IOP Conference Series: Earth and Environmental Science

Reducing computational effort of high head Francis turbines

To cite this article: Ken-Robert G. Jakobsen *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **240** 072001

IOP Conf. Series: Earth and Environmental Science **240** (2019) 072001 doi:10.1088/1755-1315/240/7/072001

Reducing computational effort of high head Francis turbines

Ken-Robert G. Jakobsen, Erik Tengs, Martin Aa. Holst

EDRMedeso AS, Leif Tronstads plass 4, 1337 Sandvika, Norway

ken-robert.g.jakobsen@edrmedeso.com

Abstract. The present work shows how a complete full-wheel model of a high head Francis turbine can be reduced directly by leaving out parts of the computational domain, but without losing accuracy in the RSI pressure that is of primary interest. Instead great savings in mesh size and consequently computational time is gained. Two main modifications have been performed: 1) Reducing the draft tube and 2) Excluding the distributor except for the guide vanes. This is performed both on separate models as well as combining them obtaining an ultimate mesh reduction of about 60%. From this process it is shown that the volute could be replaced by a very simple set of inlet boundary conditions without affecting the results as well as simplifying the meshing procedure. A close to linear speed-up with mesh size is obtained. For comparison, simulations performed on passage models have also been included and all results are validated against full-wheel transient simulations as well as experimental data. The work has been performed as part of the HiFrancis project at the Norwegian University of Science and Technology and the Norwegian Hydropower Centre.

1. Introduction

Computational Fluid Dynamics (CFD) is a powerful tool when designing and investigating hydraulic turbines. However, using CFD puts a high demand on computational resources. Acknowledging the fact that an industrial design loop must be completed within a practical time frame, typically 6-12 hours, it is crucial to reduce the computational effort. Today this is usually achieved by utilizing very simplified numerical models of the turbines, compromising on accuracy and the available information that can be retrieved from the simulation results. For high head Francis turbines, the latter is commonly justified by the need to only focus on Rotor-Stator-Interaction (RSI) to avoid resonance effects between the pressure pulsations and natural frequencies of the turbine as these are main sources for blade cracking [1]. A thorough overview of existing numerical techniques applied to high head Francis turbines can be found in [2].

Aiming to achieve more time efficient simulation procedures that are of industry relevance, the present work shows how a complete 360º model (also referred to as full-wheel model) of a high head Francis turbine can be reduced directly by leaving out parts of the computational domain, but without losing accuracy in the RSI pressure that is of primary interest. Instead great savings in mesh size and consequently computational time is gained. Two main modifications have been performed: 1) Reducing the draft tube 2) Excluding the distributor except for the guide vanes. This is performed both on separate models as well as combining them. In general, reducing the draft tube will lead to the greatest reduction in computational mesh due to the vast number of elements being contained there, but modifying the distributor will also simplify the meshing procedure and consequently reduce the time needed for mesh generation. The present investigation also includes simulations performed on some of the much faster passage models available in ANSYS CFX, utilizing the periodic symmetry of the turbine.

Existing experimental data together with CFD simulations on the same geometry and operating points are used for data validation. The latter include three-dimensional transient simulation of a full-wheel turbine model with spiral casing (also referenced as volute or distributor), guide vanes, runner, and draft tube. This mesh consists of about 42 million nodes leading to extensive simulation times before reaching a periodic state.

The work has been performed as part of the HiFrancis project at the Norwegian University of Science and Technology and the Norwegian Hydropower Centre.

2. Reference data

In the following section, reference data used to validate the reduced models will be presented. This includes both experimental and numerical assessments.

2.1. Experimental validation data

As presented in [3], an overview of the experimental setup is shown in Figure 1. The runner is a modified model of the turbine previously installed at the Tokke power plant in Norway and is used at Norwegian University of Science and Technology (NTNU) for research on pressure pulsations inside high head Francis turbines. Moreover, the runner is a so-called splitter design and consists of 15+15 blades and the distribution unit of 28 guide vanes and 14 stay vanes. To perform the measurements and capture the propagation of the pressure pulses, five pressure sensors were positioned in the middle of two of the runner channels. This is shown in Figure 2.

With the present setup combined with a runner speed of 5.54Hz, the predominant frequencies seen in this test rig become the guide vane passing at 155Hz and the blade passing at 166Hz. In addition to the RSI frequencies the system will be subjected to additional phenomena such as draft tube pulsations (Rheingans frequency), vortex shedding and elastic fluctuations in the waterway when going outside of BEP. These are included in the experimental results but are not part of the scope.

The operating point of primary interest in the present study is BEP performed at a head of 11.94m and flow rate equal 0.2m^3/s. It is referred [3] for further details on this and other load conditions.

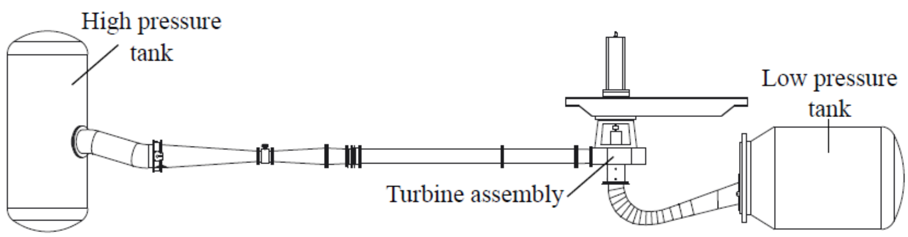

Figure 1. Overview of measurement setup.

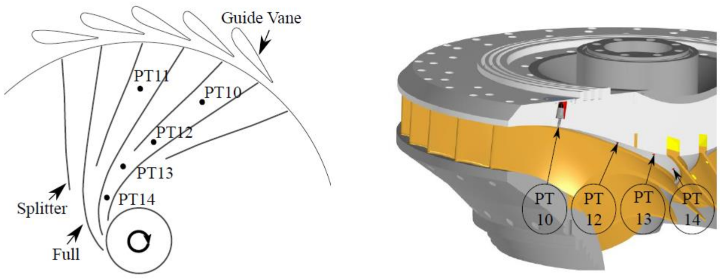

Figure 2. Position of hub mounted pressure sensors in runner channels.

2.2. Numerical validation data

Mimicking the setup of the model measurements, CFD simulations of the full-wheel model have been presented in [4] and [5]. Referred to as Base Case in the present paper, the results from these analyses are used as numerical validation data for the reduced models, both in terms of accuracy and computational speed. The simulations are transient utilizing a sliding mesh interface between the runner and volute (rotor/stator).

A mesh independent solution consisting of approx. 42 million nodes has been utilized, where the volute consists of mesh with tetrahedral elements (size), while the runner and draft tube are generated with hexahedral elements (size). The industry standard k-w SST turbulence model has been applied for all simulations with the boundary conditions as shown in Figure 3. Performed in ANSYS CFX, the simulations were run for three revolutions after steady periodic flow behaviour had been achieved. Three distinct operating points were considered, however, only BEP will be presented here.

In general, the full-wheel results agree very well with the experimental data discussed in the previous section. For BEP, this is exemplified through global parameters in Table 1 and the corresponding pressure signal in PT10 in Figure 4. The second harmonic is clearly underpredicted, which in recent studies has been shown to be due to resonance effects occurring at the double RSI frequency in the experiments.

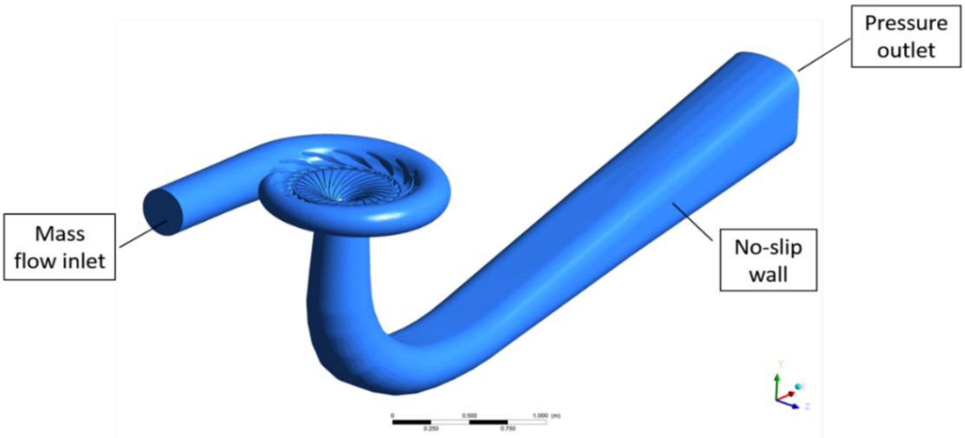

Figure 3. Computational domain for full-wheel simulations [5].

Table 1: Global parameters for full-wheel simulation and experiments at BEP [5].

Case	Net head [m]	Torque to generator [Nm]	Inlet pressure [kPa]	Outlet pressure [kPa]	Hydraulic efficiency [%]
Experiment	11.94	620.7	215.6	111.1	92.2
CFD transient	12.08	640.8	218.4	111.8	94.1
Deviation	1.17%	3.24%	1.30%	0.63%	2.06%

IOP Conf. Series: Earth and Environmental Science **240** (2019) 072001 doi:10.1088/1755-1315/240/7/072001

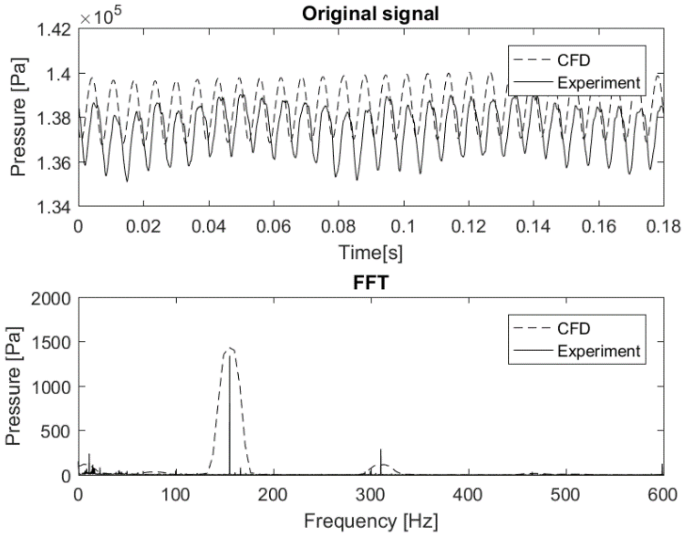

Figure 4. Pressure signal in PT10 at BEP for full-wheel simulation and experiments [5].

3. Reduced models

This section will compare different approaches to reduce the overall computational time on the high head Francis turbine. Two main methods will be discussed:

1) Reducing the total number of elements in the full-wheel model directly by leaving out parts of the volute or the draft tube domain.
2) Utilizing the rotational symmetry of the Francis turbine through different passage models available in ANSYS CFX.

The experimental and numerical investigations presented in the previous two subsections are used as validation data.

3.1. Reducing full-wheel model

The effects of removing the volute or draft tube have been investigated in the following, including one case combining both. Figure 5 shows the resulting computational domains. Same boundary conditions as for the Base Case have been used for all cases, except for the no volute/combined inlet condition for which velocity vectors have been specified at the slightly extended guide vane passage (see Figure 6). The velocity direction was determined from the stay vane angle, which is independent of the operating point.

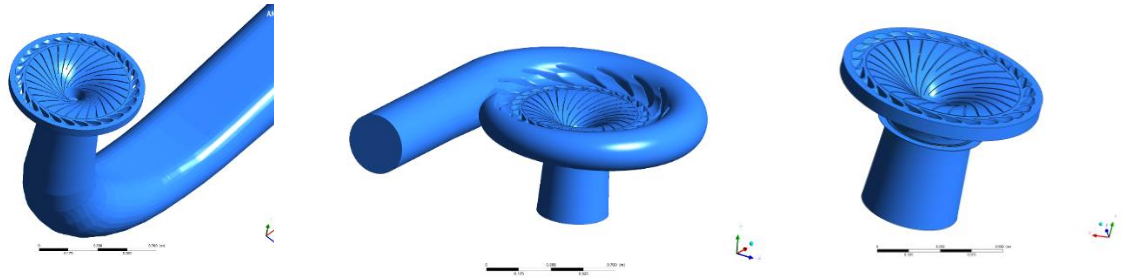

Figure 5. Computational domain of reduced full-wheel models. Left: No volute, middle: Short Draft Tube, right: Combined.

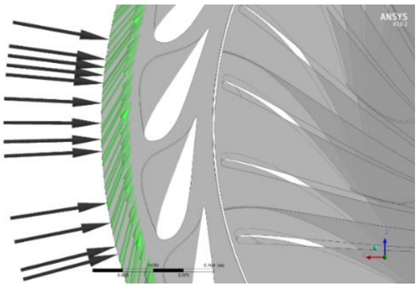

Figure 6. Constant velocity profile at inlet.

Figure 7 shows streamlines for all three cases and Figure 8 and Figure 9 give the pressure signals and corresponding FFT plots from the PT10 probe onboard the runner (see definition of sensor location in Figure 2). Clearly, none of the separate full-wheel model reductions affect the results to any significant degree relative to Base Case as both the pressure amplitudes and the pulsation frequencies match well. This is also true for the combined case. Although insignificant, it is interesting to note that the No Volute case produce a slightly higher pressure amplitude and the Short Draft Tube case sees a somewhat higher average pressure signal. These effects can be explained by the strictly uniform velocity profile substituting an otherwise periodic inlet condition and the outlet pressure condition being much closer to the measurement locations for the respective cases, respectively. The Combined case sees both effects accordingly. The small differences seen can easily be reduced further by adjusting the outlet condition at that location due to the shorter draft tube giving a lower pressure reduction from the runner to the outlet. Furthermore, a more accurate inlet condition can be obtained by for instance assuming a sinusoidal periodic velocity profile. This is expected to reduce the pressure amplitude at the measurement location. Noticeably, this is also information that can be gained prior to performing any simulations.

It is referred to section 2.2. and [5] for validation of the full-wheel simulations.

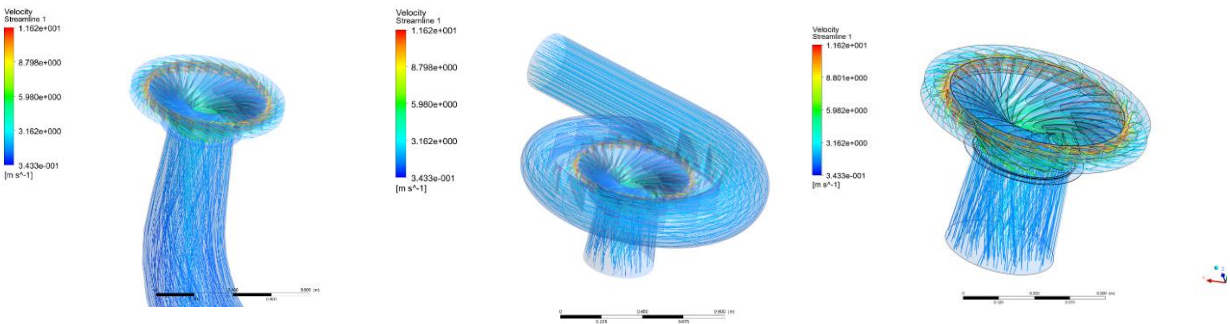

Figure 7. Streamlines on reduced model. Left: No volute, middle: Short Draft Tube, right: Combined.

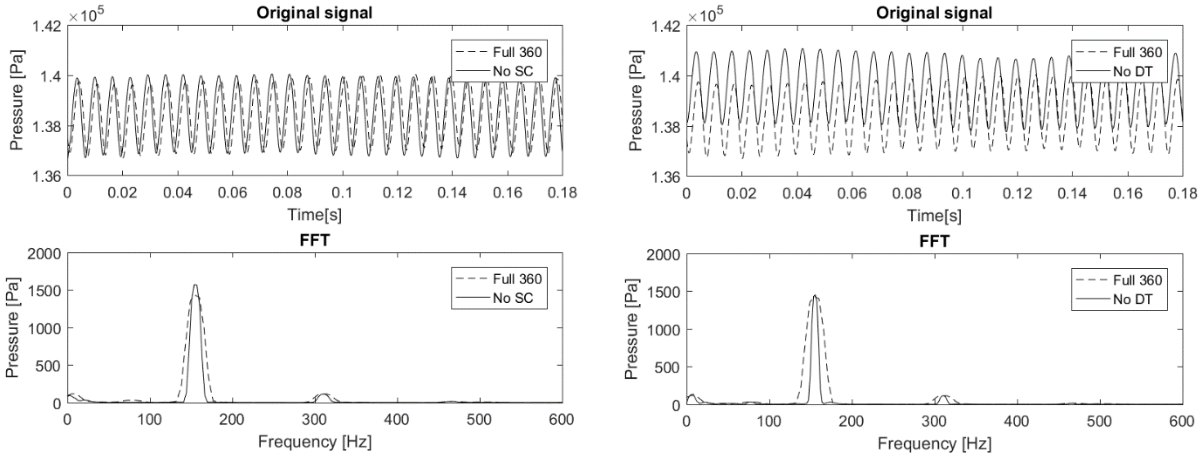

Figure 8. PT10 pressure signal. Left: No Volute, right: Short Draft Tube.

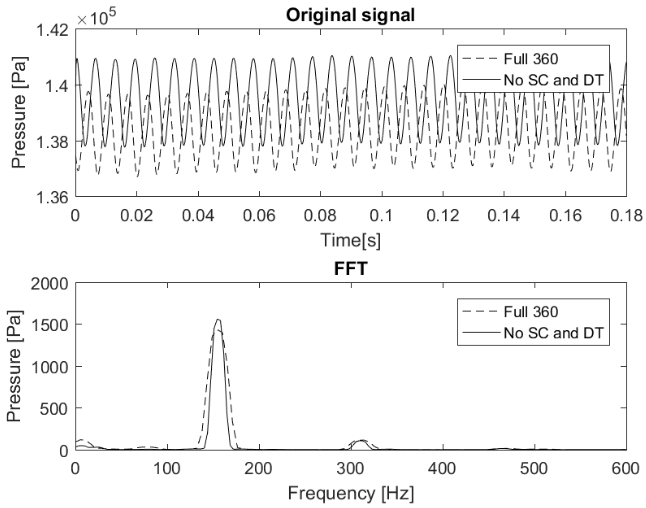

Figure 9. PT10 pressure signal for Combined at BEP.

3.2. Passage modelling
Parts of the following section is adapted from [6] with permission from author.

The turbine runner is rotationally symmetric and significant speed-up can be achieved by simulating only a section of the geometry. Challenges arise, however, when the number of stator (S) and rotor (R) components are not equal. First, periodic interfaces must be prescribed. Conventional periodic interfaces require the assumption that fluxes on one interface equals that of the other. This is not the case in a hydro turbine if S≠R. Instead, phase-shifted periodic boundaries will be present, meaning that one periodic interface will equal the other at an earlier or later instance in time. This will have to be addressed by the methods used. Furthermore, pitch change also occurs when S≠R as illustrated in the passage model in Figure 9 (S=19 and R=24). If the pitch is different from unity, some modification will be performed on the information crossing the frame change interface

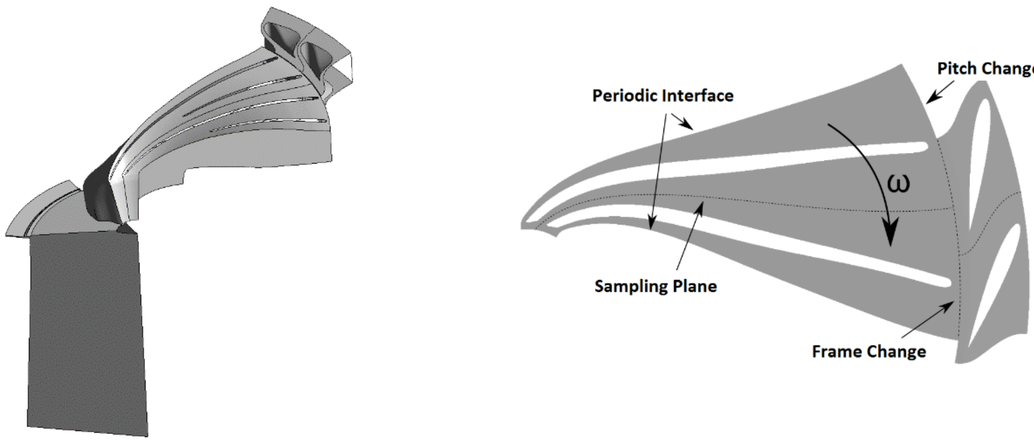

Figure 10. Computational domain of passage models. Right: definition of boundary conditions and pitch change [6].

The present investigation includes simulations on three of the passage methods available in ANSYS CFX:

1. Profile Transformation (PT)
2. Fourier Transformation (FT)
3. Frozen Gust (FG).

In all passage models, the inlet boundary conditions have been defined as total pressure and velocity directions taken from a steady simulation. The settings were otherwise as for the full-wheel simulation discussed in chapter 2.2. The direction of the velocity components imposed on the inlet to replicate the effect of the guide vane/stay vanes, was the same as the guide vane/stay vane outlet angle. This work was first published in [6]

Whereas the PT method simply scales the flow from the stator domain to match the size of the runner domain conserving mass and momentum, the FT utilizes the periodic nature of the flow through Fourier series decomposition. In the latter method, the trigonometric functions are applied directly at the rotor/stator interface resulting in a close to pitch independent methodology. Frozen Gust is an alternative to FT where only the runner passage is included in the simulation and the inlet conditions are applied directly upstream of the runner. Obviously, this results in great reduction in mesh size, but the results will depend significantly on the accuracy of the inlet conditions. Detailed descriptions of the respective methods can be found in [7].

Figure 11 shows the spectral content at pressure probe PT10 with the PT and FG methods together with the experimental measurements and full-wheel simulations presented previously. Manual Fourier coefficient sampling was implemented in the full-wheel simulation to extract the true periodic nature at the RSI interface. The profile was then reconstructed at the inlet of the Frozen Gust simulation as a periodic inlet boundary condition (FG Fourier Inlet). Clearly, the blade passing frequency is very well predicted. However, in accordance with the full-wheel and reduced model simulations, the second harmonic is consistently underpredicted by approx. 50%. Only the results from the PT and FG have been included as the FT simulation diverged after two runner revolutions and were deemed too uncertain to be included in this section.

In order to address the sensitivity of the choice of inlet condition, additional simulations were performed by applying a constant velocity profile as illustrated in Figure 6 (FG Original) as well as a sinusoidal periodicity (FG Sine Inlet). Figure 12 shows the spectral content of these simulations. Both alternatives match the blade passing frequency, but significantly overpredict the pressure amplitude of the first harmonic, especially FG Original.

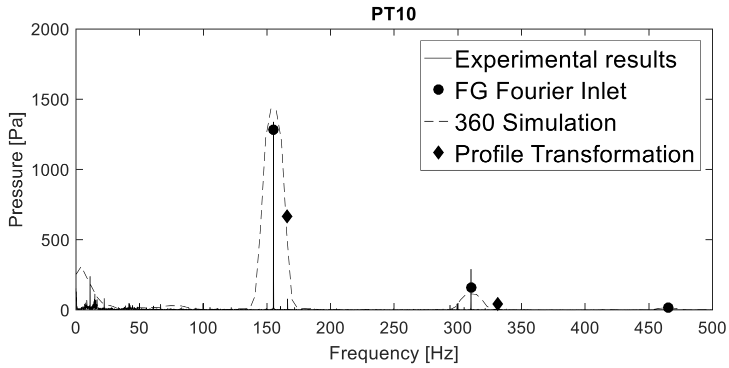

Figure 11. PT10 spectral content for Profile Transformation and Frozen Gust [6].

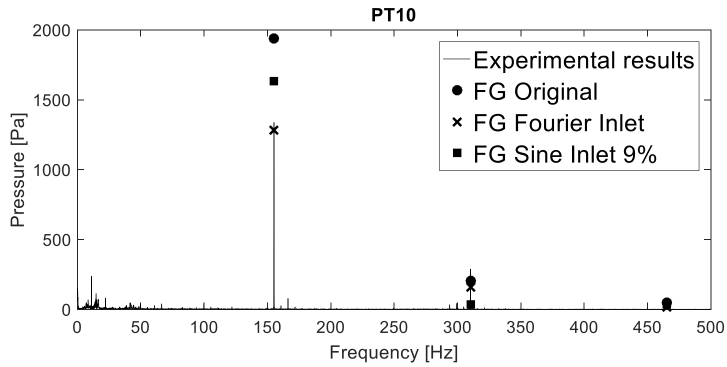

Figure 12. PT10 spectral content for Frozen Gust with different inlet conditions [6].

4. Speed up

The main purpose of this investigation has been to reduce computational time without compromising on accuracy to any significant degree. The present section will thus address the speed-up gained by the various methods presented.

Table 2 summarizes the node count for the reference case (Base Case) and the three modified cases for the full-wheel reductions together with relative speed up for each case normalized by number of time steps and compute cores. Comparing the No Volute and Short Draft Tube cases, most is gained by excluding the draft tube leading to a node reduction of 44%. And obviously, combining both effects results in the highest speed up of all cases. Nevertheless, all reductions show a close to linear speed-up with respect to mesh size and as discussed in section 3.1 only minor effects on the results were experienced by performing these simplifications. A more time efficient meshing procedure can also be expected, especially for the No Volute and Combined cases for which the meshing of the spiral casing has now been made redundant.

Table 2 also shows the relative speed-up gained by utilizing the various passage models. Here, the FT method has been included for comparison. Normalized by the number of runner revolutions, time steps and compute cores, clearly higher speed-up is achieved that for the full-wheel reductions, but only the FG Fourier Inlet method proved to give results at an acceptable level of accuracy. Remembering also that the results were obtained using information from an existing full-wheel simulation, this highlights the main advantage of the full-wheel reduction method. That is, despite the lower speed-up, no extra information is needed to achieve the level of accuracy presented and the full frequency content of the flow field is retained when moving from the Base Case to the simplified solutions.

Table 2. Speed-up of reduced models.

	Global Number of Nodes	Mesh size factor	Relative speedup [-]
Base Case	42476565	1	1
No Volute	36021993	1.18	1.2
Short Draft Tube	23822372	1.78	1.7
Combined	17367800	2.45	2.2
FT	-	-	3.7
FG	-	-	5.5
PT	-	-	8.3

5. Concluding remarks

Aiming to adopt to an industry time frame, the present work shows how a complete 360º simulation model of a high head Francis turbine can be reduced directly without losing accuracy in the RSI pressure that is of primary interest. The reduction is achieved by: 1) Reducing the draft tube and 2) Excluding the distributor except for the guide vanes. Consequently, great savings in mesh size and computational time is gained. This has been applied both on separate models as well as combining them, obtaining an ultimate mesh reduction of about 60% and a relative speed-up factor equal 2.2. A nearly linear speed-up with mesh size was achieved for all cases.

In general, reducing the draft tube will lead to the greatest reduction in computational mesh due to the vast number of elements being contained there, but modifying the distributor will also simplify the meshing procedure and consequently reduce the time needed for mesh generation. Despite a moderate speed-up compared to the passage models, the reduction of the full-wheel model does not lose any of the frequency content in the flow during this process. This is an essential aspect in describing the high level of accuracy that has been achieved. It is also shown that the entire volute could be replaced by a very simple set of inlet boundary conditions without affecting the results to any significant degree.

References

[1] Seidel U, Hübner B, Löfflad J and Faigle P 2012 *Evaluation of RSI-induced stresses in Francis runners* 26th IAHR Symp. Hydr. Machinery and Systems (Bejing)
[2] Trivedi C, Cervantes M J and Dahlhaug O G 2016 *Numerical techniques applied to hydraulic turbines: A perspective review* Applied Mechanics Reviews **68**
[3] Agnalt E 2016 *Pressure measurements inside a Francis turbine runner* Master thesis (NTNU)
[4] Jakobsen K-R G and Holst M 2017 *CFD simulations of transient load change on a high head Francis turbine* J. Physics: Conference Series **782**
[5] EDRMedeso AS 2018 *Recommended Practice HiFrancis WP3.1* (HiFrancis project)
[6] Tengs E, Storli P-T and Holst M 2017 *Effects of passage modelling in high head Francis turbines* Int. J. Hydropower & Dams (Hydro 2017 conference)
[7] ANSYS INC 2017 *ANSYS CFX Reference Guide Release 18*