XL CILAMCE
IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING
NOVEMBER 11-14, 2019
Praiamar Natal Hotel & Convention
Natal, RN-BRAZIL

TRANSIENT CFD ANALYSIS OF FLOW OVER A CIRCULAR CYLINDER

Luiz F. C. de Oliveira
Mario R. Freitas
Lineu j. Pedroso
lfelipecarvalhedo@gmail.com
mariofreitas.enc@gmail.com
lineujp@gmail.com
Universidade de Brasília
Campus Universitário Darcy Ribeiro, 70910-900, DF - Brasília, Brazil

Abstract. The flow of air over a circular cylinder can be observed in many civil engineering problems. Tall buildings, towers and silos can be subject to strong winds that can impact the structure local or global stability. The flow over a cylindrical structure is a dynamic problem with transient response characterized by vortex shedding. If the Reynolds number is sufficiently high, an alternate vortex shedding pattern is formed, known as von Karman vortex street. In this scenario, the flow applies dynamic drag and lift forces on the structure. This paper presents a methodology to simulate the fluid flow over a circular cylinder and compute the drag and lift coefficients using Computational Fluid Dynamics (CFD). A two-dimensional cross-section of a structure is modelled in ANSYS Fluent applying the proper boundary conditions. The pressure fields are computed from the simulations and numerical integration is used to obtain the drag and lift coefficients. From the simulations, the streamlines, and pressure fields can also be obtained and plotted. Geometry, mesh and time step size independence studies are conducted to ensure convergence. The results are compared to numerical data found in the literature.

Keywords: Computational Fluid Dynamics, Vortex, Cylinder

1 Introduction

Following the global trend of reducing pollution, it is necessary to study ways to produce energy that minimizes environmental damage. However, it is important not to decrease energy productivity, nor elevate costs in a way that would make the project unfeasible. Wind power generation is a good energetic alternative but possesses structural challenges. Thus, in this project, the effects of wind in a wind turbine were studied, in a simplified way with Computational Fluid Dynamics (CFD) using the software ANSYS Fluent, minimizing the need for tests. Therefore, it seeks to reduce the costs of the implementation studies for these structures.

The problem of air flow around a wind power tower is highly complex. However, the tower structure can be simplified for this simulation without problems. In this study, the structure is assumed with constant shaft diameter, further on it is considered a plane analysis. The shaft diameter is equal to 1 meter. Then, the domain is created and validated. The mesh refinement is also validated.

In the literature, there are many works on the flow around a cylinder using CFD. Souza [1] studied the generation of von Karman vortices in the water flow around a cylinder and a semicircular section as a simplified way of studying the generation of vortices in the swim of a fish. Arakaki Jr. [2] studied the water flow around circular submarine ducts emphasizing the vibration phenomenon induced by vortices. Lopes [3] studied the influence of the aspect ratio of a cylinder in the flow results. Carneiro [4] studied the flow-induced vibration in offshore structures. Hallak [5] and Costa [6] used CFD to study the wind flow around a bridge.

In the Dynamics and Fluid-Structure Group (Grupo de Dinâmica e Fluido-Estrutura) from the University of Brasília (UnB), some works in CFD were also developed. Ferreira [7] studied the water flow effects in an offshore platform using ANSYS CFX. Santos [8] studied the wind flow around a tall building. Silva [9] studied air flow around a wind turbine with low Reynolds number assuring a steady-state flow. Freitas [10] used CFD to calculate hydrodynamic pressures in a dam subject to overtopping and calculate the structure stability. Pedroso [11] shows the basic formulation used by ANSYS in acoustic cavity problems.

This work seeks to present a methodology to use CFD to analyze the flow around basic civil engineering structures. Because it is an introductory study, the Reynolds number analyzed is not similar to those that really need to be studied. However, this study can be the base for more advanced simulations.

2 Theoretical Background

2.1 Fundamental Equations

The equations that govern fluid flow are the continuity equation and the Navier-Stokes equations. The continuity equation represents, mathematically, the fact that, in a control volume, there is mass conservation, meaning that the quantity of mass that enters the control volume is equal to the one that exits it. For incompressible flow, the continuity equation is given by:

$$\nabla \cdot \vec{v} = 0. \tag{1}$$

The Navier-Stokes equations represent the sum of the forces in the three dimensions (x, y, z) applied to a control volume. Fluent uses the following vector form of the equation [12].

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho\vec{v}\vec{v}) = -\nabla p + \nabla \cdot (\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F}. \tag{2}$$

Where ρ is the fluid's density, $\vec{v}$ is the velocity, t is the time, p is the static pressure, $\rho\vec{g}$ is the gravitational force, $\vec{F}$ is the external force and $\bar{\bar{\tau}}$ is the stress tensor, given by Eq. 3.

$$\bar{\bar{\tau}} = \mu\left[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla \cdot \vec{v}I\right]. \tag{3}$$

Where μ is the dynamic viscosity and I is the unit tensor.

3 Computational Model

The adopted geometry is a function of the cylinder diameter and has the dimensions L1, L2 and H (Fig. 1.a) equal to 15D, 30D and 15D respectively. For the mesh, the size of the cells further from the cylinder was set to 0.3 m. Later in this paper, the geometry and mesh validations are presented.

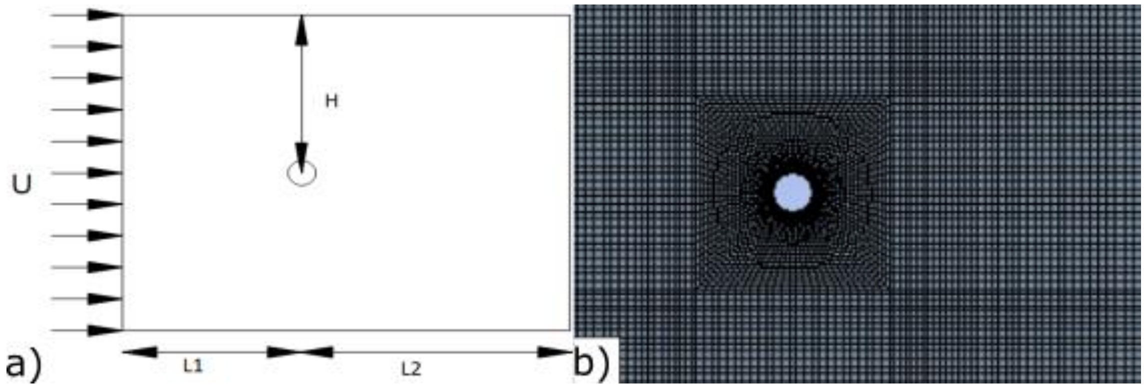

Figure 1. Computational model: (a) Geometry, (b) Mesh

The fluid used was air and its properties were taken at a pressure equal to 1 atm and temperature of 25 °C. That gives density (ρ) equal to 1.184 kg/m^3 and dynamic viscosity (μ) of 1.849 x 10^{-5} kg/ms (Çengel and Cimbala [13]). Then, using Reynolds equation, the velocity to be input in the model was obtained (Eq. 4).

$$Re = \frac{\rho V D}{\mu} \rightarrow V = \frac{Re\mu}{\rho D} \tag{4}$$

Where Re is the Reynolds number; D is a characteristic dimension of the flow, equal to the diameter of the cylinder, 1 m; V is the velocity to be found in m/s. Therefore, for Reynolds equal to 200, the velocity applied to the domain must be 0.006247 m/s.

The outlet condition in Fluent is a pressure outlet and, in this case, it was used 0 Pa. Meaning that all pressure values gotten in this study are relative to the outlet pressure of the flow. On the walls parallel to the flow, it was considered null friction. On the cylinder, it was set to no-slip condition, that is, the velocity of the air in contact with the cylinder is equal to zero.

Because it is a flow with vortex shedding, the calculation method must be transient. The time step was set to 20 s. However, as it is a laminar vortex street [9,14], the method utilized in Fluent must be laminar.

4 Convergency Studies

To validate the computational model, it is necessary to do verifications to assure that the dimensions of the domain are not influencing the results, being necessary to study the tendency of some parameters. Therefore, the dimensions L1, L2 and H were varied, one at a time, to assure the convergence of the results. For the mesh, the size of the cells was varied to validate. The parameters analyzed were: mean drag coefficient (Cd,med); maximum lift coefficient (Cl,max); and root mean square (RMS) of the lift coefficient (Cl,rms).

In the verification, it can be noted that L1, L2 and H converge at dimensions 15D, 10D and 15D respectively, in addition to the mesh converging with 7539 cells (Fig. 2). This way, it is possible to assume that the values adopted for the geometry (L1, L2 and H equal to 15D, 30D and 15D respectively) and for the mesh (18706 cells) satisfy the simulation.

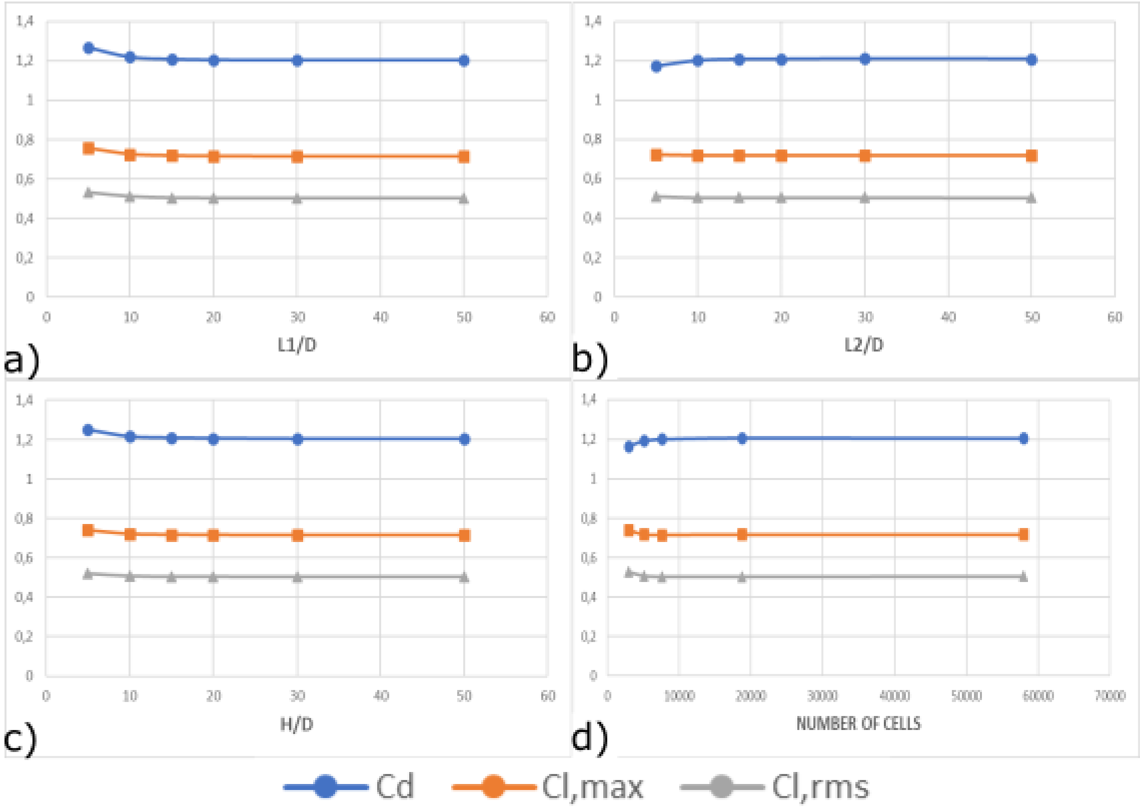

Figure 2. Geometry convergence study: (a) L1, (b) L2, (c) H and (d) mesh convergence

5 Results

After simulating with ANSYS, it is possible to generate multiple results from the software. The most basic one is the progression of Cd and Cl through time. Those coefficients vary in a sinusoidal curve with defined periods. In Fig. 3, it is possible to see the values of Cd and Cl through time.

Because of the Reynolds number adopted, an alternate periodic vortex shedding, known as von Karman vortex shedding, is observed. The lift coefficient period is equal to the vortex shedding period and double the drag coefficient period.

It is clear the tendency of stabilization of the graphics in curves with defined periods. Only the steady-state section of the graph is analyzed. The mean drag coefficient found was 1.21, the maximum lift coefficient was 0.72 and the root mean square of the lift coefficient was 0.51. The Strouhal number was 0.22. Franke et al. [15] simulated the flow around a similar cylinder and found, for the same Reynolds number, mean drag coefficient of 1.31, a maximum lift coefficient of 0.65 and Strouhal number of 0.19. Therefore, the relative error between the results was around 10%, which is reasonable for a first approximation.

The flow has a vortex shedding period of 740 seconds and, in Fig. 4, it is possible to see the representation of the streamline when Cl is close to zero. The pressure field on the cylinder in the same instant is also plotted (Fig. 4).

From the results, it is possible to compute the loads applied to the cylinder due to the air flow at the studied velocity. Because of this, it is possible to determine which would be the stress of a wind power tower structure, for example.

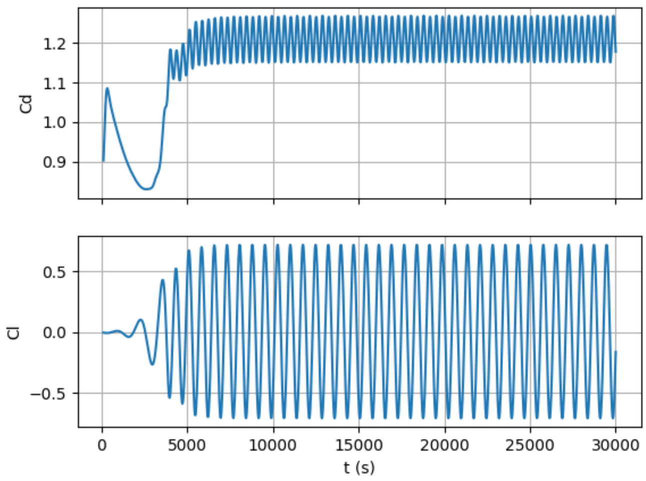

Figure 3. Cd and Cl through time

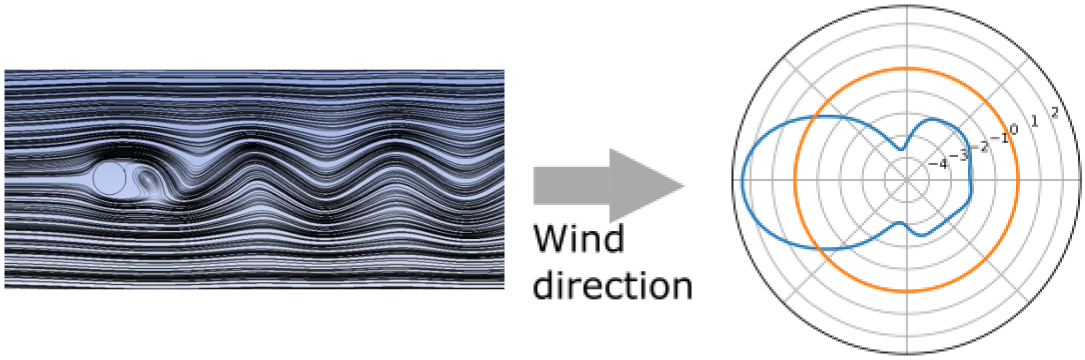

Figure 4. Streamline (left) Pressure, in 10^{-5} Pa, around the cylinder (right)

6 Conclusions and Recommendations

This work presents a methodology to use CFD to simulate the flow around a cylinder and obtain the resulting pressure fields for Reynolds number equal to 200. The obtained values are relatively close to those found in the literature. That way, simulations could be used to generate the stress in the structure without problems, increasing the feasibility of implementing wind power. However, as the work tried just to understand the flow around a cylinder and to introduce vortices in the structure, the results obtained here cannot be applied to real structures, due to the high velocities of the flow in a real situation compared to what was simulated. Either way, the methodology applied here can be used to simulate real conditions of wind and to obtain the expected pressures.

Beyond that, a domain size relative to the dimensions of the cylinder and a mesh that can be used in other simulations efficiently were presented, minimizing computational cost without loss of accuracy. This computational model can be used for higher Reynolds flows.

Acknowledgements

The authors thank the financial support of the Conselho Nacional de Desenvolvimento Cientifico e Tecnologico (CNPq) in the form of the scientific initiation scholarship (PIBIC) of the first author.

References

[1] SOUZA, T. DA S. Geração de Vórtices de von Kármán com Modelagem em CFD. Dissertação de Mestrado, Escola Politécnica, USP, 2016.
[2] ARAKAKI JR., H. Estudo dos esforços de correnteza marítima em risers com uso de CFD.
[3] LOPES, P. P. S. DE P. A CFD Investigation on the Flow around a Low Aspect Ratio Vertical Cylinder: Modeling Free Surface and Turbulent Effects. Dissertação de Mestrado, Escola Politécnica, USP, 2019.
[4] CARNEIRO, D. L. Análise de Vibrações Induzidas por Vórtices em Estruturas Offshore Utilizando Modelos Numéricos Tridimensionais no Domínio do Tempo. Dissertação de Mestrado, COPPE, UFRJ, 2007.
[5] HALLAK, P. H. Parâmetros Aeroelásticos para Pontes via Fluidodinâmica Computacional. Dissertação de Mestrado, COPPE, UFRJ, 2002.
[6] COSTA, L. M. F. Investigação numérica de modelos de turbulência no escoamento do vento em pontes suspensas. Dissertação de Mestrado, Escola Politécnica, USP, 2018.
[7] FERREIRA, J. L. Um Estudo de Ações Dinâmicas em Plataformas Offshore Utilizando Dinâmica dos Fluidos Computacional. Monografia de Graduação, Departamento de Engenharia Civil e Ambiental, Universidade de Brasília, 2012.
[8] SANTOS, T. C. Um Estudo de Ações Dinâmicas do Vento em Edifícios Altos. Monografia de Graduação, Departamento de Engenharia Civil e Ambiental, Universidade de Brasília, 2017.
[9] SILVA, A. C. DA. Análise Numérica do Escoamento em Torno de Um Cilindro. Monografia de Graduação. Departamento de Engenharia Civil e Ambiental. Universidade de Brasília, 2018.
[10] FREITAS, M. R. CFD Modelling for the Study of Structural Stability of Dams and Spillways Subject to Overtopping. Dissertação de Mestrado, Departamento de Engenharia Civil e Ambiental, Universidade de Brasília, 2019.
[11] PEDROSO, L. J. Rudimentos de CFD Baseado No Programa ANSYS. Brasília, Universidade de Brasília, 2017.
[12] ANSYS. ANSYS Fluent Theory Guide. ANSYS, Inc, 2018.
[13] ÇENGEL, Y.A.; CIMBALA, J.M. Mecânica dos fluidos: Fundamentos e aplicações. 3. ed. São Paulo: Mc Graw Hill Education, 2015.
[14] KHALIL, R.V. Estudo numérico de escoamento ao redor de um cilindro circular fixo. Monografia de Graduação, Curso de Engenharia Naval e Oceânica, Escola Politécnica, Universidade Federal do Rio de Janeiro, 2016.
[15] FRANKE, R.; RODI, W.; SCHÖNUNG, B. Numerical Calculation of Laminar Vortex- Shedding Flow Past Cylinders. Journal of Wind Engineering and Industrial Aerodynamics, v. 35, p. 237–257, 1990.