Modelling of saline intrusion in a long sea outfall
N.R. Shannon, P.A. Mackinnon & G.A. Hamill
Department of Civil Engineering, Queen’s University Belfast, BT7 1NN, United Kingdom
E-Mail: p.mackinnon@qub.ac.uk

Abstract

In this paper, the results of an investigation into the internal hydraulics of a long sea outfall are presented. The investigation focuses on the mixing and stratification of the saline receiving water and the discharged wastewater within the outfall diffuser. The research methods include a combination of numerical modelling and laboratory testing.

The paper provides details of an enhanced numerical model of outfall operation. The model is designed to reproduce the internal hydraulics of the outfall including buoyancy and stratification, so that the interaction between the saline sea water and the effluent may be predicted.

The numerical model has been developed using the FLUENT/UNS computational fluid dynamics (CFD) package. Mixing and transport of the two fluids at the saline/wastewater interface are modelled using ‘chemical species’, which simulate convection and diffusion on a molecular level.

The paper refers to physical model tests carried out using a scale model of a typical modern outfall diffuser discharging, under simplified conditions, into still receiving waters. Also included is a discussion on the dominant physical processes and the methods of scaling used to recreate saline intrusion under laboratory conditions.
1 Introduction

1.1 Saline intrusion in long sea outfalls

The discharge of wastewater through long sea outfalls is often inhibited by the intrusion of the saline receiving water into the outfall diffuser. 'Primary intrusion', a transient feature, may occur naturally when the wastewater discharge ceases and the higher density of the saline receiving water causes it to enter the outfall pipe. When pumping resumes, the performance of the outfall is impaired until the saline water is fully displaced (a process known as 'purging'). A more serious problem, known as 'secondary intrusion', sometimes occurs when the discharge resumes. Secondary intrusion results in the continuous ingress of saline water through some of the diffuser ports, and can cause a significant reduction in operational efficiency. In the case of either primary or secondary intrusion, the rapid displacement of the saline water is desirable.

1.2 Processes affecting intrusion

Previous investigations [1, 2, 3] have revealed that the occurrence of saline intrusion depends on various factors, including the geometric configuration of the outfall, the physical characteristics of the effluent and the receiving water, and the discharge pattern. The ease with which purging is achieved depends largely upon the ability to break down the saline/wastewater interface which forms within the outfall [4]. The main processes are reported to be diffusion and turbulent mixing [5]. Although engineers are now familiar with these phenomena, and have developed some means by which intrusion can be overcome, designs are largely based on empirical methods.

1.3 Previous models of saline intrusion

Previous numerical models [6, 7] describe the operation and hydraulic characteristics in one dimension. These models include two fluids of different densities. However, they do not include provision for the formation of an interfacial mixing layer. In a later model [8], mixing is incorporated by using a localised two-dimensional method. The authors report that the model does not fully address stratification and mixing.
1.4 Aims of the investigation

Although progress has been achieved in the modelling of saline intrusion and purging, it appears that a model which is capable of reproducing stratification within the outfall would provide a valuable contribution towards understanding of the flow mechanisms.

The aims of the investigation reported in this document were: to observe and provide a detailed quantitative description of saline intrusion and purging within a physical model of a long sea outfall, and to simulate in further detail the key processes of intrusion and purging using a numerical model. The ultimate aim of the research is to provide a practical means of predicting and monitoring hydraulic performance of full scale outfalls in operation using computational fluid dynamics techniques.

2 The numerical model

The specific aims of the numerical model were: to simulate the key processes in saline intrusion and purging; to quantify the changes in pressure, velocity and density that influence these phenomena; and, in doing so, to define locations of specific interest in the outfall diffuser. The performance of the numerical model is currently being evaluated using data collected from the corresponding locations on a physical model.

2.1 Computational method

2.1.1 The finite volume method

The numerical model was created using the FLUENT/UNS CFD package [9], which is based on the finite volume method. In this method, the domain of flow is divided into cells, known as ‘control volumes’. The equations governing fluid flow are integrated over each control volume, allowing definition of parameters such as pressure and velocity within individual cells. Unknown properties, such as convection and diffusion, are estimated to give a system of algebraic equations which are then solved iteratively. Versteeg [10] provides further detail on the finite volume method.

2.1.2 Species modelling

The modelling of ‘chemical species’ (or different fluid types) is carried out by incorporating a set of governing equations for each species to be modelled in any cell. Each equation includes a term describing the fractions of the different species in that cell. The sum of the mass fractions for each species within a cell is equal to unity.
When two or more species mix, a term in the convection-diffusion equation is used to define the mass flux (between adjacent cells) due to diffusion. This diffusive mass flux, $J$, is calculated using equation (1):

$$ J = -\rho D \frac{\partial m}{\partial x} \quad (1) $$

where $\rho$ is the species density, $m$ is the mass fraction of the species, $x$ is distance in one direction and $D$ is a user-defined diffusion coefficient (sometimes known as the mass diffusivity).

2.1.3 Modelling of turbulence
When the flow is turbulent, additional terms relating to the turbulence intensity $\mu_t$ and turbulent Schmidt number $Sc_t$ [9] are required to establish the diffusive mass flux term. Equation (2) shows the extended formula used for turbulent flow:

$$ J = - \left( \rho D + \frac{\mu_t}{Sc_t} \right) \frac{\partial m}{\partial x} \quad (2) $$

Additional terms may be incorporated into these equations to take account of any depletion of a species by chemical reaction, creation from a source or diffusion by temperature gradients.

2.2 Application details

The geometry of the numerical model and the flow characteristics used in tests were designed to match those of a physical model of the outfall. The model was constructed to operate within 19.5m flume, rectangular in section with a width of 0.75m and a maximum water depth of 0.7m.

2.2.1 Model scaling
The 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 [11], it was considered important to
maintain similarity (in the model and prototype) of the densimetric Froude number:

$$Fr_d = \frac{v}{\sqrt{egD}}$$  \hspace{1cm} (3)

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

$$\varepsilon = \frac{(\rho_r - \rho_e)}{\rho_r}$$  \hspace{1cm} (4)

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.

2.2.2 Model geometry

Figure 1 shows the general layout of the numerical model. The model outfall included a 110mm by 110mm square duct which represented the diffuser. The square section was chosen to overcome potential difficulties with the instrumentation system used in physical model tests. The diffuser had four vertical risers, each one square in section with internal dimensions 50mm by 50mm. 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 to prevent flow through these openings.

Figure 1: General layout of numerical model
2.2.3 Mesh generation
The mesh for the numerical model was defined using the preBFC pre-
processing software package [12]. A preliminary outline of the diffuser
and receiving water body was created using points and curves. Node
points were used to control volumes; these were concentrated in the areas
of most interest. Interpolation was used to avoid large changes in area
between adjacent cells. A section of the mesh generated is shown in
Figure 2.

![Figure 2: Section of mesh for numerical model](image)

2.2.4 Boundary conditions
The boundary conditions in the model were designed to replicate those used
in the physical model. The limits of the expanse of saline receiving water
were modelled as hydrostatic pressure outlets. These outlets were placed a
sufficiently far from the diffuser section that they had negligible effect on
the solution.

The free water surface was modelled using a ‘symmetrical’ boundary
condition which gives zero flux of all quantities across it. The other
features modelled (including the outfall pipe, the wall thickness and the
roughness) correlated directly with the equivalent features in the physical
model. The model boundaries are shown in Figure 1.
2.2.5 Physical parameters
The species used in the model were defined as freshwater and saltwater. Densities of 1003 kg/m$^3$ and 1018 kg/m$^3$ respectively were specified as input to the model. The dynamic viscosity was defined as 1.1 x 10$^{-3}$ N.s/m$^2$ and appropriate relative viscosities for the different species were computed within the model.

2.2.6 Flow regimes
The fluid flow was defined in the model as unsteady, with initial conditions in which the outfall was completely intruded by saline water.

The rate of flow was predetermined to correlate with the flows used in the physical model, to incorporate both laminar and turbulent flow conditions. Tests were carried out with freshwater discharges through the outfall of between 0.2 l/s (Reynolds number 1650, laminar flow) and 1.0 l/s (Reynolds number 8200, turbulent flow). In each case, the freshwater flow was specified as constant throughout the test.

The timesteps used in the simulations varied between tests, depending on with the differing flow rates. Where possible, they were designed to match the sampling rate used in the physical model tests.

3 Results of numerical modelling
A numerical test was performed under laminar conditions, with only the innermost riser open. A series of density contour plots was obtained as the freshwater progressed along the outfall pipe. These plots were studied to provide suitable locations at which the results from the numerical and physical models would be compared. Figure 3 shows a longitudinal section through the outfall 300 seconds after discharge commenced. The selected monitoring locations are marked and the results for these locations are presented in Figure 4.
Figure 4a: Pressure at point A

Figure 4b: Horizontal component of velocity at point B

Figure 4c: Density at point C
4 Discussion

The results presented in Section 3 show distinct temporal changes in three of the key parameters used to define the flow patterns within an intruded long sea outfall. The pressure (Figure 4a) at location A, within the main outfall pipe, initially shows a constant value which is followed by a series of fluctuations. A small increase appears at the time when the saline/freshwater interface passes A. This is followed by an immediate reduction in pressure as discharge through the riser commences. The observed fluctuations in pressure correspond with the concepts reported by Guo and Sharp [8] in their investigations of outfall hydraulics.

The horizontal component of velocity (Figure 4b) initially shows a steady seaward flow, with a magnitude reflecting a uniform distribution across the section. This continues for a short period until disrupted by a sudden increase in velocity at location B, near the top of the duct. This increase occurs as the lower density freshwater travelling along the top of the outfall passes B. The increase in velocity suggests that the freshwater is confined to the top quarter of the duct.

The modelled values of density (Figures 4c) also reflect the presence of a distinct saline/freshwater interface in the outfall. The density at location C, within the riser, indicates the presence of saline water for the first 160 seconds. Shortly after the change in horizontal velocity at B, a sudden drop in density is shown at C. This drop in density corresponds with the movement of the saline/freshwater interface past C.

The results indicate that the saline/freshwater interface encountered in primary intrusion is reproduced in the model. The effect of increasing the flow (and generating turbulent conditions) is currently under investigation and the overall accuracy of the results under various flow regimes is being examined using the physical model.

Although the initial tests have been successful in reproducing the expected flow patterns, it is also apparent that the values produced by the model are somewhat sensitive to mesh configuration. The results of tests with two different meshes (Figure 4a, 4b, 4c) show some differences in the timing of the changes associated with the interface. These and other related results suggest the use of hybrid unstructured meshing techniques which are available within the FLUENT/UNS package and which allow grid refinement on the moving interface. This issue is currently under further investigation.
5 Conclusions

A two-dimensional numerical model of unsteady laminar flow in a long sea outfall has been established using the FLUENT/UNS CFD package. Preliminary tests indicate that the model is capable of reproducing the main flow mechanisms that occur during primary intrusion in such a system.

Further numerical investigations are in progress to examine the effect of turbulent flow in the outfall and to establish the flow patterns generated when more than one diffuser port is in operation. Physical model tests are currently under way to determine the accuracy of the numerical model in both laminar and turbulent flows.

6 Acknowledgements

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

References


