Numerical and experimental assessment of frequencies and amplitudes when swirling excitation of bending vibrations of construction structures

Nikita Ageev, Olga Poddaeva, Anastasia Fedosova and Oleg Egorychev

Moscow State University of Civil Engineering, Yaroslavskoe shosse, 26, Moscow, Russia

E-mail: mgsu-misi@list.ru

Abstract. In the paper a numerical simulation of the unsteady flow around a quasi-two-dimensional stationary model based on RANS methods that are closed by the SST turbulence model was performed. After that, the amplitudes and frequencies of the aerodynamic forces were estimated, then the speed of the appearance of the resonance of the structure was searched and the amplitudes were calculated. A comparison is made with the data of experimental modeling. To eliminate the contradiction between the experimental data and the results of numerical modeling using RANS methods, oscillations were also considered within the framework of the vortex-resolving approaches LES and DES. It is shown that the technique of modeling a quasi-two-dimensional compartment allows one to quickly obtain a result that qualitatively and in order of magnitude corresponds to experimental data, but cannot provide a quantitative result. To ensure quantitative estimates, it is necessary to carry out physical modeling in wind tunnels or design studies with high spatial and temporal resolution based on modern eddy-resolving methods of computational aerodynamics.

1. Introduction

At present, numerical and experimental methods are used to model wind effects on aeroelastic structures. Experimental are considered more reliable, since they do not introduce errors associated with the discretization of time and space. However, numerical methods have ample opportunities for visualization, as well as the ability to model structures of natural dimensions.

When using numerical methods, it is necessary to know the capabilities of the methods. This requires methodological research and verification. In the present work, methodological computational studies are carried out and their results are compared with experimental data.

When modeling the flow in some cases, we can assume that the movement of the object does not significantly affect the aerodynamic characteristics. This can significantly simplify the formulation of the problem of computational research, since otherwise it is necessary to solve the related problem: at each time step, calculate the flow field, evaluate the dynamics of the object and change the calculation model.
2. Research

2.1. Formulation of the problem
In the present work, given technique is applied to a bridge, the cross section of which is depicted in Figure 1.

![Figure 1. The cross section of the bridge](image)

2.2. Numerical simulation
A numerical solution of the RANS equations with the SST turbulence model are carried out. The boundary condition on the surface of the bridge is no slip wall. The bridge has stiffeners lying in planes perpendicular to the axis of the bridge. As a result, the cross section of the bridge is variable.

Two mesh options were built: option 1 for sections of the bridge on its main part and option 2 for sections in the region of stiffeners. The options are shown in Figure 2. The grid is constructed in the computational domain in which the distance from the front edge of the bridge model to the input boundary is 5b (the b-dimension of the profile chord, in this case, from the front edge of the front fairing beak to the rear edge of the rear fairing beak), from the back to the output 15b, from the top to the top and from bottom to bottom 10b. The thickness of the region is 1 meter in full scale. The flow velocity is set at the inlet boundary and is 20 m / s.

![Figure 2. The options of meshing.](image)

The calculation grid for both options is made according to the C-H topology. The outer part of the grid has a C-topology, and the inner, located in the immediate vicinity of the object of study, has an H-topology. The characteristic cell size in the field of H-topology is 70 mm. The boundaries of the H-
topology region in all directions, except for the downstream direction, are 1b from the object. Downstream, the H-topology region adjoins the output boundary. The calculation grid for option 1 totals 1,239,068 nodes, for option 2 - 1,231,590 nodes. A sufficiently large number of nodes is associated with a small cell size near the object and a small growth rate of the cell size in the wake of the model in the flow direction - 1.015.

2.3. Results
A comparison of the flow patterns at zero angle of attack is shown in Figure 3. It can be seen that for both variants of the cross sections, the flow patterns almost coincide. In the future, only the results obtained for option 1 will be considered. Upon receipt of the results, the licensed ANSYS Fluent software package was used. We used a non-stationary segregated Pressure-based solver with an implicit computational scheme. We also conducted test launches with a coupled solver, which showed that for obtaining similar results using coupled solver it takes 7 times more time. The solver settings are as follows: Simple algorithm, calculation of the gradient according to the Least Squares Cell-Based method, second-order scheme for all equations, except for the turbulence model. For equations of the turbulence model was used a first-order counterflow scheme. The time step was 0.01 second; one step could account for up to 20 internal iterations.

![Figure 3. Comparison of instant flow patterns for option 1 and option 2 of bridge sections.](image)

Next, section 1 option was investigated at angles of attack of -10, -5, -3, 0, +3 and +5 degrees. The flow fields are shown in Figure 4. At angles of attack of -10, -5, -3, 0, and +5 degrees, a periodic vortex gathering is clearly visible. In this case, the characteristic wavelength at an angle of attack of +5 degrees is significantly greater than at other angles of attack. At an angle of attack of +3 degrees, the flow around is completely stationary. This is also confirmed by the time dependences of the lift coefficient shown in Figure 5. A total of 100 seconds of real time (full-scale, speed of 20 m/s) was simulated, which required 10 thousand iterations. On a server equipped with two Intel Xeon processors (36 cores in total) this requires about 8 hours of time to simulate one flow around at one angle of attack, which is acceptable for preliminary computational studies. According to the schedule, you can also see that the mode is reached in about 50 seconds. To perform the Fourier transform, the implementations of the last 50 seconds of simulation was taken.
Figure 4. Fields of flow around the model of the bridge section at different angles of wind attack.
Figure 5. Temporal dependences of the lift coefficients obtained by numerically solving the non-stationary RANS equations in a quasi-two-dimensional formulation.

Performed at the NRU MGSU experimental studies show that when the angle of attack is +3 degrees, fluctuations in the cut-off model of the bridge are observed. In addition, non-stationary results obtained from an aerodynamic balance also show fluctuations in the magnitude of the aerodynamic force acting on the model. Numerical studies of the quasi-two-dimensional model in the framework of the RANS equations did not show oscillations of the driving aerodynamic force at a given angle of attack. To eliminate this contradiction, oscillations within the framework of the vortex-resolving approaches LES and DES are also considered. In these approaches, for large vortices (the size of which exceeds the size of the cell), Reynolds averaging (over time) is not introduced and their direct numerical simulation takes place. Vortices smaller than a cell are modeled by homogeneous isotropic turbulence. The corresponding coefficient of turbulent viscosity is calculated by semi-empirical relations (for example, the Smagorinsky model). The DES approach differs from the LES approach in that the near-wall layers use the SST turbulence model. This allows a more correct description of the flow in the boundary layer. Moreover, turbulence is a substantially three-dimensional flow (the so-called turbulent cascade). For this reason, the flow was simulated near the simplified cut-off model (Figure 6). The grid was extruded in the direction of the axis of the bridge into 20 cells each 200 mm in size. The mesh for LES has 4,240,480 nodes, the mesh for DES has 5,262,220 nodes.

Modeling by LES and DES methods required to significantly reduce the time step even taking into account some roughening of the computational grid in comparison with RANS. A time step of 0.004 seconds was used, which is 2.5 times less than for RANS. The use of DES also caused additional difficulties with the launch of the solver, which were overcome by using the SIMPLEC algorithm with correction for cell slanting.
Figure 6. Calculation grids for modeling by LES method (above) and DES (bottom).

The visualization results are shown in Figure 7. A significant difference and the formation of a cascade of turbulent vortices are visible.

Figure 7. Comparison of visualization of calculation results at an angle of attack of +3 degrees by various methods.

Figures 8–15 show the time dependences, amplitude – frequency characteristics, and estimates of the velocity and amplitude of the vortex resonance. Estimates of the resonance velocity for LES and DES are not presented due to the absence of a dominant vortex convergence frequency in the spectrum.
Figure 8. Temporal dependences of the lift coefficient.

Figure 9. Amplitude spectra of the lift coefficient.

Figure 10. Temporal dependences of the lift obtained by the LES method.
Figure 11. Lift Amplitude Spectra (LES).

Figure 12. Amplitude Spectra of Lift coefficient (RANS).

Figure 13. Wind resonance velocity value depending on the wind attack angle.
Estimates of the amplitude of oscillations during resonance are based on the well-known formula for a one-dimensional linear oscillator, in which some effective mass acts as the mass (in these calculations, the linear mass of the bridge divided by the root of two is taken), and the amplitude is the half-amplitude of the lift oscillations. It can be seen that the velocity estimate gives results close to experimental values with an error of 5-15% (except for the angle of attack of +3 degrees, at which a significant decrease in amplitude was found in experimental studies, and there are no driving forces in the calculated vibrations).

Estimation of the amplitude of oscillations gives the correct order of magnitude, but is unsuitable for quantitative values. The values of the average values of the lift coefficient for modeling the flow around the bridge without detailed resolution of the boundary layer as a whole give a qualitatively experimental, but quantitatively unsatisfactory result. The resolution of the boundary layer requires considerable calculation time and/or the use of large computing power (about 2 weeks per point on a 36-core settlement server).
3. Conclusions
For the same computational grid, setting a different type of boundary conditions weakly affects the simulation result. A boundary condition of the no slip wall type spreads energy over a larger spectral region and, in addition, slightly increases the height of the main peak of the amplitude in frequency while simultaneously reducing the height of the second peak and decreasing the value of the bridge average time-lifting coefficient. A significantly larger impact on the results is exerted by the computational grid.

The influence of the Reynolds number on the frequency and amplitude is quite small. We can say that with a decrease in the Reynolds number, the frequency and amplitude of the oscillations increase slightly.

The screen has a damping effect. This must be borne in mind when designing and testing, and if there is a likelihood of a significant decrease in the water level in the pond, it is necessary to conduct tests with or without a screen.

References
[1] Yao, Z., Zhang, N., & Xia, H. (2018). Numerical simulation on aerodynamic performance of vehicle-bridge system subjected to cross wind. Paper presented at the ICRT 2017: Railway Development, Operations, and Maintenance - Proceedings of the 1st International Conference on Rail Transportation 2017., 2017-July 861-870.
[2] Ibrahim Tolba, K., & Morgenthal, G. (2017). Pseudo three-dimensional simulation of aeroelastic response to turbulent wind using vortex particle methods. Journal of Fluids and Structures, 72, 1-24. doi:10.1016/j.jfluidstructs.2017.04.001
[3] Zhan, Q., Zhou, Z., & Ge, Y. (2017). Numerical and experimental study of the mechanism of torsional flutter for open-cross-section composite beams. Tongji Daxue Xuebao/Journal of Tongji University, 45(4), 466-471 and 581. doi:10.11908/j.issn.0253-374x.2017.04.002
[4] He, X., Zhou, L., Chen, Z., Jing, H., Zou, Y., & Wu, T. (2019). Effect of wind barriers on the flow field and aerodynamic forces of a train–bridge system. Proceedings of the Institution of Mechanical Engineers, Part F: Journal of Rail and Rapid Transit, 233(3), 283-297. doi:10.1177/0954409718793220
[5] Jeong, W., Liu, S., Jakobsen, J. B., & Ong, M. C. (2019). Unsteady RANS simulations of flow around a twin-box bridge girder cross section. Energies, 12(14) doi:10.3390/en12142670
[6] Yao, Z. -., Zhang, N., Xia, H., & Li, X. - (2018). Study on aerodynamic performance of three-dimensional train-bridge system based on overset mesh. Gongcheng Lixue/Engineering Mechanics, 35(2), 38-46. doi:10.6052/j.issn.1000-4750.2017.03.0254
[7] de Miranda, S., Patrano, L., Ricci, M., & Ubertini, F. (2015). Numerical study of a twin box bridge deck with increasing gap ratio by using RANS and LES approaches. Engineering Structures, 99, 546-558. doi:10.1016/j.engstruct.2015.05.017
[8] Diana G. et al. Wind tunnel tests and numerical approach for long span bridges: the Messina bridge //Journal of Wind Engineering and Industrial Aerodynamics. – 2013. – T. 122. – C. 38-49.
[9] Diana, G., Bruni, S., Rocchi, D. (2005). A numerical and experimental investigation on aerodynamic non linearities in bridge response to turbulent wind. In: Proceedings of the EACWE4. Prague.

Acknowledgments
This work was financially supported by the Ministry of Science and Higher Education of the Russian Federation (Project: Theoretical and experimental design of new composite materials to ensure safety during the operation of buildings and structures under conditions of technogenic and biogenic threats #FSWG-2020-0007).