STORMWATER MANAGEMENT SYSTEMS DESIGN WITH A LITTLE HELP FROM CFD

By Geneviève Kenny, P. Eng., Natasha Lee E.I.T, Bernardo Majano, P. Eng., R.V. Anderson Associated Limited

In the evolving field of complex stormwater management, large volumes of water often have to be conveyed and managed in unique ways that challenge the imagination of designers. Many of these challenges can be encountered in the design of stormwater management infrastructure in a high-density urban area such as Toronto. In many cases, the proposed infrastructure must meet its intended purpose while respecting site and space constraints. Conventional stormwater management solutions such as shallow and large wet or dry ponds are space prohibitive in high cost real-estate areas. Innovative usage of tunnels for conveyance, tunnel shafts for storage and high rate treatment systems are required to achieve design team and owner goals. Many of these systems must be designed with less than ideal hydraulic operating conditions and other constraints that require a careful and thorough hydraulic analysis to ensure proper performance.

Computational Fluid Dynamics (CFD) is the application and approximate solution of fluid dynamics equations to predict the behaviour of a fluid within a specified boundary. Typically, fluid parameters such as pressure, velocity, flux and turbulence are calculated in the model. CFD modelling is becoming an increasingly popular tool in the infrastructure design industry, and there are many areas where CFD models can be employed to refine design, and optimize existing configurations.

This article will highlight three examples where CFD modelling and analysis was used as a very important tool to achieve successful designs of stormwater management infrastructure. In these aforementioned situations, conventional design analysis approaches or best-practice design guidelines and standards would have not been able to provide concrete answers.

Storm Sewer Vortex Inlet

In this example, CFD analysis was utilized in the design of a vortex inlet for a new stormwater sewer located in Toronto, Ontario, Canada. A vortex inlet was required as a transition piece between a 2-metre by 2-metre concrete channel and a drop shaft with a height of 23 metres that feeds a stormwater conveyance tunnel which transports stormwater for treatment.

The purpose of the vortex inlet was to direct the water into a swirl pattern, circling along the wall of the drop shaft preventing a ‘water fall’ effect, where water would plummet to the base of the drop shaft causing erosion. Because any erosion at the base of the drop shaft would negatively impact the operation of the flushing system in the tunnel, it was critical that the vortex inlet perform correctly.

These unique design constraints required hydraulic modelling using a trial and error approach with several pilot scale models. Following the testing of the pilot scale models, a CFD model was developed to verify the design. This CFD model was initially validated by comparing the results with a physical model and, once validated, the CFD model was run at multiple flow rates to evaluate the performance of the vortex inlet.

The vortex inlet model mesh is shown in **Figure 1**, with points indicating the corners of each mesh cell. This model geometry was created using Salome-Meca, an open source CFD geometry program and meshed using SimpleFoam Wizard, an open source meshing program. The CFD simulation was performed with the SimpleFoam algorithm using the K-Epsilon turbulence model, within the OpenFoam program.

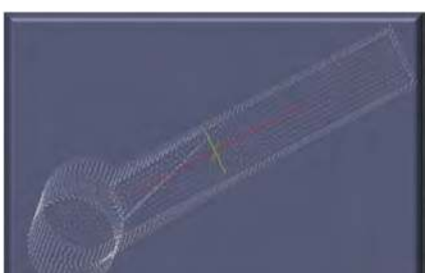


Figure 1 - Vortex inlet model.

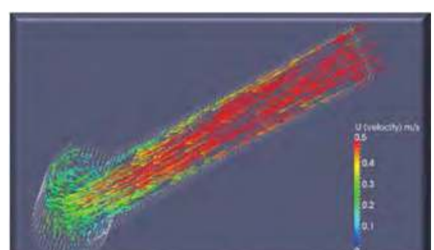


Figure 2 - CFD simulation results at average annual storm event flow.

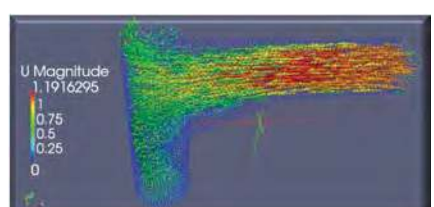
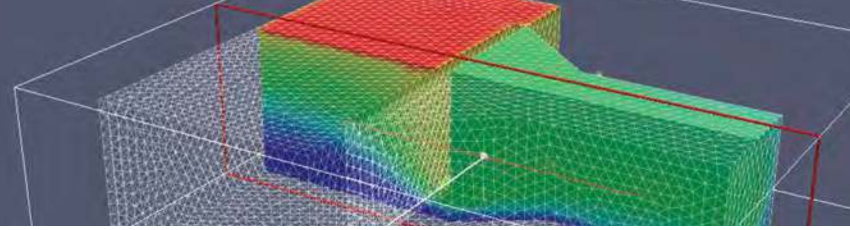


Figure 3 - CFD simulation results for 2 year storm flow.

Click HERE to return to Table of contents



PART 1: MODELLING

"THE CFD MODEL THAT WAS DEVELOPED FOR THE VORTEX INLET, ALLOWED THE DESIGNERS TO TEST THE MODEL GEOMETRY AT VARYING FLOWRATES, AND ACHIEVE GREATER COMFORT WITH THE DESIGN."

Figures 2 and **3** show the CFD simulations results of the vortex inlet model at varying flow rates. Given that the vortex inlet was to be located in a storm sewer, there was high degree of variability in the flow. Running CFD simulations for the wide range of flow rates allowed for the design team to gain increased confidence in the design. A weir incorporated into the sloped portion of the vortex inlet achieved a swirl pattern during the average storm events, yet the weir height was such that it did not inhibit the swirl pattern for the higher storm flows.

The CFD model that was developed for the vortex inlet allowed the designers to test the model geometry at varying flowrates, and achieve greater comfort with the design. Another benefit of the CFD model was that small design changes, proposed during the construction process could be easily tested using the CFD model to ensure that there were no adverse impacts.

Stormwater Shaft Sump Design

A 25-meter deep shaft originally used for tunnel construction was also designed to be converted into a vertical stormwater attenuation tank. Initially constructed in 2012, the original design consisted of using two pumps (each rated for 200 L/s), with the sump designed accordingly, including space for a grit basket to allow operators to easily remove accumulated grit as part of their operation and maintenance duties. During the detailed design phase in 2014, the concept was revised for ease of removal and maintenance by reducing the size and weight of the pumping system with the use of three smaller pumps, each rated for 133 L/s.

A uniform and steady inlet flow to the pump is recommended for optimal pump operation as there are several hydraulic phenomena that can adversely impact the performance of pumps. Therefore, mitigation measures should be incorporated into the design of the intake structure. The adverse hydraulic phenomena are the following:

- non-uniform spatial distribution of velocity at the entrance to the pump impeller;
- pre-rotation (swirl) of flow entering the pump impeller;
- temporal variations in velocity and swirl;
- submerged vortices;
- free-surface vortices; and
- entrained air or gas bubbles.

Hydraulic Institute (HI) guidelines provide recommended sump design parameter values for approach velocities, inlet bell velocities, and cross flow velocities. In addition to the HI guidelines, the swirl angle and visual vortices are commonly-used metrics to interpret CFD simulations.

Swirl angle, *a*, is commonly used to quantify the amount of swirl at the pump inlet. It is defined as *a*=arctan(vt/va) where vt is the tangential velocity and va is the axial velocity. The swirl angle computed in a CFD model is averaged over the cross-section. The maximum recommended value of the swirl angle is about 5-7 degrees for trouble-free operation of the pumps.

Occurrence of vortices in CFD models can be identified using velocity vectors (either total or its components) which are used to visualize vortices.

Because the shaft and sump have already been constructed and were designed for a two-pump configuration, any modifications to the sump (e.g., extending any of its dimensions) or shaft other than addition of baffle walls or anti-vortex baffles would have a high cost. In order to retrofit the sump to operate adequately with the three-pump arrangement, a baffle system has been proposed in order to distribute the flow more evenly to the three pumps.

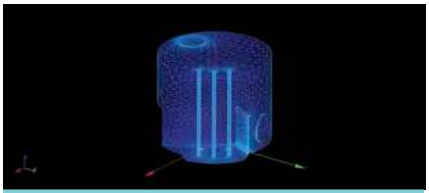


Figure 4 - Model geometry for the base of the stormwater shaft.

CFD modelling was carried out to validate the effectiveness of the preliminary sump design which included the addition of a baffle wall, in providing acceptable hydraulic conditions at the pump inlets. A model geometry, depicted in **Figure 4** was created for the base of the shaft, sump and three pumps. The CFD simulation was performed using the SimpleFoam algorithm and a K-Epsilon turbulence model.

Various scenarios with different numbers of pumps in operation were modelled. The results from the model were used to visualize the occurrence of vortices as well as estimate swirl angles.

The velocity profiles in the pump intake channels demonstrated approach velocities that exceeded recommended HI parameters; however the general CFD results did not indicate any adverse hydraulic phenomena, therefore these increased approach velocities should not be of concern. Swirl angles were also analyzed carefully at the pumps' inlets and they were determined to be within acceptable values.

The preliminary results from the CFD modelling and analysis indicate that the proposed baffle wall system will allow the system to operate adequately with the three-pump arrangement. The ultimate solution will have to be reviewed and discussed with the successful pump supplier.

Click HERE to return to Table of contents

"AS PART OF THE OVERALL BUILDING DESIGN PROCESS, THE OWNER ALSO ASKED THE DESIGN TEAM TO REVIEW SUSTAINABLE AND INNOVATIVE DESIGN CONCEPTS."

Lake Water Pre-Cooling System

For the same stormwater treatment system described in the previous example, the client requested the team to provide dehumidification of ventilation air inside the treatment facility in an effort to reduce the harmful effects, such as corrosion, that high humidity environments have on mechanical equipment. As part of the overall building design process, the owner also asked the design team to review sustainable and innovative design concepts. Since the shaft is connected to the lake, it was suggested that lake water could be used to provide partial dehumidification of outside air during the summer. This would provide significant building energy savings. Ideally, the water should be extracted from the deep lake portions; however, for a relatively small system such as this, the capital cost would be fairly high with a lengthy return on investment.

A more cost-effective alternative is to extract the lake water dehumidification system supply (LWDH supply water) from the storage shaft portion connected to the Lake. The LWDS supply can be disposed of through the building's plumbing sewer system, but this requires a permit to take water. A better alternative would be to return the hot lake water dehumidification system return (LWDH return water) into the shaft. However, there is an obvious thermodynamic constraint. If the water is returned to the shaft, LWDH supply water will also be mixed with the hot LWDH return water, thus reducing the performance of the system. However, the cooler water from the facility's outfall (outfall return water) will also be discharged in this area and mixed with the other two flows.

The outfall return water temperature will be cooler than the LWDS return water. This is a key design parameter for proper operation of the system, and its thermodynamic analysis is described as follows: during any given storm event, the theoretical rainfall temperature value is very close to the air wet bulb temperature and cannot be used to dehumidify the air. However, this rainfall will become runoff in the catchment areas. The collected stormwater in the catchment areas is transported to the management facility through deep conveyance tunnel sewers and into the storage shaft. During most storm events, the concentration time of the runoff is fairly high, and other than for large storm events, the water is stored in the deep shaft for a significant period of time; thus the water would cool down significantly to a point closer to the ground temperature than the theoretical day's wet bulb temperature. By being cooler than the air's wet bulb temperature, the water in the shaft and through the treatment (outfall return water) could be used for partial dehumidification as long as the cooling mixing effect is sufficient to allow the system to absorb all coil-extracted heat from the LWDS return water. Thus, a proper understanding of this mixing effect is a key design parameter to ensure proper hydraulic and thermodynamic operation of the system.

A CFD analysis of the proposed system was used to obtain a more detailed analysis of the temperature mixing effect. The purpose of the model was to determine at what depth the LWDS supply water must be extracted from the shaft in order for the system to have enough capacity to absorb the LWDS return water extracted heat from the coils. In addition, the model would also allow the team to determine overall mixed temperature within the shaft.

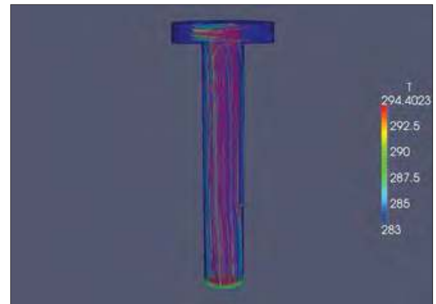


Figure 5 - CFD model results of temperature mixing effect in stormwater shaft.

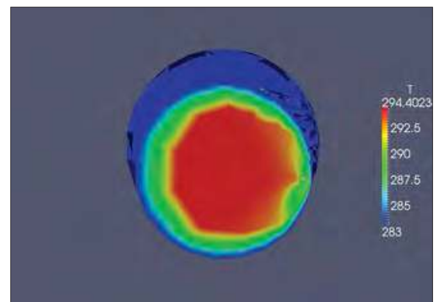


Figure 6 - Cross sectional view of temperature mixing results.

Results from the model indicate that if the LWDS supply water pipe is located approximately 10 m below the water surface level, the system will operate in steady state. Although the LDWS supply can be placed further down in the shaft, placing it too close to the bottom of the shaft can make it susceptible to extracting debris or solids from the bottom. Logically, both the LWDS return water and outfall return water pipes would have to be placed as close to the surface and away from the LWDS supply water pipes as practically possible.

Conclusions

The three design cases we have presented demonstrate that CFD modelling has a variety of applications in stormwater process designs. CFD analysis and modelling is a valuable complementary design tool that allows the design engineer to obtain a better understanding of fluid dynamics phenomena such as velocity profiles and temperature mixing.

Click HERE to return to Table of contents