simulation of flow around a 1:5 rectangular cylinder. EACWE 5 Florence, Italy 2009.

4. D. Yu, A. Kareem. Two-dimensional simulation of flow around rectangular prisms, Journal of Wind Engineering and Industrial Aerodynamics 62 (1996) 131-161.

5. Kai Fan Liaw, Simulation of Flow around Bluff Bodies and Bridge Deck Sections using CFD; Flow around Rectangular Cylinders. Dissertation Submitted for the degree of Doctor of Philosophy (PhD), University of Nottingham, England (2005).

6. Kishan B. Shah, Joel H. Ferziger. A fluid mechanicians view of wind engineering: Large eddy simulation of flow past a cubic obstacle. Journal of Wind Engineering and Industrial Aerodynamics 67&68 pg 211-224 (1997).

7. L.Bruno, N. Coste, D. Fransos. Analysis of the separated flow around a 5:1 rectangular cylinder through computational simulation. EACWE 5 Florence, Italy July 2009.

8. Mustafa Sarioglu, Tahir Yavuz. Vortex Shedding From Circular and Rectangular Cylinders Placed Horizontally in a Turbulent Flow. Turk J Engin Environ Sci 24, 217-228 (2000).

9. Nakaguchi H, Hashimoto K, Muto S, An experimental study on aerodynamics drag of rectangular cylinders. J. Japan Society of Aeronautical and Space Sci., 1968, Vol.16, p. 1-5.

10. Norberto Nigro, German Filippini, Gerardo Franck, Mario Storti and Jorge D'Elıa flow around a sharp-edged surface-mounted cube by large eddy simulation. In Mecanica Computacional, vol. XXIV, (2005).

11. O. Almeida, S. S. Mansur, A. Silveira-Neto. On the flow past rectangular cylinders: physical aspects and numerical simulation.

 

Engenharia Térmica (Thermal Engineering), Vol. 7, No 01, p. 55-64 (2008).

12. Okajima A, Strouhal number of rectangular cylinders. Journal of Fluid Mechanics, Vol.123, p. 379-398 (1982).
13. Okajima A, Numerical simulation of flow around rectangular cylinders. Journal of Wind Engineering and Industrial Aerodynamics, vol 33, 171-180 (1990).
14. Rokugou A, Okajima A , Kamiyama K, Numerical Analysis of Flow around Rectangular Cylinders with Various Side Ratios. Japan Society of Mechanical Engineers vol.68; no.670; page.1608-1613 (2002).
15. S. Berrone, V. Garbero, M. Marro. Numerical simulation of low-Reynolds number flows past rectangular cylinders based on adaptive finite element and finite volume methods, Computers & Fluids 40 (2011) 92–112.
16. Stefano Malavasi, Nicola Trabucchi, Numerical investigation of the flow around a rectangular cylinder near a solid wall. In BBAA VI International Colloquium on: Bluff Bodies Aerodynamics & Applications Milano, Italy, July, 20-24 (2008).
17. Sumeet Thete, Kaustubh Bhat and M. R. Nandgaonkar. 2D Numerical Simulation of Fluid Flow over a Rectangular Prism. CFD Letters Vol. 1(1) 2009.
18. Yu Dahai, Ahsan Kareem, Parametric study of flow around rectangular prisms using LES. Journal of Wind Engineering and Industrial Aerodynamics 77&78 p. 653-662 (1998).
19. Zhigang Wei, Ahsan Kareem. A benchmark study of flow around a rectangular cylinder with aspect ratio 1:5 at Reynolds number 1.E5. Global Center of Excellence founded by the MEXT, Japan (2009).
20. Charles Meneveau, Turbulence: Subgrid-Scale Modeling, Johns Hopkins University Scholarpedia, 5(1):9489, 2010.

 