CFD open-source code validation for fluid-structure interaction in building analysis

Valerio S. Ameida[1], José E. S. Montiel[2], Edson B. Ferreira[2], Fernando A. Kurokawa[2]

[1]*Dept. of Structural and Geotechnical, Escola Politécnica of Universidade de São Paulo*
Av. Professor Almeida Prado, 83, 05508-070, São Paulo, Brazil
valerio.almeida@usp.br
[2]*Dept. of Construction Engineering, Escola Politécnica of University of São Paulo*
Av. Professor Almeida Prado, 83, 05508-070, São Paulo, Brazil
jose.montiel@usp.br, bispo@usp.br, fernando.kurokawa@usp.br

Abstract. With the advancement of computational technology and numerical techniques, Computational Fluid Dynamics (CFD) has been playing a fundamental role in solving problems of wind-structure interaction. Aiming at the optimization of the structural design from the reduction of the costs of experimental tests directed with computer simulations, this article presents an analysis of the feasibility in the use of an open access CFD tool, OpenFOAM, for the calculation of the dynamic pressures due to the action of the wind in buildings. CFD simulation was coupled in conjunction with the structural analysis model to evaluate the efforts and displacements by using the Finite Element Method.

Keywords: computational fluid dynamics, finite element method, fluid-structure interaction, OpenFoam.

1 Introduction

In context of optimization search, regarding the structural design and dimensioning, the more realistic evaluation of the wind action and its interaction with the structures is very significant at each structural element response generated by the dynamic pressure effects. Li et al. [1] developed a simple expression, generated by wind tunnel tests, to evaluate coefficients for the wind-induced torques on L-buildings. Li et al. [2] presented a study that evaluate experimentally the surface pressure schemes for non-uniform buildings in real problems, Huang et al. [3] presented a computer vison-based vibration measurement method for wind tests of aeroelastic high-rise building models and Zou et al. [4] investigated torsional aeroelastic effects on high-rise buildings due to forced vibration wind tunnel tests. However, there are some disadvantages of applying these experimental tests, difficulty of considering some physical parameters like temperature gradient, difficulties in accuracy data measurements, scale effects, high cost for operation and demands a long time for execution.

With the difficulties in clarifying the complex 3D flow fields around a building using wind tunnel procedures and the growth of high performance and low cost of computers, the computational wind engineering (CWE) paradigm has become a powerful field of studies in fluid-structure interaction, where for the 3D flow fields, it is mandatory to understand both numerical analysis and turbulence theory. In this scenario, the Computational Fluid Dynamics (CFD) methodologies coupled with some potential and flexible method for evaluating the behavior of deformable solids, like the Finite Element Method (FEM) or Boundary Element Method (BEM) are the most profitable numerical tools for evaluating the combined spatial time PDEs in the complex above-mentioned problem and, for general application, they are superior than analytical or experimental procedures of analysis, where the term is Fluid-Structure Interaction (FSI).

The vast majority of CFD system available with these foregoing characteristics are commercial/licensed software with have high cost and "closed access", that is, do not allow the owner to make any change into the source code, as introducing a new experimental turbulence model or coupling external FEM that simulates the deformable building for aeroelastic effect assessments. In this sense, the objective of this article is to analyze the feasibility of using OpenFOAM for the calculation of dynamic pressures due to the action of wind in buildings, where it is calibrated and compared the CFD analysis with experimental model by wind tunnel database developed by Tokyo Polytechnic University (TPU). In addition, it is coupled this CFD open-source in conjunction with a homemade system via FEM based on displacements methods to evaluate the structural response. The FEM is presented in dynamics formulation with the implicit Houbolt method [5] for the solution of the motion equations, and Euler-Bernoulli beam theory is considered for the spatial frame elements for simulating beams and columns, which is the most adequate structural elements applied for design building.

2 Methodology

The motivation for the development of this work was the need to optimize the structural design from a reduction in the costs of experimental tests directed with computer simulations, using open access CFD and FEM tools. The code chosen for processing the CFD simulations was OpenFOAM, version 9.0, which has pre and post processing to solve problems of mechanics of continuous media. This simulation system is a non-commercial (free) tool in which it has C++ source code, the main reason for this choice, as it allows the user to modify, implement or create custom objects such as boundary conditions, turbulence models, solvers, coupling with an external physical model, etc.

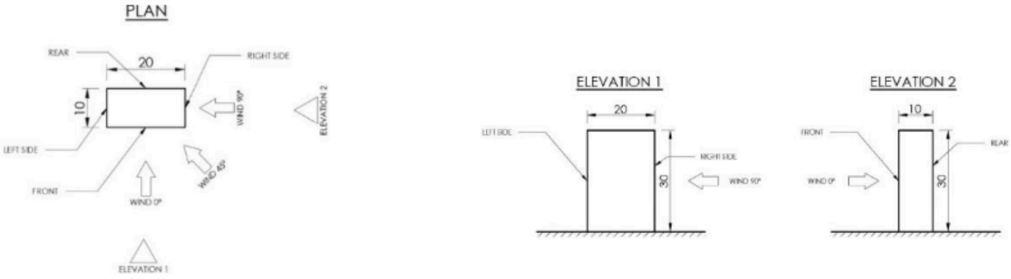
2.1 Governing equations and computational approach

The OpenFOAM system recurs to Eulerian approach, in which fluid is treated as continuum movement as expressed by mass conservation and Navier-Stokes equations. By considering that wind ow is turbulent, the choice for a turbulence model is mandatory. In the OpenFOAM tool, several turbulence models of the type RANS (Reynolds Average Navier Stokes) and LES (Large Eddy Simulation) are available. These models cover a relatively large set of flow problems that include time-dependent variables, fluid-structure interaction and heat transfer. The turbulence model adopted to perform the simulations in this work was the standard $\kappa - \varepsilon$ in which is classified as two-equations model based on the Boussinesq hypothesis. It is the most commonly used in which presents satisfactory results for fluid flows around obstacles and requires low processing consumption when compared to other turbulence models and is relatively accurate.

In order to simulate the turbulent wind flow, this system of partial differential equations is solved numerically in the OpenFOAM using the finite-volume method (FVM). The method for calculating the numerical solution employed by the tool is Pressure-Implicit with Splitting of Operators (PISO). However, a modification was made to the tool's solver (Piso FOAM) to calculate the numerical solution. The objective was to change the time step on-the-fly based on stability and convergence conditions, that is, to reduce the number of iterations with the automatic adjustment of the time step during the conditioned processing by analyzing the Courant number. The transient terms of equations are approximated by a Euler Implicit method and the central differences scheme (second-order discretization) are used for both the advection and diffusive terms. To solve the linear systems of the coupled equations, the Geometric-Algebraic Multi-Grid (GAMG) iterative method with Gauss-Seidel scheme was adopted for the pressure equations.

2.2 Domain dimensions and initial and boundary conditions

The building adopted for the computer simulation has 10 floors considering the floor-to-floor distance of three meters. The building has a ratio of width and depth of 1:2 and a ratio of depth and length of 2:3 according to Figure 1.



(a) (b)

Figure 1. Building dimensions in (a) plan and (b) elevation. Units: meters

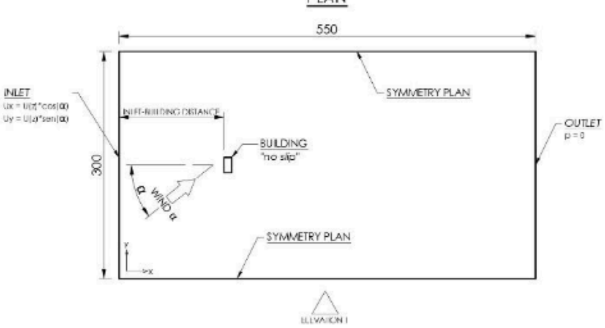


Figure 2. Boundary conditions in plan. Units: meters

The correct choice of dimensions between the domain boundary and the building surfaces is extremely important for the simulation, where the flow can propagate without causing neighbor influence. For initial conditions, the Dirichlet condition is used for u and p. In the case of turbulent kinetic energy (κ) and turbulence dissipation rate (ε), the initial conditions are estimated in the same way as they are done in several works in the literature. In this work, three types of boundary conditions were considered: inlet, outlet and rigid-wall boundaries. At the inlet section (fluid entrance), the wind velocity profile $u(z)$ is imposed. At outlet section (fluid exit), it is imposed homogeneous Neumann (fully developed flow) conditions for all variables. For top and lateral planes, it was adopted the symmetry boundary condition. The rigid walls and ground were modeled with boundary condition without slip (no-slip) i.e, zero velocity conditions were used in these contours (Figure 2). In addition, to avoid using too many grid points in the viscous sub-layer, the wall function is used near the walls for the turbulent variables κ and ε.

2.3 Calculation of the pressure coefficient through CFD and experimental model by wind tunnel

The pressure distribution on the model surface can be represented in terms of the pressure coefficient given by:

$$C_{pe} = \frac{P_x - P_\infty}{P_d}, \tag{1}$$

in which P_x represents the relative pressure of a given point on the building facade, P_∞ is the freestream undisturbed pressure and P_d is the freestream dynamic pressure defined by: $P_d = 0.5\rho V_r^2$. The static pressure sensor used to monitor P_∞ was positioned on the upper part of the domain at the beginning of the inlet section of CFD model.

An aerodynamic database was developed by Tokyo Polytechnic University as part of the Effects of Wind on Buildings and Urban Environment, the 21st Century Center of Excellence Program, funded by Japan's Ministry of Education, Culture, Sports, Science and Technology. The objective of this database is to provide wind tunnel test parameters in several different geometries of high and low buildings, in addition, providing results on the effects of buildings in the neighborhood. All the necessary statistical values for the pressure coefficients were tested for 394 cases. These values include statistical data to be applied in the boundary conditions, graphs and temporal series for the experimental model. These data were used to compare with the results of the CFD simulations. Since many low-rise buildings are located in suburban areas in Japan and other countries, suburban terrain, corresponding to terrain category III of the AIJ [6], was chosen as the tested wind field. This category has an average exponent of the wind velocity profile of 0.20 m/s and a gradient height of 450 m. The test was carried out with turbulence-generating towers, roughness elements and a floor mat upstream of the wind tunnel test section. In addition, the wind tunnel testing laboratory at Tokyo Polytechnic University carried out essay with angles of 0° to 100° directions for each 5° interval.

2.4 Structural model and fluid-structure interaction coupling

The building is formulated by using spatial frame finite elements, without considering slab rigidity, by the classical beam engineering theory (Euler-Bernoulli hypotheses) in linear geometrical and material theory. The

FEM concepts can be generated by Galerkin method [7], and considering inertia forces, then the governing equations for the dynamics system are written in matrix notation as:

$$K\mathrm{x}_n + C\dot{\mathrm{x}}_n + M\ddot{\mathrm{x}}_n = F_n \quad (2)$$

with n being the total number of degrees of freedom, K, C and M, indicate, respectively, stiffness, damping and mass matrices [8]. The terms $\mathrm{x_n}$, $\dot{\mathrm{x}}_\mathrm{n}$ and $\ddot{\mathrm{x}}_\mathrm{n}$ are nodal displacement, velocity and acceleration vectors, respectively, and F_n is the nodal force vector that contains permanent loadings and wind forces that are automatically updated with the CFD analysis. Equation (2), for generical applications, for example, nonlinear formulation, must be solved by direct integration methods that use step-by-step numerical integration via finite differences scheme. It is applied the unconditionally stable Houbolt method [5] with implicit integration schemes which considers cubic curves for $\dot{\mathrm{x}}_\mathrm{n}$ and $\ddot{\mathrm{x}}_\mathrm{n}$ in time $t_{n+1} = t_n + \Delta t$ where $t_{i-2}, t_{i-1}, t_i, t_{i+1}$ and t_{i+2} are the time considered, with i being the present time and Δt the time step.

The fluid pressure state (p) at a point acting on the external boundary of the structure $(\bar{S})$ generates traction $(\bar{t})$ component which is given by $\bar{t} = p\eta$ where η represents the components of the direction cosines, in matrix notation. The nodal force can be expressed by the classical FEM relation:

$$F_n = \int_{\bar{S}} N^T \bar{t} d\bar{S} + \int_{\hat{S}} N^T q d\hat{S} \quad (3)$$

where N^T is the transpose vector of the shape functions; q is the permanent loading vector applied in all the boundary building ($S = \bar{S} \cup \hat{S}$), with $\bar{S}$ and $\hat{S}$ being the boundary on the facade and in the whole building.

The present multiphysics FSI problem is evaluated with the partitioned treatment approach, following the Felippa et al. [9] definition, which describes the advantages of customization, independent modeling, software reuse and modularity facilities, in that can maintain the OpenFOAM and structure FEM programs decoupling but developing intercommunication codes to connect them. The flowchart depicted in Figure 3 resumes the simple but effective procedure applied in this paper for FSI using OpenFOAM and a homemade structural FEM program.

In fact that in Brazilian code requirements demands for design building small horizontal displacements and low frequency vibration limits, less than 1 Hz, aeroelastic analysis is not taken into account herein. In consequence, the building deformable configuration is not updated along the time step to the fluid scheme. Therefore, it is simulated by rigid solid but the whole pressure actioning on the facade building is stored at each time and then after being pos-processed in the structural FEM formulation. The present monoway interaction makes the FSI independent, where the building design can be revaluated as many times as necessary, without CFD coupling again.

3 Numerical results and analyses

3.1 Validation: CFD simulation

In order to validate the building model with CFD methodology, four meshes (A, B, C and D) were designed, structured with hexahedral elements with 74850, 153708, 224448 and 304320 elements, respectively. The positioning of building inside of domain must ensure that the results of the CFD simulation are not influenced by the domain boundaries. The distance between the upstream building and the domain inlet should not be too small, but it should not be too large due to the computational cost for processing the problem. Thus, to guarantee a good numerical solution with lower computational cost, the optimal distance considered after several tests was of 145 meters.

The inlet velocity profile $u(z)$ of the simulation domain corresponds to the same velocity profile used in the Tokyo Polytechnic University experimental test. This experimental test was used with the wind direction at zero degrees in relation to the axis orthogonal to the front facade of the building. The total time taken for each simulation was $t = 120\ s$ and the maximum Courant number was limited to 0.5 and the time step was initialized to 10^{-4}. The tolerance of the algebraic system solution for the variables u, v, w, p, κ and ε was $10^{-\varepsilon}$. It is also observed that, as the mesh is refined, the numerical solution converges to a solution very close to the experimental one. This is identified by calculating the relative error of the C_{pe} temporal average in various positions of the front facade, as shown in Tables 1.

Table 1. Relative error of the temporal average of C_{pe} for four mesh

Mesh	Horizontal position [m]:					
	5	**7**	**9**	**11**	**13**	**15**
A	18.92%	14.29%	12.82%	11.69%	12.00%	14.29%
B	9.46%	7.79%	6.41%	5.19%	5.33%	4.29%
C	6.76%	5.19%	3.85%	2.60%	2.67%	1.43%
D	2.70%	1.30%	0.00%	1.30%	1.33%	2.86%

It can be inferred that the mesh D (304320 elements) presented consistent results in relation to the experimental test, with a relative error (average) of 1.6%, therefore the refinement used in the following simulations had a minimum order of 3×10^{-6} elements. A qualitative comparison is depicted in Figure 3, showing that the most relevant level with the greatest pressure variations occurs at two thirds of the total height of the building, the level being used for quantitative analysis.

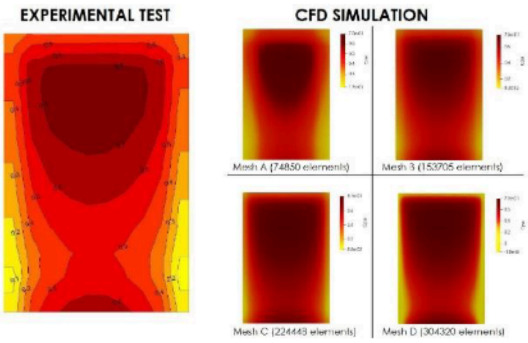


Figure 3. Isovalue curves of C_{pe} in front of building, obtained from CFD simulations for all the present meshes and the experimental test

3.2 Wind simulation at different angles of attack

The wind incidence angle (α), measured between the wind direction and the largest side of the building, can change the criticality of the structure in a given hypothesis. Thus, it is essential to perform several cases of wind incidence with a certain range of parameter α. The simulations were performed considering four angles of wind incidence: 0°, 30°, 45° and 60°. Boundary conditions in these cases are the same as those applied in the previous simulations for model validation Sect. 2.2. The distance between the upstream building and the domain inlet was considered to 145 meters and total time of 120 seconds.

Tables 2 shows the calculation of the absolute and relative error of the C_{pe} temporal average obtained by the CFD simulation in the four wind incidences. It should be noted that the experimental results on the front facade with wind in the 60° direction are close to zero or equal to zero (according to position $x = 5\ m$). In this way, the relative error can be high even that absolute error is low, as occurs in the position $x = 7\ m$, for example, which the experiment resulted in 0.05 and the CFD simulation resulted in 0.06. In this case, the absolute error is 0.01, which indicates consistent results, despite the relative error (average) being 12.5%.

Table 2. Relative error of C_{pe} for all wind incidences

α	Horizontal position [m]:					
	5	7	9	11	13	15
0°	2.70%	1.30%	0.00%	1.30%	1.30%	2.90%
30°	0.00%	1.90%	0.00%	1.50%	1.40%	4.10%
45°	4.60%	0.00%	0.00%	2.20%	2.00%	0.00%
60°	-	20.00%	10.00%	6.30%	14.30%	12.00%

3.3 Fluid-structure interaction results

In the present FSI interaction, it is considered zero velocity at the solid-fluid interface on the rigid building surfaces, which indicates no-slip boundary condition. The whole model is not monolithic, but the small deformations occurred in the building take to small modifications in the volume control, which is not considered this variation in the CFD simulation at each time step. Following the procedure presented in the FSI, the deformable building was modeled with the same dimensions as the CFD simulations, with 10 floors with ceiling height of 3 meters each, all columns with dimensions $19\ cm \times 100\ cm$ and beams $19\ cm \times 60\ cm$, permanent loading $q = 10\ kN/m$ applied in all beams.

Figure 4 illustrated the element positions and the wind direction. The properties of the material used are elasticity modulus $E = 28\ GPa$, density $\rho = 25\ kN/m^3$, Rayleigh damping $C = 0.02\ K$, lumped mass matrix, total time of $t = 120\ s$ and time step $\Delta t = 10^{-4}s$. The wind profiles obtained by Standard NBR 6123:1988 model [10] and CFD aerodynamics analysis are depicted in Figure 5.

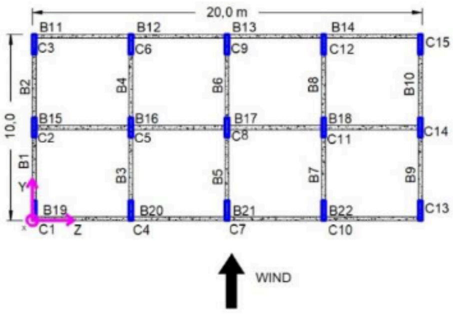


Figure 4. Building plan with wind force direction and local axes

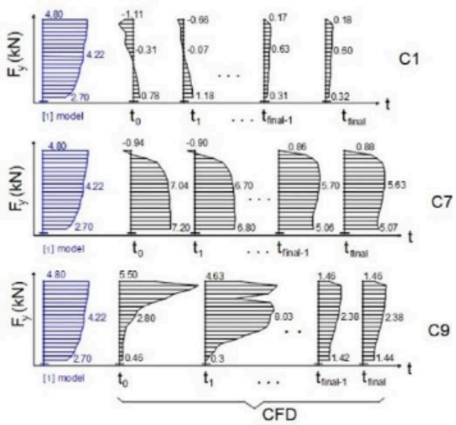


Figure 5. Wind-force time-history profiles on the building facade due to CFD analysis

Internal forces on the base of the columns C1, C7 and C9 are showed in Table 3 where the steady state pressures are used for the final FSI responses. It is noticed that the greatest differences occurred in C1, for normal and M_z moment efforts, with relative difference of 12% and -19%, respectively.

Table 3. Efforts on the base of columns due to wind forces from NBR [10] and CFD steady state analysis.

Column	Normal force N (kN) Code/CFD	Bending moment My(kNm) Code/CFD	Bending moment Mz (kNm) Code/CFD
C1	342.9/383.5	2.1/2.1	120.0/97.2
C8	951.8/951.8	0.0/0.0	136.7/111.7
C9	957.9/917.3	0.0/0.0	131.4/108.5

Finally, the maximum horizontal displacements in direction y, which occurred on the top of the column 7 (C7), are plotted in Figure 6 along the time history for the FSI and the Standard NBR 6123:1988 [10], showing higher value for the last procedure, with a relative difference of 23%, with 1:35 cm for NBR 6123:1988 [10] procedure and 1.04 cm for the FSI coupled program evaluation.

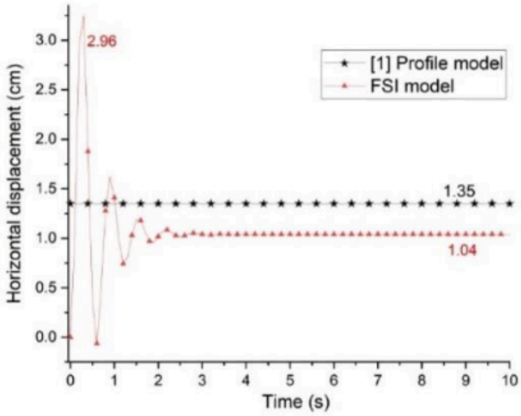


Figure 6. Horizontal displacements using NBR 6123:1988 [1] model and FSI model

4 Conclusions

This work has presented an effective coupling fluid-structure using CFD open-source code, OpenFOAM, and homemade software based on the FEM for the calculation of dynamic pressures due to the action of wind in buildings. In order to validate the building model with CFD method, the numerical results were compared with experimental data by wind tunnel developed by Tokyo Polytechnic University. According to the computed results, the values of the external pressure coefficients on the front and rear façade presented satisfactory accuracy results. It is concluded that the OpenFOAM CFD tool together with the FEM analysis developed herein in this paper is a good and low-cost alternative to estimate the dynamic pressures, efforts and displacements in buildings efficiently and accurately.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.

References

[1] Y. Li, Q. S. Li and F. Chen. Wind tunnel study of wind-induced torques on L-shaped tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 167, pp. 41-50, 2017.

[2] B. Li, L. Wang, Y. Liu and J. Liu. Drag distributions of non-uniform buildings from surface pressure measurements in wind tunnel. *Building and Environment*, vol. 143, pp. 618-631, 2018.

[3] M. Huang, B. Zhang and W. Lou. A computer vision-based vibration measurement method for wind tunnel tests of high-rise buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, vo. 182, pp. 222-234, 2018.

[4] L. Zou, F. Li, S. SoLiang, O. ng. Mercan and T. Shi. Investigation of torsional aeroelastic effects on high-rise buildings using forced vibration wind tunnel tests. *Journal of Wind Engineering and Industrial Aerodynamics, vol.* 200, pp. 104158, 2020.

[5] J. C. Houbolt. A recurrence matrix solution for the dynamic response of elastic aircraft, *Journal of the Aeronautical* Sciences, vol. 17, pp. 540-550, 1950.

[6] Architectural Institute of Japan. Recommendations for loads on buildings: Tokyo, 2004.

[7] B. G. Galerkin. Series solution of some problems of elastic equilibrium of rods and plates (in Russian). *Vestnik inzhenerov i tekhnikov*, vol. 19, pp. 897-908, 1915.

[8] W. Wunderlich and W. Pilkey. Mechanics of structures. Variational and Computational Methods. second edition. CRCPress, 2002.

[9] C. A. Felippa, K. C. Park and C. Farhat. Partitioned analysis of coupled mechanical systems. *Computer Methods in Applied Mechanics and Engineering,* vol. 190, pp. 3247-3270, 2001

[10] D. I. Robinson, H. A. Hassan, Two-equation turbulence closure model for wall bounded and free shear flows. *AIAA Journal,* vol. 115, pp. 147-152, 1998.