Investigation of the influence of various factors on the results of a calculation-experimental assessment of frequencies and amplitudes during vortex excitation of bending vibrations of building structures

O I Poddaeva, N D Ageev and A N Fedosova

Moscow State University of Civil Engineering, Yaroslavskoe shosse, 26, Moscow, 129337, Russia

Abstract. In the paper influence of different factors on the results of a calculation for numerical simulation of span aerodynamics was investigated. A family of structured multi-block hexahedral computational grids was built. The technology of meshing is given. Numerical simulation based on RANS methods for each mesh was carried out. A comparison with experimental data was performed. A significantly impact on the results is exerted by the computational grid. Reynolds effect and screen effect were investigated.

1. Introduction

As a rule, building structures are high-drag body with sharp edges. For such bodies, the position of separation of the boundary layer is rigidly fixed. In this regard, among researchers there is a view of the non-requirement of a detailed modeling of the boundary layer for the correct playback of the main aerodynamic characteristics in accounting studies. However, the answer to this question is not obvious. The presence of a boundary layer may affect to the main flow. For example, it is proposed to use this in wind tunnels to minimize the influence of the walls of the wind tunnel. It is necessary to study the influence of types of boundary conditions on the results of numerical modeling. The modeling of the boundary layer requires both setting the corresponding boundary conditions (such as no slip wall instead of free slip wall) and a computational grid that can correctly resolve all the main scales of the flow near the wall. The influence of the two mentioned factors on the basic aerodynamic characteristics of the bridge span in a quasi-two-dimensional formulation was investigated.

2. Research

2.1. Meshing

A family of structured multiblock computational grids was built (Figure 1), including:
- a grid for modeling flow around a bridge model without a boundary layer — 1,239,068 nodes in a quasi-two-dimensional state (a);
- a grid for modeling the flow around the bridge model with a stream with a boundary layer - 2,584,274 nodes, respectively (b);
a grid for modeling flow around a bridge model with a flow with a boundary layer with doubled in each direction (except normal to the flow plane) number of cells - the total number of cells is increased 4 times - the corresponding number of nodes is 10 322 640 (c).

The methodology for constructing the computational grid (b) is as follows. The block structure of a two-dimensional grid is built. The external block structure has a C-topology. Immediately near the span (at a distance of 1 span width), an H-topology is embedded in it, which directly describes all the details of the span. And in the immediate vicinity of the individual parts of the bridge, O-topologies have been created to describe the boundary layer itself. The grid (a) does not have the corresponding O-topologies. The characteristic size and number of cells in O-topologies is determined as follows: the width of the O-topology is chosen so that it completely captures the entire boundary layer. The thickness of the real boundary layer is estimated by the thickness of the turbulent boundary layer on a flat plate:

\[ \Delta = 0.37 \times \text{Re}^{-0.2}, \]  

where to calculate the number Re and determine the x coordinate, the span width is taken to be 28.4 m, and the flow velocity is 20 m/s. It is also desirable that the external and internal corresponding edge of the O-topology differ by 1.3-1.5 times. All these parameters correspond to an O-topology thickness of 0.2 meters.

The thickness of the first cell corresponds to \( y^+ \) equal to 1. In the case of a particular bridge (SS width 28.4 m) for a flow velocity of 20 m/s, the thickness of the first cell corresponding to \( y^+ = 1 \) is 0.018 mm. The number of cells falling on the smallest simulated structural element should be about 20. In this case, the smallest vertical fencing elements account for 19 nodes.

2.2. Numerical simulation

Numerical modeling on the grids (a) and (b) was carried out in the framework of the RANS equations with closure according to the SST turbulence model. It was not possible to carry out numerical
modeling on the grid (c) using the described methodology (due to instability, the solution loses its stability). It was necessary to change under-relaxation factors: by pressure from 0.3 to 0.25, by density - from 1 to 0.8, by forces - from 1 to 0.8, by momentum - from 0.7 to 0.5, by kinetic energy of turbulent pulsations - from 0.8 to 0.6, according to the characteristic dissipation rate, from 0.8 to 0.6, and according to the turbulent viscosity, from 1 to 0.8. Ultimately, simulation on the grid (c) showed oscillations with a characteristic time of about 16 seconds of the span lift coefficient at an angle of attack of 0 degrees (Figure 2), which does not correspond to the results obtained in the course of experimental studies. Thus, it is shown that excessive fineness of the grid can lead to problems with the stability of the solution, and their elimination, in turn, can lead to unphysical results.

Figure 2. Time dependence of the lift coefficient at a speed of 20 m/s and an angle of attack of 0 degrees for the grid (c)

Of considerable interest are the results obtained using a mesh of type (b) – Figure 3. Moreover, in addition to the influence of the computational grid, it is possible to evaluate the influence of the type of boundary conditions. For correctness, it is necessary to carry out two calculations on the same calculation grids with different types of boundary conditions. This was done. The Fourier transforms of the time dependences of the span lifting force coefficients for boundary conditions such as slip wall and no slip wall are presented. In addition, for comparison, the graph shows the amplitude-frequency characteristics of the oscillations modeled using grid (a).

A significantly larger impact on the results is exerted by the computational grid. Thus, the computational grid (a) gives a completely different value of the main frequency of the driving force: 0.66 Hz versus 0.42 Hz for the grid (b) with a simultaneous increase in amplitude from 0.01362 to 0.06788 (almost 5 times). The results shown in the simulation using the computational grid (b) are significantly closer to reality. It was also necessary to consider the entire range of angles of attack, in which the occurrence of vortex resonance is possible, which was done. In the graphs shown in Figure 4, the results obtained using the grid (b) and the boundary of the type no slip wall are shown in comparison with the results obtained on the grid (a) and the boundary of the type free slip wall.

Figure 3. The amplitude-frequency characteristics of the oscillations obtained on different computational grids under different types of boundary conditions. Dotted line shows grid (a) with free slip wall; red line shows grid (b) with free slip wall; blue line shows grid (b) with no slip wall.
Figure 4. Dependents from attack angle: a) vibration excitation rates; b) oscillation amplitudes; c) average lift factors

Thus, it was shown that grid (b) allows one to obtain results that are much closer to the experimental data. The vortex excitation rates obtained during its use fully correspond to the results given in the
The oscillation amplitudes are also much closer to the experimental data. At zero angle of attack, there is complete agreement, with a change in the angle of attack there is some discrepancy. This may be due to the fact that vortices descending from small structural elements, at zero angle of attack for a long time after the descent, are in the zone of condensation of the computational grid, and at nonzero angles of attack they are carried away from the condensation zones. The values of the average lift coefficients are quite close. However, the derivative of the lift coefficient used for the Den-Hartog criterion in the study of galloping, when applying the grid (b) better matches the experimental data. Apparently, the big difference in the results obtained on different grids is due to the fact that the presence of a relatively small vortex track behind small bridge elements leads to the excitation of an already much more powerful vortex track behind the span. The vortex cascade mechanism, well known in the aerodynamics of turbulent flows, works in the opposite direction. In this case, mesh (a) does not model the vortex track behind small structural elements, and mesh (b) models (Figure 5).

![Figure 5. Fields of flows during modeling using various grids: at the top - grid (a), at the bottom – grid (b)](image)

The visualization of the velocity fields obtained for all angles of attack except zero is shown in Figure 6. It can be seen that at an angle of attack of -10 degrees the flow is pressed to the upper surface of the bridge and the vortices move along it, in the zone of increased resolution of the grid and, in addition, along the grid lines. This leads to a fairly exact match with the results of experimental studies.
2.3. Space considerations
One of the unique possibilities that numerical modeling gives in comparison with the physical one is the possibility of a substantial change in the Reynolds number. This is especially important for estimating errors that occur during physical modeling on a reduced scale. A numerical simulation of the flow around the span model was carried out at the numbers $Re = 1$ and $0.01$ of the full-scale model. The Reynolds number was varied by formally changing the dynamic viscosity coefficient. Flow patterns are shown in Figure 7.

The time dependences of the lift coefficients on time are also shown (Figure 8).
Figure 8. Comparison of the time dependences of the coefficients of lift with full-scale and 100 times smaller Reynolds number.

It can be seen that the average value with a decrease in the Reynolds number by a factor of 100 also decreases slightly (by 0.048). In wind tunnel, the Reynolds number as a rule is about 0.01 full-scale and, therefore, numerical simulations have shown that the average values, characteristic frequencies and amplitudes of the forces obtained in wind tunnel correspond to real ones, despite a large change in the Reynolds number.

2.4. Screen effect

Often, when conducting aerodynamic tests of bridges, a screen model is used. A numerical assessment of the effect of the screen effect is made. To do this, at a distance of 9.4 meters in height from the lower point of the PS model, there is a horizontal surface on which the adhesion condition is specified. This corresponds to the actual position of the bridge above the water level. Flow patterns at zero angle of attack are shown in the figure. Temporal dependencies are shown in the graph.
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
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

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

de Miranda, S., Patruno, 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

**Acknowledgments**

The work was financially supported by the Ministry of Education and Science of the Russian Federation within the framework of the state #7.6075.2017/BCh. All tests were carried out using research equipment of The Head Regional Shared Research Facilities and Unique Scientific Installation Large Research Gradient Wind Tunnel of the Moscow State University of Civil Engineering.