An experimental investigation of saline intrusion in a long sea outfall

PA Mackinnon, NR Shannon, GA Hamill & BM Doyle
School of Civil Engineering, Queen's University Belfast, United Kingdom

Abstract

The discharge and subsequent dispersion of wastewater through long sea outfalls is often inhibited by the intrusion of saline receiving water into the outfall diffuser. Until recently, numerical modelling of intrusion and purging of saline water has been limited to one-dimensional (1-D) and enhanced 1-D models, which do not adequately describe the complex hydrodynamic processes generated within these devices. Recent research [1] has resulted in the development of a two-dimensional (2-D) numerical model to simulate the internal hydraulics of a long sea outfall with a saline intrusion. The model, which was developed using the FLUENT computational fluid dynamics (CFD) package, is designed to reproduce the effects of buoyancy and stratification, so that the interaction between the saline sea water and the effluent may be more accurately predicted.

In this paper, details of an experimental investigation to assess the performance of the numerical model are presented. The investigation used a 1 in 30 scale model of a typical modern outfall diffuser discharging, under simplified conditions, into still receiving waters. Information is provided on model scaling, experimental facilities, instrumentation, and measurement procedures. The paper documents the results of an initial test series, in which discharge was permitted through only one of the outfall’s four vertical risers. The results include measurements of velocity and water density within the model outfall as the conditions change from a steady state with the outfall fully intruded, through the transitory purging period, to normal operation.

A brief overview of the numerical modelling technique is provided and a comparison of the results obtained using the physical and numerical models is presented.
1 Introduction

1.1 Saline intrusion in long sea outfalls

Long sea outfalls are now widely used for the discharge of treated waste water into coastal receiving waters. However, despite their apparent simplicity (they normally consist of a pipe with a diffuser to distribute the effluent at the discharge location), they exhibit some complex hydraulic characteristics. Since it was first recognised in the 1970s, the problem of saline intrusion in operational long sea outfalls has been investigated by various researchers. In its simplest form, known as 'primary intrusion', it is essentially the blockage or partial blockage of the outfall pipe by the saline sea water in the period between successive discharges of waste water. However, a more complex and detrimental form, known as 'secondary intrusion', can result in the continuous ingress of saline water into the outfall pipe, even while waste water discharge is under way. The latter may cause substantial changes in the outfall's discharge capacity and is therefore an important concern for design engineers and outfall operators alike.

1.2 Assessment of saline intrusion for design purposes

Early guidelines addressing the internal hydraulics of such systems, including those published by Fischer [2] were followed by investigations by Wilkinson [3], Burrows and Davies [4], Charlton [5] and Munro [6]. Later work by Mort [7] and by Larsen, Burrows and Engedahl [8] and Guo and Sharp [9] addressed the possibility of numerical modelling to solve the problem. Although progress was made, these last authors concluded that a one-dimensional model was insufficient to adequately describe the mixing processes that take place within a long sea outfall. However, recent developments in numerical modelling have greatly increased the capability of the engineer to simulate complex hydraulic processes. It appears that some of the latest techniques may have the potential to overcome the difficulties previously encountered in modelling saline intrusion.

1.3 CFD modelling of saline intrusion

Using commercial computational fluid dynamics (CFD) software, the authors of the current document set out to evaluate this potential. This paper relates to the validation of a numerical model of flow within a long sea outfall, describing preliminary experiments carried out to assess the overall performance of the model. The following sections detail the numerical model, the physical model and the laboratory facilities used for physical tests. The outcome of selected tests is also reported.
The initial stages of the research involved the development of a two-dimensional numerical model to simulate the internal hydraulics of an operational long sea outfall. The model was created using the FLUENT/UNS CFD package [10], which utilises the finite volume method. After manual specification of the main geometrical features, a non-uniform mesh of cells was generated to describe the overall geometry of a simplified long sea outfall. The preBFC pre-processing package [11] was used during mesh generation.

Boundary conditions were carefully selected to replicate the conditions within the physical model (described in section 3.3). The distant limits of the receiving waters were modelled as hydrostatic pressure 'outlets' which, despite their name, allowed flow in and out of the model domain. The water surface was modelled using a 'symmetrical boundary' which prevents flux of any parameter across it. The pipe walls and tank floor were specified as 'wall shear factors'. The inlet to the diffuser was specified as a 'velocity inlet'. The turbulence characteristics were also defined as part of the velocity inlet boundary condition.

The modelling of fluids of different density (the waste water within the outfall and the saline receiving water) was achieved by using different 'chemical species'. The model permitted simulation of both laminar and turbulent conditions. A detailed description of the model is given by Shannon et al [1].

The numerical model was used to predict the flow patterns that would occur within a simplified long sea outfall, the geometry of which is described in detail in Section 3. The results of numerical tests are presented in Section 4 along with the results of corresponding physical tests.

3 Physical modelling

3.1 Test facilities

The performance of the numerical model was evaluated by comparing the results with equivalent data obtained using a physical model. The facilities used for physical modelling consisted of a 19.5m flume, rectangular in section with a width of 0.75m. The maximum depth of water in the flume was 0.7m. The flume permitted different combinations of fresh and saline water flows but, for the purpose of the initial tests described in this document, it was used to create a reservoir of stationary saline water.

3.2 Model scaling

For this particular investigation, a physical model was designed by hydraulic scaling from a recently designed prototype outfall. Due to practical constraints imposed by the testing facilities, the number of diffuser ports was reduced from sixteen (in the prototype) to four (in the model). In order to recreate the key
physical processes of intrusion and purging [7], it was considered important to maintain similarity (in the model and prototype) of the densimetric Froude number:

$$\text{Fr}_d = \frac{v}{\sqrt{\varepsilon gD}}$$

where $v$ is velocity, $g$ is acceleration due to gravity, $D$ is the pipe diameter, $\varepsilon$ is defined by:

$$\varepsilon = \frac{(\rho_r - \rho_e)}{\rho_r}$$

and $\rho_r$ and $\rho_e$ are the densities of the receiving water and the effluent respectively. The design resulted in a model approximately thirty times smaller than the prototype.

### 3.3 Model outfall

![Physical model of outfall diffuser](image)

The model outfall consisted of a 3 inch diameter circular PVC pipe, leading into a 110mm by 110mm square perspex duct which represented the diffuser. The square section was chosen to overcome potential difficulties with the instrumentation system (which is discussed later). The diffuser had four vertical risers, each one square in section with internal dimensions 50mm by
The risers were spaced at 450mm centres, and extended 350mm vertically from the diffuser centreline. The depth of water over the diffuser ports was initially set at 200mm, but varied between this and 270mm depending on the specific test under way. For the purpose of the first series of tests (described in this document), the three outermost risers were sealed using perspex plates. Figure 1 shows the model within the flume.

During the experiments, three parameters – pressure, velocity and density – were used to quantify the flow characteristics. Pressure measurements were carried out using a series of transducers fitted to the model at key positions, which were identified in advance of physical tests after studying the results from the numerical model. Velocities were recorded using a two-component Laser Doppler Anemometer (LDA) with a fibre optic fitting. In order to measure density, salinity was monitored at various locations inside the model using twin wire conductivity probes. In initial tests, powder dye was used as a tracer to highlight the water discharged through the outfall and also to provide seeding for the LDA.

As freshwater and wastewater have similar density characteristics, freshwater was used to simulate a discharge of domestic effluent through the model outfall. In order to control the flow, the freshwater was allowed to gravitate through an electrically actuated butterfly valve close to the outlet of a small header tank. The header tank was connected to the mains supply and the water within it was maintained at a constant level, approximately 1m above the water level in the flume. Figure 2 illustrates the hydraulic system used. Measurements were recorded using a Viglen 486 DX266 PC running Signal Centre data logging software. Each of the signals was scanned at a frequency of 10 Hz.

![Figure 2: Schematic diagram of flume and model](image-url)
3.4 Conditions at start of tests

In each test, the ambient salinity of the receiving water was adjusted to 1018 kg/m³ at the start of the test. As mentioned earlier, the water discharged through the model outfall was taken from the mains supply. Its salinity was not adjusted, but was normally found to be around 1003 kg/m³. At the outset of each test, the model outfall was completely intruded by saline water.

4 Results of physical and numerical modelling

4.1 Experimental test under low flow conditions

The first of various tests was carried out under conditions designed to create laminar flow. A quantity of 0.2 l/s was discharged continuously through the outfall. Saline intrusion was observed in the single operational riser and in the outfall pipe delivering water to the riser. The flow was insufficient to purge the saline water fully, so a degree of intrusion persisted throughout the test.

4.2 Comparison with laminar model

A corresponding test was carried out using the numerical model. For this particular test, the flow was specified as laminar. The results showed general agreement with the main events of the experimental test. Figure 3 shows modelled contours of density and the corresponding observed values under steady state conditions, 600 seconds after the test commenced. The figure provides an overall visual illustration of the capability of the model to reproduce the main features of primary saline intrusion. Although the trends shown in this instance are comparable, the phase and duration of the changes that took place prior to this showed some significant differences.

Figure 3: Contours of density 600 seconds after start of test
Figure 4 shows the temporal changes in density at the location where the base of riser joins the main pipe soffit. The results show the density of water within the physical model to be higher than the initial condition specified in the model. Despite careful control of the density in the receiving water, this problem occurred with some persistence during the test programme. It was found to be caused by stratification generated while waiting for the saline intrusion to establish. The subsequent results from both the numerical and physical models show a rapid drop in density as the saline/freshwater interface passes the measurement location. Both models show this to happen concurrently, but the numerical model showed the change to be more sudden. This is thought to be related to assumptions regarding turbulence, and is discussed again in section 4.3.

Despite the transient discrepancies in density, the measured and modelled velocities within the outfall pipe show remarkable agreement during the period when the saline/freshwater interface passes a given location. Figure 5 shows a profile of the longitudinal component of velocity at a section approximately 100mm upstream of the riser. The profile shows the distinct differences in velocity between the upper and lower layers of the water. Both the model and the measured values show the outward flow of freshwater close to the top of the pipe. The results obtained using both methods show good agreement regarding the extent of this outward flow. The modelled values show a small but significant upstream flow of saline water close to the base of the pipe; this suggests a form of secondary intrusion. The measured values show this to be much less significant. This is probably due to turbulent mixing, a factor not represented in the laminar numerical model.
Distance from invert of outfall pipe

Pipe soffit

Pipe invert

Velocity component (m/s)
(positive indicates outward flow)

■ measured value
◆ value from numerical model

Figure 5: Longitudinal velocity component approximately 100mm upstream of riser at 150 seconds after start of test

4.3 Use of turbulence model

On further examination of the results, the assumption of laminar flow was found to be inaccurate for this particular test. Due to the extent of stratification in the outfall pipe, the freshwater flow was restricted to the pipe soffit, hence reducing the effective diameter and increasing the Reynolds number. Further examination of the flow confirmed the presence of eddies at the saline/freshwater interface, with turbulent effects transmitted to other areas of the flow. Further numerical tests were therefore performed, on this occasion incorporating turbulence in the flow. For the purpose of numerical modelling, the turbulence intensity was specified using values measured in advance within the main outfall pipe.

The inclusion of turbulence within the numerical model resulted in increased mixing within the outfall pipe. This was evident in the density versus time plot (the equivalent of Figure 4), in which the numerical values decreased much more gradually than in the laminar test. It also affected the predicted velocities, reducing the magnitude and extent of reverse flow, so that the profiles were closer to those recorded. Initial tests showed, however, that the model was over estimating the turbulent effects and therefore some further refinement of the method is required.
5 Discussion

As with any experimental programme, the repeatability of tests proved to be an important factor in establishing the validity of the results. In this investigation, it was particularly significant as temporal changes in velocity could only be established for one location during any test. Velocities recorded in successive tests at different locations could therefore be affected by changes in the flow patterns from one test to another.

Where experimental conditions were rigorously controlled, successive tests were found to be repeatable, but this was dependent on the accuracy of the flow and the ambient conditions. Changes in density of as little as 5 per cent were found to produce differing flow patterns. Likewise, changes in flow of 10 per cent were found to affect the intrusion process. As a result, extreme care had to be taken in preparing the experimental facilities for each individual test. The most problematic feature in this respect proved to be the adjustment of the partially automated inlet valve to achieve accurate flows at such low values.

Pressure measurements also created some difficulty, in that the arrangement used (which measured differential rather than absolute pressure) was relatively insensitive to rapid fluctuations. Despite this, the trends produced in pressure measurements were useful in establishing similarity between experimental and numerical models.

The preliminary laminar and turbulent tests examining flow in a single riser were useful in establishing the capabilities and limitations of the numerical model with regard to the main processes encountered in the physical model. The results of the early turbulence modelling showed that it had a significant effect on the predominant flow patterns within the outfall, affecting the generation of secondary intrusion.

Since recording data on flow through a single operational riser, further physical and numerical tests have shown the capability of the numerical model to reproduce the more complex conditions that occur when two or more risers operate. Secondary intrusion has been identified in both physical and numerical results.

6 Conclusions

The results of detailed measurements of flow patterns within a simplified long sea outfall are reported. The observed values confirm that previous assumptions of uniformity of flow across the outfall section are unlikely to be adequate when carrying out numerical modelling of such a device. The results from the reported experimental programme have been used as part of an extensive assessment of the validity of a new two-dimensional model of saline intrusion and purging in a long sea outfall.
7 Acknowledgements

The authors wish to acknowledge the contribution of the Department of Education for Northern Ireland in providing sponsorship for this research.

References


