HR Wallingford
Working with water

HRPP 391

Dense jet assessment procedure

M. Wood & C. Mead

Reproduced from a paper presented at:
5th International Conference on Marine Waste Water Discharges and Coastal Environment, 2008
3rd International Exhibition on Materials, Equipment and Services for Coastal Environmental Projects, 2008

DENSE JET ASSESSMENT PROCEDURE

***Matthew J Wood* (1)*, Christopher T Mead* (2)**

[1] *Matthew J Wood, Mr., Senior Scientist., HR Wallingford Ltd, Howbery Park, Wallingford, Oxfordshire, OX10 8BA, UK +44 (0)1491 822496 – m.wood@hrwallingford.co.uk – www.hrwallingford.co.uk*
[2] *Christopher T Mead, Dr., Principal Scientist., HR Wallingford Ltd, Howbery Park, Wallingford, Oxfordshire, OX10 8BA, UK +44 (0)1491 822496 – c.mead@hrwallingford.co.uk – www.hrwallingford.co.uk*

Key Words: Dense jets, desalination, wastewater discharge, mixing zone, initial dilution, modelling

Introduction

HR Wallingford has long experience of the assessment of dilution and dispersion of positively-buoyant marine discharges from, for example, power stations and refineries. However, increasingly, assessments of negatively-buoyant discharges are required, particularly for:

- Saline discharges from proposed desalination plants.
- Saline discharges from salt cavern leaching operations.
- Cold discharges from Liquefied Natural Gas (LNG) regasification plants.

Studies of dense discharge dispersion usually involve two-stage approaches: a near-field dilution assessment, based on mixing zone software, such as CORMIX; and a mid- to far-field dilution/dispersion assessment to check longer-term build-up of background concentrations, such as can be carried out using a three-dimensional (3D) advection-diffusion model, for example TELEMAC-3D. HR Wallingford has developed an assessment procedure for undertaking such two-stage analyses of dense discharges, which can be used until the application of fully coupled near- and far-field models becomes practical.

This paper considers current modelling techniques, identifies alternative approaches where appropriate, and presents the above assessment procedure for the analysis of dense discharge dispersion and dilution.

The dense jet assessment procedure was established in the light of experience, and after review of previous studies. Work undertaken recently by various authors is summarised in the early sections of this paper, to introduce an example application of the assessment procedure presented in the later sections.

Recent advances in dense jets research: 1) Initial dilution and mixing

Whilst many previous studies have considered the mixing and dilution of positively buoyant jets, considerably fewer studies have focused on jets with negative buoyancy. This is likely to be due in large part to market needs; dense discharges still make up only a small percentage of discharges worldwide.

Of the research that has been undertaken into the mixing of dense jets, most has focused on discharge of an upwardly angled jet into a stationary ambient environment. Whilst this configuration should be the most simple to analyse, in reality the situation is complicated, as fully steady-state behaviour cannot be simulated in most laboratory conditions; an effluent layer develops at the bed, with no ambient current to carry it away from the discharge site. The situation is further complicated if experiments involve the use of a relatively confined test tank, which can cause the dense layer to build up artificially, resulting in complex 3D mixing patterns. Dilutions within this dense layer are usually of key environmental concern for real applications, because the layer is in direct contact with benthic flora and fauna.

The earliest study considered in detail in this literature review is that of Roberts et al. [1], who present analytical formulae based on the results of laboratory experiments to predict the geometry and dilution of dense jets angled upward at $\theta_0 = 60^o$ to the horizontal. The normalised empirical relationships established in this study (presented in Table 1) depend solely on the jet densimetric Froude number, $F = U_0/(g_0'\ d)^{1/2}$ where, for a discharge of density ρ_0 into ambient fluid of density ρ_A, $g_0' = g(\rho_0 - \rho_A)/\rho_A$ is the initial reduced gravity of the discharge (ms^{-2}), d is the nozzle diameter (m) and U_0 is the jet exit velocity (ms^{-1}).

Roberts et al. suggest that the layer of effluent that forms in the vicinity of the bed reaches an ultimate dilution at a certain distance from the nozzle. This ultimate minimum dilution is shown to be about 60% higher than the minimum dilution upon impact with the bed. It is suggested that this limit in dilution is caused by collapse of turbulence in the spreading layer, with the density current essentially forming a stable "relaminarised" region of low mixing. This analysis is discussed by Doneker and Jirka [2], who suggested that more in-depth three-dimensional analysis of the near-field plume behaviour is required. These uncertainties have not yet been resolved, and to the present authors' knowledge the thickness and mixing within this dense layer have not been considered in detail in any subsequent study.

Several authors have produced and refined similar formulae to those of Roberts et al. The formulae are summarised in Table 1, and the studies are described below.

Since the work of Roberts et al. [1] (and perhaps earlier still by Zeitoun et al. [3]) it has generally been assumed that jets angled upward at $\theta_0 = 60°$ result in the longest trajectories before impact with the seabed and, hence, the highest dilutions on impact with the bed. As such, outfall designs for dense discharges often assume a 60° angle by default. However, experiments conducted for stationary ambient environments by Nemlioglu and Roberts [4] found that impact dilutions show little variation for angles in the range 30° to 75°. Jets oriented upward at 15° and 90° (vertical) do show significantly reduced dilutions; vertical jets have the lowest dilutions as they fall back on themselves, limiting mixing with fresher ambient water, and at 15° jet dilutions are reduced due to the shorter jet path length and increased interaction with the bed.

Cipollina et al. [5] noted the lack of data for dense jets at discharge angles other than $\theta_0 = 60°$. They undertook an experimental investigation into jets angled at 30°, 45° and 60° to the horizontal, and used the results to establish empirical relationships between the jet Froude number and key geometrical properties of the discharge. However, dilutions were not recorded as part of the study. The experiments were undertaken using a continuously flushing tank (with an approximate current speed of 1.5mm/s) to prevent build-up of an effluent layer near the bottom of the tank.

Shao and Law [6] investigated the optimum discharge angle for shallow water conditions. They state that whilst a 60° upward angle produces long jet trajectories in deeper water, in shallow water (<10m for most coastal applications) such a discharge angle may cause the effluent to impinge on the water surface. This can result in inhibited dilution, and may be undesirable aesthetically for some discharge sites. The results of the study suggest that in such situations, upward discharge angles in the range 30° to 40° may be more appropriate. Analytical formulae are presented for the jet centreline terminal rise height, location of the downstream impact point and impact dilution. (These ideas are developed in the numerical studies of Jirka [7] and Bleninger and Jirka [8], described in a later section.)

Table 1 Analytical formulae from previous studies for dense jet discharges into stagnant ambient conditions

Parameter	**Normalised equation**	**Coefficient**							
		Roberts et al. [1]	**Nemlioglu and Roberts [4]**	**Cipollina et al. [5]**			**Shao & Law [6]**		**Kikkert et al. [10]**
Applicable range of upward discharge angles, θ°		60	30-75	30	45	60	30	45	0 to 75
Range of Froude numbers tested		20.5-35.7	21.2-24.1	23-76	14-216	18-89	10-31	10-32	14-99
Terminal rise height (top of jet), Z_t	$z_t/d\mathrm{F} = C_{1a}$	2.2	0.68 to 3.1 (1)	1.08 (3)	1.61 (3)	2.32 (3)	1.05	1.48	0.3 to 2.5 (4)
Centreline rise height, z_t	$Z_t/d\mathrm{F} = C_{1b}$	-	-	0.79 (3)	1.17 (3)	1.77 (3)	-	-	0.2 to 1.7 (5)
Impact point dilution, S_i	$S_i/\mathrm{F} = C_2$	1.6 ± 12%	1.8 ± 8%	-	-	-	1.37	1.26	(6)
Ultimate minimal dilution, S_m	$S_m/\mathrm{F} = C_3$	2.6 ± 15%	-	-	-	-	-	-	-
Location of impact point, x_i	$x_i/d\mathrm{F} = C_4$	2.4	1.9 to 3.4 (2)	3.03 (3)	2.82 (3)	2.25 (3)	2.97	2.83	0 to 3 (7)
Location of ultimate minimum dilution (length of "mixing zone"), x_m	$x_m/d\mathrm{F} = C_5$	9.0	-	-	-	-	-	-	-
Bottom layer thickness, z_L	$z_L/d\mathrm{F} = C_6$	0.7	-	-	-	-	-	-	-

Notes

\- Parameter did not form part of the investigation

[1] Depends on θ_0. For example, for θ_0 = 30°, 45° and 60°, $C_{1a} \approx 1.5$, 2.25 and 2.75 respectively. For θ_0 = 60° this is therefore 20-40% higher than the value obtained by Roberts et al. [1]

[2] Depends on θ_0. For example, for θ_0 between 30° and 60°, C_4 lies approximately in the range 3.2 to 3.4. For θ_0 = 60°, C_4 is 30-41% higher than that presented in Roberts et al. [1]

[3] Reference states that the formulae can be extended for any angle in the range 30-60°

[4] Depends on θ_0. For θ_0 = 30°, 45° and 60°, C_{1a} = 1.1, 1.6 and 2.2 respectively, which is in good agreement with the values obtained by Cipollina et al. [5]

[5] Depends on θ_0. For θ_0 = 30°, 45° and 60°, $C_{1b} \approx 0.63$, 1.14 and 1.7 respectively, which is in reasonable agreement with the values obtained by Cipollina et al. [5]

[6] Presents the *integrated* impact dilution, which does not depend on Froude number. If the discharge reaches the bed as a plume, then the integrated dilution is dependent on the discharge angle. The discharge reaches the bed as a jet for angles approximately $\theta_0 \leq 20°$. Integrated dilution = 0.88 for $\theta_0 < 20°$, and 1.0 to 1.6 for $\theta_0 \geq 20°$

[7] Depends on θ_0. For θ_0 = 45°, $C_4 \approx 3$, which is around 25% higher than the value obtained by Roberts et al. [1], and 6% higher than the value obtained by Cipollina et al. [5]

Ferrari [9] carried out an experimental investigation of some important factors in good outfall design for dense discharges. The results show that the most important parameters for design are the jet Froude number and the angle of discharge to the horizontal. The paper recommends that the Froude number should be made as high as possible (keeping within design constraints, for example navigation issues and avoiding excess head), and the upward discharge angle should be in the range 60-70°, to maximise the path length of the effluent before impact with the bed. This is largely in agreement with the earlier work of Roberts et al. [1]. However, the recommendations assume that the discharge is deeply submerged, with no surface interference. Shallow ambient conditions may necessitate smaller discharge angles, as discussed above.

Kikkert et al. [10] undertook a detailed experimental and analytical study to develop formulae for the prediction of both geometrical properties and dilutions of dense discharges at a range of upward discharge angles (0° to 75°). Their formulae cover both the jet region (momentum-dominated) and plume region (buoyancy-dominated) for dense discharges. They found that that their analytical formulae predicted the discharge's maximum rise height and upper edge position with reasonable accuracy, but the lower spread of the jet from its centreline is underestimated. They also find the minimum dilutions at the centreline maximum rise height and impact point are underestimated. They suggest that this is probably due to the complex and unstable structure of the dense jet, which is not intrinsically represented by standard formulations. This is discussed further in a later section.

Recent advances in dense jets research: 2) Linking with the mid- to far-field

Dispersion studies undertaken at HR Wallingford for both positively and negatively buoyant discharges usually involve a two-stage approach. Firstly, a near-field dilution assessment is carried out, based on mixing zone software, such as CORMIX (Doneker and Jirka, [11]), or analytical formulae as described above. These provide prediction of the near-field behaviour of the discharge, and key parameters such as the dilution and geometry of the near-field plume are predicted.

Secondly, mid- to far-field dispersion modelling is carried out to check longer-term build up of background concentrations, and dilution/dispersion outside the mixing zone, using a hydrodynamic model such as TELEMAC-3D. TELEMAC is a finite element system, developed by LNH-EDF, that represents the model area on a mesh of triangles of variable size and orientation. TELEMAC-3D uses a fixed number of layers based on sigma co-ordinates, where a given plane in the model is at a fixed proportion of the water depth. Active "tracers", such as temperature or salinity, are used in an equation of state to vary the fluid density. The model includes the effects of buoyant spreading and suppression of vertical turbulent mixing in the presence of stratification.

The dispersion modelling determines the wider impact in the surrounding waters, and assesses whether the discharge is likely to be retained in the vicinity of the outfall or is likely to reach sensitive receivers, etc.

It is important to introduce dense discharges into any mid- to far-field model in a way that is consistent with its near-field behaviour. In studies undertaken in recent years by HR Wallingford, for both positively- and negatively-buoyant discharges, the following approaches have been adopted in some studies:

i. The flow and undiluted source concentration are specified at a computational node at the exact location of discharge (for example, if the diffuser is to be located at mid-depth, then an undiluted dense source is placed at mid-depth);

ii. The flow and undiluted source concentration are specified at a computational node at the terminal level of the discharge, as indicated by the near-field dilution modelling (for example, if the outfall is to be located at mid-depth, but initial dilution modelling indicates that the discharge will sink to the bed, then an undiluted dense source is introduced at the bed).

Choi and Lee [12] refer to these approaches as i) "Actual Source" (AS), and ii) "Undiluted Source at Trapped Level" (USTL). Note that if an outfall is located at the same height as the terminal layer of the plume (for example as might occur if an outfall for a dense discharge was located at the bed), then the AS and USTL approaches are equivalent.

However, for studies of dense discharges where buoyancy effects are important (such as salt cavern leaching or combined power and desalination plant projects), the above approaches can lead to inaccurate dilutions being applied at the source, as the mass flow rate is divided over the volume of water associated with one computational node. The modeller therefore has limited control over the actual dilution that is applied. In some cases, HR Wallingford has found that using the USTL approach for dense discharges at the bed can result in artificially low initial dilution, with corresponding artificially high concentrations at the bed. This requires careful interpretation/explanation in any results presentation. In contrast, it has also been found that using the AS approach for a dense discharge introduced at mid-depth can lead to an artificially high dilution being applied at the source, to the point at which the simulated effluent is no longer sufficiently dense to sink to the bed (despite near-field modelling clearly indicating a tendency of the discharge to plunge and form a near-bed density current).

One potential improvement to this is the following approach:

i. An increased flow and diluted source concentration are specified at a computational node at the terminal level of the discharge, as indicated by the near-field dilution modelling. If the source has a concentration of C_0, and a flowrate of Q_0, and the dilution factor predicted at the bed is S_i, then the source is introduced at a concentration of C_0/S_i, at a flowrate of $S_i \times Q_0$.

Choi and Lee ([13] and [12]) refer to this approach as "Diluted Source at Trapped Level" (DSTL). Whilst the approach conserves the pollutant mass, conservation of water mass is not achieved. More specifically, too much water is introduced to the model (by a factor of S_i) as the volume of ambient water entrained into the discharge during initial dilution is not removed from the model domain. Choi and Lee address this through the development of the "Distributed Entrainment Sink Approach" (DESA), which represents the "loss" through entrainment of ambient water, as well as the increased flow of diluted effluent. The plume action on the ambient flow is modelled by a collection of sinks along the trajectory and a diluted source is introduced at the terminal level. Entrainment sink/source terms are determined by the embedded JETLAG near-field model. JETLAG is a Lagrangian jet model that predicts the mixing of an arbitrarily-inclined round buoyant jet in a stratified crossflow, with a 3D trajectory. It tracks the evolution of the average properties of a plume element by conservation of horizontal and vertical momentum, conservation of mass accounting for shear and vortex entrainment, and conservation of mass/heat (Lee et al. [14]).

The DESA is both water and pollutant mass conservative. Lee [15] compares the DESA and AS methods for an angled dense jet discharging near the bed. Cross-sections of pollutant concentrations show the vertical structure of the DESA-represented plume to be qualitatively realistic when compared with the AS approach, although explicit quantitative comparisons are not presented.
Bleninger [16] coupled the near-field dilution model CORMIX with the hydrodynamic model Delft3D. The coupled model uses time-varying near-field results from CORMIX to describe the source term for Delft3D. The coupling approach firstly classifies field-data and CORMIX timeseries results for the near- and intermediate-fields and computes input and source files for both CORMIX and Delft3D according to the far-field model grid resolution and intermediate-field plume geometry and concentration. At the time of writing, it is understood that neither the coupled model nor the individual CORMIX modules required to link with other hydrodynamic models (for example, CorTime) are commercially available.

Recent advances in dense jets research: 3) Numerical approaches: applications and limitations

Several authors have presented the results of numerical models, such as CORMIX and VisJet, for dense jets. Whilst some present good comparisons between numerical predictions and experimental results, others discuss the limitations of the underlying models.

Jirka [17] presents a detailed validation of CORMIX for discharges into a variety of water bodies. The study focused on positively-buoyant discharges, apart from a number of observations for dense jets:

- CorJet, the jet integral model within CORMIX, shows good agreement with data for the terminal rise height and minimum dilution, for vertical jets in stagnant conditions. The study notes that for cases where the jet densimetric Froude number, F, is less than five, turbulent mixing is weak and the jet falls back on, and entrains, itself, reducing dilution significantly. The jet integral model formulation is not valid under these conditions, and is therefore only recommended for jets with F>5 for the vertical case.
- The author refers to the original equations of Roberts and Toms [18], for the maximum elevation of the jet centreline, z_t, for discharges with $\theta_0 = 60°$. Roberts and Toms find that $z_t = 2.2L_m$, whereas CorJet uses $z_t = 1.9L_m$, (where L_m is the jet-to-plume-transition lengthscale, defined by $L_m = M_0^{3/4} / J_0^{1/2}$, $M_0 = U_0^2 d^2 (\pi/4)$, and $J_0 = U_0 g_0' d^2 (\pi/4)$, are the initial fluxes of momentum and buoyancy for a jet with initial exit velocity U_0).

- The author presents good agreement, in terms of trajectory and geometry ("visual width"), for jets with $\theta_0 = 55°$, as studied by Hutter and Hoffer [19]. Good agreement is also shown for dense jets in crossflow with data from Anderson et al. [20] for trajectory, dilution and visual width.
- The trajectories of vertical jets into crossflow also show good agreement with the data of Chu [21].

For the multiport diffuser (plane-jet) case, Jirka [22] presents validation of CORMIX2 against data for positively-buoyant jets and states that no experimental data have been reported in the literature for dense plane jets.

Limitations of typical modelling approaches for upwardly-angled dense jets have been discussed by several authors, including Ferrari [9] and Kikkert et al. [10]. Whilst in the immediate vicinity of a discharge outlet concentration profiles are axi-symmetric and Gaussian across the jet, the lower boundary of the plume rapidly becomes unstable, as the initial momentum and buoyancy of the upwardly-angled jet act in opposite directions,. This results in the collapse of the axi-symmetry of the jet, creating additional mixing and dilution of effluent in the lower half of the jet that is not represented by traditional integral model approaches. Kikkert et al. [10] show that the predictions of standard jet integral models (CorJet and VisJet) are conservative in terms of dilutions at the maximum jet rise height location, and the horizontal and vertical location of the maximum jet height. This may be due to the fact that this additional region of unstable mixing is not represented. They also show that the predicted integrated dilutions at the maximum rise height have a dependence on the angle of discharge, which is not supported by their experimental data.

Jirka [7] presents a parametric study using the CorJet model for dense jets discharged in the range 0° to 90° to the horizontal, for variable ambient bed slopes, with stagnant ambient conditions. The study is used to produce a design procedure for optimising the location and configuration of outfalls for dense discharges, based on the results of a large set of CorJet simulations. As part of the study, a validation exercise is undertaken against a limited range of experimental data. The predicted jet geometries are generally in reasonable agreement with the observed data. A limited comparison of CorJet results with dilution data is presented, which supplements the validation exercises already presented in Jirka [17]. The paper states that, given the lack of data (particularly dilution measurements) for dense discharges, the recommendations are considered preliminary.

Selection of available approaches for a dense jet assessment procedure

The assessment procedure presented here has been developed using the experience gained on recent projects of the following types:

- Salt cavern leaching studies around the UK
- Desalination plant studies in the UK and elsewhere
- Salt water cooling towers in the Middle East
- LNG regasification plant studies worldwide

Full details of the potential approaches available have been presented in the preceding sections of this paper.
Until coupled near- and far-field modelling systems become available, through in-house development or other means, the main tools available to HR Wallingford for the assessment of dense jets are:

- Initial dilution: analytical formulae and CORMIX
- Mid- to far-field dispersion: TELEMAC-3D

For initial dilution, CORMIX is the first choice calculation method at HR Wallingford, in line with the findings of an internal review of mixing zone software, that CORMIX generally provides the most conservative dilution predictions over a reasonably wide range of applications. CORMIX is believed to be the only model which ensures applicability before it executes a simulation, and is perceived to be the most widely-accepted near-field dilution model. The model has recently been updated with DHYDRO, which is a simulation module for dense brine and/or sediment discharges, and can now predict the formation of a density current flowing down a bed slope.

Like any model, CORMIX has limited applicability for certain conditions, for example, dilutions calculated for weak ambient currents are considered to be particularly conservative. For such conditions, the CORMIX results may be supplemented with analytical formulae, such as those shown in Table 1. However, it is important that caution is used when following this approach; if the CORMIX results suggest that prediction is not possible, or that predictions are unreliable, then this may be indicative of a particular discharge or flow phenomenon that must be taken into account (for example, near-field instabilities, discharge re-entrainment, coastal impingement, etc).

The near-field assessment should establish several features of the discharge. Firstly, the behaviour of the discharge at the edge of the near-field (for example, whether the discharge is fully mixed through the water column, or has restratified to form a density current at the bed). If the near-field assessment suggests that the discharge is likely to become fully mixed through the water column for the entire range of ambient conditions (and the discharge flow rate is sufficiently small to ignore buoyancy-induced spreading effects), full 3D mid- to far-field modelling may not be necessary. In such cases, it may be sufficient to use a depth-averaged, two-dimensional (2D) advection-diffusion model, such as TELEMAC-2D, or a Lagrangian model such as HR Wallingford's PLUME-RW(2D) model. However, in practice, most dense discharges encountered on consultancy projects do tend to form layers at the bed for at least some of the tidal cycle. Therefore, a 3D modelling mid- to far-field approach is usually appropriate.

The near-field assessment should establish the dilution at the edge of the near-field. In most dense discharge applications, HR Wallingford takes this to be the minimum dilution at the point of impact with the bed. Once the value of the initial dilution has been determined for a range of tidal conditions at the site, a decision must be made as to how the source term is specified in the mid- to far-field dispersion model. A conservative approach might be to use the worst-case initial dilution predicted at the bed over the range of tidal conditions at the site (often, the worst-case is at slack water, when the weak ambient currents limit the rate of dilution).

HR Wallingford usually uses near-field results to prescribe a dilution to the source, which is placed at the terminal layer. The source is therefore specified using the DSTL approach, with an increased flow rate (to represent the entrained ambient fluid) at a lower concentration (to represent the initial dilution). This approach maintains the correct total quantities of heat and/or salt introduced to the model.

The question of what constitutes "initial dilution" in this approach requires some consideration. In initial tests of this method by HR Wallingford, the minimum centreline dilution was used, and it was assumed that this applied over the entire initial zone of impact (that is, the entire plume cross-section at the bed). However, this may be overly conservative for some applications. In such cases, it is possible to integrate the dilution over the cross-section of the effluent plume, assuming a Gaussian concentration profile (the so-called bulk or flux-averaged dilution). According to Jirka [17], the bulk dilution can be approximately 70% greater than the minimum impact dilution at the centreline. This is unlikely to be important outside the near- to mid-field transition region, but sensitivity tests may be carried out if appropriate.

As the discharge is likely to be confined to the near-bed in the mid- to far-field model, it is important that the model vertical resolution (that is, the number of horizontal planes used to represent the vertical), and pollutant diffusivity coefficients are suitably defined. The sensitivity of the predictions to both of these parameters usually requires a number of sensitivity tests, as well as consideration of the hydrodynamic conditions at the site.
The assessment procedure for dense jets is summarised in Figure 1.

Example application of the dense jet assessment procedure

The procedure is presented here for a prototypical application. The opportunity was taken to demonstrate the sensitivity of the results to some key aspects of model application that can be easily overlooked. The test site was a typical open coastline, with tidally reversing currents and gradually sloping bathymetry. Tidal currents varied between less than 1cms^{-1} at slack water up to around 0.4ms^{-1} at peak current speeds.

A source of dense effluent was introduced near the seabed, at a location approximately 500m offshore, as shown in Figure 2. The average water depth in the vicinity of the discharge was about 9m. The excess salinity of the discharge was about 150 parts-per-thousand (ppt), and the discharge flow rate was around 0.1m^{3}s^{-1}. This concentration is typical of those associated with salt cavern leaching, with a density at the upper end of the normal range.

The outfall consisted of four ports equally spaced along a diffuser of length 30m. The densimetric Froude number of the individual ports was 20.7 (in line with the recommendation of Jirka [7] that the Froude number should lie between 20 and 25). The port spacing of this prototype diffuser design ensured minimal interaction between jets, although in reality a more complex design may facilitate higher dilutions. Ports were located 1m above the bed, and were angled upward at 60° to the horizontal. This angle was chosen to allow comparison with a number of the analytical formulae and to potentially maximise the jet trajectory. CorJet (CORMIX) was initially used to calculate the range of potential initial dilutions of the jet. Due to the variation in tidal currents, the range of achievable dilutions upon impact of the discharge with the bed was less 17:1 at slack water to 53:1 at peak current speeds. A comparison of the range of near-field predictions from the analytical formulae is shown alongside the CorJet predictions in Table 2.

In line with conservative assumptions, the DSTL approach was used, using the worst-case impact dilution prediction (from CORMIX), namely the centreline dilution S_i = 17. The source for the DSTL run was therefore introduced at four computational nodes at the bed, these corresponding to the length of the prototype diffuser, at a rate of 17x the original flow rate, at 1/17th of the original discharge concentration. Sensitivity tests were carried out on the method of source representation, using a similar test with AS approach.

The standard turbulence model used to represent vertical turbulence in TELEMAC-3D is a mixing length formulation. This computes a temporally- and spatially varying vertical tracer diffusivity. However, the user is required to specify a minimum "cut-off" value, which is used when the model computes a zero value. The minimum value taken varies from site to site, and a value is specified at a new site only after appropriate sensitivity testing and consideration of the local hydrodynamics.

Table 2 Near-field predictions of CorJet and various analytical formulae for the example procedure application

Parameter	**Normalised equation**	**Prediction in stagnant ambient conditions**				
		Roberts et al. [1]	**Nemlioglu and Roberts [4]**	**Cipollina et al. [5]**	**Kikkert et al. [10]**	**CorJet**
Terminal rise height (top of jet), Z_t	$z_t/d\mathrm{F} = C_{1a}$	$C_{1a} = 2.2$ ➔ $z_t = 3.4$m	$C_{1a} = 2.75$ ➔ $z_t = 4.3$m	$C_{1a} = 2.32$ ➔ $z_t = 3.6$m	$C_{1a} = 2.2$ ➔ $z_t = 3.4$m	$z_t = 2.5$m (centreline + Gaussian 1/e jet half-width)***
Centreline rise height, z_t	$Z_t/d\mathrm{F} = C_{1b}$	-	-	$C_{1b} = 1.77$ ➔ $Z_t = 2.8$m	$C_{1b} = 1.7$ ➔ $Z_t = 2.\ 6$m	$Z_t = 2.2$m
Impact point dilution, S_i	$S_i/\mathrm{F} = C_2$	$C_2 = 1.6 \pm 12\%$ ➔ $S_i = 29$ to 37	$C_2 = 1.8 \pm 8\%$ ➔ $S_i = 34$ to 40	-	*	$S_i = 17$
Ultimate minimal dilution, S_m	$S_m/\mathrm{F} = C_3$	$C_3 = 2.6 \pm 15\%$ ➔ $S_m = 46$ to 62	-	-	-	**
Location of impact point, x_i	$x_i/d\mathrm{F} = C_4$	$C_4 = 2.4$ ➔ $x_i = 3.7$m	$C_4 = 3.2$ ➔ $x_i = 5.0$m	$C_4 = 2.25$ ➔ $x_i = 3.5$m	$C_4 = 2.7$ ➔ $x_i = 4.2$m	$x_i = 3.6$m
Location of ultimate minimum dilution (length of "mixing zone"), x_m	$x_m/d\mathrm{F} = C_5$	$C_5 = 9.0$ ➔ $x_m = 14.0$m	-	-	-	**
Bottom layer thickness, z_L	$z_L/d\mathrm{F} = C_6$	$C_6 = 0.7$ ➔ $z_t = 1$m	-	-	-	**

- Parameter did not form part of the investigation

*** Does not depend on Froude number. Integrated dilution = 0.88 for $\theta_0 < 20°$, and 1.0 to 1.6 for $\theta_0 \geq 20°$**

**** CORMIX does not predict beyond the CorJet module, due to the fact that steady state behaviour does not exist for stagnant ambient conditions**

***** The CorJet terminal rise height is the sum of the centreline rise height and the Gaussian 1/e jet half-width. Note that this may not be comparable with the terminal rise heights predicted by the analytical formulae, which were most likely derived using concentrations much lower than the 1/e half-width.**

The sensitivity to the vertical resolution (that is, the number of vertical planes, and the spacing between neighbouring planes) was tested for the prototype application by undertaking model simulations using either 5 or 11 equally-spaced planes, or 11 planes with enhanced resolution nearer to the bed. For the enhanced bed resolution tests the vertical layering was defined by 11 planes at the following proportions of the total water depth: 0 (seabed), 0.037, 0.074, 0.111, 0.148, 0.185, 0.222, 0.4165, 0.611, 0.8055 and 1 (surface). This is a typical configuration that HR Wallingford has found to give acceptable representation of the dense plume near the bed, for similar sized discharges into similar water depth at other sites.

The range of test conditions is shown in Table 3.

Table 3 TELEMAC-3D test cases for the example application of the dense jet assessment procedure

Test	Source type, based on nomenclature of Choi and Lee ([13] and [12])	Number of vertical planes	Vertical plane spacing	Physical height of the 2nd model plane above seabed at mid-tide* near the outfall (m)
1	AS / USTL	11	near-bed resolution	0.33m
2	DSTL	11	near-bed resolution	0.33m
3	AS / USTL	11	equal	0.9m
4	AS / USTL	5	equal	2.25m

***Approximate total water depth of 9m near the outfall at mid-tide**

The maximum simulated excess salinity contours at the bed are shown for Tests 1 and 2 in Figure 3. It should be noted that these plots do not represent the excess salinity concentrations at a particular time; they can be thought of as the plume footprint.

It can be seen that the overall footprints are largely similar, emphasising the well-known fact that far-field results for a given flow/concentration are often essentially independent of the near-field details of the source. The effect of the source specification is limited to the region immediately around the outfall (~100m radius), which is often the most important for regulatory purposes. Peak concentrations at the bed for the AS approach are around 40ppt, so that the introduction of the source into the TELEMAC-3D model gives around a 4:1 dilution to the effluent for a source of this magnitude. The peak concentrations are still over 40% higher than initial dilution results suggest is likely. Peak concentrations for the DSTL approach are around 9ppt, which is consistent with the initial dilution results. This suggests that the introduction of the larger source of flow into the model results in less "artificial" dilution being applied.

In order to assess the effect of introducing an excess volume of water into the model (as the DSTL approach conserves the mass of pollutant, but not water), depth-averaged simulated flow vectors and speed difference contours are shown at slack water and during peak currents for Tests 1 and 2 in Figure 4. The effects of the DSTL approach on the depth-averaged current speeds is generally less than 5mm/s, with a maximum speed difference of less than 1.5cm/s at slack water. For most applications this difference would be acceptable, although in less energetic ambient environments (or at higher discharge rates), sensitivity tests might be appropriate to assess any artificial effects on currents.

Maximum simulated excess salinity contours at the bed for the vertical resolution sensitivity tests are shown in Figure 5 (Tests 1, 3 and 4). Each of the tests uses the AS representation of the source. The use of either 5 or 11 equally-spaced planes is shown to afford insufficient resolution near the seabed to represent adequately the vertical structure of the plume, as significant changes result from further resolution enhancements. For example, the offshore extent of the 5ppt contour is reduced from about 500m in Test 1, to just over 200m in Test 3, to below 100m in Test 4. The dispersion patterns presented demonstrate the importance of carefully considering the 3D structure of the dense plume, and an appropriate choice of vertical resolution.

It is clear in this example that the principal effects on the extent and vertical structure of the far-field plume are due to the model configuration, in particular the resolution of the water column structure, as opposed to the source representation.

Conclusions and further work

A procedure for the assessment of dense discharge dispersion in the sea has been developed through experience and on the basis of the available literature, and is now in use at HR Wallingford. The procedure can be summarised as follows:

- Determine whether the discharge is required to comply with near-field regulatory constraints.
- If the near-field is not a particular concern, then:
  - 3D mid- to far-field dispersion modelling should be undertaken, using a hydrodynamic model such as TELEMAC-3D.
  - AS source representation should be used.
  - The modelling should assess the level of dispersion away from the near-field, and should establish any potential for build-up of background concentrations due to the discharge.
  - Sensitivity tests should be undertaken to assess the sensitivity of the results to, for example, horizontal and vertical resolution.
- If the near-field is a key issue, then:
  - Near-field analysis should be undertaken using CORMIX, possibly in combination with the analytical formulae summarised in this paper.
  - The near-field analysis should assess the initial geometry and dilution of the discharge.
  - If the discharge restratifies at the edge of the near-field, then:
    - 3D mid- to far-field modelling should be undertaken, using a hydrodynamic model such as TELEMAC-3D
    - DSTL source representation should be used
    - The modelling should assess the level of dispersion away from the near-field, and should establish any potential for build-up of background concentrations due to the discharge.
    - Sensitivity tests should be undertaken to assess the sensitivity of the results to, for example, horizontal and vertical resolution, and the mixing/dispersion coefficients used.
    - If ambient currents at the discharge site are particularly weak, then it may be appropriate to carry out additional sensitivity tests using the AS approach.
  - If the discharge does not restratify at the edge of the near-field, then:
    - Mid- to far-field analysis may be undertaken using a depth-averaged 2D approach, such as TELEMAC-2D, or PLUME-RW(2D).
    - The modelling should assess the level of dispersion away from the near-field, and should establish any potential for build-up of background concentrations due to the discharge.
    - Sensitivity tests should be undertaken to assess the sensitivity of the results to, for example, the horizontal model resolution and the mixing/dispersion coefficients used.

The procedure has been demonstrated using near-, and mid- to far-field hydrodynamic modelling for an example test case. As might be expected, near-field pollutant concentrations were shown to be highly dependent on the choice of source representation, whilst far-field dispersion patterns showed little sensitivity to the method used. Simulated dispersion patterns were found to be more influenced by other key model parameters, in particular the choice of vertical resolution.

As noted by Jirka [7], there is limited experimental and field data available to validate modelling procedures, and it is highly desirable that appropriate monitoring exercises of dense discharges are undertaken. With the current increase in dense discharges worldwide (for example, through desalination and salt cavern leaching projects), there should be an abundance of sites available for such monitoring.

Modelling of the initial dilution of positively-buoyant discharges using Computational Fluid Dynamics (CFD) techniques has been considered by some authors (for example, Davis [23], Watt and Mead [24]); this is relatively new field, in which interest is increasing as computer processors become faster. The extension of CFD techniques to dense discharge simulation has been attempted (for example, Ortiz et al. [25]), though few studies have been published to date. CFD may have a role in near- and far-field model coupling but, at the present time, this is computationally impractical.

Physical modelling of dense discharges is usually not practical for consultancy projects, due to the time and costs involved and, as such, has not been discussed in this paper. However, in relatively complex cases, where the validity of numerical/analytical approaches may be brought into question, physical modelling could provide appropriate support to computational models.

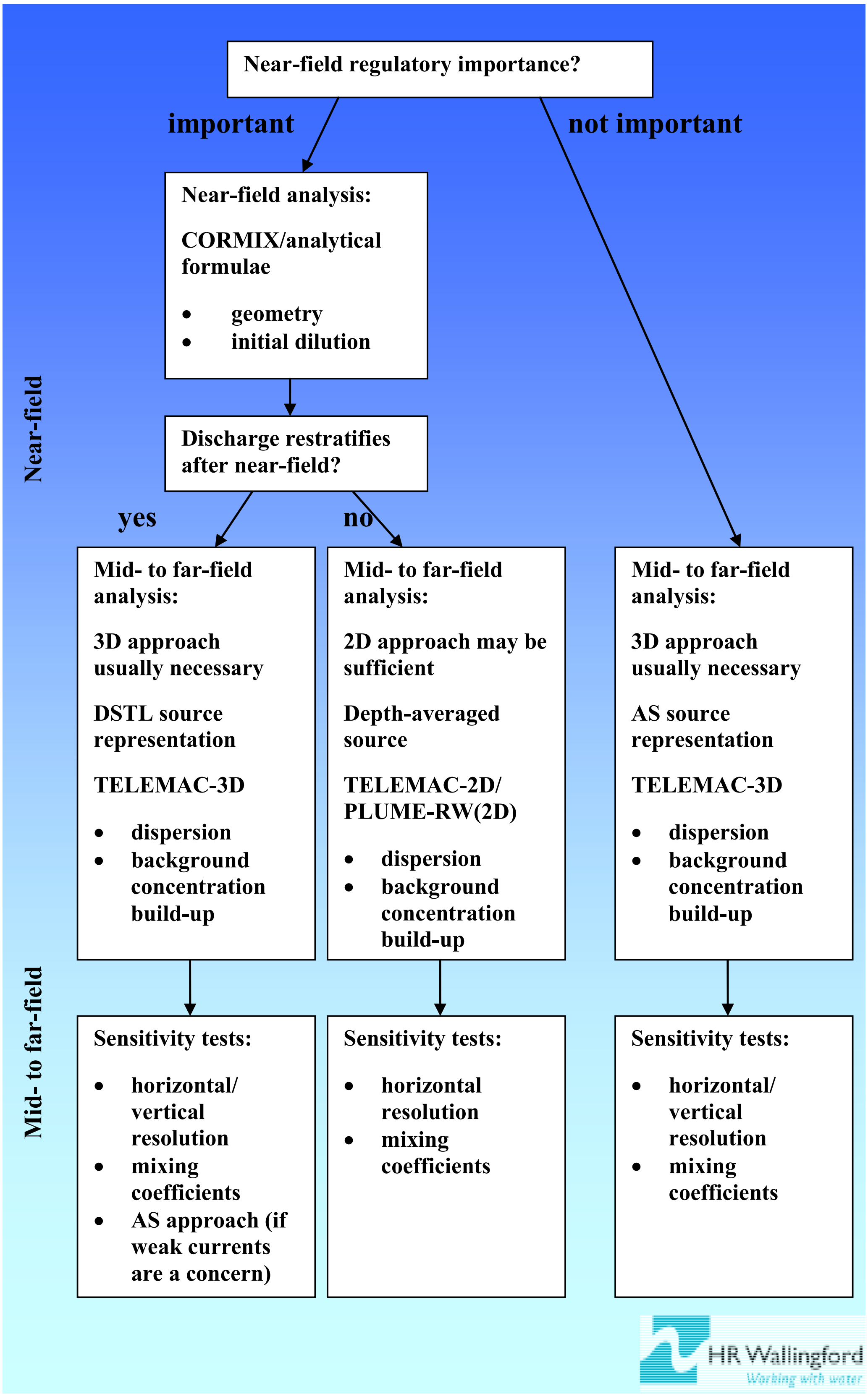


Figure 1 Decision tree for the dense discharge assessment procedure

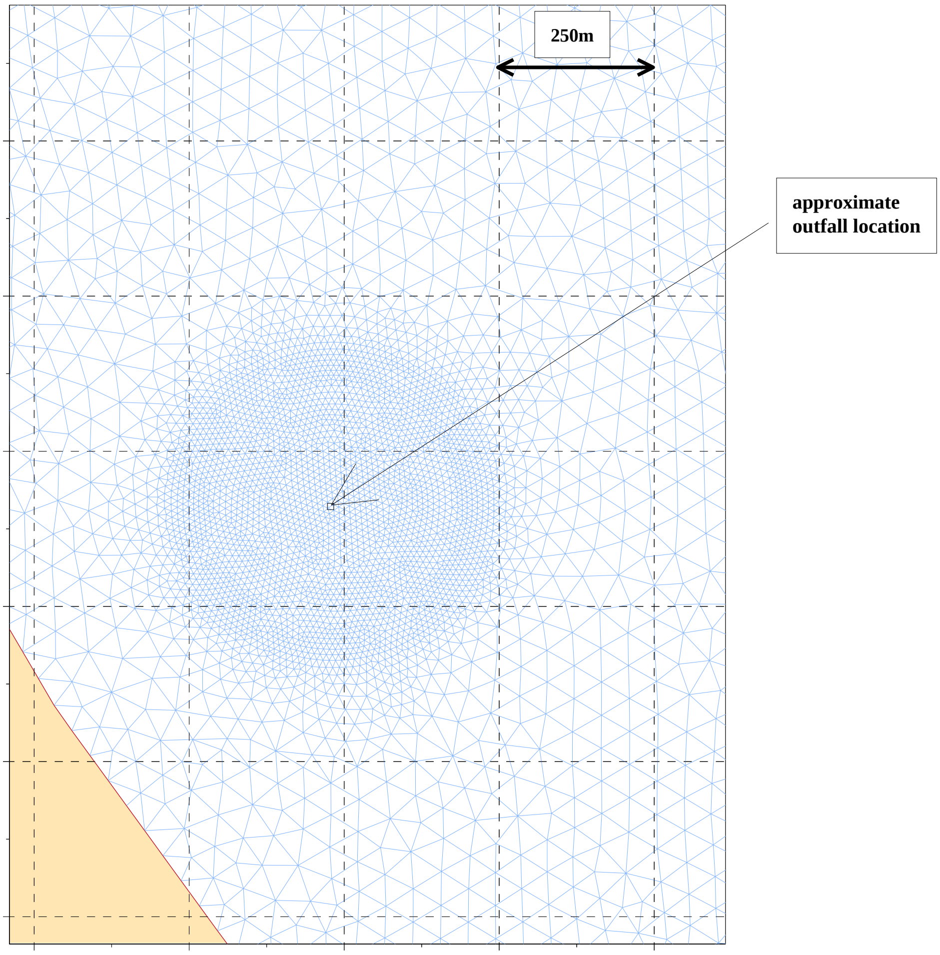


Figure 2 Example outfall location and model computational mesh

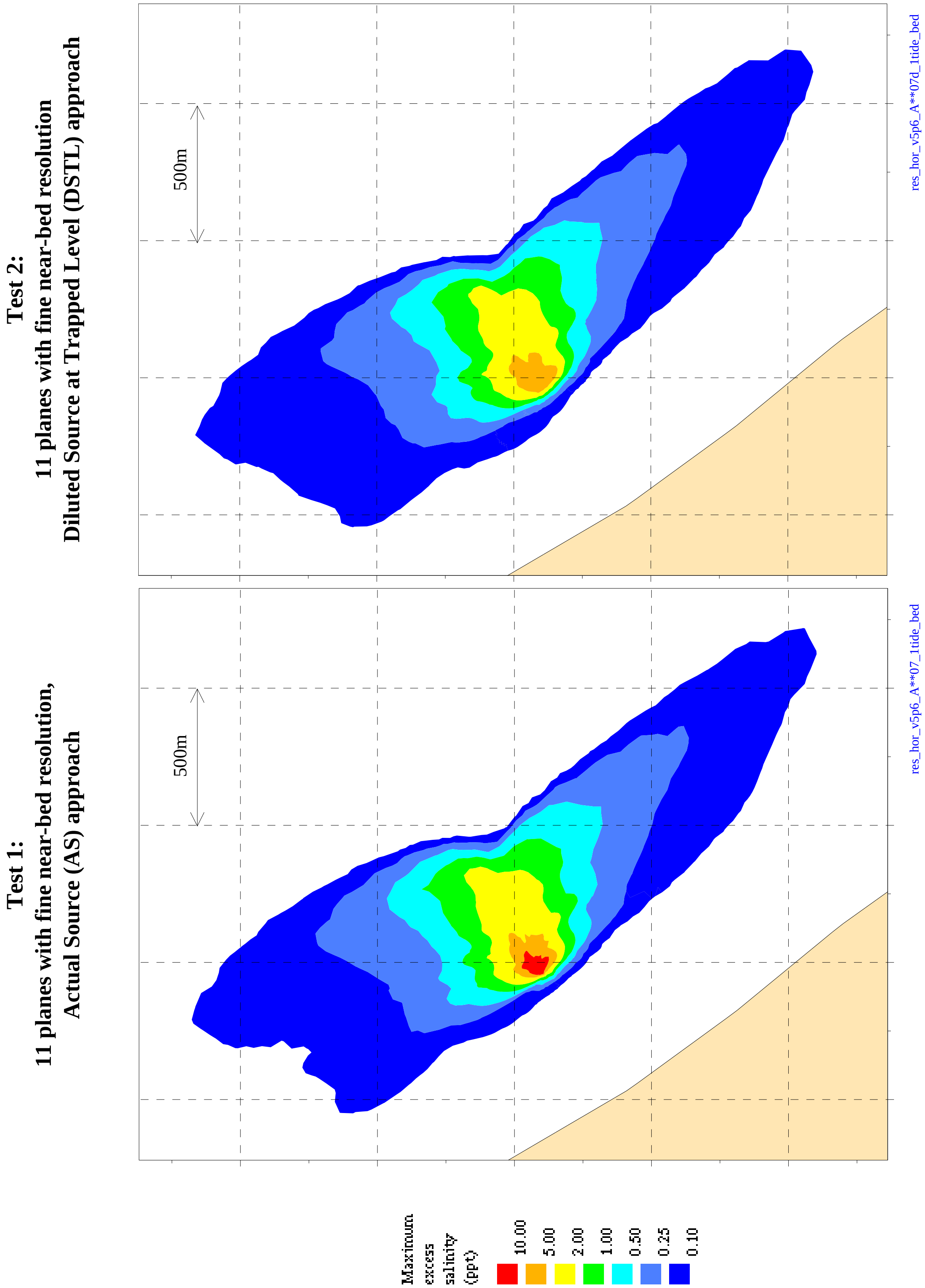


Figure 3 Maximum simulated excess salinity at the seabed, source representation sensitivity tests

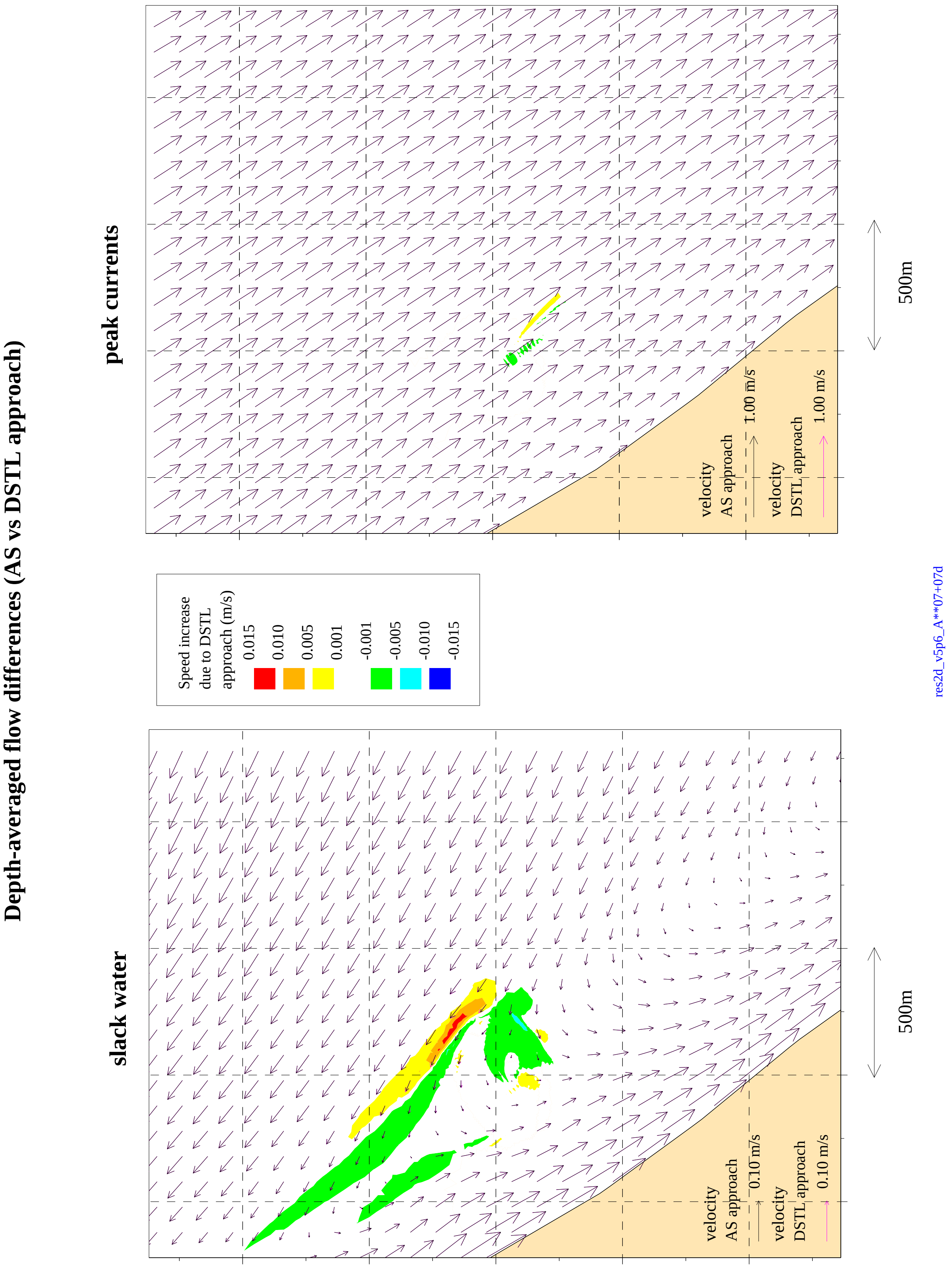
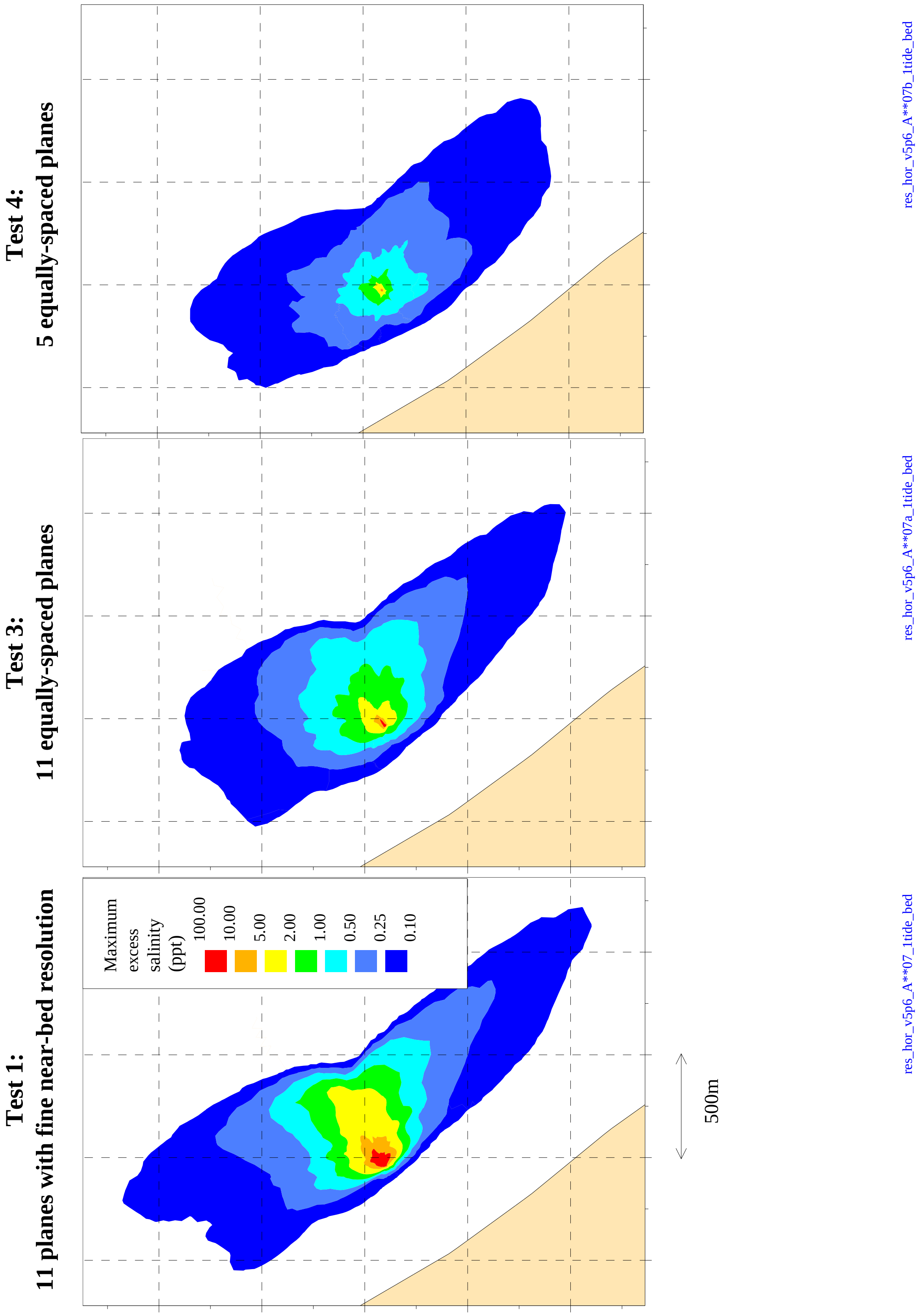


Figure 4 Simulated ambient current differences due to the method of source representation (note the different velocity scales in the left and right hand panels)



Test 1: 11 planes with fine near-bed resolution

Test 3: 11 equally-spaced planes

Test 4: 5 equally-spaced planes

Figure 5 Maximum simulated excess salinity at the seabed, vertical resolution sensitivity tests

Matthew Wood is a scientist within the Hydrodynamics & Metocean Group of HR Wallingford Ltd, and specialises in environmental hydraulics and pollutant dispersion modelling. He has over five years' experience in conducting and managing a range of numerical modelling studies investigating water quality, thermal impact, pollutant and sediment transport, and the effects of reclamations in marine waters, worldwide. Before joining HR Wallingford, Matthew gained an MSc in Theoretical and Applied Fluid Dynamics and a BSc (Hons) in Mathematics.

Dr Christopher Mead is a Principal Scientist, and team leader within the Hydrodynamics & Metocean Group of HR Wallingford Ltd. The team is responsible for all aspects of studies relating to marine discharge dispersion and water quality. Dr Mead has a PhD in physical oceanography, and some 25 years' experience in marine mathematical model application and development, including more than 20 years' specializing in marine dispersion studies at HR Wallingford. He developed the Company's PLUME-RW suite of Lagrangian dispersion models from first principles, and is now responsible for undertaking, supervising and managing a range of projects for water organisations and industrial clients.

HR Wallingford

HR Wallingford is an independent company offering specialist environmental and engineering consultancy, applied research services and software systems worldwide. Established in 1947, it has delivered innovative solutions to complex hydraulic problems ever since, and has gained a well-deserved international reputation for scientific and engineering excellence in the management of water related risks.

References

[1] Roberts, P. J. W., Ferrier, A., and Daviero, G., "*Mixing in inclined dense jets*", Journal of Hydraulic Engineering, 123 (8), p693-699, 1997

[2] Doneker, R. L., and Jirka, G. H., "*Mixing in inclined dense jets, discussion*", Journal of Hydraulic Engineering, p317-319, March, 1999

[3] Zeitoun, M. A., Reid, R. O., McHilhenny, W. F., and Mitchell, T. M., "*Model studies of outfall systems for desalination plants. Part III. Numerical simulations and design considerations*", Research and Development Progress Report No. 804, Office of Saline Water, U.S. Department of Interior, Washington, D.C., 1972

[4] Nemlioglu, S., and Roberts, P. J. W., "*Experiments on Dense Jets Using Three-Dimensional Laser-Induced Fluorescence (3DLIF)*", Proc. MWWD2006, 4th International Conference on Marine Waste Water Disposal and Marine Environment, Antalya, Turkey, November 2006

[5] Cipollina, A., Brucato, A., Grisafi, F., and Nicosia, S., "*Bench scale investigation of inclined dense jets*", Journal of Hydraulic Engineering, 131 (11), p1017-1022, 2005

[6] Shao, D., and Law, A. W-K., "*Desalination discharges in shallow coastal waters*", Proc. MWWD2006, 4th International Conference on Marine Waste Water Disposal and Marine Environment, Antalya, Turkey, November 2006

[7] Jirka, G. H., "*Improved discharge configurations for brine effluents from desalination plants*", Journal of Hydraulic Engineering, 134 (1), p116-120, 2008

[8] Bleninger, T., and Jirka, G. H., "*Modelling and environmentally friendly sound management of brine discharges from desalination plants*", Desalination, 221, p585-597, 2007

[9] Ferrari, S., "*Design of diffusers for heavy fluid discharge*", Proc. MWWD2006, 4th International Conference on Marine Waste Water Disposal and Marine Environment, Antalya, Turkey, November 2006

[10] Kikkert, G. A., Davidson, M. J., and Nokes, R. I., "*Inclined negatively buoyant discharges*". Journal of Hydraulic Engineering, 133 (5), p545-554, 2007

[11] Doneker, R. L., and Jirka, G. H., "*CORMIX User Manual. A hydrodynamic mixing zone model and decision support system for pollutant discharges into surface water.*", Office of Science and Technology, U.S. Environmental Protection Agency, Washington, DC 20460, 2007

[12] Choi, K. W., and Lee, J. H. W., "*Distributed entrainment sink approach for modelling mixing and transport in the intermediate field*", Journal of Hydraulic Engineering, 133 (7), 2007

[13] Choi, K. W., and Lee, J. H. W., "*A new approach to effluent plume modelling in the intermediate field*",. Proc. 31st IAHR congress, Seoul, Korea, p4303-4311, September 2005

[14] Lee, J.H.W., Cheung, V., Wang, W.P., and Cheung, S.K.B., "*Lagrangian modeling and visualization of rosette outfall plumes*", Proc. Hydroinformatics 2000, Iowa, July 23-27, 2000

[15] Lee, J. H. W., "*DESA - A New Approach to Effluent Plume Modelling in the Intermediate Field*", Proc. MWWD2006, 4th International Conference on Marine Waste Water Disposal and Marine Environment, Antalya, Turkey, November 2006

[16] Bleninger, T., "*Coupled 3D hydrodynamic models for submarine outfalls: Environmental hydraulic design and control of multiport diffusers*", Doctoral Thesis, Universität Fridericiana zu Karlsruhe, 2006

[17] Jirka, G. H., "*Integral model for turbulent buoyant jets in unbounded stratified flows. Part I: The single round jet* ," Environmental Fluid Mechanics, 4, p1-56, 2004

[18] Roberts, P. J. W., and Toms, G., "*Inclined dense jets in a flowing current*", Journal of Hydraulic Engineering, 113 (3), p323-341, 1987

[19] Hutter, K. and Hofer, K., "*Freistrahlen im homogenen und stratifizierten Medium – ihre Theorie und deren Vergleich mit dem Experiment*", Mitteilungen der Versuchsanstalt für Wasserbau, Hydrologie und Glaziologie, ETH Zürich, Nr. 27, 1978

[20] Abraham, G., "*Jets with negative buoyancy in homogeneous fluid*", Journal of Hydraulic Research, 5 (4), p235-248, 1976

[21] Chu, V.H., "*Turbulent dense plumes in a laminar crossflow*", Journal of Hydraulic. Research, 13, p263–279, 1975

[22] Jirka, G. H., "*Integral model for turbulent buoyant jets in unbounded stratified flows. Part II: Plane jet dynamics resulting from multiport diffuser jets*", Environmental Fluid Mechanics, 6, p43-100, 2006

[23] Davis, L.R., Davis, A. L., and Frick, W., "*Computational Fluid Dynamic Applications to Diffuser Mixing Zone Analysis - Case Studies*", Proc. MWWD2004, 3rd International Conference on Marine Waste Water Disposal and Marine Environment, Catania, Italy, September-October 2004

[24] Watt, G. A., and Mead, C. T., "*CFD calculations of initial dilution*", Proc. MWWD2006, 4th International Conference on Marine Waste Water Disposal and Marine Environment, Antalya, Turkey, November 2006

[25] Ortiz. J. P., Fortis, R., Lamparelli, C., Nieto, R., "*Computer modeling dispersion plume of ammonia from Petrobras submarine outfall*", Proc. MWWD2006, 4th International Conference on Marine Waste Water Disposal and Marine Environment, Antalya, Turkey, November 2006

Fluid thinking...smart solutions

HR Wallingford provides world-leading analysis, advice and support in engineering and environmental hydraulics, and in the management of water and the water environment. Created as the Hydraulics Research Station of the UK Government in 1947, the Company became a private entity in 1982, and has since operated as a independent, non profit distributing firm committed to building knowledge and solving problems, expertly and appropriately.

Today, HR Wallingford has a 50 year track record of achievement in applied research and consultancy, and a unique mix of know-how, assets and facilities, including state of the art physical modelling laboratories, a full range of computational modelling tools, and above all, expert staff with world-renowned skills and experience.

The Company has a pedigree of excellence and a tradition of innovation, which it sustains by re-investing profits from operations into programmes of strategic research and development designed to keep it – and its clients and partners – at the leading edge.

Headquartered in the UK, HR Wallingford reaches clients and partners globally through a network of offices, agents and alliances around the world.

HR Wallingford
Working with water

HR Wallingford Ltd
Howbery Park
Wallingford
Oxfordshire OX10 8BA
UK

tel	+44 (0)1491 835381
fax	+44 (0)1491 832233
email	info@hrwallingford.co.uk

www.hrwallingford.co.uk