Frozen Rotor and Sliding Mesh Models Applied to the 3D Simulation of the Francis-99 Tokke Turbine with Code_Saturne

To cite this article: N Tonello et al 2017 J. Phys.: Conf. Ser. 782 012009

View the article online for updates and enhancements.

Related content
- 3D Model Retrieval Based on Vector Quantisation Index Histograms
  Z M Lu, H Luo and J S Pan
- Study on vibration phenomena of guide vane inside bend by fluid structure interaction analysis
  S Tomimatsu, N Ohshima, K Uranishi et al.
- Airborne ultrasonic transducer using polymer-based elastomer with high output-to-weight ratio
  Jiang Wu, Yosuke Mizuno, Marie Tabaru et al.

Recent citations
- Numerical modeling of eccentric mass rotation in chamber filled with fluid
  AV Savchenko et al.
Frozen Rotor and Sliding Mesh Models Applied to the 3D Simulation of the Francis-99 Tokke Turbine with Code_Saturne

N Tonello¹, Y Eude¹, B de Laage de Meux², M Ferrand²

¹ Renuda, 329-339 Putney Bridge Road, London, SW15 2PG, UK
² EDF R&D, MFEE, 6, quai Watier -BP 49, 78401 Chatou, France

nicolas.tonello@renuda.com

Abstract. The steady-state operation of the Francis-99, Tokke turbine [1-3] has been simulated numerically at different loads using the open source, CAD and CFD software, SALOME [4] and Code_Saturne [5]. The full 3D mesh of the Tokke turbine provided for the Second Francis-99 Workshop has been adapted and modified to work with the solver. Results are compared for the frozen-rotor and the unsteady, conservative sliding mesh approach over three operating points, showing that good agreement with the experimental data is obtained with both models without having to tune the CFD models for each operating point. Approaches to the simulation of transient operation are also presented with results of work in progress.

1. Introduction

The Francis-99 workshops series is motivated by the need for numerical means of carrying out accurate and realistic computer modelling of high-head Francis turbines, to support the design of high efficiency turbines over the expanding operational range demanded by operators [1, 2]. Due to the size of the turbines, their complexity, with moving parts and vanes, tight tolerances, and complex, three-dimensional flows, such simulations remain challenging in terms of physics modelling, numerical stability and efficiency, computer requirements, and the size and duration of the calculations. To make it possible to validate numerical models and establish recommended practices, define the state of the art, and highlight areas where progress would be required, a set of experimental data obtained on the laboratory scale, Tokke turbine has been made publically available [3]. In addition, workshops have been organized for researchers to present the results of their simulations on the turbine test cases, with a focus on specific topics. As part of the Second Workshop, five different topics have been proposed, ranging from the influence of numerical and physical modelling in steady-state operation to uncertainty quantification.

In this paper, we present the research carried out on the influence of numerical modelling on the steady operation of the Tokke turbines for the three operating points studied experimentally: Part Load (PL), Best Efficiency Point (BEP), and High Load (HL). All the CFD modelling, simulations, and analysis have been performed with open source software. The tools in the SALOME platform [4] were used for pre- and post-processing, and the general, 3D, CFD solver, Code_Saturne [5] was used for the CFD modelling and the calculations. The setting up of the models was carried out on multi-
core, local workstations. Full simulations were carried out on up to 224 compute cores on EDF’s Porthos cluster. All the simulations have been conducted in 3D, over the full, 360° geometry. Simulations have been carried out using both the frozen rotor approximation and, full unsteady, sliding meshes technique. Results are compared for each case between the two modelling options and with the experimental data supplied for the workshop [3]. A further effort is also in progress to simulate the turbine in transient operation, and a summary of this research to date is also presented.

2. CFD Modelling
The CFD models were prepared based on the mesh files made available for the Second Workshop, and further adapted for Code_Saturne v4.0.

2.1. Mesh Preparation
Complete, 360° meshes were built for the simulations using the ‘cgns’ files available for the spiral casing, distributor, runner and draft tube. A view of the complete mesh is given in figure 1 below.

Figure 1: View of the surface of the full computational mesh and the different turbine components.

For the runner, the full wheel was rebuilt based on the single blade and splitting blade passage available in the mesh ‘RU-AG5-HighRe.cgns’. The initial mesh was imported in the ‘SMESH’ module of SALOME and then copied and rotated around a full revolution, with the individual meshes merged into a unique mesh for the wheel. All the operations were carried out using SALOME Python scripts. Further examination of the mesh revealed nodes duplicates in the original mesh. Those were also removed with SALOME, by automatically fusing duplicate nodes (figure 2).
Having assembled the full 3D mesh, further work was undertaken to build a new interface between the runner and the draft tube. Sliding mesh trials with the initial interface showed that the large difference in discretisation between the runner and the draft tube (figure 3a) at the position of the interface, and the resulting gaps between the two meshes at the inner and outer edges of the interface (figure 3b) were preventing adequate face joining during the mesh sliding operations in Code_Saturne.

SALOME scripts were programmed to create a thin slice in the initial mesh, remove the original cells and rebuild a conformal interface, joining the upper side of the new interface mesh to the runner and the bottom side to the draft tube (figures 4a-b).

**Figure 2:** Zoom on duplicate nodes found in the runner mesh.

**Figure 3a-b:** Views of the initial meshes at the interface and during rotation.
Figures 4a-b: Views of the new, two layers, conformal interface mesh.

This final mesh was used for all the calculations, with the frozen rotor and the sliding mesh method.

2.2. CFD Solver Setup

The calculations were setup and run with version 4.0 of the open-source, general CFD solver Code_Saturne, developed by EDF R&D. The solver is based on a Finite Volume discretisation on collocated meshes. The Navier-Stokes equations can be solved in three-dimensions, with either RANS or LES approaches for turbulence. A large number of RANS models are available, from algebraic to Reynolds stress or, alternatively, LES can be performed for incompressible or weakly compressible flows. The equations are solved with an implicit, iterative, PISO type algorithm. At the prediction step, the convective and diffusive fluxes are centred, second-order accurate in space. In the correction step, an algebraic, multigrid method is applied to the solution of the pressure equation. The time integration is first-order accurate.

For turbomachine calculations, Code_Saturne makes it possible to run both approximated, frozen rotor, and full unsteady, sliding mesh simulations. The frozen rotor simulations are based on the Multiple Reference Frame (MRF) methodology while for sliding meshes approach, the rotation of the rotor mesh is explicitly taken into account.

Two options are available in Code_Saturne to account for the rotor mesh rotation. The older methodology consists in coupling two separate instances of Code_Saturne running on separate meshes. The interfacial coupling is then handled through specific boundary conditions formulations for the two-instances. The newer method uses a single instance running on the complete mesh, including both static and rotating parts, and uses an advanced mesh-processing algorithm in order to perform the meshes joining. That way, the cell faces on the rotor/stator interface are treated as internal faces, which makes it possible to conserve all the fluxes (mass, momentum, etc.) through the interface. With this sliding mesh method, the rotor mesh coordinates are first updated at the beginning of each
time step, and then the mesh joining is performed between the prediction and correction steps of the algorithm, just before the mass flux are updated.

The crux of the conservation properties is the coupled solver which has been implemented in Code_Saturne. Rather than solving them one after the other, in the prediction step the momentum equations for all three velocity components are solved together at the same time. This results in one single matrix, which is three times larger than the individual components’ equations’ matrices but makes it possible to obtain a system where more terms can be written in an implicit, accurate manner. In particular, the boundary conditions at walls and no-slip boundaries can be applied on several components together.

In validation tests, EDF has shown that this new algorithm helped to reduce both solution oscillations and calculation times compared to the original coupling method. Therefore, it is the recommended approach for mesh rotation, provided that the mesh discretisation is similar on each side of the sliding interface in order to allow adequate face joining operations. The algorithm is fully parallelised and has demonstrated linear scaling ability in studies realised by Renuda and EDF.

The sliding mesh simulations discussed in this paper were carried out using this method, which is part of the newly developed turbomachinery module of Code_Saturne.

2.3. Numerical and Physical Model

For the single-phase simulations of the Second Workshop, water was modelled as an incompressible fluid with a density of 999.8 kg/m$^3$ and a viscosity of 9.57×10$^{-7}$ m$^2$/s. The RANS k-ω SST model was applied to simulate the effects of turbulence. Figure 5, taken from the calculations results described in Section 3 below shows that the non-dimensional wall distance $y^+$ of the first computation node is located either in the viscous sub-layer or in the log-region. Therefore, the k-ω SST model is well-suited to this mesh and it has been preferred to other turbulence models available in Code_Saturne.

![Figure 5: Non-dimensional wall distance of the first computation node (BEP).](image)

Both frozen rotor and sliding mesh simulations were carried out with an unsteady time marching scheme with a time-step of 10$^{-4}$ s. The maximum CFL in the calculations was approximately 10, with an ensemble average over the cells lower than one and a spatial average of the order of 10$^{-2}$.

Three seconds of physical time were simulated for frozen rotor computations at each operating points, corresponding to about 16.5 revolutions of the runner. In spite of the absence of runner mesh rotation in the frozen-rotor calculations, some unsteadiness of the flow was observed in the results, especially in the draft tube elbow or downstream of the runner at part load. Thus, a time averaging of the results was performed during the final 4.4 revolutions (0.8 seconds) in order to compute the averaged flow.
The sliding mesh computations were then performed for an additional 1.5 physical second, starting from the corresponding frozen rotor computation, for each operating point. The last 0.8 second was used for the time averaging.

2.4. Computing Resources
The simulations were carried out on the EDF cluster Porthos and 224 compute cores. Typically, computations took 24 hours for each second of real time. The calculations times with the sliding mesh modelling were approximately 25% longer than with the frozen rotor approach.

3. Steady-State Operation
Simulations were performed for the three operating conditions (PL, BEP, HL). The numerical results are presented below for the two modelling approaches and compared to the experimental results.

3.1. Velocity
The velocity magnitude computed with the sliding mesh model is shown in figures 6 and 7 below for the BEP and the two off-design conditions, respectively. The flow structure is similar with the frozen rotor model. At BEP, the flow is uniform, without noticeable recirculations. However, at PL and more so for the HL conditions, the vortex tube is unstable and flaps between the walls of the draft tube. At HL, significant non-uniformity and losses can be observed in the spiral cone. Calculating this type of flow on a mixed grid made up of polyhedral and hexahedral cells is challenging for CFD solvers.

Figure 6: Instantaneous velocity field, sliding mesh simulation. BEP operating condition.
The numerical results are now compared to the experimental data measured along lines 1 and 2 [3] (figure 8).

Figure 8: Positions of the experimental measurements lines 1 and 2. From [3].

The results are plotted below (figure 9) for each operating condition, and with the three sets of results on each graph (experimental in black, sliding mesh in green, frozen rotor in red).
Comparing the calculated velocities for the vertical velocity to the experimental measurements shows that both sets of simulations successfully capture the changes in velocity profiles between the three operating conditions and along both measurement lines below the cone. At the Best Efficiency Point, the best results are obtained with the sliding mesh simulations. For all three operating points, the sliding mesh results agree well with the experimental results away from the core of the draft tube ($0 \leq \tilde{r} \leq 0.4$, $0.6 \leq \tilde{r} \leq 1.0$). For the Part and High Load operating points, the magnitude of the experimentally measured velocity nearer the centre of the draft tube is significantly smaller than for the BEP. The flow is more still and, for the HL point, nearly reverses. At Part Load, the sliding mesh results underpredict the velocity in the core. However, the thickening of the central region of lower velocity downstream of the cone shown from Line 1 to Line 2 is predicted correctly. At High Load, the velocity predicted with the sliding mesh simulations overestimates the experimental results, although, again the edges of the central region and the inflexion in the velocity profile at the edges of the central zones are predicted correctly. For the PL and HL conditions, overall the frozen rotor calculations give results which are closer to the experiments.

The calculated and measured radial velocity along lines 1 and 2 are compared in figure 10 below. The experimental data is discontinuous with peak and troughs, and is not symmetrical. The radial velocity magnitude is an order of magnitude lower than the axial velocity. The overall trends are correct in the numerical results, with the sliding mesh model giving slightly better results than the frozen rotor. However, for both models the calculated velocities show variations which are significantly larger than the variations in the measured velocities. A strongly refined mesh would probably be required in order to better resolve this very challenging region of the flow in the wake of the impeller blades, where velocities are low and variations of velocities of the same order of magnitude as the velocities themselves.
3.2. Pressure and Hydraulic Head

The net head is calculated as the total pressure change through the turbine, excluding the hydrostatic effects:

\[ H = \frac{p_{abs,1} - p_{abs,2}}{\rho g} + \frac{v_1^2 - v_2^2}{2g} + \Delta z, \]

where \( \Delta z = 1.0715 \) corresponds to the height difference between the inlet and outlet reference planes (figure 11).

Figure 10: Numerical and experimental radial velocity vs. normalised radius along lines 1 and 2.
The calculated pressures and net hydraulic heads are compared to the experimental values for the three operating points in Tables 1-3 below.

Table 1: Pressure and hydraulic head. Part Load operation.

<table>
<thead>
<tr>
<th>PL</th>
<th>VL2 [kPa] (error [%])</th>
<th>Pin [kPa] (error [%])</th>
<th>H [m] (error [%])</th>
</tr>
</thead>
<tbody>
<tr>
<td>experiment</td>
<td>168.97</td>
<td>218.46</td>
<td>11.88</td>
</tr>
<tr>
<td>frozen rotor</td>
<td>161.60 (-4.36)</td>
<td>221.17 (+1.24)</td>
<td>12.18 (+2.53)</td>
</tr>
<tr>
<td>sliding mesh</td>
<td>161.51 (-4.41)</td>
<td>223.29 (+2.21)</td>
<td>12.35 (+3.96)</td>
</tr>
</tbody>
</table>

Table 2: Pressure and hydraulic head. Best Efficiency Point operation.

<table>
<thead>
<tr>
<th>BEP</th>
<th>VL2 [kPa] (error [%])</th>
<th>Pin [kPa] (error [%])</th>
<th>H [m] (error [%])</th>
</tr>
</thead>
<tbody>
<tr>
<td>experiment</td>
<td>173.17</td>
<td>215.93</td>
<td>11.97</td>
</tr>
<tr>
<td>frozen rotor</td>
<td>163.54 (-5.56)</td>
<td>217.20 (+0.59)</td>
<td>12.10 (+1.09)</td>
</tr>
<tr>
<td>sliding mesh</td>
<td>164.67 (-4.91)</td>
<td>219.49 (+1.65)</td>
<td>12.33 (+3.01)</td>
</tr>
</tbody>
</table>

Table 3: Pressure and hydraulic head. High Load operation.

<table>
<thead>
<tr>
<th>HL</th>
<th>VL2 [kPa] (error [%])</th>
<th>Pin [kPa] (error [%])</th>
<th>H [m] (error [%])</th>
</tr>
</thead>
<tbody>
<tr>
<td>experiment</td>
<td>178.87</td>
<td>212.94</td>
<td>11.88</td>
</tr>
<tr>
<td>frozen rotor</td>
<td>172.05 (-3.81)</td>
<td>212.51 (-0.20)</td>
<td>11.88 (-0.04)</td>
</tr>
<tr>
<td>sliding mesh</td>
<td>168.07 (-6.04)</td>
<td>216.58 (+1.71)</td>
<td>12.18 (+2.52)</td>
</tr>
</tbody>
</table>

The evolution with the different operating conditions and the overall magnitude of the pressure in the vaneless space upstream of the blades (VL2) are captured very well by both numerical models. Likewise, the inlet pressure and heads are very similar in the simulations and the experiments. Overall, the frozen-rotor results are in slightly better agreement with the experimental ones than the sliding mesh results.

Both models tend to overestimate the head, which can be explained by the fact that losses such as gap losses are neither modelled nor implicitly taken into account in the present CFD model. In fact, the larger values of hydraulic head computed with the sliding mesh model indicate that the numerical losses are smaller for the sliding mesh model than the frozen rotor one.

The pressure fluctuations in the draft tube (DT5 and DT6 sensors) were not captured in the simulations and the experiments.

4. Transient Operating Conditions

In addition to the simulations of the steady-state operation, work is also currently in progress to perform unsteady, frozen-rotor simulations of the transient operating conditions. The methodologies
being developed and evaluated are based on the ALE solver already available in Code_Saturne, to impose displacements on the mesh nodes of and around the guide vanes.

In the first approach trialed, a global transformation was defined to prescribe the motion of all the nodes in a specific radial region, based on the desired variation of vane angle. Views of a proof of concept calculation are shown in figure 12 below for a simplified test case. Note that this global transformation does not coincide with the exact periodic transformation and leads to a deformation of the guide vanes.

![Figure 12: Validation of the mesh displacement for simplified stator. Displacement magnitude before (left image) and after (right image) displacement of the stator vane.](image)

Nevertheless, the methodology was then applied to the Tokke case (load reduction, from BEP to PL) to evaluate if simulations could be carried with transient vanes motion. The results showed that the simulations were stable and results could be obtained, with the vanes moving over time, but that the nodes motion which had been prescribed resulted in excessive deformation of the vanes (figure 13).

![Figure 13: Snapshots of the transient vanes operation at three different vanes positions.](image)

However, these results are encouraging and have provided the impetus for the development of a more robust approach based on new mesh-processing capabilities, to move individual nodes by interpolating their position over time between the initial and final mesh.

5. Conclusions
Modelling and simulations of the Tokke turbine at three steady-state operating points have been carried out using the open-source software SALOME for pre- and post-processing and Code_Saturne for the CFD simulations. The simulations on the full 3D model of the turbine have been run with two different methodologies, the frozen-rotor approximation and a moving mesh for the runner on over 200 compute cores.

Results obtained with the k-ω SST RANS model for turbulence show that the CFD solver can be used successfully to obtain an accurate local description of the flow at various operating conditions,
without making changes to the computational model. The main characteristic of the turbine are also correctly estimated.

When considering which modelling approach to choose, the comparisons between the frozen rotor and sliding mesh results show that the sliding mesh approach does not offer significant advantages over the frozen rotor model for predictions of the flow in the draft tube and the global characteristic of the turbine. This is perhaps not overly surprising considering the large number of runner and guide vanes blades and the fact that stator/rotor interactions have a lesser impact in the draft tube. However, for more accurate simulations involving losses for example, or to look at the effect of rotor-stator interaction in detail for the different operating conditions, the sliding mesh modelling would be better suited. In the present state, the accurate prediction of pressure fluctuations appears to be beyond the scope of an incompressible flow solver.

The turbomachinery functionalities of Code_Saturne are already being used for hydraulic pump simulations. The reported simulations aimed at examining the capability of the code to simulate hydraulic turbine flows, including complex phenomena occurring away from design conditions and transient operations. The test results present an excellent opportunity to push and validate the solver and, whilst the results obtained are encouraging, they open the door to new developments for transient conditions and to performing more detailed flow analysis of such conditions and their impact on turbine design. Further work would also be of interest to better resolve the radial velocity component near the runner outlet.

Acknowledgements
The authors would like to thank the organisers of the Second Workshop for the use of their CAD, meshing, and experimental data.

References
[3] https://www.ntnu.edu/nvks/test-case