DINAME 2023 - Proceedings of the XIX International Symposium on Dynamic Problems of Mechanics,
A.S. De Paula, A.T. Fabro and M.R. Machado (Editors), ABCM.
Pirenópolis, Go, Brazil, February 26th to March 3rd, 2023

Francis modeling over generating units applying advanced fluid dynamics methods

Leandro J. L. Stival[1], **João Marcelo Vedovotto**[1], **Rodrigo Cavalcanti Ribeiro**[1], **Freddy A. P. Morales**[1], **Leonardo Campanine Sicchieri**[1,2] **and Aldemir A. Cavalini Jr.**[1]

[1] Federal University of Uberlândia, Av. João Naves de Ávila, 2121 - Uberlândia, Minas Gerais, Brazil

[2] Technical University of Denmark, Anker Engelunds Vej 101 2800, Kongens Lyngby, Denmark

Abstract: The use of Large Eddy Simulation o (LES) has increased for rotor simulations due to the increased computational power available, where mathematical formulation over a generating unit virtual machine are possible nowadays. The numerical framework used in the simulations performs LES under a Cartesian block-structured mesh that is dynamically refined via an adaptive mesh refinement (AMR) to increase accuracy and reduce computational costs, even with an initial mesh configuration with approximately 13 million control volumes. Preliminary analyses demonstrate that the LES-IB methodology is capable of simulating the interaction within the Francis turbine and flow structures, and well capturing the blade's passage frequency. It is important to emphasize that the result presented here is pioneer in the sense of considering a transient simulation, with dynamic modeling of the turbulence of a Francis turbine, considering the current preliminary results, the analyzes are promising in the developing context.

Keywords: Large eddy simulation, Francis turbine, immersed boundary, adaptive mesh refinement

INTRODUCTION

The development of Computational Fluid Dynamics (CFD) has expanded to problem application complexes involving fluid-structure interaction. Numerical modeling has been used in energy projects to solve the properties of turbulent flows in detail. In particular, the use of Large Eddy Simulation (LES) has increased for rotor simulations due to the increased computational power available. Because of that, the application of LES by renewable energy research groups is increasing and stimulating the evolution of the methodology. LES is based on a spatial filtering process of the governing equations, which allows an explicitly solution to the largest turbulent structures on computational meshes and to model only the smallest scales. This method, in conjunction with the Immersed Boundary (IB) method, is the state-of-the-art approach to simulate flows with high Reynolds numbers, with fluid-structure interactions involving complex and moving geometries. The approach is known as immersed type methods, in which the immersed boundary method (IB) is a classical example (Peskin, 2002). The IB method arose as a strong and effective simulation tool for dealing with complex flows, because of its capacity to take care of complex structures without the need for expensive and complicated dynamic meshing techniques (Sotiropoulos and Yang, 2014). Due to the meshing flexibility, the IB approach has acquired significant popularity over the past two decades for several problems, especially for heavy movements in solids or large deformation in the fluid (Mittal and Iaccarino, 2005a).

In this article, the results of three-dimensional simulations of the complete system will be presented, considering the flow through the spiral case and the rotor (Francis turbine) in motion. Thus, the concepts involved in the mathematical formulation over a generating unit virtual machine (UG). Moreover, details about the mechanical, electrical, and hydraulic forces that act on the system and a previous sensitivity analysis will be presented to demonstrate the development of an innovative project on the dynamic responses of a UG. In the simulations, the rotor and the upstream and downstream flow will be instrumented with virtual pressure and velocity probes to evaluate the intensity of turbulence and determine information such as emission frequencies and vortex passage in specific regions of interest.

METHODOLOGY

Mathematical and numerical modeling

The numerical framework used in the simulations performs LES under a Cartesian block-structured mesh that is dynamically refined via an adaptive mesh refinement (AMR) to increase accuracy and reduce computational costs. The framework employed to produce the present work is MFSim, developed in the Fluid Mechanics Laboratory of the Federal University of Uberlândia, Brazil. It has been henceforth expanded into a multi-disciplinary code for simulating 3D problems implicating turbulent flows (Damasceno *et al.*, 2015), fluid-structure interaction (Neto *et al.*, 2019; Souza *et al.*, 2022; Stival *et al.*, 2022), multiphase flows (Barbi *et al.*, 2018; Pinheiro *et al.*, 2019, 2021), gas-solid and gas-liquid flows (Santos, 2019), and chemically-reactive flows (Damasceno *et al.*, 2018; Castro *et al.*, 2021).

This work employs the immersed boundary (IB) method to represent the unit generation (Vedovoto *et al.*, 2015; Neto *et al.*, 2019; Souza *et al.*, 2022; Stival *et al.*, 2022). The methodology requires two different meshes: first, a block-structured Eulerian field where the transport equations are solved utilizing a finite volume technique with second-order discretization, and then an unstructured Lagrangian domain describing the immersed unit generation. Distinct methods of IBs can be encountered in several reviews concerning the topic (Peskin, 2002; Mittal and Iaccarino, 2005b; Iaccarino and Verzicco, 2003; Sotiropoulos and Yang, 2014; Mohammadi *et al.*, 2018). The major distinctions between the different published IB approaches proposed in the literature are the way to calculate the force and the interpolation stage. Incorporating an IB technique with a block-structured Cartesian mesh and AMR allows efficient solvers to be employed within Cartesian blocks while permitting better efficient refinement of the mesh exclusively in areas demanded by the flow characteristics.

The modeling of the flow is based on the Navier-Stokes equations, a set of four partial differential equations that can be solved for a description of the velocity and pressure fields. In the present work, these equations are solved using the LES methodology. This methodology is particularly suitable for the simulation of turbines since it enables the selection of the large eddies of the flow to be calculated explicitly. In contrast, the energy exchange between modeled and resolved structures is calculated based on the sub-grid scale (SGS) models, which are smaller than the grid.

In cartesian coordinates and using index notation, for $i, j = 1,2,3$, applying the LES filtering process over the mass and momentum balance equations, and assuming the cumulative property over both operators, it is possible to obtain the equations for the filtered velocities. These equations may be written as:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0. \tag{1}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \tau_{ij}\right] + f_i, \tag{2}$$

where ν (m^2/s) is the kinematic viscosity, u_i (m/s) is the i component of velocity vector, p (N/m^2) is the pressure, ρ (kg/m^3) is the fluid specific mass,μ (kg/ms) is the dynamic viscosity, and f_i (N/m^3) is the i component of the Eulerian dynamic force vector representing the immersed boundary method.

Note that the filtering process introduces a new variable in the second last term of the right-hand side of the momentum equation, Eq. (2). This term is a sub-filter tensor known as Reynolds Stress Tensor (RST), which represents the contribution of the dynamics of the sub-filter turbulent fluctuations on the resolved scales of LES, written as:

$$\tau_{ij} = \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right), \tag{3}$$

where μ_{SGS} is the subgrid viscosity. For the simulation of turbulent flows with a high Reynolds number, the Large Eddy Simulation methodology was used with the Germano Dynamic Subgrid-Scale model (Germano *et al.* (1991); Lilly (1992)). The turbulent viscosity is calculated using the tensor strain rate $\overline{S}_{ij}$, the length scale Δ, which is usually defined by the mesh size and from the proportionality function $c(\vec{x},t)$. The turbulent viscosity is calculated using Eq. (4):

$$\nu_t = c(\vec{x},t)\Delta^2 \left|\overline{S}\right|, \tag{4}$$

In connection with the Eulerian formulation, which is represented by the LES method, the Langrangian formulation of the work is based on the immersed boundary method framework. The immersed boundary method uses an independent grid to define the body inside the fluid flow. Fig. 1 shows an example of an Eulerian domain Ω, which is used to model the fluid flow, and a Lagrangian domain Γ that models a solid sphere immersed in the fluid domain. The volume $\Delta\Omega$ of the Eulerian element is h^3, where h is the volumes' length in the three directions. The Lagrangian mesh is constructed in such a way that the Lagrangian volume $\Delta\Gamma$ is equal to the Eulerian volume (Uhlmann (2005)); i.e., $\Delta\Gamma = \Delta A \cdot h = h^3$, where ΔA is the area of the Lagrangian element, then $\Delta A = h^2$.

The main advantage of this methodology is the capability of simulating flows over complex geometries applying, for instance, a cartesian grid to solve the balance and transport equations on Eulerian referential. The coupling of the two referential is possible through the use of source terms, as presented in Eq. (5). In the present work, the method retained is a modification of the multi-direct forcing method of Wang *et al.* (2008). The modifications are fully presented elsewhere in Vedovoto *et al.* (2015). This methodology applies an iterative process of the direct forcing method and is better detailed in works as Neto *et al.* (2019).

The force term, f_i, in the momentum equation controls the definition of the immersed boundaries. In order to calculate this force is commonly utilized a distribution function:

$$f_i(\vec{x}) = \sum_K \vec{F}(\vec{x}_K) D_{ij}(\vec{x} - \vec{x}_K) \Delta V(\vec{x}_K), \tag{5}$$

where $\vec{x}$ is the Eulerian coordinate, $\vec{x}_K$ is the Langrangian element coordinate, $\Delta V(\vec{x}_K)$ the volume of the control volume i and D_{ij} represents the distribution function. This work applies the hat function, where Δ is the characteristic length of the

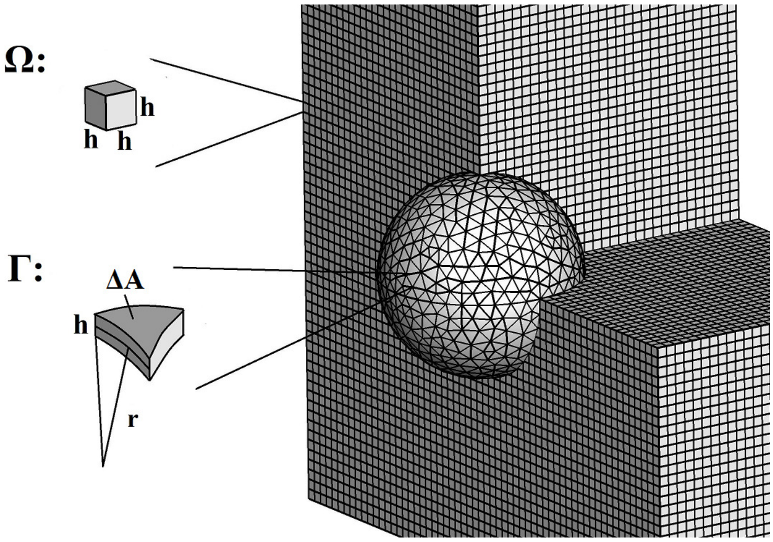


Figure 1: Eulerian and Lagrangian domains applied in the MFSim

Eulerian grid:

$$D_{ij}(\vec{x}) = g(\vec{x}_K - \vec{x})\, g(\vec{y}_K - \vec{y})\, g(\vec{z}_K - \vec{z}). \tag{6}$$

This function presents an essential description when the Eulerian volumes are far from the Lagrangian points, and the force value is distributed at those points. Besides that, a crucial property of it is related to the integration over r, obtaining a unitary value, which is due to the conservative distribution process of the function (Mittal and Iaccarino, 2005a). Therefore, $\vec{F}(\vec{x}_K)$ stands for the force at a Lagrangian point, which will be distributed over the Eulerian field to delimit the boundary. Moreover, the momentum equation is still valid in each of the Lagrangian points (Vedovoto *et al.*, 2015).

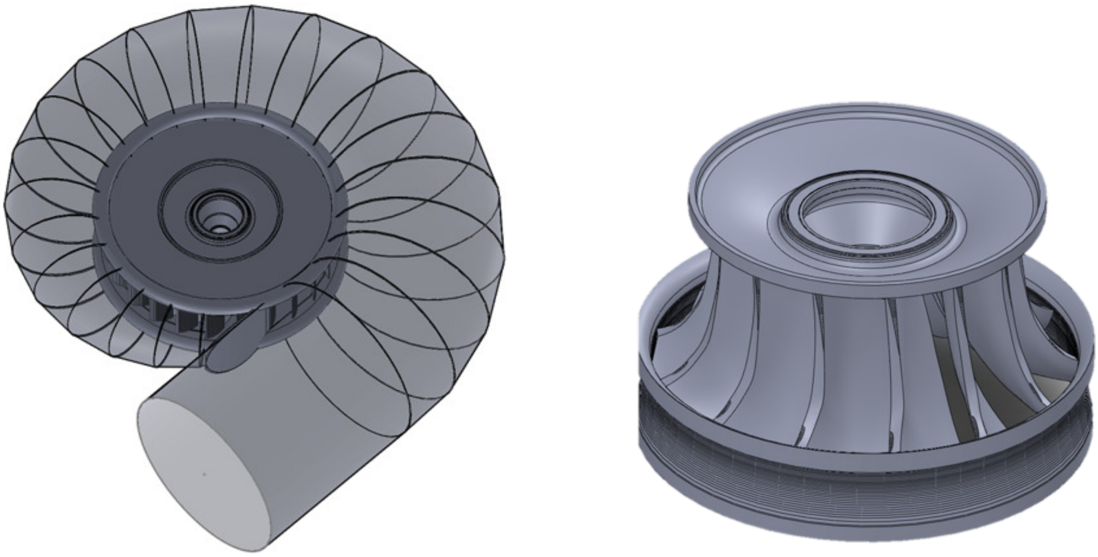
The above paragraphs show that the mathematical formulation involving the LES and IB methods is based on two different referential. The first approach correlates with the Eulerian coordinate system, while the second brings the Lagrangian perspective of the immersed body distributed over the flow. For the Eulerian referential, the momentum and mass balance equations provide the filtered velocity and pressure fields. However, the velocity field and position provided from Eulerian referential are transferred to Lagrangian points to calculate the velocity of the body movement at the Lagrangian points, after that, the source terms F_i are evaluated at those points. Further on, this source term is also distributed over the Eulerian domain, and the cycle continues iteratively for the subsequent time step until the convergence criteria reach the minimum residual established for the case. Therefore, the main advantage of employing an iterative process is improving the precision that the calculation can accomplish.

Computational domain and boundary conditions

CFD simulation uses the fundamental laws of mechanics, which govern fluid dynamics equations, to formulate a physical problem mathematically. Once developed, computing resources use numerical methods to solve the equations using MFSim to obtain numerical solutions for the physical properties involved. The accuracy of the simulation in MFSim depends on the fidelity of the model, approximations and assumptions used, experimental validation, and available computational resources. It is essential to plan the simulation in MFSim to use it as an effective tool in design and analysis.

In this work, the smallest element is a cube with edges with a dimension of 75 mm, and the computational domain was subdivided into 24 x 28.8 x 19.2 m of the control volume. Numerically, this means that for the solution of the transport equations, at each iteration (advance of time step), about 13 million equations must be solved simultaneously if we use a uniform mesh, even though parallel processing alleviates the computational cost of solving these equations. However, we are using a block-structured mesh with local refinement around the cells of the Lagrangian mesh. With that, there is a significant reduction in the number of volumes to be simulated. Fig. 2 demonstrate the geometry of the spiral case and rotor, while Figs. 3 and 4 demonstrate the correlation between the Eulerian mesh and the superficial Lagrangian mesh representing the solid domain, and Fig. 5 shows the turbine example GIF of the rotating body inside the domain through the AMR technique.

The simulations aim to be performed with an adaptive mesh of 4 levels. The initial mesh arrangement is configured with 20 x 24 x 16 volumes for the coarsest grid (base level) in the x, y, and z directions using the hexahedral uniform grid, obtaining an initial calculation with approximately 13 million control volumes. From this initial configuration, a mesh refinement was implemented around the blades. The inlet flow condition was performed using a nominal value of 483 m_3/s, inlet velocity was u=9.42, v=0, and w=0 m/s, in the MFSim code, characterizing a type of Dirichlet boundary



(a) (b)

Figure 2: (a) Spiral case and (b) Francis turbine geometry

condition. The boundary conditions for the side planes (xz-planes), upper plane (xy-plane), and lower plane (xy-plane) establish a Neumann boundary condition.

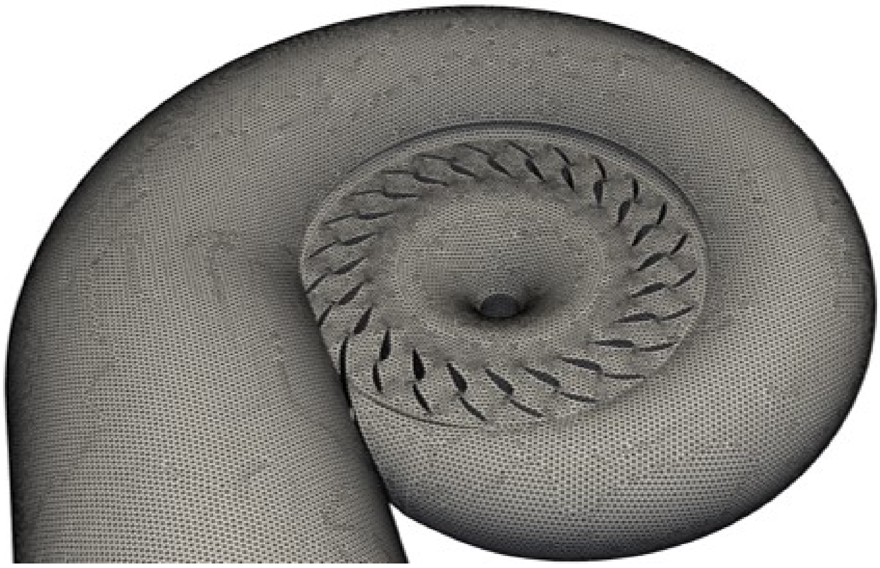


Figure 3: Lagrangian mesh that will represent the immersed boundary UG

RESULTS AND DISCUSSION

Analysis of the Flow Turbulent Structures

This section presents analyses to demonstrate that the LES-IB methodology is capable of simulating the interaction within the Francis turbine and flow structures. Fig. 6 shows a dynamic representation of the iso-surfaces colored by streamwise velocity, in order to provide important visualization of eddies occurring over the turbulent flow. Vortex structures occur in the region close to the blade being transported over the flow, and the vortices structure patterns are straight connected to the turbine's operational parameters.

Moreover, in order to restate the qualitative analysis, Figure 7 displays instantaneous iso-surfaces colored by vorticity magnitude with a 3D visualization. Figure 7 shows values of vorticity, over 60 (1/s), are occurring in the near the blades and downstream of the flow develops. While, lower values of vorticity occurs over the spiral case due the flow pattern, where the maximum range can reach values around 40 (1/s). In addition, Figure 7 displays the highest values of vorticity

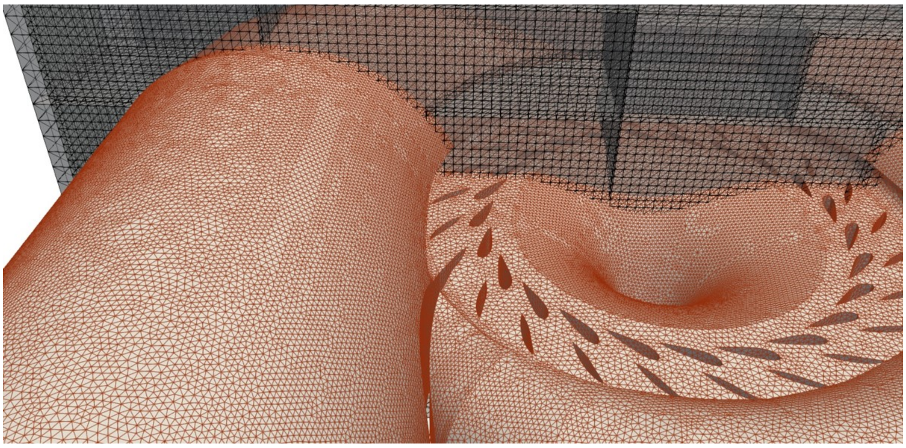


Figure 4: Connection between the Eulerian and Lagrangian domains applied in the MFSim

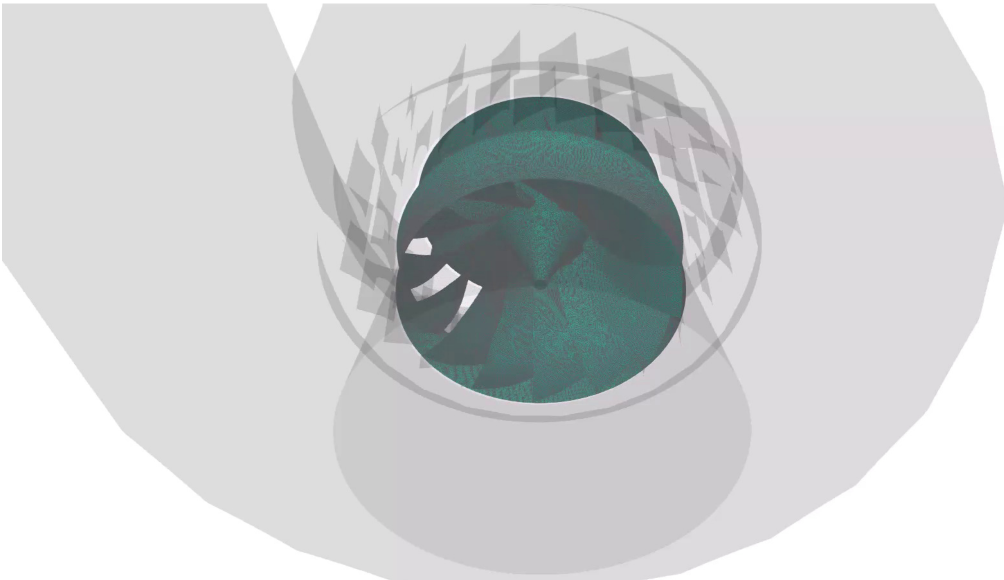


Figure 5: Francis turbine rotating due the AMR in the MFSim

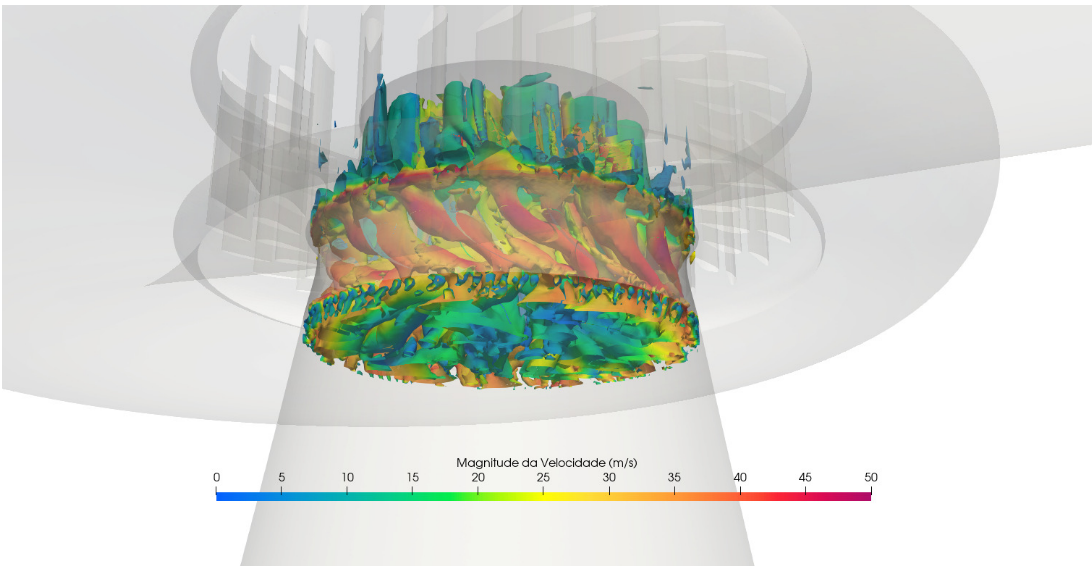


Figure 6: Vortex visualization coloured by velocity magnitude over the domain

around the blades top close to 160 (1/s).

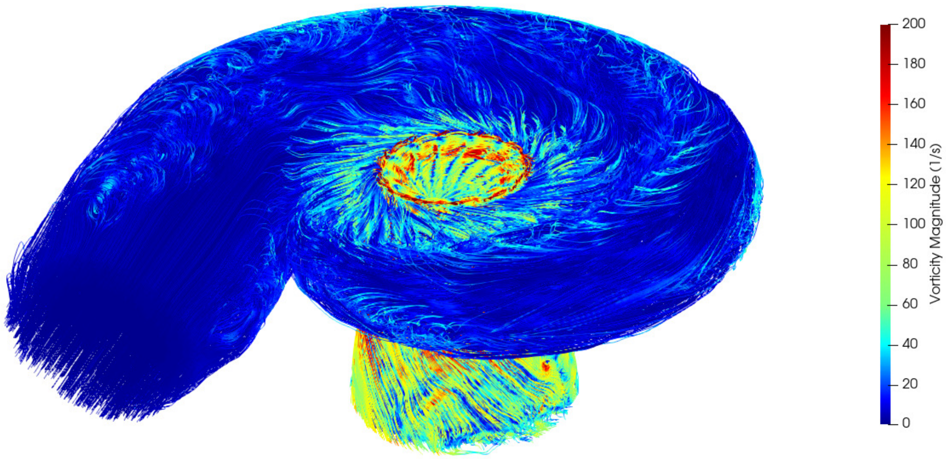


Figure 7: Vortex visualization using iso-surfaces coloured by velocity

Energy Spectrum

According to the Pope (1999), approximately 80% of the turbulent kinetic energy needs to be resolved in a well-resolved LES. A partial verification of the numerical results can be performed by computing the power spectral density of the turbulent kinetic energy (Chen *et al.*, 2022). The following section describes the power spectral density (PSD) of the turbulent kinetic energy, $E(f)$, $E_{(f)}$energy spectrum as a function of a wave number for the LES simulations within the wind turbine wake to study the wake instability quantitatively. The $E(f)$ time series of distinct points in the wake region were recorded and used to analyze the PSD distribution, and the power law decay $k^{-5/3}$ was added for better comparison.

From preliminary analysis, the simulation was able to capture correctly the blade's passage frequency, close to 19.5 to 20 Hz. In order to exemplify the preliminary agreement of the simulations results with the real Francis turbine, Fig. 8 presents the turbulent kinetic energy spectrum, $E(f)$, around the turbine, in order to capture the blade's passage.

Evaluation of Power Generation

This section presents the results obtained from the temporal evolution of the torque and power of the UG. Since, the geometry is incorporated into the fluid domain by a force term in the Navier-Stokes equations, by the immersed boundary method, an advantage is to obtain, with post-processing, the torque and power generated by the turbine over a structure immersed in the flow. The steps of this implementation consisted of:

1. To calculate the distance from the wind turbine rotation reference position to the center position of each Lagrangian cell:

$$r_x = x_c - \tilde{x}_k, \tag{7}$$
$$r_y = y_c - \tilde{y}_k, \tag{8}$$
$$r_z = z_c - \tilde{z}_k, \tag{9}$$

   where x_c, y_c, z_c are the rotation reference positions of the immersed boundary and $\tilde{x}_k, \tilde{y}_k, \tilde{z}_k$ are the center positions of each Lagrangian cell in each direction. x_{ic}rotation reference position of the immersed boundary $\tilde{x}_{ik}$center positions of each Lagrangian cell

2. To calculate the Torque through the summation in the Lagrangian space of the cross product of force and distance in x, y and z.

$$T = \sum_{\Gamma} r \times F, \tag{10}$$

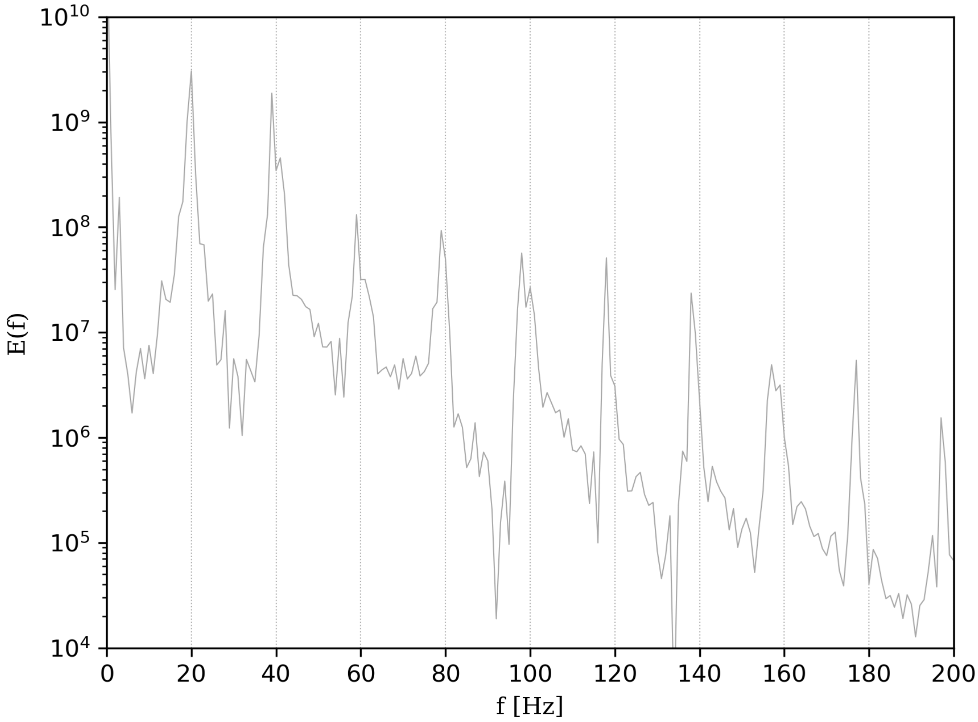


Figure 8: Blade's passage frequency

where F represents the force magnitude of the Lagrangian field that promotes the immersed boundary rotation. Also illustrate on Fig. 9.

$$r \times F = \begin{vmatrix} i & j & k \\ r_x & r_y & r_y \\ F_x & F_y & F_z \end{vmatrix} \tag{11}$$

3. Calculating mechanical Torque, the mechanical Power generation is obtained.

Hence, in terms of torque represented the MFSim results presented a mean and standard deviation of 24.3±10.2 (MNm), while the power generation showed mean values and standard deviation for the MFSim simulation corresponds to 229.5±11.3 (MW).

Finally, it is important to emphasize that the result presented here is pioneer in the sense of being one of the first complete, transient simulations, with dynamic modeling of the turbulence of a Francis turbine, considering the current preliminary results, the analyzes are promising in the developing context.

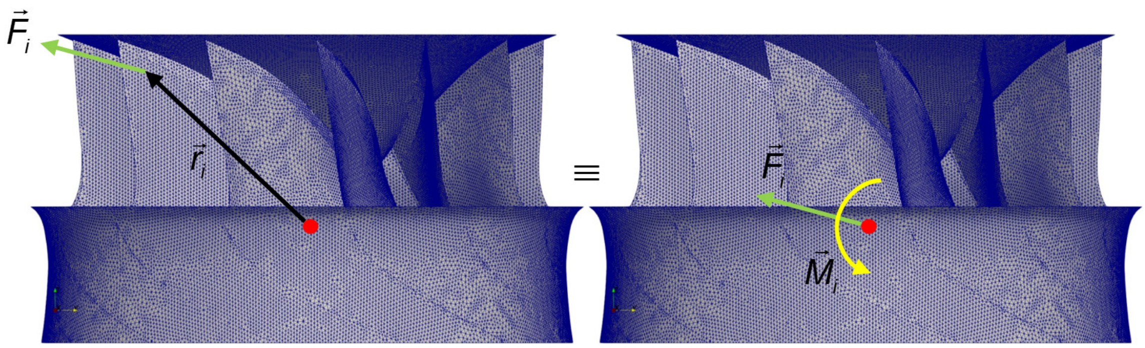


Figure 9: Torque calculation in the simulation

ACKNOWLEDGMENTS

The authors gratefully acknowledge technical and financial support from Foz do Chapecó, Baesa, Enercan e Ceran pelo apoio técnico e financeiro, através do projeto de Pesquisa e Desenvolvimento PD-02949-3007/2021– "Solução integrada para o diagnóstico de defeitos, análise dinâmica e monitoramento contínuo de unidades gerado-ras francis" com recursos do programa de PD da ANEEL.

REFERENCES

Barbi, F., Pivello, M.R., Villar, M.M., Serfaty, R., Roma, A.M. and Silveira Neto, A.d., 2018. "Numerical experiments of ascending bubbles for fluid dynamic force calculations". *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, Vol. 40, No. 11, p. 519. ISSN 1806-3691. doi:10.1007/s40430-018-1435-7. URL https://doi.org/10.1007/s40430-018-1435-7.

Castro, L.P.d., Pinheiro, A.P., Vilela, V., Magalhães, G.M., Serfaty, R. and Vedovotto, J.M., 2021. "Implementation of a hybrid lagrangian filtered density function–large eddy simulation methodology in a dynamic adaptive mesh refinement environment". *Physics of Fluids*, Vol. 33, No. 4, p. 045126. doi:10.1063/5.0045873.

Chen, G., Liang, X.F. and Li, X.B., 2022. "Modelling of wake dynamics and instabilities of a floating horizontal-axis wind turbine under surge motion". *Energy*, Vol. 239, p. 122110. ISSN 0360-5442. doi:https://doi.org/10.1016/j.energy.2021.122110. URL https://www.sciencedirect.com/science/article/pii/S0360544221023586.

Damasceno, M., Vedovoto, J. and Silveira-Neto, A., 2015. "Turbulent inlet conditions modeling using large-eddy simulations". *CMES - Computer Modeling in Engineering and Sciences*, Vol. 104, pp. 105–132.

Damasceno, M.M.R., de Freitas Santos, J.G. and Vedovoto, J.M., 2018. "Simulation of turbulent reactive flows using a fdf methodology – advances in particle density control for normalized variables". *Computers Fluids*, Vol. 170, pp. 128 – 140. ISSN 0045-7930. doi:https://doi.org/10.1016/j.compfluid.2018.05.004. URL http://www.sciencedirect.com/science/article/pii/S0045793018302494.

Germano, M., Piomelli, U., Moin, P. and Cabot, W.H., 1991. "A dynamic subgrid-scale eddy viscosity model". *Physics of Fluids A: Fluid Dynamics*, Vol. 3, No. 7, pp. 1760–1765.

Iaccarino, G. and Verzicco, R., 2003. "Immersed boundary technique for turbulent flow simulations". *Applied Mechanics Reviews*, Vol. 56, No. 3, p. 331. doi:10.1115/1.1563627.

Lilly, D.K., 1992. "A proposed modification of the germano subgrid-scale closure method". *Physics of Fluids A: Fluid Dynamics*, Vol. 4, No. 3, pp. 633–635.

Mittal, R. and Iaccarino, G., 2005a. "Immersed boundary methods". *Annual Review of Fluid Mechanics*, Vol. 37, No. 1, pp. 239–261.

Mittal, R. and Iaccarino, G., 2005b. "Immersed Boundary Methods". *Annual Review of Fluid Mechanics*, Vol. 37, No. 1, pp. 239–261.

Mohammadi, M., Sotiropoulos, F. and Brinkerhoff, J., 2018. "Moving least squares reconstruction for sharp interface immersed boundary methods: Mls reconstruction in the immersed boundary method". *International Journal for Numerical Methods in Fluids*, Vol. 90. doi:10.1002/fld.4711.

Neto, H.R., Cavalini, A., Vedovoto, J., Neto, A.S. and Rade, D., 2019. "Influence of seabed proximity on the vibration responses of a pipeline accounting for fluid-structure interaction". *Mechanical Systems and Signal Processing*, Vol. 114, pp. 224–238.

Peskin, C.S., 2002. "The immersed boundary method". *Acta Numerica*, Vol. 11, pp. 479–517. doi: 10.1017/S0962492902000077.

Pinheiro, A.P., Rybdylova, O., Zubrilin, I.A., Sazhin, S.S., Sacomano Filho, F.L. and Vedovotto, J.M., 2021. "Modelling of aviation kerosene droplet heating and evaporation using complete fuel composition and surrogates". *Fuel*, Vol. 305, p. 121564. ISSN 0016-2361. doi:https://doi.org/10.1016/j.fuel.2021.121564. URL https://www.sciencedirect.com/science/article/pii/S0016236121014459.

Pinheiro, A.P., Vedovoto, J.M., da Silveira Neto, A. and van Wachem, B.G., 2019. "Ethanol droplet evaporation: Effects of ambient temperature, pressure and fuel vapor concentration". *International Journal of Heat and Mass Transfer*, Vol. 143, p. 118472. ISSN 0017-9310. doi:https://doi.org/10.1016/j.ijheatmasstransfer.2019.118472. URL https://www.sciencedirect.com/science/article/pii/S0017931019309214.

Pope, S.B., 1999. "A perspective on turbulence modeling". pp. 53–67. doi:10.1007/978-94-011-4724-8$_5$.*URL*

RESPONSIBILITY NOTICE

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