



25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

PRESSURE ANALYSIS OF A BUCKET WHEEL RECLAIMER SUBMITTED TO A STORM WIND LOAD

Moisés de Paula Gouvea
Rubens Perdigão Diz Oliveira
Hudson Oliveira
Jánes Landre Júnior
Cristiana Brasil Maia

Pontifícia Universidade Católica de Minas Gerais – Av. Dom José Gaspar, 500 Coração Eucarístico - 30535-901 - BH, Brasil
moisesgou@gmail.com
rubensperdigao@hotmail.com
hudson.oliveira@gmail.com
janes@pucminas.br
cristiana@pucminas.br

Abstract. *The modeling of pressure generated by unilateral wind loads in CFD simulations has been a subject widely covered in the automobile, aeromechanics and civil scope regarding the structures of automobiles, elevated buildings and pressure load in airplanes. However, there are fields not well covered, as an example, the mining field, which has large equipments driven by wind loads and can suffer from damage, sudden stop and loss of production. The objective of this work is the analysis of the wind load that crosses the structure of a bucket wheel reclaimer positioned in a harbor by the development of a numerical model of the flow in a critical speed load. The model was developed using the Ansys-Fluent software considering the conditions of permanent regime, wind velocity of 30,6 m/s, Mach 0,09 equivalent, constant density and segregated flow with Reynolds 195.556.*

Keywords: *pressure analysis, CFD, 3D modeling, permanent regime, mining equipment.*

1. INTRODUCTION

The mining sector has great economic importance for Brazil, directly influencing on its gross domestic product (GDP), corresponding to 4.3% of the total (IBRAM, 2015). In order for the production and transportation of the ore to be high-scale, it is essential to use ore recovery systems, which can recover thousands of tons per hour, having thus, dimensions and weight elevated, provoking structural tensions during their duty. Other generators of loads are the winds and according to recent news, Brazil are receiving huge storm winds events in the coast and harbors (Gazeta Online, 2019) reaching speeds of 110 km/h (30.6 m/s) (Estado de Minas, 2019) in locations where these equipment act.

The influence of wind loads is studied through Computational Fluid Dynamics (CFD) technique. It is possible to calculate the deformation of the mast of a 30-foot yacht (Bak *et al.*, 2013), to study the wind loads in a ship of loaded containers (Janssen, 2017), to test the resistance of buildings (Nozu *et al.*, 2015), trains and bridges under the action of lateral winds (Braun and Awruch, 2008). Authors around the world is also studying the impact phenomena of lateral winds in cyclists (Fintelman *et al.*, 2015) and the drag force with a cyclist followed by a car (Blocken *et al.*, 2015), the wind flux in sand dunes changing its format (Liu *et al.*, 2011), evaluating the side wind impact in a car (Tsubokura *et al.* 2010) or in a truck (Rocchi *et al.* 2012) or even analyzing the car body drag of an hybrid vehicle (Ramasamy *et al.*, 2017), the pressure generated in the top of high buildings (Zang *et al.*, 2012), the pressure load of wind in loads raised by a crane, wire rope supported (Han; Han, 2011), the wind load characteristic in a tower crane (Wei, 2018) and in a portal crane (Qing *et al.*, 2017). Papers focused in CFD simulation of mining equipment and harbor equipment submitted to wind are rare turning the present study valuable. There is a large camp of study of turbulent wind passing through structures with different velocities, considering external flow with high values of Reynolds modeling the problem in almost every job with $k - \epsilon$ model.

According to NBR6123, it is possible to describe the existing forces due to the impact of the wind in buildings, but is not yet adopted an exclusive criterion for the mining equipment. Thus, the objective of this work is to analyze the pressure distribution on the structure of an ore recovery when subjected to a constant wind load of 110 km/h. The reclaimer studied is a model of the mining company Vale, located in the maritime terminal of Ponta da Madeira. The Ansys-Fluent commercial code will be used to solve the conservation equations and the numerical results will answer questions about the possibility of a machine drag by the wind or simply pressure accumulated area causing abnormal stress to the structure.

2. MATHEMATICAL MODEL

A model concerning the structure of a bucket wheel reclaimer applying CFD best practices and a methodology developed with Vale were done to run tests and achieve the necessary results and conclusions. The machine virtual model drawing, object of this study, can be seen in the image shown in Fig. 1.

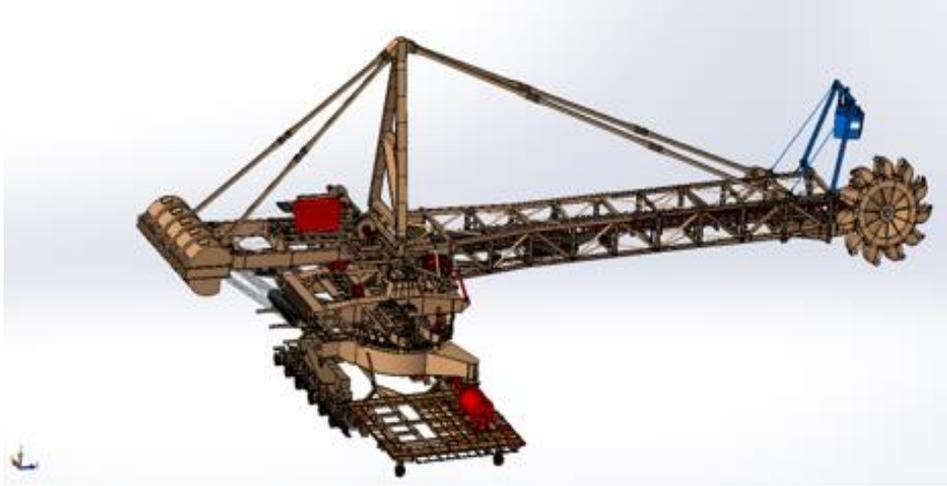


Figure 1. Bucket wheel reclaimer.

The three-dimensional model developed was created from the virtual geometric model of the reclaimer, which has approximate dimensions of 81 m in length, 36 m in width and 35 m in height with a payload of 1,40E6 kg. The control volume was modeled with the dimensions respecting 3% locking (Fintelman *et al.*, 2015), defined from the stipulated size for the equipment and all the 36 contacts with the rails by rolling wheels were modeled as it is.

The static pressure is the most valuable result of this paper because it shows what the impact of the wind results in the structure. For a large bucket wheel reclaimer even with technology to avoid accidents a wind blast of 30.7 m/s could be a problem as it is the manufacturers guarantee all the security for working in about 20 m/s (Vale, 2018).

The fluid in the control volume is modeled by the assumption of constant density, with properties evaluated at a temperature of 25 °C, relative humidity of air 75% to 85% (Vale, 2018) and atmosphere pressure of 1atm, density of air 1.184 kg/m³, the dynamic viscosity assumed was 1.85E-5 kg/m.s and an steady state regime to solve the flow. The estimated Reynolds for this case is 195.556, turbulent regime for external flow.

To define the numerical field and solve the problem proposed it is necessary to approach the equations that govern the problem considering a permanent state flow that is the mass conservation equation, as in Eq. (1) and the linear momentum equation, as in x axis by Eq. (2), y axis in Eq. (3) and z axis in Eq. (4) given by Versteeg and Malalasekera (2007) due to Reynolds Averaged Navier Stokes equations:

$$\nabla \cdot (\rho \cdot \vec{v}) = 0 \quad (1)$$

$$\nabla \cdot (\rho UV) = -\frac{\partial P}{\partial x} + \nabla \cdot (\mu \nabla U) - \frac{\partial(\overline{\rho u^2})}{\partial x} - \frac{\partial(\overline{\rho u'v'})}{\partial y} - \frac{\partial(\overline{\rho u'w'})}{\partial z} + S_u \quad (2)$$

$$\nabla \cdot (\rho VV) = -\frac{\partial P}{\partial y} + \nabla \cdot (\mu \nabla V) - \frac{\partial(\overline{\rho v'u'})}{\partial x} - \frac{\partial(\overline{\rho v'^2})}{\partial y} - \frac{\partial(\overline{\rho v'w'})}{\partial z} + S_v \quad (3)$$

$$\nabla \cdot (\rho WV) = -\frac{\partial P}{\partial z} + \nabla \cdot (\mu \nabla W) - \frac{\partial(\overline{\rho w'u'})}{\partial x} - \frac{\partial(\overline{\rho w'v'})}{\partial y} - \frac{\partial(\overline{\rho w'^2})}{\partial z} + S_w \quad (4)$$

The three-dimensional, transient and random fluctuations of the properties, characteristics of the turbulent regime, require the application of a turbulence model. In the present study, the standard k - ε model was used. It works with the principle of turbulent viscosity (μ_t) (Eq. 5).

$$\tau_{ij} = -\overline{\rho u'v'} = \mu_t \frac{\partial \bar{U}}{\partial y} \quad (5)$$

where τ_{ij} is the Reynolds specific tension tensor and ρ is the specific mass. The turbulent kinetic energy is given by Eq. (6) and Eq. (7) for dissipation rate:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k \quad (6)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (7)$$

The reclaimer surface firstly passed by a geometric cleaning to remove shallow parts, pins, gears, thin structures and redundant components maintaining a uniform and close to real structure to the act of the air flow.

After that, the cleaned virtual geometry was loaded in the fluid dynamics software Ansys-Fluent meshing and automatic turned into a surface mesh model. Then, the model passes through a mesh cleaning and treatment for a second surface mesh, simpler than the primer one at the same time that connects all the parts and components in groups (Fig. 2).

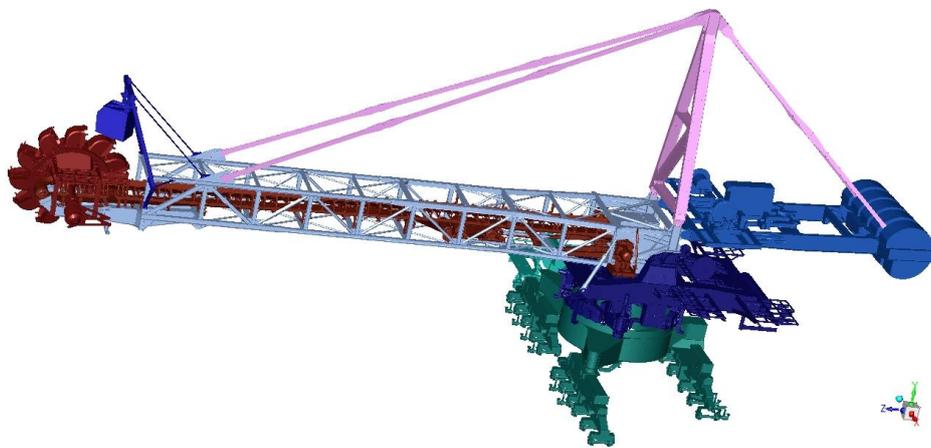


Figure 2. Geometry surface mesh with all parts connected.

A lateral positioning was chosen due to the higher moment of inertia that consequently generates higher loss of load and drag, providing a critical situation. The fluid zone was detached and a volumetric mesh was generated in the defined control domain, iterative simulation given the contour conditions and preprocessing of the results.

The physical models applied for simulation were transient, air as ideal gas, flow in turbulent regime, using the turbulence model $k-\varepsilon$ as in recent work of Fintelman *et al.* (2015), Zhu and Zhigang (2015) and Han and Han (2011). The face facing the side of the equipment was configured as a speed input with a constant value of 110 km/h, 30.6 m/s. The speed also corresponds to Mach 0,09.

Figure 3 shows the configuration of the virtual tunnel. The lateral walls and the ceiling were configured as plane of symmetry in sliding condition and ambient pressure equal to zero. The floor was configured in two parts, as slip surface in ground1 and non-slip surface in the ground 2 due to develop fluid from the inlet in it's the limit layer. The equipment has been configured with non-slip condition.

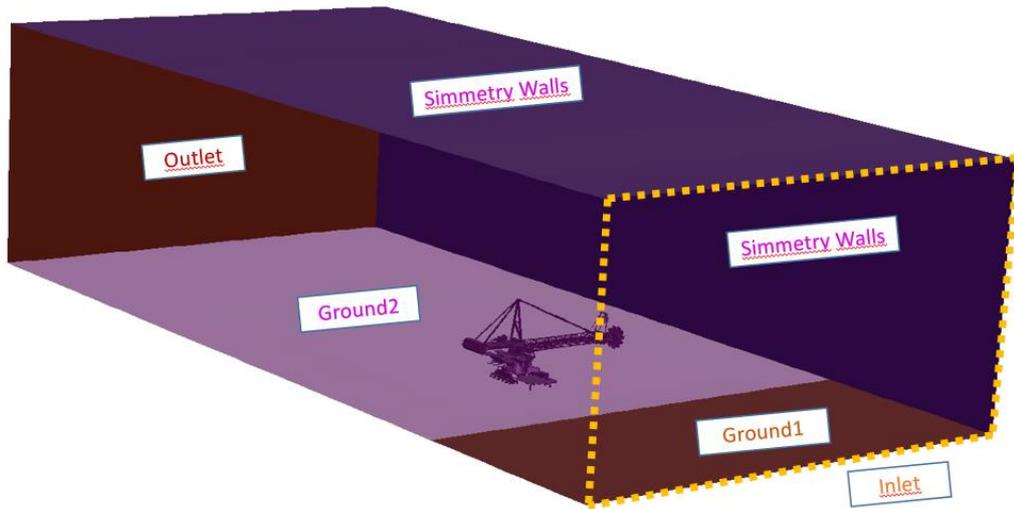


Figure 3. Configuration of the wind tunnel.

To define the mesh applied in the control volume, a coarse, a medium and a refined mesh were test using tetrahedral mesh. Table 1 shows the mesh configurations. Two refinement boxes, high and low, were used to help the iteration calculus convergence and to produce a volumetric mesh that attends the drag vortices close and behind to the equipment.

Table 1. Coarse, medium and refined mesh testing.

Mesh Type	Mesh Size (mm)	High Refinement Box (mm)	Low refinement Box (mm)	Tunnel Refinement (mm)	Volume elements
Coarse	1200(max)	2400(max)	5000(max)	10000(max)	4 million
Medium	600(max)	1800 (max)	4000(max)	8000(max)	12 million
Refined	300(max)	900(max)	1500(max)	4000(max)	38 million

The mesh test consisted in running the three models with different meshes until the residues drop down to 10^{-3} and by monitoring pressure and velocity of the 18 points represented in Fig. 4. The medium mesh was chosen.

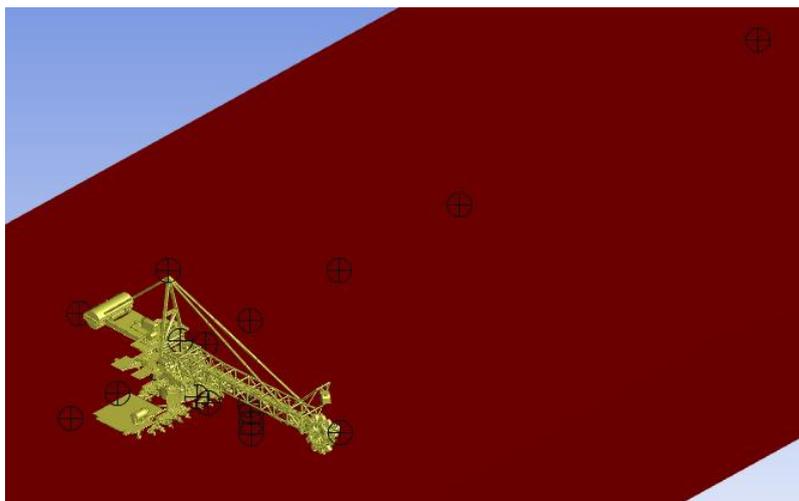


Figure 4. Monitoring points on the ore recovery and wind tunnel.

In order to obtain correct results close to the wall, it turns necessary to check the wall law, defined by von Karman (1930), which determines the average velocity of a turbulent flow in a fluid region. For this, the value of y^+ (Eq. 12), dimensionless parameter related to the distance of the wall to the center of the cell, the friction velocity and the kinematic viscosity (Schlichting and Gerstein, 2000) was used as the evaluation criterion.

$$y^+ = \frac{\Delta y^* U_t}{\nu} \quad (12)$$

As U_t the friction velocity, ν the kinematic viscosity and Δy^* is the distance from the wall.

Since the simulation is an external flow, to ensure good convergence, y^+ was guaranteed within the acceptable range, between 30 and 300 (Fluent, 2019). Considering the velocity stream of 30.6 m/s, the kinematic viscosity of the air about $1.562 \times 10^{-5} \text{ m}^2/\text{s}$ and desirable y^+ in about 100, the estimated first height of the wall in about 2mm. To conform the necessary prism layer with the base mesh 6 prisms were added with scaled growth from the necessary Δy^* to the mesh used.

The medium tetrahedral mesh chosen with 6 prism layers creates a mesh with 15 million elements (Fig. 5).

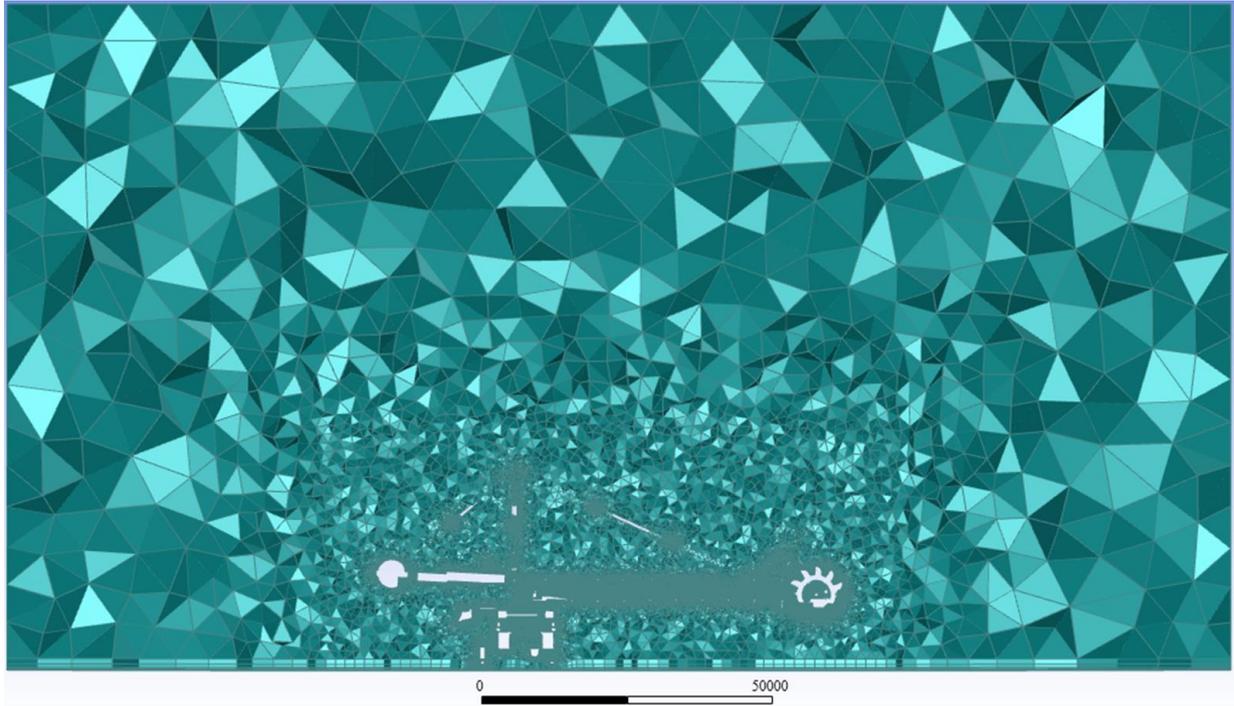


Figure 5. Bucket Wheel reclaimer tetrahedral mesh.

As the model did not involve temperature variations, the energy equation was not solved, and the expected results were pressure and velocity in the airflow around the reclaimer and its structure. It was expected that the model obtained convergence around 2000 ~ 5000 iterations (Fintelman *et al.*, 2015; Peren *et al.*, 2015).

3. RESULTS AND DISCUSSIONS

The residues stabilized between 10^{-3} and 10^{-5} after 1600 iterations. To process all the 15 million tetrahedral mesh running in a desktop with Intel Xeon E3-1270 v6 with 8 cores and 32gb of memory the time cost was about 10 hours. The y^+ average value found was around 180 in the reclaimer structure, as expected, between 30 and 300.

After the running process iteration and convergence confirmation, the post processing data was taken. The results indicate as expected, a pressure accumulation on the faces of the reclaimer that offers a barrier to the flow (Fig.6) with the wind speed of 30.7m/s as an inlet input. The maximum pressure value obtained was around 780 Pa in spots close to the cabin pointed as 1, in the first part of the spear structure pointed as 2 and in the end of the spear pointed as 3 in the Figure 6. The average pressure in the front of the structure was around 600 Pa.

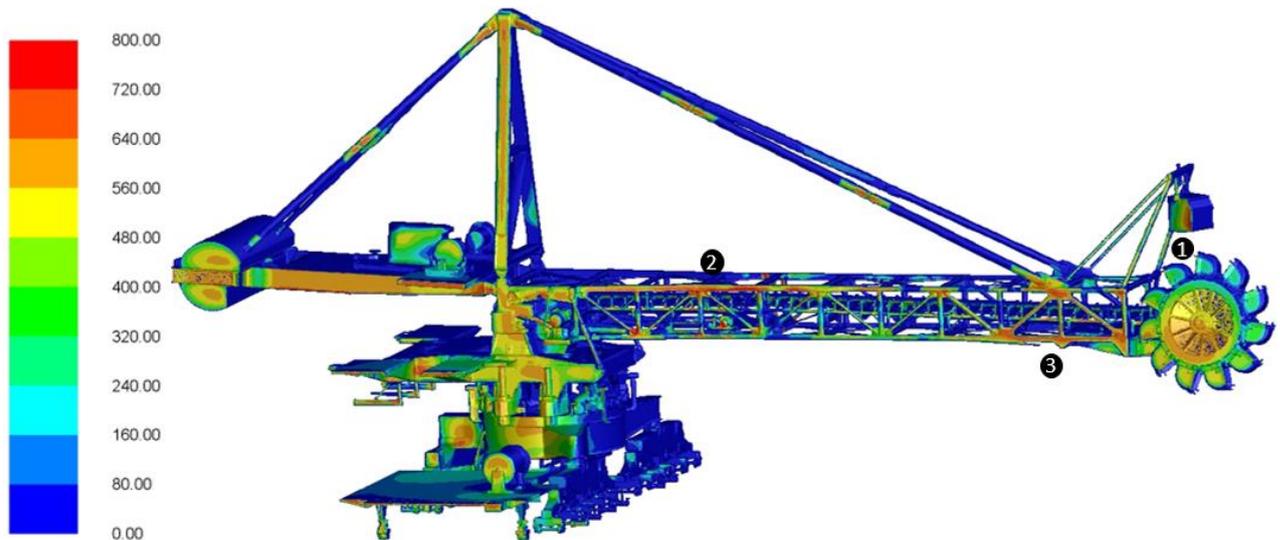


Figure 6. Static pressure result on the face of the bucket wheel reclaimer.

As the flow only passes through the easy way it runs off the regions with high pressure profile increasing the velocity on the sides (Fig. 7). At the same time that the pressure grows in front of the reclaimer the pressure behind the structure falls due to the bigger front area blockage creating a long turbulence vortex in the wind direction.

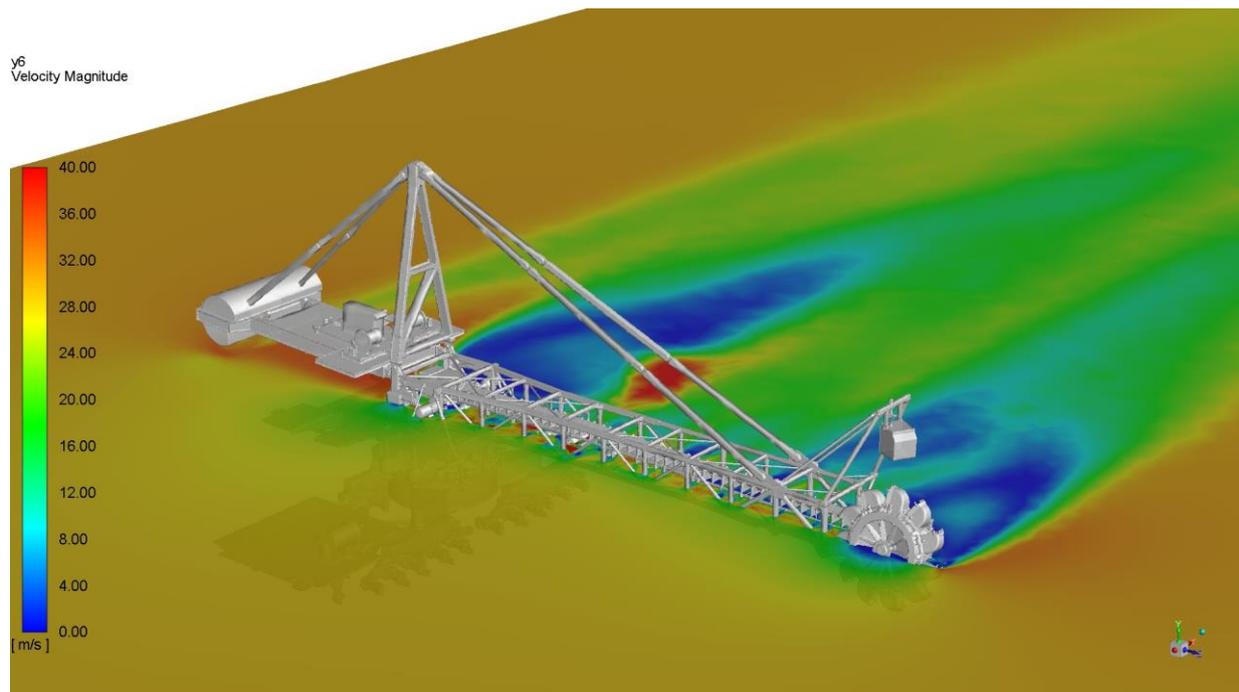


Figure 7. Velocity Magnitude results in a y-plane in the middle of the structure.

The air flow shocks with the structure creates a pressure difference in all the paths and geometry connections between the face that blocks the wind and the face behind it creating a high to low pressure shape profile facilitating the air to pass through this passages with a velocity addition and assume a different form depending on the blockage (Fig. 8 and Fig.9).

This intermittent flow behind the structure mix with the air with zero velocity and creates different groups volumes surrounding by high speed air in the turbulence vortex. When the vortex gain distance from the structure in x direction it merges with the flow ending like a conic tail.

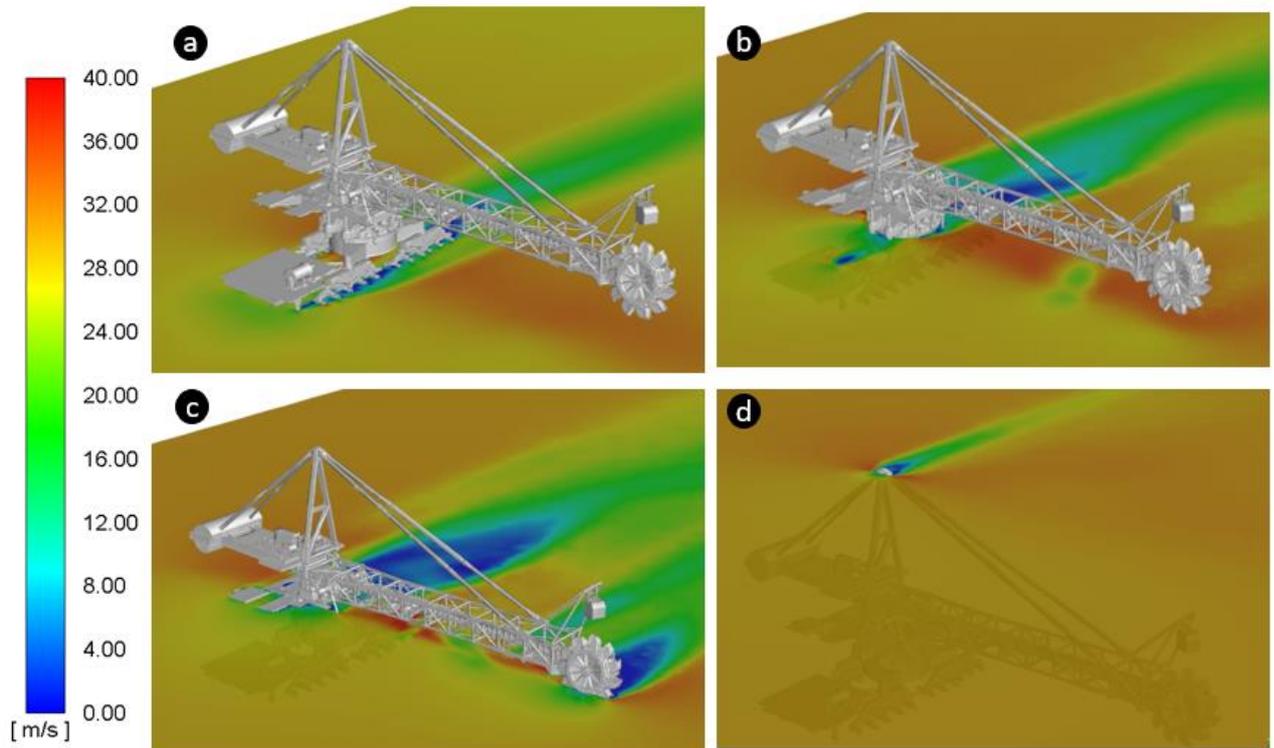


Figure 8. Velocity Magnitude results in 4 planes in y axis. (a) reclaimer feet; (b) reclaimer belt; (c) Reclaimer spear; (d) Reclaimer mast.

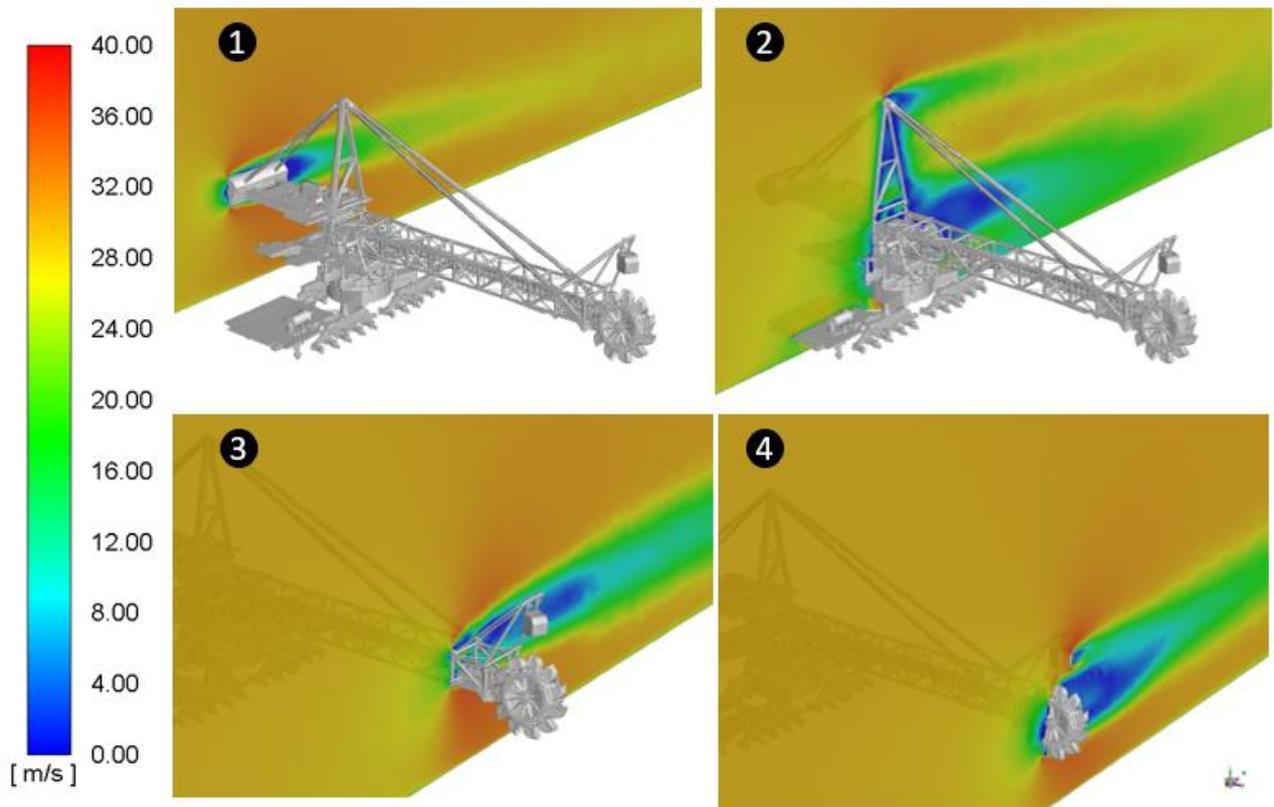


Figure 9. Velocity Magnitude results in 4 planes in z axis. (1) reclaimer counterbalance; (2) reclaimer middle; (3) Reclaimer braces connection with spear; (4) Reclaimer bucket wheel.

As it's possible to see in Figure 8, (b) and (c) and in Figure 9, (3) and (4), the air flow that finds the structure increases its velocity mainly in the spots close to the structure. The resultant of a 30.7 m/s air speed in inlet is a maximum 40 m/s speed close to the bucket wheel reclaimer in the tunnel, like 30% increase of air speed.

As the static pressure the most valuable result in where this paper is focused, the results presented an average 600Pa of pressure around the frontal area in contact with the storm wind. As it was said in the start of this analysis the resultant pressure obeys what's expected producing the high values in places with more blockage as shown in Figure 10.

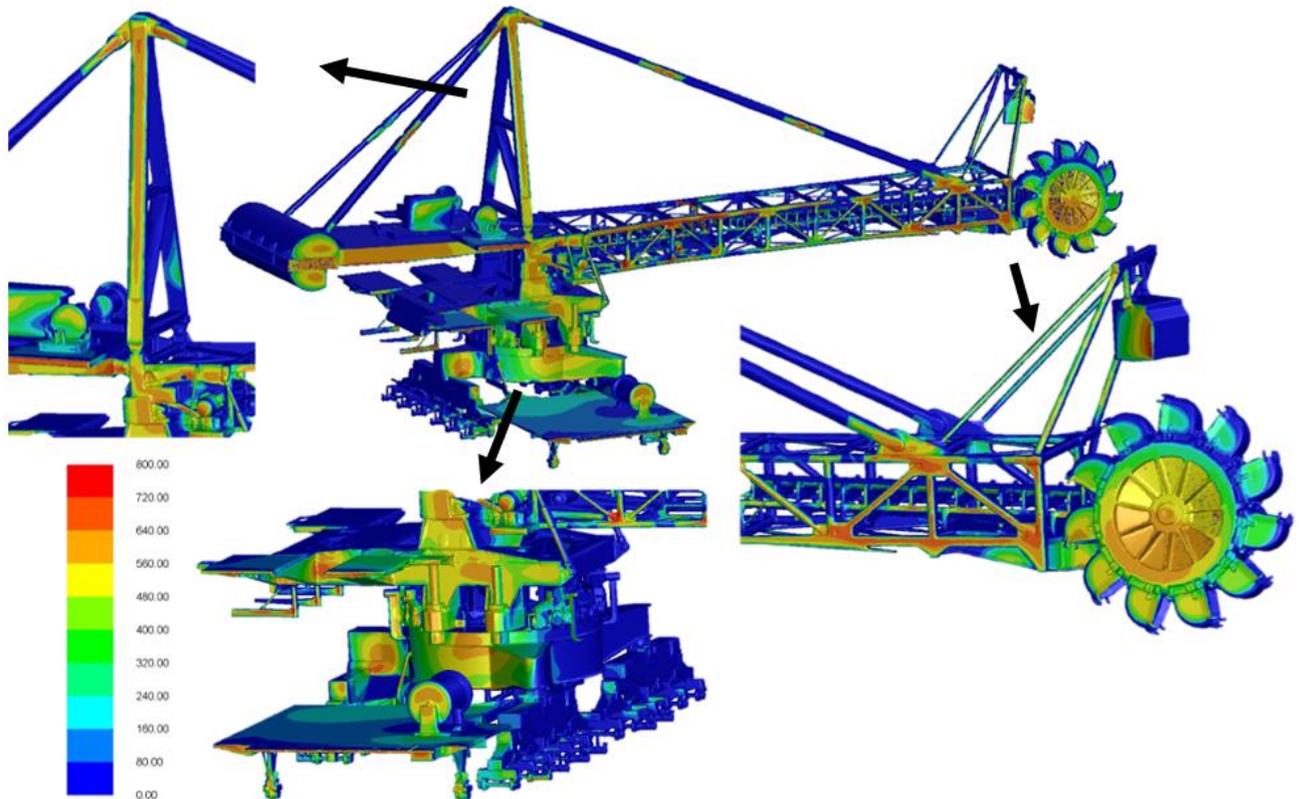


Figure 10. Static pressure results of the Bucket Wheel Reclaimer with details.

The bucket wheel is about 54 meters far from the base, or better saying, offering a 54 meters' lever arm, with a 600 Pa applied on the extremity creating a spot with high moment applied. As the reclaimer bucket wheel, the mast, and the counterbalance parts offers more inertia and moment by the force resultant of the wind pressure. This places aggregated with the pressure in the base could made the structure win the frictional force in the rails and be dragged, twist or topple. Couple the results found with structural analysis in a fluid structure interaction (FSI) could aggregate value to the results found. Furthermore, it's important to realize experimental tests to confront the results found in the simulation in order to correlate the model and validate the methodology.

4. CONCLUSIONS

The necessity to understand the dynamics of wind crossing a huge structure that actuates in harbors within a critical velocity motivates this paper. The model based in the real geometry of an ore recovery from Vale. To the simulation it was chosen a position of work that presents a large area blockage, all the geometric parts were prepared into a CFD model and it was assembled in a wind tunnel with a tetrahedral mesh. After a mesh test and the testimony of results in an acceptable level of convergence the medium mesh was chosen. The medium geometry offers a better coast-benefit because it offers a lower time per iteration than the refined one, and a lower size occupation with almost the same results.

Within the analysis of the results, it was concluded that the resultant of a 30.7 m/s air speed in inlet is a maximum 40 m/s speed close to the bucket wheel reclaimer in the tunnel, like 30% increase of air speed found in some leaked spots and structure corners, where the wind escapes. As expected, the turbulence generated created vortices behind the recovery with a large profile of velocities in a large volume in x axis direction, which, in some places, was close to zero, almost behind the structure and in the face that receives the wind-blown. This air stoppage was caused by the instant decrease of the air velocity, creating a pressure rise and accelerating the air in the corners, where it's easy for the air to scape.

The static pressure profile found in the frontal area presented an average distribution around 600 Pa. Since the bucket wheel reclaimer a big sized equipment with a heavy weight it's not simple to topple or being dragged once it has technology to avoid accidents, but, the momentum generated by the wind exciting the structure and a lever arm from the base to the bucket wheel in about 54 meters could generate force enough to a structure drag along the rails by overcoming the frictional force and clamping between the rail and the reclaimer rolling wheels or a structure twist changing the gravity center and made the structure topple. A fluid structure interaction (FSI) could present results to aggregate value to the results found on this paper. Aiming the validation of the model and method it's important to run experimental tests to confront the results found in the simulation and correlate it. This will be done in future studies.

5. ACKNOWLEDGEMENTS

This study was financed in part by the Coordenação de Aperfeiçoamento de Pessoal de Nível Superior - Brasil (CAPES) - Finance Code 001. The authors also thank to FAPEMIG, CNPq and Vale for their financial support.

6. REFERENCES

- Bak, S., Yoo, J., Song, C., 2013. "Fluid-structure interaction analysis of deformation of sail of 30-foot yacht". *Int. J. Naval Archit. Ocean Eng.*, Vol. 5, pp. 263-276.
- Blocken, B., Torpalar, Y., A following car influences cyclist drag: CFD simulations and wind tunnel measurements. *Journal of wind engineering and Industrial Aerodynamics*. Ed.145, p.178–186, 2015.
- Braun, A.L., Awruch, A.M., 2008. "Finite element simulation of the wind action over bridge sectional models: Application to the Guamá River Bridge (Pará State, Brazil)". *Finite Elements in Analysis and Design*, Vol. 44, pp. 105 – 122.
- Estado de Minas. "Temporal com ventos de mais de 100 km/h causa estragos no Rio". 28 Mar. 2019. <https://www.em.com.br/app/noticia/nacional/2019/02/06/interna_nacional,1028532/temporal-com-ventos-de-mais-de-100-km-h-causa-estragos-no-rio.shtml>.
- Fintelman, D., Hemida, H., Sterling, M., Li, F., 2015. "CFD simulations of the flow around a cyclist subjected to crosswinds". *Journal of wind engineering and Industrial Aerodynamics*, Vol. 144, pp. 31-41.
- Fluent, "Datashet and Help". June 2019, Fluent version 19. Available only for computers with the software installed.
- Gazeta Online. "Marinha comunica formação de ciclone e ventos podem chegar a 88 km/h". 28 Mar. 2019 <<https://www.gazetaonline.com.br/noticias/cidades/2019/03/marinha-comunica-formacao-de-ciclone-e-ventos-podem-chegar-a-88-km-h-1014173580.html>>.
- Han, D., Han, G., 2011. "The difference in the uplift force at each support point of a container crane between FSI analysis and a wind tunnel test". *Journal of Mechanical Science and Technology*, Vol. 25, pp. 301-308.
- IBRAM. 2015. "Informações sobre a economia mineral brasileira 2015". *Relatório do Instituto Brasileiro de Mineração – IBRAM*.
- Janssen, W.B., Blocken, B., Wijhe, H.J., 2017. "CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications". *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 166, p. 106-116.
- Liu, B., Qu, J., Zhang, W., & Qian, G. (2011). Numerical simulation of wind flow over transverse and pyramid dunes. *Journal of Wind Engineering and Industrial Aerodynamics*, 99(8), 879-888.
- Nozu, T., Tamura, T., Take-shi, K., Akira, K., 2015. "Mesh-adaptive LES for wind load estimation of a high-rise building in a city". *J. Wind Eng. Ind. Aerodyn.*, Vol. 144, pp. 62–69.
- Peren, J.I., van Hoff, T., Leite, B.C.C., Blocken, B., 2015. "CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: Impact of roof angle and opening location". *Building and Environment*, Vol. 85, pp. 263-276.

- Qing, G., Zhangyan, Z., Gaoyao, D., & Zhijian, Z. (2017). CFD-based analysis on wind load characteristics of portal crane. *Hoisting and Conveying Machinery*, (3), 9.
- Ramasamy, D., Mohanesan, K., Kadirgama, K., Noor, M. M., & Rahman, M. M. (2017). Hybrid electric vehicle car body drag analysis using computational fluid dynamics. *International Journal of Automotive and Mechanical Engineering*, 14, 4496-4507.
- Rocchi, D / Rosa, L. / Sabbioni, E. / Sbroisi, M. / Belloli, M. A numerical–experimental methodology for simulating the aerodynamic forces acting on a moving vehicle passing through the wake of a bridge tower under crosswind. *Journal of Wind Engineering and Industrial Aerodynamics*. Ed. 104, p.256–265,2012.
- Schlichting, H, Gersten, K., 2000. *Boundary-Layer Theory*, Ed Springer, Bochum, 8th edition.
- Tsubokura, M., Nakashima, T., Kitayama, M., Ikawa, Y., Doh, D.; Kobayashi, T. “Large eddy simulation on the unsteady aerodynamic response of a road vehicle in transient crosswinds”. *International Journal of Heat and Fluid Flow*, v. 31, p. 1075-1086, 2010.
- Vale. “Regulamento do terminal marítimo de ponta da madeira”. 15 Mar. 2018 <http://www.vale.com/PT/business/logistics/shipping/ship_tracking/Documents/regulamento_terminal_ponta_madeira-PT.pdf>.
- Versteeg, H.K., Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics – The Finite Volume Method*. Pearson Education Limited, Edinburgh Gate Harlow, 2^a ed.
- Von Kármán, T., 1930. “Mechanical Similitude and Turbulence”, *Technical Memorandum NACA*, no. 611.
- Wei, C. H. E. N., Xian-rong, Q. I. N., & Zhi-gang, Y. A. N. G. (2018). Wind load characteristics analysis of mast and jib of tower crane. *Journal of ZheJiang University (Engineering Science)*, 52(12), 2262-2270.
- Zang, Y., Borna, A., Khurram, R., Habashi, W., 2012. “Hybrid RANS/LES method for FSI simulations of tall buildings”. *The 2012 World Congress on Advances in Civil, Environmental, and Materials Research (ACEM' 12)*. Seoul, Korea.
- Zhu, H., Zhigang, Y., 2015. “Fluid–structure interaction study of three-dimensional vehicle model under crosswind”. *Advances in Mechanical Engineering*, Vol. 7, pp. 1-10.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.