3D COMPUTATIONAL MODEL INCLUDING TOPOGRAPHIC DATA FOR THE PREDICTION OF DIRE WIND REGIMES AND VORTEX SHEDDING IN AN ATTEMPT TO IMPROVE TALL BUILDING RELIABILITY

Drishtysingh Ramdenee[(1)], Pierre Luc Paradis[(2)], Hussein Ibrahim[(3)], Hemraj Joyram[(4)], Adrian Ilinca[(5)]

[(1)]Institut Technologique de la Maintenance Industrielle, Cegep de Sept Îles
[(2)]T3e, École Technologique supérieur, Montréal
[(3)]Technocentre Éolien, Gaspé, Québec
[(4)]Sir Abdool Raman Osman, Phoenix, Mauritius
[(5)]Wind Energy Research Laboratory, 300 allée des Ursulines, Rimouski. G5L3A1

drishtysingh.ramdenee@itmi.ca

ABSTRACT

A structure's wind resistance largely determines its Extended Coverage Endorsement (ECE) and is an important factor in determining total insurance costs. Insurance policies and costs are often not very representative or prejudicial to either the insurer or the policy holder due to uncertainties in the ability to predict structures' wins resistance capacities. In other words, according to International Standards (ISO 2394:1998(E)) all structures including tall buildings must abide to the criteria of serviceability. Serviceability means the ability of a structure or structural element to perform adequately under normal use, expected actions or bounded actions (actions which have a limited value which cannot be exceeded and which is exactly or approximately known). Consequently, the ISO rules stipulate that choice of structural system, design for durability and implementation of quality policy need to be accordingly set up as to appropriate degree of reliability, which, in turn should be judged with due regard to the possible consequences of failure. In the case of tall buildings, mostly as to what concerns claddings, it is becoming more and more important to define proper reliability framework to increase certainty and render insurance policies more tailor made and unbiased. When it comes to static or quasi-permanent value solicitations on buildings such as known winds, collisions, etc, a high level of certainty exists in the calculations and models predicting failure and damage. However, when it comes to loads with high variability, the reliability index becomes very low. In many countries, skyscrapers are rapidly growing in restrained areas such that the formed corridors accelerate the winds and increase formation of vortices. Furthermore, in cases of rapidly changing gusts, the wind loads can be quite dangerous and cause damage and even failure to the buildings. Unfortunately, limited work enables very precise prediction of such loads such that insurance policies are accordingly unspecific. In this paper, work conducted at the Wind Energy Research Laboratory (WERL) to model pressure and velocity fields around buildings in a test area is presented. In most studies, proposed results are only for one building only and the wind regime do not account for the topography around. This present article, wishes to propose a full scale 3D simulation of the pressure, turbulence and velocity regimes around multiple buildings in the

same city whilst accounting for topography and canopy information of the region. Further to being a tool to uncertainty prediction in wind modeling, this highly complex CFD (computational Fluid dynamics) model proposes vortex modeling in high gradient regions. These are, moreover, compared and calibrated using a Matlab model.

Keywords: Wind modeling, turbulence, tall building, simulation, insurance policy

1. INTRODUCTION

Wind is a highly complex and variable phenomenon as it interacts at all times with structures, gets modified according to different canopy and is inherent of turbulence of vortices in many cases. Modeling of the effects of gusts on structures is uncertain and hazardous due to turbulence induced rotational eddies. This turbulence nature tends to be more important in the lower atmosphere as the wind interacts with rough ground and decreases with height. Therefore, slender skyscrapers are often subjected to variable winds both in time but also in altitude. The lower parts of the buildings are subjected to slower but highly turbulent winds while the uppermost parts are subjected to higher speed but less turbulent winds. These variable characteristics of wind make it difficult to model and impose dangerous dynamic loads on the buildings. In case of significant eddies as compared to the building size, well established pressures develop in these regions whereas for small eddies (as compared to the building size), variable uncorrelated pressures develop with distance separation. These effects bring risks of occurrence of very dire aeroelastic effects like flutter. In other words, if there is a positive retroaction of the turbulent dynamic loads with the elastic response of the buildings, the latter may start to oscillate with expansive amplitude until failure. One such example is the Tacoma bridge failure in 1940 at wind speed of only 19 m/s. Bridge failure as a result of resonance was modeled by D.Ramdenee et al. of the WERL team in [1]. We can, hence, clearly see that the need to properly model aeroelastic and aerodynamic effects on building is very important. The breakthrough of our work is that we make use of very precise CFD methods coupled with complete consideration of the terrain aerodynamics around to find very accurate results. The terrain aerodynamics consideration enables us to set as realistic as possible wind regimes in the computational domain. Furthermore, Matlab support enables us to compare and calibrate the model via vortex shedding consideration. Such precise analysis can be very interesting tools to mend knowledge around reliability and serviceability of buildings so as to improve building norms and adjust the insurance policies around buildings.

2. WIND CHARACTERISTIC

Loads modeling (pressure and wind speed) on buildings require accurate modeling of approaching wind. However, such is quite difficult and in many cases require time and money costly experiments. For example, as we can see in a study by Deaves and Harris (1978) [2], the extensive full scale data and classic logarithmic law based wind model in neutral stability conditions has been the fruit of very tedious work. It will be very cumbersome as method to build a model for different places using such methods. Such models are expressed in the following equation with specific coefficients A, B, C, D and E.

$$\overline{V_z} = \frac{u^*}{A}\left[ln\left(\frac{z}{z_0}\right) + B\left(\frac{z}{z_g}\right)^1 + C\left(\frac{z}{z_g}\right)^2 + D\left(\frac{z}{z_g}\right)^3 + E\left(\frac{z}{z_g}\right)^4\right] \quad (1)$$

Where

$\overline{V_z}$ is the hourly mean wind velocity at a height of z measured in ms^{-1}

u^* is the friction velocity and is given by:

$$u^* = \sqrt{\frac{surface\ friction\ shear\ stress}{atmospheric\ density}}$$

z_g is the stretch version of gradient length in *m*

z_o is the average ground roughness in *m*

The above model allows modeling of the wind profile according to different altitudes. However, it does not take into account canopy, atmospheric thermal stratification (only significant for very tall buildings) and roughness effects which are very important. In our model, the wind profile is inherent of all these characteristics via modeling of wind profiles within buildings region from far unperturbed boundary winds in Windmodeller [3] software. The output wind profile expressions (inherent of all the above described phenomena) are used as boundary inlet conditions in modeling the wind profile around the buildings.

3. PRESENT CONSTRUCTION FRAMEWORK AND POLICIES RELEVANT TO WIND LOADS

Studies conducted by the Division of Building Research, National Research Council of Canada, in an attempt to rationalize the calculation of wind loads on buildings are exclusively based on generalization of smooth wind tunnels experimental results. These allow us to define potential hazards and a broad range of domains where the latter might appear. However, no tools have been developed that can actually model with satisfactory precision the pressure distribution around buildings, wakes effects, interaction of multiple flows due to buildings and acceleration in corridors. In many cases, as design criteria for buildings and to evaluate failure probabilities, static analysis is made use of. This method uses the criteria of the peak pressure to calculate classic structural failure like stress, shear or torsion. The peak pressure is taken to be the product of the gust dynamic wind pressure and the mean pressure coefficients. The mean pressure coefficients are calculated in wind tunnel experiments such that the values are specific to a given context. This kind of analysis, furthermore, assumes fully correlated progression of the variations in the upwind velocity such that a peak wind speed value will automatically imply a peak value in the pressure or load on the structure. Such kind of analysis is used despite surprisingly low level of accuracy because it offers simplicity, continuity with previous practice and allows direct use of existing meteorological data on wind gusts. However, the method is not suitable for tall structures or those with significant dynamic response. Furthermore, the near quasi-steady assumption fails in many cases where the mean pressure coefficient is near zero. Also, this assumption fails in many cases of vortex shedding whereby a rapid rotational phenomenon triggers a short lived but high gradient low pressure region (disastrous for claddings) or rapidly changing wind velocities (considering magnitude and direction) which interferences in a positive retroaction with the buildings structural dynamics. The actually used models are, thus, clearly, insufficient to adequately model and provide guarantee on the ability of building to withstand different wind regimes. Hence, the uncertainty in relevant wind induced hazards on buildings make it difficult for completely relevant insurance policies to be applied or construction framework devised. The wind association for Wind Engineering makes it clear that there is need for improvement in this field. Advancements in proper wind modeling and effects on structures will, further, enable different stakeholders to assess risks for potential projects and increase precision in cost management analysis. [3 to 9] provide further details of limitations prevailing in the framework of wind loads effects on buildings.

4. METHODOLOGY ADOPTED IN THIS WORK

In this work we wish to propose a generic tool capable of modeling wind distribution (pressure and velocity) around tall structures (in this case building). The idea behind is to propose results that can allow us to evaluate potential hazards and hence accordingly set insurance policies and construction norms. In further work, aeroelastic modeling will be performed. In this paper, the final simulation permits the definition of a tool that allows simulation of wind field distribution over a city skyline. The simulation is 3D and is transient. The reasons for modeling a whole city and in transient state (with time dependent wind) follow: the closeness of tall buildings allows us to see if the tool satisfactorily model wind acceleration in the corridors and interaction of wakes with neighboring buildings. In many present tools and studies such phenomena are not taken into consideration. Transient state modeling allows us to see the presence of vortex shedding as the wind changes its profile over the buildings. Furthermore, to these rotational phenomena, analysis of changing pressure field allows us to predict hazards for claddings or risk of positive retroaction with the buildings structural dynamics. Such study has been done through successive steps whilst calibrating and sustaining our CFD model. In a preliminary study, the domain calibration, mesh optimization and turbulence model calibration was performed using a steady flow simulation on a single building. The idea behind has been to simulate a most common flow on simple structure. This simulation will allow us to see if the domain is sufficiently large to prevent interaction of the boundary walls with the simulation (creating artificial vortices or pressure gradients), to refine the mesh size and type until any additional change has no or very insignificant change on the results and a study of the results to choice which turbulence model better performs in such cases. In a second simulation, a transient flow modeling was performed on the same building with optimized domain, mesh and turbulence model properties. In this case, we wish to see if our model offers precise results with transient modeling. In order to validate such, we have set up a Matlab code capable of calculating the pressure changes and simulate the vortices in the rear of the buildings according to the time dependent wind. Comparison of the CFD obtained vortices and those by the Matlab code will enable us to see if the tool performs well. Once the model having been calibrated and supported, we performed a transient wind distribution simulation of wind field around buildings in a city for a given wind regime. Finally, we add the model additional precision from Windmodeller software (topography, canopy and wind profiles) to add realism to our model and propose a very accurate tool.

5. DOMAIN, MESH AND TURBULENCE MODEL CALIBRATION

The calculation domain is defined by a cuboids of length L, height H and with W. This was inspired from works from both Bhaskaran presented in the Fluent tutorial and from Nathan Logsdon [10-14]. As an objective we only vary L, H and W to see how the distance between the boundaries limits and the building influence the results. As these three parameters will vary, the number of elements will also vary. In order to define the optimum calculation domain, we created different domains linked to a preliminary arbitrary one by a homothetic transformation with respect to the centre of the cuboids. For the city a 1:400 scale was used on a city of 5 skycrapers. The average size of the real skyscrapers is 40m by 40 m by 300m and the size of the city (horizontal plane) 1000m by 1000m. The size of the domain and other relevant sizes are illustrated in table 1 below:

Table 1: Relative sizes of real entities and CFD model in optimised domain

Entities	Real life Size (m)	CFD model size (m)
Average length of building	40	0.1
Average width of building	40	0.1
Average height of building	300	0.75
Length of city	1000	2.5
Width of city	1000	2.5
Length of domain	4000	10
Width of domain	1400	3.5
Height of domain	400	1

The working domain and buildings are shown in figure 1 below.

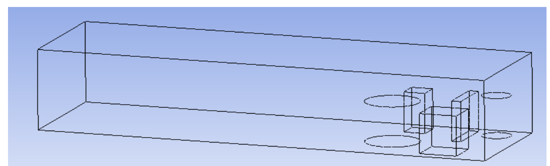

Figure 1: CFD model and domain of wind flow over city

For the mesh optimization, the one building steady flow simulation was run with increasingly refined meshes until the results changed only insignificantly. We, hence, made use of mesh sizes as defined in table 2.

CFX proposes several turbulence models for flow over structures resolution applications. Documentations from [15] advise the use of three models for such kind of applications namely the k-ω model, the k-ω BSL model and the k-ω SST model. The Wilcox k-ω model is reputed to be more accurate than k-ε model in the near wall layers. It has been successfully used for flows with moderate adverse pressure gradients, but does not succeed well for separated flows. The k-ω BSL model (Baseline) combines the advantages of the Wilcox k-ω model and the k-ε model but does not correctly predict the separation flow for smooth surfaces. The k-ω SST model accounts for the transport of the turbulent shear stress and overcome the problems of k-ω BSL model. The k-ω SST model has been chosen as previous works and analysis like [14] leads to us to believe that this turbulence model will better model intrinsic effects like turbulent shear stress transport along the walls of the structures.

Table 2: Table of optimized mesh size

Description	Value (m)
Size of elements along the buildings	0.001
Size of first element in the boundary layer	0.00008
Size of the elements on the boundary limits	0.3
Number of layers in the boundary layer	17
Inflation factor in the boundary layer	1.19
Inflation factors near the boundary limits	1.19

6. TRANSIENT MODEL ANALYSIS WITH ONE BUILDING

Another simulation was performed: A transient flow simulation was run over a single building using the same domain, mesh and turbulence model as in the previous case. The wind profile used only varies in time and not in space. This simulation was performed in an attempt to validate our model in transient state. As no particular experimental validations exist, we have modeled an identical transient flow simulation in Matlab. The aim has been to be able to see the same vortex shedding in the rear of the building in the two simulations thus confirming same transient pressure distribution results and velocity field. This will validate the capacity of our model to simulate transient flow over buildings. To do so, in an attempt to see the vortices very well, the domain size was reduced. The length, width and height of the CFD domain were made very close to that of the building. The simulation resembled a flow in a duct with a barrier as the building. Such was done to increase the presence of vortices (more visible) and ease the programming of a Matlab code. The size of the used domain was 4m*0.2m*1m. The building was at the centre of the domain. In the Matlab code, the Euler equations for the two dimensional inviscid flow are written in the integral form for a region Ω with the boundary δΩ as :

$$\frac{\delta}{\delta t}\iint w dx dy + \oint (f dy - g dx) = 0 \qquad (2)$$

Where x and y are Cartesian coordinates and:

$$w = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix}, f = \begin{pmatrix} \rho \\ \rho u^2 + p \\ \rho u v \\ \rho u H \end{pmatrix}, g = \begin{pmatrix} \rho v \\ \rho u v \\ \rho v^2 + p \\ \rho v H \end{pmatrix} \qquad (3)$$

p, ρ, u, V, E and H denote the pressure, density, Cartesian velocity components, total energy and total energy considered in our situation (as in the CFD modeling) as an ideal gas. The discretization procedure follows the method of lines in decoupling the approximation of the spatial and temporal terms. The domain is divided into quadrilateral cells and the system of differential Euler equations is applied to each cell separately. Each quantity, such as u_1 or ρu1 is then evaluated as the average of the values in the cells of the two sides of the face:

$$(\rho u)_1 = \frac{1}{2}(\rho u)_{i,j} + \frac{1}{2}(\rho u)_{i+1,j} \qquad (4)$$

Bounday conditions such as pressure at the velocity at the entry, free slip walls and atmospheric pressure at exit were specified. Improper treatment of the boundary conditions can lead to serious errors and perhaps instability. Fourth order Runge Kutta was used to solve the equations. The used speed was 200*sin (30*t) ms^{-1}. This velocity has very high amplitude and variability. The used velocity (both in Matlab code and CFD modeling) were the same. The pressure in the vortices at the same position (in the wake of the highest region of the building) were compared. The results are illustrated and discussed below.

First of all, we will illustrate the vortices obtained at different time steps via Matalb code as compared to the simulation results for time t=30 seconds on the ANSYS-CFX code. The aim of this is only to see the importance of the pressure discrepancies and vortex presence from a qualitative point of view.

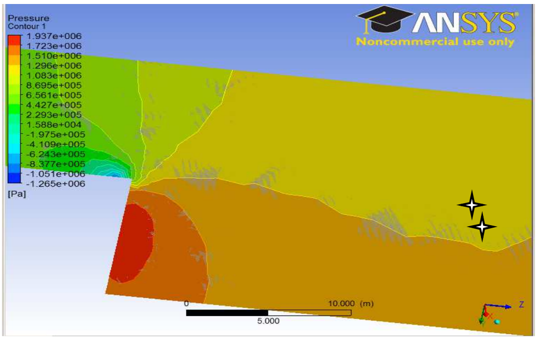

Figure 2: pressure and vortices in the wake of the building

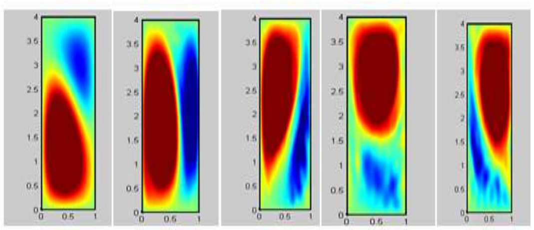

Figure 3: Presence of vortices at different time intervals using Matlab code

The pressures are measured at the two points illustrated by stars in figure 2 at different time intervals in both the CFD based and Matlab based codes. The results are illustrated in table 3.

First of all, we note that both the Matlab code pressure results and the CFD generated pressure results are very close. This is quantitatively supported by extracted results illustrated in table 3. This firmly supports the validity of our tool in transient modeling. Furthermore, we note that, on average the Matlab code generates results which are larger than those generated by our CFD tool. This was expected as our Matlab code did not cater for dissipative terms such that a slight over prediction of pressure was anticipated. The dissipative terms suppress the tendency for odd and even point decoupling and prevent the appearance of wiggles in regions containing severe pressure gradient.

Table 3: Pressure at different points by Matlab model

Time (s)	Pressure at point 1 obtained via ANSYS (Pa)	Pressure at point 1 obtained via Matlab (Pa)	Pressure at point 2 obtained via ANSYS (Pa)	Pressure at point 2 obtained via Matlab (Pa)
0	1010000	1333200	1010000	1515000
0.5	1240071,84	1785703,45	1240071,84	1612093,39
1	660434,411	911399,488	631188,675	315594,338
1.5	1311049,67	1586370,1	1311049,67	1678143,57
2	902158,002	1055524,86	893135,608	1178939
2.5	872802,857	1152099,77	872802,857	1056091,46
3	1326296,02	1685722,24	1352758,32	2029137,48
3.5	666624,617	859279,131	666624,617	791283,42
4	1215420,24	1391899,26	1232606,33	1599923,01
4.5	1041264,84	1500670,89	1041264,84	1468183,43
5	757076,719	1022053,57	735916,377	838944,67

7. TRANSIENT MODEL ANALYSIS ON THE WHOLE CITY USING WINDMODELLER IN BOUNDARY CONDITIONS DEFINITION

The previous sections of this article have pondered on the importance and relevance of making dynamic aerodynamic over large structures in an attempt to evaluate risks of aeroelastic effects and other dynamic hazards. The aim of this study is to verify the ability of CFD models to predict pressure distributions for different wind flow regimes over large structures so as to better support the implementation of reliability factors and insurance policies. A CFD tool has been developed at the WERL to predict such. The article, has till now focused on specific cases in an attempt to calibrate and validate the model. This achieved, we will now illustrate the model as such. The model can use both as input user defined winds (e.g. 5 ms^{-1}) or very accurate wind profiles defined using Windmodeller software. Windmodeller software has uses as boundary conditions velocities at different angles defined at several particular altitudes. The software then models the velocity profile while taking into account, thermal stratification, the topography, roughness and canopy. In modeling the pressure and velocity distribution in a particular city, it is interesting to use as boundary condition, the velocity or velocities from a region of low velocity gradient and variability, for example having winds from the sea or large water systems where the velocity is quite constant. These velocities are used as boundary conditions in Windmodeller to model the wind profile in the vicinity of the city. The flow of the air is transported by the software till the city region and the output is a set of velocity profiles which can be then used in the CFD model to calculate the pressure and velocity field in the city. Figure 4 below shows the velocity input in a region of low wind variability in Windmodeller. These velocities will be transported into the inner circle as illustrated in figure 4 which represents the outskirts of the city. Therefore, the input boundary conditions of the city are, in fact, the output from Windmodeller which define high accuracy wind profiles that take into account the topography, roughness, canopy and thermal stratification.

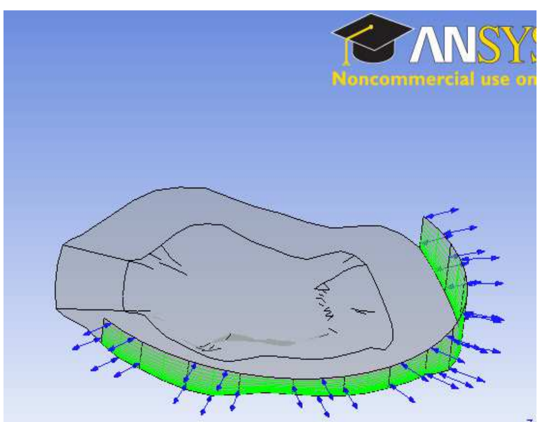

Figure 4: Boundary conditions treatment in Windmodeller

We used our model to simulate the pressure distribution around a hypothetical city comprising of 5 skycrapers. We will present results of the modeling and discuss that intrinsic observed phenomena and improvements that need to be added to the CFD model.

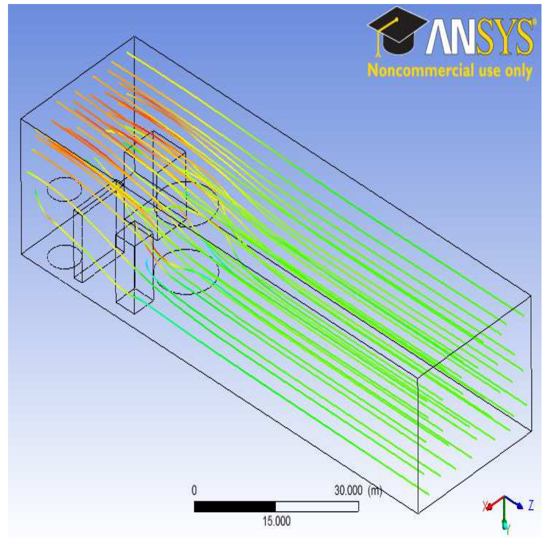

Figure 5: 3D modeling of the wind distribution over the hypothetical city

Figure 6 below shows velocity fields in different horizontal planes at different altitudes. From these illustrations, we can clearly see the variation of the velocity near the buildings, rapid retardation, general acceleration, corridor acceleration and wake phenomena.

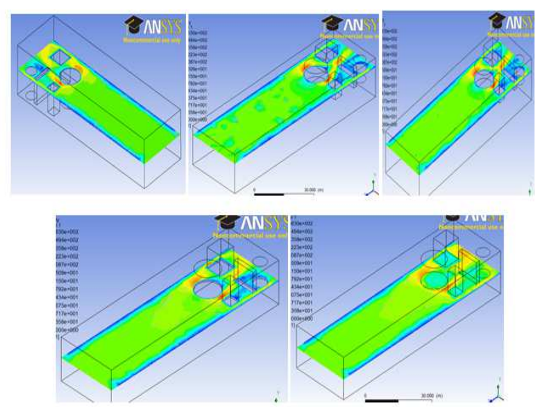

Figure 6: 3D modeling of the wind distribution over the hypothetical city at different horizontal planes

The different illustrations in figure 6 shows the wind velocity field for t=5s at different altitudes. We observe that in all cases, we see acute corridor acceleration between the buildings.

These zones (red color) represent zones of low pressure and even negative pressure which can be dire for the cladding. Furthermore, we note a pressure disparity at different levels for the same horizontal coordinates. In case of intensive winds of high variability such pressure disparity may bring rapidly changing dynamic solicitations on the building and, in case of positive retroaction, cause flutter like events.

An analysis of the transient behavior of the wind was performed on a vertical plane passing through 2 skyscrapers at the same time. The idea behind was to see if there is a significant variation of the pressure distribution for a wind varying within Windmodeller boundary limits as follows: at t=0 s, v=50 kmh^{-1} ,at t=5 s, v=100 kmh^{-1} and at t=10 s, v=150 kmh^{-1} . All the velocities have been entered at a reference height of 10 m and a direction along the length of the domain. The obtained pressure distribution profile is the same for the different velocities; however, the quantitative data are different. Therefore, only the pressure distribution illustration for t=10 s is shown in figure 7 to avoid redundancy.

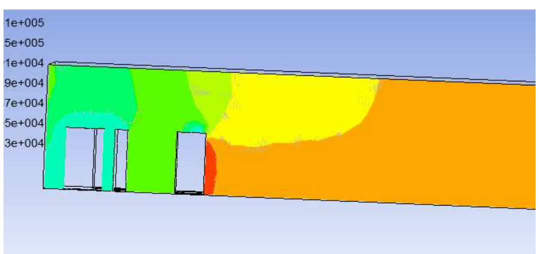

Figure 7: Pressure distribution on a vertical plane at t=10 s

We note from figure 7 that the pressure disparity between the front and the back of the building facing first the wind is quite important. This will create an appreciable torque on the structure. Rapidly changing wind direction, but with similar magnitude (in case of cyclones) will cause the torque to change direction very often and might lead to some positive retroaction of such movements.

8. CONCLUSION

In this article, we have proposed a CFD tool capable of simulating the pressure, velocity (and any other relevant parameters like turbulence intensity) fields. The importance of such modeling has been underlined and attributed to the need to better quantify the reliability and serviceability of structures when it comes to dynamic solicitations. This tool can be used to analyze the pressure distribution according to different winds and evaluate the loads and risks of failure or damage on cladding. In future work, we would like to extend this analysis to an aeroelastic one where the tool would be able to directly evaluate the structural response of the building according to the aerodynamic loads. Nowadays static models that use peak pressures are mostly used. This paper shows the possibility of using breakthroughs in wind modeling to better predict building serviceability and set a framework for insurance policies. However, this model only predicts the pressure distribution around the buildings but does not predict the dynamic and elastic behavior and response of the buildings when subjected to such pressure differences. In future work, the building should be designed with parameterized soil fixture, material properties and elastic nature and couple the purely fluid simulation with a structural simulated with ANSYS-CFX dual simulation.

REFERENCES

[1] D.Ramdenee et al. " Static, dynamic and spectral response of a suspended bridge under known solicitations" Processing, Performance and Failure Analysis of Engineering Materials conference, Luxor, Egypt. Nov 2011

[2] Deaves D.M., Harris, R.I. (1978) A mathematical model of the structure of strong winds. Report No. 76. United Kingdom Construction Industry Research and Information Association.

[3] D. Ramdenee et al. "Modeling wake effects on wind farm for inefficiency Assessment using computational fluid dynamics" ASME 2012 International Mechanical Engineering Congress & Exposition IMECE2012 November 9-15, 2012, Houston, Texas, USA

[4] National Institute of Building Sciences (NIBS). 2006. Whole Building Design Guide.

[5] National Weather Service (NWS). 2005. Tropical Cyclone Report for Hurricane Katrina. National Hurricane Center. December 20, 2005. 2006

[6] FEMA. 2006. Recommended Residential Construction for the Gulf Coast: Building on Strong and Safe Foundations, FEMA 550. Publication available May .
[7] Sachs, P., “Wind forces in engineering”. Pergamon Pres, 1978.
[8] Saunders, J.W., and Melbourne, W.H., “Tall rectangular building response to cross–wind excitation”. Proc. 4th International Conference on Wind Effects on Buildings and Structures, Cambridge University Press, September 1975, pp. 369–379.
[9] Saunders, J.W., and Melbourne, W.H., “Buffeting effects of upwind buildings”. 5th International Conference on Wind Engineering (Fort Collins), U.S.A. Pergammon Press, 1979.
[10] Nathan Logsdon, Dr. Gary Solbrekken, A Procedure For Numerically Analyzing Airfoils And Wing Sections, doctoral diss., University of Missouri – Columbia, December 2006.
[11] T. Tardif d‟Hamonville, A. Ilinca Modélisation et Analyse des Phénomènes Aéroélastiques pour une pale d’Éolienne. Master s Thesis, UQAR/LREE
[12] T. Tardif d‟Hamonville, A. Ilinca. Modélisation de l‟écoulement d‟air autour d‟un proil de pale d‟éolienne,Phase 1: Domaine étude et Maillage. Rapport Technique, UQAR/ LREE-05, Décembre 2008
[13] T. Tardif d‟Hamonville, A. Ilinca. Modélisation de l‟écoulement d‟air autour d‟un proil de pale d‟éolienne,Phase 2 : Modèles Turbulents. Rapport Technique, UQAR/ LREE-06, Janvier 2009.
[14] T. Tardif d‟Hamonville, A. Ilinca. Dynamic stall and transitional turbulent model, Technical Report UQAR/LREE-05, Décembre 2008
[15] ANSYS CFX, Release 11.0