Quantification of the Effects of Turbulence in Wind on the Vortex-induced Vibrations

Tajammal ABBAS
PhD candidate
Bauhaus University
Weimar, Germany
tajammal.abbas@uni-weimar.de

Tajammal Abbas, born 1984, received his civil engineering degree from the UET Lahore, Pakistan. He did his MSc from Bauhaus University Weimar, Germany before starting his PhD at the same university. His main area of research is related to wind engineering and aerodynamics.

Guido MORGENTHAL
Professor
Bauhaus University
Weimar, Germany
guido.morgenthal@uni-weimar.de

Dr Guido Morgenthal, born 1975, is Chair Professor for Modelling and Simulation of Structures at the Institute of Structural Engineering.

Summary

The Vortex-induced Vibrations (VIV) for structures are strongly influenced by the turbulence in the wind. Generally, the turbulence in the wind decreases the VIV amplitudes at lock-in. Therefore, it is important to consider it for the accurate prediction of the VIV phenomenon. This study aims at quantifying the effects of turbulence on the VIV. For this purpose, numerical simulations were performed and an approach to generate turbulence was used by placing geometrical shapes in the flow. This is analogous to well accepted laboratory wind tunnel approach. The generated turbulent flow was studied and the flow characteristics were quantified. An example is presented for the case of a circular cylinder considering different flow situations. The study shows that the free stream turbulence has a substantial and systematic effect on cylinder response. The merits and limitations of the approach are also discussed.

Keywords: vortex shedding; vortex-induced vibrations; turbulence; bluff body aerodynamics; numerical simulations; CFD.

1. Introduction

The study of VIV is useful in various engineering applications such as design and analysis of long-span bridges, towers, chimneys and heat exchangers. Wind tunnel tests are the basis for the design such structures under wind actions. However, these tests are expensive and cannot reproduce fully the physics of full-scale problem. The usefulness of numerical methods has been now widely accepted. Numerical simulations using Vortex Particle Method (VPM) are virtual wind tunnels which simulate 2D wind flow over a cross-section and allow the vortex shedding process and its interaction with the structural motion. Traditional numerical solvers operate with the uniform incident flow. In this study, the focus is on the approach by which approximation to the atmospheric boundary layer wind turbulence could be introduced in the numerical flow simulation by placing set of blocks on the upstream side of the main section. Defining some geometric shapes in the flow of the simulation appears to be relatively simple but the main challenge is to get the turbulent flow with the representative flow characteristics.

Numerical investigations were carried out, in this study, using a solver VXFlow based on VPM. Here the aerostatic and aeroelastic simulations were performed on a circular cylinder in uniform and in turbulent flow. Some of the important factors which affect the flow past cylinder and its vibrations are Reynolds number, ratio of vortex shedding frequency to the structural frequency, structural damping and mass of cylinder, surface roughness and level of free-stream turbulence. Turbulent intensity is the ratio of the standard deviation or the root-mean-square of the turbulent velocity fluctuations, at a particular location over a specified period of time, to the mean wind.

2. Numerical Simulation on Circular Cylinder

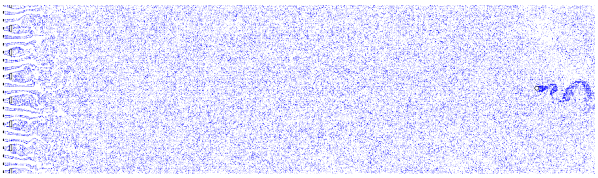
In the actual wind tunnel, turbulence can be introduced by some disturbance upstream of the test

https://doi.org/10.2749/222137813808627514
Distributed by structurae

region or by surface roughness on the floor of the wind tunnel. An analogous approach to the upstream disturbance to create turbulent flow in the wind tunnel was considered in this study for the numerical simulations. A number of rectangular cross-sections in the flow were placed on the upstream side of the main section, i.e. circular cylinder, to generate turbulence in the incident uniform flow. One of the main effects of blocks on the flow regime is the reduction of mean wind speed due to blockage. As a result roughly 10-25% blockage was observed depending on the block spacing from 1-10 times the block depth. The important parameters to be considered here are the mean wind speed, turbulent intensities in longitudinal and lateral direction and the frequency characteristics of flow. Aerostatic simulations involve a fixed circular cylinder defined in the way that no displacement or rotation is allowed in response to body forces. The fundamental difference between fixed and oscillating bluff bodies is that the motion of the cylinder can take control of the oscillation mechanism that leads to VIV.

3. Results

In case of fixed cylinder, the vortex shedding frequency increases with the increase in wind speed for both uniform and turbulent case. Whereas, in spring supported cylinder, the lock-in region is clearly visible for a band of wind speeds for both situations of uniform and turbulent flow. At lock-in, the cylinder oscillates with large amplitudes in uniform flow but amplitudes decrease in turbulent flow. It is also to be noted that the reduction in amplitudes at lock-in due to turbulence depends on the amount of turbulence and frequency characteristics of flow. Up to 75% reduction in maximum amplitudes at lock-in was observed for the cylinder in presented example.



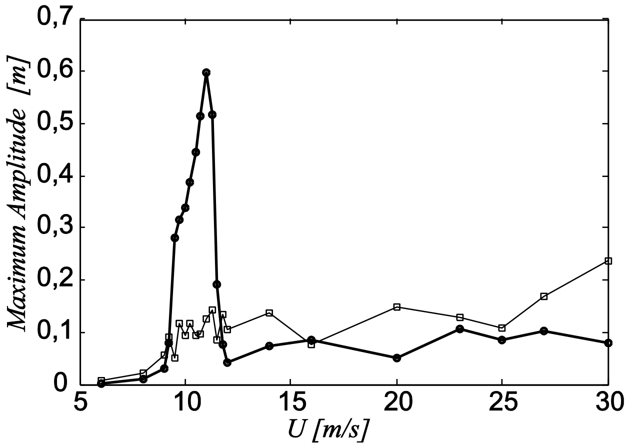


Fig. 1: (Left) Instantaneous view of particle stream; cylinder in a turbulent flow. (Right) Maximum response amplitudes of spring-supported cylinder (—o— = uniform flow, —□— = turbulent flow).

4. Conclusions

The approach presented here was to represent the turbulent flow in the numerical simulations which is subject to 2D limitations. However, it has been shown to provide bases for capturing 2D approximation of 3D wind flow. The approach can also provide engineers with the insight into the complex aerodynamic and aeroelastic phenomena. Turbulent flow was studied and modelled with the help of different geometrical shapes in CFD simulations and the flow characteristics were quantified. The turbulent flows were generated in the simulations for several cases of interest. The phenomenon of VIV was studied for these conditions and the effect on circular cylinder was quantified. The comparison was made for amplitudes of the cylinder in uniform flow as well as in turbulent flow situations. It is shown that the free stream turbulence has a substantial and systematic effect on the amplitudes. The effect on the frequency of the vortex shedding was also studied. The approach is computationally feasible and very similar to the accepted approach in the wind tunnel tests but important is to meet the requirements of the atmospheric wind flow characteristics. The approach can also be used to study other phenomena, such as flutter, to have better understanding of turbulent flow interaction with instability problems.