THREE DIMENSIONAL NUMERICAL MODELLING FOR FLOW ANALYSIS INSIDE A PUMPING STATION

Olivier, Bertrand[1]
ARTELIA Eau & Environnement, 6 rue de Lorraine, 38130 Echirolles, FRANCE
olivier.bertrand@arteliagroup.com

Pierre-Etienne Loisel, Caroline Girard,
Pierre-François Demenet, Julien Schaguene
ARTELIA Eau & Environnement, 6 rue de Lorraine, 38130 Echirolles, FRANCE
pierre-etienne.loisel@arteliagroup.com, caroline.girard@arteliagroup.com
pierre-francois.demenet@arteliagroup.com, julien.schaguene@arteliagroup.com

KEY WORDS

Numerical modeling, pumping station, free surface, vorticity, turbulence

ABSTRACT

The constant improvement of numerical tools, especially in terms of speed of execution and reliability, can address new issues for complex flows. Widely encountered in drinking water projects, waste water or industrial process, pumping stations must be designed and dimensioned in particular to ensure adequate water supply conditions for pumps, in order to ensure their efficiency and lifetime. ARTELIA Eau & Environnement regularly realizes design or layout of pumping stations on the base of 3D hydraulic modeling, either physical or numerical. Three-dimensional numerical modeling allows to understand more and more phenomena with both a global and exhaustive visualisation of flows and more detailed analyzes by computing additional parameters such as vorticity. Through the presentation of different studies, contributions and limitations of the numerical model compared to the physical one, for this type of structure, are presented.

1. INTRODUCTION

As an independent engineering, project management and consulting group, Artelia operate in nine markets which are building construction, water, energy, environment, industry, maritime, multi-site projects, transportation and urban development. The sector dedicated to the water and environment works on 4 of these markets (Water, Maritime, Energy, Environment). Our Maritime activity possesses a unique range of facilities to support its project teams: a numerical modelling unit dedicated essentially to the development of high-quality computation codes focusing mainly on hydrodynamics, a world-renowned physical modelling laboratory, a navigation simulator and a shiphandling training centre. All these tools give the possibility to propose studies focused on the objectives of the client and on the faisability or the optimization of the project.

For some of these nine markets, lot of industries need cooling system, which requires water intakes, and for most of them a pumping station. As example, pumping station are encountered in drinking water plants, waste water plants or power plants. for most of these industries. Most large-capacity desalination or power plants use open-ocean intake systems with onshore or offshore inlets. In these cases, intakes are one of the critical component. For these coastal facilities, problems with seawater intakes and corrosion are the two primary causes of unscheduled downtime.

[1] Corresponding author

2. PUMPING STATION

From upstream to downstream, the inlet, the forebay and the pump bay are the three parts of a pumping station that are hydraulically significant. Some documents and publications [1, Sanks 1998] [2, ANSI/HI 1998] are widely accepted as international standards for pumping station design and are used for hydraulic design by a lot of companies.

Ideally, the flow at the pump inlet should be uniform and steady, without swirl, vortices or carried air. Non-uniform flow at the pump intake can reduce efficiency and cause pulsating loads on the propeller blades, resulting in noise and vibrations. Unsteady flow can also cause fluctuating loads, noise and vibrations. Swirl in the intake can change the head, flow, efficiency and power in undesirable ways. It can also augment vortices. Experience with designs already in use provides valuable guidelines for the design of new pumping stations.

3. PHYSICAL AND NUMERICAL TOOLS

Common practice is to build scaled prototypes of the pumping station in order to simulate the working conditions. The constant improvement of numerical tools, especially in terms of speed of execution and reliability, can address new issues for complex flows observed in these facilities. The aims of the models are to check the shape of the pump sump and bays, and approach conditions, avoiding unacceptable pre-swirl, vortices and air intake at the pumps.

3.1 Physical model

For the physical model, where surface or sub-surface vortices may be a possible problem, it is important to exercise care and judgment when selecting the model scale. The scale of the model must be selected in order to achieve dynamic similarity with the prototype, and to correctly reproduce any vortex action at the pump suctions. However, it is impossible in a reduced scale model to satisfy simultaneously the conflicting requirements of the three scaling laws defined by the Froude, Reynolds and Weber numbers (inertial to gravitational, viscous and surface tension effects respectively).

A large number of studies on this problem have been carried out by [3, Anwar] and other researchers. It has been shown that vortex motion can be correctly reproduced in a Froudian operated model provided that the radial Reynolds number and the Weber number are kept sufficiently high so as to eliminate significant viscous and surface tension effects. Other independent studies have confirmed that adopting such a strategy leads to satisfactory modelling of pump [4, Knauss]. A large amount of research has also concluded that models operated according to Froudian scaling give reliable results when compared to the prototype [5, Hecker].

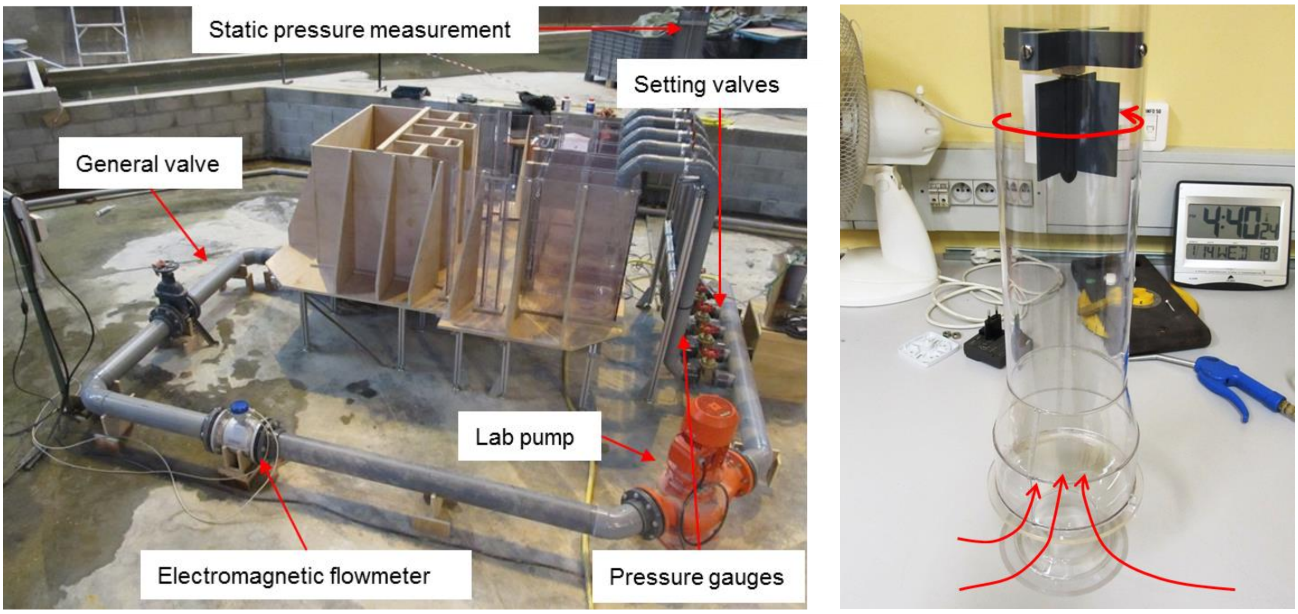

Figure 1: (left) Pictures of a physical scale model; (right) Pump bell mouth equipped with swirl meter

The physical model can aim, as example, at reproducing the formation of vortex of low "type" (see Figure 1), without air entrainment. After study of the geometrical and physical parameters and dimensions of the pumping station, it can be found satisfactory to choose a specific scale factor. Justifications for this choice can be detailed as below:

Scale factor may lead to non-dimensional numbers which are close to values recommended by international standards:

- Viscosity effects: Reynolds number greater than 10^4
- Surface tension effects: Weber number (= $\rho.V^2.D/to$) higher than 120

The aim is not to reproduce strong air core vortices (type 5 or 6), which needs a scale factor of 1/10 or even less, but is to check that no vortex stronger than type 2 (ARL classification – see Figure 2) appears, and a scale factor of 1/20 or even more could be sufficient.
Generally, the model scale is chosen in order to allow accurate flow and levels measurements and in order to allow easy visual observation. The accurate value may be also chosen in order to match with standard pipe diameters.

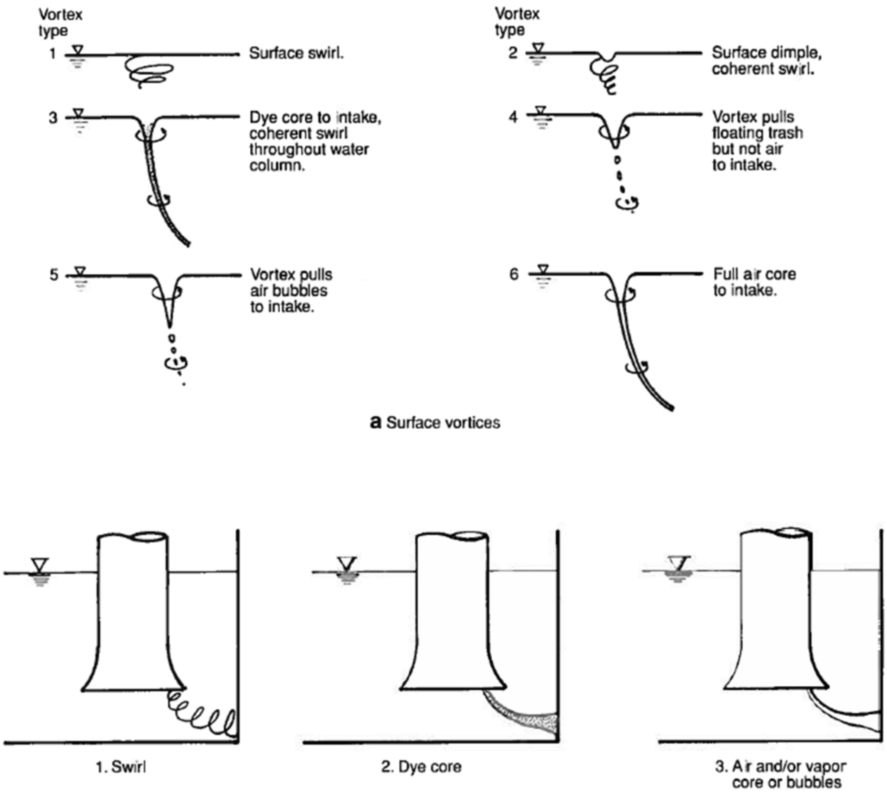

Figure 2: ARL vortex type classification scale developed at the Alden Research Laboratory, Massachusetts, USA

3.2 Numerical model

For the numerical model, according to the geometry but also to the flow and particularly the evolution of the free surface, we normally use two kind of models.
The hydrodynamic opensource Telemac3d model is developed by EDF R&D (Electricité de France - Research and Development department) and a core group of developers. Telemac3d is a three-dimensional model solving the Navier Stokes equations on an unstructured grid; it can take into account of the bottom friction due to the bathymetry, tidal phenomena, wind, rivers flow, radiation stress induced by waves…
The model is built of prismatic elements with a triangular base, feature offering the possibility to use large elements in general areas of the model (as offshore areas) and much smaller elements in the areas of interest, as around the intake and pumping structures. Charge flow cannot be solved with Telemac3d and the free surface must be smooth (without breaking or spashing for example).

For more complexe flows or geometries, like the 3D modeling of the pump lines, we use the opensource OpenFOAM software. It is a CFD software released and developed primarily by OpenCFD Ltd since 2004. OpenFOAM has a large user base in most areas of engineering and science, both for commercial use (the system is operational and competitive) and academics (the system is reliable and validated). It offers a wide range of characteristics to solve multiphysics problems as complex fluid flow involving chemical reactions, turbulence and heat transfer.
Meshes are constructed of mainly cubic cells of different sizes depending on the areas of the model and the necessary accuracy. These cells can become polyhedral in areas near the boundaries and obstacles in the model to adapt to the forms encountered.

4. 3D NUMERICAL MODEL STUDY FOR INTAKE STRUCTURE

For a project along the maroccan coast, we realized a numerical study by using three different models as described hereafter. An upstream forebay model (for the cooling water intake forebay), built using Telemac3D, in order to assess the flow conditions at inlet of each independent pumping line. Input data for this model are sea water level and operating pumps. Outputs are flow conditions at inlet of each pump line. The aim is to check if the upstream condition for the two other models is homogeneous for all configurations (depending only on the sea water level) or if some important discrepancies are identified.

The two other models are dedicated, one to the Cooling Water (CW) pump line model and one to the Flue Gas Desulfurization (FGD) pump line. These two models are built using OpenFOAM, in order to assess the flow into the pump chamber and to analyze the risk of vortex. Input data are sea water level and related unit flow rate, upstream flow condition (provided by forebay Telemac3D model), headlosses at the trash rack and travelling band screen. Outputs are flow conditions within the CW and FGD pumping lines, particularly in the vicinity of the pumping bells.

4.1 Forebay model

The computational domain of the 3D model covers the entire intake structure forebay, delimited by a CW intake outer breakwater and a CW intake counter breakwater (Figure 3). It presents a dredged channel at -7mNGM leading to the intake basin, which floor is -10.5m depth.

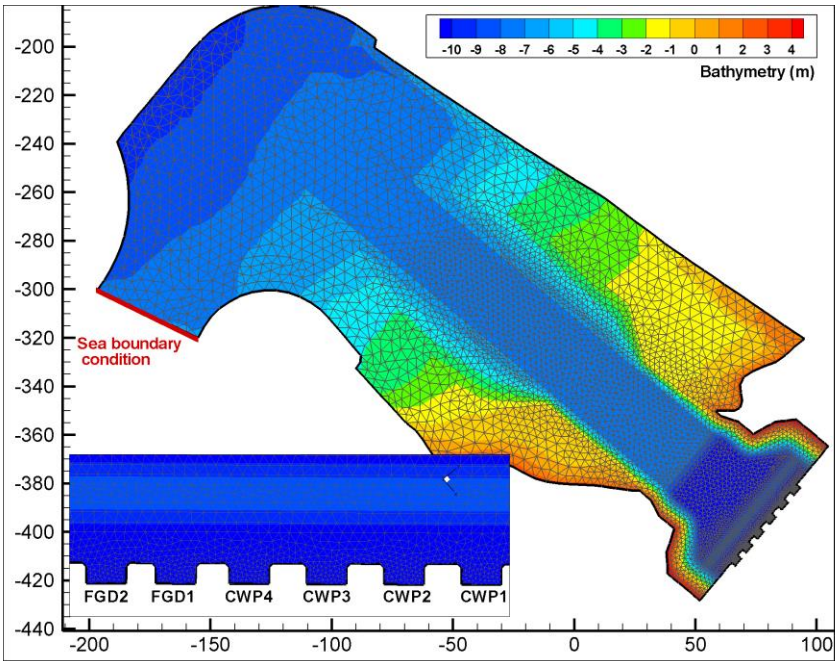

Figure 3: Mesh and bathymetry of the model of the Forebay

The mesh consists of 8 plans over the vertical water column, each containing more than 5,000 calculation nodes. Mesh size is about 5m in the forebay and is particularly refined in the vicinity of the pumps intakes, reaching 0.4m.
Five scenarios are simulated on the forebay Telemac3D model, aiming to cover the main types of operating conditions: normal operating conditions (occurring most of the time), maximal operating conditions (most severe conditions in terms of flow velocity, with high or low water levels), maintenance operating conditions (most severe conditions in terms of inflow velocity balance).
The Figure 4 below shows the flow velocity in the CW intake forebay for Low Water Level and four operating pumps (2 CW and 2 FGD pumps). Velocities in the deepest part of the forebay are low (<20cm/s) but raise to 40cm/s in the pumps intake channels. Flow in CWP4 intake channel is almost homogeneous, whereas flow in CWP1 intake channel mainly comes from right side. Flow in FGD pumps intake channels are slightly unbalanced, flow coming from left side.

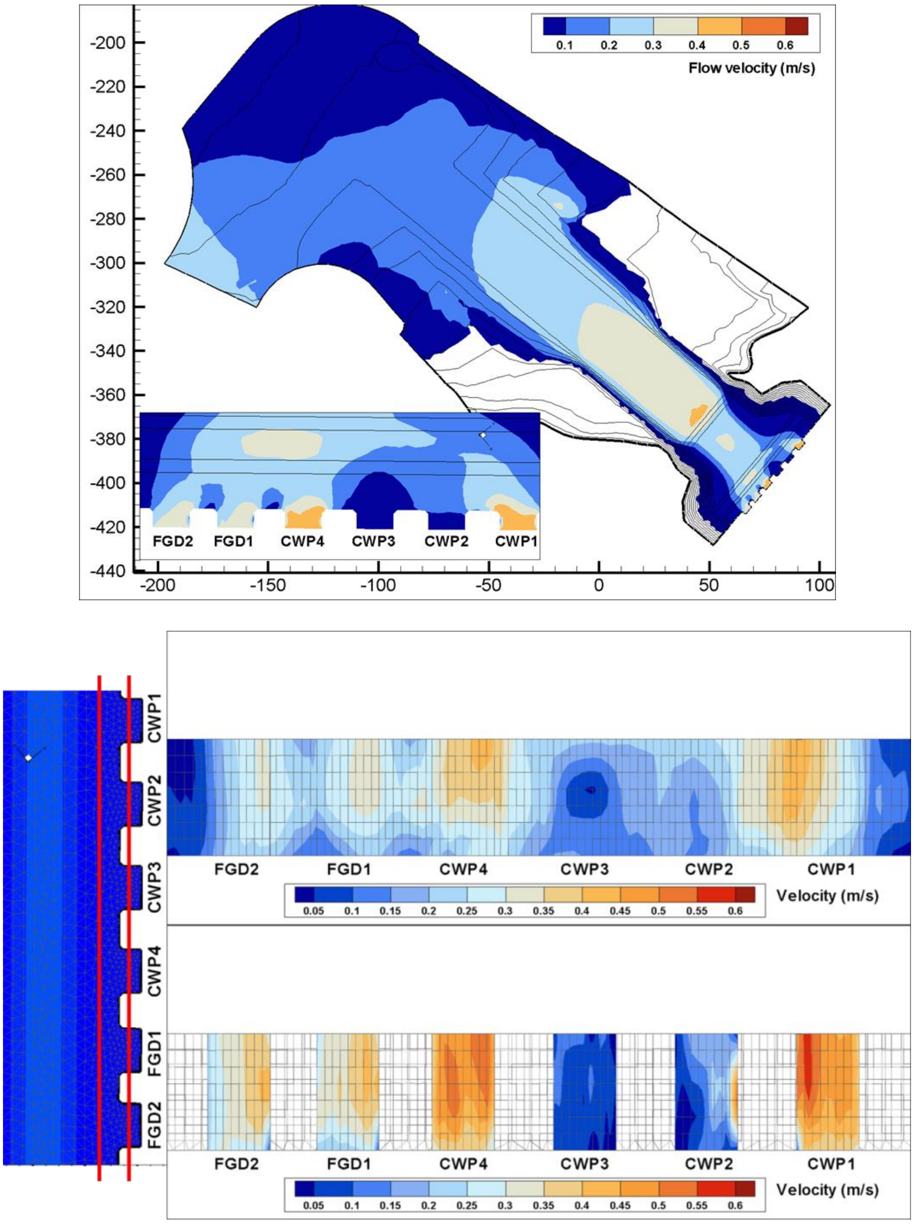

Figure 4: Forebay flow velocity – Vertical mean & Pump lines intake velocity

For this model, we concluded that regardless of water level, velocities in the deepest part of the forebay are lower than 40cm/s and flow patterns do not present any turbulence or skewness problems. The maximum speed criterion in CW Intake forebay (0.5m/s) is fulfilled for all studied cases.

4.2 Cooling Water Pumps line model

The CWP line model represents the entire pumping line. The upstream end of the model geometry is 4 m upstream from the line intake, and the downstream end is located 10 pump pipe diameters downstream from the suction bell.
The opening shape of the travelling band screen has been slightly simplified (rectangular shape instead of trapezoid). However, the openings area is kept identical (i.e. same speed, same head loss). This simplification does not impact the hydraulic downstream behavior since it allows for a good reproduction of the flow patterns coming from the band screen.
The model is discretized with cells, which dimensions go from 20cm at the intake to 5cm (or less) in the vicinity of the suction bell and in the pump pipe. The Figure 5 shows the extent and mesh of the CWP line model

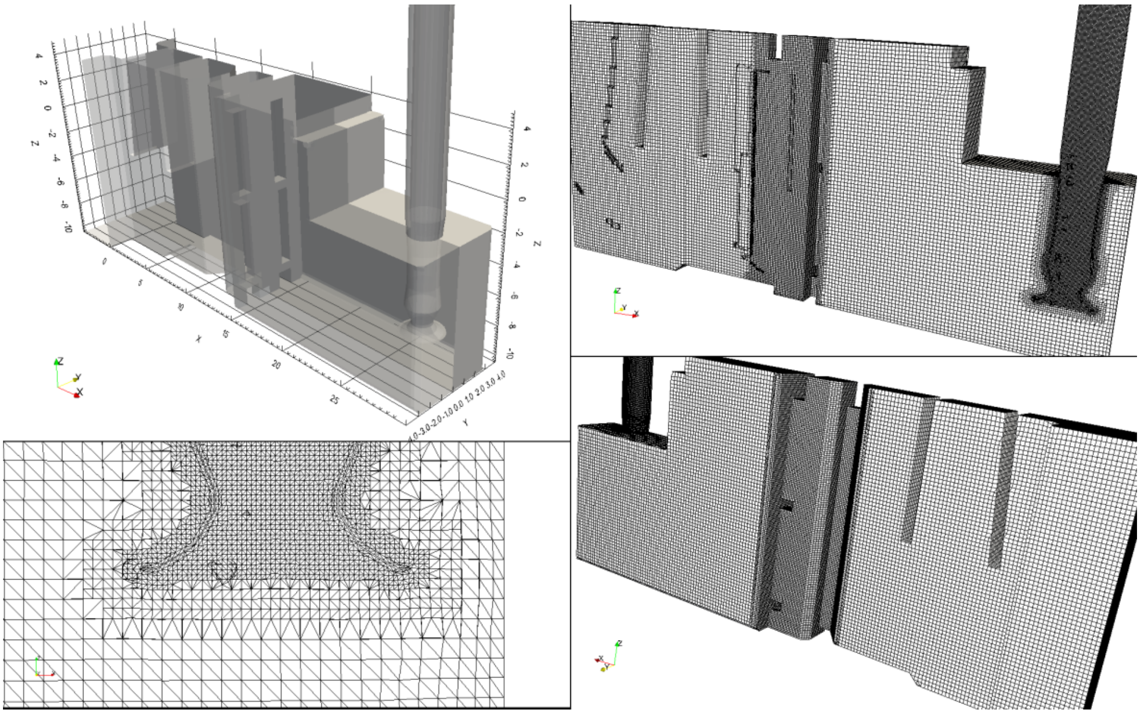

Figure 5: CWP line – Extent and Mesh

Each operating condition tested on both CFD models (CWP and FGD) is analyzed in the same way. Our analysis of flow patterns and acceptability is based on the following elements: velocity fields; swirl angle; Q criterion.
This analysis is made over 10 minutes of simulation time after initialization of the flow, in order to have a sufficient time sample to evaluate the potential transient and permanent phenomena.

Flow velocities and patterns (horizontal and vertical) are observed, in order to check for non-time-dependent standing water area, recirculation area and high speed area. Vertical flow observation is focused on a transversal section located two chamber width upstream the suction bell. Horizontal flows inside the chamber are analyzed observing velocities for two different elevations (EL -2.0m and -7.0m for HWL scenarios, EL -5.5m and -8 for LWL scenarios).

Vortices observation is based on the study of Q criterion. It is a parameter developed to visualize vortex (Hunt, Wray & Moin (1988)). Q is expressed as follow:

$$Q = \frac{1}{2}(\Omega^2 - S^2)$$

Where:
Ω = vorticity tensor (s^{-2})

S = rate-of-strain tensor (s^{-2})
Q criterion allows exploring surface and subsurface vortices in a CFD model. The 10 minutes simulations are analyzed with this Q criterion in order to check vortices which stand all along the simulation.
Each figure presenting Q criterion (by contour of positive value of Q) result is colored by vorticity. If Q is positive and vorticity low, the vortex is weak. The higher vorticity is, the stronger the vortex is.
Finally, and according to ANSI-HI 2012-9.8, the swirl angle corresponds to a ratio between axial and orthoradial speed and can be determined on the scale models thanks to the swirl meters by the following equation :

$$\theta = \tan^{-1}\left(\frac{U_{theta}}{U_Z}\right)$$

Where:
Utheta: tangential velocity = n.2π.D/2 (n = Swirl meter rotation speed (rds/sec))
Uz: mean axial velocity along the pipe = Q/Section (Q=flow, Section=pipe section)
The temporal evolution of this angle is analyzed through two major criteria:

- Its average value over 15s should not exceed 5 degrees;
- The duration in which swirl angle exceed 5 degrees should not exceed 10% of the simulation time (here 10mn).

On the physical scale models, the swirl meter is implemented about 4 diameters downstream from the pipe entrance, with a tip-to-tip vane diameter of 75% of the pipe diameter.
However, when dealing with CFD model, it is not so obvious to calculate such an angle as there are different ways to average spatial velocity. The method developed to calculate CFD swirl angle is the following, keeping in mind that it should remain as closed as possible from the physical model measure :

- Extracting velocities on a regular grid in a section located 4 diameters downstream from the pipe entrance;
- Points from the grid which are not inside the circle of 75% of the pipe diameter are excluded;
- For each point a rotational speed is calculated and an average rotational speed is deduced;
- This average rotational speed multiplied by the pipe radius gives an average tangential velocity from which swirl angle can be calculated.

Results, obtained for one operational condition (low water level, asymmetric inflow), are reported below. A careful observation of the flow pattern and behavior over the time simulated gives the following findings (Figure 6):

- No surface vortex is observed
- Numerous wall, floor and submerged vortices can be seen, lasting up to 1mn and reaching the pump intake.
- Small weak (low vorticity) transient vortices are observed at the vicinity of the pump bell, and near the screen outlets. Most of them last less than 10s.

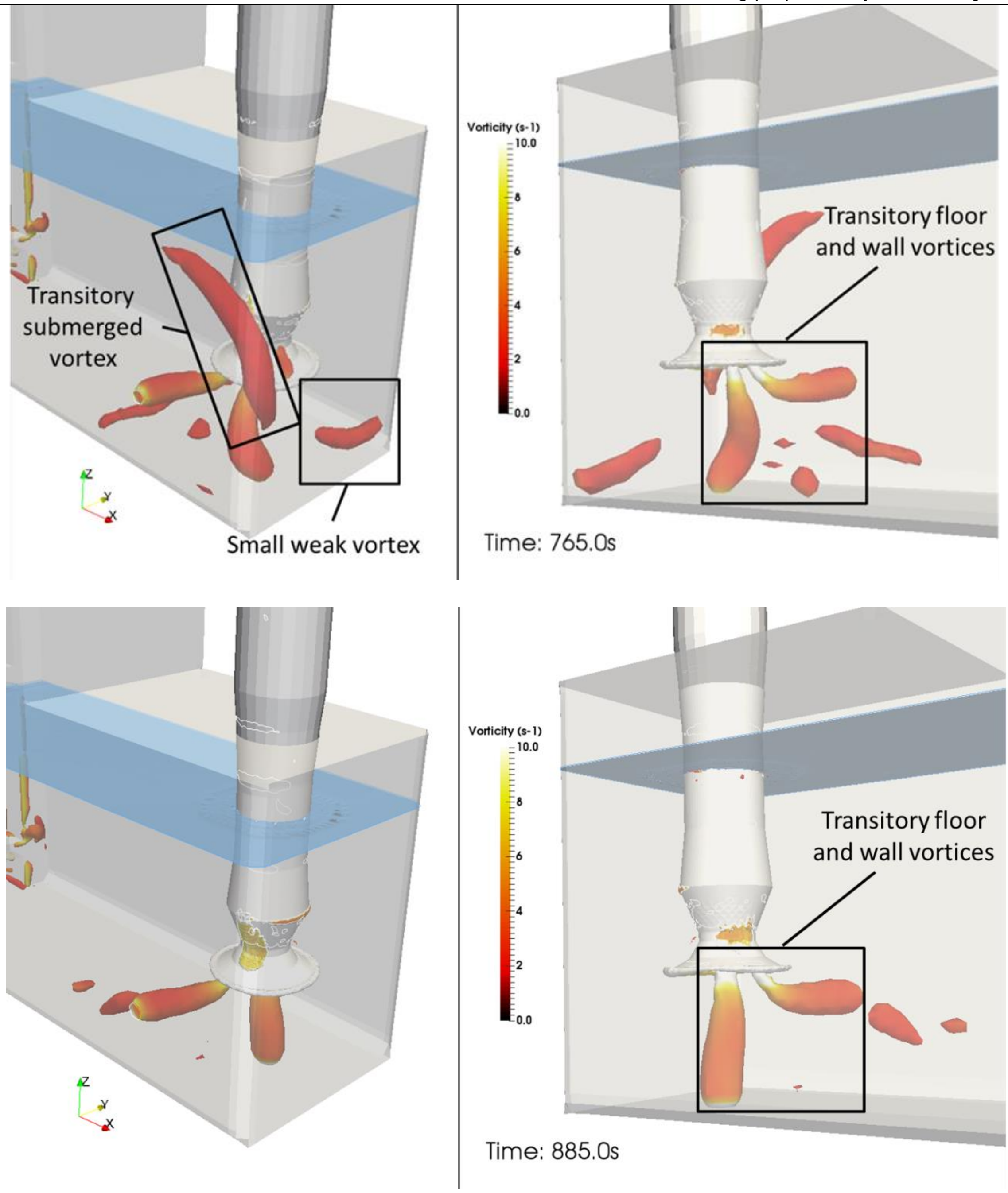

Figure 6: CWP line (Low Water Level – Asymmetric inflow) – Vortices observation

Considering the four conditions simulated on the CWP model, the main results concerning the swirl angle are summarized below:

- The maximum absolute value reached along the 10 minutes simulation is about 3.5°, and the critical value of 5° is never reached.
- The maximum average value is 1.5°.
- The swirl angle remains very low, globally below 1°, for the tested scenario with high water level.
- The scenarios, for which the water level is lower, show higher values of swirl angle, due to higher inflow velocity and therefore higher turbulence.

As a conclusion, the swirl angle criteria are respected for every scenario studied on CW pump line.

Concerning the flow criteria, inhomogeneous flow pattern is observed within the pump chamber, with dead flow areas and velocity fluctuations, due to the narrow outlets of the band screens. However, as the pump chamber is long enough to globally dissipate the water jets from these outlets, flow velocity in the vicinity of the pump does not show important values and can be considered satisfactory.

No surface vortices are observed. However, many subsurface vortices appear during the simulation, that can last up to two minutes. These vortices are almost critical as they reach class 2 (dye core) and although non-permanent, they occur for about 20% of the time.

On the basis of the results, two types of devices were implemented in the CFD model (Figure 7):

- Anti-Vortex Devices (AVD), which are recommended by pump vendor. It is composed of:
    - a longitudinal vertical backwall splitter plate and a longitudinal vertical floor splitter plate, in the axis of the pump chamber,
    - and a "U-shaped" transversal plate, crossing the pump chamber in the transversal axis of the pump.
- Flow distributor, located approximately at half length of the pump chamber, composed by vertical columns supported by middle and bottom slabs.

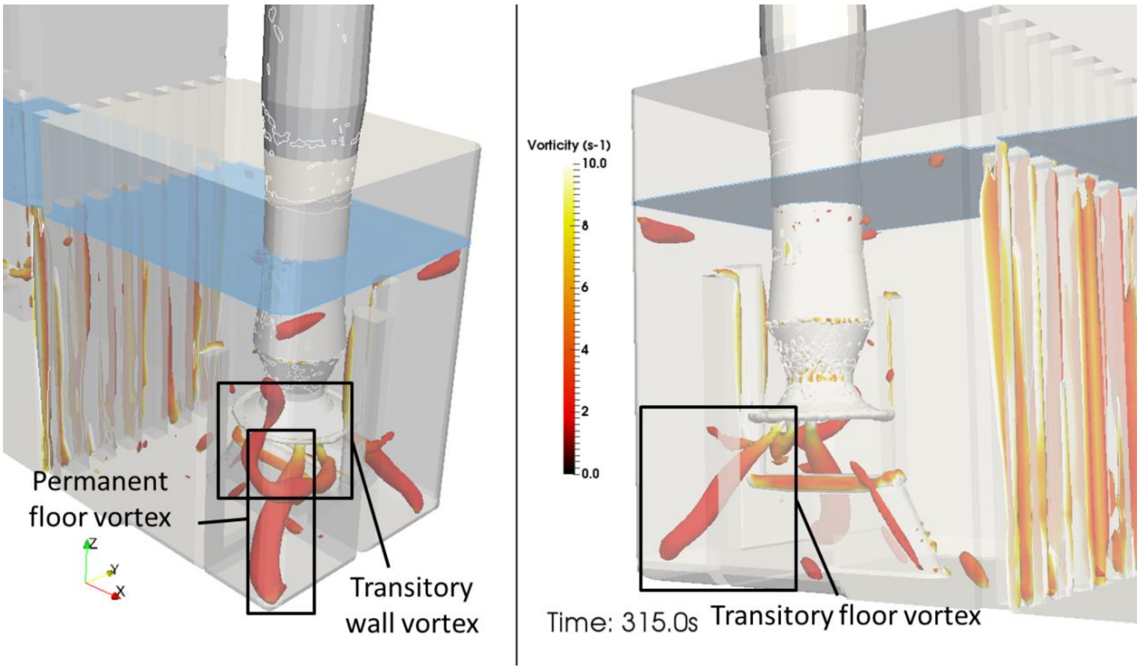

Figure 7: CWP line with AVD & flow distributor – Vortices observation

Addition of AVD and flow distributor highly reduces swirl angle (Figure 8).

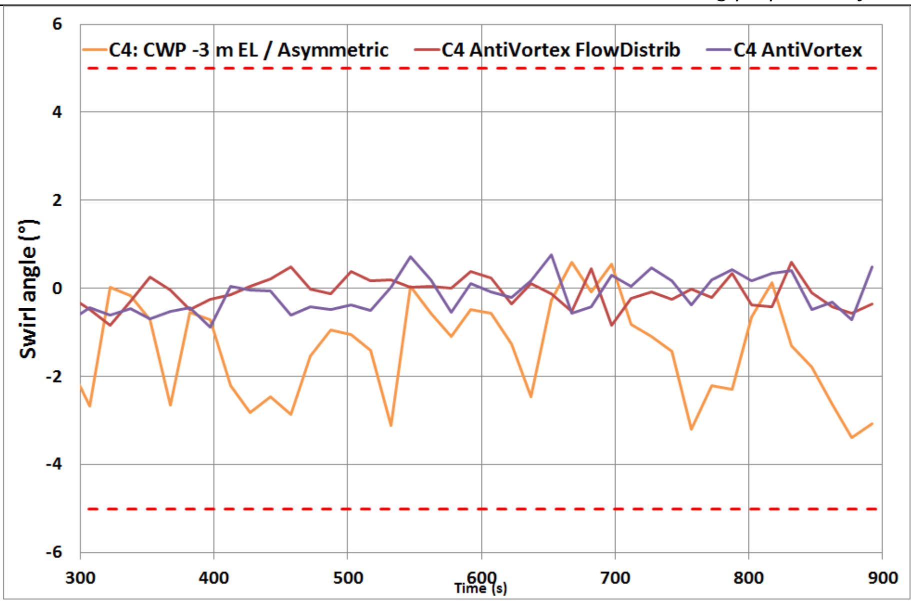

Figure 8: CWP line – Comparison of Swirl Angle – Original Version / AVD / AVD & Flow Distributor

The same analysis on the FGD pump line leads to a permanent floor vortex of class 2 (dye core) for original version. The addition of AVD solved this flow issue.

5. CONCLUSIONS

Widely encountered in drinking water projects, waste water or industrial process, pumping stations must be designed and dimensioned in particular to ensure adequate water supply conditions for pumps, in order to ensure their efficiency and lifetime.
ARTELIA Eau & Environnement regularly realizes design or layout of pumping stations on the base of 3D hydraulic modeling, either physical or numerical. Common practice remains the scaled prototypes of the pumping station in order to simulate a lot of working conditions. Numerical model can be used as one of precondition to perform physical model. If huge modification of the layout is mandatory, the numerical model can be more adaptable than the physical one.
For a project along the maroccan coast, different models built under Telemac3D and OpenFOAM are presented.

We show with this article that three-dimensional numerical modeling now allows understanding more and more phenomena with both a global and exhaustive visualisation of flows and more detailed analysis by computing additional parameters such as vorticity.
As example, and by comparing an initial layout with an adapted one (with AVD and flow distributor) numerical tools are able to quantify the eddies evolution.

REFERENCES AND CITATIONS

[1] Robert L. Sanks, Pumping station design, 2nd Edition, Butterworth Heinemann, 1998.

[2] Hydraulic Institute, ANSI/HI 9.8-1998 – American National Standard for Pump Intake design.

[3] Anwar H.O, Weller, J.A and Amphlett, M.B. 1978. Similarity of Free-Vortex at Horizontal Intake. Journal IAHR, 1978

[4] Knauss, Jost, IAHR Hydraulic Structures Design Manual 1, Swirling flow problems at intakes. A.A Balkema, Rotterdam, 1978, ISBN 90 6191 6437.

[5] Hecker, G.E. 1981. Model-Prototype Comparison of Free Surface Vortices. ASCE Jnl Hyd Div, HY10, 1981