International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 10 Issue 3 ǁ 2022 ǁ PP. 51-58

Numerical investigation on the flow around blunt bodies with different turbulence modeling

[1,2]Yuguang Bai*, [1]Sheng Zhang, [1]Lixin Cao
*[*1]Advanced Technology for Aerospace Vehicles of Liaoning Province, Dalian University of Technology, Dalian, China*
[2]School of Aeronautics and Astronautics, Dalian University of Technology, Dalian, China
Corresponding Author: Yuguang Bai

Abstract
Numerical investigations of the flow aerodynamics around blunt bodies with different three dimensional turbulence model are presented in this paper to find an effective method for numerical simulations of long-span structure aerodynamics. The cross section of a long-span bridge and an inverse-U shaped beam section are modeled by three dimensional CFD method. The fluid-structure interaction effect is included. Three dimensional turbulence models including RANS, LES and hybrid RANS/LES are employed. Aerodynamics fluid force coefficients of the two structures are computed and the turbulence flows around them are analyzed. Numerical simulation results with numerical computations and wind tunnel test results are presented for comparison. It can be found from the computed results that with the proposed coupling algorithm and mesh control method, RANS model is effective enough for the streamlined blunt body while LES and hybrid RANS/LES are still necessary for blunt body structures with sharp edges. These investigations have important application potential in numerical analysis of long-span structures.

Keywords: *blunt body, turbulence modeling, aerodynamics, flutter computation.*

Date of Submission: 18-02-2022 Date of acceptance: 03-03-2022

I. INTRODUCTION

Flows around blunt body structures such as long-span bridges play an essential role in determining the environment of the populace which use them and sometimes the integrity of the structures themselves. Wind-induced vibration including fluid-structure interaction (FSI) is one of the challenging problems related to these structures. Tacoma bridge in USA was damaged due to such phenomena in 1940 when the wind velocity is only 19m/s [1]. Thus there are strong incentives to find effective ways of modeling these phenomena.

From the last century, wind tunnel experiments have been the most popular way to test the wind-induced vibration problem, but the cost cannot be taken by most engineers and designers working for industrial wind engineering. Additionally, sometimes these experiments can be influenced by unexpected factors such as model geometrical, measurement complexity which can produce the error of the test results. The most recent trend is computational fluid dynamics (CFD). For the past decades, the power of computers has been increasing continuously and many experiments can be partly replaced by numerical simulations. Now CFD has been recognized as an effective analysis tool for interdisciplinary numerical investigations [2].

In wind engineering, Transient or steady wind loadings are usually necessary inputs for structure designs. For long-span structures like bridges, it is often necessary to consider the FSI effect, in which the wind-induced vibration can feedback into the flow itself, causing at worst amplification of the forces, leading to structural failure. As a start step of the eventual study of complex FSI problems, streamlined structures like cylinders and airfoils can be treated as an initial methodology [3]. An efficient FSI coupling algorithm is necessary and mesh or meshless methods also attract much attention. For wind-induced vibration problems of long-span bridges, a most famous meshless method is the discrete vortex method (DVM) [4]. It employs the classical meshless method to solve the bridge FSI problem with moving boundaries and is computationally efficient though it is criticized due to that it is not easily extend to three dimensional (3D) flows. Bai et al. have proposed a method based on Gauss-Seidel block-iterative coupling algorithm to solve FSI problem and applied it on aerodynamic analysis of NACA0012 airfoil [5]. The computed results have achieved good agreements with the experiments results and the theoretical values. But 3D numerical simulations of aerodynamics of blunt body structures are still demanding more accurate CFD methods due to that flows around these structures like Tacoma Bridge with sharp edges are complex to be simulated [6].

One of the factors of these difficulties is turbulence modeling. Flows around long-span structures are highly turbulent and high Reynolds (Re) number is usually a concern. Without proper turbulence modeling, the resultant atmosphere cannot be representative of physical flows. As what is well known, the biggest progress in turbulence re-search is in turbulence modeling. Actually, CFD industry is based on turbulence modeling. Direct numerical simulation (DNS) is a theoretical accurate method but the computational demand is huge. At the beginning of the turbulence modeling research, two dimensional (2D) turbulence models were most popular due to the computational ability. Reynolds-averaged Navier-Stokes (RANS) modeling is a conventional method. Launder and Spalding [7] presented that computational economy, range of applicability and physical realism of early turbulence models in which the magnitudes of two turbulence quantities were studied. Spalart [8] proposed the advantages of RANS modeling for the computational complexity of 3D CFD studies. But large eddy simulation (LES) appears as the most promising candidate to predict unsteady phenomena appearing in complex 3D flows [9]. Considering the computer power and to decrease the LES resolution requirements, a hybrid RANS/LES model named detached-eddy simulation (DES) was proposed by Spalart et al. [10]. The topic of this model is to solve near-wall flow with RANS modeling and far-wall flow with LES. It can achieve an effective balance between computational complexity and accuracy. Considering that computer power increases by a factor of 5 every five years, for external aerodynamic purposes, the 'readiness date' is approximately 2045 for LES and 2070 for DNS.

In order to find an effective CFD method accompanied with appropriate 3D turbulence modeling to simulated the flow around blunt body structures at high Re numbers, this paper employs RANS, LES and hybrid RANS/LES models based on Gauss-Seidel FSI coupling algorithm and efficient mesh control method to analyze aerodynamics of two different blunt body structures. The computed results will be compared to wind tunnel experimental results and the computed values by commercial software. From the comparisons of aerodynamic force parameters and flutter derivatives, a usage characteristic of turbulence modeling of streamlined blunt body structures and the ones with sharp edges can be concluded.

II. TURBULENCE MODELING

2.1 TURBULENCE MODELING METHOD

This paper uses three turbulence models: RANS, LES and DES. For the presented investigations, the flow is incompressible and turbulent, so the mass conservation and the momentum equation can be expressed as follows:

$$\frac{\partial \rho \bar{u}_i}{\partial t} + \frac{\partial \rho \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + 2\frac{\partial (\eta_t \bar{S}_{ij})}{\partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{1}$$

$$\frac{\partial \bar{u}_j}{\partial x_j} = 0 \tag{2}$$

with

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

and τ_{ij} is the sub-grid scale stress tensor resulting from the filtering operation

$$\tau_{ij} = \overline{\rho u_i u_j} - \rho \bar{u}_i \bar{u}_j = 2\nu_t \bar{S}_{ij} + \frac{1}{3}\tau_{kk}\delta_{ij}.$$

where p is the pressure, u_i is flow velocity in the direction x_i,η_t is dynamic viscosity coefficient and ρ is the fluid density. S is the modulus of the mean rate-of-strain tensor, defined as

$$S = \sqrt{2S_{ij}S_{ij}}, \tag{3}$$

The $\overline{(\cdot)}$ operator is a average for a RANS or a filter for a LES computation. For the former case, ν_t is the turbulent viscosity provided by RANS $k-\varepsilon$ model; and in the other case ν_t represents the sub-grid scale viscosity. δ_{ij} is the Kronecker coefficient [7].

This paper uses three turbulence models: RANS, LES and DES. For the presented investigations, the flow is incompressible and turbulent. There are some popular turbulence models for CFD computations. Many works have discussed advantages and disadvantages of them.

Unsteady RANS models are still the most popular in industrial application, especially for aeronautical engineering related to initial evaluations of aircraft aerodynamics. But for blunt body like structures with sharp edges, eddy simulations are recognized to be necessary. Some comparisons between RANS, LES and DES are shown in Table 1. It is seen that if unnecessary use for flows that RANS or LES can handle, also it is in terms of computational accuracy and efficiency, DES is a very good choice.

For the detail situation of each kind of turbulence models, the *k-ω* SST RANS model has some advantages than the *k-ε* RANS model: the *k-ε* model does not allow direct integration through the boundary layer and also produces excessive turbulence kinetic energy at impingement on the wall, which may significantly affect the flow patterns; and in contrast, the *k-ω* model allows direct integration through the boundary layer and benchmark testing shows that the *k-ω* model is particularly superior for the wall layer simulation. Now the *k-ω* SST RANS model is recognized as a better one to simulate separated flows. The DES *k-ω* SST model is thus used in this paper to keep relevance among the three kinds of turbulence models. For the LES simulations, the fluid solver is based on the finite volume method using the standard Smagorinsky sub-grid model with the sub-filter of the LES solver was switched on.

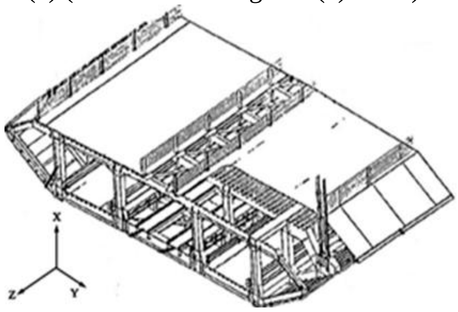
The second-order upwind and the second-order backward-Euler schemes were used for the advection and time domain integration.

Table 1: Turbulence model comparisons.

Model	Mesh level	Dependence on Re	Time steps	Usable time (year)
2D RANS	10E5	weak	10E3.5	1980
3D RANS	10E7	weak	10E3.5	1995
LES	10E11.5	weak	10E6.7	2045
DNS	10E16	strong	10E7.7	2080
DES	10E8	weak	10E4	2020

2.2 STRUCTURES

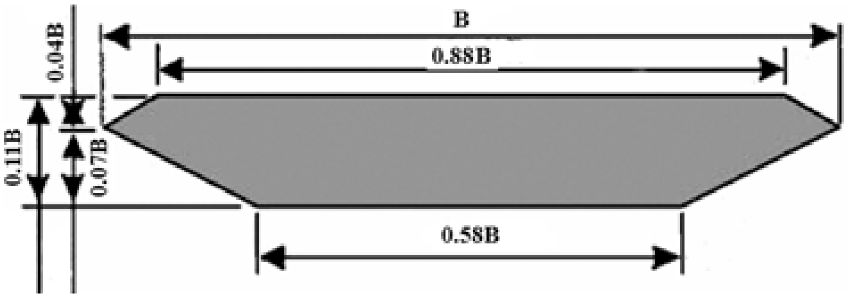
This Paper used two structures, as shown in Figure 1. The first one is the cross section of the classical bridge in civil engineering, see Figure 1(a) [11]. This structure has been used in the designs of many long-span bridge over the world. It was considered as a streamlined blunt body. The second one is an inverse-U shaped beam, see Figure 1(b). A wind tunnel experiment was accomplished with it. It is a blunt body with sharp edges, around which the flow features are complex. The geometry size of the two structures is shown in Figure 2. For the cross section of Tsing Ma bridge, this paper used a reduced scale of the real structure. The chord length B is given a value of 1m, see Figure 2(a). The length (i.e. perpendicular to the section shown on Figure 2) L is also equal to 1m. For the inverse-U shaped beam, B=0.7m and L= 0.7m according to the experiment, as shown in Figure 2(b) (i.e. the unit in Figure 2(b) is cm).



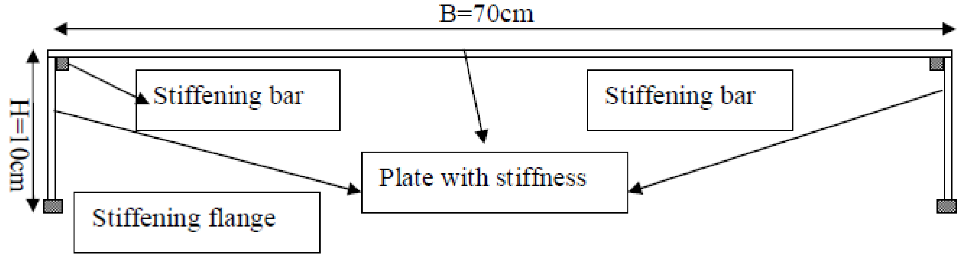
(a)

(b)

Figure1: Structures used.



(a)



(b)

Figure2: Geometry sizes of the structures used.

2.3 MESH GENERATION

Mesh generation in this paper makes use of the rigid plane assumption fundamental in classical beam theories [5]. The cylinder region under consideration is divided into a rigid region with $R \leq R_1$ and a buffer region with $R_1 < R \leq R_2$, see Figure 3(a) and 3(b).

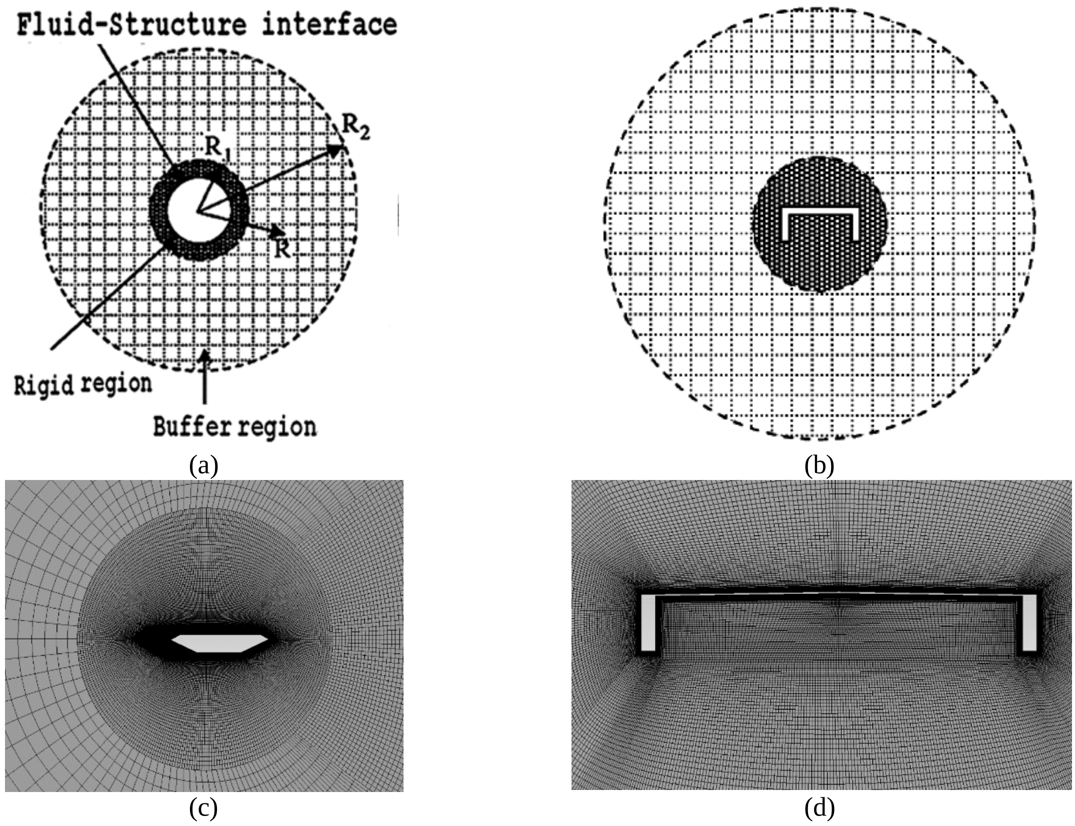


Figure3: Mesh generation

Figures 3(a) and 3(b) shows the meshes for the cross section of the bridge and the inverse-U shaped beam. It is cylindrical, centred on the structure and has $R_1 = 1.5B$ and $R_2 = 16B$. The meshes used are shown in Table 2 (i.e. for simplifications, this paper uses Bridge to represent the cross section of Tsing Ma bridge and U Beam to represent the cross section of the inverse-U shaped beam, respectively). The structured meshes used in this paper are all hexahedral cells which can ensure the accuracy of the computations.

Demanded meshes may be different among those of URANS, LES or DES. This paper wanted to employ consistent meshes to compare these three models for blunt body aerodynamics, while a most possible balance could be found.

Mesh independence tests were taken through different meshes of RANS and DES, as shown in Table 3. (i.e. the time increment used was 0.0003s for the U Beam (approximately a fiftieth of the time unit d/U, and U was 10m/s). Compared to the results of the U Beam form the wind tunnel experiments, workable meshes in the Table 2 could be found.

Table 2: Mesh generation.

Structure	Total meshes
Bridge	3,364,392
U Beam	2,433,472

The fluid velocity U can be computed from

$$Re = \frac{\rho U d}{\eta} \tag{4}$$

where d is the scale of the simulation domain which is equal to 16B in this paper; the fluid density ρ in this paper is equal to 1.185kg/m3; and η represents the dynamic viscosity of the fluid and the value of it is equal to 1.831E-5 in this paper. This paper uses a Re number 10E5 for the Bridge. The inlet velocity for the U Beam is equal to 10m/s according to the wind tunnel experiment and the corresponding Re number is equal to 7.25E6. The viscous boundary layer over the structure surface is well resolved by the fine mesh with the overall y-plus less than 2. The sub-viscous layer is resolved by the meshes.

III. NUMERICAL EXAMPLES AND DISCUSSIONS

3.1 AERODYNAMIC FORCE COEFFICIENTS

For the design of every bridge, the steady aerodynamic force coefficient must be tested or computed [12]. The wind flows from the left side to the right side of the structures in Figure 2. The aerodynamic force coefficients of the Tsing Ma Bridge can be computed from Equation (5) [11]:

$$C_d = \frac{F_D}{\frac{1}{2}\rho U^2 DL}, \; C_l = \frac{F_L}{\frac{1}{2}\rho U^2 BL}, \tag{5}$$

where D = 0.11B, as shown in Fig. 2(a). And those of the U Beam can be computed from Equation (5) which are proposed by the wind tunnel laboratory:

$$C_d = \frac{F_D}{\frac{1}{2}\rho U^2 HL}, \; C_l = \frac{F_L}{\frac{1}{2}\rho U^2 BL}. \tag{6}$$

In these two equations, C_d is the drag coefficient and C_l is the lift coefficient; B is the chord length of the section; and F_D is the aerodynamic drag force and F_L is the aerodynamic lift force. H=0.01m in Equation (5), as shown in Figure 2(b) (i.e. it should be noted that D in Equation (5) and H in Equation (6) both represented the vertical height of the section of the bridge and the U beam, respectively, as shown in Figure 2; and L in Equation (5) and Equation (6) are equal to 1m and 0.7m, respectively). The fluid velocity U for the bridge is 1.545m/s which is computed by Equation (4). The inlet fluid velocity for the U Beam is 10m/s as mentioned above. The time increment used was T =0.002s for the Bridge and 0.0003s for the U Beam (i.e. approximately a fiftieth of the time unit *d/U*).

Every time step the section surface pressure distribution is computed and integrated along the contour to form the time traces of drag, lift and moment. Figure 4 shows partial simulated time traces for the aerodynamic force coefficients obtained from 3D CFD simulations with RANS and DES models of the Bridge and the U Beam, respectively. Tables 3-6 show the computed results of the aerodynamic force coefficients for the two structures. It can be found that for the Bridge, 3D CFD simulations with the current numerical method can obtain similar results no matter which turbulence model is used. But when using commercial software (i.e. ANSYS workbench), the results of 3D CFD simulations with RANS are quite different those with LES and DES models. For the U Beam, 3D CFD simulations with the current method can obtain good agreements with wind tunnel experiments when LES and DES models are used. The RANS model is not successful to be applied when obtaining aerodynamic force coefficients. When using commercial software, only 3D CFD simulations with LES model can obtain satisfactory results. Additionally, there is a false stall phenomenon that the lift coefficient began to decrease while RANS model was used.

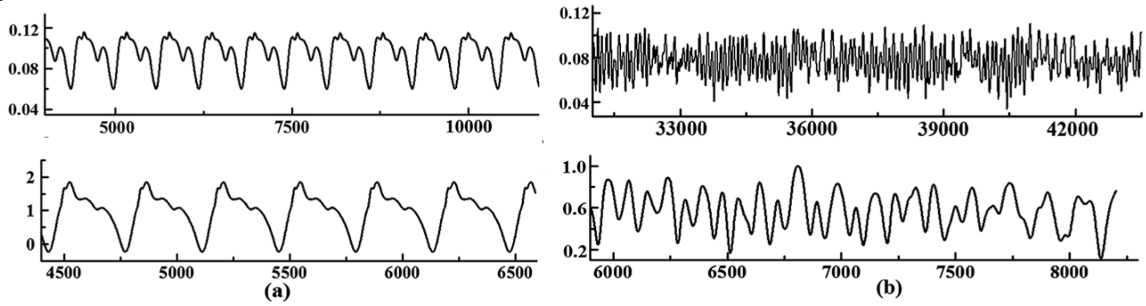


Figure4: Time trace of the aerodynamic drag force of the structures (the upper line is for the Bridge and the other line is for the U Beam) from 3D CFD simulations with the current numerical method: (a) with RANS model; (b) with DES model.

Table 3: Aerodynamic drag force coefficient of the Bridge.

	Angle of attack		
Results	0°	4°	8°
RANS	0.067	0.079	0.186
LES	0.061	0.071	0.174
DES	0.061	0.072	0.173
Commercial software (RANS)	0.086	0.113	0.293
Commercial software (LES)	0.067	0.076	0.179
Commercial software (DES)	0.071	0.096	0.184

Table 4: Aerodynamic lift force coefficient of the Bridge.

	Angle of attack		
Results	0°	4°	8°
RANS	-0.026	0.405	0.811
LES	-0.021	0.435	0.870
DES	-0.022	0.416	0.869
Commercial software (RANS)	-0.046	0.384	0.624

Commercial software (LES)	-0.021	0.431	0.863
Commercial software (DES)	-0.029	0.403	0.857

Table 5: Aerodynamic drag force coefficient of the U Beam.

	Angle of attack		
Results	0°	4°	8°
Wind tunnel experiments	1.268	1.882	2.919
RANS	1.615	2.383	2.688
LES	1.209	1.817	2.896
DES	1.197	1.789	2.877
Commercial software (RANS)	1.921	2.466	3.498
Commercial software (LES)	1.192	1.993	3.087

Table 6: Aerodynamic lift force coefficient of the U Beam.

	Angle of attack		
Results	0°	4°	8°
Wind tunnel experiments	-0.277	0.561	0.958
RANS	0.447	1.197	0.845
LES	-0.241	0.587	0.942
DES	-0.216	0.593	0.925
Commercial software (RANS)	0.564	1.393	1.362
Commercial software (LES)	-0.219	0.61	1.106

3.2 FLOW FEATURES

The Eddy viscosity distributions of the Bridge and the U Beam are shown in Figures 5 and 6, respectively. It can be seen form Figure 5 that there is no obvious flow separation around the Bridge, and the flow features obtained from the three turbulence models are similar. Thus the aerodynamic characteristics of this kind cross sections are suitable for the design of long-span bridges.

It can be seen from the Figure 6 that the flow around the U Beam is instable. Flow features obtained by 3D CFD simulations with RANS model obtains are quite different with those obtained by simulations with DES and LES models. There are obvious flow separations around the U Beam and the aerodynamic stability becomes worse after the angle of attack is changed. So it can be concluded that 3D CFD simulations based on eddy details are necessary for the aerodynamic analysis of the blunt body structures with sharp edges. RANS model cannot simulate this phenomenon though 3D CFD modeling is used.

3.3 COMPUTATIONAL EFFICIENCY

The comparison of the computational efficiency is shown in Table 7 (i.e. the angle of attack is 0°). It is seen that simulations with LES model have the most computational de-mands. With the proposed coupling algorithm and the mesh control method, the computational cost can be decreased significantly than that of commercial software. Especially for 3D CFD simulations with DES or LES models, the current numerical method can achieve better computed results with less computational times.

Table 7: Comparisons of the computational efficiency (Unit: hours).

	structure	Current numerical method	Commercial software
RANS	Bridge	163	180
	U Beam	171	235
DES	Bridge	295	423
	U Beam	384	571
LES	Bridge	856	1755
	U Beam	912	2067

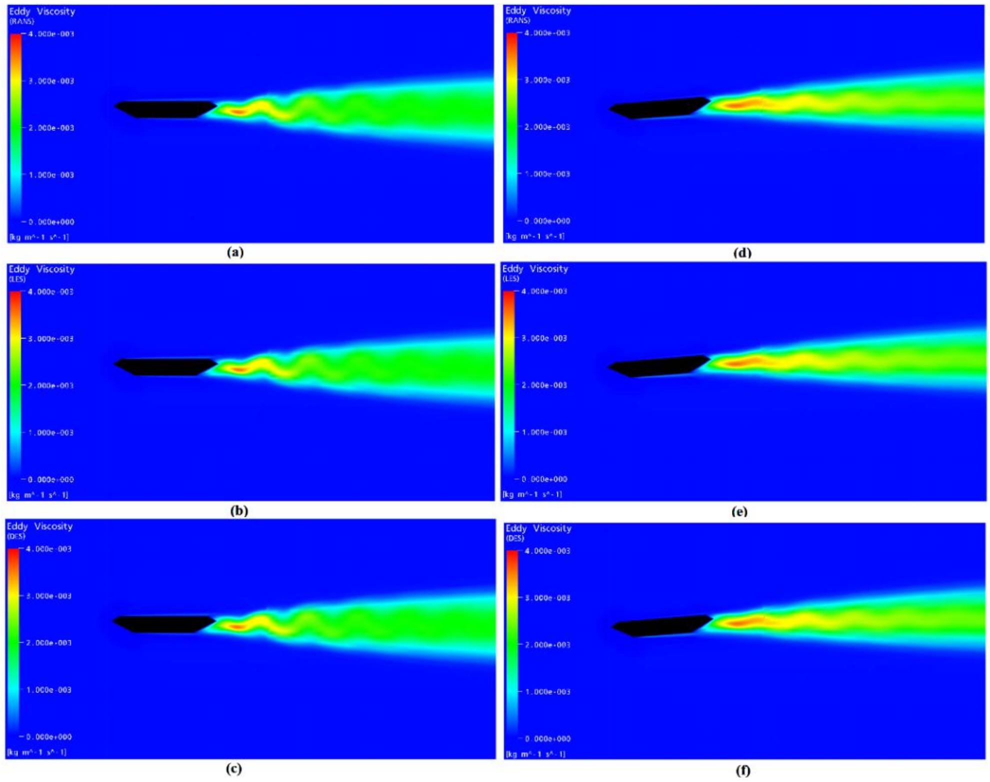


Figure5: Eddy distributions around the Bridge for different angle of attack and turbulence models: (a) angel of attack 0° and RANS model; (b) angel of attack 0° and LES model; (c) angel of attack 0° and DES model; (d) angel of attack 4° and RANS model; (e) angel of attack 4° and LES model; and (f) angel of attack 4° and DES model.

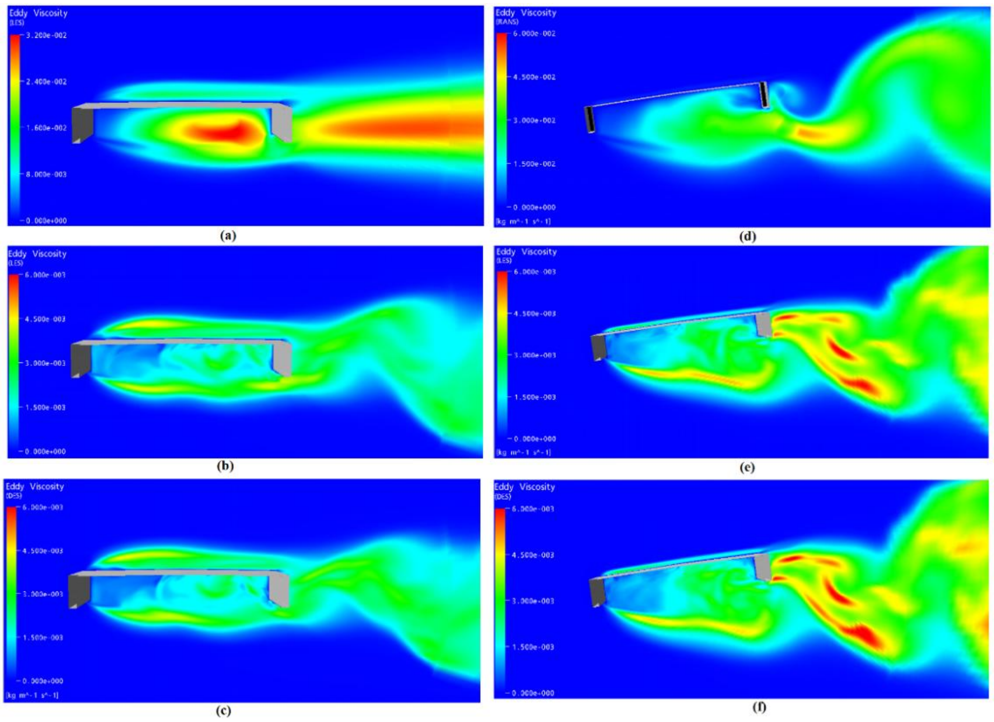


Figure6: Eddy distributions around the U Beam for different angle of attack and turbulence models: (a) angel of attack 0° and RANS model; (b) angel of attack 0° and LES model; (c) angel of attack 0° and DES model; (d) angel of attack 8° and RANS model; (e) angel of attack 8° and LES model; and (f) angel of attack 8° and DES model.

IV. CONCLUSION

A streamlined blunt body structure which is the cross deck section of classical bridges and a blunt body structure named inverse-U shaped beam are modeled with different 3D turbulence modeling based on CFD method. An efficient mesh control method is employed based on the classical beam theory. The aerodynamic force coefficients are computed from the current numerical method with RANS, LES and DES turbulence models. The computed results are compared with the wind tunnel experimental results and the computed values of the commercial software. The flow aerodynamics around the structures for different angle of attack and turbulence models are analyzed and the comparisons of the computational efficiency are presented.

It can be concluded:

a) for the streamlined blunt body structure like the cross section of classical bridges, 3D CFD numerical simulations with RANS turbulence model and efficient coupling algorithm and mesh control method can obtain accurate aerodynamic results;
b) for the blunt body structures with sharp edges, DES model should be used at least. And 3D CFD numerical simulations with this turbulence model can obtain satisfactory results with balanced computational demands than those with LES model;
c) The aerodynamic features around the structure with sharp edges are more complex than those of the streamlined blunt body structures. Many flow separations occur and appropriate 3D turbulence models are necessary to capture such phenomenon.
d) With efficient numerical method, less computational cost can be achieved.

The numerical simulations in this paper can be applied to the designs of long-span structures with different kinds of cross sections, which have significant application value in finding effective turbulence modeling and numerical analysis for wind engineering.

REFERENCES

[1]. Matsumoto, M., Shirato, H., Yagi, T., Shijo, R., Eguchi, A., Tamaki, H (2003) 'Effects of aerodynamic interferences between heaving and torsional vibration of bridge decks: the case of Tacoma Narrows Bridge'. Journal of Wind Engineering and Industrial Aerodynamics, Vol. 91(12-15), Pp. 1547-1557.
[2]. Mavriplis, C. (1999) 'Interdisciplinary CFD' International Journal of Computational Fluid Dynamics, Vol. 26(6-8), Pp. 333-335.
[3]. Tamura, T.(1999) 'Reliability on CFD estimation for wind–structure interaction problems' Journal of Wind Engineering and Industrial Aerodynamics, Vol. 81(1-3), Pp. 117-143.
[4]. LeMaitre, O. P., Scanlan, R. H. and Knio, O. M. (2003) 'Estimation of the flutter derivatives of an NACA airfoil by means of Navier–Stokes simulation' Journal of Fluids and Structures, Vol. 17(1), Pp. 1-28.
[5]. Bai, Y. G., Sun, D. K. and Lin, J. H. (2010) 'Three dimensional numerical simulations of long-span bridge aerodynamics, using block-iterative coupling and DES' Computers & Fluids, Vol. 39(9), Pp. 1549-1561.
[6]. Nguyen, C.H., Nguyen, D.T., Owen, J.S., Hargreaves, D.M. (2021) 'Wind tunnel measurements of the aerodynamic characteristics of a 3:2 rectangular cylinder including non-Gaussian and non-stationary features' Journal of Wind Engineering and Industrial Aerodynamics, Vol. 220, No. 104826.
[7]. Launder, B.E. and Spalding, D.B. (1974) 'The numerical computation of turbulent flows' Computer Methods in Applied Mechanics and Engineering, Vol. 3 (2), Pp. 269–289.
[8]. Spalart, P. R. (2009) 'RANS modelling into a second century' International Journal of Computational Fluid Dynamics, Vol. 23(4), Pp. 291-293.
[9]. Xu, L., Zhao, X., Xi, L., Ma, Y., Gao, J., Li, Y. (2021) 'Large-Eddy Simulation Study of Flow and Heat Transfer in Swirling and Non-Swirling Impinging Jets on a Semi-Cylinder Concave Target' Applied Sciences, Vol. 11, No. 7167.
[10]. Spalart, P. R., Jou, W. H., Strelets, M., Allmaras, S. R. (1997) 'Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach'. In: Advances in DNS/LES, Greyden Press, Pp.137-147.
[11]. Xu, Y. L., Sun, D. K., Ko, J. M., Lin, J. H. (2000) 'Fully coupled buffeting analysis of Tsing Ma suspension bridge' Journal of Wind Engineering and Industrial Aerodynamics, Vol. 85(1), Pp. 97–117.
[12]. Zhang, Z.B. and Xu, F.Y. (2020) 'Spanwise length and mesh resolution effects on simulated flow around a 5:1 rectangular cylinder' Journal of Wind Engineering and Industrial Aerodynamics, Vol. 202, 104186.