A History of Vortices

GE Power is a world leading supplier of solutions for power generation, from engineering to manufacturing. Detecting and mitigating critical flow structures in water pumping stations is a complex engineering task, that has always been based on experimental activities. Now GE Power can rely also on CFD modelling and on the support of EnginSoft

Power Generation

Introduction

Vortex generation in pumping stations must be prevented or limited to superficial and low intensity vortices, because deep and intense ones can severely damage the pumps or / and affect their performance (flow, head & efficiency) due to non uniform velocity distribution at the pump impeller inlet and their air content. This kind of dangerous vortices might appear in some operational conditions related for example to low water levels or high rotational speeds and high flow rate values.

Vortex generation must be carefully studied and verified during the design phase of pumping stations, and measures aimed at preventing vortices or reducing their intensity must be taken.

One of the key factor to prevent vortices is the submergence, that is the minimum immersion depth required by the pump to prevent vortices. It varies according to the size of the pump and the rotational speed. As an indication, it is typically 2.5 times the diameter of the hydraulic part, but it must be adapted to the pump NPSHr conditions as well as the pump and pumping station geometry since this rule is very general.

Common practice is to build scaled prototypes of a portion of the pumping station, typically one single pump room is built, in order to simulate the real working conditions and to verify the presence and nature of vortices. Vortices are detected visually, from the observation of the free surface curvature and with the support of colored tracers, that can make the flow structures visible and help the classification of vortices according to the HIS vortex classification shown in Figure 1. This classification is essentially based on visual identification and characterization of structures such as swirls, ripples and air bubbles. This kind of activity requires experienced engineers to distinguish between Type1-2, that are not dangerous, and the other types, that must be avoided. The complexity of the classification is also associated to the transient nature of these vortices that tend to be unstable and to appear and disappear. In this frame the use of Computational Fluid Dynamics (CFD) can support and supplement the experimental activities by providing detailed information about the flow behavior in the whole pumping station and about vortices and fluid structures in the proximity of the pumps.

It must be pointed out that also the CFD simulation of pumping stations and pump rooms is not a simple task. The accurate solution of multi-phase flows on complex geometries, with air entrainment and transportation in unsteady conditions requires high quality computational mesh, high order accuracy and parallel computing. Moreover the identification and classification of flow structures requires the definition of quantitative criteria during the post-processing of CFD results.

Nonetheless GE Power and EnginSoft accepted the challenging task of developing and running CFD models of a pumping station with the aim of comparing the simulated vortices with the ones detected during experimental tests on a scaled model. If CFD demonstrated to be able to identify and classify vortices with the same accuracy of experimental tests, it could be used to speed up the design process, for example to detect critical operating conditions or to compare alternative designs to mitigate vortices intensity.

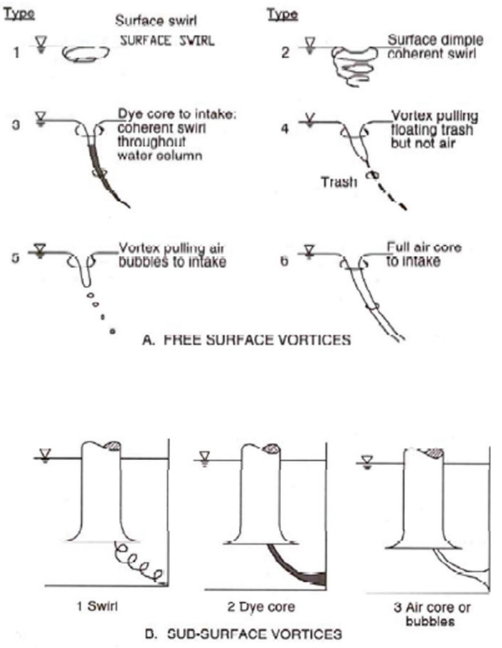

Figure 1 – HIS vortex classification

Pumping Station

GE-Power and EnginSoft developed real and virtual prototypes of the cooling water pumping station for the Narva Power Plant (2 x 300 MW) of Eesti Energia in Estonia. The main cooling water pumps are sized for a nominal flow rate of 19.000 [m3/h] each, while the auxiliary pumps are sized for a nominal flow rate of 1.500 [m3/h]. Figure 2 shows the CFD model of the pumping station, with four main pumps and four water inlets. Considering also the auxiliary pumps and the filters sections, located just downstream the inlets, the resulting flow path includes many disturbances and obstacles from the inlet to the pumps. However the area interested by vortices is close to the main pumps intake, where water velocity is higher than in other areas.

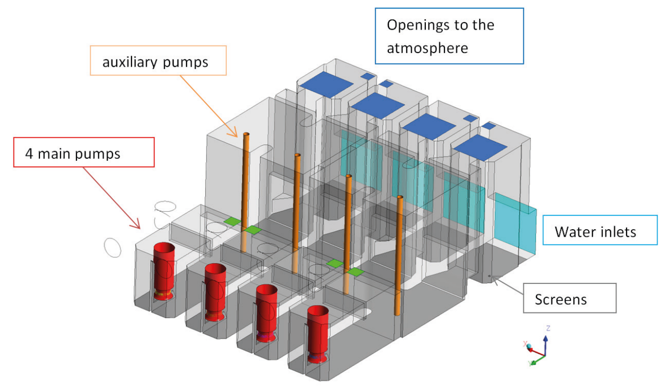

Figure 2 – Pumping station CFD model

For an accurate solution of the free surface and of vortices a fine and high quality hexahedral mesh must be generated. Figure 3 shows the mesh on a section plane of one of the pump rooms, with mesh refinement at the water free surface.

The fact that the pumps are located in separated rooms and that the potential vortex generation takes place locally in proximity of the pumps allows reducing the model to one single pump room. A global CFD model of the pumping station can be used to calculate and visualize the global flow behavior, above all when some of the inlets and pumps are off, but the use of such a large model is not suited for detailed studies related to local vortex generation. Figure 4 shows the CFD model of two different configurations of the same pump room. In the second configuration a skimmer wall is located upstream the pump to break the vortices and reduce their intensity. GE Power made built scaled physical models of the two pump rooms in an external laboratory and verified the beneficial effect of the skimmer wall. The validation and the quality of the CFD results depend on the capacity of the CFD model to simulate the same types of vortices that were detected in the laboratory in the two cases: Case I,

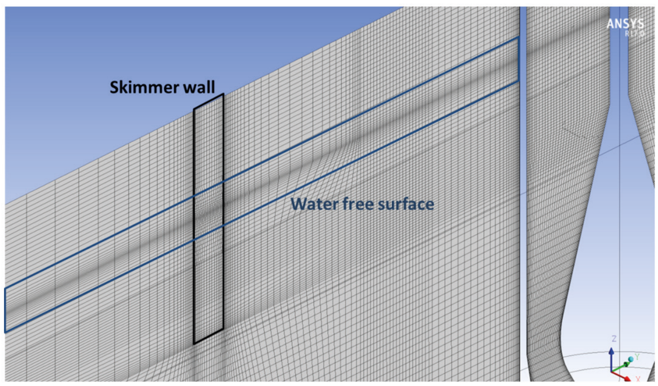

Figure 3 – Mesh on a cut plane

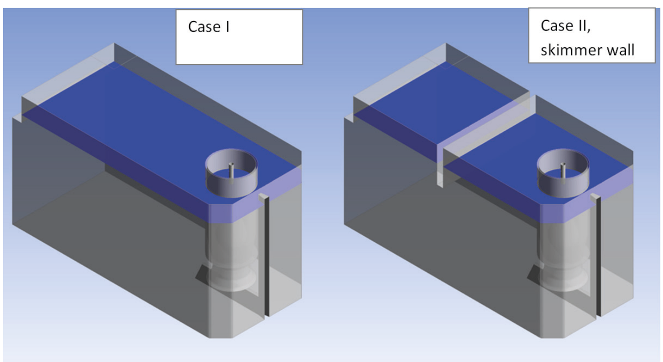

Figure 4 – Pump room CFD model: Case I (left), Case II, with skimmer wall (right)

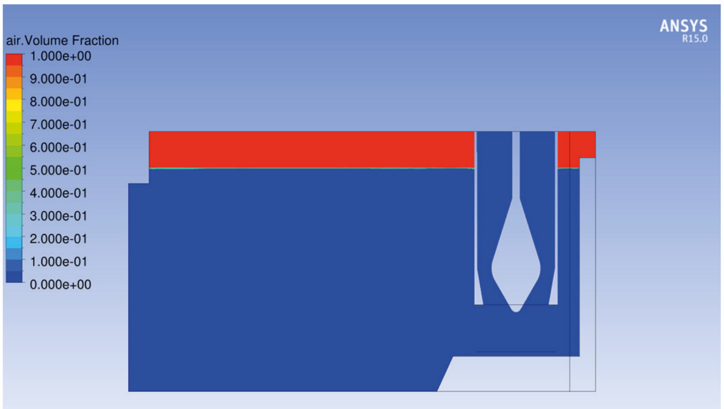

Figure 5 – Contour showing air (red) and water (blue) distributions on mid vertical plane

without skimmer wall and Case II, with skimmer wall.
The CFD procedure is based on two steps: the initial one is a steady state simulation that is used to initialize the flow field, the second one is a transient simulation, about 30[s] of physical time are simulated. The transient simulation is needed to resolve the transient nature of the vortices.
A free surface multi-phase model is used, with water and air. A pressure profile is applied at the inlet of the pump rooms, while a flow rate value is applied at the outlet of the pumps. It must be pointed out that, like in the experimental facility, only the pumps external walls are present, while the impeller and its rotational effect are not present in the CFD model.

Vortex Detection and Classification

Figure 5 shows the air and water distributions on a section plane of Case I (no skimmer wall). At first sight the free surface seems flat and no vortex or other structures are visible. Actually dangerous vortices are there.

In order to detect and classify vortices different ways can be used, the most intuitive one is to create iso-surfaces of the water volume fraction at different levels. Figure 6 shows the water volume fraction at 99.5%. While no structures are visible in Figure 5, Figure 6 clearly shows two vortices on the two sides of the pump. The same vortices are made

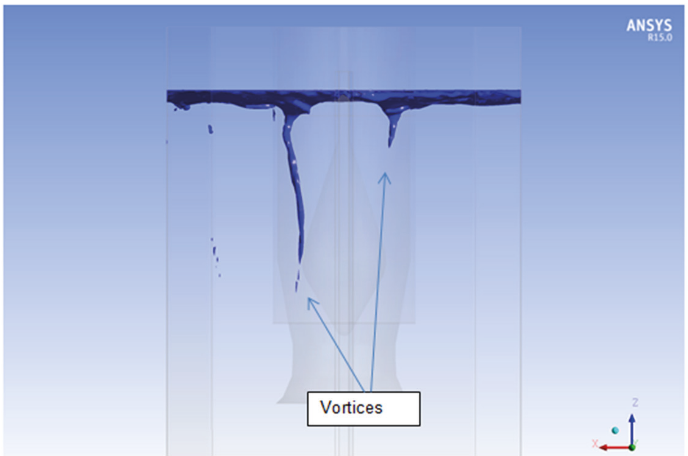

Figure 6 – Isosurface of water volume fraction at 99.5%, lateral view

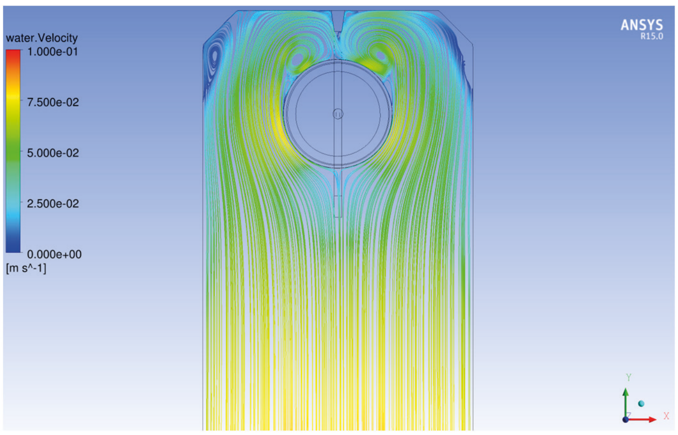

Figure 7 – Streamlines on a plane located 14.5mm beneath the free surface

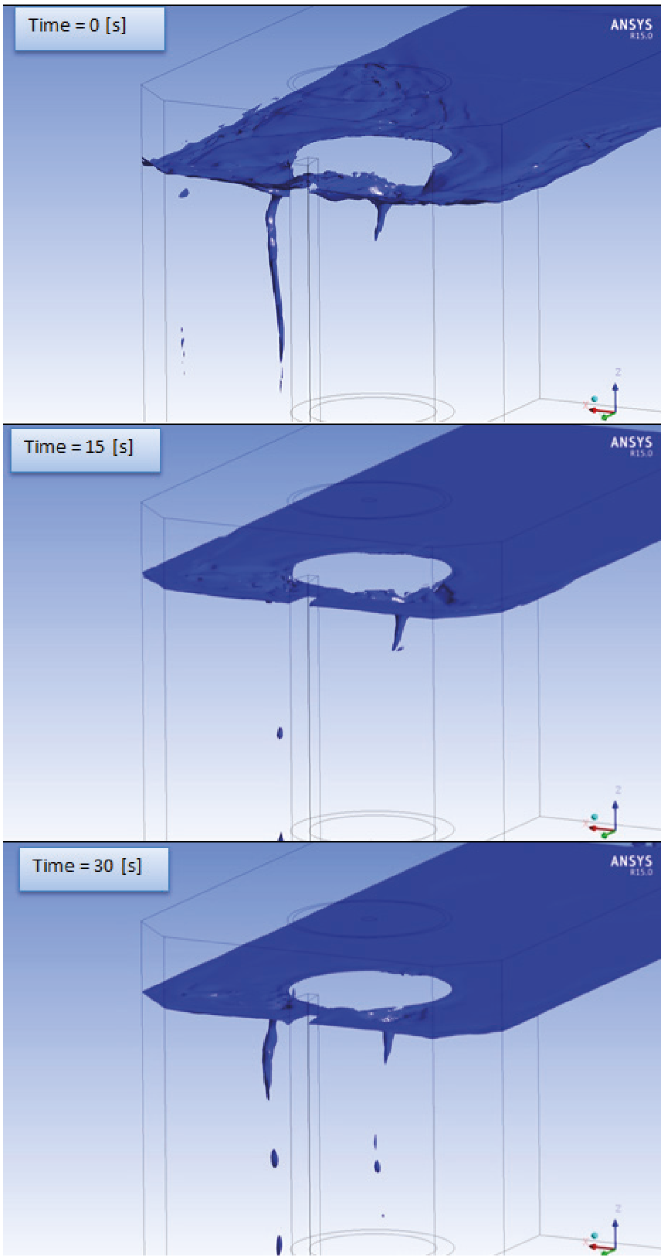

Figure 8 – Isosurface of water volume fraction at 99.5%, at time = 0[s] , 15 [s] and 30 [s]

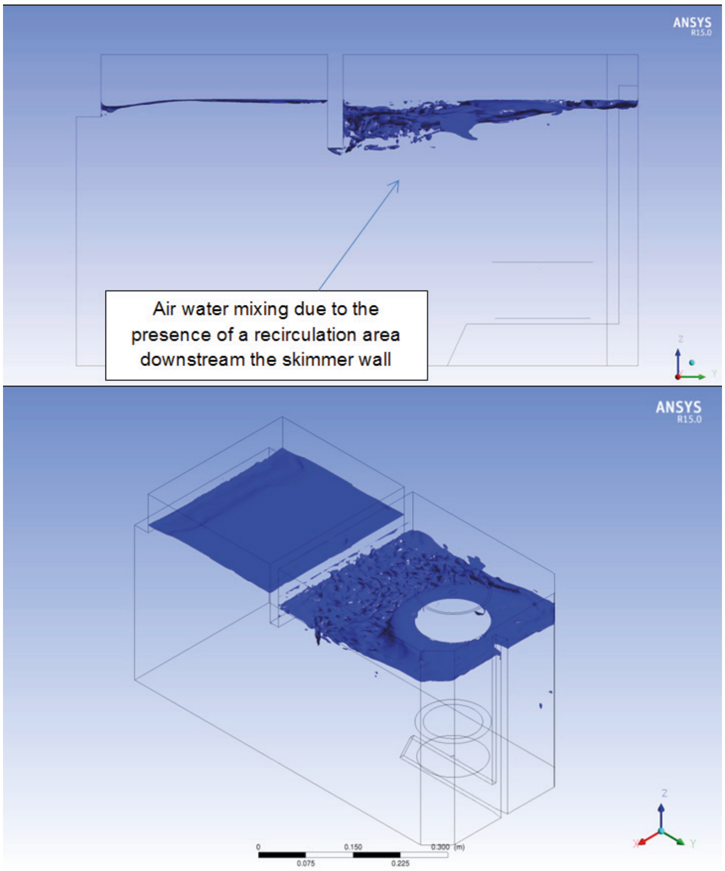

Figure 9 – Isosurface of water volume fraction at 99.5%

evident by water streamlines in Figure 7. The two vortices are almost symmetric and located between the pump and the rear wall of the room.
Figure 6 and Figure 7 actually show the solution of the CFD steady state simulations.
The same kind of visualization can be done on the transient CFD results. Figure 8 highlights that the vortices that are initially present tend to disappear, change in depth, position and intensity and then tend to re-appear after about 20 [s].
All these pictures clearly show the presence and unstable nature of the two vortices in Case I. In this case the structures are classified as Type 3 according the HIS vortex classification shown in Figure 1. This type of vortex is dangerous and must be avoided because of its depth and air content. The same kind of structure and classification type are identified using the laboratory testing facilities. Hence CFD demonstrated to be in good agreement with physical testing.
The same kind of CFD analysis is carried out on Case II. Figure 9 shows that in this case a vertical vortex is created just downstream the skimmer wall. The vortex increases the turbulence level locally, breaks other structures and entrains air that tends to re-circulate in a superficial mixing zone.
The skimmer wall prevents the generation of deep vortices of type 3 or higher. Only minor superficial vortices with no air content can be detected in Case II. These vortices are classified as Type 1 or 2 according the HIS vortex classification and they are not dangerous for the pump. Also in this case CFD predictions are in line with experimental tests, thus demonstrating that CFD can be confidently used to identify vortices and to compare the effect of different design solutions.

Conclusions

GE Power and EnginSoft carried out a validation study on CFD models for the detection and classification of vortices in cooling water pumping stations. The comparison of CFD results and experimental tests demonstrated that the CFD methodology can correctly identify vortices according to the HIS vortex classification.
This means that virtual prototyping can be used both for the study of the global flow behavior in pumping stations and for the detailed study of critical flow structures in the proximity of the pumps. The big advantage of CFD is the insight into the way vortices forms and how to intervene in order to remove them or to reduce their intensity.

Alessandro Arcidiacono, Massimo Galbiati - EnginSoft

For more information:
Massimo Galbiati, EnginSoft
m.galbiati@enginsoft.it

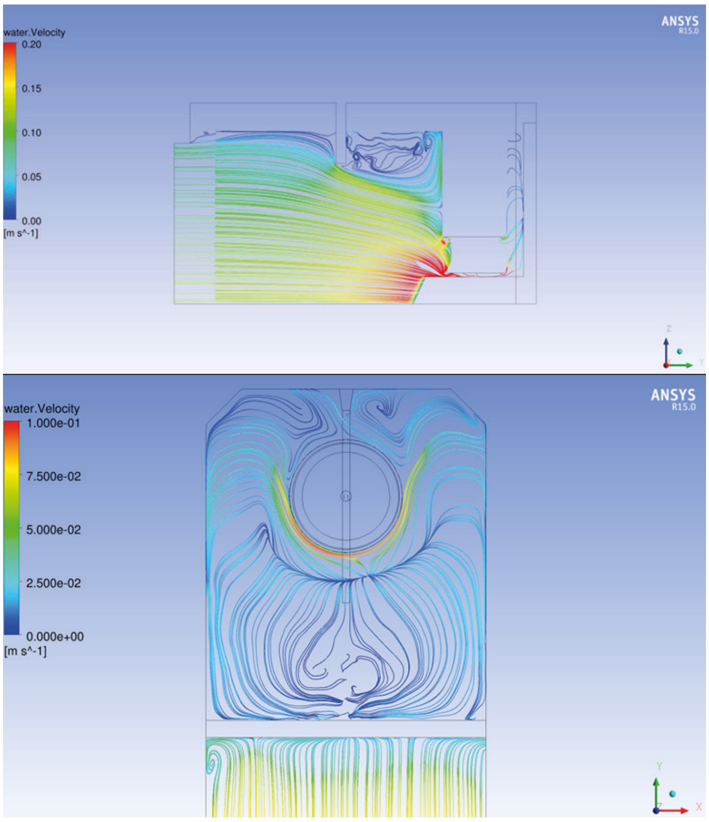

Figure 10 – Streamlines on a vertical plane and on a plane located 16.5mm beneath the free surface.