International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

Wind Effects on Bridge Deck: A Computational Fluid Dynamics Study

Vishav Murti[1], **Dr. Esar Ahmad**[2]

[1]Scholar, Department of Civil Engineering, Mewar University Chittorgarh, Rajasthan, India

Abstract: ***The interaction between wind and bridge structures has significant implications for their design, stability, and safety. This dissertation presents a comprehensive study of the effects of wind on bridge decks using Computational Fluid Dynamics (CFD) simulations. This research aims to analyze the aerodynamic behaviour of a typical bridge deck under various wind conditions and assess its structural response. The study begins with a detailed literature review that explores the historical development of bridge aerodynamics and the significance of wind-induced effects on bridge design. The simulation data provide insights into potential vibration modes, stress concentrations, and areas of concern for the integrity of the bridge. This research contributes to the development of more resilient and stable bridge designs.***

Keywords: Wind Effects, Bridge Deck, Computational Fluid Dynamics, Aerodynamics, Structural Analysis, Turbulence Models

1. Introduction

The instability caused by aerodynamics is a significant topic of study in the construction of bridges with long spans in the modern period because strength is not the sole consideration for slender structures. This study was conducted because of the wind's ability to change the direction or size of the load and how that affects the structure. Many bridge collapses in the history of bridge engineering have occurred as a result of an inadequate understanding of how the structure will behave under a dynamic load from wind, as in the Tacoma Narrow bridge failure. As everyone is aware, the 853-meter-long Tacoma Narrow Bridge, which was inaugurated on July 1st of that year, fell on November 7th of that same year in a spectacular fashion at a pace of only 18 meters per second. The bridge was the third-longest suspension bridge in the world, behind the illustrious George Washington and Golden Gate bridges, when it was built or even at the time of its demise. At the time, the impending demise of a recently built bridge put designers in a very difficult situation. All the experts in this field had no idea how a bridge that had been carefully designed and checked for all parameters could collapse under a light wind, and this gave rise to the new field of bridge aerodynamics. As many academics have studied the Tacoma Narrow bridge collapses, it has been discovered that flutter stability is the most frequent problem with the construction of bridges with long spans. In order to control this instability, two distinct trestle designs have been employed in the UK and USA.

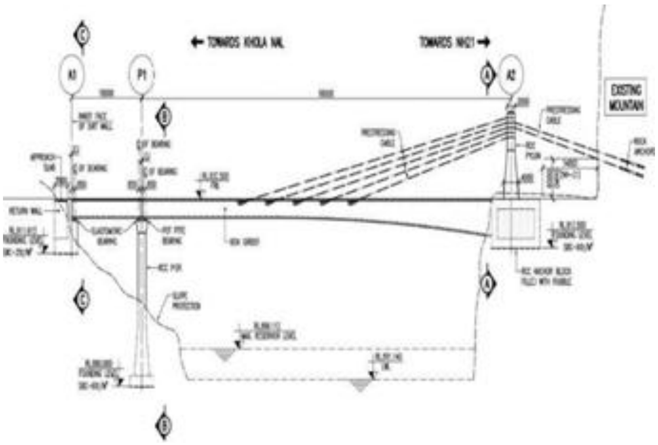
A new field, bridge aerodynamics, is learned as a result of this bridge's failure, and it is from this place that we also learn about bridge aeroelasticity, which provides insight into the connection between wind force and structural motion. Since 1940 till the present, designers and the bridge industry have been greatly impacted by the Tacoma Narrow Bridge research. Now that science and technology have advanced daily, there are more and more demands for longer spans, greater strength, beautiful look, and improved aerodynamic performance. However, as we continue to lengthen the span of the bridge, it gets more flexible and causes more flutter instability; to put it another way, the danger of failure rises quickly and safety declines.

- **Analysis Methodology of Bridge Aerodynamics** There are two different approaches that are frequently used to solve problems relating to the aerodynamics of bridges and their interaction with wind flow. The first strategy focuses on researching the theoretical growth that produced formulae for assessing aeroelastic characteristics. The second scientific approach uses wind tunnel testing to determine critical speeds. In order to mimic instability brought on by flutter in the structure, simpler finite element models of bridges are created and subjected to static and dynamic loads using numerical simulation.

In this section, the second approach, which is the experimental study, will be discussed. Wind tunnel testing is employed in this method to determine the aerodynamic characteristics of bridges. Presently, this method stands as the most dependable means to validate results for issues concerning bridges with long spans. Both closed and open circuit wind tunnel facilities are used to simulate the atmospheric boundary layer, providing a natural wind pressure environment for testing the wind tunnel model. The data obtained from these tests are thoroughly analyzed, and the behaviour of the actual structure is predicted based on the results.

- **Introduction to cable-stayed Bridge** Bridge engineering is evolving along with technology. The cantilever beam, beam bridge, truss bridge, arch bridge, and cable-supported bridge are the five primary types of bridges. Cable-supported bridges are commonly used when long-span bridges are required. One notable example is the Russky Bridge, located in Russia, which was inaugurated in 2012. This bridge boasts a primary span of 1104 meters. On the other hand, the Akashi Kaikyo Bridge holds the title of the world's longest suspension bridge, with a central span of 1991 meters.
-
- **Hanoi Cable Stayed Bridge** The Hanoi cable-stayed bridge is a significant infrastructure project currently under development in the Kathua district of Jammu & Kashmir. It will be the first cable-stayed bridge in the region, connecting the Basoli and Billawar areas and enhancing communication

between them. The construction of the bridge is currently in its design phase. The total span of the bridge is 113 meters, with an RCC Pylon that rises 14 meters above the deck level, as shown in Figure 1.2. A twin-cell prestressed box girder supports the structure at a height of 34.39 meters above the reservoir level. To improve weight transmission, the girder and pylon are connected by pre-stressed stay cables.



2. Objective

The objective of this research is to assess the aerodynamic coefficient of the bridge deck, as it plays a critical role in determining the aerodynamic forces acting on the structure. The investigation employs the fast procedure for Fluid-Structure Interaction (FSI) analysis on the bridge. Considering the information mentioned above, the subsequent discussion outlines the research's main goal and focus.

- Using the ICEM program, a three-dimensional Hexa mesh is created for the bridge deck.
- Modal analysis is performed on a 3-dimensional finite element model created with ANSYS software and given all the structural and geometrical features of a real structure.
- Wind flows around the deck, and pressure changes on the deck's surface.
- The structure's dynamic reaction to the applied wind stress.

3. Assumptions

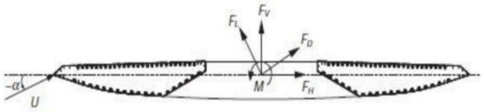
Some assumptions are made in this research work in order to reduce the robust computing efforts and the computational time, ensuring that the study's results are practical.

- To save on computing costs, a 3-D segment of the bridge is chosen for Fluid-Structure Interaction rather than the entire structure.
- Young's modulus is decreased to achieve a greater degree of deformation than steel.
- Gravity and the structural damping effect are not considered in the structure section.

4. Literature Review

The primary objective of this project is to develop a computational method to address the challenges posed by wind effects on the bridge deck. This problem involves multiple physics, namely structural dynamics and fluid flow. Understanding the interplay between these two phenomena is essential for effectively tackling the issue.

In bridge aerodynamics, the mean wind load can be represented in two different ways: one is using the structural coordinate, and the other is using the wind coordinate.



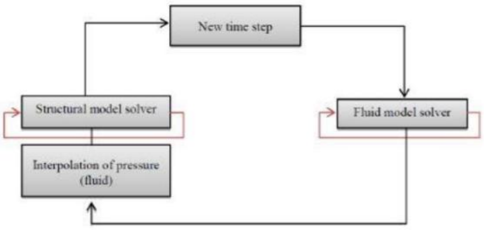
It is quite challenging to determine the force coefficient for bluff bodies like bridge decks since it depends on the shape, or, as we say, the geometrical characteristics of the shape as well as the wind factors. As we know, the bridge deck has three force components. These are additional effects of how the wind is behaving at an angle of attack.

$$F_D(\propto) = 1/2\ \rho U^2 B C_D(\propto)$$
$$F_L(\propto) = 1/2\ \rho U^2 B C_L\ (\propto)$$
$$(\propto) = 1/2\ \rho U^2 B C_M(\propto)$$

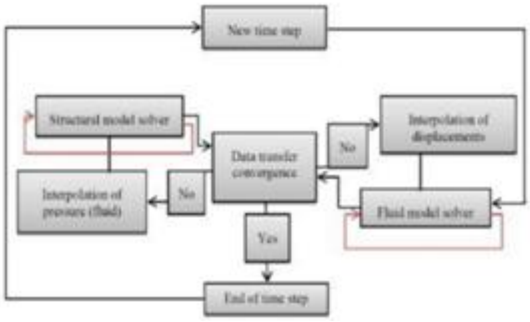
Where ρ is the density, B is the characteristic dimension, U is the wind velocity,

$C_D(\propto)$, C_L $(\propto)$, $C_M(\propto)$ are the drag lift and moment coefficient respectively.

One-Way Coupling Approach During a simulation, the fluid motion affects the structure, causing deformations in the structure that do not, in turn, affect the fluid flow. This indicates that the structure's deformation has no impact on the fluid's flow. The figure provided below illustrates the entire working process of this approach.



Two-Way Coupling Approach Simulating the two-way fluid-structure interaction (FSI) takes more time compared to the one-way FSI. In the two-way FSI, the fluid flow impacts the structure, causing deformation, and the resulting deformation in the structure, in turn, influences the fluid flow by disturbing its flow pattern. The given graphic provides a detailed illustration of the two-way FSI simulation procedure.



Different Computational Approach

In fluent, there are many different models that may be utilized to address computational issues, and the simulation process is applied depending on the issues at hand as well as the system efficiency requirements.

Reynolds-Averaged Navier-Stokes (RANS) models

a) The Navier-Stokes time-averaged solution is offered.
b) Models of all turbulent length scales are created using it.

This method is most typically used to solve the industrial flow.

Large Eddy Simulation (LES)

a) This approach solves the spatially averaged form of the Navier-Stokes. Only large eddies are solved.
b) It is an efficient process as compare to the DNS and used for most practical problems and required high amount computational effort.

Direct Numerical Simulation (DNS)

Very high cost due which it is not used for industrial flows and in fluent it is also not available.

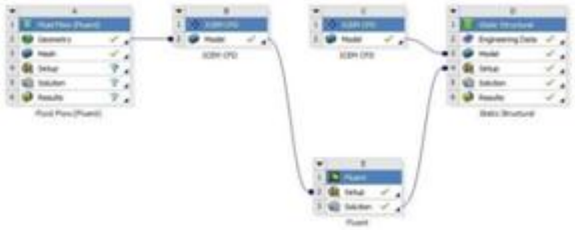
SST K-ω Model

The conventional K- ω model may not perform adequately in the free stream or boundary layer under negative pressure gradients. However, the K- ω model offers improved results in both the free stream and boundary layer. Menter et al.'s (2003) innovative Shear Stress Transport (SST) K- ω model combines the formulation of the classic K- ω model with that of the K- ω model, yielding superior outcomes, particularly in the near-wall region. This approach helps to address turbulence challenges in a more accurate manner, making it a favourable choice for certain flow conditions.

5. Methodology

In pursuit of the study's objectives, it is imperative to engage in fluid-structure interaction. To comprehend the structure's response, the initial approach employs the one-way coupling technique to establish a connection between the structural and fluid solvers. This study utilizes Fluent as the fluid solver, while the mechanical system is resolved using a static structure solver. To surmount this challenge, both a fluid structural solver and a static structural solver are harnessed on a unified platform known as ANSYS WORKBENCH. This platform's notable attributes include the ability to integrate multiple solvers of diverse natures and the seamless exchange of information between these solvers. Facilitated by the solver, pressure is transmitted to affect the coupling of the system.

- **Tool Used:** All the tools employed in this study are comprehensively providing details on their specific applications. Prudent consideration was dedicated to the selection of solvers, recognizing their pivotal role in the study's execution. The assortment of tools was meticulously curated, drawing from diverse sources such as prior research and empirical data.\
- **Approach Towards FSI:** In this context, the Fluid-Structure Interaction approach is achieved by utilizing ANSYS' Fluent and Static Structural modules in tandem. Figure 3.1 visually depicts the sequential steps of the coupling process for the fluid-structure interaction technique, as facilitated within ANSYS Workbench.



All the geometries employed in this study have been meticulously designed in a modular fashion, encompassing both the fluid domain and the structural components. These modular designs can be readily accessed within Ansys Workbench, serving as the platform for geometry construction. With Ansys Workbench serving as the overarching platform for the entire project, seamless communication of information and results between solvers is effortlessly facilitated. Thus, both solutions share the same geometry, enhancing consistency and coherence.

- **Geometry:** To delve into Fluid-Structure Interaction (FSI) within this research, a model of the Hanoi cable-stayed bridge is meticulously constructed, as depicted in Figure 3.2. However, given time limitations and the desire to circumvent computationally burdensome intricacies, our focus centres on a specific segment of the bridge deck, measuring 22 meters in length (Figure 3.3). To streamline operations and minimize computational expenses, a series of simplified geometrical assumptions are also incorporated, facilitating the avoidance of complex meshing processes.

Figure 3.2 Hanogi cable stayed bridge

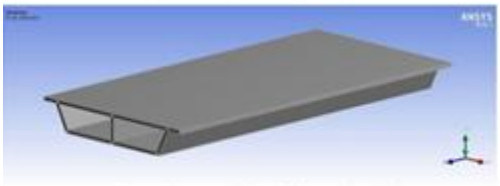


Figure 3.3 Isometric view of 22 meter long deck

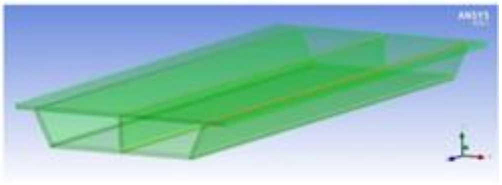


Figure 3.4 Transparent Isometric view with six prestressed wires

6. Simulation

From this juncture, the task will be bifurcated into two distinct sections. The initial phase entails the fluid stage, encompassing the resolution of all fluid flow-related physics and the subsequent discussion of the resultant flow parameter outcomes. The subsequent component involves the simulation of the structural solver, wherein the structural aspect is meticulously addressed, and an array of parameters are derived through this analysis, utilizing the pressure data derived from the fluid flow simulations. The ensuing paragraphs offer a succinct delineation of the comprehensive simulation procedure.

Coupling Setup

The final and pivotal element of the simulation is the establishment of a connection. In this regard, a one-way coupling methodology is employed, and comprehensive details on this approach are furnished in the thesis file. The graphical user interface (GUI) of the one-way coupling process is illustrated. This step proves advantageous in importing the fluid flow's pressure onto the structural system and transferring the relevant data from the fluid solver to the structural solver.

References

[1] Ashtiani Abdi, I. (2011). Experimental & Numerical Investigation of the Wind Effects on Long Span Bridge Decks.
[2] Bosch, H.R. & Dhall, Y. (2008) Aerodynamic Investigation of Generic Bridge Deck Section Models.
[3] Gu, M., Xiang, H., & Lin, Z. (1999) Identification of Bridge aerodynamic coefficients from measured displacement or acceleration.
[4] Theory of Structure, Dhahpat Rai & Sons.
[5] Design of Reinforced Structure, N. Krishnaraju.
[6] Design of cable stayed pedestrian bridge, T. Nelson Ponnu Duria, Dr P. Asha & R. Vinoth Kumar.