Prediction of high Reynolds number flow over a twin box bridge using LES

WEI Zhigang, GE Yaojun
(SLDRCE of Tongji University, Shanghai 200092, China)

Abstract: Suspension bridge with twin box girder has a better aerodynamic stability and hence has been adopted in many designs for super-long-span bridges. Yet the Reynolds number of aerodynamic research based on wind tunnel section model testing can only reach up to around 600,000. Focusing on Reynolds number effect, several cases with different Reynolds numbers less than 1,000,000 have been modeled first, and the numerical results including drag, lift and moment coefficients have been compared with corresponding experiments conducted in Wind Tunnel in Tongji University. Then with a high Reynolds number up to 4,000,000, flow over a twin box bridge using Large-eddy simulation with wall modeling has been modeled. The drag, lift and moment coefficients, pressure distribution, and wake vortices around the deck surface have been given and discussed. It is observed that LES method with proper wall modeling is able to assess the Reynolds number effect of twin box bridge.

Key Words: Reynolds number effect, twin box bridge, LES, wall modeling, aero-elastic model, drag coefficients

0. Introduction

The objective of this study is to assess the viability and accuracy of large-eddy simulation (LES) with wall modeling for high Reynolds number twin box bridge. It is well known that suspension bridge with twin box girder has much better aerodynamic performances, and are suggested in the future design of many super-long-span bridges (Xiang Haifan, Ge Yaojun, 2005). For instance, Messina Bridge is designed to have a girder section with three boxes. The slot destroys the process of flow’s reattachment and vortices’ formation, then changes the wind-induced periodical forces on the deck, and pronouncedly increases the aerodynamic stability of the bridge.

Since both the reattachment and distribution of pressure is very sensitive to the Reynolds number, therefore the effect of Reynolds number remains to be a hot topic in bridge design. Yet, the Reynolds number is usually high in most practical situations, for bluff bodies with sharp edges, the Reynolds number similitude is often relaxed in current wind engineering practice. It is generally assumed that Reynolds number effects are negligible when a Reynolds number of a few thousand is reached. Typical scale model studies of wind effects are thus conducted at $Re$ in the $10^3$~$10^4$ range while the prototype structure would be in a flow regime at $Re$ greater than $6\times10^6$. However, evidences of $Re$ effects for bluff bodies with sharp edges have been reported, from studies conducted e.g. in a pressurised wind tunnel (Schewe, 2001), or on a large model in a low-speed wind tunnel (Matsuda et al., 2001; Larose et al., 2003) or through full-scale measurements (Hoxey et al., 1998; Kubo et al., 1999).

Before the investigation of complicated structure, people tend to pay close attention to the behavior of some simple bluff bodies in uniform flow that is of the practical importance. The fundamental fluid dynamics problems of bluff bodies such as circular and rectangular cylinders have been examined extensively in both numerical and experimental, especially at low and moderate Reynolds numbers. In recent years, researchers’ attention has turned to the use of LES for studying the turbulent flow around bluff bodies at higher Reynolds numbers. Concerning the numerical techniques, the influence of a finer grid, shorter time step and the larger computational aspect ratio on their results was also studied.

The reason for this increasing focus on LES for the study of the flow around bluff bodies has to do with poor results, when one uses statistical turbulence models. Most probably, this has to do with complicating factors such as a strongly retarded stagnation flow, massive flow separation, streamline curvature, transition from laminar to turbulent flow, recirculation, vortex shedding and, perhaps most important, the existence of inherent three-dimensional flow structures.
A large number of numerical and experimental investigations have been performed for a wide range of Reynolds numbers. Details on the subject of a circular cylinder (spanning a wide range of Reynolds numbers in uniform flow can also be found in the book of Zdravkovich (1997). Moreover, investigation of vortex response of a twin box bridge section at high and low Reynolds numbers was performed by Larson (2007). The effect of Reynolds number is deeply studied by Li Jiawu (2005).

The present study follows in the footsteps of the works cited above. Its focus is to depict $Re$ effects for twin box cross-sectional shapes used in civil engineering structures and for which $Re$ similitude would normally be relaxed. The ultimate goal is to provide a good description of the influence of $Re$ on the fluid-structure interaction. It would then be possible to predict when effects are likely and to evaluate the error margin of testing at low $Re$ when unavoidable.

The format is organized in the following sequence. After introducing our motivation and objective, the numerical method LES with its wall modeling considered is described in Section 1. The experiments conducted in wind tunnel are stated in Section 2. Results and discussion are given in Section 3. Finally, a conclusion and acknowledgement are given.

1. Numerical method and procedure

1.1 Governing equations and Dynamic eddy viscosity model (Described in the full paper)

The subgrid-scale stress (SGS) tensor is modeled using the dynamic SGS model (Germano et al. 1991; Lilly 1992).

1.2 Wall modeling

The LES calculation is sensitive to the modeling of the near deck wall region, because the separation depends on the details of the attached boundary layer whether laminar or turbulent (in the present conditions laminar). In order to avoid the resolution of the thin boundary layers, one can use wall functions. However, this is a delicate task when the separation point is not fixed by the geometry and is far from being settled. For the present computations, the non-slip condition was applied at the cylinder surface as a local refinement technique was used to obtain high resolution in the near wall region as explained below. The van-Driest damping function should be used together with the Smagorinsky model in order to reproduce the correct asymptotic behaviour of the sgs viscosity close to the wall.

1.3 Computational details

The incompressible finite volume method, based on a non-staggered grid arrangement, is used to discretize the governing equations. All spatial terms in the momentum equations are discretized using the second-order central differencing scheme (CDS). While the second-order accurate Crank-Nicolson method is employed to advance the terms in time, with the exception of the pressure term, which is treated fully implicitly, i.e. it was evaluated at the new time step, instead of being split into two terms, one on the old and one on the new time step. The PISO algorithm is used to deal with the pressure-velocity coupling between the momentum and the continuity equations.

The no-slip boundary conditions are assigned on the surfaces of the decks, as follows:

1) Inflow boundary condition: $u = U_\infty$ and $v = 0$;
2) Outflow boundary condition: $p=0$;
3) The upper and the lower, front, and back boundary conditions: $u = 0$ and $v = 0$;
4) The initial velocities in the flow domain are specified as $u = 0$ and $v = 0$ at $t = 0$.

In case of unsteady flow calculations, suitable time steps and the modified Crank-Nicolson scheme are employed. The non-dimensional time step is selected in order to avoid the nonphysical oscillations and obtain a smooth time-dependent solution. The sketch of computational region is shown in Fig.1. The Prototype of twin box is shown in Fig.2. The detailed case description is listed in Table.1.

2. Experiments set up (Described in the full paper)

3. Results and discussion

Both time-averaged and unsteady data are discussed. The pressure-tap data provide a detailed understanding of the unsteady flow, including vortex shedding, around the cylinder in different flow
regimes. The vortex existed in the slot, as can be seen in Fig.3, varies its shape and position as Re ranging from 400~400000.

![Fig.1 Sketch of computational region](image1)

![Fig.2 Prototype of Numerical Simulation (Box girder sections of Xihoumen Bridge)](image2)

<table>
<thead>
<tr>
<th>Case</th>
<th>(M_{\text{total}})</th>
<th>(Re)</th>
<th>Turbulence Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>96400</td>
<td>40</td>
<td>DS</td>
</tr>
<tr>
<td>B</td>
<td>96400</td>
<td>400</td>
<td>DS</td>
</tr>
<tr>
<td>C</td>
<td>96400</td>
<td>4,000</td>
<td>DS</td>
</tr>
<tr>
<td>D</td>
<td>96400</td>
<td>40,000</td>
<td>DS</td>
</tr>
<tr>
<td>E</td>
<td>289200</td>
<td>400,000</td>
<td>DS</td>
</tr>
<tr>
<td>F</td>
<td>289200</td>
<td>4,000,000</td>
<td>DS</td>
</tr>
</tbody>
</table>

\(M_{\text{total}}\) is the total number of cells, DS stands for Dynamic Smagorinsky

4. **Concluding Remarks**

A bold numerical experiment has been carried out to compute the flow around a twin box bridge at high Reynolds numbers up to 4 million using LES. The simulation is made possible by the use of a wall-layer model, which alleviates the near-wall grid resolution requirements. Preliminary results are promising in the sense that they correctly predict the delayed boundary-layer separation, and the reduction of drag coefficients is consistent with wind tunnel testing measurements.
5. Acknowledgement
6. References