Transitent CFD Simulation of a Francis Turbine using OpenFOAM

Master Thesis / February - August 2010

1. Motivation

The purpose of this project is to evaluate the possibilities offered by OpenFOAM (OF), an open-source software, in order to perform a transient simulation of a full Francis turbine. The interest of OpenFOAM is to avoid license cost. The full geometry is needed to compute dynamic loads without simulations.

In order to provide a proper initialization for the transient analysis, steady state simulations are performed and a preliminary software comparison is investigated with these results. Then, the transient simulations are performed in order to focus on the rotor-stator interaction.

The results are compared to a famous commercial code (CC) for Computational Fluid Dynamics (CFD) applications and also, when possible, to experimental measurements provided by Voith Hydro.

2. Numerical Setup

The mesh is created by an in-house mesh generation tool provided by Voith Hydro. After the import of the structured mesh in OF, the 360° geometry is created. This mesh corresponds to a relative coarse resolution with 10107808 cells and y⁺ values around 25 on blades and 40 on walls. The boundary conditions are set according to the discharge for the inlet and rotational speed for the runner. The Generalized Grid Interface (GGI) is used to couple the different zones.

In order to decrease the computational time, a domain decomposition is applied. Therefore, the use of cluster is necessary. Voith Hydro and Stuttgart University provided these computational resources.

3. Steady State Simulations

The comparisons of velocity contours in XY and XZ planes highlight a good agreement of the flow patterns and maximum velocity magnitudes between the OF and CC results. A comparison of characteristic values is also performed between the two software. The table gives the specific energy between inlet and outlet, discharge, extracted power between the two GGIs (computed with the Euler equation), force on the shaft, torque on the shaft and efficiency of the turbine between inlet and outlet.

The difference between OF and CC is always less than 2%, which highlights consistent results.

<table>
<thead>
<tr>
<th>Characteristic values</th>
<th>Revert</th>
<th>gH2 = gH1</th>
<th>Q/Q_{ref}</th>
<th>P/P_{ref}</th>
<th>T_{c} [kN]</th>
<th>\eta [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>OF</td>
<td>CC</td>
<td>CC</td>
<td>OF</td>
<td>OF</td>
<td>OF</td>
<td>OF</td>
</tr>
<tr>
<td>1250</td>
<td>1250</td>
<td>0.9994</td>
<td>0.9997</td>
<td>0.9996</td>
<td>0.9997</td>
<td>0.9994</td>
</tr>
</tbody>
</table>

4. Transient Simulations

As for the steady state simulations, the velocity contours and characteristic values are in good agreements.

In order to analyze the rotor-stator interaction, the pressure evolution is recorded by four probes. The amplitude and mean value of the signal are computed with the least square method. A good agreement of the mean values is shown. However, large differences in amplitude computation are observed.

5. Comparison with LDV Measurements

In order to complement the CFD investigation, Laser Doppler Velocimetry (LDV) measurements are provided by Voith Hydro along a line in the diffuser.

The qualitative flow field for both OF and CC is well represented on this line. Moreover, it is difficult to define which of OF or CC is the closest from the experimental data.

6. Outlook

This study highlights the possibility to use OpenFOAM for the characteristic parameters computation. Indeed, a difference less than two percents is observed between OpenFOAM and the commercial code.

Moreover, the flow field is well represented despite some differences with the experimental velocity profiles for both software. However, further investigation on implementation of proper scalable wall function has to be done for OpenFOAM in order to improve the velocity contour comparison in the diffuser.

Furthermore, improvement of the transient OpenFOAM solver has to be done in order to improve the convergence, and so, to allow the use of OpenFOAM for industrial applications.