Optimization of a hydrodynamic separator using a multi-scale computational fluid dynamics approach

Vivien Schmitt[1] , Matthieu Dufresne[1], José Vazquez[1], Martin Fischer[1], Antoine Morin[2]

[1] Fluid and Solid Mechanics Institute (CNRS, University of Strasbourg, ENGEES, INSA), 2 rue Boussingault, 67000 Strasbourg, France, vivien.schmitt@engees.unistra.fr, [2] Hydroconcept, ZA Trappes Elancourt, 46 Avenue des frères Lumière, 78190 Trappes, France

EXTENDED ABSTRACT

Hydrodynamic separators are structures currently used to remove sediments from water in order to protect receiving watercourses. Several types of hydrodynamic separators exist, each having its own operating process (USEPA, 1999). The CycloneSep® works on the tangential separation mechanism along a grid. After passing through the grid, the effluent reaches the central part of the device and is then discharged into the environment (Figure 1). The objective of this structure is to maximize the retention of particles in the part located around the grid. The main constraint is to avoid the clogging of the grid.

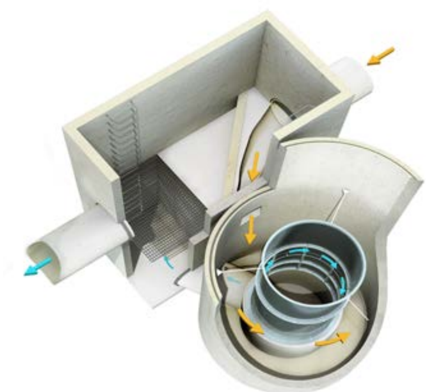

Figure 1. The CycloneSep®

The aim of this study is to optimize the device by using Computational Fluid Dynamics (CFD). Two approaches can be followed. A first solution is to modify the shape of the grid. The idea is to observe the pressure effects by testing different characteristics (size, angle and shape of the orifice). A second solution is to increase the velocity near the grid with the modification of the shape of the device, and consequently the hydrodynamic behaviour of the device.

Because of the complexity of the geometry, a large number of cells would be necessary to model the complete device. To overcome this difficulty, a multi-scale approach is used (Figure 2).

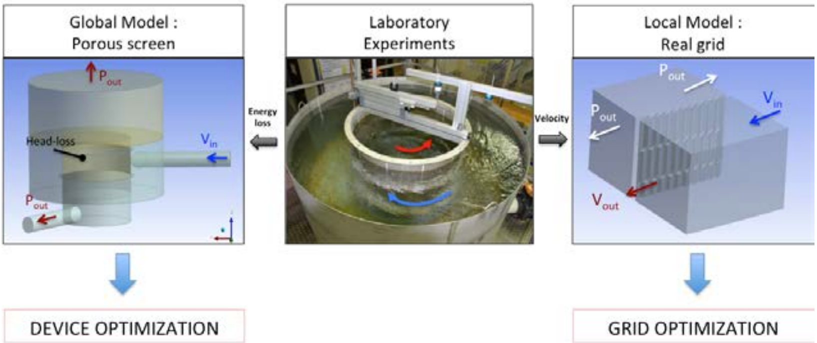

Figure 2. The multi-scale approach for the optimization of the CycloneSep® separator

The local model is built in order to simulate the hydrodynamic in the vicinity of the grid. This model will be used to investigate different shapes of the grid (angle, aperture sizes and type of the grid). The results show that the effects of pressure near the grid can be changed by using different parameters. The difference of pressure between the edge and the middle of the aperture will increase with an expanded metal grid. It is also available by decreasing the angle of the metal strip and by decreasing the aperture size.

The global model is built in order to simulate the main hydrodynamic structures of the tank. The global model consists in the whole geometry but the grid is modelled using a porous wall, following a conceptual approach. This approach introduces a source term that reproduces the energy loss of the grid. By comparing with experimental results, the CFD model globally reproduces the velocity around the grid. The highest velocities are encountered near the grid (0.73 m/s experimentally and 0.84 numerically). Near the external wall, the velocity is much smaller: about 0.60 m/s experimentally and 0.62 m/s numerically. The global model is also used to optimize the hydrodynamic behaviour. The installation of deflectors is investigated to observe their influence in the aim of increasing the velocities near the grid and also to favour the non-clogging effects. Concerning the use of six horizontal deflectors, the results show that the discharge in circulation decreases. The orientation of the deflectors will favour the flow passage in the screen. In contrary, the installation of a vertical deflector (disk plate) under the grid did not decrease the discharge or the velocity. The advantage of this configuration concerns the velocity under the grid (almost 0.3 m/s), that probably avoids resuspension of deposited waste on the bed of the structure.

This study is the basis for a wider prospective work about the optimization of the CycloneSep.

References

Office of Water and US Environmental Protection Agency. (1999). *Stormwater Technology fact sheet hydrodynamic separator*.