COBEM 2023
27TH INTERNATIONAL CONGRESS OF MECHANICAL ENGENEERING

ABCM
Brazilian Society of Mechanical Sciences and Engineering
27th ABCM International Congress of Mechanical Engineering
December 04-08, 2023, Florianópolis-SC, Brazil

# COB-2023-0081
# Francis Generating Unit Modelling Through High-fidelity CFD Simulations

**Leandro José Lemes Stival**
**Leonardo Sicchieri**
**João Marcelo Vedovotto**
**Aldemir Ap. Cavalini Jr.**
Federal University of Uberlandia, School of Mechanical Engineering, Av. João Naves de Ávila, 2121, Uberlândia, MG, 38408-196, Brazil.
stival@ufu.br, leosicchieri@ufu.br, vedovotto@ufu.br, aacjunior@ufu.br
**Ricardo Lourencini Assumpcão**
CPFL Energia, Rodovia Engenheiro Miguel Noel Nascentes Burnier, Km 2,5 - Parque São Quirino - Campinas - SP, 13088-900, Brazil
rlassumpcao@cpfl.com.br
**Odair Morona**
Foz do Chapecó Energia S.A., Estrada Alto Alegre, S/N - Zona Rural Alpestre - RS - 98480-000, Brazil
odair.morona@fozdochapeco.com.br

***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 22 million control volumes. Preliminary analyses demonstrate that the LES-IB methodology is capable of simulating the interaction within the Francis turbine and flow structures. 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:*** *Francis turbine, computational fluid dynamics, large eddy simulation, immersed boundary method.*

## 1. INTRODUCTION

The use of Computational Fluid Dynamics (CFD) has expanded to complex fluid-structure interaction problems. Energy projects have utilized numerical modeling to solve turbulent flow properties. Large Eddy Simulation (LES) is increasingly used for rotor simulations due to improved computational power. The application of LES by renewable energy research groups is growing and encouraging methodology evolution. LES involves a spatial filtering process of governing equations, explicitly solving the largest turbulent structures on computational meshes and modeling only the smallest scales. The Immersed Boundary (IB) method, in conjunction with LES, is the state-of-the-art approach for simulating flows with high Reynolds numbers and fluid-structure interactions involving complex and moving geometries (Vedovoto *et al.*, 2015). The IB method is a strong and effective simulation tool, popular for dealing with complex structures without expensive and complicated dynamic meshing techniques. Due to its meshing flexibility, the IB approach has gained popularity for solving problems like heavy movements in solids or large deformation in fluid (Sotiropoulos and Yang, 2014; Wang *et al.*, 2008).

This article discusses the findings of simulations that include the flow through the spiral case and the motion of the Francis turbine rotor in three dimensions. It explores the mathematical concepts involved in generating a virtual machine for a generating unit (UG). Additionally, it delves into the mechanical, electrical, and hydraulic forces that affect the system and presents a sensitivity analysis as evidence of a groundbreaking project on the dynamic responses of a UG. The simulations use virtual pressure and velocity probes to measure turbulence intensity and provide information about emission frequencies and vortex passage in specific regions of interest.

### 1.1 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 (**?**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, 2005a; Iaccarino and Verzicco, 2003; Sotiropoulos and Yang, 2014; Mohammadi *et al.*, 2018). In the literature, various IB approaches have been proposed, with the main differences being in how the force is calculated and the interpolation stage. By combining an IB technique with a block-structured Cartesian mesh and AMR, it's possible to use efficient solvers within Cartesian blocks and refine the mesh only in areas where flow characteristics require it. This allows for more efficient use of resources.

To model the flow of a turbine, we use the Navier-Stokes equations, a set of four partial differential equations that describe velocity and pressure. To solve these equations, we use the LES methodology, which allows us to explicitly calculate the large eddies in the flow. However, we also use sub-grid scale (SGS) models to calculate the energy exchange between modeled and resolved structures, which are smaller than the grid. This approach is ideal for turbine simulation.

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 $\nu$ $(m^2/s)$ is the kinematic viscosity, $u_i$ $(m/s)$ is the $i$ component of velocity vector, $p$ $(N/m^2)$ is the pressure, $\rho$ $(kg/m^3)$ is the fluid specific mass,$\mu$ $(kg/ms)$ 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 $\mu_{SGS}$ 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 $\Delta$, 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):

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

The Lagrangian formulation of work is used in connection with the Eulerian formulation, which is represented by the LES method. The immersed boundary method framework is used in the Lagrangian formulation. This method uses an independent grid to define the body inside the fluid flow. An example of an Eulerian domain $\Omega$, which models the fluid flow, and a Lagrangian domain $\Gamma$, which models a solid sphere immersed in the fluid domain, is shown in Fig. 1. The volume of the Eulerian element is $h^3$, where $h$ is the volume's length in the three directions. The Lagrangian mesh is constructed so that the Lagrangian volume $\Delta\Gamma$ is equal to the Eulerian volume (Uhlmann (2005)). This means that $\Delta\Gamma = \Delta A \cdot h = h^3$, where $\Delta A$ is the area of the Lagrangian element, and $\Delta A = h^2$.

One major advantage of this methodology is its ability to simulate flows over complex geometries using a cartesian grid to solve balance and transport equations in an Eulerian reference frame. The two reference frames are coupled through source terms, as shown in Equation (5). The current study uses a modified version of the multi-direct forcing method from Wang *et al.* (2008). This methodology involves an iterative process of the direct forcing method and is further explained in works such 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:

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

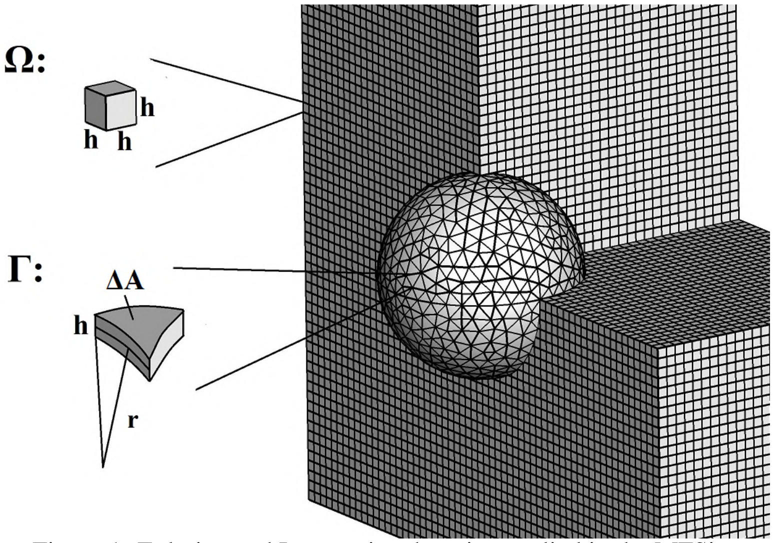

Figure 1: Eulerian and Lagrangian domains applied in the MFSim

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

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

This function is important in situations where the Eulerian volumes are distant from the Lagrangian points and the force is distributed at those points. A significant feature of this function is its conservative distribution process, which results in a unitary value when integrated over r (Mittal and Iaccarino, 2005b). The force at a Lagrangian point, represented by $\vec{F}(\vec{x}_K)$, is distributed over the Eulerian field to define the boundary. Furthermore, the momentum equation remains valid at each Lagrangian point (**?**).

The paragraphs above explain how the mathematical formulation involving LES and IB methods is based on two different reference points. The first approach uses the Eulerian coordinate system, while the second involves the Lagrangian perspective of the immersed body distributed over the flow. The momentum and mass balance equations provide filtered velocity and pressure fields for the Eulerian reference point. However, the velocity field and position provided by the Eulerian reference point are transferred to Lagrangian points to calculate the velocity of the body's movement at those points. After that, the source terms $F_i$ are evaluated at those points. This source term is then 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, using an iterative process improves the calculation's precision, which is the main advantage.

### 1.2 Computational domain and boundary conditions

The process of CFD simulation involves applying the principles of fluid mechanics to a physical problem and creating a mathematical model. This model is then solved using numerical methods through MFSim, resulting in numerical data for the physical properties being studied. The accuracy of the simulation depends on factors such as the quality of the model, the approximations made, and the resources available for computation. Proper planning is key to utilizing MFSim effectively for design and analysis.

The work involves using cubes with 75 mm edges as the smallest element, and the computational domain is divided into control volumes measuring 90 x 45 x 60 m. To solve the transport equations, around 22 million equations must be solved simultaneously at each iteration (advance of time step) if a uniform mesh is used. However, using a block-structured mesh with local refinement reduces the number of volumes to be simulated. The spiral case and rotor's geometry is shown in Fig. 2, while Figs. 3 and 4 demonstrate the correlation between the Eulerian and superficial Lagrangian meshes. Additionally, Fig. 5 shows an example of a turbine rotating inside the domain using the AMR technique.

To ensure accurate simulations, we utilize an adaptive mesh consisting of 5 levels. At the base level, we configure the initial mesh arrangement with 24x12x16 volumes in the x, y, and z directions using a hexahedral uniform grid. This results in an initial calculation with approximately 22 million control volumes. To refine the mesh around the blades, we implement mesh refinement. The inlet flow condition is set at a nominal value of 483 m³/s with an inlet velocity of u=9.42, v=0, and w=0 m/s using the MFSim code. This is classified as a Dirichlet boundary condition. For the side planes (xz-planes), upper plane (xy-plane), and lower plane (xy-plane), we establish a Neumann boundary condition as

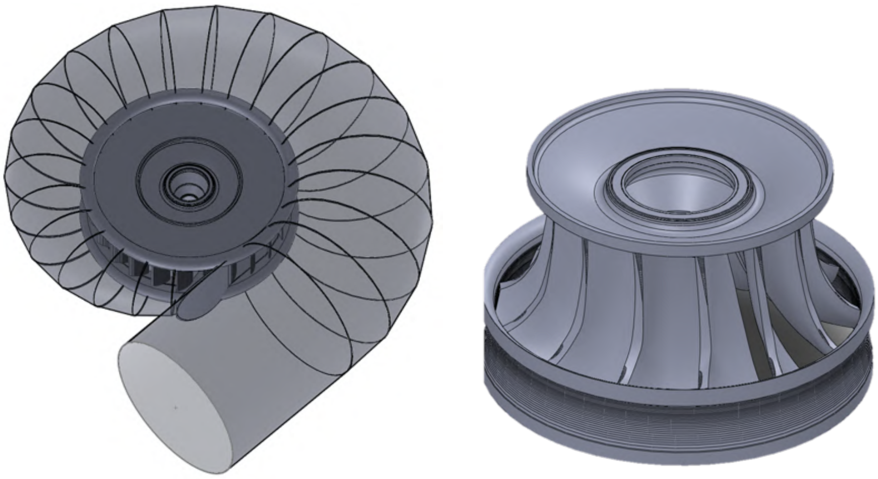

Figure 2: (a) Spiral case and (b) Francis turbine geometry

the boundary conditions.

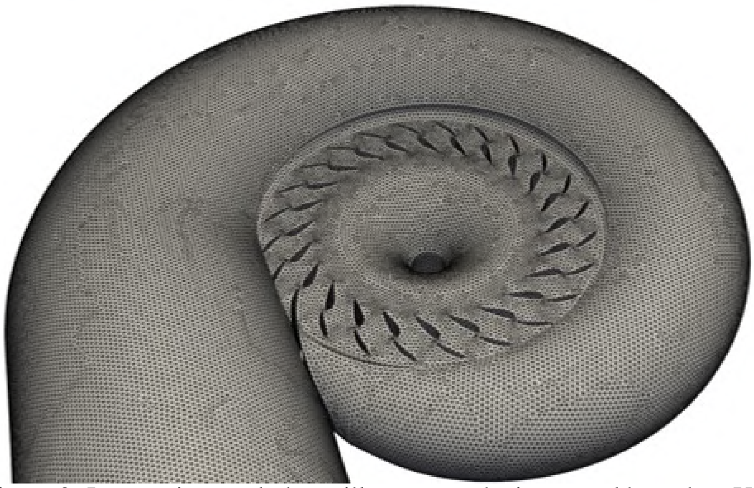

Figure 3: Lagrangian mesh that will represent the immersed boundary UG

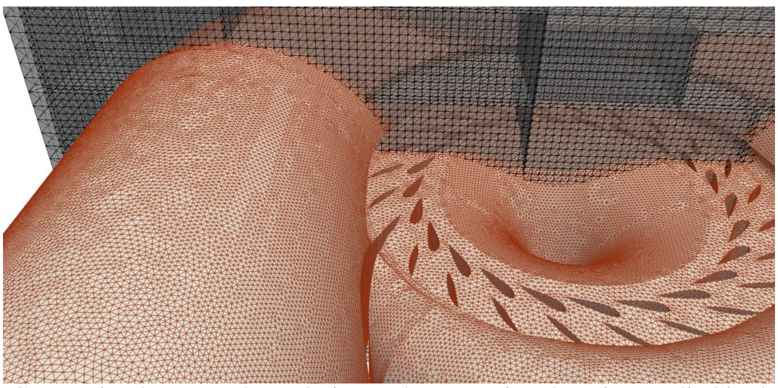

Figure 4: Connection between the Eulerian and Lagrangian domains applied in the MFSim

## 2. RESULTS AND DISCUSSION

### 2.1 Analysis of the Flow Turbulent Structures

This section presents analyses demonstrating that the LES-IB methodology can simulate the interaction between the Francis turbine and flow structures. Fig. 6 shows a dynamic representation of the iso-surfaces colored by streamwise velocity to provide important visualization of the turbulent flow. Essential streamlines occur in the region close to the

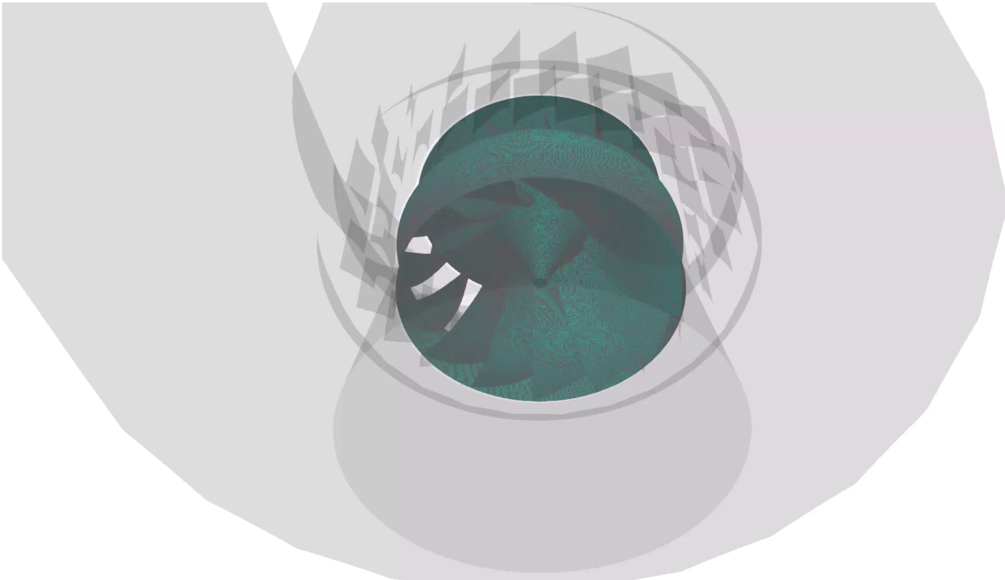

Figure 5: Francis turbine rotating due the AMR in the MFSim

blade and being transported over the flow showing some structures occurring right after the turbine, and those vortices structure patterns are straight connected to the turbine's operational parameters, such as flow rate, turbine speed and pressures.

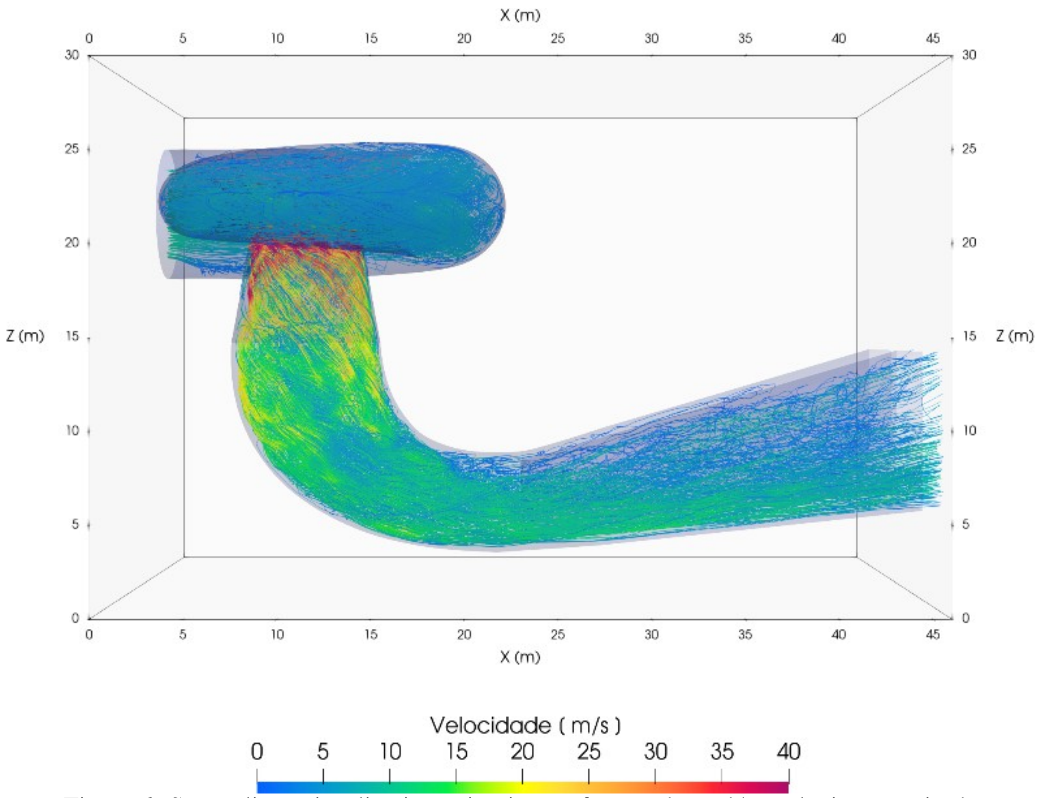


Figure 6: Streamlines visualization using iso-surfaces coloured by velocity magnitude

From 6 it is possible to identify that the highest speeds occur around the turbine due to the turbine rotation movement around 90 RPM in nominal operating conditions. Another important point is how the flow loses speed after the turbine and entry into the cone and subsequent development to the suction tube, where the speeds go from 40 m/s to 15 m/s and later to less than 10 m/s.

To summarize the qualitative analysis, Figure 7 presents a 3D visualization of instantaneous iso-surfaces color-coded by vorticity magnitude. The figure indicates that vorticity values exceeding 60 (1/s) are concentrated near the blades and downstream of the flow. On the other hand, lower vorticity values (around 40 (1/s)) are observed over the spiral case due

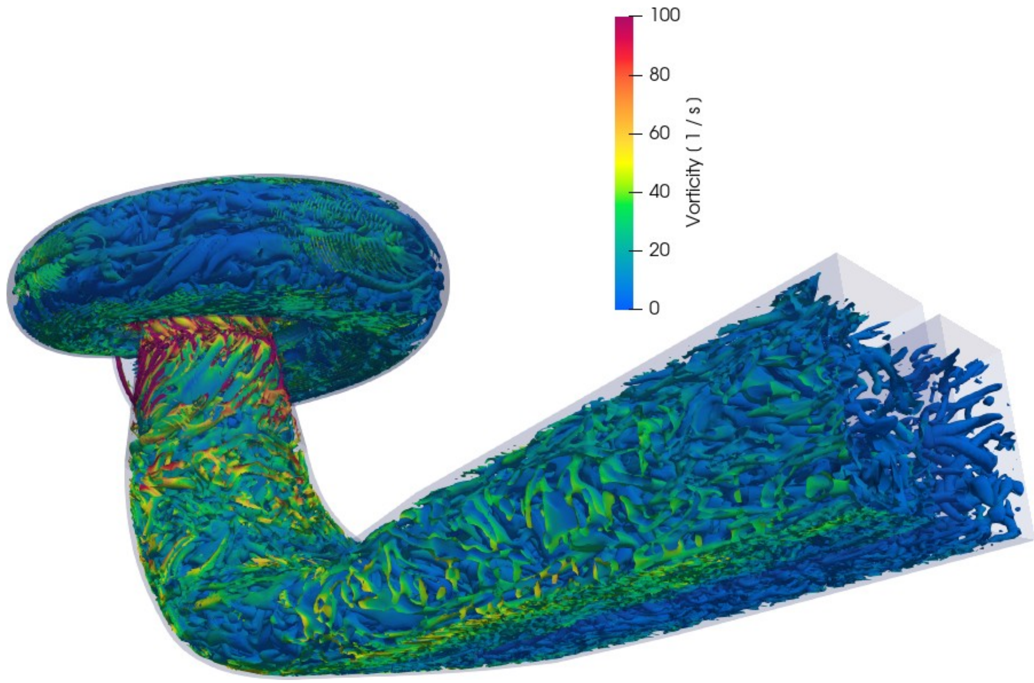


Figure 7: Vortex visualization coloured by vorticity magnitude over the domain

to the flow pattern. Furthermore, the figure highlights the highest vorticity values (around 160 (1/s)) near the blade top.

## 2.2 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, $E(f)$, 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 passing 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.

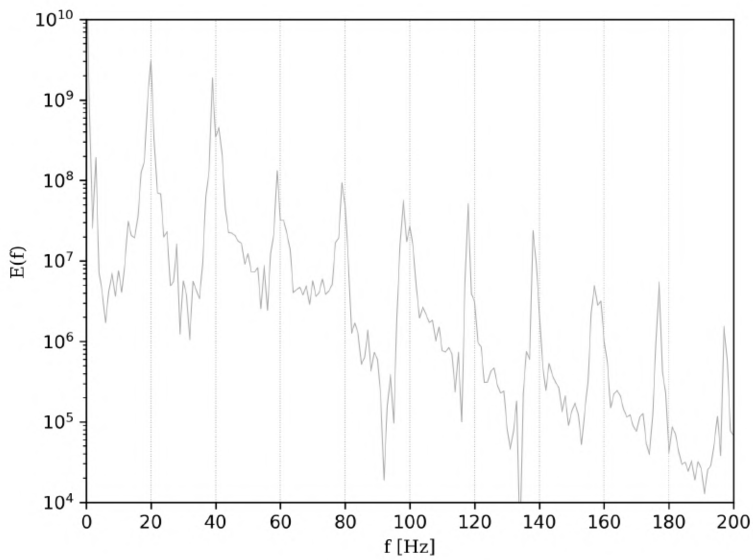


Figure 8: Blade's passage frequency

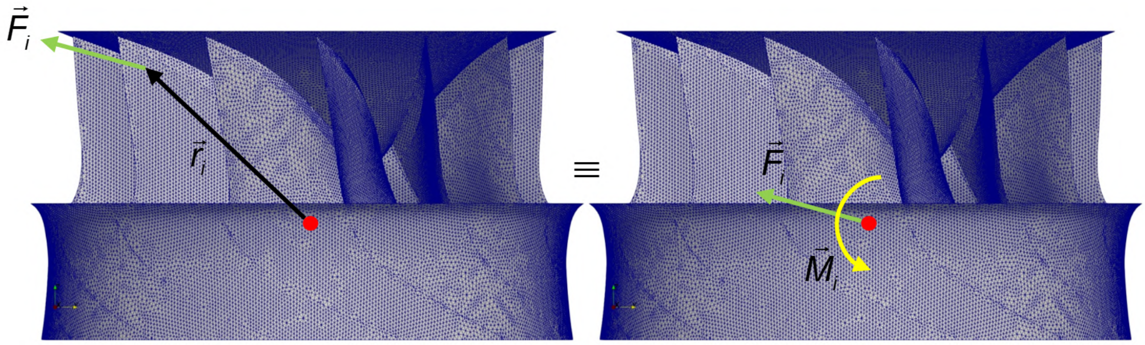


Figure 9: Torque calculation in the simulation

### 2.3 Evaluation of Power Generation

In this section, we will discuss the findings regarding the UG's torque and power over time. By utilizing the immersed boundary method, the geometry is introduced into the fluid domain with a force term in the Navier-Stokes equations. This allows us to calculate the torque and power generated by the turbine over a structure within the flow through post-processing. Below are the steps involved in this implementation:

1. To determine the distance between the turbine rotation reference position and 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 $z$.

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

   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. When you calculate mechanical torque, you can get the mechanical power.

## 3. CONCLUSION

The MFSim results showed that the torque generated had a mean of 24.3 $\pm$ 10.2 $(MNm)$ and the power generated had a mean value of 229.5 $\pm$ 11.3 $(MW)$. It is important to highlight that the outcome shown here is groundbreaking as it is among the earliest complete, time-bound simulations that employ dynamic modeling of the turbulence in a Francis turbine. Based on the current initial findings, the evaluations show potential in the developmental context.

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. The obtained results show the ability of the implemented model to represent the behavior of the components under analysis. The sequence of this work will be with grouping different models and their application for diagnosing defects in the considered GU.

## 4. ACKNOWLEDGEMENTS

The authors gratefully acknowledge technical and financial support from Foz do Chapecó, Baesa, Enercan and Ceran by the financial and technical support from RD PD-02949-3007/2021– "Solução integrada para o diagnóstico de defeitos, análise dinâmica e monitoramento contínuo de unidades geradoras francis" with program resources from ANEEL RD, and also to FAPEMIG and Petróleo Brasileiro S.A. (Petrobras).

## 5. 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$_5.URL$

## 6. RESPONSIBILITY NOTICE

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