HEFAT 2010
7 th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
19-21 July 2010
Antalya, Turkey

NUMERICAL AND EXPERIMENTAL ANALYSIS OF WIND LOADS ON CLADDING OF TALL BUILDINGS

Levent ÇOLAK[a*], Emre ÖZTÜRK[b], Mehmet N.BALCI[a], Serkan B.KÖRPE[a]
*Author for correspondence
[a]Başkent University ,Department of Mechanical Engineering, Asst. Professor
Baskent University Mechanical Engineering Department Bağlıca Campus, Ankara, TURKEY
[b]Anova Engineering Ltd.
Silikon Blok Teknokent ODTÜ, Ankara, TURKEY
E-mail: lcolak@baskent.edu.tr

ABSTRACT

Calculating wind loads is important in design of the wind force resisting systems, including structural members and cladding against shear, sliding over-turning and uplift actions.

Simple quasi-static treatment of wind loading which is universally applied to design of typical low to medium rise structures can be unacceptably conservative for design of very tall buildings.

Due to the significant cost of typical facade systems in proportion to the overall cost of tall buildings, engineers cannot afford the luxury of conservatism in assessing design wind loads.

For determining the type and thickness of cladding material, the selection criterion is maximum pressure that it can withstand, where maximum pressure is stagnation of wind on the building. Although stagnation pressures can be roughly calculated by simple Bernoulli equations, the atmospheric boundary layer and blockage effect of the surrounding buildings cannot be resolved by such primitive approaches.

Wind tunnel testing to assess design loads for cladding materials is now usual industry practice, with the aim of minimizing initial capital costs and maintenance costs associated with structural failure.

Since experimental analysis is not always feasible and practical, creating mathematical model and selecting the most appropriate cladding material and thickness by using CFD analysis is more effective and cheaper approach.

The aim of this study is to validate the numerical analyses results against the experimental data and then create a 1-1 scale numerical model to find the wind loads on the claddings to make optimizations on the thicknesses when possible to reduce the material cost of the construction.

For this purpose, a tall building in Ankara was selected and the building model and adjacent buildings were created with an appropriate scale by using CATIA V5. The CAD model created in CATIA was used to build up the mathematical model for the computational fluid dynamics (CFD) analyses. Mathematical model was analyzed by commercially available CFD program Ansys Fluent. Also, a mock-up was produced for the wind tunnel testing.

In order to create the atmospheric boundary layer effect, some small obstacles were placed in front of the building. Pressure values were recorded via pressure transducers and collected data were compared with numerical results.

Since the experimental and numerical analysis results were in good agreement, the mathematical model was validated by an acceptable error margin. Therefore, by using this derived mathematical model, cladding thickness selection due to wind loads can easily be done cost effectively for different tall buildings by using CFD techniques.

INTRODUCTION

Aerodynamic effects should be considered for the design of high rise buildings as well as the architectural and the structural concerns.

Studies have started with Jack E. Cermak who is considered as the “father of wind engineering” in 1959[1]. Since then several studies have been done. In 1970, Barriga *et. al.* [2] and Roberson and Crowe [3] did experimental studies in turbulent flow conditions. And to give a recent example, Ahmad and Kumar [4] worked on the wind forces on low rise buildings.

In this study, for optimization cost of the cladding material due to variable pressure distribution on façades of tall buildings, a CFD model has been built up.

The created model was three dimensional representations of the tall building in concern and its adjacent buildings. CFD analyses were performed with Ansys Fluent, a finite volume solver for Navier-Stokes equations. Various numbers of cells were created and grid independency was checked with coarse, medium and fine meshes. Pressure based solutions with second order upwind schemes were done. The boundary conditions were chosen from the data obtained from the State Meteorological Institute of Turkey. The wind speed and direction data was recorded on a daily basis for past years.

The critical conditions were considered to be the highest wind speed and this was entered as one of the boundary conditions for the CFD model. All of the solid boundaries were modeled as no-slip walls. The realizable k-epsilon turbulence model with enhanced wall treatment was used to predict the turbulence effects correctly.

For the validation of the CFD model, mock-up have to be produced and tested in a wind tunnel. The CFD methodology can be used to model the full scale geometries upon the agreement of the test data with numerical results.

For this purpose, several tall buildings in Ankara were examined, like, TÜBİTAK Central Building, Marriott Hotel Ankara and Bayraktar Tower Business and Shopping Center. As per the simplicity and the available geometric data, Bayraktar Tower was chosen as the building to be studied in this research.

Figure 1 Bayraktar Tower - Ankara

EXPERIMENTAL STUDY

Considering the Ankara Wind Tunnel test cell cross section area, Bayraktar Tower was modeled with a 1/144 scale by using Catia V5. For the consistency of the experiment and the numerical solutions, both the mock-up and the mathematical model were created in the same size. The other dimensions were also designed with using this scale which ensured that the numerical model to be analyzed was exactly the same as the physical model to be tested. While the original height of the building is 115 m, the mockup and the numerical model height is 0,8 m. In order to get correct flow distribution the adjacent structures around the building were also modeled.

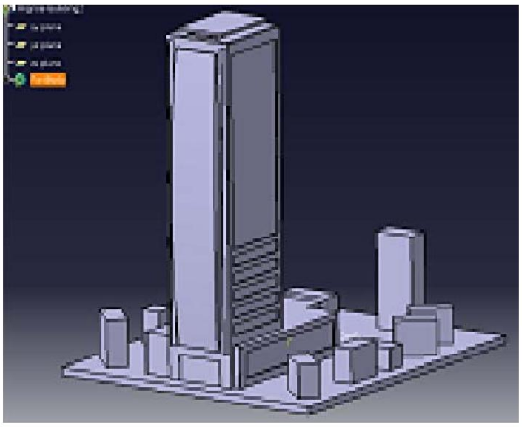

Figure 2 Tall building with adjacent structures were modeled

For wind tunnel testing, mock-up was produced by according to the designed model dimensions. Mock-up was produced in Başkent University, machine tool laboratory of Mechanical engineering department.

Figure 3 The mock-up of designed model

In order to get the correct boundary conditions, the wind characteristics data were obtained from the State Meteorological Institute of Turkey for the concerned region. The critical conditions were considered to be the highest wind speed of 25 m/s which was recorded in 1984. This value was used as the free stream velocity in the wind tunnel and also one of the boundary conditions for the CFD analysis.

Tests were performed in two directions with three different wind velocities. Wind velocities were 25 m/s, 12 m/s and 6 m/s from North-East (NE) and South-East (SE) directions.

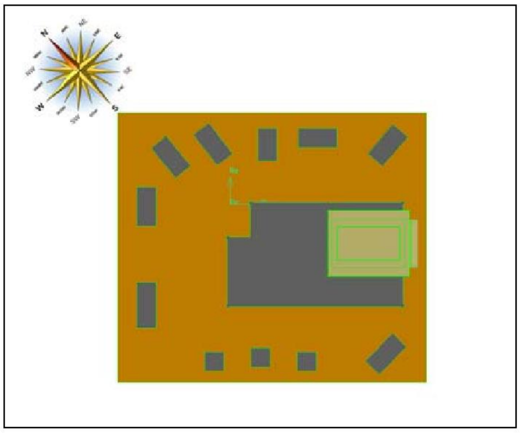

Figure 4 Orientation of the tall building

Experimental analyses were done in TÜBİTAK-SAGE (The scientific and Technological Research Council of Turkey) Ankara Wind Tunnel (AWT). The AWT is a closed circuit wind tunnel with the test chamber dimensions of 3,05 m x 2,44 m x 6,10 m. The 750 kW fan can provide free stream velocities up to 90 m/s in the test area. The turbulence intensity ratio is less than 0,5 %.

Figure 5 Wind tunnel scale model

In this experiment, pressures on the building model surface were measured by pressure tubes which were fixed on three facades of building. 45 pressure tubes were used totally, for this measurement.
15 pressure tubes on the South-East face, 14 pressure tubes on the North-East face and 16 pressure tubes on the North-West face were placed with 4 cm of spacing.

Pressure tubes were connected to a multiple channel pressure transducer which is capable of measuring negative pressures.

Figure 6 Assembling pressure tubes and cyclones with model

Figure 7 Connection the marked cyclones with transducer

In the first part of experiment, wind stream with three different velocities was provided from South-East direction of the building model.

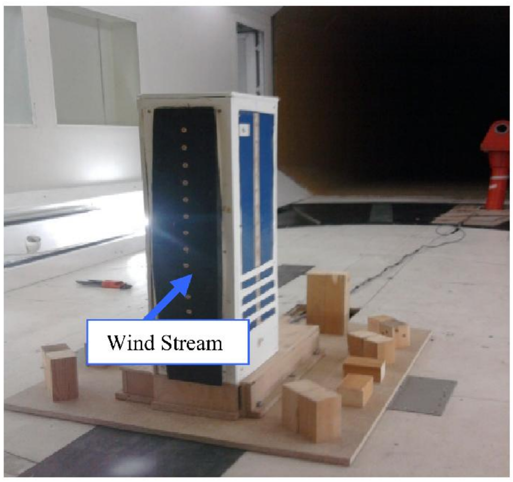

Figure 8 South-East wind streams in wind tunnel

In the second part of the experiment, wind stream with three different velocities was provided from the North-East direction of building model.

Figure 9 North-East wind streams in wind tunnel.

Results of Experiments

The pressure values recorded for the North-East wind stream of 25 m/s on the North-East, South-East and North-West facades of the building were given in Figures 10, 11 and 12.

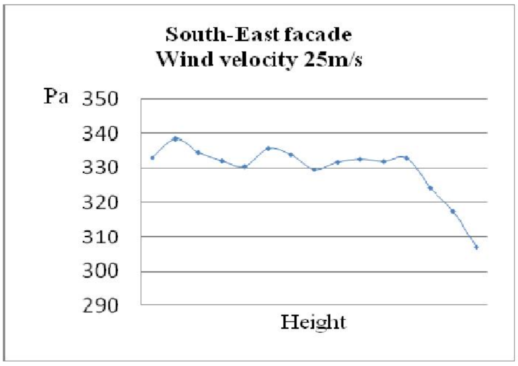

Figure 10 Pressure distribution on South-East facade - wind velocity 25 m/s

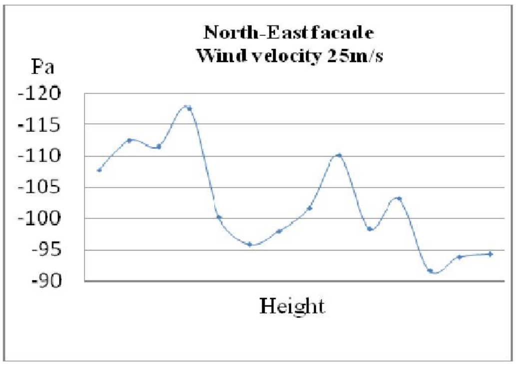

Figure 11 Pressure distribution on North-East (side) facade – wind velocity 25 m/s

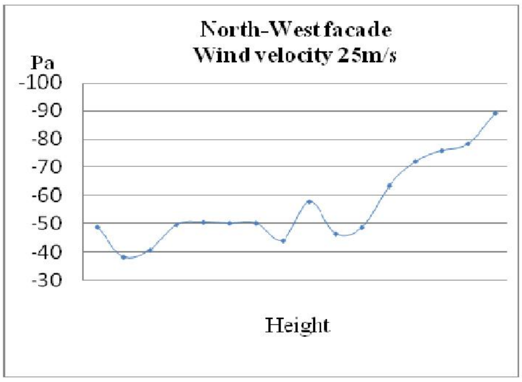

Figure 12 Pressure distribution on North-West (back) facade – wind velocity 25 m/s

Pressure values on the front face which was exposed to direct wind load reached their maximum because of the rise of the pressure on the stagnation points. However back and side of the buildings have negative pressure values due to separation of flow.

NUMERICAL ANALYSES

Numerical analyses were performed at Ansys Fluent v12. 3 different velocities from the South East side were given as boundary conditions.

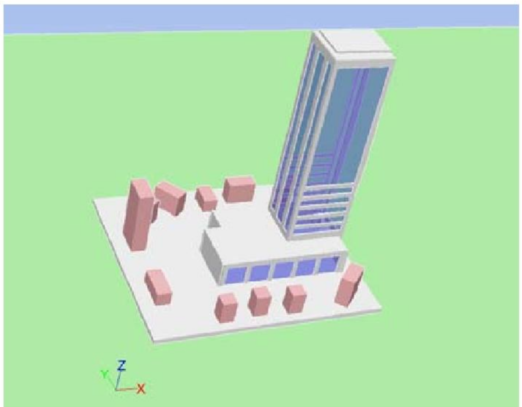

Figure 13 Numerical model

Turbulent flow was solved with the realizable k-epsilon turbulence model and the enhanced wall treatment was used to capture the boundary layer. The mesh as seen in Figure 14 was fine enough to resolve the near-wall high gradients. Total number of cells in the model was around 1,7 million. The cells in the boundary layer were prismatic and all the rest were tetrahedral.

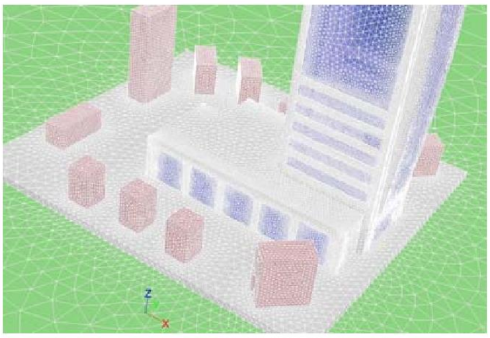

Figure 14 Surface meshes on the numerical model

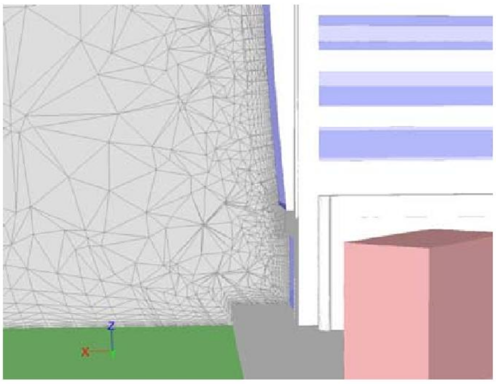

Figure 15 Boundary layer meshes on the cross section of the numerical model

The solution was started from First Order Upwind discretization scheme which was an easier way to converge but then switched to Second Order Upwind to obtain more accurate results. The convergence history was shown below.

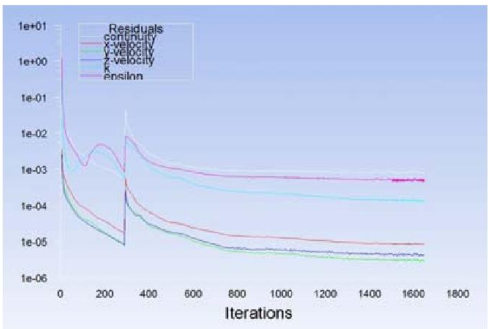

Figure 16 Convergence history

Results of Numerical Analyses

The pressure distribution and the velocity vectors on the mid-plane were shown on the below figures. As expected, the stagnation regions have the maximum pressure values which were recorded on the front face. The separation of flow and the vortices in the wake of the building can be seen from the velocity vectors (Figure 18).

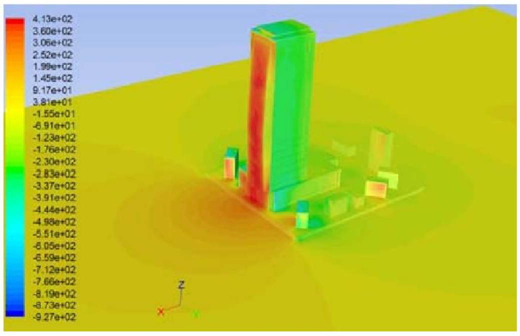

Figure 17 Pressure distributions on the numerical model

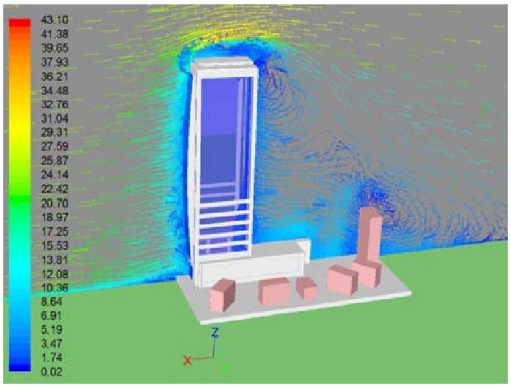

Figure 18 Velocity vectors on the cross section of the numerical model

In order to compare the numerical results with the experimental ones, the pressure values on the nodes which correspond to the physical positions of the pressure tubes were plotted and given in the Figures 19, 20 and 21. The effect of free stream velocity on the pressure values was given in Figure 22.

The pressure tube numbers shown on the figures 19, 20 and 21 start from 17 cm above the base of the building which was designated as number 1 and point locations were increased by 4 cm at each tube. Number 15 on the figure was for the pressure tube placed 73 cm above the base while the total height of the mockup was 80 cm.

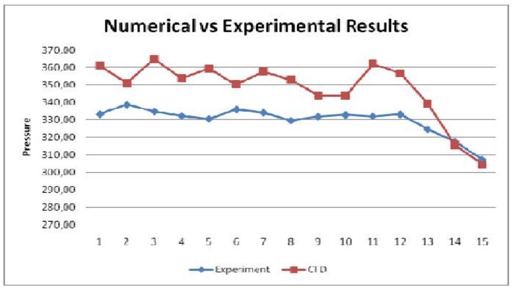

Figure 19 Numerical vs experimental results on the South-East facade for 25 m/s free stream velocity

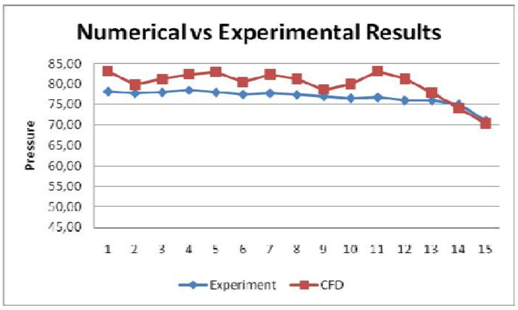

Figure 20 Numerical vs experimental results on the South-East facade for 12 m/s free stream velocity

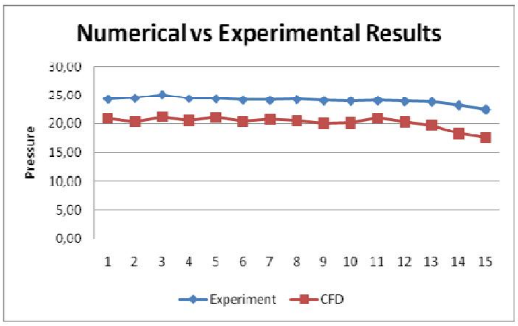

Figure 21 Numerical vs experimental results on the South-East facade for 6 m/s free stream velocity

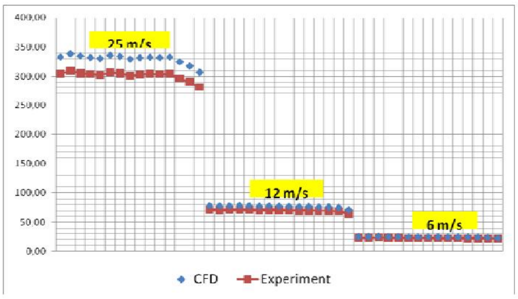

Figure 22 The effect of free stream velocity on the pressure values on the South-East facade

CONCLUSIONS

The numerical and the experimental results were in good agreement for the different velocities. The maximum difference for the pressure values were less than 20 % and the average differences were around 8 %. The differences have aroused especially due to the mockup surface quality and the unavoidable numerical errors.

The results could be improved by using a better mockup like a one manufactured from rapid prototyping techniques as SLA, SLS etc on the experimental side. And also using a finer mesh and a higher order turbulence model could improve the numerical results.

The methodology proposed in this study will help the tall building constructors to select the appropriate thicknesses for cladding by performing either wind tunnel tests or CFD analyses in a cost effective way. As the wind tunnel tests cannot be performed in such a way to ensure the Reynolds Analogy, the wind tunnel tests can be used to validate the numerical model and numerical models can be created in real scale which will provide useful engineering results.

Another outcome of this study is to understand the flow path lines around the buildings on the ground level and therefore when there are wind corridors or any unwanted region in terms of the comfort level of pedestrians, this can also be spotted and building orientations and architectural designs can be improved to prevent such regions.

ACKNOWLEDGEMENTS

The authors are thankful for the support of TÜBİTAK SAGE for letting them use the Ankara Wind Tunnel for the experimental study. Without their help this study could not have been realized.

REFERENCES

[1] Karem, A., A Tribute to Jack E. Cermak-Wind Effects on Structures: A Reflection on the Past and Outlook for the Future, Nathaz Modeling Laboratory, University of Notre Damme 156 Fitzpatrick Hall, USA.

[2] Barriga, A.R., Crowe, C.T., Roberson, J.A., Pressure Distribution on a Square Cylinder at a Small Angle of Attack in a Turbulent Cross Flow, Wind Forces on Buildings and Structures, 89-93, 1976.

[3] Roberson, J.A., Crowe, C. T., Pressure Distribution on Model Buildings at small Angles of Attack in Turbulent Flow, Proc. 3rd U.S. Natl.Conf. On Wind Engineering Research, University of Florida, Gainesville, 289-292, 1978.

[4] Ahmad, S., Kumar, K., Interference Effects on Wind Loads on Low-Rise Hip Roof Buildings, Engineering Structures, Vol. 23, 1577-1589, and 2001.