Annual Journal of Hydraulic Engineering, JSCE, VOL.44, 2000, February

EFFECTS OF FINITE DIFFERENCE SCHEMES ON LES OF TURBULENT FLOW AROUND BLUFF BODY

Sankara N.VENGADESAN[1] and Akihiko NAKAYAMA[2]

[1]Research Associate, Division of Global Development Science, Graduate School of Science and Technology, Kobe University
[2]Professor, Division of Global Development Science, Graduate School of Science and Technology, Kobe University
(1-1, Rokkodai Nada, Kobe 657-8501)

LES simulations have been performed using different finite difference schemes derived by a general procedure for discretising the convective terms to evaluate their importance in the results of the simulation. Two typical subgrid-scale stress models have also been examined. The flow past a square cylinder is taken as a benchmark test case of general flow around a bluff body. The code was initially validated at different low Reynolds number and then calculation results are compared with experimental data. Dynamic Smagorinsky model is seen to improve the solution marginally. Upwind-biased scheme of appropriately high order has been found to give satisfactory results.

Key words: *LES, square cylinder, finite difference schemes*

1. INTRODUCTION

As fast and large-capacity computers are becoming affordable nowadays, large computational loads can be tolerated in many practical calculations of turbulent flows, particularly if accuracy and universality are gained. Large-Eddy Simulation (LES) is one such technique that once thought to be too expensive for practical calculations has a potential to be exploited in various engineering and environmental applications and certainly to many flows appearing in hydraulics. Though there have been many attempts of using this potentially useful technique, there still is a lack of uniformly accepted methodology. Different methods are proposed, but they yield different results. The main element of LES method has been though to be modelling of subgrid stresses and many models have been proposed[1-3], but the outcome of recent workshop attended by leading developers indicates that the results depend on many other factors like grid resolution, numerical schemes and boundary conditions[4]. While there are many elements in the numerical solution of the equations of motion, such as how the pressure is computed and time advancing is done, but the most important is the discretisation of the convective terms due to its non-linearity and dominancy at high Reynolds number. While discretising this term, the leading truncation error of even order has dissipative and that of odd order has dispersive influence on the solution and higher order ones lessen these effects. So far there has not been a conclusive study reported on the influence of various numerical aspects on LES, since it has been difficult for an individual to study the effect of all numerical aspects.

In the present study we examine the importance of finite difference schemes to be used in typical LES of flows around bluff body, taking the flow past a square cylinder as a benchmark test case. This is a case for which detailed experimental data are available and is the test case of the workshop reported by Rodi et al.[4]. Using a general procedure, finite difference schemes of arbitrary orders of accuracies are derived for the discretisation of the convective terms. The results are examined for a range of Reynolds number and different grid resolution with typical subgrid stress models. Finally, calculation of turbulent flow past a square cylinder at the Reynolds number of 22,000 is performed and detailed comparisons are made.

2. NUMERICAL METHODS

The basic equations used in LES are three-dimensional, time dependent, Navier-Stokes equations, filtered in order to separate the large scale and the small scale motions. In this study incompressible flow is considered and the code solves the filtered governing equations along with closure subgrid-stress model by finite difference procedure. As the focus of the present work is a study on the discretisation and model influence, governing equations are not described here and

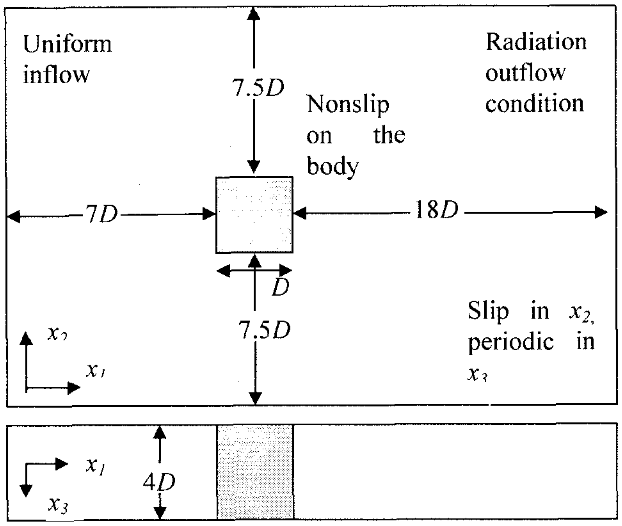


Fig. 1 Computational domain and boundary condition

Table 1. Computation grid

Grid Size	$\Delta x_{1,\min}$ $\Delta x_{2,\min}$	points inside the cylinder	Stretching Ratio Sr_{xu}, Sr_{xd}, Sr_y
101x 91x21	0.05 D	21x21	1.12, 1.06,1.06
130x111x21	0.04 D	26x29	1.05
256 x 161	0.02 D	51x51	1.03

they can be found in Nakayama and Noda[5] and vairous literature. Numerical procedures employed to solve the equations are explained in this section.

(1)Calculation domain and computational grid

The flow configuration considered is shown in Fig.1. It is uniform flow past a square cylinder and the associated boundary conditions for the numerical computations are as described. As the body is geometrically simple, cartesian co-ordinate system with staggered mesh arrangement, is used. The grids are uniformly spaced inside the cylinder and for short region just outside and then unequally spaced determined by the stretching factor. Based on these and for the present computational domain, two sizes of grid are arrived at: i) 101x91x21 and ii) 130x111x21 for all 3-D cases, referred to as G1 and G2 respectively in this study. x_1-coordinate origin is fixed on the rear side of the cylinder and that of x_2-coordinate coincides with the axis of the cylinder.

It is reported[6] that very fine grid resolution near solid wall enabled by 2-dimensional simulation, may be more important than representing 3-dimensional motion by a 3-D LES, which limits the grid size. Hence, 2-D LES is also performed. To adopt the same code developed for 3-D to 2-D easily, number of grid points in the spanwise direction is reduced to the minimum required, computation domain extends from -6*D* to 16*D* in the x_1-direction and 15*D* in the x_2 direction. and a grid of 256x161 is chosen for 2-D case. Table 1. gives relevant information for all the chosen grid.

(2) A general procedure for the discretisation of the convective terms

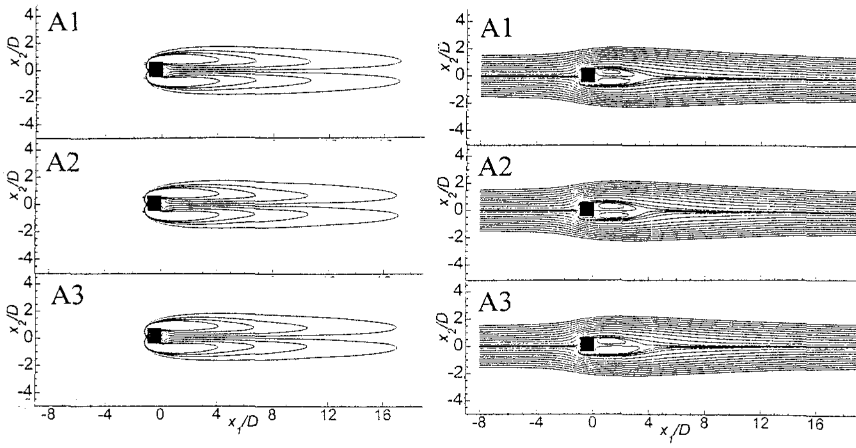
In the governing equations, convective terms are expressed in gradient form. They are discretised by finite difference schemes by a general procedure in the program as explained here. If $f(x)$ is the continuous function to be represented by discrete values $f(x_i)$, the finite difference formula for the *m*th derivative at $x=x_0$ and for any order of accuracy (*N*), the problem is reduced to finding the weights ($\delta^m_{n,v}$). The general finite difference formula for the derivative at $x=x_o$ is writen as a function of $f(x)$

$$\left.\frac{d^m f}{dx^m}\right|_{x_0} = \sum_{v=0}^{n} \delta^m_{n,v} f(\alpha_v),\ m=0,1\ldots M;\ n=m,m+1,\ldots N$$

where, *m* is order of derivative and *N* is order of accuracy. For the convective term discretisation, the order *m* is set to 1. The weights are calculated (Fornberg[7]), by taking the relative position of neighbouring points (α_v). The order in which the neighbouring grid points (α_v) are described and number of neighbouring points considered, decide the particular differencing scheme with required accuracy. So, by one control input from outside, the same code switches to the desired scheme. For the present study, four schemes were considered viz., a) second order central, b) fourth order central, c) third order upwind-biased and d) fifth order upwind-biased and are referred to as CD2, CD4, UPB3 and UPB5 respectively.

(3)Other numerical details

Viscous terms are discretised by second order accurate centeral differencing. Uniform inflow is specified at the upstream end and radiation outflow condition is applied at the downstream end. The periodic conditions and slip conditions are assumed for the spanwise and cross-flow directions, respectively. The nonslip boundary conditions are applied on the body surface. HSMAC scheme is used for pressure calculation step. As it is required to capture resolved fluctuations in time, time advancing is performed by second order accurate explicit, Adams-Bashforth method for the convective terms, which is stable for any value of CFL number. For other terms implicit Euler method is followed. All the



(a) Vorticity

(b) Streamlines

Fig.2. Results for laminar case, Re=50

Table 2. Laminar test cases

Re	Grid	Scheme	Key
50	G1	CD2	A1
	G1	CD4	A2
	G1	UPB3	A3
100	G1	CD2	B1
	G2	CD4	B2
	G1	UPB3	B3

Table 3. Effect of subgrid-stress test cases

Re	Grid	Scheme	Key
100	G1	CD2	C1
	G2	UPB3	C2
200	G1	CD2	D1
	G1	UPB3	D2
	G2	CD2	D3
	G2	UPB3	D4

calculations are performed with the non-dimensional time increment of 0.001 for 3-D case and with 0.0005 for 2-D case.

(4)Subgrid stress models

The filtering procedure introduces the non-resolvable subgrid scale stresses which describe the influence of the small scale structures on the larger eddies. For modelling these subgrid stresses, two different models are used namely, (a) well known Smagorinsky model[8] and (b) dynamic model proposed by Geramano et al.[9]. In order to resolve viscous sublayer and to apply the nonslip boundary condition on the wall, the first grid point next to the wall in the normal direction should be set close to the wall (interms of non-dimensional wall distance should be less than 20). This point becomes physically closer and closer to the wall, as the Reynolds number increases. But, it is found from approximate calculation for both the grid used in this study, that the first point was not in this region. So, Van-Driest type near-wall damping for eddy viscosity is not used. Thus, Smagorinsky constants - C_s and C_k, are being set to the constant value of 0.13 and 0.094 respectively. For the dynamic procedure, additional filtering is performed by top-hat filter. Following a suggestion of Lilly[10], a least-squares approach is used to determine values of C_s^2. Additionally, negative eddy viscosities are clipped. Moreover, computations have been performed without any subgrid-stress model.

3.PERFORMANCE OF THE METHOD

Before calculating the actual test case, it is necessary to validate all the schemes and the model incorporated in the code. So, at first calculations are performed at different low Reynolds number. All the calculation results shown here are at the non-dimensional time of tU_{in}/D=20, after the time marching calculation is started from the uniform-flow initial conditions.

(1)Validation by laminar flow calculation

For this purpose, Smagorinsky's constant are set to zero, which means, the subgrid stresses are neglected and the flow is laminar at such low Re. Table 2 gives the list of different cases run and the keys used to refer to them. Fig. 2(a) shows the results for Re=50, represented by the plots of contours of constant spanwise component vorticity. At this low Reynolds number, the flow is steady and symmetric with a separation bubble with twin vorticies. It is seen that all the calculation results agree with one another and show anti-symmetric vorticity distribution around the *x*-axis accurately.

The results represented by streamlines in Fig.2(b) also indicate the same trend and nearly symmetric separation bubbles of almost the same length. These are traces of constant stream function and shapes of the streamlines depend very sensitively on the value of the stream function. At Re=100, the flow starts to be a little unsteady and the flow pattern start to depend on time. The results using different finite difference scheme shown in Fig.3, indicate that the results of different method still agree very well. The CD4 scheme using the coarse grid G1 diverged and the results shown here are for the fine grid G2. Higher order schemes show more undulations indicating sensitivity to stability. These results indicate that the laminar flow at low Reynolds number is computed correctly

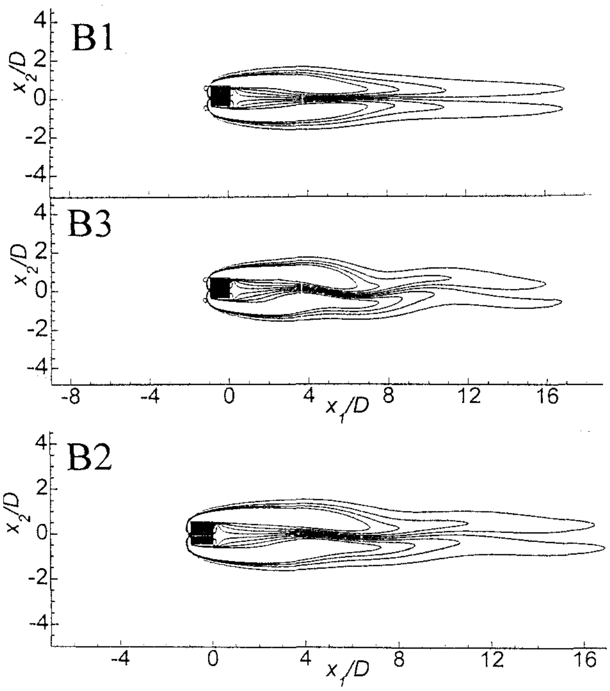


Fig.3. Vorticity for laminar case, Re=100

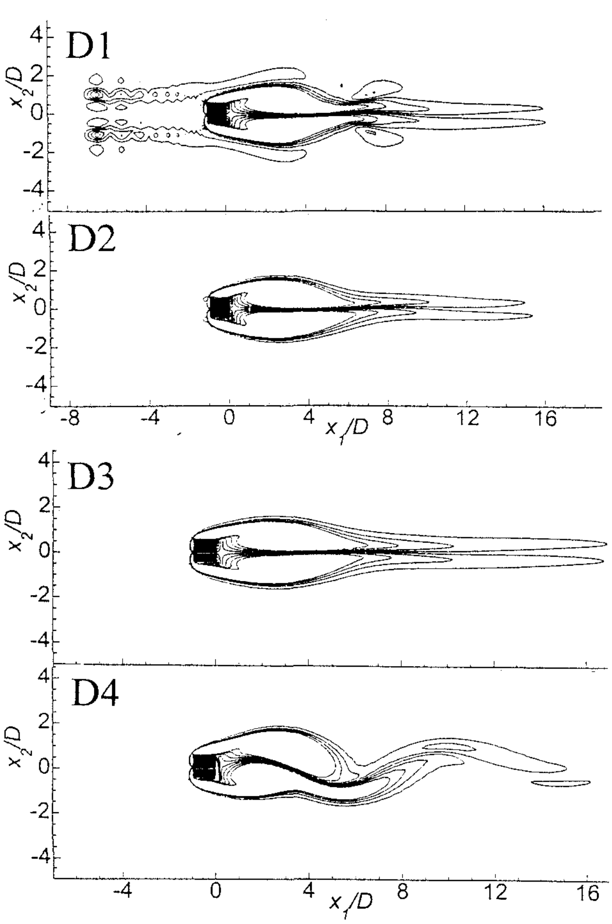


Fig.4. Vorticity for turbulent case, Re=200

(2)Effects of subgrid-stress model

Table 3 shows the calculation cases for flows with the subgrid-stress model included. The case of Re=100 is also considered, which is low enough so that subgrid stresses are not important. Results of this test case confirm that calculation by the same code, by setting the Smagorinsky constant to zero can be consistant with that the flow is laminar. Fig. 4(a) is the corresponding results for the case of Re=200. Higher-order schemes on fine grid are seen to show start of vortex shedding. However, it is to be noted that CD2 scheme on coarse grid shows spurious oscillations in vorticity upstream of the cylinder and (not shown) streamlines show only small disturbances, which vanishes when used with fine resolution. CD4 scheme has not worked at all.

4.VALIDATION BY EXPERIMENTAL DATA

In this section results are discussed for the case of turbulent flow at Re=22,000 for which experimental data are available[11,12]. This case has been studied extensively and considered as the benchmark problem for LES calculation. For this case, in addition to the third order, fifth order upwind-biased scheme is also considered as a higher order scheme reduces the effect of numerical viscosity. Also, 2-D LES is also performed with one grid size and with third order upwind-biased scheme. Both time and spanwise averaging have been done for the statistical quantities.

It is to be mentioned here that second order central difference scheme has not worked for this high Reynolds number even with present fine grid resolution. Fig.5 shows example of instantaneous flow pattern in terms of streamlines and vorticity. Fig. 6(a) is a plot of the mean velocity along the center-line behind the cylinder. Fig.6(b) compares velocity profile on the cylinder. In fig. 7(a), (b) and (c), turbulent quantities are plotted from both calculation and experiment. Fig.8 shows variation of mean surface pressure computed by present method and the experimental results of Lee[13] and Otsuki et al.[14]. Table 4 gives the comparison of bulk quantities for this case.

(1)Three dimensional effect

Because more points are provided very close to the wall, 2-D LES is seen to predict better in the boundary layer velocity profile on the cylinder side. The mean centerline velocity, however is grossly under-predicted. This may be due to the

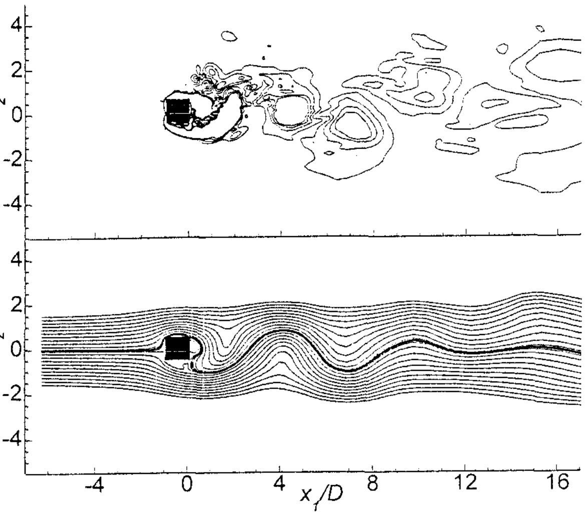
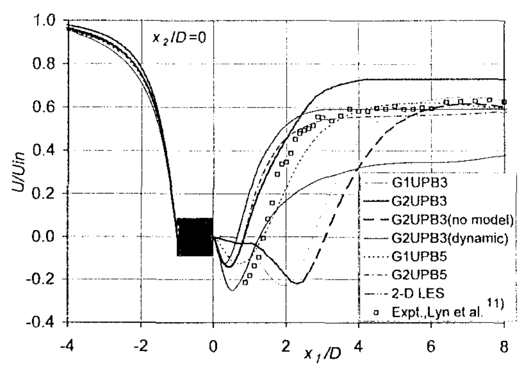
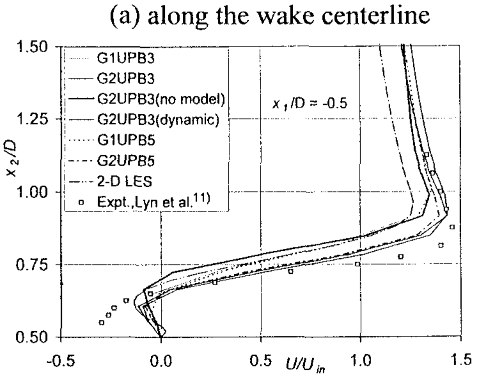


Fig.5. An example of instantaneous vorticity and streamlines for case G2UPB5

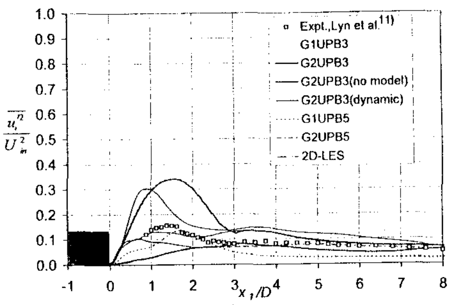


(a) along the wake centerline

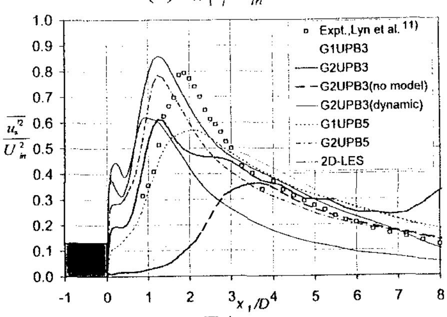


(b) on the cylinder side

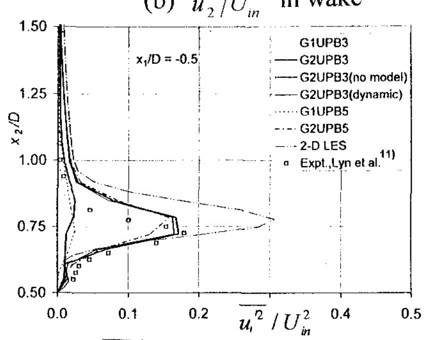
Fig.6. Comparison of mean velocity



(a) $\overline{u_1'^2}/U_{in}^2$ in wake



(b) $\overline{u_2'^2}/U_{in}^2$ in wake



(c) $\overline{u_1'^2}/U_{in}^2$ on the cylinder side

Fig.7. Turbulent stresses

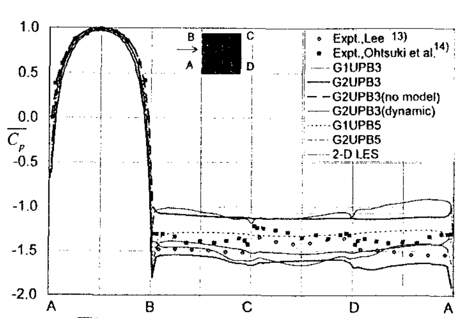


Fig. 8. Mean surface pressure

delayed vortex shedding as seen from the calculated Strouhal number. Pressure on the rear side of the cylinder and sides is overpredicted, reducing the drag coefficient. The increased resolution near wall provided by a 2-D LES does not seem to improve the overall predictive ability.

(2) Influence of discretisation

Examination of the effects of different discretisation schemes is the main subject of the present study and is done in this subsection.

It is seen from Fig.6(a) that the results improve with the order of accuracy, if it is of odd-order and that the results using finer grid is better.

About the normal stress components both streamwise and crosswise, from Fig. 7(a) and Fig.7(b), it can be seen that, UPB5 scheme capture the peak value very close to that of experiment and all the scheme predicts the decay of the stress in the wake well.

UPB3 scheme on the coarser grid calculates mean drag coefficient lower. But for three cases, the values of the Strouhal number (St) fall in a close range of experimental data. This observed phenomena matches with comment reported in Rodi et al.[4], as the accurate prediction of this quantity need not be an indication of quality simulation.

(3) Influence of the subgrid-stress model

It is seen from the results that without the subgrid model that the numerical viscosity due to upwind difference alone is inadequate. On the other hand, both the conventional and dynamic Smagorinsky model using higher-order differencing scheme appear to be very close, indicating that the refinement in the subgrid stress model is not as important as the proper difference scheme or grid densitv.

Table 4. Comparison of Bulk parameters

Case	C_d	L_R	St
Expt.	2.1	1.38	0.132
Sub-Cases			
G1UPB3	1.78	2.41	0.150
G1UPB5	2.02	1.32	0.125
G2UPB3	2.20	1.28	0.135
G2UPB3 (no model)	1.88	3.00	0.110
G2UPB3 (dynamic)	2.24	1.22	0.136
G2UPB5	2.14	1.28	0.135
2-D LES	1.78	1.99	0.18

C_d - mean drag coefficient; L_R - Reattachment length

5. CONCLUSIONS

A general code to discretize the non-linear convective term for arbitrary order of accuracy also incorporating two typical subgrid-scale stress model is written and a detailed study of their influence on LES was made. Other numerical aspects of computation were kept same for all the cases considered. Code developed was initially validated with laminar cases and then for subgrid scale model at relatively low Reynolds number cases. It is found from this study that central difference schemes have stability problems and sensitive to grid resolution. While upwind-biased schemes work without any problem, higher order central differencing scheme fails to work computationally even at moderatively high Reynolds number. On comparing the simulation results at Re=22,000 with experiment, it is noted that on the whole, higher order upwind-biased scheme on fine grid reproduces the experimental results reasonably well. This better performance can be attributed to higher order accuracy and lesser magnitude of numerical dissipation. 2-D calculation results clearly underline the need for 3-D calculation in LES type of simulation, even though one can resolve the viscous sublayer by having a very fine grid. Although dynamic subgrid-stress model with lower order differencing scheme predicts better, it is computationally 1.3 times expensive than using higher order upwind-biased scheme. Hence, it is suggested that for engineering purposes, fifth order upwind-biased scheme with conventional Smagorinsky subgrid-stress model will do the needful.

REFERENCES

1) Horiuti, K. : A new dynamic two-parameter mixed model for large eddy simulation, *Phys. Fluids*, Vol 9, n11, pp.3443-3464, 1997.
2) Murakami, S.: Overview of turbulence models applied in CWE - 1997, *J.Wind Engineering and Industrial Aerodynamics*, Vol.74-76, pp.1-24, 1998.
3) Sohankar, A., Davidson, L. and Norberg, C.,: A dynamic one-equation subgrid model for simulation of flow around a square cylinder, *Engineering Turbulence Modelling and Experiments* - Vol.4, (Eds.,) W.Rodi and D.Laurence, pp.227 - 236, 1999.
4) Rodi, W., Ferziger, J.H., Breuer, M.and Pourquie, M.: Status of Large Eddy Simulation:Results of a Workshop, *J. of Fluids Engineering*, Vol.119, pp.248-262, 1997.
5) Nakayama, A. and Noda, H.: LES simulation of flow around a bluff body fitted with a splitter plate, *J.Wind Engineering and Industrial Aerodynamics*, 1999, (accepted for publication)
6) Bouris, D. and Bergeles, G.: 2D LES of vortex shedding from a square cylinder, *J. Wind Engineering and Industrial Aerodynamics*, Vol.80, pp.31-46, 1999.
7) Fornberg, B.: Generation of finite difference formulas on arbitrarily spaced grids, *Mathematics of Computation*, Vol.51, 184, pp.699-706, 1988.
8) Smagorinsky, J.: General circulation experiments with primitive equations-I, The basic experiment, *Mon. Weather Review*, Vol.91, pp.99-165, 1963.
9) Germano, M., Piomelli, U., Moin, P. and Cabot, W.H. : A Dynamic subgrid scale eddy viscosity model, *Phys. Fluids* A, Vol.3, n7, pp.1760-1765, 1991.
10) Lilly, D.K. : A proposed modification of the Germano subgrid -scale closure method, *Phy. Fluids* A, Vol.4, n3, pp.633-635, 1992.
11) Lyn, D. A.and Rodi, W.: The flapping shear layer formed by flow separation from the forward corner of a square cylinder, *J. Fluid Mech.* Vol.267, pp.353-376, 1994.
12) Lyn, D. A., Einav, S., Rodi, W. and Park, J. H.,: A laser Doppler velocimetry study of ensemble-averaged characteristics of the turbulent flow near wake of a square cylinder, *J. Fluid Mech.*, Vol.304, pp.285-319, 1995.
13) Lee, B. E.: The effects of turbulence on the surface field of a square prism, *J. Fluid Mech.*, Vol.69, pp.263-282, 1975.
14) Otsuki, H., Fujii, K., Washizu, H. and Ohya, A.: On the characteristics of three components of aerodynamic forces and pressure distribution around a fixed rectangular cylinder in a uniform flow, *5th Symposium on wind loads on structures*, pp. 169-175, 1987, (in Japanese).

(Received September 30,1999)