3D Numerical Modeling of a Supercritical Intake With a Flow Diversion Barrier

L. Chang  
*Hong Kong University of Science and Technology, lu.chang.hk@gmail.com*

S.N. Chan  
*Hong Kong University of Science and Technology, treechansn@ust.hk*

Joseph H.W. Lee  
*Hong Kong University of Science and Technology, jhwlee@ust.hk*

Follow this and additional works at: [https://digitalcommons.usu.edu/ishs](https://digitalcommons.usu.edu/ishs)

**Recommended Citation**


This Event is brought to you for free and open access by the Conferences and Events at DigitalCommons@USU. It has been accepted for inclusion in International Symposium on Hydraulic Structures by an authorized administrator of DigitalCommons@USU. For more information, please contact digitalcommons@usu.edu.
3D Numerical Modeling of a Supercritical Intake with a Flow Diversion Barrier

L. Chang¹, S.N. Chan¹,² & J.H.W. Lee¹
¹ Department of Civil and Environmental Engineering, Hong Kong University of Science and Technology, Hong Kong, China
² Institute for Advanced Study, Hong Kong University of Science and Technology, Hong Kong, China
E-mail: treechansn@ust.hk

Abstract: The Hong Kong West Drainage Tunnel (HKWDT) system consists of 34 storm water intake structures designed to intercept storm water runoff in upland catchments. Each intake intercepts and diverts the upstream supercritical flow into a bottom rack chamber connected to a supercritical vortex drop. To satisfy the minimum environmental flow requirement, a small flow is intercepted by an inclined barrier across the main channel and conveyed to the downstream drainage system through a 300mm wide low flow channel (LFC) along one side of the intake. Observations during rainstorms suggest that the storm water runoff being conveyed via the low flow channel might be more than originally designed. The hydraulic performance of the intake structure is assessed via three-dimensional Computational Fluid Dynamic (CFD) modelling using the Volume-of-Fluid (VOF) method. The predicted un-intercepted flow compares well with experimental measurements on a 1:12 undistorted model. The CFD computations show that in a typical rainstorm event, the supercritical inflow impinges on the barrier and is deflected upwards, resulting in complex cross-currents between the main stream and the LFC. It is found that the ratio of un-intercepted flow to the total flow decreases with increasing inflow. Supported by field observations the CFD model predicts that the low flow channel flow is significantly higher than designed; the effect of the barrier is less significant for higher flows.

Keywords: Intake structure, supercritical flow, transverse barrier, storm water interception, volume-of-fluid, CFD.

1. Introduction

Located in the subtropical region, the densely populated and highly urbanized coastal city of Hong Kong has an average annual rainfall of over 2000 mm, concentrated in the summer season (May to September). Global climate change has resulted in more frequent extreme rainfall events. Together, with sea levels rising, the low-lying urban areas of Hong Kong Island have become more vulnerable to flooding due to the increased runoff from its steep hinterland. As traditional methods of upgrading existing drainage systems (such as widening culverts) are not practical or effective in the congested urban areas, an innovative upstream diversion approach—the Hong Kong West Drainage Tunnel (HKWDT) system—is designed to enhance the flood protection standard for the urban and business districts. Completed in 2012, the HKWDT system consists of 34 intake structures to intercept storm water runoff from the steep upstream hinterland in the hilly areas of Hong Kong Island to a drainage tunnel for discharge to the sea (DSD 2003). Each intake structure is designed to intercept flow into a bottom rack chamber connected to a supercritical vortex drop shaft (average height 82 m), leading to the underground drainage tunnel. During dry weather, a small environmental flow is conveyed through a 300 mm wide low flow channel (LFC) next to the intake structure to the channel downstream of the intake (Fig. 1). The dry weather flow is diverted to the LFC by a 10cm high inclined barrier across the main stream, with a design capacity of 20 L/s. However, when the intakes are in operation during rainstorms, significant upward deflection and spilling of the flow is observed due to the interaction of the supercritical inflow with the inclined barrier, suggesting that the storm water runoff conveyed via the LFC may be more than originally designed.

The flow interception barrier significantly affects the performance of the intake structure. Supercritical flow over obstacles like barriers and sills has been studied for the design of stilling basins for spillways. For example, Hager et al. (1986) studied experimentally and theoretically the incipient hydraulic jump caused by a horizontal supercritical flow impinging on a vertical sill perpendicular to a rectangular channel. Ohtsu et al. (1996) examined the flow features of a horizontal supercritical flow impinging on a vertical sill and developed a criteria of hydraulic jump formation based on the height of the sill and tailwater level. These studies were conducted in a horizontal channel with a barrier or sill perpendicular to the flow direction. Demetriou (2009) carried out experiments to investigate the flow characteristics downstream of a sill in a sloped open channel, but the sill was also perpendicular to the flow direction. It is observed that an inclined flow diversion barrier could result in a change of flow direction.
With rapid advances in computational power, three-dimensional (3D) Computational Fluid Dynamics (3D CFD) modeling has become a useful tool for the design and performance assessment of hydraulic structures. The Volume-of-fluid (VOF) method (Hirt and Nichols 1981) had been used extensively for studying spillways and hydraulic jumps (Bombardelli et al. 2011; Valero et al. 2016; Bayon et al. 2016; Valero and Garcia-Bartual 2016). Nevertheless, there have been hitherto very few related studies on numerical simulation of supercritical flow over an obstacle.

The supercritical flow over a 45-degree inclined bottom-mounted barrier is investigated for the first time. The flow interception performance for a typical storm water intake in the HKWDT system is studied using a 3D CFD model. The model set-up is first described. The predicted flow field and depth across the entire intake are then presented and compared with experimental data. The results are discussed in relation to the flow interception performance.

2. Numerical Model

2.1. Governing Equations

The VOF method computes an air-water, two-phase flow by solving the continuity equation of water phases and the shared momentum equation of the two phases (Hirt and Nichols 1981). The continuity equation of the water phase is:

$$\frac{∂}{∂t}(\alpha_w \rho_w) + \nabla \cdot (\alpha_w \rho_w \mathbf{U}) = 0$$  \hspace{1cm} (1)

where $\alpha_w$ is the volume fraction of water phase, $\rho_w = 998.2$ kg/m$^3$ is water density, and $\mathbf{U}$ is the phase averaged velocity of air and water mixture. In one computational cell, the total volume fraction of air and water is equal to unity, i.e. $\alpha_w + \alpha_a = 1$, where $\alpha_a$ is the volume fraction of air.

The two phases share the same momentum equation, which can be written as (Patankar 1980; Rodi 1980):

$$\frac{∂}{∂t}(\rho \mathbf{U}) + \nabla \cdot (\rho \mathbf{UU}) = -\nabla P + \nabla \cdot [\mu_t (\nabla \mathbf{U} + (\nabla \mathbf{U})^T)] + \rho \mathbf{g}$$  \hspace{1cm} (2)

where $\rho = (1 - \alpha_w) \rho_a + \alpha_w \rho_w$ is the phase-averaged density, $P$ is the pressure, $\rho_a$ is the air density = 1.225 kg/m$^3$, and $g = (0,0,-9.81)$ m/s$^2$ is the gravitational acceleration. The turbulent dynamic viscosity $\mu_t$ is predicted using a standard $k-\varepsilon$ turbulence model (Rodi, 1980), where $k$ is the turbulent kinetic energy and $\varepsilon$ is its dissipation rate.

The governing equations are solved using the CFD software ANSYS FLUENT (ANSYS 2013). Pressure-based implicit solver is chosen to solve the unsteady governing equations. The PISO algorithm (Issa 1986) is used for pressure-velocity coupling, and the skewness correction and neighbor correction are set to be 0 and 1 respectively. The PRESTO method and the modified HRIC scheme are used for the discretization of pressure terms and volume fraction equation respectively (ANSYS 2013). The second-order upwind and first-order upwind discretization schemes are used for momentum and turbulence parameters respectively.

2.2. Model Geometry and Grid

A representative intake in the HKWDT system is studied as a generic design for assessing the interception capacity and flow characteristics. A physical model of 1:12 undistorted Froude scale is designed and constructed according to as-built drawings of the intake, covering the approach channel, the inclined barrier, the low flow channel (LFC), and the bottom rack. Experiments have been performed on a number of flow scenarios corresponding to different rainstorm return periods. The un-intercepted flow for each scenario is measured. Detailed model set-up and measurement techniques can be found in Lee et al. (2017).

The CFD model is developed according to the exact dimensions of the physical model experiment (Fig. 1a). The approach channel consists of two parts: a 1000 mm long channel with a triangular bottom, which models the natural channel upstream of the intake, and a 167 mm long rectangular approach channel with a slope of 1:5.5. At the end of the approach channel, a 45 degree inclined transverse barrier of height $h = 8$ mm spans across the channel until it meets the LFC of 25 mm width, located to the left of the bottom rack (viewing downstream). The majority of the incoming flow passes through the bottom rack and plunges into the chamber below (the intercepted flow), while a portion of the inflow passes through the LFC onto downstream (the un-intercepted flow). For this supercritical flow diversion
problem, the bottom rack bars and details of the connection to the vortex intake are not modelled as they can be shown to have insignificant influence on the flow interception.

The geometry and computational mesh of the model is developed using the GAMBIT software. The mesh consists of a total number of 127,668 cells (Fig. 1b). The barrier is discretized by 30 grids cells along its length and 3 grid cells across the width with a minimum grid size less than 3mm. Grid sizes gradually increase from the channel bed and the barrier region to the top atmospheric boundary and the upstream/downstream. The grid resolution is similar to that of a CFD study on a supercritical bottom rack intake (Chan and Lee 2018).

2.3. Boundary and Initial Conditions

The water flow rate and depth is imposed at the upstream boundary. The inflow water depth is estimated using a head-discharge relationship determined from the physical model experiments (Lee et al. 2017). Turbulence quantities at the inlet are estimated as

\[ k = \frac{3}{2} (u_{avg} I)^2, \quad \varepsilon = C_{\mu} \frac{2}{3} \frac{k^3}{l} \]  

(Rodi 1980), where \( I = 0.16Re^{-1/8} \) is the turbulent intensity of fully developed wall bounded flow; \( Re = u_{avg}D / v \) is the Reynolds number of the inflow, \( u_{avg} \) is the mean inflow velocity, \( l = 0.07D \) is a turbulence length scale, and \( D \) is the water depth at the inlet; and \( C_{\mu} = 0.09 \) is a semi-empirical constant for eddy viscosity in the standard turbulence model. Zero gauge pressure is specified for the top atmospheric boundary and the two outlet boundaries. All the solid boundaries are taken as no-slip walls with the roughness height of 0.05mm and roughness constant of 0.5.

All the simulations begin at time zero with a dry bed, and the flow develops from the inlet. The maximum Courant number is fixed to 1, and the minimum time step is 0.0001s. The initial velocity is set to be zero, and the turbulent kinetic energy and turbulent dissipation rate are set to be 1% of the values at the inflow boundary. The model simulations are carried out for six discharge conditions, ranging from 4L/s for typical rainstorm condition to 41.8L/s for a flood with a 200-year return period with climate change (Fig. 3).

3. Results and Discussion

3.1. Flow Profile

The CFD model predictions of the depth profile are validated against experimental data. Water depth above the barrier is measured using a point gauge in the laboratory model. The comparison of the numerical prediction of water level above the interception barrier (taken as 50% air volume fraction) with data is shown in Fig. 2 for two typical flow
scenarios. It can be seen that the depth of water decreases gradually with distance from the upstream end of the barrier towards the LFC. The CFD model prediction shows the same trend as the data, although the observed water levels in all cases are somewhat higher than the numerical prediction near the upper side-wall area, which may be due to the uncertainty in wall friction.

3.2. Un-intercepted Flow

The CFD predicted un-intercepted flow can be determined from the total water flow rate at the downstream end of the LFC, while the intercepted flow is obtained from the flow rate that passes over the barrier into the dropshaft (Fig.1). The predicted un-intercepted flow of the six typical scenarios are shown in Fig. 3. The predicted un-intercepted flow compares well with the laboratory experiment results, although it is in general smaller than the observed values. This may be attributed to the exclusion of bottom rack bars in the CFD model; the bar racks present a flow resistance and interaction between the flow above the racks and the LFC flow. A small portion of flow can also be conveyed downstream on top of the rack bars, in reality. It can be seen that the un-intercepted discharge increases with the increasing total inflow but not in proportion with the total discharge. For example, about 13% of flow is un-intercepted for \( Q = 4 \) L/s; however, the ratio of un-intercepted flow decreases to about 5% for a high inflow of \( Q = 41.8 \) L/s. When scaled to prototype flows, the computed un-intercepted flow ranges from 255 to 900 L/s, which is much larger than the designed maximum of 20 L/s. This confirms the field observations that the storm water conveyed to the downstream drainage system is more than originally designed.

3.3. Flow Field

The majority of the supercritical flow from the upstream approach channel comes across the barrier before going into the underground tunnel, while a small portion of flow goes directly to the LFC near the side. Fig. 4 shows the top views of the flow field for \( Q = 13.9 \) L/s as a typical example. It can be seen that flow is rather uniform along the approach channel before it encounters the barrier which leads to a flow direction change near the channel bottom (Fig. 4a) but has little effect on the flow direction near the surface (Fig. 4b). It suggests that the lower part of flow is diverted into the LFC along the barrier, while the upper part of the flow skims across the barrier and is intercepted by the intake.
dropshaft. Under a lower flow condition of $Q = 4$ L/s, the water depth is smaller, and a relatively larger portion (~10%) of the flow is diverted to the LFC by the barrier. For high flow condition of $Q = 41.8$ L/s, the larger flow depth results in a smaller portion (~5%) of diverted flow by the barrier. The effect of the barrier diminishes with increasing flow discharge or depth.

Fig. 5 shows the cross-sectional views of flow field at the intake structure and the LFC at different streamwise positions. It can be seen that near the inclined barrier, the water level inside the LFC is lower than the free surface level outside the channel, thus the flow goes into the LFC (Fig. 5a). The complex transverse flow field downstream of the leading edge of the transverse barrier can be seen in Fig. 5b. Beyond the downstream end of the barrier, the flow plunges onto the chamber below; driven by lateral free surface gradients, a small part of the LFC flow can be returned back to the intake (Fig. 5c). Fig. 5d shows the flow in the centerline longitudinal section—the incoming high velocity flow impinges on the barrier and is deflected upwards with a weak recirculation zone immediately downstream of the barrier.

**Figure 4.** CFD predicted flow field, top view ($Q = 13.9$L/s). (a) $Z = 0.001$m (near bed), (b) Free surface. $Z$ is the normal distance from the channel bed.

**Figure 5.** (a)-(c) CFD predicted flow field at different cross-sections of the intake ($Q = 13.9$L/s). (a) $X = 0$m, (b) $X = 0.008$m, (c) $X = 0.018$m, (d) along channel centerline. $X$ is the downstream distance from the head of the dry weather flow channel.
4. Conclusion
A 3D numerical study of the supercritical flow over an interception barrier of a storm water diversion intake structure has been carried out using a CFD model with the VOF method. Model predictions of the water depth and un-intercepted flows are in satisfactory agreement with measurements on a physical model. The flow interception performance has been analyzed under six storm water runoff conditions. As the total inflow increases, the un-intercepted flow increases; the percentage of the un-intercepted flow to the inflow, however, decreases. The inclined barrier across the channel results in significant flow deflection and spilling over the barrier and complex three-dimensional cross-currents between the main stream and the low flow channel. The near-bed flow is diverted into the low flow channel by the inclined barrier, contributing to a larger un-intercepted flow rate than designed. The numerical study provides insights into possible improvement measures for the intake structures.

5. Acknowledgements
This work is supported by a study commissioned by the Drainage Services Department (DSD) of the Hong Kong Special Administrative Region (HKSAR) Government and AECOM Hong Kong (CE 55/2013(DS)).

6. References
Drainage Service Department (DSD) (2003). Stormwater Drainage Master Plan Study in Northern Hong Kong Island—Executive Summary. The Government of Hong Kong Special Administrative Region, China.