

FLUID-STRUCTURE INTERACTION AND AEROLASTIC BALANCE ON THE ANALYSIS OF A TALL BUILDING WITH IRREGULAR GEOMETRY

ABSTRACT

Aims: In this work the effect of wind on a building with irregular geometry is analyzed. Today, many cases of computational fluid-structure phenomena are validated with Benchmark models; nevertheless, the structure analyzed in this work does not have an associated benchmark model. Numerical algorithms applied in this work are also validated with a physical scale model under wind loads on a subsonic wind tunnel.

Study design: Developed procedure involves two coupled field models corresponding to fluid and mechanical phenomenology, each one governed by different methodology: Fluid field associates an Eulerian Configuration, meanwhile solid model is described on Lagrangean configuration. These two models require a third model named ALE to interact each other; this last model is built on the interfaces between them. This work develops the numerical procedure as well experimental ones for solving these complex structures.

Place and Duration of Study: Graduate Engineering Lab. Universidad Autonoma de Queretaro, Mex., Civil Engineering Department. Universidad de Guanajuato, Mex., Design Lab. Construction, Torreon Cohauila, Mex., between September 2012 and July 2014.

Methodology: The numerical model was analyzed through the software ANSYS and also DINES + GiD while the experimental test was performed in a subsonic wind tunnel. The prototype consists of structural steel sections while walls are made of glass panels; physical model belongs to an elastic model where the effect of twisting is not included.

Results: Main results obtained in this work are the bending moment at the basement of the building, the displacement at the top of the structure and the non-uniform pressure distribution on the structural walls.

Conclusion: Dynamic fluid-structure interaction (IDFE) problems have attracted the attention of a large number of researchers. IDFE plays an important role in the structural engineering: on the analysis of high-rise buildings, large bridges, industrial plants, chimneys, transmission towers, etc., all these structures may collapse due to the aeroelastic instability caused by wind effect. The goal of this work is provide a methodology for real structures of complex geometry.

Keywords: *Fluid-structure interaction, Structural Dynamics, Coupled Fields, Aeroelastic Modeling*

1. INTRODUCTION

In recent years various scientific areas have had an important development, one of the areas where an important progress has been seen on the analysis of dynamic fluid-structure interaction (IDFE) problems which has attracted the attention of a large number of researchers [1,2,3]. The IDFE studies play an important role in the structural engineering as on the analysis of high-rise buildings, large bridges, industrial plants, chimneys, transmission towers, etc., all these structures may collapse due to the aeroelastic instability caused by wind effect [4,5]. The complexity of this phenomenology leads to research on experimental procedures (usually in wind tunnels) as well in numerical algorithms through computational fluid dynamics (CFD), taking this into account, two groups of research are clear, the first one

25 addressed to obtain the maximum effects of wind occurring in the structure, while the second
26 focuses on the evaluation of maximum flow effects [6].

27
28 These kinds of problems are treated as coupled models [7], they are multi-physical problems
29 governed by phenomenologies interacting simultaneously [8], Most researchers consider
30 IDFE phenomenology as a case including three main fields: solid (structural), fluid (wind)
31 and interface (border) between domains [5]. The solution of these cases keeps some
32 difficulty because the total domain has to be divided into structural subdomain as well as
33 fluid subdomain which associate different configurations: Lagrangian and Eulerian ones,
34 which are not compatible each other [9]. Therefore generated geometries tend to be
35 complex, whereas the boundary conditions between fluid and structure are nonlinear due to
36 movements of the interface, besides, the position of the interface is part of the solution
37 [10,11].

38
39 The key for solving these problems is a method consisting of coupling these two
40 phenomenologies: structural and fluid dynamics, in addition it must be included movement
41 techniques of nonstructural meshes for the fluids in order to represent the movement of the
42 interface. It is also required stabilization methods and segregation of pressures for the fluid
43 domain as well a block method for solving the coupling, all this together with computational
44 procedure for large scale solutions [12].

45
46 Related to experimental tests in wind tunnels, the aeroelastic "stick" model is popular for its
47 efficiency in the design, manufacture, calibration and measurement procedures. In most
48 structures under wind effects has been observed that the fundamental modes of vibration
49 are the most significant, this feature promotes simplification of the "stick" model by which
50 only two fundamental modes of translation is enough to include for obtaining a direct
51 similarity between the model and prototype structure [13].

52
53

54 **2. MATERIAL AND METHODS / EXPERIMENTAL DETAILS / METHODOLOGY**

55 **2.1 Phenomenological Continuum Equations**

56
57 The equilibrium equation of a continuous medium is given by the following relation (1), while
58 equation of local mass conservation is expressed by the relation (2), [14]:

$$59 \quad \text{div } T + b = \rho \dot{v} \quad (1)$$

$$60 \quad \rho' + \text{div } (\rho v) = 0 \quad (2)$$

61 T is the Cauchy stress tensor, b is the body force field, ρ density field, velocity field v.
62 Moreover, the constitutive equation for a Newtonian fluid is expressed by the next
63 relationship:

$$64 \quad T = -\pi I + 2\mu D \quad (3)$$

65 here π pressure is a scalar field, D is the spatial gradient tensor of velocity, μ is the viscosity
66 fluid and I is a identity tensor. Substituting equation (3) in (1) and (2) the Navier-Stokes
67 equation for a Newtonian fluid are obtained:

$$68 \quad \rho[v' + (\text{grad } v) v] = -\text{grad } \pi + \mu \Delta v + b \quad (4)$$

$$69 \quad \text{div } v = 0 \quad (5)$$

On the other hand, the constitutive equation of a homogeneous isotropic elastic solid is given by the equation:

$$S = 2\mu E + \lambda(\text{tr}E)I \quad (6)$$

where E is the tensor of infinitesimal deformation, λ and μ are Lamé constants.

2.2 Dynamic Models.

2.2.1 Solid

The matrix equation governing the dynamic behavior of a solid is given by the following equation [15]:

$$[M]\ddot{u} + [C]\dot{u} + [K]u = F_b + F_t \quad (7)$$

Being $[M]$, $[C]$, $[K]$ mass, damping and stiffness matrices respectively, also u , $\dot{u}$, $\ddot{u}$, F_b y F_t are the vectors of displacement, velocity, and acceleration in addition to the force vectors and surface forces respectively, the latter ones commonly as a time-dependent vector. This equation (7) associates a set of linear or nonlinear equations depending on the structure of the matrix involved in the process. A typical solution of (7) is given by the method of Newmark where displacement and velocity fields are approximated by the following relationships [4,15]:

87

$$u(t)^{n+1} = u^n + \dot{u}^n \Delta t + \left[\left(\frac{1}{2} - \beta\right)\ddot{u}^n + \beta\ddot{u}^{n+1}\right]\Delta t^2 \quad (8)$$

$$\dot{u}(t)^{n+1} = \dot{u}^n + [(1 - \gamma)\ddot{u}^n + \gamma\ddot{u}^{n+1}]\Delta t \quad (9)$$

β and γ are parameters of stability of the method, Δt is the time step [16].

2.2.1 Fluid

Analogously, the discrete equations governing the behavior of the fluid are described below:

$$[M]\ddot{u} + [K]_u(\dot{u})\dot{u} - [G]p + [K]_\mu\dot{u} = F_b + F_t \quad (10)$$

$$[G]^T\dot{u} = 0 \quad (11)$$

Here $[M]$, $[K]_u$, $[G]$, $[K]_\mu$ represent the mass, convection, pressure and viscosity matrix respectively, $\dot{u}$, $\ddot{u}$, p , F_b y F_t are the velocity, acceleration, pressure, body forces and surface force vector respectively. Equations (10) and (11) are a set of nonlinear equations. The solution of these equations is based on the implicit integration scheme of Euler [5,18]:

$$[M] \left[\frac{3\dot{u}_{n+1} - 4\dot{u}_n + \dot{u}_{n-1}}{2\Delta t} \right] + [K](\cdot)[\dot{u}_{n+1}] - [G]p_{n+1} = F_{n+1} \quad (12)$$

$$[G]^T\dot{u}_{n+1} = 0 \quad (13)$$

The process of solving of the discrete model described in these equations (12) and (13) requires a method of stabilization, particularly Ansys program uses SUPG (Streamline-upwind/Petrov–Galerkin) and PSPG (Pressure stabilized/Petrov–Galerkin) [19], whose methodology is based on the following equation:

$$\int_{\Omega} [N + \delta^h + \varepsilon^h]^T (\rho[v' + (\text{grad } v)v] + \text{grad } \pi - \mu \Delta v - b + \text{div } v) d\Omega \quad (14)$$

The terms δ^h y ε^h are the stabilizers terms called SUPG y PSPG, The latter equation (14) is solved using the scheme named "SIMPLE" (semi-implicit method pressure linked equations) where grades of freedom corresponding to the velocity are solved with the algorithm "TDMA" (tri-diagonal matrix algorithm) and the pressure are solved with the method of residual conjugate pre-conditioned. The integration process (14) is quite similar to that indicated in the equations (10) and (11). In contrast, the DINES + GiD software uses the stabilization method OSS (orthogonal sub-scale) expressed by the relation [5,20]:

$$[M]\ddot{u} + [K]_{\dot{u}}(\dot{u})\dot{u} + [K]_{\mu}\dot{u} - [G]p + [G]^T\dot{u} + \tau(\cdot) = F_b + F_t \quad (15)$$

Here $\tau(\cdot)$ is the stabilization parameter described in [5], then a fractional step scheme is used. The key is to use the auxiliary variable u' and the following approximation [21]:

$$K(\cdot)[\dot{u}_{n+1}] \approx K(\cdot)[\dot{\tilde{u}}_{n+1}] \quad (16)$$

Thus, equations (12) and (13) are transformed into the following equivalent relationship:

$$[M] \left[\frac{3\dot{\tilde{u}}_{n+1} - 4\dot{u}_n + \dot{u}_{n-1}}{2\Delta t} \right] + [K](\cdot)[\dot{\tilde{u}}_{n+1}] - [G]p_n + E_1 = F_{n+1} \quad (17)$$

$$-\frac{2}{3}\Delta t [G]^T [M]^{-1} [G](p_{n+1} - p_n) + E_2 = [G]^T \dot{\tilde{u}}_{n+1} \quad (18)$$

$$[M] \left[\frac{3\dot{u}_{n+1} - 3\dot{\tilde{u}}_{n+1}}{2\Delta t} \right] - [G](p_{n+1} - p_n) = 0 \quad (19)$$

Where E_1 y E_2 are stabilizers terms of the OSS scheme corresponding to the momentum and continuity equations respectively. The resulting equations from this methodology of fractional steps are solved by the method of Picard.

2.3 Coupled Model

In previous pages the structural and fluid models were raised in its Lagrangian and Eulerian domain respectively, however, when considering the interaction between these domains, incompatibilities can be solved through a third description called ALE (Algorithm Lagrangian Eulerian) [22]. This procedure refers a mesh movement method complying the law of geometric conservation, there are several methods of moving mesh, one of the most used is the Laplacian method [8,23]:

$$\Delta u_m = 0 \quad (20)$$

u_m is the movement of mesh nodes in areas near of the structural domain. As the nodes of the mesh of the fluid domain can move arbitrarily, is necessary then to consider the convective velocity c defined by [24]:

$$c = v - \bar{v} \quad (21)$$

136 Being v and $\bar{v}$ the material flow velocity and the velocity of the mesh respectively, therefore,
137 considering (21) is obtained the following equation [5]:

$$138 \quad [M] \ddot{u} + [K]_{\dot{u}}(\dot{u})c - [G]p + [K]_{\mu}\dot{u} - F + [G]^T\dot{u} + \Delta u_m + E_1 + E_2 = 0 \quad (22)$$

139 which is the set of discrete equations on the domain of the fluid. The integration procedure
140 for this equation (22) is similar to the equations of fluid mechanics indicated above. Mesh
141 movement is considered only in the vicinity of fluid domain with the structural domain.
142 Similarly, equation (11) is modified as follows:

$$143 \quad [M]\ddot{u} + [C]\dot{u} + [K]u = F_b + F_t + H \quad (23)$$

144 Here H is the effect of the fluid domain over the structural domain. Boundary kinematics and
145 dynamics conditions required in the solution of the coupled problem are indicated as follows:

$$146 \quad x_0^s + d = x, \quad \dot{d} = u \quad (24)$$

$$147 \quad T_n = S_n \quad (25)$$

148 Where d is the structural displacement during the interaction, x_0^s is the initial position of the
149 structural domain and n is a unit normal vector. The equation (22) and (23) are solved by the
150 block method Gauss-Seidel [25]:

$$151 \quad x_{k+1}^{n+1} = F(x_k^n, y_k^n) \quad (26)$$

$$152 \quad y_{k+1}^{n+1} = E(x_{k+1}^{n+1}, y_k^n) \quad (27)$$

153 F and E are abstract representations of equations (38) and (39) respectively, x and y are the
154 unknowns of fluid domains (velocity and pressure) and structural (displacement), n
155 represents the time step and k is a counter for controlling the iterations sets.

156 2.4. Experimental Procedure

157 2.4.1 Aeroelastic Balance

158 When considering the effect of Aeroelastic Structure, we can rewrite equation (7) as follows
159 [13,26]:

$$160 \quad m\ddot{u} + c\dot{u} + ku = F(t, u, \dot{u}, \ddot{u}) \quad (28)$$

161 where the force vector F is a function of time, movement and its derivatives. The form of the
162 first mode of vibration of the structure to be analyzed can be approximated by the following
163 exponential equation [13]:

$$164 \quad \varphi(z) = \alpha \left(\frac{z}{H} \right)^\beta \quad (29)$$

165 α and β are normalization coefficients. For a linear mode shape, considering scaling
166 laws described in [13,27], together with relation (28), and according to [16,28] we obtain:

$$167 \quad \ddot{u} + \left(\frac{1}{K_\zeta} \right) 4\pi f_m \dot{u} + (2\pi f_m)^2 u = \frac{M_m/H_m}{m_m} \left(\frac{1}{K_{CM}} \right) \left(\frac{1}{K_{\rho a}} \frac{K_\rho}{K_L^3} \right) \left(\frac{K_L}{K_t K_V} \right) \quad (30)$$

Here f_m the fundamental frequency in the wind direction, M_m is the bending moment at the basement, H_m is the total height of the structure, the subindex m indicates that these three parameters are related to the physical scale model. Equation (46) refers the scaling laws regarding damping, force coefficient, generalized mass and stiffness. These scaling laws are functions of K_L , K_V , K_t , K_{CM} , $K_{\rho a}$, K_ρ and K_ζ representing geometry similarities, wind velocity, frequency, force coefficient, density, mass and damping between the physical scale model and the prototype [13].

Based on these scaling laws, the model represents a direct similarity with the prototype and can be obtained the bending moment at the basement of the scale model as well as the displacement at any point in the model.

$$K_M = \frac{k_{\rho a} K_L^5}{k_t^2} \quad (31)$$

2.4.2 Wind tunnel. Settings

The mass of the physical model is obtained by applying equation [13]:

$$m(z)_i = \frac{m_{pi}(K_{\rho a})(K_L)^3}{\varphi_i^2} \quad (32)$$

m_{pi} is the general mass of prototype and φ the fundamental mode shape in the wind direction. Once the model was built, its damping is determined by the logarithmic decrement method [28]:

$$\xi = \sqrt{\frac{\delta^2}{4\pi^2 + \delta^2}} \quad (33)$$

Where δ is the logarithmic decrement and ξ the damping factor. After built the Physical model, its fundamental frequency f_{1m} . Wind velocity in the tunnel is determined by the following relationship:

$$v_m = (K_L) \left(\frac{1}{K_t} \right) v_p \quad (34)$$

v_m y v_p are the wind velocity applied to the model (on the wind tunnel) and the corresponding velocity of the prototype. Before making the measurements in the wind tunnel is required to establish a calibration equation based on the displacement of the model resulting of a known force and its point of application, this same equation will be applied to the measurements on the wind tunnel with the difference that the force will be produced by the wind. The moment of the prototype is given by the equation (31):

$$M_p = M_m \frac{(K_t)^2}{(K_L)^5} \quad (35)$$

M_p y M_m are the bending moment at the basement of the prototype and the model respectively, in (35) has been considered that the air density in the model and prototype are identical.

2.5 Test Models

2.5.1 Properties

The structural prototype has a prismatic shape depending on the height, the structural elements are made of steel and concrete, the building is covered with glass plates to prevent the flow of wind inside (Fig.1). The total height is 72.0 meters, it has 18 slabs (including the roof one) uniformly spaced. Table 1 shows the mechanical properties of the prototype applied in this work while in Table 2 are shown the geometrical properties of the structural sections. Numerical models were developed in ANSYS [29], and DINES + GiD [30,31] software and the experimental test was carried out on a subsonic wind tunnel.

Table 1. Mechanical Properties

Material	E (N/m ²)	ν (-)	ρ (kg/m ³)	μ (kg/m ² s)
Steel	2.00×10^{11}	0.250	7,800	-
Concrete	2.25×10^{10}	0.100	2,400	-
Glass	$1.00 \times 10^{11*}$	0.225	0.500	-
Wind	-	-	1.185	1.831×10^{-5}

Table 2. Structural Sections

Section	d (m)	b (m)	t_f (m)	t_w (m)
W18x35	0.4496	0.1524	0.0108	0.0076
HSS16	0.4064	0.4064	0.0127	-
HSS14	0.3556	0.3556	0.0127	-
HSS12	0.3048	0.3048	0.0127	-
HSS10	0.2540	0.2540	0.0127	-
Slab	0.1100	-	-	-
Glass	0.0050	-	-	-

All beams are sections W18x35, all levels to a height of 12.00 meters are supported with columns HSS16, next levels up to a height of 24.00 meters are supported by columns HSS14, the following levels to a height of 48.00 meters are HSS12 columns sections, the rest of the levels are supported by columns sections HSS10. The slab and glass sections are identical in all levels. Each column-column connections and beam-column are rigid type. All supports at basement of building have been considered fixed.

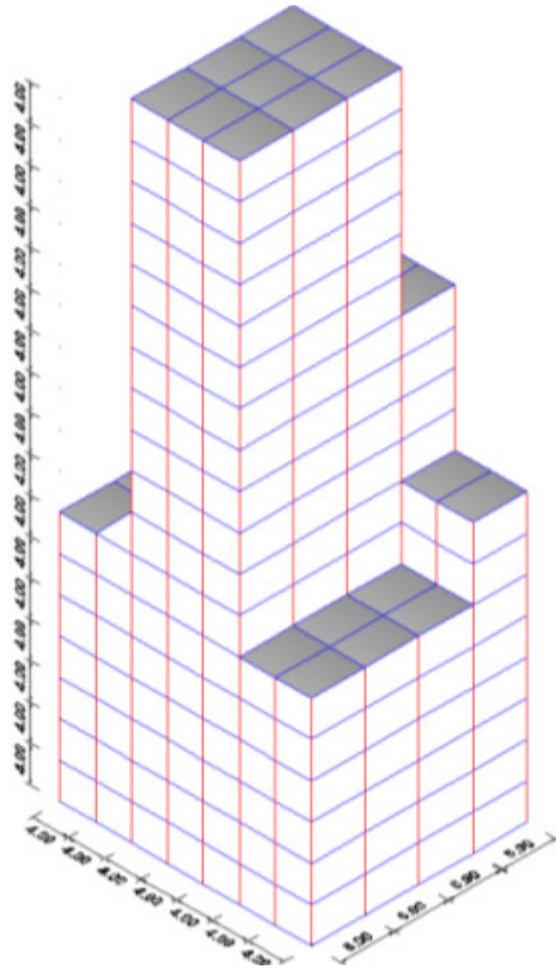


Fig.1. Main dimensions of the Irregular Building composed of steel structure and reinforced concrete slabs

The domain of the fluid surrounds the structural domain is shown on Figure 2, the fluid domain size and boundary conditions considered in this work are indicated. In all areas dividing the two domains is considered a velocity $v = 0 \text{ m / s}$. The hatched area on Figure 2 shows the structural domain.

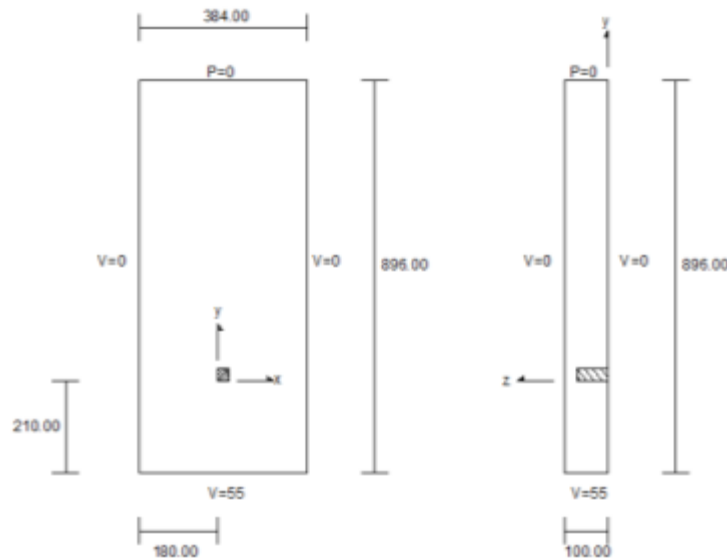


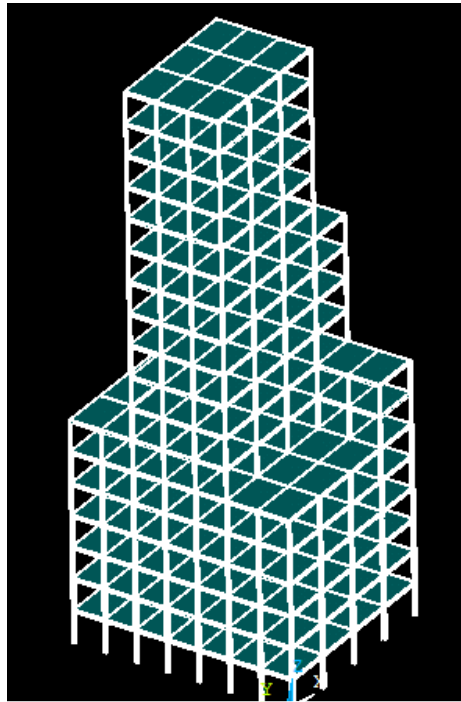
Fig. 2. Domain fluid on planes "xy" and "yz" respectively

2.5.2 Computational Modeling in Ansys

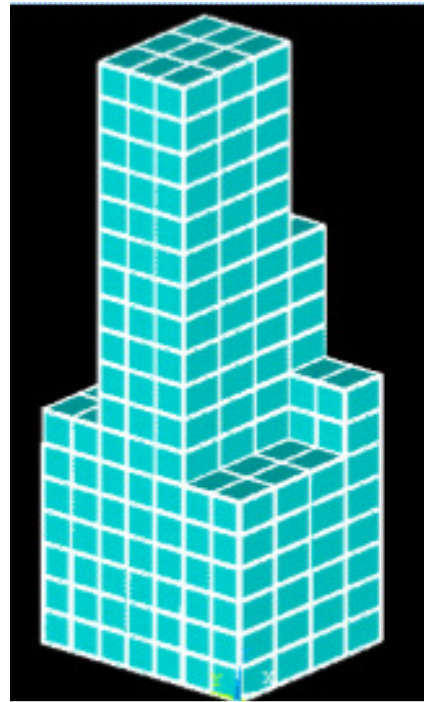
The discrete model was built with 50,228 finite elements. For beams and columns have been used elements "beam" with 6 degrees of freedom per node. For concrete slabs as well as cover glass on windows have been used elements "shell" with 12 degrees of freedom (see Fig 3). To model the fluid, finite elements "brick" of 24 degrees of freedom were used. The structural model solution obeys the equation (23) with $\beta = 1/6$ and $\gamma = 1/2$. The fluid domain use equations (12), (13), (14), (22), while coupling of fields applied equations (20), (26) and (27). In both domains was used a step time of 0.01 seconds, having carried out a total of 500 iterations, the convergence criteria used for displacements was 1×10^{-3} , whereas in velocities 1×10^{-6} and for pressures 1×10^{-3} . Figures 3(a) and 3(b) show the internal structure geometry together with elements "beam" and "shell". Figure 3(c) shows the external geometry of the structure including the domain of fluid that surrounds the structural domain.

2.5.3 Computational Modeling with DINES + GiD

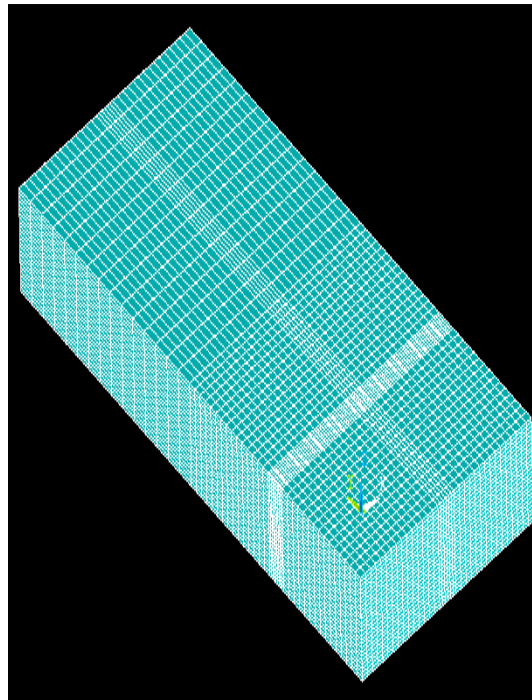
In this case only was modeling the fluid domain, structural domain has been considered as a rigid body. A total of 113.000 finite elements were required. The fluid was modeled with elements "brick" of 24 degrees of freedom. Fluid domain was solved by equations (12), (13), (15), (17), (18), (19), and (22). The time step used was 0.01 seconds, 1,000 iterations in total were performed, and the convergence criteria were in velocities equal to 1×10^{-6} while in pressures 1×10^{-3} . Figure 4 shows the domain of the fluid which surrounds the structural domain.



(a)



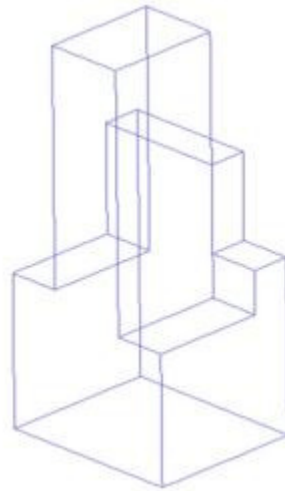
(b)



(c)

Fig. 3. Structural and fluid domain in ANSYS: (a) Beam steel structure and slabs of reinforced concrete. (b) Side covers of building. (c) Fluid domain enveloping the building.

271



(a)

272
273



(b)

274
275
276
277
278
279
280

Fig. 4. Structural and fluid domain in Dines + GiD: (a) Building as a rigid body. (b) Fluid Domain around the building

2.6 Test on wind tunnel

281 The fundamental frequency of the building in wind direction was initially determined turning
282 out to be 0.51 Hz and the mode shape associated as shown in Figure 5. The scale model
283 (Figure 6) was made with PVC foamed material and scaled from the first fundamental
284 vibration mode. The prototype analyzed has not a linear mode shape, however, in this work
285 was linear considered taking parameters $\alpha = 1.00$ and $\beta = 0.914$ according to (29). In Figure
286 5 are shown both modes.

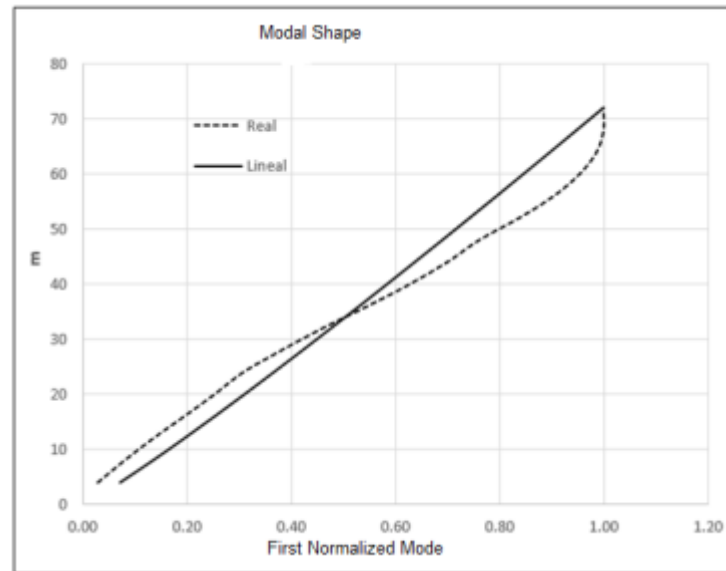


Fig. 5. Mode Shape of Prototype



Fig. 6. Scale Model made of PVC foamed material

Considering the fundamental linear mode in (32) and a geometric scale 1: 200 it has been obtained a mass scale model of 0.293 kg (while the prototype has a mass of 2345×10^3 kg). Using equation (33) we obtain a damping factor of 0.0418, this value was also used in ANSYS modeling. The frequency of the model was obtained experimentally proving to be of 20.81 Hz, and by means of equation (34) has been obtained the wind velocity inside the tunnel of 11.11 m / s. Equipment used for physical experimentation is shown in Figure 7 and 8. Table 3 shows a comparison between the physical parameters of the model and prototype used.

Table 3 Physical Parameters

Parameter	Model	Prototype
Mass (kg)	0.293	2,345x10 ³
Height (m)	0.36	72.0
Period (s)	0.048	1.96
Frequency (Hz)	20.81	0.51
Damping (-)	0.0418	0.0418
Wind Velocity (m/s)	11.11	55.0



Fig. 7. Wind tunnel and instrumented scale model



Fig. 8. Equipment used on test measurements

3. RESULTS AND DISCUSSION

The model inside the wind tunnel was instrumented in wind direction; hence the results that will be validated with the numerical results will be only the ones corresponding to this direction. On the other hand, the bending moment at the basement obtained with Ansys [29]

was 47.504 kN-m with a displacement at the top of 0.936 meters in a time of 1.30 seconds. Figures 9 and 10 show some of the numerical results obtained with Ansys.

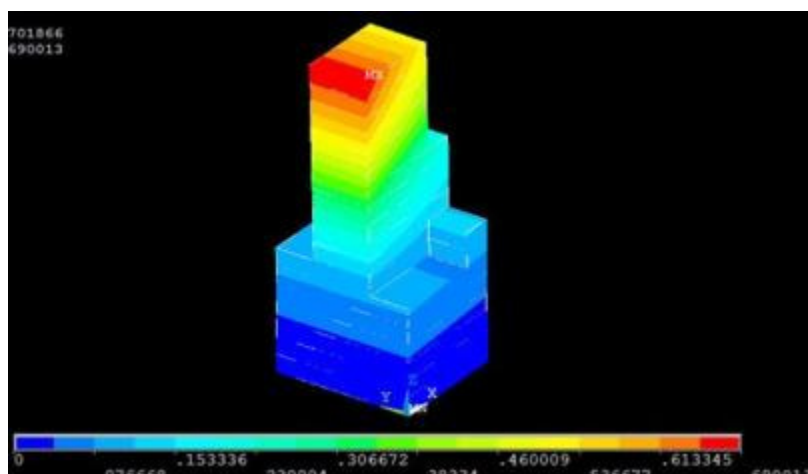


Fig. 9. Displacement Results for $t=1.0$ sec.

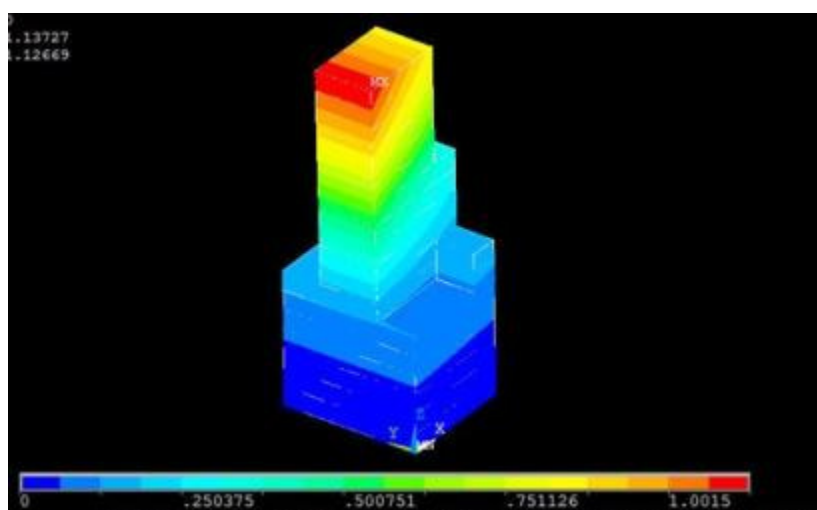


Fig. 10. Displacement Results for $t=1.1$ sec.

Through Dines + Gid software [30,31], it has been obtained at the Building basement a bending moment of 53.706 kN-m in a time of 1.06 seconds. Figures 11-12 show some results corresponding to the pressure generated by the bending moment at the basement.

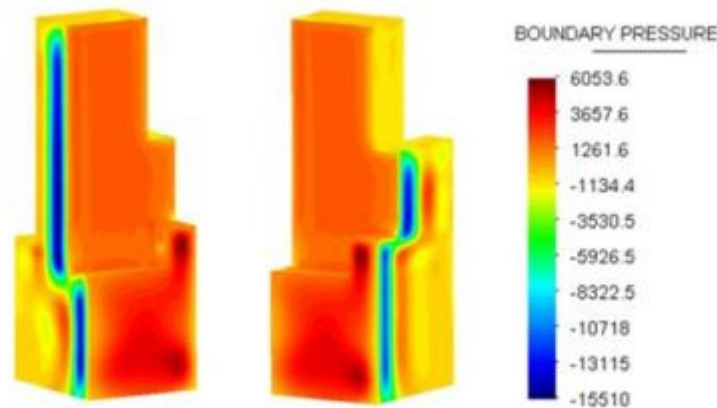


Fig. 11. Pressure contours for a time $t=1.0$ sec.

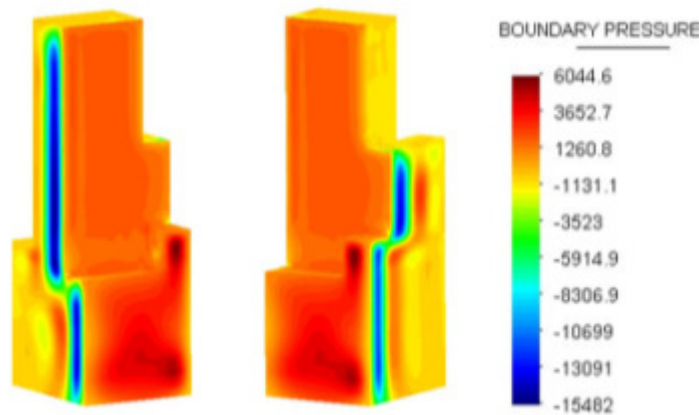


Fig. 12. Pressure contours for a time $t=1.3$ sec.

In the wind tunnel the bending moment at the basement of the model was obtained as 0.291 Nm and a displacement at the top of model of 0.00458 meters. Applying the scale law (35) and considering a geometrical scale 1: 200 the bending moment at the basement of prototype will be 55,397 kN-m while the corresponding displacement at the top of the building of 0.916 meters.

The test in the wind tunnel allows us to compare results with those obtained from computational modeling. Regarding DINES + GiD software, here it is only possible to compare the maximum moment obtained with respect to the wind tunnel. On the other hand ANSYS solutions allow as compare displacement and moment at the basement. We proceed to make a comparison (value: e) between experimental obtained results with respect to the computational modeling.

$$e = \frac{55.4 - 47.5}{55.4} = 0.142 = 14.2 \% \text{ using ANSYS}$$

$$e = \frac{55.4 - 53.7}{55.4} = 0.031 = 3.1 \% \quad \text{using DINES + GiD}$$

356 4. CONCLUSIONS

357 The above results observe differences in part by the different algorithms applied on each
358 program. Some remarks resulting from this work are as follow:

359 The physical scale model used in this work is quite similar to that shown in Ansys, which is in
360 turn based on the twist that occurs in the basement of the model due to the wind effect, so,
361 the physical model behaves like a rigid body. On the other hand the torsion effect it was not
362 considered in our scale model, but from ANSYS Simulation it is observed to be significant
363 the torsion effect in the highest points of the structure.

364 In Dines + GiD, the structural model is considered as a rigid body fixed at the basement, so
365 it has been possible to run the wind algorithm turbulently because a quantity of finite
366 elements within the fluid domain were enough to achieve the corresponding convergence
367 criterion, this number of elements was possible due to the no transfer process of pressure
368 towards another domain.

369 In ANSYS both domains were modeled, but in the flow domain fewer elements were used
370 due to the fact that was proposed that only the fluid pressure on the external nodes of the
371 tridimensional frame structure will be transferred, this proposal was suggested because it
372 was no possible introduce real material properties of plates cover glasses, in another case,
373 this plates will be highly deformed causing early failures. ANSYS algorithm requires a
374 structural surface to be adjacent to the fluid domain to calculate the displacement in the
375 boundary between domains, because of this, was not possible to propose a finer mesh to
376 achieve convergence criteria in a turbulent fluid model and therefore it was chosen a laminar
377 flow.

378 Despite the large numerical and experimental development nowadays, difficulties prevail for
379 solving this kind of structures having a complex geometry. This work can be useful in
380 choosing criteria for analogous analysis cases of high buildings subject to wind effects.

381

382

383 REFERENCES

384

- 385 1. Breuer, M., G. De Nayer, M. Münsch, T. Gallinger, R. Wüchner. 2012. Fluid-
386 structure interaction using a partitioned semi-implicit predictor-corrector coupling
387 scheme for the application of large-eddy simulation. Journal of fluids and Structures
388 29 107-130
- 389 2. Wood, C., A.J. Gil, O.Hassan, y J. Bonet. 2010. Partitioned block-Gauss-Seidel
390 coupling for dynamic fluid-structure interaction. Computers and Structures 88 1367-
391 1382.
- 392 3. Huerta, A., J. Sarrate, y J. Donea. 2001. Arbitrary Lagrangian-Eulerian formulation
393 for fluid-rigid body interaction. Comput. Methods Appl. Mech. Engrg. 190 3171-3188.
- 394 4. Drinkler, D., B. Hübner, y E. Walhorn. 2004. A monolithic approach to fluid-structure
395 interaction. Comput. Methods Appl. Mech. Engrg. 193 2087-2104.
- 396 5. Valdés-Vázquez, J.G. 2007. Nonlinear analysis of orthotropic membrane and shell
397 structures including fluid-structure interaction. PhD Thesis, Universitat Politècnica de
398 Catalunya, España.
- 399 6. Belver, A.V., A. L. Ibán, y C. E. Lavín-Martín. 2012. Coupling between structural and
400 fluid dynamic problems applied to vortex shedding in a 90 m steel chimney. J. Wind
401 Eng. Ind. Aerodyn. 100 30-37.

- 402 7. Matthies, H. G., R. Niekamp, y J. Steindorf. 2006. Algorithms for strong coupling
403 procedures. *Comput. Methods appl. Mech. Engrg.* 195 2028-2049.
- 404 8. Masud, A., y R. A. Khurram. 2006. A multiscale/stabilized formulation of the
405 incompressible Navier-Stokes equations for moving boundary flows and fluid-
406 structure interaction. *Comput. Mech.* 38, 403-416.
- 407 9. Richter, T., y T. Wick. 2010. Finite elements for fluid-structure interaction in ALE and
408 fully Eulerian coordinates. *Comput. Methods Appl. Mech. Engrg.* 199, 2633-2642.
- 409 10. Wall, W. A., S. Genkinger, y E. Ramm. 2007. A strong coupling partitioned approach
410 for fluid-structure interaction with free surfaces. *Computers & Fluids* 36 169-183.
- 411 11. Tezduyar, T.E., S. Sathe, R. Keedy, y K. Stein. 2006. Space-time finite element
412 techniques for computation of fluid-structure interactions. *Comput. Methods Appl.*
413 *Mech. Engrg.* 195 2002-2027.
- 414 12. Zhang, Q., y B. Zhu. 2012. An integrated coupling framework for highly nonlinear
415 fluid-structure problems. *Computers & Fluids* 60 36-48
- 416 13. Zhou, Y., y A. Kareem. 2003. Aeroelastic Balance. *Journal of Engineering*
417 *Mechanics* 129 283-292
- 418 14. Gurtin, M. E. 1981. *An Introduction to Continuum Mechanics*. Academic Press.
- 419 15. Slone, A. K., K. Pericleous C. Bailey, y M. Cross. 2002. Dynamic fluid-structure
420 mesh procedures. *Computers and Structures* 80 371-390.
- 421 16. Cook, R. D. 1995. *Finite element modeling for stress analyses*. Wiley.
- 422 17. Bathe, K. J., y M. M. I. Baig. 2005. On a composite implicit time integration
423 procedure for nonlinear dynamics. *Computers and Structures* 83 2513-2524.
- 424 18. Peric, D., y W. Dettmer. 2003. An analysis of the time integration algorithms for the
425 finite element solutions of incompressible Navier-Stokes equations based on a
426 stabilized formulation. *Comput. Methods Appl. Mech. Engrg.* 192 1177-1226.
- 427 19. Fries, T. P., y H. G. Matthies. 2006. A stabilized and coupled meshfree / meshbase
428 method for the incompressible Navier-Stokes equations-Part I: Stabilization.
429 *Comput. Methods Appl. Mech. Engrg.* 195 6205-6224.
- 430 20. Codina, R., O. Soto. 2004. Approximation of the incompressible Navier-Stokes
431 equations using orthogonal subscale stabilization and pressure segregation on
432 anisotropic finite element meshes. *Comput. Methods Appl. Mech. Engrg.* 193 1403-
433 1419.
- 434 21. Badia, S., y R. Codina. 2006. Fluid-structure iterative algorithms using pressure
435 segregation methods and their application to bridge aerodynamics. In: P. Wesseling,
436 E. Oñate, y J. Periaux (Eds.) *European Conference on Computational Fluid*
437 *Dynamics*. Netherlands.
- 438 22. Ponthot, J. P., y T. Belytschko. 1998. Arbitrary Lagrangian-Eulerian formulation for
439 element-free Galerkin method. *Comput. Methods Appl. Mech. Engrg.* 152 19-46.
- 440 23. Wick, T. 2011. Fluid-structure interactions using different mesh motion techniques.
441 *Computers and Structures* 89 1456-1467.
- 442 24. Borst, R., E.Kuhl, y S. Hulshoff. 2003. An arbitrary Lagrangian Eulerian finite-
443 element approach for fluid-structure interaction phenomena. *Int. J. Numer. Meth.*
444 *Engrg.* 57 117-142.
- 445 25. Matthies, H. G., y J. Steindorf. 2002. Partitioned but strongly coupled iteration
446 schemes for nonlinear fluid-structure interaction. *Computers and Structures* 80
447 1991-1999.
- 448 26. Boggs, D.W., 1992. Validation of the Aerodynamic Model Method. *Journal of Wind*
449 *Engineering and Industrial Aerodynamics* 41-44 1011-1022.
- 450 27. ASCE Standards, 2012
- 451 28. Craig Jr., R. R. 1981. *Structural dynamics*. Wiley.
- 452 29. ANSYS Inc. Software. Versión 14.0, 2011.

- 453 30. DINES 2007. Algoritmo numérico para interacción fluido-estructura. Jesús Gerardo
454 Valdés Vázquez. Universidad de Guanajuato, México.
455 31. GiD. 2007. Software for pre-process and post-process of the finite element method.
456 R. Ribo, M. Pasenau, E. Escolano, J. Suit, y A. Coll. CIMNE, Barcelona.