Available online at www.sciencedirect.com

ScienceDirect

Procedia Engineering 145 (2016) 774 – 781

Procedia Engineering

www.elsevier.com/locate/procedia

International Conference on Sustainable Design, Engineering and Construction

# Evaluation and Analysis of Wind Flow for a Football Stadium

Weijun Mei[a], Ming Qu.[a]*

*Purude University, Lyles School of Civil Engineering, 550 stadium mall Dr. CIVILG243, West Lafayette, IN 47906, United States*

**Abstract**

Wind influences many aspects in the design of modern stadiums. Computational Fluid Dynamics (CFD) modeling, as one of the numerical analysis methods, can assist architects and engineers in finding solutions for a safe and efficient design for a stadium. This paper presents the wind analysis process for a football stadium in West Lafayette IN by using CFD to evaluate its future retrofit design. The current and future design concept models, include the stadium, surrounding buildings, and topography, were built using AutoCAD. Based upon these models, steady state RANS (Reynolds-averaged Navier-Stokes) CFD mathematical models were developed using ANSYS Fluent software to evaluate the wind flow patterns around and inside the stadium. Using the current stadium as a baseline case, the future design concept was simulated to predict its performance. Simulation results suggest that there will be bigger area of high velocities and vortexes occurring in the future design concept due to the closed shape of the future design. A modification of having an opening on the first floor was proposed, and the resultant wind flow was simulated. The final results show in most cases the area of high velocities and vortexes is much smaller compared to that of the design concept resulting from the opening. The approach used in the study can be generally applied in the wind analysis for different types of buildings or infrastructures.


Peer-review under responsibility of the organizing committee of ICSDEC 2016

*Keywords:* Wind; Stadium; Computational fluid dynamic; ANSYS Fluent

* Corresponding author. Tel.: +1 765 494 9125; fax: +1 765 269 7903.
*E-mail address:* mqu@purdue.edu(M. Qu)


Peer-review under responsibility of the organizing committee of ICSDEC 2016
doi:10.1016/j.proeng.2016.04.101

## 1. Introduction

Wind is an important factor affecting building performance, especially for stadiums. Typically, stadiums have huge open spaces, and as a result, they are susceptible to surrounding infrastructures. Open space means there is no wind protection from envelops. Thus, surrounding buildings sometimes protect stadiums but sometimes accelerate the wind. Wind velocity also has a significant impact on the performance of football players. “I always thought that it was really wind that affected the passing game the most, and not temperature”[1], said by Brian Burke, an advanced football analyst.

A computational fluid dynamics (CFD) mathematical method is usually applied for wind flow analysis[2]. “Detailed aerodynamic information can be obtained using Computational Fluid Dynamics (CFD), which offers considerable advantages compared to wind tunnel testing”[3], according to B. Blocken. For a large football stadium in an urban environment, a steady state RANS (Reynolds-averaged Navier-Stokes) mathematical model is applied when establishing the numerical model[4]. This method has been receiving strong support[4]. In B. Blocken and J. Persoon’s research[4], they used three-dimensional steady state RANS CFD simulation in combination with a wind comfort standard to assess pedestrian wind comfort around a large football stadium. In T.Van Hooff’s research[5], they used RANS model to determine the wind flow pattern to evaluate the influence of stadium geometry. All these studies and researches verified that the RANS model is a reasonable method to simulate wind flow patterns. A quick and straightforward assessment is often required in engineering projects. To meet these requirements, this paper presented the wind simulation process for a stadium design and addressed a possible problem of the design.

The stadium studied is located in West Lafayette, IN, facing southeast with a capacity of 57236. It is a huge open stadium surrounded by a lot of buildings, as shown in Fig. 1a. A renovation plan was developed in 2015 by Purdue University. The design team was planning to add a multifunctional extension on the southeast part of the stadium and a huge screen on the northwest side, as indicated in the red circles shown in Fig. 1b. Also, one of the surrounding buildings was planned to expand, which could also affect the wind flow pattern around and inside the stadium. The renovation plan is shown in Fig. 2.

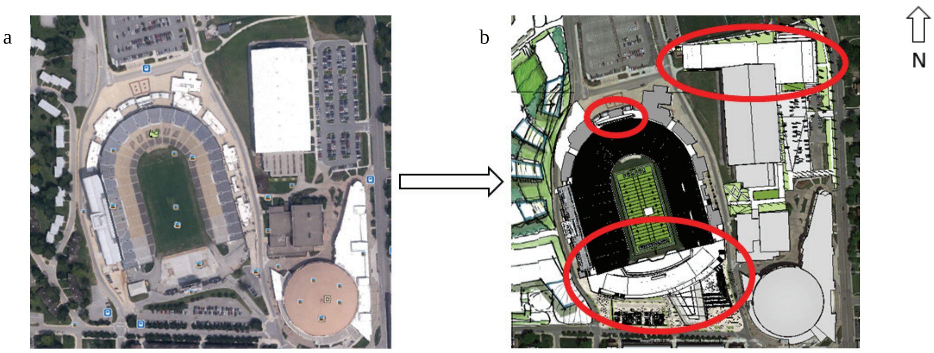



Fig. 1. Plans for (a) current stadium and (b) renovation

a b

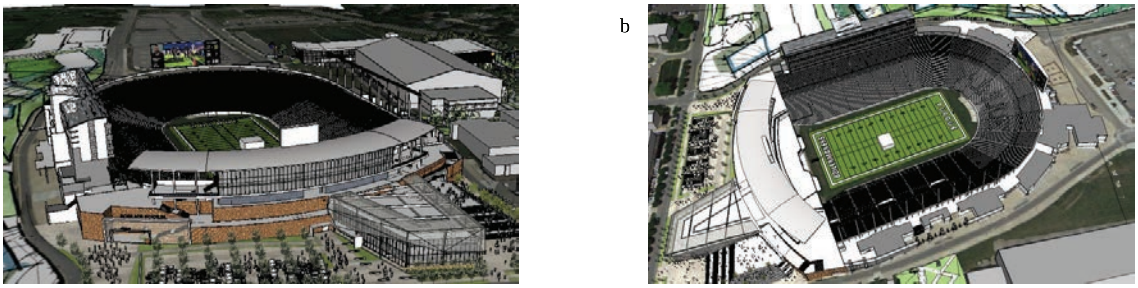


Fig. 2. (a) View from southeast; (b) Bird's view of the stadium of the renovation

The renovation of the stadium will be a closed circle with an open to the sky. Wind flow simulation is necessary for the design team to evaluate their design concept and visualize the wind field in their decision making process.

## 2. Method

### 2.1 3D simulation model

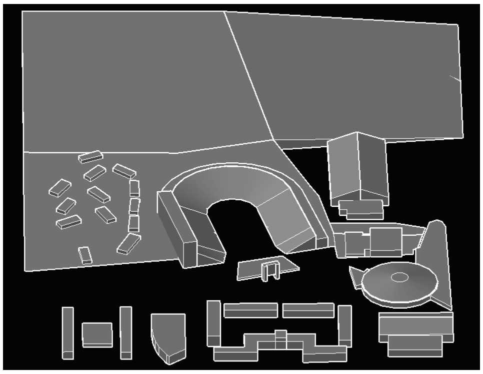
To assist in the design of the renovation, we developed a CFD model to evaluate the wind field and analyzed the performance of the new design using ANSYS, an integrated finite element analysis software with different modules that are capable of many different types of simulation, including structural, mechanical, electronics, etc. Fluent was integrated into ANSYS several years ago, and it is now more powerful.

For a CFD simulation, a 3D model must be well developed to maximize its accuracy. AutoCAD was selected in this study to develop the 3D model because AutoCAD is highly compatible with ANSYS. Almost all details of the site, including edges, overlapping, and others, can be remained as the same in AutoCAD when the 3D model in AutoCAD is exported to ANSYS.

Based on the existing stadium and surrounding buildings, we developed a 3D model in AutoCAD, including the stadium, surrounding buildings, and the topography of the site. Dimensions of the stadium and surrounding buildings were strictly retrieved from the SketchUp model and Revit model provided by the design team and Google Earth Pro data. Since it was unpractical to have the topography of the model strictly following the real topography, due to complicated shapes, it was reasonably simplified according to the topography drawing provided by the design team and Google Earth Pro data.

The design concept model was also developed in AutoCAD using the same method. The topography was connected with the stadium as one solid body. The 3D models built in AutoCAD for current stadium and the renovation are shown in Fig 3.

a



b

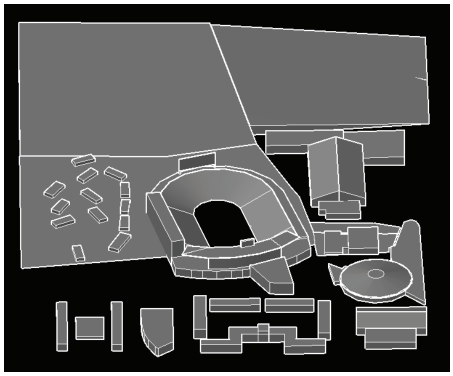
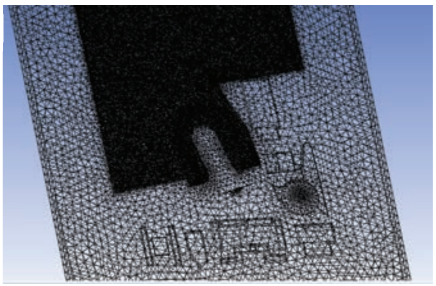


Fig. 3. 3D models in AutoCAD for (a) Current Stadium (b) Design Concept model

## 2.2 Models in ANSYS

The models developed in AutoCAD were exported to ANSYS for CFD study. The calculation areas for both models were 1100m*850m*50m. It was also developed in AutoCAD as a solid body. The stadium and surrounding buildings were all included in it. Under the AutoCAD commend “subtract”, the stadium and surrounding buildings were subtracted from the calculation area. That is, in the calculation solid body, the stadium and surrounding buildings were substituted with empty space in the same shape as their solid body 3D model. Air was set as the study material and other surfaces were treated as walls. Then the wind flow patterns on surfaces and space of the stadium with surrounding buildings can be simulated.

a



b

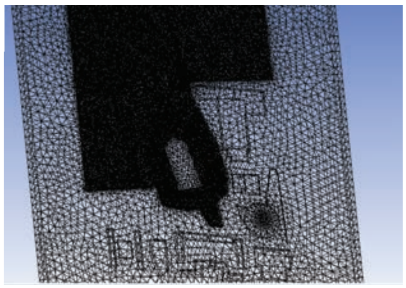


Fig. 4. (a) Current stadium mesh setting in ANSYS; (b) Design Concept mesh setting in ANSYS

The stadium is embedded in the topography. This is why the topography and the stadium were connected together as one solid body in the model in AutoCAD, as well as in mesh. In ANSYS, the size of mesh element for the topography area and the stadium area were set as 10 meters and 5 meters, respectively. The mesh element size in other areas was provided by ANSYS meshing.

## 2.3 Simulation in Fluent

The wind blows from one direction to another. Also, the simulations were conducted in the steady state. The calculation area was 1100m*860m*50m where 1100m was the length and 860m was the width. As the length and width are much larger than the height, the calculation area is like a plate. As discussed in introduction, the steady state RANS method has been receiving strong support from other researches, therefore, a steady state RANS model was selected for the simulation in the present paper.

According to NEN 8100(NEN, 2006a)[6], in the range 0-5m/s, the comfort levels for the activities of traversing, strolling, and sitting are considered beyond moderate. Also, according to the design team's requirements and discussion, we considered the range of 0-14 mph (0-6.2m/s) as an acceptable wind speed range for football games. Therefore, we take 0-14 mph (0-6.2m/s) as an acceptable wind speed range for the stadium. In this case, the comfort level for the activity of sitting is considered as poor. As football player’s performance is the focus, the range 0-14 mph (0-6.2m/s) was selected for simulation purposes.

The wind speed selection was based on weather data. WeatherSpark data showed that in West Lafayette area, over the course of a year, typical wind speeds vary from 0 mph to 18 mph (calm to fresh breeze), rarely exceeding 25 mph (11m/s) (strong breeze). In this paper, 25mph (11m/s) was selected as the highest magnitude combined with various directions, being the conditions for worst case simulations. We also studied normal cases, in which wind velocity was 8mph, the average mean velocity from September to November. In these normal cases, west (26%) and southwest (18%) as primary directions were used in simulations, and results of primary directions are shown in this paper.

## 3. Results

Four different height values of 2 meters, 5 meters, 15 meters, and 20 meters were selected for observing wind field conditions. The height of 2 meter represents the level of the playground area. The 5 meter height is the possible football flying height. The height of 15 meters is the possible height of barbecue smoke, which could occur on the second floor of the southeast extension. The 20 meter height is the highest seat area. Wind field conditions at the four height values were analyzed and compared. The wind field modeling results are typically presented in a figure with three scenarios: a) current stadium, b) concept design, and c) the solution and for each scenario, each with four wind field diagrams for the four elevations at 2, 5(upper right),15(lower left), and 20 meters.

In the worst case, when wind was at 25mph (11m/s), in the current stadium model, high velocities (higher than 13.4m/s) were found at 20 meters’ height when it blows from west (primary direction, Fig. 5a), south, and northwest. In the design concept model, high velocities were found when wind blows from west (primary direction, Fig. 5b), south, southwest (primary direction, Fig. 6b), southeast, and northwest. Also, vortexes were found when wind blows from southwest, southeast, and northwest.

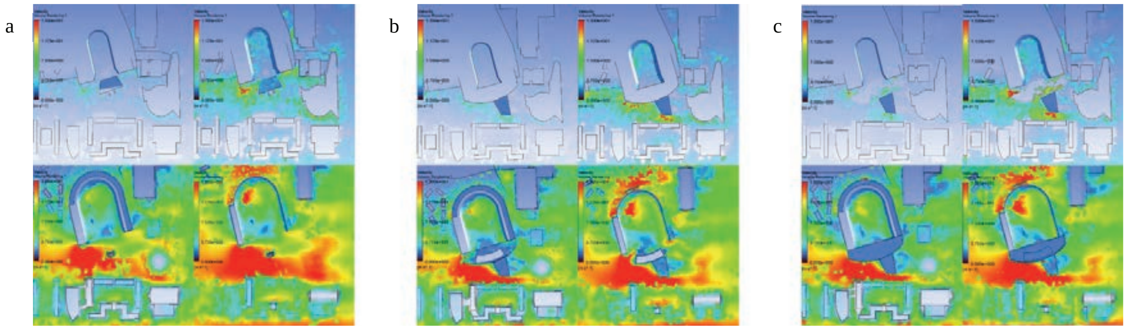


Fig. 5. Wind blows from west at 25mph (11m/s), (a) current Stadium; (b) design concept; (c) solution

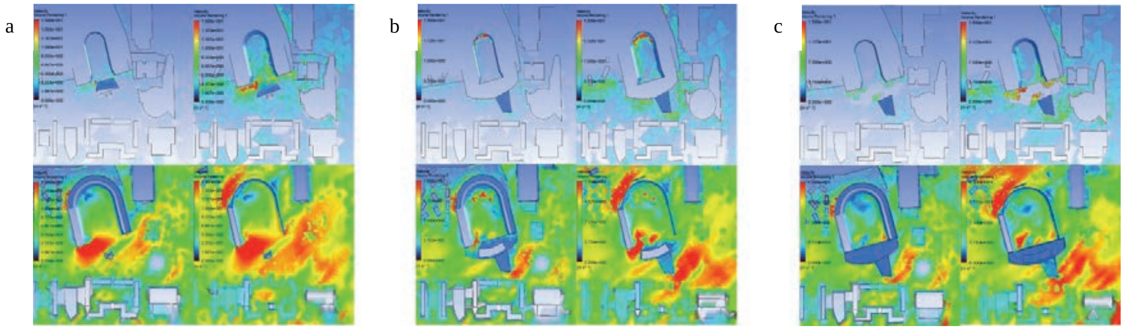


Fig. 6. Wind blows from southwest at 25mph (11m/s), (a) current Stadium; (b) design concept; (c) solution

In normal cases, when wind was at 8mph (3.6m/s), blowing from west and southwest, there was no high velocity or vortex found in the current stadium, as shown in Fig. 7a and 8a. However, when wind was from west and southwest, there were high velocities in the design concept model, as shown in Fig 7b and 8b. Besides, at a height of 15 meters, the wind velocities were found to be less than 10mph (4.5m/s) in both cases. This means the platform is suitable for barbecue activity.

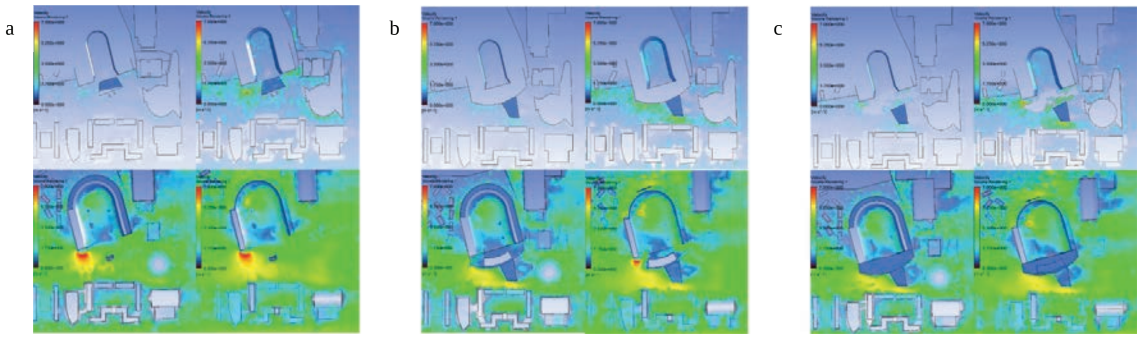


Fig. 7. Wind blows from west at 8mph (3.6m/s), (a) current Stadium; (b) design concept; (c) solution

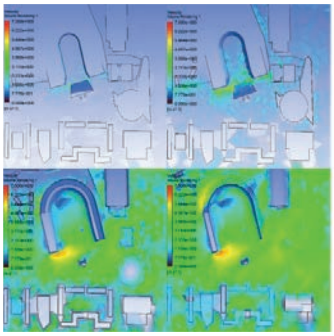
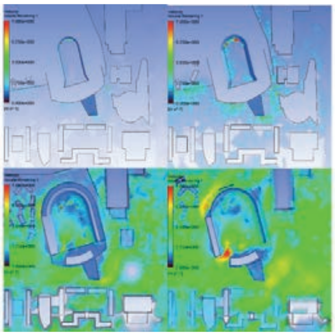
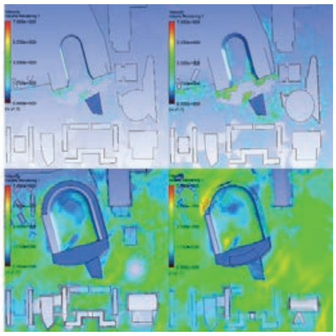
a

b

c


Fig. 8. Wind blows from southwest at 8mph (3.6m/s), (a) current Stadium; (b) design concept; (c) solution

## 4. Discussion

The high velocities and vortexes appeared in the cases where wind was from west, south, southwest, southeast, and northwest in the design concept model. However, in the current stadium model, the areas of high velocities were much smaller than that of the design concept and no vertex occurred. Considering the topography and position of the stadium, we believe that the designers used the topography and slope to protect the open stadium from high velocities and vortexes 90 years ago. Such phenomenon of high velocities and vortexes occurred in the design concept was analyzed. Based on general understandings of wind engineering, we found the following reasons for the high vertexes occurred in the renovation design.

1. The extension located at the southeast end blocked the wind way from north to disperse.
2. The huge screen in the northwest side blocked the wind way to disperse.
3. The space between the seats and the big screen accelerated the wind velocity.
4. The space between seats and southeast side extension acted like a valley, which accelerated the wind speed.
5. The whole shape was like an open egg, which had no roof protection but blocked the wind way to disperse, as the wind cannot disperse vertically.

Among the above mentioned possible causes, the first guess was considered as the main reason in this study. A modified model was developed to verify this assumption. We moved the extension building at the southeast end upwards by 7.5 meters, which was as same height as the seat area. In this case, the stadium remained open at the ground level. The modification of the model in AutoCAD is shown in Fig. 9, and the raised extension at the southeast end was shown in the red circle.

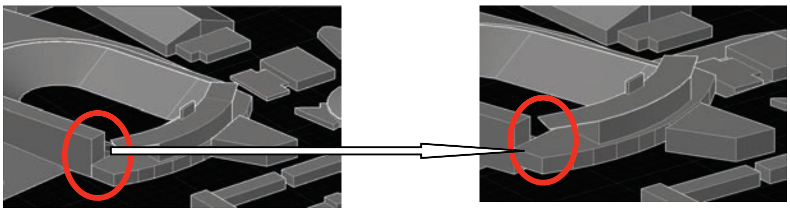


Fig. 9. Modified model

Afterwards, another simulation was conducted in ANSYS Fluent with the same settings. Calculation results showed that high velocities appeared when wind blowed from west (Fig. 5c), southwest (Fig.6c), southeast, and northwest. However, compared to the design concept model, the areas of high velocities were much smaller. The solution model was closer to the current stadium model. It was also found that when wind blowed from west and northwest, even with the huge screen, the areas of high velocities in the solution model were much smaller than those in the design concept model.

## 5. Conclusion

The steady state RANS CFD simulation method was used to investigate the wind flow patterns around and inside the current stadium and future renovation models. Simulation results indicated that larger areas of high velocities and vortexes were found in the future design concept model compared to the current stadium, which are not acceptable for a stadium. The closed shape was regarded as the major reason for these high velocities and vortexes. A modified model was developed to verify this assumption. With the same settings, simulation results indicated that, when the extension at the southeast end is moved up by 7.5 meters, the areas of high velocities and vortexes were much smaller compared to those in the design concept and closer to the current stadium model. Such results confirmed our assumption. Therefore, a recommendation was proposed to the design team. The entire process, from site assessment, developing 3D model, to the simulation in ANSYS Fluent, can be used as a general method for engineering project in wind field analysis.

## 6. Acknowledgement

The authors would like to thank POPULOUS design team for their models, topographies, discussions and other information, which made this study possible.

## 7. References

1. Burke, B. Weather effects on passing. http://archive.advancedfootballanalytics.com/2012/01/weather-effects-on-passing.html
2. Reiter, S. Assessing wind comfort in urban planning. *Environment and Planning B: Planning and Design* **2010,** *37*, (5), 857-873.
3. Blocken, B.; Janssen, W. D.; van Hooff, T. CFD simulation for pedestrian wind comfort and wind safety in urban areas: General decision framework and case study for the Eindhoven University campus. *Environmental Modelling & Software* **2012,** *30*, 15-34.
4. Blocken, B.; Persoon, J. Pedestrian wind comfort around a large football stadium in an urban environment: CFD simulation, validation and application of the new Dutch wind nuisance standard. *Journal of Wind Engineering and Industrial Aerodynamics* **2009,** *97*, (5–6), 255-270.
5. Van Hooff, T.; Blocken, B.; Van Harten, M. 3D CFD simulations of wind flow and wind-driven rain shelter in sports stadia: Influence of stadium geometry. *Building and Environment* **2011,** *46*, (1), 22-37.
6. NEN, Wind comfort and aind danger in the built environment. Nederlands Normalisatie Instituut Publications: 2006.