Large-eddy simulations of a swirling flow in a model Francis turbine

E V Palkin¹, M Yu Hrebtov¹,², and R I Mullyadzhanov¹,²

¹ Institute of Thermophysics SB RAS, 630090 Novosibirsk, Russia
² Novosibirsk State University, 630090 Novosibirsk, Russia

rustammul@gmail.com

Abstract. We performed Large-eddy simulations of the flow in a model air Francis turbine in a range of low-load regimes with a swirler rotating at fixed frequency. All investigated regimes revealed the presence of coherent helical vortex structure in the draft tube: the precessing vortex core. We identified the frequency of this instability and obtained mean flow velocity fields to be utilized in further works.

1. Introduction
In global energy consumption a steady tendency appears towards increasing the share of renewable natural resources such as sun, wind and water. Hydroelectric power plants are the most stable renewable energy source that allows flexible control of electricity generation. Radial-axial turbines are among the world most common types of turbines, which can operate at partial or forced generator loads. However, in such non-optimal low-load operating modes, hydrodynamic instabilities can arise in the flow path of a hydraulic turbine, the most dangerous of which is a precessing vortex core (PVC) [1]. It can be represented as a coherent helical vortex structure that arises behind the turbine flow body in the form of a spiral oriented along the axial direction of the expanding conical flow path of the hydraulic turbine. Its rotational frequency may match the natural frequency of hydroelectric station parts causing resonance [2]. Moving away from the optimal efficiency regime by decreasing the volumetric flow rate the size of this stagnant region increases [3] and may get to the runner area. Recent investigations are directed towards PVC suppressing in stationary [4] and transient flow regimes [5]. It is required to thoroughly investigate low-load flow regimes in the Francis turbine before expanding the range of permissible operating modes of hydraulic turbines by applying control techniques to suppress the PVC in the future.

2. Geometry and computational details
The Francis-99 model air turbine model geometry is used for numerical simulations and corresponds to that investigated experimentally in IT SB RAS [2] and is shown in figure 1. This model turbine uses two vane swirlers to replicate a velocity field behind the runner of a real turbine. This approach greatly reduces the costs to investigate the flow in Francis turbine both numerically and experimentally. The swirler blade configuration in the model geometry is composed to output optimum performance at volumetric flow rate \( Q_c = 0.049 \) m³/s and angular frequency of the runner of 40.53 Hz. The draft tube design of Francis turbines remains unmodified. The investigated operating parameters are ranged for the flow rate from 0.30 \( Q_c \) to 0.65 \( Q_c \), which corresponds to under-load regimes with a strong PVC present with maximum pressure fluctuations on the walls of the suction pipe among those considered...
exponentially. The inlet diameter $D = 0.1$ m is used for Reynolds number definition. Main flow parameters are displayed in Table 1. The coordinate system is located at the tip if the runner axis and $x$ corresponds to the streamwise direction.

| Table 1. Parameters of three operating flow regimes experimentally investigated in IT SB RAS on a model air Francis turbine. |
|---------------------------------------------------------------|
| $Q/Q_c$ | $U_b$ m/s | $Re$ |
|---|---|---|
| regime 1 | 0.30 | 1.85 | 11 600 |
| regime 2 | 0.50 | 3.08 | 19 300 |
| regime 3 | 0.65 | 4.01 | 25 100 |

Figure 1. Francis turbine model geometry (top row) and computational domain with the mesh used for LES simulations (bottom row). All pink surfaces correspond to impermeable walls, the guide swirler and the runner rotating around its axis are highlighted in green and blue, accordingly. Different mesh blocks are marked with colors: green is the inlet pipe, orange is a block rotating at constant frequency and purple is the draft tube with the outlet meshed with multi-block OH-topology.

We use the unstructured finite-volume open-source computational code OpenFOAM [6] of the second-order accuracy both in time and space on a cell-centered collocated mesh. The code solves the three-dimensional spatially filtered incompressible Navier–Stokes equations relying on a Large-eddy simulation approach with a dynamic Smagorinsky subgrid-scale model [7]. This model maintains unsteadiness and captures essential flow features unlike axisymmetric or Reynolds-averaged equations approaches with linear eddy-viscosity models [3, 8]. The MUSCL TVD scheme was used to approximate the convective terms, which provides the necessary compromise between numerical stability and approximation accuracy. Central differences were used for viscous terms. The time approximation was performed according to the second-order Crank-Nicholson scheme. The PISO method was used to correct the pressure-velocity using the predictor-corrector scheme [9].

To construct a finite-volume mesh, non-conformal blocks were used to provide a transition between the region of geometry with the runner blades and fixed parts of the geometry. Between the blocks, flat boundaries (arbitrary mesh interfaces) were used where the values of the calculated velocity and pressure fields were interpolated between the cells of the mesh blocks located on opposite sides of the boundary. The weights for this interpolation were calculated dynamically at each time step. Interpolation was performed with the second order of accuracy. A similar approach to multi-
block mesh with an arbitrary mesh interface was recently elaborated on another swirling flow and proved to be robust [10]. The entire computational domain consists of three non-conformal blocks (see figure 1). The stationary inlet block includes the inlet pipe and the first swirler. It has a common surface with the block rotating clockwise at specified angular velocity. The mesh in the third block has a multi-block OH-topology in cross-section and is uniform in the longitudinal direction. The first block contains 1.5 mln cells, the second (rotating) block contains 0.45 mln cells, and the third block (draft tube) contains 5.6 mln cells. Directly behind the runner the characteristic longitudinal cell size increases exponentially with the increment equal to 1.05. In the longitudinal direction the mesh has 79, 48 and 187 elements in the 1st, 2nd and 3rd blocks, respectively. In total the mesh consists of 7.6 million hexagonal cells. At the inlet a homogeneous velocity profile is set. A recent computational study on a Kaplan hydroturbine [8] suggests that the resolution is sufficient for the LES approach.

3. Results

Typical PVC structures formed after the runner in the draft tube obtained by the LES method represented as the pressure isosurface for \( Q/Q_c = 0.50 \) flow regime can be seen in figure 2. It can be observed that large-scale vortical structure is a main contributor to turbulent pulsation.

![Figure 2. Pressure isosurface colored with streamwise velocity. Full PVC visualization is available at https://youtu.be/VosegjxeqTU](https://youtu.be/VosegjxeqTU)

To find the precession frequency we investigated the pressure difference signal for all the computational regimes shown in figure 3. A large-scale pressure fluctuation corresponds to PVC regime. A Fourier analysis of this signal is also performed and the characteristic frequency corresponding to the rotation of the vortex core is \( f_D/U_b = 1.14, 0.545 \) and \( 0.518 \) for \( Q/Q_c = 0.35, 0.50 \) and 0.65, respectively. The frequency obtained experimentally by PIV method for \( Q/Q_c = 0.50 \) is \( f_D/U_b = 0.55 \), which indicates a good agreement between the data and experiment and the adequacy of the chosen calculation methods. It can be noticed that for \( Q/Q_c = 0.50 \) the PVC harmonic is dominant and signal is nearly periodic, while for other flow regimes it is more stochastic. All spectra also indicate the presence of the runner blades rotational frequency \( f_D/U_b = 2.19, 1.31 \) and 1.01 for \( Q/Q_c = 0.35, 0.50 \) and 0.65, correspondingly, and their multiple of five frequencies: 10.9, 6.56, and 5.05.
Figure 3. Time-history of the pressure difference between points 0: (-0.1, -0.1, 0.5) D and 1: (-0.1, 0.1, -0.5) D which are located at opposite sides of a cross-section x = -0.1 D in the draft pipe just behind the runner.

Time-averaged streamwise and radial velocity fields for different flow regimes are presented in figure 4. A vast recirculation zone is present in all the cases although for the case with the lowest volumetric flow rate the recirculation zone propagates to the internal area of the runner which suggests that it is the most undesirable flow regime.
4. Conclusion
We performed a number of LES simulations of the flow in the model Francis turbine with the swirler and the rotating runner in three different flow regimes with a partial load. Simulations required arbitrary mesh interfaces to account for the rotation showed good agreement with experimentally obtained PVC frequency. As turbine efficiency decreases (lower volumetric flow rate) the size of this recirculation region increases and eventually propagates into the runner. The flow velocities and pulsations obtained in this work will be used to analyze the coherent structures behind the runner by Proper Orthogonal Decomposition method or Linear Stability Analysis to develop an affordable control strategy which attempts to suppress PVC mode in the model Francis turbine geometry.

Acknowledgments
This work is partially funded by the Russian President Scholarship for young scientists, grant No. 29.2021.1, and the Russian Foundation for Basic Research, grant No. 20-58-12012, while the development of the computational code is performed under the state contract with IT SB RAS. The computational resources are provided by Siberian Supercomputer Center SB RAS, Joint Supercomputer Center of the Russian Academy of Sciences and Supercomputing Center of the Novosibirsk State University.

References
[1] Gallaire F, Ruith M, Meiburg E, Chomaz JM, Huerre P 2006. J. Fluid. Mech. 549 71–80
[2] Litvinov I, Shtork S, Gorelikov E, Mitryakov A and Hanjalic K. 2018. Exp. Therm. Fluid Sci. 91 410–422
[3] Foroutan H and Yavuzkurt S 2014. J. Appl. Mech. 81(6)
[4] Pasche S., Avellan F and Gallaire F. 2019. J. Fluids Eng. 141 081203
[5] Litvinov I, Suslov D, Gorelikov E and Shtork S 2021. Energies 14 1240
[6] OpenFOAM website. 2004. Available at http://www.openfoam.com.
[7] Lilly DK 1992. Phys. Fluids 4(3) 633–635
[8] Minakov A, Platonov D, Litvinov I, Shtork S and Hanjalic K 2017. *J. Hydraul. Res.* **55**(5) 668–685

[9] Moukalled F, Mangani L and Darwish M 2016. *Springer: New York*

[10] Hrebtov M, Palkin E and Mullyadzhanov R 2020. *J. Phys. Conf. Ser.* **1677**(1) 012012