An experimental and computational investigation of performance of the Green Gully for reusing stormwater

Sharmina Begum, M. G. Rasul, R. J. Brown, N. Subaschandar and Phil Thomas

ABSTRACT

A new stormwater quality improvement device called 'Green Gully' has been designed and developed in this study with the aim of reusing stormwater for irrigating trees and other plants. The main purpose of the Green Gully is to collect road runoff/stormwater, make it suitable for irrigation and provide an automated network system for watering roadside plants and irrigational areas. This paper presents the design and development of Green Gully along with experimental and computational investigations of the performance of Green Gully. Performance (in the form of efficiency, i.e. the percentage of water flow through the gully grate) was experimentally determined using a gully model in the laboratory first, then a three-dimensional numerical model was developed and simulated to predict the efficiency of Green Gully as a function of flow rate. Computational fluid dynamics code FLUENT was used for the simulation. GAMBIT was used for geometry creation and mesh generation. Experimental and simulation results are discussed and compared in this paper. The predicted efficiency was compared with the laboratory measured efficiency. It was found that the simulated results are in good agreement with the experimental results.

Key words | CFD simulation, Green Gully, stormwater management, stormwater quality improvement devices, stormwater reuse

INTRODUCTION

In Australia, the recent drought and concerns about climate change have all highlighted the need for managing water resources in a more sustainable manner (Department of Environment and Conservation NSW 2006). While urban water supplies may comprise only around 30% of total Australian water use, according to the Commonwealth Scientific and Industrial Research Organisation (CSIRO 2001), they have significant impacts on the catchments in which they are situated. At present, many people in the world are likely to suffer from the lack of clean water, particularly those who live in hot, arid countries where reliable water supplies are only available during part of the year. Two-thirds of the earth's surface is covered by water but 97% of it is saline seawater that is not suitable for drinking, irrigation, industry or household use. The remaining water is not easily accessible because nearly three-quarters of it is either frozen in polar ice caps or present as ground moisture. Less than 1% of the world's water is in freshwater lakes and rivers. With the world's population growing alarmingly, the available freshwater is not sufficient to meet human demands and this presents a huge problem (Frederick 1995).

Therefore, there has been increasing interest in the use of water resources generated within the urban boundary for potable supply substitution, as a means of augmenting current supply capacity. It is now a vital issue to access alternative sources of water. Stormwater can play a significant role as an alternative source of water. This is now acknowledged as a valuable resource for irrigation and watering gardens after the required level of treatment. Expanding the use of

doi: 10.2166/wrd.2011.010
stormwater runoff to add to the water supply and reduce water pollution are important objectives (Begum et al. 2008a).

Stormwater treatment is the most important issue in the reuse of stormwater. A hierarchy of stormwater treatment levels based on the dominant treatment processes are: (1) primary, (2) secondary and (3) tertiary. Generally, the greater the level of treatment the greater the reduction in pollutants, and the fewer restrictions there are on the potential reuse. Primary treatment normally involves screening out gross pollutants and sediment to remove coarse particulate matter. Secondary treatment removes organic matter and lighter solids by biological and mechanical means. Tertiary treatment removes nutrients and finer suspended particulate matter by one or more means, including carefully controlled biological processes, chemical processes and filtration (James 1994; Begum et al. 2008b). The level of treatment depends on the application and the risk to the public. For many applications, secondary treatment (i.e. physical treatment to remove solids and biological treatment to remove organics) with disinfection may be adequate. For higher contact applications, tertiary treatment (i.e. secondary treatment plus removal of nutrients) may be required (James 1994; Begum et al. 2008c). Recycled water is directed to major industrial consumers of water such as power stations to release existing raw water supplies for urban consumption (Traves et al. 2008).

A new stormwater quality improvement device (SQID), named the ‘Green Gully’, has been designed and developed in this study with an aim reuse stormwater to irrigate trees and other plants. Green Gully can collect water from storms, remove pollutants from road runoff and offer an automatic network for watering plants and irrigating nearby land. The Green Gully serves two purposes. Firstly, it diverts stormwater from the roadways to the diverter channel by filtering litter, and secondly it waters the roadside plants with the stormwater that is collected from the diverter channel (Begum et al. 2008c).

This study aims to determine the performance (i.e. the percentage of water flow through the gully grate) of a Green Gully model through experimental and computational techniques for different design and operating conditions. A three-dimensional model was developed to simulate experimental results using the computational fluid dynamics (CFD) packages GAMBIT and FLUENT. Experimental results are compared with simulated results and discussed.

**GREEN GULLY: DESIGN AND DEVELOPMENT**

In general, road gullies comprise gully grates (screens) and a drainage system that is usually situated at the side of the road by the kerb. The main purpose of the gullies is to take the surface water run-off from the road and to prevent any gross pollutants and sediment from being carried into the drainage system. The Green Gully, designed in this study, is an updated version of road gullies that direct water from a road gutter to the drainage channel by filtering out gross pollutants. It consists of a gully screen or a runnel with a V-shaped base wall for filtering litter from stormwater before it enters the diverter channel. It also includes an irrigation unit which directs stormwater to irrigate plants grown in the vicinity of the irrigation unit. Different components of the newly designed Green Gully are shown in Figures 1(a) and 1(b), which consists of:

- a channel member (‘3’ in Figures 1(a) and 1(b));
- a kerb member (‘4’ in Figures 1(a) and 1(b)) extending along one side of the channel member;
- a gully inlet (‘7’ in Figures 1(a) and 1(b)) formed within the kerb member and adjacent to the channel member. This gully inlet directs water into a stormwater drain;
- a diverter channel (‘11’ in Figure 1(b)) formed within a side wall of the gully inlet before the stormwater drain (the diverter channel provides an alternative passageway for the water);
- the gully inlet (an elongated opening);
- a filter (‘10’ in Figure 1(a)) associated with the diverter channel to prevent debris from entering the diverter channel (the filter includes a gully screen (‘8’ in Figures 1(a) and 1(b)) located at or adjacent to an opening in the diverter channel);
- a removable grill (‘6’ in Figures 1(a) and 1(b)) positioned behind the gully screen to alter the aperture of the gully screen (the grill has fan-like blades to direct water into the diverter channel);
- the filter, which includes an elongated and V-shaped base wall (‘13’ in Figure 1(b)) to collect and direct debris (the V-shaped base wall is located adjacent to the channel member);
- a runnel member (‘9’ in Figure 1(a)) located within the gully inlet. The runnel member directs water to the...
stormwater drain or diverter channel (the runnel member has an inclined base wall that slopes downward toward the stormwater drain or diverter channel; the base wall has a V-shaped portion for collecting and passing debris);

- a V-shaped side portion that supports one side edge of the gully screen, and the side wall (‘12’ in Figure 1(b)) of the gully inlet that supports the opposite side edge of the gully screen;

- the channel member which has the channel opening (‘6’ in Figure 1(a) and 1(b)), providing access to the stormwater drain (alternatively, the kerb members have a kerb opening providing access to the stormwater drain). A removable screen can be positioned over the kerb of the channel opening.

Figure 2 shows the schematic diagram of the newly designed Green Gully with its dimensions. The Green Gully model is made of black mild steel; its form is an irregular combination of rectangular and triangular shapes. The Green Gully was designed with the intention of allowing the maximum amount of flow to pass through the gully screen. The base of the left-hand side wall of the gully is inclined 0.5° downward from the base of the right-hand side wall.

The inlet width of the Green Gully is smaller than the middle width. This is so that the upstream flow, after entering into the gully, can spread over the gully middle area, and both water and litter can move to the left-hand side wall where most of the water will pass through the gully screen into the diverter channel and litter will be retained on the screen. There are two outlets provided with the Green Gully. The first outlet is straight from the inlet and the second outlet is 0.2 m down from the base of the Green Gully on the left-side wall. The second outlet is connected to the pipe provided for watering the roadside plants. The width of the first outlet ($Q_1$) is the same as that of the

Figure 2  | Schematic diagram of Green Gully model with dimensions.
gully’s inlet. The middle section of the gully is slightly inclined (0.5°) so that the maximum amount of water has a tendency to flow to the screen (Figure 3).

Three removable rectangular screens (210 mm × 90 mm; cross-diagonal, perpendicular and square apertures) were used to measure the variation in performance of the Green Gully in order to identify the operating conditions to achieve best performance. The varying aperture sizes are shown in Figure 4. The cross-diagonal, perpendicular and square aperture screens were made of black poly (plastic) 1 mm thick, brushed stainless steel 0.9 mm thick and stainless steel 1.58 mm thick, respectively.

**EXPERIMENTAL INVESTIGATIONS OF PERFORMANCE**

**Laboratory setup and procedures**

Experiments were done using three types of pollutant filtering gully screens: cross-diagonal, perpendicular and square.

Different flow conditions, such as using only freshwater flow and fresh water flow with mixed litter, were used to conduct experiments. Each experiment was done for two flume base angles: 0° – horizontal level, and 1° – inclined. Flume base angles of 2° and more were not considered in this study as it was observed during preliminary experiments that, at more than 1° only a small proportion of the runoff water flowed through the gully grate, which may not be acceptable to stakeholder in reality.

A schematic diagram of the experimental setup is shown in Figure 5. The model of the Green Gully was installed from the middle to the end section of the laboratory’s existing water flume. The flume length, height and width are 7.76, 0.45 and 0.297 m, respectively. Water is supplied to the flume for experiments by a 10HP pump. The flow pipe of the pump has a diameter of 16 cm with a capacity of 5 L/s.

The volumetric flow rate of water was measured at three positions to calculate the efficiency of the Green Gully. They are: flow at the Green Gully entry \( Q \) (i.e. total flow from the pump), flow downstream of Green Gully \( Q_1 \) and flow through the gully screen \( Q_2 \) (Figure 5). A Pitot static tube was used to determine the upstream fluid flow velocity. A differential manometer was used in the Pitot tube. \( Q \), \( Q_1 \), and \( Q_2 \) were measured for at least six different inflow rates. For each value of \( Q \), at least three sets of readings were taken at approximately three minute intervals, and the average of those were considered as a single value of data set.

The difference in height of the fluid in the manometer was converted to pressure difference using Equation (1). Velocity was calculated from the pressure difference using Equation (2). The upstream flow rate (\( Q \)) was determined by multiplying the velocity by the cross-sectional area.

\[
P = \rho gh = P_s - P_o \quad (1)
\]

\[
V = \left( \frac{2(P_s - P_o)}{\rho} \right)^{1/2} \quad (2)
\]

Thus:

\[
P = \rho gh = P_s - P_o = \sqrt{\left( \frac{2(P_s - P_o)}{\rho} \right)}
\]

where \( h \) = difference in height (m); \( \rho \) = density of water (kg m\(^{-3}\)) and \( g \) = gravitational force (m s\(^{-2}\)).
Flow through the gully screen ($Q_2$) was measured by collecting water in a tank for a certain time. Flow rate was calculated by dividing the volume of water by time. Flow through the gully downstream ($Q_1$) bypassing the gully trap (see Figure 3) was determined by subtracting $Q_2$ from $Q$. The efficiency of the device is defined as the ratio, expressed as a percentage, of the flow rate through the gully screen ($Q_2$) to the total flow rate ($Q$), given in Equation (3).

$$\eta = \frac{Q_2}{Q} \times 100$$

(3)

**Experimental results**

Laboratory experiments were done considering two flow conditions: using only freshwater flow and using mixed litter with freshwater flow. The results are discussed separately for each type of screen. The empirical formula for the trend line of each plot is shown in the respective figures below.

**Cross-diagonal screen, without litter**

Performance of the experimental Green Gully, i.e. water capture efficiency of Green Gully is plotted as a function of flow rate for flume angles of 0° and 1° (Figure 6). In general, efficiency (calculated using Equation (3)) decreases exponentially with increase in flow rate, as was predicted. Efficiency of approximately 46% was achieved at a flow rate of 8 L/s for both flume angles. However, with increase in flow rate, efficiency decreases more rapidly at a flume angle of 1° compared to a flume angle of 0°. At a flow rate of 35 L/s, efficiency decreases to 20% for a flume angle of 0°, while the efficiency decreases to 10% for a flume angle of 1°. Since there is no litter in the flow at the inclined
flume angle in this case, the water has a greater tendency to pass straight through in the channel and bypass the gully trap (lower efficiency) compared to the situation for the horizontal level flume base.

**Cross-diagonal screen, with litter**

Performance is plotted as a function of flow rate for flume angles of 0° and 1° (Figure 7). It can be seen from Figure 7 that, at a flow rate of 3 L/s, efficiency of 48% was achieved with a flume angle of 0° whereas efficiency of only 28% was achieved with a flume angle 1°. It can also be observed that efficiency decreases at a slower rate for a flume angle of 1° compared to a flume angle of 0° as the flow rate increases. Efficiency of about 10% was achieved at 15 L/s for a flume angle of 0°, whereas the same efficiency was achieved at 35 L/s for a flume angle of 1°. It can be seen from Figure 7 that about 22% efficiency was achieved for both flume angles (0° and 1°) at a flow rate of 9 L/s. During experiments it was observed that, at a flume angle of 0°, litter accumulated very quickly on the screen and formed a blockage so that water could not pass through the screen. As a result, the maximum volume of upstream water went straight through, bypassing the gully trap and flowing downstream in the kerb channel; hence the reason for efficiency dropping so quickly. On the other hand, at a flume angle of 1°, a small amount of litter stuck to the screen, mostly wet litter (dry litter flowed downstream with the water and bypassed the gully trap), thus explaining the reason for efficiency decreasing at a slower rate than for a flume angle of 0° as flow rate increases.

**Perpendicular screen, without litter**

Performance is plotted as a function of flow rate for flume angles of 0° and 1° (Figure 8). Efficiency was higher (53%) at a flume angle of 1° at a flow rate 6 L/s. The aperture size of the perpendicular screen is larger than that of the cross-diagonal and square screens. As a result, at a flume angle of 1°, when flow rate was increased the maximum volume of water passed through the screen. The lowest efficiency (20%) was achieved with a flow rate of about 32 L/s. Efficiency decreased exponentially with increased flow rate. Efficiency decreased (40% to 25%) at a flume angle of 0°, due to low flow velocity. At that time, most of the water flowed straight downstream bypassing the gully trap and very little passed through the screen.

**Perpendicular screen, with litter**

Performance is plotted as a function of flow rate for flume angles of 0° and 1° (Figure 9). The overall efficiency was
significantly low (18% to 5%) with a small difference in flow rate (5–18 L/s) at a flume angle of 0°. This was similar to what was found with the cross-diagonal screen at a flume angle of 0° (Figure 7). It was observed that, at a 1° flume angle, when litter reached the screen most of the wet leaves were broken and the fragments adhered to the screen and blocked it. Other litter such as cigarette butts and paper contributed to the blockage. As a result, efficiency was very low (10% to 7%). In the perpendicular screen the internal bars have sharp edges that catch the litter, and this increases the potential to form an obstruction on the screen. It is seen from Figure 9 that there is very little difference in efficiency at a given flow rate and across all flow rates studied (5–30 L/s).

Square screen, without litter

Performance is plotted as a function of flow rate for flume angles of 0° and 1° (Figure 10). Maximum efficiency was similar, 40 and 42%, for flume angles of 0° and 1°, respectively. Efficiency was slightly higher for the latter because the inclined flume base increased flow velocity. Minimum efficiency (24% to 22%) was achieved for flow rates in the range of 4–20 L/s. With the flume base horizontal level (angle of 0°), efficiency was relatively lower than for angle of 1° because of screen aperture arrangements. The screen with square apertures is relatively thick and appears this combination formed an obstacle to the flow. From Figure 10 it can be clearly seen that the efficiency vs. flow rate curve follows a similar trend for both 0° and 1° flume angles.

Square screen, with litter

Performance is plotted as a function of flow rate for flume angles of 0° and 1° (Figure 11). Efficiency at a flume angle of 0° varied significantly (48 to 11%) over a small range of flow rates (4–13 L/s); a similar trend was found in both the cross-diagonal and perpendicular screens (Figures 7 and 9). Efficiency decreased very quickly when litter was trapped on the screen by an increased flow rate. Wet litter, especially small leaves and cigarette butts, adhered to the screen very quickly and formed a blockage so that water could not pass through the screen. At a flume angle of 1°, efficiency varied from 25 to 7% for flow rates ranging from 5 to 18 L/s.

Comparison of results considering litter accumulation

Flume base angle plays a significant role in obtaining higher efficiency. A detailed analysis has been done using three types of gully screens (cross-diagonal, perpendicular and
square) for two flume angles considering water flow with mixed litter. Figures 12 and 13 compare the efficiency based on flume base angles of the experimental data.

Flume angle: 0

From Figure 12 it can be seen that the highest efficiency of about 48% was achieved with the cross-diagonal and square screens at about 3–4 L/s, and the lowest efficiency of 18% was achieved with the perpendicular screen at about 18 L/s. When the flume angle was 0°, the perpendicular screen was blocked by more litter than the cross-diagonal and square screens. Some of the litter was broken when it reached the perpendicular screen and blocked the apertures. With the cross-diagonal screen, however, the litter was at first jammed on the screen area but moved on downstream as water flow continued. In this case, the litter did not enter the screen, so a permanent blockage did not occur. It can be seen from Figure 12 that, for all the screens, efficiency decreased rapidly within a small range of flow rates.

Up to a flow rate of 5 L/s, both cross-diagonal and square screens performed equally, but in general the cross-diagonal screen performed better compared to the other two screens across higher (>5 L/s) flow rates.

Flume angle: 1°

In general as shown in Figure 13, the efficiency was lower at a flume angle of 1° than at a flume angle of 0°. At an angle of 1°, the highest efficiency achieved was 29%. It was observed that, when the flume base angle was set at 1°, flow rate increased rapidly. As a result, the maximum volume of flow passed straight through the gully directly downstream, and only a small volume passed through the gully screen. Again the cross-diagonal screen performed better across all flow rates studied.

Discussion and interpretation of experimental investigation

The experiments were done using different types of litter such as leaves, cigarette butts, bottle-caps, small plastic bags and papers as mixed litter. The litter showed different performance according to category and characteristics. Mixed litter was used both dry and wet. Wet litter stacks more quickly on the screen than dry litter. In the case of leaves, two types of leaves were used: large and small, but not any regular shape and size. Different leaves showed different behaviours, such as wet and small leaves being trapped in the screen more than dry and large leaves. It was found that small leaves create more blockage then large leaves. Most of the cigarette butts and bottle-caps did not get trapped on the screen. Plastics bags and papers were trapped moderately. Bottle-caps never created any blockage of the screen.

In summary, the efficiency of the experimental model varies with flow rate. For all screens and flume base angles it was found that mixed litter decreased the efficiency when it was mixed with water flow. The cross-diagonal screen at a flume base angle of 0° achieved the highest efficiency, while the perpendicular screen performed well when no litter was mixed with the flow. It is also suggested...
that the material of the cross-diagonal screen (black poly) contributed to its superior performance. The maximum efficiency of 46% was achieved when experiments were done with only water flow, and 48% was achieved when experiments were done with mixed litter in water flow. The flume base angles play a very important role in achieving the highest efficiency. The highest efficiency was achieved with a flume base angle of 0° (horizontal level). For flume base angles of 1°, the majority of the water continued straight through downstream from the gully trap, hence resulting in lower average efficiencies.

The water capture efficiency of Green Gully generally indicates the potential of recycling/reuse of stormwater. That is for a low rainfall event where stormwater flow rate is low (<1 L/s for the cross-diagonal screen). The implementation of the Green Gully installed horizontal to road surface is more appropriate for capturing more stormwater, i.e. possibility of more recycling/reuse of stormwater. In the case of the cross-diagonal screen, Green Gullies can be installed 1 km apart each other for a standard two-lane road. On the other hand for high rainfall events where stormwater flow rate is higher (>9 L/s, with cross-diagonal screen), the use of the inclined Green Gully (i.e. Green Gully inclined 1° to the surface of the road) is recommended for capturing more water for recycling/reuse. In this case, installation of Green Gully at 500 m apart seems appropriate. Alternatively, installation of a longer (twice than the one installed here) Green Gully may be more appropriate. If road is wider, i.e. four-lane road, in a heavy rainfall area, installation of more Green Gullies may be required. Similar explanations will be applicable for the Green Gully with a perpendicular or square screen for both 0° and 1°. To confirm the better performance of the cross-diagonal screen, a CFD simulation was done to validate the experimentally measured data using only fresh water for flume base angles of 0° and 1°.

COMPUTATIONAL INVESTIGATIONS OF PERFORMANCE

A three-dimensional model was developed for simulation purposes to predict the efficiency of the Green Gully as a function of flow rate. The commercial CFD packages FLUENT 6.3.26 and GAMBIT 2.3.16 geometry/meshing software were used for model development.

Governing equations for simulation

The basic governing equations for any single phase, non-reacting fluid process are the Navier–Stokes equations, i.e. conservation of mass (Equation (4)) and momentum (Equation (5)). These equations were solved.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{U}) = 0 \tag{4}
\]

\[
\frac{\partial \mathbf{U}}{\partial t} + \mathbf{U} \cdot \nabla \mathbf{U} = -\nabla p + \nu \nabla^2 \mathbf{U} + \mathbf{g} \tag{5}
\]

The standard k-ε model was used in this study because this is the simplest complete model of turbulence in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. Robustness, economy and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations.

The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). It is a linear eddy viscosity model in which k, the turbulence kinetic energy per unit volume, and ε, the turbulence kinetic energy dissipation rate per unit volume, are solved by transport equations in order to determine the turbulence velocity and length scales respectively. The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. The standard k-ε model is therefore valid only for fully turbulent flows. In this study, the flow was fully turbulent (Reynolds number, Re, 9.23 × 10^4). The governing equation of the standard k-ε model can be written as follows (Fluent 2005):

For turbulent kinetic energy, k:

\[
\frac{\partial}{\partial \xi} \left( \rho k \right) + \frac{\partial}{\partial \xi_i} \left( \rho k u_i \right) = \frac{\partial}{\partial \xi_i} \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial \xi_i} + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{6}
\]
For dissipation, $\epsilon$:

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_i} \left( \mu + \mu_t \frac{\partial \epsilon}{\partial x_i} \right) + C_1 \epsilon \left( G_k + C_3 \epsilon G_b \right) - C_2 \epsilon \frac{\epsilon^2}{K} + S_\epsilon$$

(7)

The turbulent (or eddy) viscosity, $\mu_t$, is computed by combining $k$ and $\epsilon$ as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\epsilon}$$

where $C_\mu$ is a constant. The model constants were taken from the default values, thus:

$$C_\mu = 0.09; \quad C_{1e} = 1.44; \quad C_{2e} = 1.92; \quad \sigma_k = 1; \quad \sigma_\epsilon = 1.3.$$

Model geometry

A rectangular part (outlined in red in Figure 14) of the Green Gully laboratory model was considered for modelling to avoid complications of geometry. To allow only a part of the Green Gully model to be used for simulation, total inflow and outflow was checked with that of the full model. The geometry of the computational domain is shown in Figure 15. GAMBIT 2.3.16 was used to create the model geometry. The geometry was created with the original dimensions. The length and width of the computational domain were 1 and 0.6 m, respectively.

Water and air zones were considered separately at the inlet (inflow, $Q$) of the computational domain as the simulation process involves the interaction between air and water. As shown in the Figure 15, the zone of the upper section at the inlet is air and the lower section is water. The top of the model was covered with a surface to create a volume for 3D model development. The $Q_2$ outflow location (i.e. the screen area) was considered to be a porous medium. According to CFD phenomena, the porous-jump zone must be a type of internal face zone. So the screen area was considered as a porous medium, and the screen outlet (for outflow, $Q_2$) was considered as being 2 mm away from the porous medium (shown in Figure 15).

Two volumes were created to generate volume mesh. The first volume included the entire model except the screen outlet area. The second volume includes the extended part for the screen outlet (outflow, $Q_2$).

Meshing the geometry

As the domain being considered is 3D, volume meshing was applied for meshing the geometry. Volume meshing of the computational domain was done by using a constant interval size option. The element type was considered as Tet/Hybrid, and the TGrid mesh type was used because the mesh is composed primarily of tetrahedral elements that include hexahedral and wedge elements. An interval size of 0.013 was used for meshing. The mesh of the computational domain is shown in Figure 16. The mesh was examined for the presence of any distorted elements, which may be occur because of improper interval sizes for different edges or by not choosing the proper mesh type for a given geometry. A good
mesh should not contain any distorted elements as the CFD analysis can be used only for regular element shapes. The mesh was then exported to FLUENT. The generated mesh contained a total of 73,183 nodes and 396,568 elements.

Grid independency

A study was performed to check the accuracy of the grid and to establish the cell size. Three grids were generated using different interval sizes. Each of the grids were exported to FLUENT. Simulation was performed for each grid for the screen outflow \( (Q_2) \). The predicted outflow for the three grids were compared. It was observed that the difference of predicted flow rate was less than 10% which supported the established grid. A comparative statement of grid independency check outcomes is shown in Table 1. Grid of interval size 0.013 with 396,568 elements was selected for simulation from the point of view of accuracy and computational time. The calculation time for the selected grid was approximately 48 hours for 1,500 iterations.

<table>
<thead>
<tr>
<th>Interval size</th>
<th>No. of elements</th>
<th>Variation of simulation results</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.014</td>
<td>3,193,70</td>
<td>9.6%</td>
</tr>
<tr>
<td>0.013</td>
<td>3,96,568</td>
<td>0%</td>
</tr>
<tr>
<td>0.0125</td>
<td>4,31,565</td>
<td>7.8%</td>
</tr>
</tbody>
</table>

Boundary conditions

Boundary types were defined in GAMBIT. Following these boundary types, boundary conditions with required parameters were applied in FLUENT. They are:

- There were two outlets (flow straight through to downstream of the gully trap and flow to the gully screen) and one inlet considered in this study. The boundary conditions at the inlet (for water and air separately) and outlets were specified by magnitude of velocity and volume flow rates. Other parameters such as specification method, turbulent intensity and hydraulic diameter were given as inputs for the corresponding models. The pressure outlet boundary condition is specified for the screen and the opposite side of inlet which are open.

- Wall boundary conditions were used for sides, bottom and top of the domain.

- In GAMBIT, the screen area was declared as an internal boundary type as there was no option for porous medium. After exporting to FLUENT, the screen area changed from internal to porous-jump.

Multiphase model

This study considered water and air flow as multiphase flow. In order to predict the free surface of water and air, the volume of fluid (VOF) method (Hirt & Nichols 1981) in FLUENT was used. The VOF formulation was designed for cases where there two or more immiscible fluids were used. The method tracked the volume fraction of each phase in each computational cell. The fields for all variables and properties were shared by the fluids present in each cell, and represent volume-averaged values. The tracking of the interfaces between the phases is computed when the VOF model is used. The tracking of the interface(s) between the phases is accomplished by the solution of a continuity equation for the volume fraction of one (or more) of the phases (FLUENT 2005).

Computational techniques

Using FLUENT, the solver was selected with default parameters (such as Pressure based, Implicit, 3D, Unsteady). As the flow considered was multiphase, the VOF model with
two phases including 'Implicit Body Force' was selected from the Multiphase option. Regarding the model selection, $k-\epsilon$ (standard) was considered. Other model parameters (i.e. Standard wall functions and Model constants, such as $C_{mu}$: 0.09, C1-Epsilon: 1.44, C2-Epsilon: 1.92, TKE Prandtl Number: 1 and TDR Prandtl Number: 1.3) were left at default. In the Materials option, water and air were added with relevant parameters of density and viscosity. Operating condition was set to default (i.e. Operating Pressure: 101,325 pa and reference pressure location: 0, 0, 0 for $x$, $y$, $z$, respectively). Boundary types were determined in GAMBIT and boundary conditions were applied with parameters in FLUENT. In the Solutions panel, some problem solving techniques were applied for this particular type of problem. One such technique is the relaxation technique, which was used only for solving elliptic partial differential equations. The relaxation technique used the finite difference method and may be either explicit or implicit. Another technique which was used to solve the Navier–Stokes equations is the pressure correction technique, which involved the calculation of a guess velocity field based on the guess pressure. The corrected pressure was then obtained from the continuity equation and the velocity field values were updated based on the new pressure. The pressure–velocity coupling method was selected as SIMPLE (Semi Implicit Method for Pressure Linked Equations) scheme (Patnakar 1980). A region was marked to identify the water and air flow. At the end of the model setup, iteration was done by 1e-06s time steps; the maximum iteration per time step was 10. A minimum of 1,500 time steps were done for each simulation.

**Post-processing/simulation**

Post-processing is one of the targeted goals of CFD simulation. Once the equations are solved to obtain the flow variables, they need to be analysed to see if they meet the requirements of the process such as velocity profiles. There are several kinds of plots in CFD analysis; a few of them are listed below.

**Contours**

Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, etc.). A profile plot draws these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface (FLUENT 2005). This type of plot represents the global nature of the variables and can be used for both vector and scalar quantities. The contour of volume fraction of different phases (water and air) at the inlet and centre-plane (at $y = 0.2$) of the rectangular duct is shown in Figure 17. Water and air volumes are shown in red and blue colours respectively.

**Velocity vector**

Vector distribution plots present both the direction and magnitude of the velocity. As the name proposes, these types of plots can be used to represent only vector quantities. The velocity vectors at the outlet screen ($Q_2$) are shown in Figure 18. The screen face is represented by the red colour. The velocity vectors were drawn for the phase as a mixture using auto scale and auto range. From Figure 18, it is clear that water passes through the screen. The screen was defined as a porous medium in FLUENT. The face permeability was calculated as 1e-08 m$^2$ using Equation (8) and the porous medium thickness was measured as 0.001 m.

$$\Delta p = -\left(\frac{\mu}{\alpha} + C_2 \frac{1}{2} \rho \nu^2\right) \Delta m \quad (8)$$

where $\mu$ is the fluid viscosity, $\alpha$ is the face permeability of the medium, $C_2$ is the pressure-jump coefficient, $\nu$ is the velocity.
normal to the porous face and $\Delta m$ is the thickness of the medium.

Velocity magnitude and direction of flow were clearly identified by the colour and arrows. It is clearly seen from Figure 18 that there was no reverse flow and the flow behaviour was good. The magnitude shows a small variation because flow was interrupted by the porous medium (screen area) that lowers the velocity. The model was optimised by changing parameters in the relaxation factor and residual monitors.

**RESULTS AND DISCUSSION**

Simulation was performed for different velocities at flume base angles of 0° (horizontal level) and 1° (inclined) considering only freshwater flow through the computational domain. Volume flow rates were obtained from the simulation. Efficiency of the Green Gully screen was determined with respect to total flow rate and compared with experimental data. Efficiency vs. flow rate for experimental and predicted data is compared in Figures 19 and 20 for flume base angles of 0° and 1°, respectively.

Average values were taken from three measurements to plot the experimental data in Figures 19 and 20. Error bars have been plotted using the range of maximum and minimum results of experimental measurements. The deviation between the predicted result and the measured data are in the range of 3–10% for a flume base angle of 0° and 1–16% for a flume base angle of 1°. It is clear from Figures 19 and 20 that the predicted results follow similar trends to the experimental results. In the case of a flume angle of 0° (horizontal level), predicted efficiency was very close (3–10%) to experimental efficiency. For a flume angle of 1°, predicted efficiency was within a good range (1–16%) of experimental efficiency.

Predicted results were found to be in very good agreement with experimental data for the efficiency of the Green Gully screen (Figure 19 and 20). It is therefore suggested that the experimentally measured efficiency represents the actual efficiency of the Green Gully.

**CONCLUSIONS AND RECOMMENDATIONS**

This article contributes to the understanding of gully screens in relation to water flow and to screen type and material. Tests were conducted on a laboratory model of the Green Gully. The article presents the performance results of three
different screens, tested with freshwater and water mixed with litter. The performance of the screens is reported as a function of flow rate. The experiment was conducted for flume angles of 0° and 1°. It is concluded that the efficiency of the experimental model varies with flow rate. The cross-diagonal screen at a flume angle of 0° achieved the highest efficiency, while the perpendicular screen performed well when no litter was mixed with the flow. It is also suggested that the material of the cross-diagonal screen (black poly) contributed to its superior performance.

To confirm the better performance of the cross-diagonal screen, a CFD simulation was performed to validate the experimentally measured data using only freshwater for flume base angles of 0° and 1°. A three-dimensional CFD model was developed using CFD code FLUENT. Geometry creation and mesh generation were done using GAMBIT. The standard k-ε turbulence model was used to determine the volume flow rate through the Green Gully screen. The VOF model was used to predict free surface of multiphase (water and air) flow. The efficiency was predicted from the volume flow rate through the screen with respect to total inflow to the gully. Experimentally measured velocity was used at the inlet boundary for an accurate and realistic flow simulation. Predicted results were compared with experimentally measured results. It was found that the simulated results are in good agreement with the experimental results. Therefore the model developed and simulation procedure followed can be used to determine the optimum design of the gully for achieving best performance. The CFD model, robust and reasonably accurate, will be a useful tool for designing and optimising the operation of the Green Gully.

Further research is required to improve and optimise the design of the Green Gully to achieve better performance. Several areas were identified for further study as follows:

- Experiments can be done using other types of litter (such as discarded cups, pet waste, chips and chocolate wrappings, shopping/plastic bags, pesticides and fertilisers, rubber and metal deposits from tyre wear, etc.) in mixed form.
- Simulation can be done for the full geometric model of the Green Gully.
- A discrete phase model can be applied for considering litter with flow in FLUENT.

**REFERENCES**


James, D. 1994 *Using Economic Instruments to Control Pollution in the Hawkesbury-Nepean.* Environment Protection Authority, NSW.


First received 6 February 2011; accepted in revised form 20 May 2011