

3-D LES numerical simulation researches on wind flows interfered by rough waves

Fuyou Xu^{1*}, Xuyong Ying², Zhe Zhang³, Wenjuan Lou⁴

^{1,2,3}School of Civil Engineering, Dalian University of Technology, Dalian China 116024

⁴School of Architecture Engineering, Zhejiang University, Hangzhou China 310012

Corresponding author E-mail: fuyouxu@hotmail.com

ABSTRACT: Under the actions strong wind over the sea, rough waves may roar and interfere wind flows. The flows may become extreme complicated and fluctuated and incur unusual response for structures in this region. The strong interfered performances of wind flows near sea surface by rough waves are investigated by using the 3-D LES (Large-Eddy-Simulation) numerical simulation technique. The research results indicate that the numerical simulation results are basically consistent with those by wind tunnel experiments. The difference can not be avoided and the major causes are presented. With the interference of wave, the wind flows fluctuated intensely, and the unsteady characteristics is very significant, which largely dependent upon the wave shapes. The apparent influential region by wave ranges from the wave crest to 15 times of wave height downwind horizontally and 4 times of wave height upward vertically.

KEYWORDS: 3-D LES; sea wave; aerodynamic interference; numerical simulation

1. INTRODUCTION

On the extreme (such as once every hundred years) climatic conditions, under the actions of strong wind, wild sea waves (crest may be higher than 8m, or even more) may appear. The combined effects from wind, waves and currents will have an extremely negative impact on bridges and/or offshore platforms. Wave velocity is generally lower than the wind speed, thus the wave can be regarded as an obstacle, and it will influence the downwind flow field characteristics, especially for the roaring wave, its impact is very significant, may lead to very complex wind flow field. The turbulence intensity and wind angle of attack, etc. may be completely different with the normal wind field. Therefore, unusual impact from the interfered wind flows may construct the terrible threat to the artificial structures. However, only a few researches have been carried out in references and/or documents [1-4]. Exhaustive investigations on this issue should be further conducted

In order to study the influence of wind interfered by rough waves on bridge and/or offshore platform, it is necessary to investigate the characteristics of wind flow downwind of the wave beforehand. Wind tunnel test and numerical simulation are both applicable, and the latter one is expounded and the results are compared with by the former one in this study. The properties are analyzed and summarized, and some conclusions are drawn.

2. WAVE SHAPE

For actual sea waves, enormous forms can be observed and it is impossible to investigate all of them. In this study, only the extreme cases are concerned. The waves with 8m height are considered as the examples. For such fairly rough waves, the shape is relatively simple and the distance between two wave peaks is very long (maybe longer than 500m). So for the high waves, it is rarely to observe several accompanied peaks, and one single wave may usually emerge. For the sea waves, two simplified shapes are selected and quantified with scale of 1:40. Certainly, the influence of more types of wave shapes and even groups of waves on wind flows can be studied with similar methods.

Wave shape type 1: certain function curve, $x=\exp(0.25*z)$, $y=(\text{normpdf}(0,4,x))$, $\text{plot}(x,y)$ ($3 \leq x \leq 9.9$). The shape can be achieved through stretching or panning the curve in both horizontal and vertical coordinates, and the width and height are 30cm and 20cm, respectively.

$$y = \frac{689.6552}{(0.3255x+4)\sqrt{2\pi}} e^{\frac{-8}{(0.3255x+4)^2}} - 21.7241 \quad (5.7849 \leq x \leq 24.2151)$$

Wave shape type 2: The probabilistic density function of standard normal distribution, and the interval of X axis is $[-2.5, 2.5]$. The shape can be realized through the curve, and the width and height are 60cm and 20cm, respectively.

$$y = 50 \frac{1}{\sqrt{2\pi}} e^{\frac{-x^2}{288}} \quad (-30 \leq x \leq 30)$$

The graphs of two wave shapes are plotted in Fig.1.

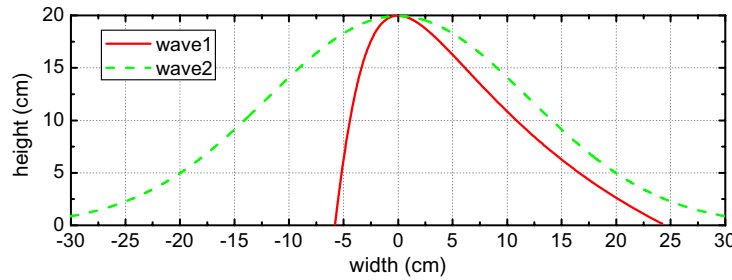
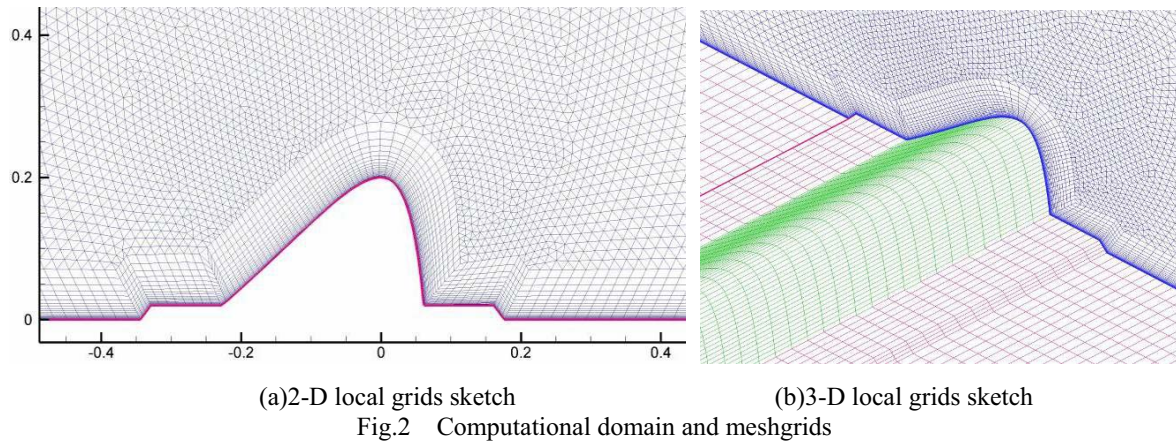


Fig.1 Wave shape graphs

3. NUMERICAL SIMULATION MODEL

3.1 Geometric modeling and meshing

The geometric ratio is 1:40 for numerical modeling, and the three-dimensional computational domain is taken as 26m (length) $\times$ 2.5m (height) $\times$ 1m (Width). The horizontal distance between wave crest and inlet is 8m, and the outlet is 18m apart from the crest. The model is 0.2m high. The computational domain for Wave 1 meshes is shown in Fig. 2, and the triangle unstructured mesh is adopted. Boundary layer mesh is used for Near-wall region meshing to ensure the requirements of stable numerical solving on the wall boundary layer. The boundary layer height of the first grid is 0.0002m. Owing to the fact of two-dimensional segmental rigid computational model, the statistical average of the flow maybe assumed in the spanwise direction, and it is divided into 30 grids. The total volume mesh number is about 1,230,750. The sketch maps of 2-D and 3-D local computational domain and meshgrids are shown in Fig.2 For Wave 2, it is similar to that of wave 1 using the same modeling approaches, and the total volume mesh number is about 1,619,430.



3.2 Computational parameters

The 3-dimensional Large Eddy Simulation model is utilized for numerical analysis. The spatial discretization can be realized using central difference, and the velocity pressure coupling may be solved using the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm based on decoupling idea. The convergence criterion for Iterative calculation is set to be the non-dimensional root mean square residual decays below 10^{-4} .

The boundary conditions for the numerical model are respectively set as follows: the inlet flow velocity is smoothly 10m/s (It need to be mentioned that the Reynolds number effect may exist, which is not considered for limitation of the context length); the outlet flows are completely developed; the top surface is free boundary; the side surface is symmetric boundary; the model surface and sea level is no-slipping wall boundary, and the standard wall function is adopted; the time interval of calculation step is 0.005s; when the flow field become stable, the calculation time is 15s, and the steps are 3000.

4 RESULTS AND ANALYSIS

The contours of different parameters are plotted in Fig.4-Fig.7, respectively.

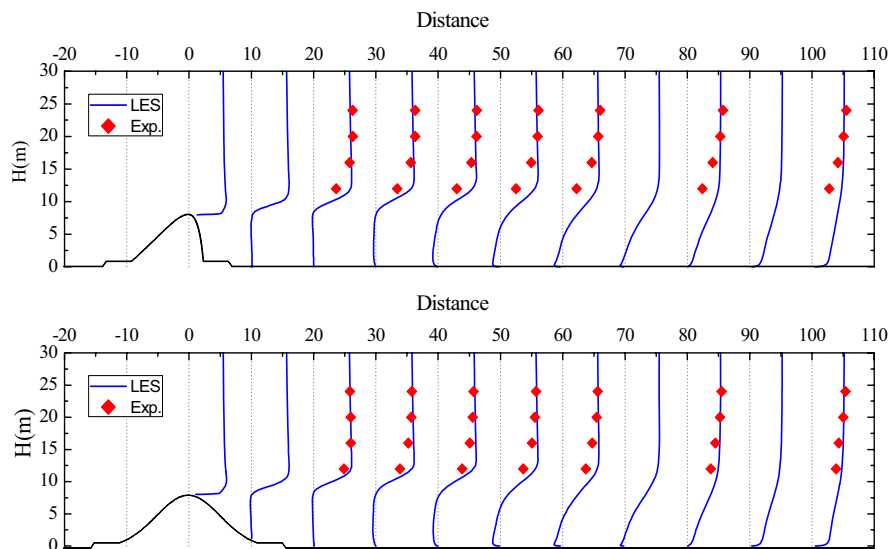
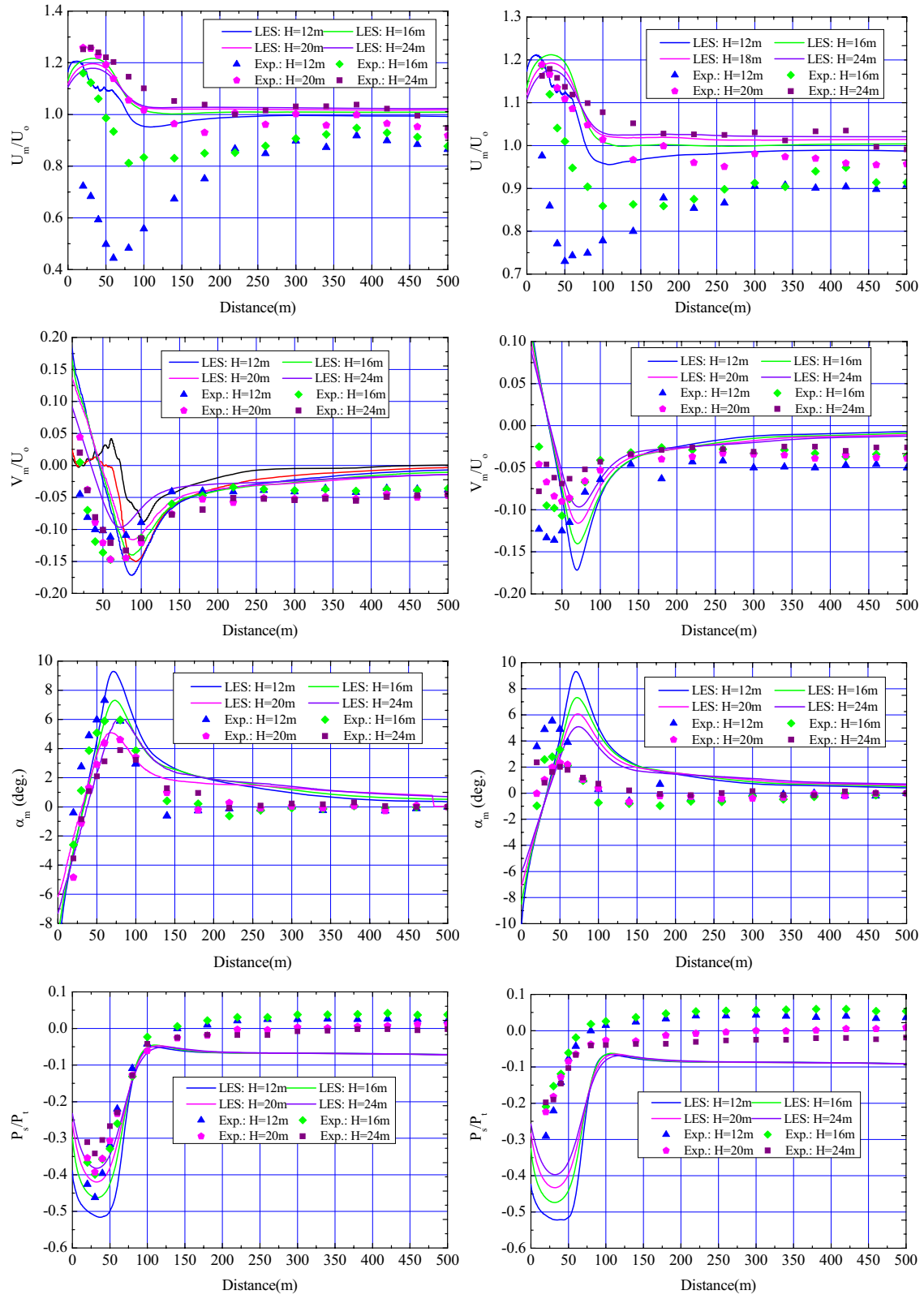
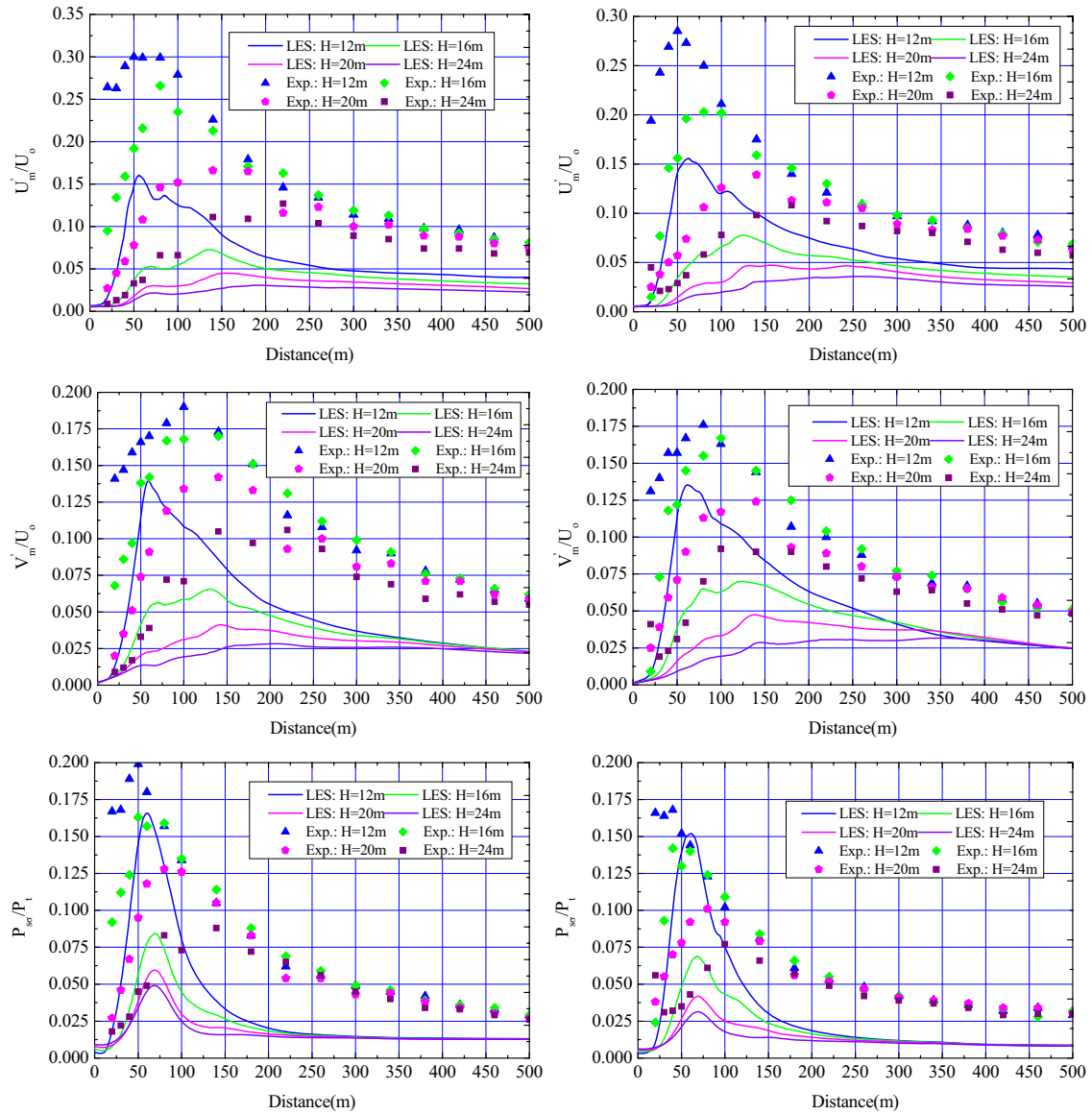


Fig.3 Horizontal mean velocity profiles

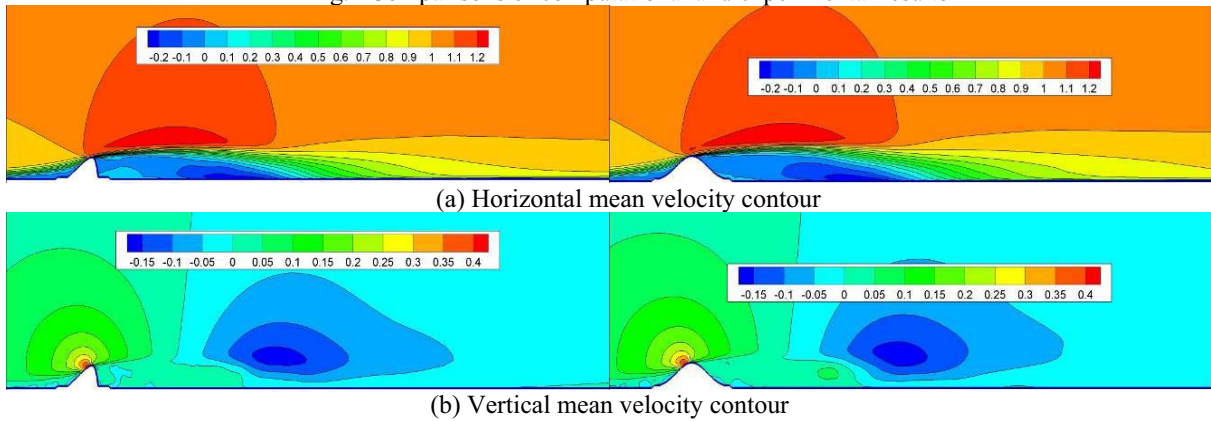


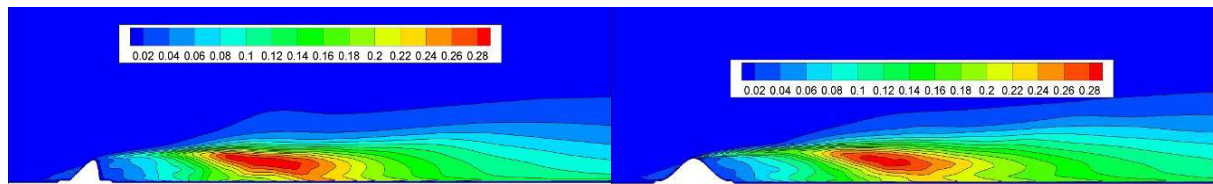
(a) Mean values



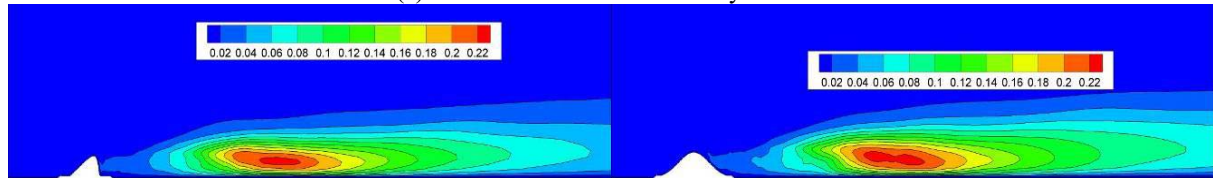
(b) Std values

Wave 1 Wave 2
Fig.4 Comparisons of computational and experimental results

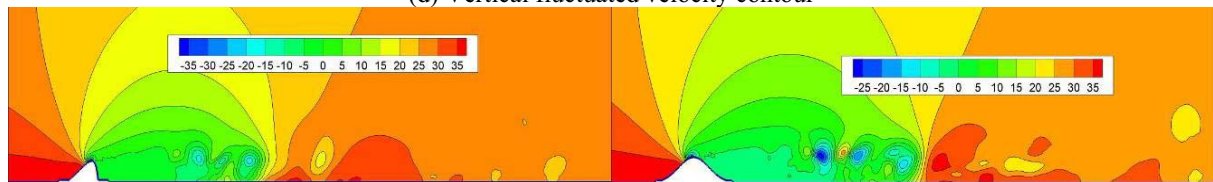




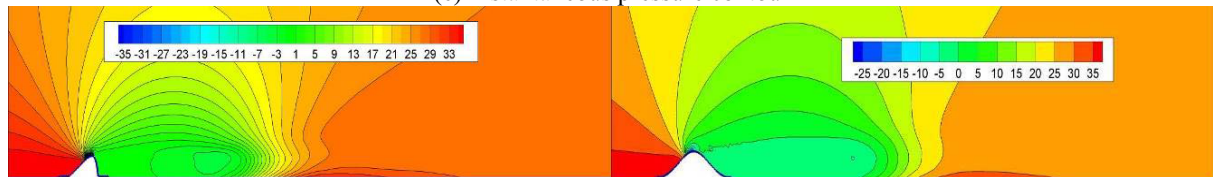
(c) Horizontal fluctuated velocity contour



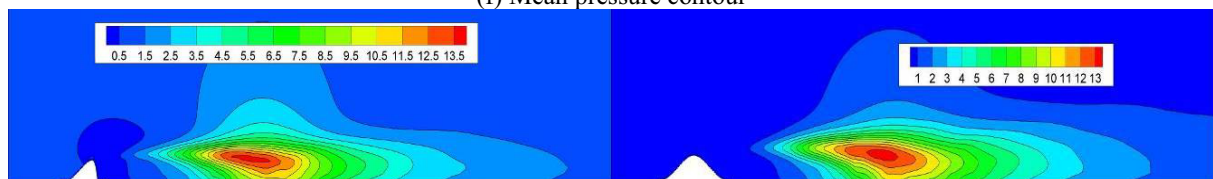
(d) Vertical fluctuated velocity contour



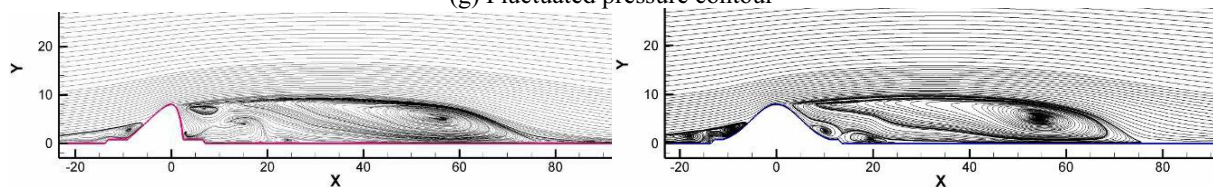
(e) Instantaneous pressure contour



(f) Mean pressure contour

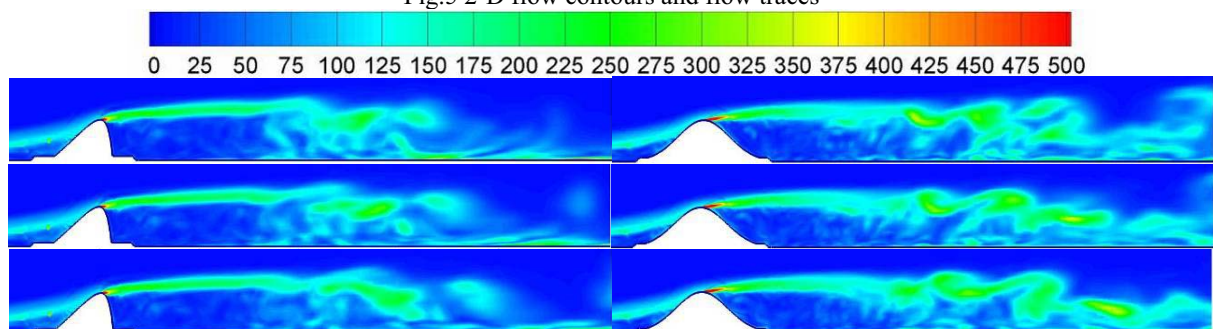


(g) Fluctuated pressure contour



(h) Mean flow traces

Fig.5 2-D flow contours and flow traces



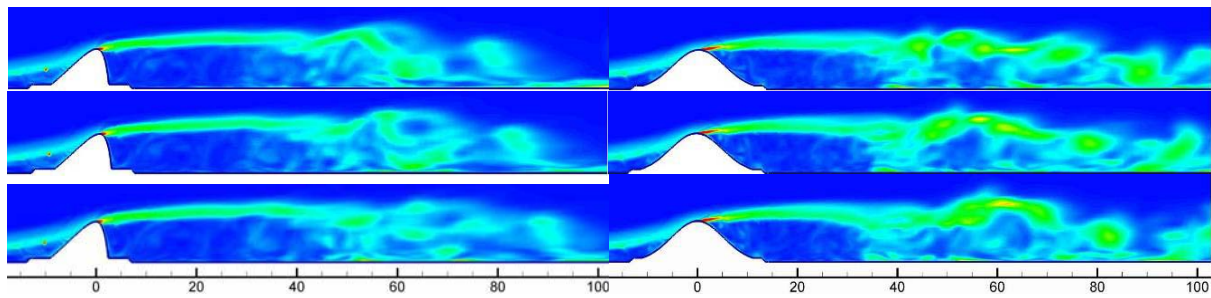


Fig. 6 Instantaneous 2-D vorticity maps at different time (Time interval: 0.025s)

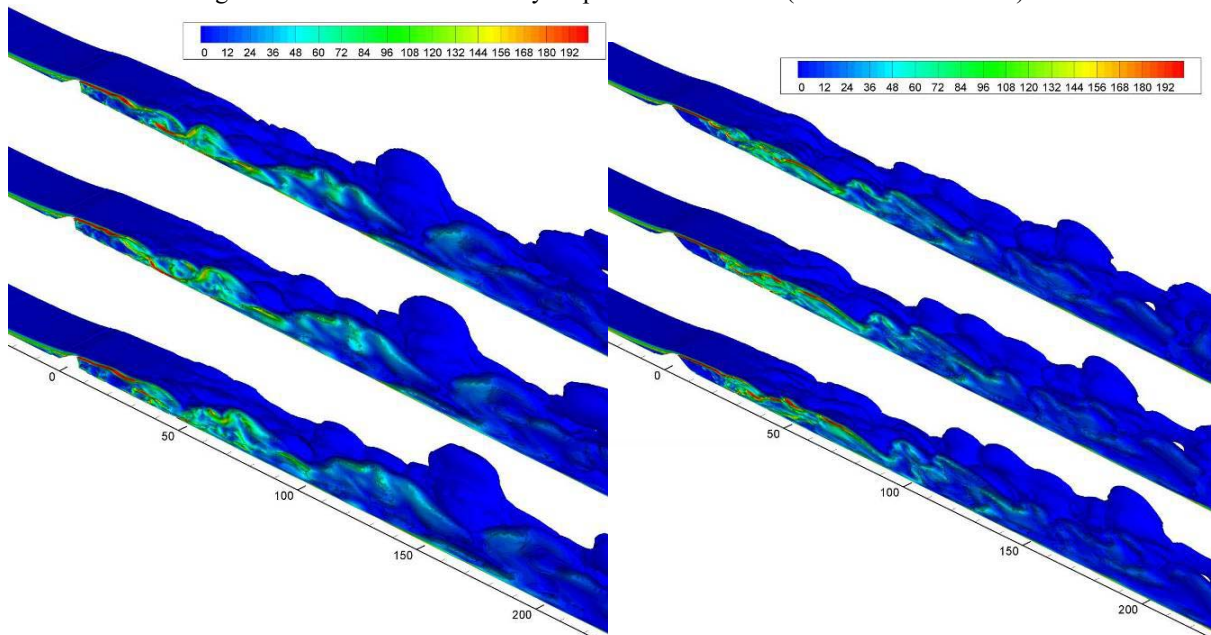


Fig. 7 Instantaneous 3-D vorticity maps at different time (Time interval: 0.05s)

From Fig.3, it can be seen that:

- (1) In front of sea wave, in the region of 3-4m upside of sea surface level, the wind velocity behaves obvious difference, and the major causes may include two aspects: on one hand, air is viscous and one boundary layer is inevitable in which the velocity increase from zero to be 10m/s; on the other hand, owing to the obstacle effect from wave, the velocity near the sea surface is relatively lower. For the influence extent from the above two aspects, it can be analyzed by means of the following two techniques: no wave, i.e. remove the wave; the wave is set at the entrance of flows. In the region outside of 40m in front of wave, the wave effect is negligible, and the gradient flows property is similar, which may be mainly caused by the first reason. For the region behind wave and the distance maybe about 30m, the mean velocity is almost zero when the height is lower than wave crest. It may be considered as a “stagnant water area”. When the mean velocity is negative, it may be regarded as a “backwater area”. Such backwater area can be detected in the region from 35-70m behind the wave. Behind this area, the boundary layer thickness increase owing to the obstacle effect of wave.
- (2) For the two heights of 30cm and 40cm, the numerical results are higher than those of experimental ones. For the two heights of 50cm and 60cm, the numerical and experimental results are almost identical. The reasons maybe provided as follows. The wind tunnel laboratory walls are not absolutely smooth, and the testing points are apart from the flow inlet entrance. Therefore, thicker boundary layer appears. Whereas, the smooth wall modeling is utilized for CFD numerical simulation. Thus, the results difference near the wall region is comparatively

apparent. In addition, differences between the wind tunnel test model and numerical simulation model can not be avoided. The testing model may be relatively rough than the computational ones. Such influence may become weaker with the distance farther from the wave, and the results tend to be more close to each other.

From Fig.4, it can be observed that:

(1) For the mean wind velocity of horizontal flow, the numerical simulation results are higher than those of wind tunnel test ones. The closer to wave and the closer to sea surface, the apparent difference. In such regions, the flow fields are more complicated, and the result accuracy is relatively unsatisfactory for both wind tunnel test and numerical simulation.

(2) For the mean wind velocity of vertical flow, in a very wide region behind the wave, the value is negative. The results difference between the numerical computation and wind tunnel test is very obvious in the region that near the wave. The vertical velocity is almost negligible when the distance is 150m far from the wave. The numerical results trend of two types of waves is almost similar.

(3) For the mean angle of incidence, the results from numerical computation and wind tunnel test are relatively close for wave 2. The numerical simulation results are comparatively higher. For wave 3, the results from two methods are distinct. The major cause is the complex flow fields and extreme fluctuation near the wave, and the accuracy is difficult to be ensured for both wind tunnel test and numerical simulation.

(4) For the mean static pressure, the numerical simulation results are lower than those of experimental ones, and the difference is more obvious for wave 3. The systematic error may happen for testing apparatus and/or numerical calculation.

(5) For the standard deviation of all parameters, the numerical simulation results are lower than those of experimental ones. The major cause may include the following two aspects. The experimental instruments and signals are inevitably interfered and contaminated by a large variety of noises. The testing conditions (eg. roughness, turbulence, and so on) can not be so realistic as the numerical simulation. In addition, the wind tunnel width is 3m, while the wave model length is 2.5m, and the more obvious 3-D around flow effect may be incurred.

From Fig.5-Fig.7, it can be detected that:

Except the region that close to wave, the interfered wind flows by two waves are fundamentally similar in other regions. All calculation conditions are basically identical. Although the wave shape is different, the wave height is the same and the configurations are smooth and a bit similar. Two complicated flow fields can be detected: one is the “stagnant water area” that locates in the interval from wave crest to 30m downwind; the other is the “backwater area” that locates in the interval from 30 to 80m downwind of the crest, and the height is within 1.5 times of the wave height. For bridge deck, the height may be higher than this area. With the interference effects of wave, the angle of incidence may deviate from zero to the formidable values, which may bring a challenging threat to the safety of bridge.

5 CONCLUDING REMARKS

The strong interfered performances of wind flows near sea surface by rough waves are investigated by using the 3-D LES numerical simulation technique. The research results indicate that the numerical simulation results are basically consistent with those by wind tunnel experiments. The difference can not be avoided and the major causes are presented. With the interference of wave, the wind flows fluctuated intensely, and the unsteady characteristics is very significant, which largely dependent upon the wave shapes. The apparent influential region by wave ranges from the wave crest to 15 times of wave height downwind horizontally and 4 times

of wave height upward vertically. So if the bridge deck and/or offshore platform are located such a region, adverse wind effects may incur and more comprehensive researches on wind-resistant performance of bridge and/or offshore platform are strongly recommended to be conducted.

REFERENCES

- 1 Sun Luzhong, Yin Hongbo, Li Xia, et al., Analysis of wind flowing around trestle bridge deck based on large eddy simulation method. *Port Engineering Technology*, 6(2008) 5-9.
- 2 LI Xia, SUN Lu-zhong, YIN Hong-bo, et al., Displacement response of flexible overhead trestle bridge subjected to coupling of wind and wave, *Journal of Vibration and Shock*, 30(2011) 117-121.
- 3 LI Xia, SUN Lu-zhong, YIN Hong-bo, et al., Influence of wind speed and wave height on displacement response of single span overhead trestle bridge, *Journal of PLA University of Science and Technology(Natural Science Edition)*, 12(2011) 172-177.
- 4 XIE Xing-kun, YIN Hong-bo, SUN Lu-zhong, et al., Detached eddy simulation of wind flow around trestle bridge deck, *Journal of PLA University of Science and Technology(Natural Science Edition)*, 12(2011) 273-278.