INDIVIDUAL LOT ON-SITE STORMWATER DETENTION

UNDERNEATH RESIDENTIAL CAR PORCH

DARRIEN MAH YAU SENG
JOHNNY NGU ONG KING

INDIVIDUAL LOT ON-SITE STORMWATER DETENTION

UNDERNEATH RESIDENTIAL CAR PORCH

INDIVIDUAL LOT ON-SITE STORMWATER DETENTION

UNDERNEATH RESIDENTIAL CAR PORCH

DARRIEN MAH YAU SENG
JOHNNY NGU ONG KING

First Publication 2017

Published in Malaysia by

UNIMAS Publisher,
Universiti Malaysia Sarawak,
94300 Kota Samarahan,
Sarawak, Malaysia.

Printed in Malaysia by

Heng Sing Brothers Press,
No. 37, Lot 75, Section 63, 1st Floor
Jalan Chan Bee Kiew, Kuching,
Sarawak, Malaysia.

Perpustakaan Negara Malaysia Cataloguing-in-Publication Data

Mah, Darrien Yau Seng, 1977-
INDIVIDUAL LOT ON-SITE STORMWATER DETENTION UNDERNEATH RESIDENTIAL CAR PORCH / DARRIEN MAH YAU SENG, JOHNNY NGU ONG KING.
ISBN 978-967-2008-45-3 (hardback) 1. Roof runoff--Research. 2. Runoff--Research.
3. Water harvesting--Research.
I. Johnny Ngu Ong King, 1991-. II. Title.
551.488

PREFACE

This is the full report for the Research Acculturation Grant Scheme RAGS/TK01(1)/1315/2015(09). The idea of having a combined pavement on top and underground detention system was first surfaced in 2012. A group of UNIMAS researchers had contributed to the methodology of setting up such a system. It took a couple of years to see the idea grew to become an actual product. This technical report is the second, hopefully in a series of reports on a product named **StormPav**. Here, the product is tried for individual residential lot as a part of its car porch from the outlook; and a part of urban stormwater drainage system hidden underground.

Malaysian generally prefers large car porch. Therefore, a residential car porch presents itself as an alternative strategy to fight against urban flash flood as news of such a flooding occurs more frequently nowadays. **StormPav** could be installed as the pavement for car porch, without noticeably the underneath is a stormwater detention tank that could capture urban runoff. Should each household play a small role to reduce a bit of the runoff volume, the accumulated effect could achieve the runoff pattern of pre-development era – less amount of urban runoff, and probability of increased infiltration to the native soil if the process is allowed.

The author would like to express gratitude to other team members who have been working hard for this project. It has been a delightful journey to be able to involve in the birth of **StormPav**, and wishes are extended to have the product succeeded against the test of time for many years to come.

Dr Darrien Mah
December 2016

Table of Contents

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

ARI	-	Average Recurrent Interval
CFD	-	Computational Fluid Dynamics
DID	-	Department of Irrigation and Drainage
MSMA	-	Manual Saliran Mesra Alam
OSD	-	On-Site Detention
SWMM	-	Storm Water Management Model
UNIMAS	-	Universiti Malaysia Sarawak
US EPA	-	United State Environmental Protection Agency
WSUD	-	Water Sensitive Urban Design

CHAPTER 1
ON-SITE DETENTION

1.1 TRENDS IN URBAN HYDROLOGY

On-site Detention (OSD) storages are becoming popular nowadays to mitigate flash floods. This trend is due to the increase in volume and peak discharges as a result of high proportion of impervious surface in the urban land areas. OSD storage is a structure designed to slow down stormwater runoff and reduce the peak discharge which is expected to solve the problem of flash floods (Haberland et al., 2012). OSD system is part of the proposed measures in Manual Saliran Mesra Alam (MSMA). OSD system has been widely applied in Sydney since the year 1991; the developers in this city provide detention storages for stormwater in their project sites to limit rates of runoff (O'Loughlin et al., 1995).

1.2 EXPLORING NEW POSSIBILITY

Even with the popularity of OSD, the problem persists when availability of land is difficult to come by. Therefore, it is an experiment to use the space underneath a residential car porch, rather than open spaces for OSD in this project. A spacious car porch is a building requirement in the Kuching municipality as each household is assumed to own more than one car for daily activities. As such, a car porch for a single house is usually quite large and takes up a sizeable area at the front of the house. An example is shown in Figure 1.1 below. The car porch area can be put into good use by constructing an underground OSD storage; stormwater from the roof area can then be directed to the OSD tank underneath.

Figure 1.1: Typical Malaysian Houses with Car Porch

Parameters and possible solutions to the engineering design of OSD storage underneath car porch are unknown at the moment. Attempts are made to store stormwater in a temporary storage under the car porch in order to slow down the surface runoff, and later it is slowly released with time in a safe rate. One important question needs to be answered before implementing such OSD: what are the flow characteristics like when the stormwater is to be directed to an OSD beneath a car porch?

At the onset of a rainfall event, instantaneous rain could be intense with a high runoff rate from roof areas. Upon landing on the drain from the roof gutter, the flow is basically supercritical in nature. Such thrusting water flows are at high speed and may come with erosive energy. This is particularly so at the inlet region. Directing this water with such a strong force to an underground chamber could be a challenge, particularly in such a way as not to cause congestion and overflow that may bring inconvenience to the property owners. Therefore, the hydraulics of a system engineered to cater for such flow need to be studied carefully. Laboratory testing coupled with Computational Fluid Dynamics (CFD) modelling efforts could assist in providing the necessary information.

CHAPTER 2

FACTS OF ON-SITE DETENTION

2.1 INTRODUCTION

Usually the rainwater that falls on an undeveloped or natural catchment area is generally infiltrated or absorbed into the ground with some excess water slowly forming overland runoff and flows to streams or water courses. However, for a catchment area that is populated, the urban development alters the natural process of stormwater runoff. Rainwater is rapidly collected on covered land surfaces. Urbanisation increases and accelerates the rate of runoff, reduces infiltration, and water quality deteriorates.

This is mostly caused by replacement of permeable surfaces with impervious surfaces. Such changes prevent the water from infiltrating into the aquifer (Chahar et al., 2011). These impervious surfaces of the built environment include rooftops, streets, parking areas, and paved areas. The rainwater collected on impervious surfaces drains directly to the ditch and then flow to streams or water courses.

From the descriptions above, surface runoff is understandably affected by the processes in the hydrological cycle, such as precipitation and infiltration, plus factors such as imperviousness and land slope. In the absence of stormwater management programmes, urbanisation usually dramatically increases surface runoff volumes and rates. However, it is an idealised goal for urban water management to restore each of the components of the hydrological cycle to its natural level (Oki, 1991).

Stormwater consists of rainfall runoff and any materials (soluble or insoluble) mobilised in its path of flow. According to the Australian Environment, Communications, Information Technology and the Arts Legislation Committee (2002), undeveloped catchments can absorb and infiltrate up to 90% of precipitation; meanwhile, in built environment, the amount of infiltration can be as little as 10% of precipitation. Figure 2.1 illustrates the impact of the said urbanisation on infiltration and runoff.

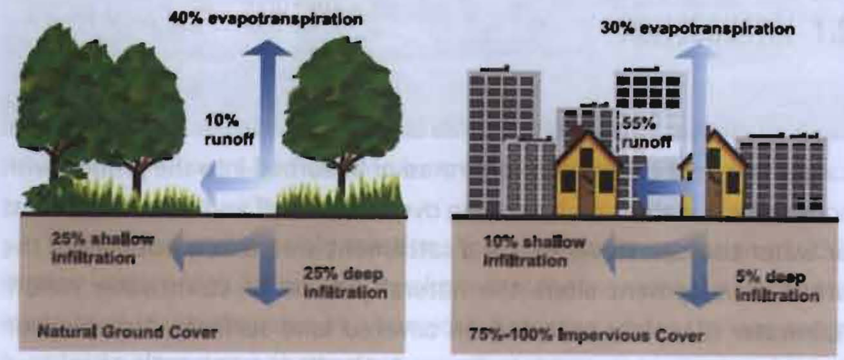

Figure 2.1: Impact of Urbanisation on Infiltration and Runoff (US EPA, 2008)

Uncontrolled stormwater runoff can significantly cause environmental problems. Most stormwater discharges are considered point sources. One of the methods of stormwater control is by using OSD at the point source. The function of the storage-oriented approach is to provide a temporary storage for the stormwater runoff at or near its point of origin. Then the captured stormwater is slowly released to the downstream system or is allowed to infiltrate into the surrounding soil. This is the approach promoted by the Malaysian Department of Irrigation and Drainage (DID) in the Urban Stormwater Management Manual for Malaysia or *Manual Saliran Mesra Alam* (MSMA), the 1st Edition of which was published in the year 2001 and superseded by the 2nd Edition in the year 2012 (DID, 2012).

2.2 ON – SITE DETENTION (OSD)

OSD is a way of collecting the rain that falls on a site and to have the volume of urban stormwater runoff put under control. In other words, OSD is a way of ensuring that changes in land use do not cause more downstream flooding: both in the local drainage system immediately downstream and along the streams and rivers further downstream. The detention concept is employed in MSMA to limit the peak outflow rate for a specific range of flood frequencies near to that which existed in the same catchment before the development (see Figure 2.2).

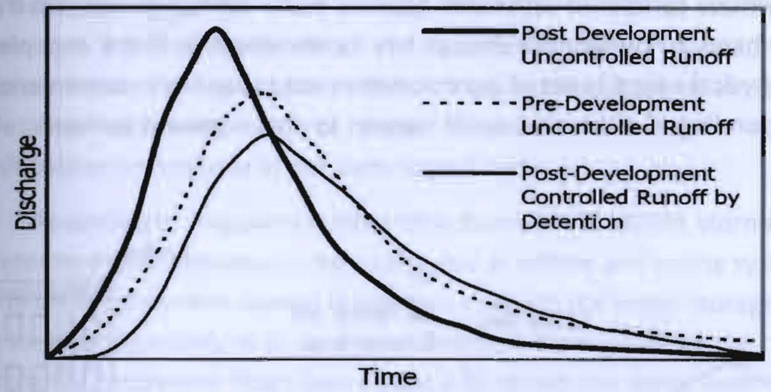

Figure 2.2: Concepts of Urban Runoff Control (DID, 2012)

Detention storage is classified into three different types based on the location and size-- on-site storage, community storage, and regional storage. On-site storage is a small storage constructed in individual residential, commercial, or industrial lots. Community storage is one that is constructed in public open space areas, or in conjunction with public recreation and sporting facilities. Regional storage is a large storage facility that is constructed at the lower end of a catchment before the water is discharged to receiving waters. In this study, on-site storage is emphasised.

At the lot scale, stormwater control measures (e.g. driveway interceptor, lawn retention basin, cisterns and bio-retention pits) have been reported effective in reducing stormwater runoff (Loperfido et al., 2014). The effect of the individual OSD system is small, but the cumulative effect is great. In this line of research, many control measures mentioned above have been explored. There are several designs of OSD such as above-ground, below-ground, or a combination of both types (see Figure 2.3). Real-world examples are presented in Section 2.3.

Above-ground storage can be easily incorporated into a site with slight modification to the design of surface features, and it is relatively inexpensive compared with other types of water storage design. On the other hand, below-ground storage has its advantage in that it occupies less physical space, is out of sight and does not cause any inconvenience with ponding of water that could happen to above-ground storages.

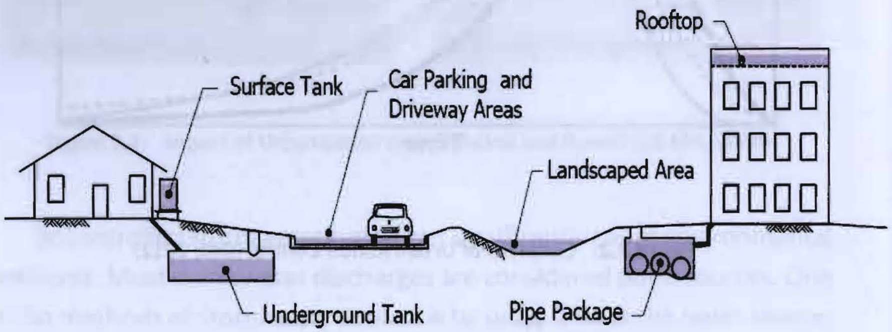

Figure 2.3: Typical OSD (DID, 2012)

However, in the Malaysian context, a stormwater control structure underneath a car porch is an opportunity for research. Site-specific information about the contribution of an individual lot to system-wide stormwater volumes could serve as a powerful public information tool to give all members of a community a stake in watershed management (Scott et al., 2014).

OSD system should be installed on redevelopment sites so that there is no adverse impact from stormwater runoff on downstream properties as a result of the ongoing developments in the catchment. Based on Zakaria et al. (2004), a few studies by the DID indicated that there is a downtrend of capabilities in the existing drainage system to cater for stormwater runoff. All real-world studies of this nature to date have been devoted to OSD technology and have led to council-administered strategies governing redevelopment applications (Clarke et al., 1997).

OSD is not designed to provide stormwater quality benefits, but it holds the runoff for short periods. The total volume of stormwater leaving a site is not reduced but the detained volume is released when downstream drainage capacity becomes available (Scott et al., 1999). Detained stormwater can be either released or allowed for natural infiltration to continue in the hydrological cycle.

According to Singapore Public Utility Board (PUB) (2010), stormwater detention tank systems can be configured as offline and online systems. The offline detention system is a system in which the water storage tank is located separately or in parallel to the drain through which the runoff from the catchment flows (see Figure 2.4). When the water level in the drain does not exceed a certain level, the runoff just flows in the drain. However, when there is excess runoff above the weir level, the running water is diverted into the detention tank. Hence, there is only a portion of the flow in the drain is conveyed into the detention tank.

For the online detention system, runoff from the entire catchment is drained to the detention tank via an inlet (see Figure 2.5). Discharge of stormwater from the detention tank can take place during or after the storm event, as long as the total peak runoff to be discharged from the development site is in compliance with the requirement of the maximum allowable peak discharge. With such systems in place, the drainage system as a whole can cater for higher intensity storms brought about by the increasing uncertainties due to climate change (Scott et al., 1999).

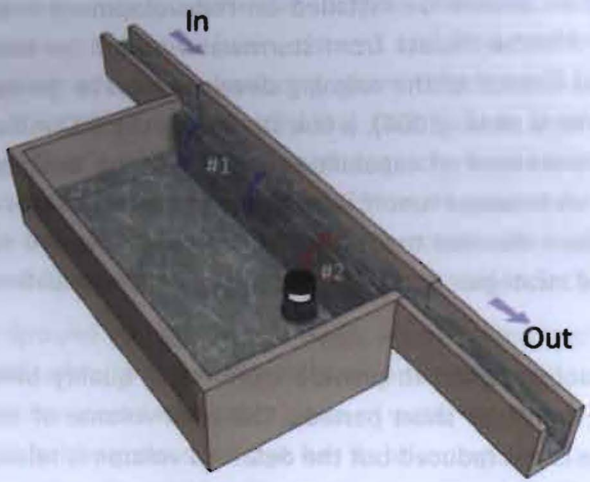

Figure 2.4: Offline Detention System (PUB, 2010)

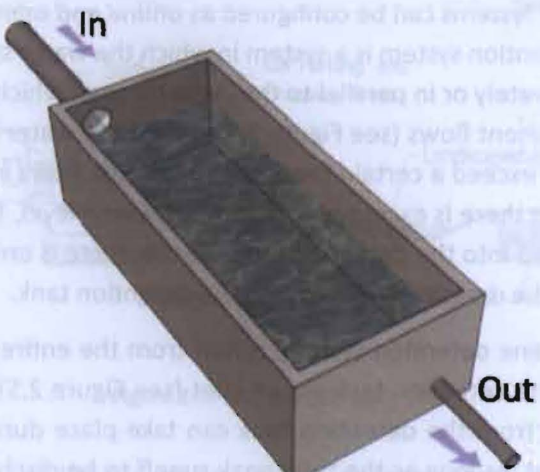

Figure 2.5: Online Detention System (PUB, 2010)

Although the detention volumes required by offline detention systems are smaller as compared with those of online detention systems, it is generally more complex to design offline detention systems due to the sensitivity of the weir levels in relation to the water levels in the diversion structure (PUB, 2010).

Detention tanks require very little maintenance after the installation. They have no moving parts and remain intact for many years (US EPA, 2001). There are some concerns that the use of corrugated steel or polyethylene pipes might crack or buckle over time due to the weight of soil surrounding them. However, a study of underground corrugated steel pipe (CSP) for stormwater detention structures in the metropolitan area of Washington, D.C. conducted by the National Corrugated Steel Pipe Association (NCSPA, 1999) found that all components of the system were performing well. There were no signs of buckling, cracking, or bending on the pipe although it had been placed for up to 25 years.

2.3 EXAMPLES OF ON-SITE DETENTION

In previous studies pertaining to OSD, centralised large size control structures were used, for example wet ponds (Chang et al., 2013; Ahmad et al., 2014) and detention basin (Vorogusshyn et al., 2012). But availability of empty land is getting more and more difficult to come by particularly in urban areas. Therefore, the trend of focusing on localised control structures in a distributed manner is gaining importance today. It is to cater for physiographic and urban development characteristics (Hamel et al., 2013). Literatures on best management practices covering various measures are widely reported (Baker et al.; Reynolds et al., 2012; Ritchie, 2014). However, utilising a car porch for the purpose of OSD is rare in the literature. Probably, this is due to the different living styles in different parts of the world; for example, many residents of certain countries would prefer a garage to a car porch.

Upper Parramatta River Catchment Trust, Australia

A case study below presents the applications of an online underground detention tank. There are approximately 1000 OSD systems installed in the Upper Paramatta River catchment, mostly within individual lots (see Figure 2.6).

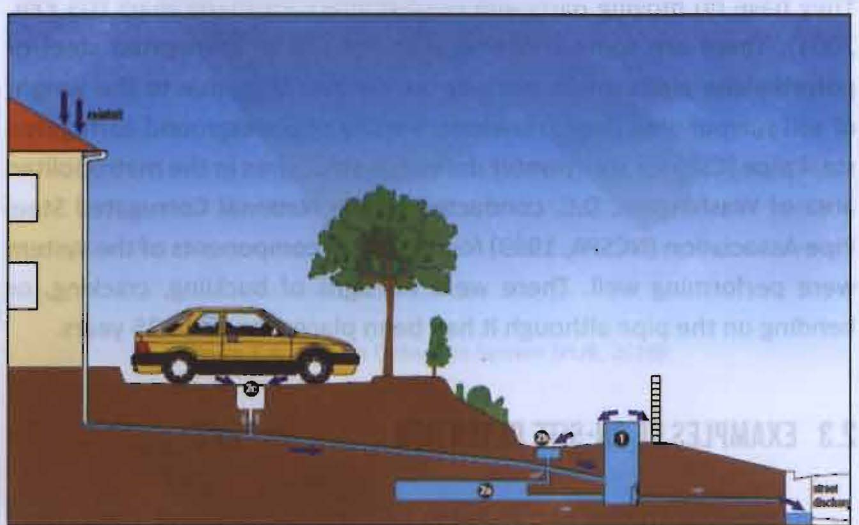

Figure 2.6: Individual Lot OSD (http://www.uprct.nsw.gov.au)

Sydney, Australia – Individual Lot Underground OSD

This is another type of online underground storage tank designed using polyethylene as the main material of the tank. This type of underground tank is designed using rounded sections with ribbing to provide maximum strength and it can last for at least 15 years. It can store amounts of rainwater ranging from 3000 to 5000 litres (Waterplex, 2014).

Figure 2.7: Bagel Toroid Underground Tank, Artarmon, Sydney, Australia (Waterplex, 2014)

The case studies above provide credible evidence that OSD under a car porch can be utilised to reduce rainfall runoffs. As a result, peak runoffs of storms can be slowed down and peaks in the hydrographs will change accordingly.

2.4 DESIGN CRITERIA

The most common type of storage facility used for detention is 'dry facility'. Such facility releases all the detained runoff after a storm. Detention of water on development sites is seen as a solution to solve the flooding problems that occur at the established areas. Generally, it is not possible to progressively enlarge drainage systems due to physical or financial constraints.

Since this study stresses the underground OSD, such design criteria are presented in the following writing. Below-ground storage can be designed using underground tanks or pipe packages (DID, 2012). Typical underground storage tanks are designed in circular or rectangular shapes and normally constructed using reinforced concrete.

Some important elements need to be taken into consideration when designing underground OSD systems; they are the size, shape, and physical characteristics of space available for the systems. All these factors will determine how a system is constructed and what type of construction material to use. Besides, consideration of site specifications will also influence the design criteria of the system such as depth of the water table, area of allowable excavation space, and construction costs.

Underground OSDs can be in different forms, ranging from simple storage pipes or chambers to complex systems consisting of multiple pipes or chambers, with accompanying joints, crossovers, multiple inlets and access points. However, each water detention system must have at least an inlet and access to the chamber.

For a larger site, it requires more than one detention system to serve the entire catchment; a large detention volume would be needed and the system would have to be placed at the most downstream end of the internal drainage network. Thus, a site may be analysed and split into various sub-catchments, adopting a distributed catchment approach (PUB, 2010). OSD storages can be analysed using any of the existing hydrograph estimation techniques (Figure 2.8), but the Rational Method is by far the most popular (DID, 2002).

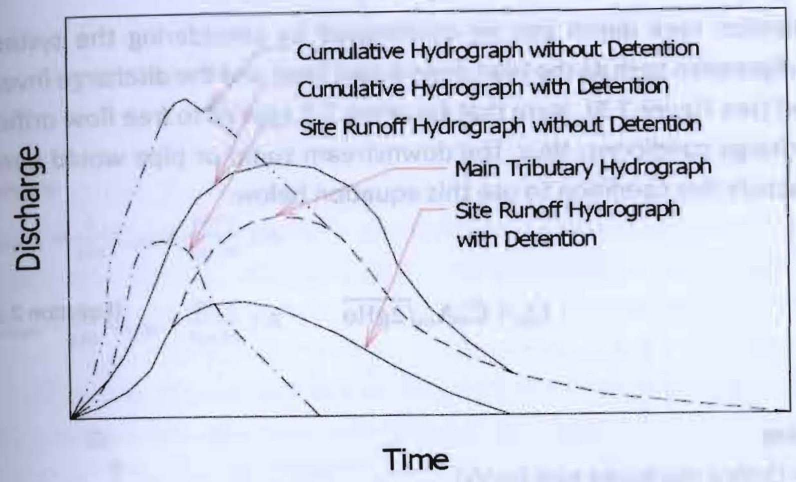

Figure 2.8: Example of Cumulative Hydrographs With and Without Detention (DID, 2002)

To calculate the flow produced by the design rainfall, Rational Method, as in Equation 2.1 is used (DID, 2012).

$$Q = \frac{C\,I*A}{360} \qquad \text{(Equation 2.1)}$$

where

Q = peak flow (m^3/s)
C = 1.0 (fully impervious)
I = rainfall intensity (mm/hr)
A = catchment area (ha)

The size of detention tank is determined based on the space available and volume of stormwater from Equation 2.1. The size of orifice outlet can be calculated based on Equation 2.2. An orifice discharge system serves as the flow regulator for the detention tank. The effective detention tank depth can be determined by considering the system configuration such as the inlet drain invert level and the discharge invert level (see Figure 2.9). Note that Equation 2.2 applies to free flow orifice discharge conditions; thus, the downstream sump or pipe would need to satisfy this condition to use this equation below.

$$Q_o = C_o A_o \sqrt{2gH_o} \qquad \text{(Equation 2.2)}$$

where
Q_o = Orifice discharge rate (m^3/s)
C_o = Discharge coefficient (m^2)
A_o = Area of orifice (m^2)
g = Acceleration due to gravity ($9.81 m/s^2$)
H_o = Maximum head to the centre of the orifice (m)

Figure 2.9: Water Storage Tank (PUB, 2010)

According to PUB (2010), the outflow hydrograph for gravity discharge detention systems is approximated by a straight line as shown in Figure 2.10. The storage volume required for particular storm duration is represented by the shaded area between the inflow and outflow hydrographs.

Storage Volume, $$V_t = Q_{inflow}(t_c + t_x) - \frac{1}{2} Q_{target}(2t_c + t_x) \qquad \text{(Equation 2.3)}$$

where

$$Q_{inflow} = \frac{1}{360} C_{post} \left(\frac{8973}{td+36} \right) A$$

$$Q_{target} = \frac{1}{360} C_{target} \left(\frac{8973}{tc+36} \right) A$$

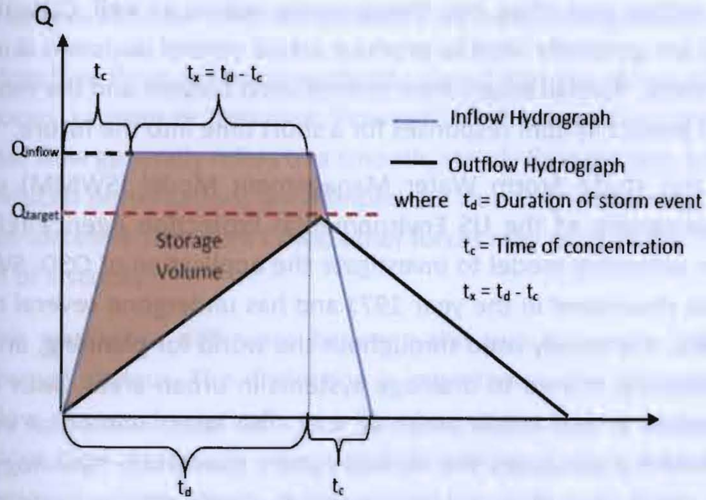

Figure: 2.10: Example of Inflow and Outflow Hydrographs for Gravity Controlled Detention System (PUB, 2010).

2.5 MODELLING OF ON-SITE DETENTION

According to Huber *et al.* (1988), there are four broad objectives of modelling which are normally used for studies of quantity and quality problems associated with urban runoff. The four broad objectives of modelling are screening, planning, design and operation. Each objective generally produces models with slightly different characteristics, and the different models overlap one another to some extent.

Screening models provide the first estimation of the magnitude of urban runoff quantity and quality problems, prior to the investment of time and resources into more complex computer-based models. Planning models are used for an overall assessment of the urban runoff problem as well as estimation of the effectiveness and costs of abatement procedures. Planning or long-term models may also be used to generate initial conditions, which are antecedent conditions, for input to design models. Design models are oriented towards the detailed simulation of a single storm event. They provide complete descriptions of flow and pollutant routing from the point of rainfall through the entire urban runoff system and often into the receiving waters as well. Operational models are generally used to produce actual control decisions during a storm event. Rainfall enters from telemetered stations and the model is used to predict system responses for a short time into the future.

In this study, Storm Water Management Model (SWMM) under the sponsorship of the US Environmental Protection Agency (EPA) is used as screening model to investigate the application of OSD. SWMM was first developed in the year 1971 and has undergone several major upgrades; it is widely used throughout the world for planning, analysis and designing related to drainage systems in urban areas, with many applications in non-urban areas as well. The latest numerical engine of SWMM 5.0 calculates the surface runoff, subsurface hydrology and assigns the current climate data at either the wet or dry hydrologic time step (Rossman, 2010).

The runoff component of SWMM operates on a collection of sub-catchment areas that receive precipitation and generate runoff and pollutant loads. The routing portion of SWMM that transports this runoff can be designed through a system of pipes, channels, storage/ treatment devices, pumps, and regulators. SWMM tracks the quantity and quality of runoff generated within each sub-catchment; the flow rate, flow depth, and quality of water in each pipe and channel during a simulation period consist of multiple time steps (Rossman, 2010).

2.6 FLOW – SUPERCRITICAL AND TURBULENCE

Fluids are substances that deform continuously and permanently when they are subjected to forces that vary spatially in magnitude or direction. The nature of the relationship between the deforming forces and the geometry of deformation varies from fluid to fluid. Fluids are treated as continuous media, and their motion and state can be specified in terms of the velocity u, pressure p, density ρ, etc., evaluated at every point in space x and time t (Pedley, 1997). The movement of liquids and gases is generally referred to as "flow," a concept that describes how fluids behave and how they interact with their surrounding environment.

Most fluid flows can be conveniently placed into one of two different categories: Laminar or Turbulent. Flow can be either steady or unsteady. Laminar flow generally refers to a smooth, steady fluid motion, in which any induced perturbations are damped out due to the relatively strong viscous forces. In turbulent flows, other forces may react to the counter action of viscosity.

The qualitative difference between the two is usually (although not always) obvious. This distinction is important as the characteristics of a flow change dramatically as a flow transitions from a laminar to a turbulent state. As already mentioned, the mixing of mass, momentum, and energy occurs much more rapidly in turbulent flows. Many applications related to such diverse areas as aerodynamics, sediment transport, combustion, acoustics, and the weather are all significantly affected by turbulence. One important factor in determining the state of a fluid's flow is its viscosity, or thickness, where higher viscosity increases the tendency of the flow to be laminar.

Laminar and turbulent flows can be characterised and quantified using Reynolds Number established by Osborne Reynolds and is given as:

$$R_e = \frac{\rho VD}{\eta} = \frac{VD}{\nu} \quad \text{(Equation 2.4)}$$

where

R_e = Reynolds Number and R_e is dimensionless

V = velocity of flow (m/s)

D = diameter of pipe (m)

ρ = density of water (kg/m^3)

η = dynamic viscosity (kg/m.s)

ν = kinematic viscosity (m^2 /s)

Reynolds Number is directly proportional to velocity and inversely proportional to viscosity. For flow around a cylinder, the flow starts separating at Re = 5. For Re below 30, the flow is stable and oscillations appear at higher Re.

Laminar flow (or streamline flow) occurs when a fluid flows in parallel layers, with no disruption between the layers. Laminar flow tends to occur at lower velocity, below a threshold at which it becomes turbulence. At low velocity, the fluid tends to flow without lateral mixing, and adjacent layers slide past one another like playing cards. There are no cross-currents perpendicular to the direction of flow, nor eddies or swirls of fluids. In laminar flow, the motion of the particles of the fluid is very orderly with all particles moving in straight lines parallel to the pipe walls. Laminar flow is a flow regime characterised by high momentum diffusion and low momentum convection.

One characteristic of turbulent flows is their irregularity or randomness. Turbulent flows are usually described statistically. Turbulent flows are always chaotic. But not all chaotic flows are turbulent. Waves in the ocean, for example, can be chaotic but are not necessarily turbulent. Turbulence is rotational and an inherently three-dimensional phenomenon. It is characterised by large fluctuations in vorticity. Most of the important characteristics of turbulent flow, such as vortex stretching and length scale reduction, are identically zero in two dimensions. This contributes to the difficulties in describing turbulence both analytically and numerically. There are situations in which the turbulent velocity field is, for some length scales, confined to two dimensions. Flow in the atmosphere is often treated as two-dimensional as it is confined to a thin layer over the surface of a planet. Although some important dynamical mechanisms are absent in two-dimensional flows, they can be highly complex and nonlinear, and they often provide a reasonable approximation to study the features of interest.

Besides the boundaries matter, there can be no normal component of velocity through the boundary. If friction is neglected, there may be free slip along the boundary, but friction has the effect of slowing down fluid near the boundary and it is observed experimentally that there is no relative motion at the boundary, either normal or tangential to the boundary. In fluids with low viscosity, this tangential slowing down occurs in a thin boundary layer, and in a number of important applications this boundary layer is so thin that it can be neglected and we can say approximately that the fluid slips at the surface; in many other cases the entire boundary layer separates from the boundary and the inviscid model is a very poor approximation. Thus, in an inviscid flow (also called the flow of an ideal fluid) the fluid velocity must be tangential at a rigid body.

Figure 2.11 shows that in a flow started from rest, no vorticity develops anywhere until viscous diffusion has had an effect there. The separation point moves upstream, increasing drag up to Re = 2000.

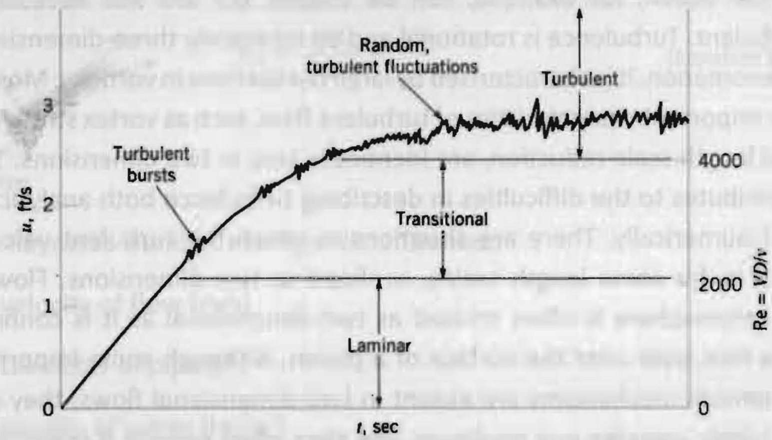

Figure 2.11: Illustration of the Transition from Laminar to Turbulence as a Function of R_e for a Cylindrical Wake (Van Dyke, 1982)

It is convenient now to restrict attention to a 2D flow associated with Figure 2.11 of a homogeneous fluid past a 2D body such as a circular cylinder (Figure 2.12).

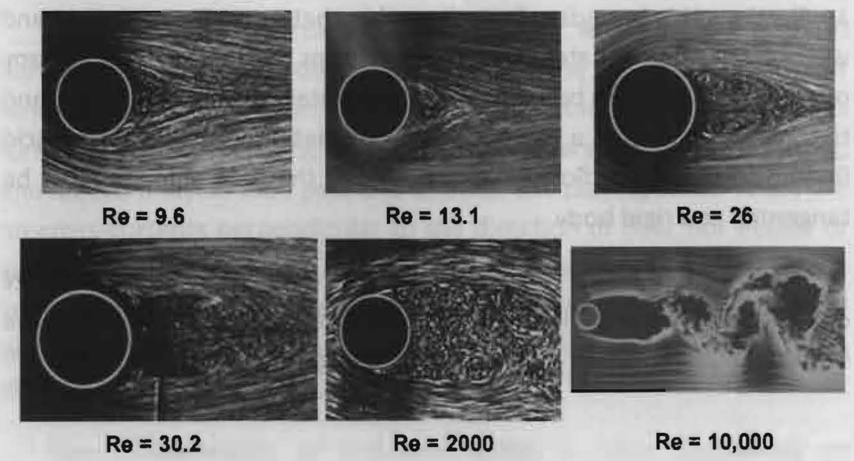

Figure 2.12: Steady Flow Past a Circular Cylinder at Different Values of the Reynolds Number (Van Dyke, 1982)

In such a 2-D flow, with velocity components $u = (u,v,0)$, functions of x, y and t, the vorticity is entirely in the third, z, direction, and is given by:

$$\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \quad \text{(Equation 2.5)}$$

There is no vortex-line stretching, and the only effect which can generate vorticity anywhere is viscosity. Assuming that the uniform stream at infinity is switched on from rest at the initial instant. Initially there is no vorticity anywhere, and the initial irrotational velocity field is easy to calculate. It satisfies all the governing equations and all boundary conditions *except* the no-slip condition at the cylinder surface. The predicted slip velocity therefore generates an infinite velocity gradient $\partial u/\partial y$ and hence a thin sheet of infinite vorticity at the cylinder surface.

The different types of "free shear flows" influence the large scale 2D structures and dominated the statistics. These structures can play a dominant role in the transport of scalar material. However, the smaller scale, three-dimensional motions come into play when mixing at molecular scales is important. The distinction between scalar transport and molecular diffusion is critical in understanding and predicting turbulent mixing processes.

Most problems in fluid dynamics are too complex to be solved by direct calculation. In these cases, problems must be solved by numeric methods using computer simulations. This area of study is called numerical or computational fluid dynamics (CFD). However, because turbulent flow tends to be nonlinear and chaotic, particular care must be taken in setting up the rules and initial conditions for these simulations. Small changes at the beginning can result in large differences in the results.

2.7 FLOW PREDICTION METHODS

In common days, engineers are used to predict the behaviour of systems to understand the relationship between the system variables. The prediction can help engineers to come up with better design of systems or understanding of their behaviour for optimising their operation. There are three approaches used by the engineers nowadays to understand the relationship between the system variables.

Typically, engineers perform experiments which are usually termed experimental techniques. This method allows them to understand the system directly or construct mathematical models that represent their systems. The most reliable information about physical phenomena is usually given by measurements. In certain situations, an experimental investigation involving full-scaled equipment can be used to predict how the equipment would perform under given conditions.

However, in most practical engineering applications, such full-scaled tests are either difficult or very expensive to perform, or not possible at all. A common alternative is to perform experiments on small-scaled models. The resulting information however, needs to be extrapolated to the full-scaled and general rules for doing this are often unavailable. The small-scaled models do not usually simulate all the features of the full-scaled system. This sometimes limits the usefulness of the test results. In addition, in many situations, there are serious difficulties in measurements and the measuring equipment can have significant errors. Besides, it also involves significant costs for some experimental techniques.

Another approach called analytical approach is to understand the system by constructing a mathematical model based on the understanding of the basic physical phenomenon that governs its behaviour and then tries to solve these models for a given set of conditions by finding a mathematical solution to the resulting system of equations. Analytical models work out the consequences of a mathematical model which represents the behaviour of a system. The mathematical model representing the physical process mainly consists of a set of differential equations. If classical mathematics were used to solve these equations, the approach is called the analytical or theoretical approach.

Analytical methods played a significant role in the past and they still play an important role now. They have helped engineers and scientists in the understanding of the fundamental rules controlling the behaviour of many engineering systems. In addition, they are used to help understand and interpret experimental results. Furthermore, they can be used as a first stage in the validation of CFD models.

However, in most practical engineering applications, various assumptions and simplifications need to be made to enable the analytical solution of the differential equations representing the physical situation. This on the one hand limits the applicability of these methods to simple type problems or limits the validity of the solutions if too many assumptions and simplifications are made.

The third approach is the use of CFD method. This method is used in this study where the differential equations governing the system are converted to a set of algebraic equations at discrete points, and then solved using digital computers (see Figure 2.13). Experimental fluid dynamics has played an important role in validating and delineating the limits of the various approximations to the governing equations. Traditionally, this has provided a cost effective alternative to full-scaled measurement. However, the design of equipment depends critically on the flow behaviour. The flow governing equations are extremely complicated such that analytic solutions cannot be obtained for most practical applications. This situation has led to an increasing interest in the development of the CFD methods.

Figure 2.13: Liquid Flow Pattern in Bubble Column

The role of CFD in engineering predictions has become so strong that today it may be viewed as a new third dimension of fluid dynamics; the other two dimensions are the above stated classical cases of pure experiment and pure theory. CFD is the branch of fluid dynamics providing a cost-effective means of simulating real flows by the numerical solution of the governing equations. The governing equations for Newtonian fluid dynamics, namely the Navier-Stokes equations, have been known for over 150 years (Abdulnaser, 2009). However, the development of reduced forms of these equations is still an active area of research, in particular, the turbulent closure problem of the Reynolds-averaged Navier-Stokes equations. For non-Newtonian fluid dynamics, chemically reacting flows and two phase flows, the theoretical development is at a less advanced stage.

All fluid flows are governed by three fundamental principles according to Hjertager (2001) and Veersteg & Malalasekera (2007), namely:

a) Conservation of mass;

b) Conservation of energy (1st law of thermodynamics); and

c) Newton's second law (force equal the rate of change of momentum).

By the use of these principles, one can derive the Navier-Stokes equations which are a set of partial differential equations that describes the flow of fluid. Determination of numerical simulations to the Navier-Stokes equation that can be propagated in time and space are the main issue of CFD. The following equations are solved on a grid inside the region of interest. The dependent variables in a 3-dimensional numerical analysis are:

a) Pressure, p;

b) Temperature, T;

c) Viscosity, μ;

d) Heat conductivity, Y;

e) Density, ρ;

f) Velocity in x-direction, u

g) Velocity in y-direction, v; and

h) Velocity in z-direction, w.

Conservation of mass (continuity equation)

$$\frac{\partial \rho}{\partial t} + \nabla(\rho \vec{u}) = 0 \qquad \text{(Equation. 2.6)}$$

Momentum balance in x, y and z direction (on vector form) is expressed as:

$$\frac{\rho D\vec{V}}{Dt} = -\vec{\nabla} P + \vec{\nabla}\vec{\tau} + \vec{F} \qquad \text{(Equation. 2.7)}$$

where

$V^{\rightarrow}$ is the velocity vector, $\tau^{\rightarrow}$ is the viscous stress vector and $F^{\rightarrow}$ is the body force.

Conservation of energy (on tensor form) is expressed as:

$$\frac{\partial(\rho h)}{\partial t} + \frac{\partial(\rho u j h)}{\partial x j} = \frac{-\partial q j}{\partial x j} + \frac{Dp}{Dt} + \tau i, j\frac{\partial u i}{\partial x j} + S \qquad \text{(Equation. 2.8)}$$

where $\dot{S}$ is the source term and h is enthalpy.

Equation of state (general transport equation) is expressed as:

$$\frac{\partial(\rho\varphi)}{\partial t} + \frac{\partial(\rho u i\varphi)}{\partial x i} = \frac{\partial}{\partial x i}\left[\frac{\Gamma\varphi\,\partial\varphi}{\partial x i}\right] + S\varphi \qquad \text{(Equation. 2.9)}$$

where φ is a general variable and Γ is the transport coefficient.

Viscosity is a measure of the internal resistance between neighbouring particles. In a Newtonian fluid, the viscous stresses and the rate of deformation are proportional. All variables are split into a mean and a fluctuating part; this is called Reynolds decomposition. The time average of the fluctuating part is introduced into the general

The purpose of a flow simulation is to find out how the flow behaves in a given system for a given set of inlet and outlet conditions. These conditions are usually termed boundary conditions.

Since the geometry in new design material is complex, it is difficult to find an analytical solution to the flow equations. For all engineering purposes, it is useful to know the basic flow quantities at a large number of discrete points spread around the geometry. This gives enough understanding of the flow behaviour and enables engineers to get the required information either for operation or design purposes.

The basic concept of CFD method is then to find the values of the flow quantities at a large number of points in the system. These points are usually connected together in what is called numerical grid or mesh. The system of differential equations representing the flow is converted, using some procedures, to a system of algebraic equations representing the interdependency of the flow at those points and their neighbouring points.

The resulting system of algebraic equations, which can be linear or non-linear, is usually large and requires a digital computer to solve. In essence, a system is set up with the unknowns being the flow quantities at the grid points. Solution of this system results is in the knowledge of these quantities at the grid points.

If the flow is unsteady, it is either due to varying boundary conditions, or due to inherent unsteadiness. The solution procedure is repeated at discrete time intervals to predict the evolution in time of the flow variables at the grid points.

With the development of fast and validated numerical procedures, and the continuous increase in computer speed and availability of cheap memory, larger and larger problems are being solved using CFD method at cheaper cost and quicker turnaround times. In many design and analysis applications, CFD method is quickly replacing experimental and analytical methods.

In addition to the speed and reduced cost of CFD method, compared with experimental procedures in most engineering applications, it also offers a more complete set of information. It usually provides all relevant flow information throughout the domain of interest. Experimental methods are mostly limited to measurements of some flow quantities at certain locations accessible by the measuring equipment.

CFD simulations also enable flow solutions at the true scale of the engineering systems with the actual operating conditions, while experimental measurements mostly require either scaling up or down. In most cases, realistic conditions cannot be economically represented and thus results need to be extrapolated. This problem does not exist in CFD simulations.

2.8 TURBULENCE MIXING AT INLET REGION

For a case depicted in Figure 2.14, the water emerges from the pipe and the kinetic energy is converted into flow or fluid energy. The pressure would increase and it is called dynamic pressure. The water is observed to swirl about at the inlet region, hence a mixing mechanism before finally coming to rest. All the kinetic energy is dissipated by friction and is expressed as a fraction of dynamic pressure. Eventually, all the dynamic pressure is lost. These are the general descriptions of turbulence mixing at inlet region.

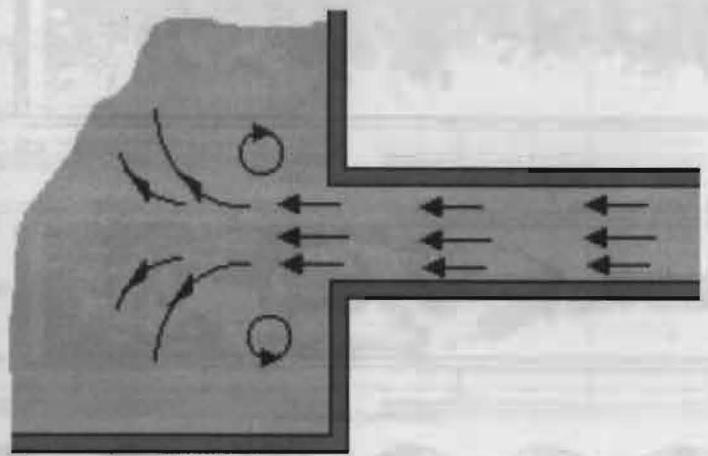

Figure 2.14: Flow at Inlet Region

Yet, scientists have made use of this short-lived turbulent mixing for a couple of industrial applications. A typical flow field within a tabular reactor of industrial geometry is depicted in Figure 2.15. A reactant is injected to a feed pipe to introduce a turbulent jet to the flow system, observable at p020 and p050. The kinetic energy is high and therefore high pressure exists in the beginning. The reactant begins to swirl after p100 as the dynamic pressure is diminishing. It tends to move towards low pressure zone, thus moving to the side away from the jet streamline. The swirling and slowing down of velocity cause the reactant to be distributed over the whole length of the feed pipe, observable at p200 to p400.

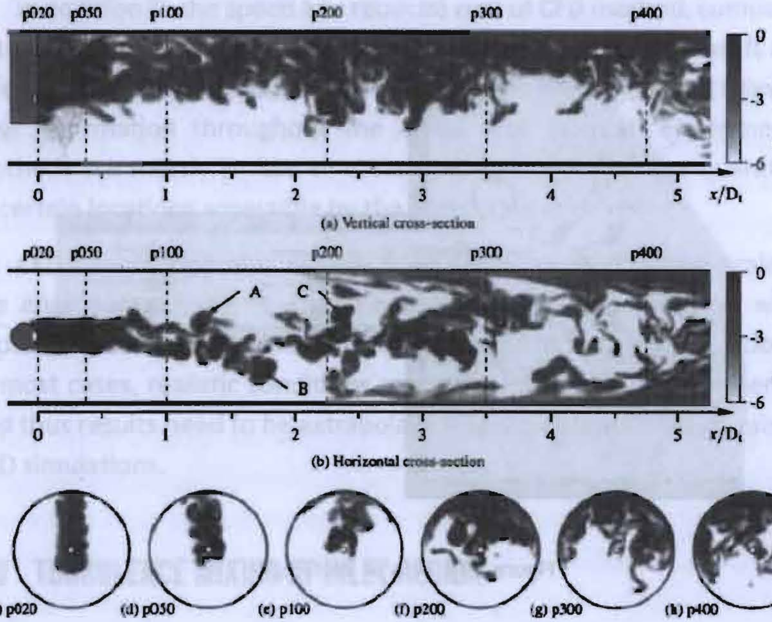

Figure 2.15: Turbulent Mixing in Tabular Reactor (Van Vliet et al., 2005)

Another example is depicted in Figure 2.16 for a confined tank. Velocities along the mid plane at 5s to 40s from commencement of filling are illustrated. Gas velocities are found to be highest in the vicinity of tank axis. It is observed that the incoming jet strikes the rear end of the tank, and flow turns along the rear surface towards the inlet forming a secondary flow region on either side of inlet jet. These secondary flow regions shift positions and vary in size with respect to time during the fill. Flow velocity gradually decreases with the increase in pressure as density of gas increases as the tank gets filled up. Flow at the inlet is turbulent throughout the fill. Development of flow field within the tank is dominated by the structure of turbulent jet flowing from the inlet. Moreover, turbulence near the inlet results in enhanced heat transfer to the wall in the region.

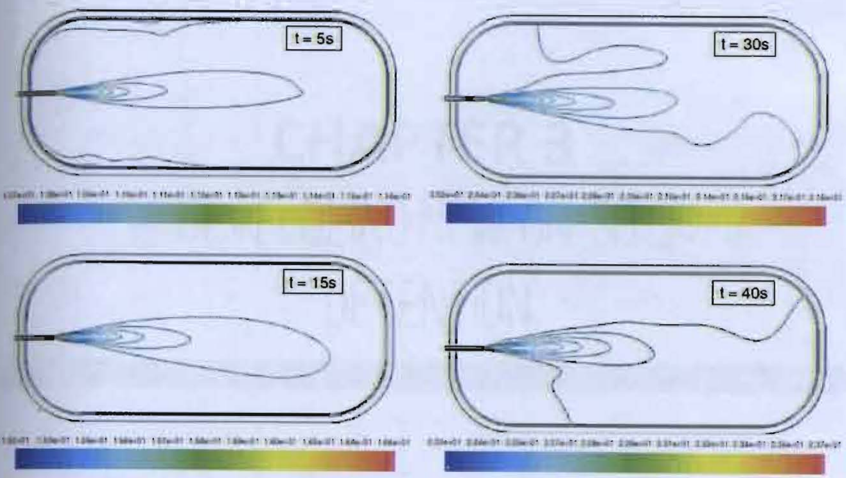

Figure 2.16: Turbulent Mixing in Confined Tank (Setoguchi et al., 2013)

Based on the descriptions above, it can be summed up in the following points that can be applicable to OSD underneath a car porch:

a) Water from turbulent jet tends to move to low pressure zone;

b) Secondary flow is expected at tight spaces and its effect varies;

c) Inlet structure influences the flow pattern.

CHAPTER 3
DEVELOPING NEW ON-SITE DETENTION

3.1 INTRODUCTION

The increase in impervious ground surfaces leads to exacerbation of flooding disasters. In order to combat this problem, the main idea of this study is to direct the runoff from the roof of a house to a temporary storage below a car porch. This does not entail any use of space either within the compound of an individual house or in the neighbourhood. As depicted in Figure 3.1, a car porch is designed to be spacious. Hence, it is practicable to install a unit of OSD water storage underneath the car porch which does not take up any space above the ground.