Waste Water Discharge Modelling With Dynamically Coupled Near Field and Far Field Models

Robin Morelissen[*], Theo van der Kaaij[*], Dr. Tobias Bleninger[**]

* Hydraulic Engineering Department, Deltares, Rotterdamseweg 185, 2629 HD Delft, The Netherlands
(E-mail: *Robin.Morelissen@deltares.nl*, *Theo.vdKaaij@deltares.nl)*
** Institute for Hydromechanics, Karlsruhe Institute of Technology (KIT), Germany
(E-mail: *bleninger@kit.edu*)

Abstract
In many cases, (processed) waste water (e.g. sewage) is discharged into the marine environment, rivers or lakes. Obviously, effects of such discharges on the environment should be as small as possible and the natural system should be able to cope with them. It is therefore important to be able to assess the expected behaviour (i.e. mixing, spreading and dilution) of the waste water outfall plume resulting from such discharges, under the influence of currents, density differences and other processes.

In order to accurately determine the dispersion, recirculation and environmental impacts of waste water outfall plumes, it is important to be able to model the different characteristics of the outfall plume in detail; from the near field (small scales, metres around the outfall) to the far field (large scales, up to several kilometres away). Models that can theoretically cover this entire range of temporal and spatial scales in one integrated computation (e.g. non-hydrostatic models) are available, but they are very computational expensive and are not yet usable for most applications (e.g. engineering). The practical solution to this is to combine different types of models (near and far field models) that each focus on a specific range of scales and which have optimised resolutions and processes, depending on which scales they need to consider. However, to properly describe the hydrodynamic processes on the different scales, it is essential to couple these models in a dynamic and comprehensive way.

To achieve this, a dynamic coupling between the open-source Delft3D-FLOW far field model and the CORMIX near field expert system is under development. This coupled modelling system is able to use the computed far field ambient conditions in the near field computations and, conversely, to use the initial near field dilution and mixing behaviour in the far field model. Preliminary results are presented here to provide a first indication of the potential of the method. Once finished, this coupling will result in a comprehensive method for modelling the complete trajectory of a waste water outfall plume, allowing an accurate assessment of the environmental effects and the design of possible mitigating measures.

Keywords
near field model, far field model, dynamic model coupling, outfall plume

INTRODUCTION
General background
In many cases, waste water (e.g. sewage) is discharged into the marine environment, rivers or lakes. Obviously, effects of such discharges on the environment should be as small as possible and the natural system should be able to cope with them. It is therefore important to be able to assess the expected behaviour (i.e. mixing, spreading and dilution) of the waste water outfall plume in the environment, under the influence of currents, density differences and other processes.

In order to accurately determine the recirculation, spreading and resulting environmental impacts of waste water outfall plumes, it is important to be able to model the different characteristics of the outfall plume in detail. This is particularly the case in weak dynamic systems (i.e. low ambient flow velocities). The processes dominating the plume dynamics occur on significantly different spatial and temporal scales and are typically characterised by three zones defined along the plume trajectory: 1) Near field, 2) Intermediate zone (or Mid field), and 3) Far field. The near field is a region where the outflow characteristics (i.e. fluxes, geometry, and orientation of outflow) dominate the plume behaviour. The far field region is where the ambient flow conditions dominate the behaviour of the plume. The intermediate zone is the transition region from the near field to far field processes. Due to the large differences in scales and processes involved, different types of models are typically utilized for the simulation of near field and far field processes. Existing models that can theoretically cover this entire range of temporal and spatial scales in one integral computation (non-hydrostatic models) are very computational expensive and are not yet usable for most practical applications (e.g. engineering).

Offline coupling approach

To be able to model the entire trajectory of the plume from the initial metres to its effects several kilometres away, separate near and far field models must be used in combination. To describe near field behaviour of a plume, expert systems like CORMIX (Jirka et al, 1996) or jet-integral models like CorJet (part of CORMIX), VISJET (Lee et al, 2000) or Jet3D (Delvigne, 1979) are often applied. To assess the far field behaviour of a discharge plume, hydrostatic far field simulation programs like Delft3D-FLOW (Lesser et al, 2004) or POM (Blumberg and Mellor, 1987) are used. The most common engineering approach is to “translate” results of near field simulations into input (sources of fresher/brine/warm water) for the far field model. Different methods exist to couple these models, though mostly offline, i.e. not dynamic and one-way, and often in an arbitrary way. A change in ambient conditions as a result of the near field mixing of the plume is *not* accounted for. In addition, the dynamics of the ambient conditions are only represented in a very limited way in the near field model. A first step towards a more sophisticated, but still one-way and offline, coupling between a near and far field model (CORMIX and Delft3D) was developed by Bleninger (2006).

Online coupling approach

To represent near field behaviour in a far field simulation, the far field three-dimensional hydrostatic (open source) simulation program Delft3D-FLOW is coupled *online* and *dynamically* to different near field models. In this context, *online* means that during a far field simulation the computational results are used to define the input for the near field simulation and that near field results are used in the far field simulation. *Dynamically* means that transferring near field results to the far field simulation and vice versa occurs at a time interval sufficiently small to account for a change in ambient conditions. With this type of coupling, near field effects will be accounted for as much as possible in the far field model and changes in the far field will be fed back to the near field.

Initially a dynamic coupling between Delft3D-FLOW (far field) was set up with the near field jet-integral model Jet3D and the CorJet module of the CORMIX system. The jet-integral near field models are however often restricted to single port diffuser type discharges and do not include boundary interaction (surface or bed) of the plume. Therefore, subsequently an online coupling between Delft3D and the CORMIX expert system is presently being developed. CORMIX is a much more comprehensive system that includes different outfall configurations and flow classes and can assess boundary interactions of the plume, resulting in a more comprehensive and realistic coupling between the near and far field models.

In this paper, first the different near and far field models are described, followed by the motivation for the development of the coupled modelling system. The development of the coupling is presented in two steps, first coupling a near field, jet-integral model to the far field Delft3D model online and secondly coupling the CORMIX system to Delft3D online. The first, preliminary results are subsequently presented followed by the conclusions and discussion.

COUPLING CORMIX TO DELFT3D-FLOW

As discussed above, the aim of this development is to couple near and far field models in an online and dynamic way, in order to account for the near field effects in the far field model as much as possible. In particular, the interest is in coupling the extensive near field CORMIX system to Delft3D-FLOW to be able to apply this coupled modelling system for many different applications, such as waste water discharges with diffusers. This will result in a better representation of the plume behaviour in the far field model and potentially more accurate plume dispersion results, especially in weakly dynamic environments.

The Delft3D-FLOW far field model

Delft3D-FLOW simulates hydrodynamic flows in three dimensions, resulting from tide, wind and density differences induced by salinity and/or temperature differences. The hydrodynamic computational core of the program is based upon the shallow water equations. One of the assumptions in deriving the shallow water equations is that vertical accelerations are small compared to gravity. This results in a vertical momentum equation that reduces to the well known hydrostatic pressure assumption. A drawback of this hydrostatic assumption is that the near field region of the plume is not described correctly since vertical accelerations resulting from buoyancy differences are not accounted for.

Besides the hydrodynamic equations, the computational core also includes an advection/dispersion equation describing the transport of substances, including heat and/or salinity. The impact of horizontal density differences resulting from salinity and/or temperature differences is taken into account in the hydrodynamic part of the simulation program.

Vertical exchange of momentum and matter follows from a k-ε turbulence closure scheme (Rodi, 1980). Two separate equations for the transport of turbulent kinetic energy (k) and the dissipation of turbulent kinetic energy (ε) are solved. From k and ε vertical viscosity and diffusivity are computed.

Delft3D-FLOW is being used extensively in consultancy outfall studies regarding the large-scale (environmental) impact of waste water discharges, heat discharged by power plants or brine discharged by desalination plants. The near field outfall plume behaviour is traditionally determined by performing a number of offline near field simulations and "translating" the results (often arbitrarily) into sources for the far field model.

The CORMIX expert system

CORMIX (Jirka & Doneker, 1996) is a USEPA-supported mixing zone model and decision support system for environmental impact assessment of regulatory mixing zones resulting from continuous point source discharges. The system emphasizes the role of boundary interaction to predict steady-state mixing behaviour and plume geometry. The CORMIX methodology contains systems to model single-port, multiport diffuser discharges and surface discharge sources. Effluents considered may be conservative, non-conservative, heated, brine discharges or contain suspended sediments. CORMIX contains a rigorous flow classification scheme developed to classify a given discharge/environment interaction into one of several flow classes with distinct hydrodynamic features (Doneker & Jirka, 2001). Furthermore, it contains a jet-integral model (CorJet), and is capable of assessing plume boundary interactions and subsequent spreading of the plume in the ‘intermediate zone’, based on analytical and empirical formulations.

The coupled modelling system

The coupling of Delft3D-FLOW with CORMIX will combine the benefits and efficiency of both models, which will therefore result in a more comprehensive coupled modelling system that can also handle boundary interactions and provides a more flexible and physically more appropriate location for the coupling.

The development of the dynamic, two-way (online) coupling between CORMIX and Delft3D-FLOW is being carried out in a collaboration between (presently) MixZon (developers of CORMIX), Deltares (developers of Delft3D) and the Karlsruhe Institute of Technology (Dr Tobias Bleninger). In both modelling systems adaptations need to be made to be able to carry out autonomous simulations of the online coupled near and far field models, which are presently under development. The coupling approach is presented below in the form of the two main steps in which the coupling is developed. The first step presents the coupling method that is used for the near and far field models, initially applied for the development of an online coupling of jet-integral near field models to Delft3D. The second step shows the coupling of the more comprehensive CORMIX system to Delft3D by means of the same coupling method.

Step 1 - Coupling a jet-integral near field model to Delft3D-FLOW

In the first step, the dynamic coupling was developed for a steady-state, jet-integral near field model, such as Jet3D or CorJet (one of the modules in CORMIX), which are typically restricted to single port diffuser type discharges and do not include boundary interaction (surface or bed) of the e.g. sewage plume. The online and dynamic coupling between Jet3D/CorJet and Delft3D-FLOW was developed based on the Distributed Entrainment Sinks Approach (DESA) by Choi et al. (2007). They rightfully state that traditional one-way “source only” coupling methods do not account for near field mixing induced by plumes. To account for this mixing, they propose to position a number of sinks in the far field hydrostatic model along the plume centreline trajectory, which is computed by also taking into account the ambient conditions (e.g. currents, stratification). The magnitude of each sink should correspond to entrainment of the plume in the spatial section it represents. At the predicted near field terminal level of the plume, a source corresponding with the *diluted* plume is introduced. Due to entrainment of ambient water into the plume, the discharge of the diluted source is an order of magnitude larger than the original source and the concentrations (salt, temperature) are correspondingly lower. The discharged ‘mass’ at the predicted near field terminal level of the plume is conserved and corresponds to the mass discharged at the outfall point (plus the potentially entrained mass in case of e.g. temperature and salinity). A more elaborate description of this method can be found in Choi et al. (2007) and in Morelissen et al. (2011).

In the implementation of this coupling method in Delft3D-FLOW, a wide array of potential sinks and sources are defined in the horizontal grid cells and vertical layers in the Delft3D-FLOW model. The coupling procedure determines the plume trajectory in the far field model domain and assigns the sink and source magnitudes time-dependently to the appropriate cells. The unused defined sinks and sources in the Delft3D-FLOW model will have a discharge equal to zero but will be activated when the plume enters those cells. This allows the outfall plume to move in the far field domain under influence of e.g. tidal forcing without the need of defining new sink/source terms every coupling time step.

Coupling time interval and location

The coupling time interval is predefined in the Delft3D model input files and is based on expert-judgement and by taking into account the variability in the ambient system. The time interval of the coupling should correspond to the time-scale of the expected variations in the far field. It is further noted that this coupling time step will also depend on the actual 'travelling time' of the plume to reach the coupling location.

The location of coupling is defined at the 'end of the near field'. Since the definitions of 'near field' and 'far field' are somewhat ambiguous, the actual location of the coupling is also not fixed. In general with the use of jet-integral models, the end of near field is defined as either 1) the location where a boundary interaction occurs (plume reaching bed or surface), since these models do not include this type of interaction or 2) when the momentum of the plume itself is smaller than the momentum of the ambient flow. It is noted that especially for the boundary interaction cases, a transition between the near and far field exists that can be important in the spreading of the plume. In jet-integral models, this interaction, e.g. the lateral spreading of a thermal plume along the water surface, is typically not included and assumptions need to be made for this intermediate zone, and the plume behaviour therein, for the proper coupling of the near and far field models (see Figure 1).

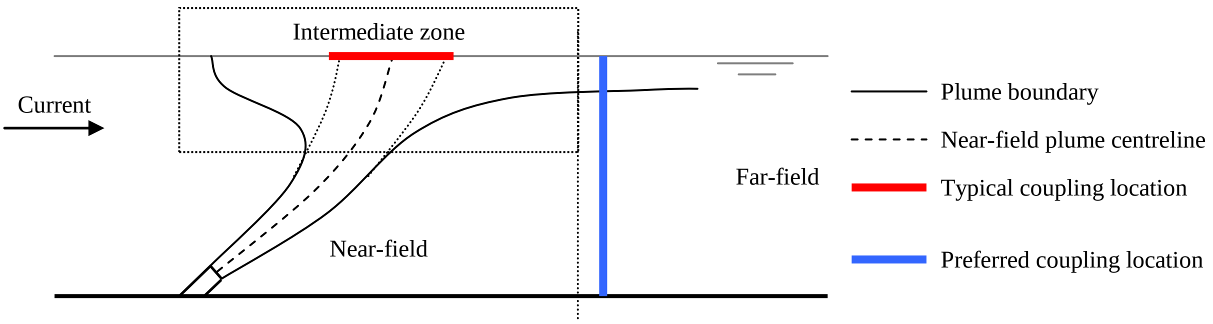

Figure 1 Typical plume behaviour including boundary interaction and lateral spreading along the water surface. The typical and preferred locations for coupling the near and far field are indicated.

Effect of the implemented DESA methodology

To demonstrate the effect of the Distributed Entrainment Sinks Approach (DESA) coupling for the Jet3D near field model in the Delft3D-FLOW far field model, a schematic case was modelled consisting of a rectangular tank of stagnant water with a depth of 5 metres. In this tank, a thermal jet was inserted at the bottom with a ΔT of 10°C. Subsequently, the jet was schematised in the far field model of the tank in two ways; 1) the traditional, typical method of coupling by inserting the (undiluted) source at the water surface and 2) by means of the DESA method and thus including sinks along the (vertical) jet trajectory and discharging the diluted source at the surface (i.e. end of the near field). It is noted that in the first way, the source was included as an undiluted source in order to be mass conserving. If this source would have been implemented as a diluted source (e.g. based on offline jet computations), the mass would not be conserved since the entrainment of ambient water that is responsible for this dilution, is not included in this way of coupling. Figure 2 shows the resulting flow patterns in a vertical cross section through the tank. The left plot shows the results of the traditional method of coupling the near and far field model with the undiluted source inserted at the surface and the right plot shows the results of the DESA coupling method. The difference in flow patterns can clearly be observed and the right plot with the DESA coupling method shows a much more realistic flow pattern, considering the jet trajectory, in which vertical flow cells are formed due to the momentum and entrainment of the jet.

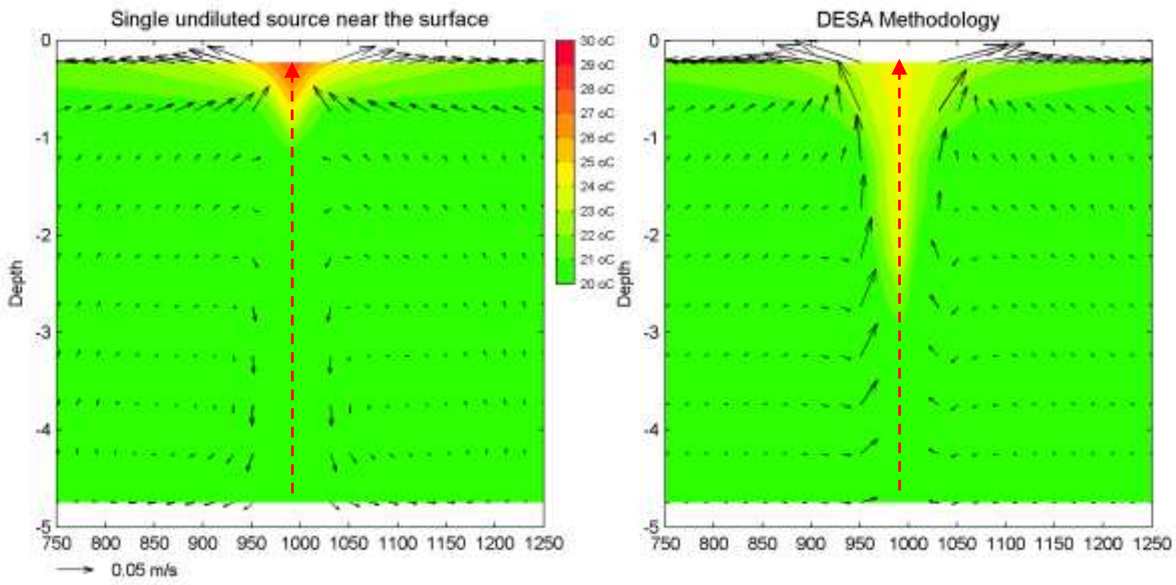

Figure 2 Comparison of flow fields (vertical cross section) in a schematic tank between the traditional, typical method of coupling (left) and the DESA coupling (right). The dashed red arrow shows the jet centreline that is schematised with the two different methods in the model.

Step 2 - Coupling the CORMIX near field expert system to Delft3D-FLOW

As presented in the previous section, the inclusion of near field entrainment effects in the far field model by means of the online, dynamic coupling (DESA), results in a more credible representation of the plume than traditional methods. However, by the use of only jet-integral models, the boundary interactions of the plume are still not described well, due to the absence of this capability in most jet-integral models. This boundary interaction is however important for the modelling of the full plume trajectory. Many of these boundary interactions cannot be modelled accurately in typical far field models either, therefore leaving a gap in the modelling of the full plume trajectory. In addition, these boundary interactions can play an important role in determining a proper schematisation of the plume dimensions and characteristics in the far field model. Therefore, a further model development is presently ongoing to overcome these issues.

Approach

The approach that is followed is a coupling between CORMIX to Delft3D-FLOW. It is based on the proprietary input and output files of the CORMIX system. Recent versions of CORMIX contain the CorTime module, in which it is possible to carry out a number of CORMIX simulations in batch, based on an input file with different model settings (e.g. outfall and ambient conditions) and a basic ‘case’ file that contains the details of the modelled outfall. From this batch, a series of CORMIX prediction files is generated.

In this extended coupling approach, Delft3D-FLOW writes the CorTime input file at a specified time interval, based on the simulated ambient flow conditions. The Delft3D-FLOW simulation will pause to allow a CORMIX simulation to run. Subsequently, Delft3D-FLOW will read and interpret the produced CORMIX prediction file and will include the computed plume characteristics in the far field model, following the DESA method (see above). At the proper coupling location, the plume characteristics (e.g. dimensions, dilution) will be translated to source terms in the Delft3D-FLOW model, to allow for an accurate coupling.

Status and first results
The present status of the coupling development is that Delft3D-FLOW is capable of writing the CorTime input files and is able to read and interpret the CorTime/CORMIX prediction (output) files. The latter is implemented for a selected number of CORMIX flow classes and is set up in such modular way that it allows for easy extension of other flow classes in the next phase of the development. The extension of the CorTime module to allow it to wait for the Delft3D-FLOW computation is presently under development. When this development is finished, the coupled CORMIX – Delft3D modelling system is able to run autonomously in dynamic interaction with each other.

At present, one of the implemented flow cases in the coupling is a single port outfall with a thermal (positively buoyant) jet in a small cross flow, which has a boundary interaction at the water surface (similar to the situation sketched in Figure 1, but with an initially horizontal discharge). The preliminary results of this coupled model simulation are presented below.

This simulation consists of a 20 km long, 600 m wide and 5 m deep channel with a buoyant (thermal) discharge at about 10 km and located about 80 m from the south bank. The Delft3D-FLOW grid size is about 25 x 25 m near the outfall location and has an equidistant layer distribution of 10% per (sigma-)layer. A constant cross-flow of about 0.16 m/s was applied to the channel. A horizontal discharge 1 m above the bed with a diameter of 1 m and a discharge rate of 2 m^3/s with an excess temperature of 15°C above the background temperature in the channel (15°C), i.e. the discharge temperature is 30°C.

The coupled simulation was run for 1 day to allow a steady-state situation to form. The coupling time step was 6 hours (i.e. every 6 hours a CORMIX simulation was carried out with updated ambient flow conditions). It is noted that in e.g. tidal cases, this coupling time step will be much smaller. In this simulation, the coupling location from near field to far field was selected to be just beyond the indicated 'end of near field' by CORMIX, similar to the preferred location indicated in Figure 1. At this location, about 50 m from the discharge, the computed dilution and dimensions of the plume are taken from the CORMIX results and 'mapped' to the appropriate cells and layers of the Delft3D-FLOW model. In this particular case, the width of the plume (~20 m) at that location still falls within one Delft3D-FLOW grid cell, but the thickness of the plume (~ 2.5 m), is divided over 5 layers (each layer corresponds to 0.5 m).

The initial results are presented as a vertical cross section in Figure 3. This figure also shows the initial plume trajectory computed by CORMIX (black line) and the coupling location (red dashed line). This figure shows that the diluted plume is indeed released over about 2.5 m of water depth in the far field model. The model results also show some additional diffusion below and upstream of the defined discharge. This is explained by the momentum of the discharge in the far field that (presently) works in all directions.

These preliminary results are compared to the results of the CORMIX computation beyond the coupling location. For that trajectory, the CORMIX model computes a top-hat concentration profile, which makes a direct comparison with Delft3D less straight-forward. Therefore, the 15.5°C contour line was computed, based on a transformation of the top-hat profile to a Gaussian concentration distribution and taking into account the dilution along its centre-line (the surface in this case). This contour line is presented in Figure 3 as the white line. Although the contour lines of the CORMIX and Delft3D models are not at the exact same location, the trends are very similar. The parameter settings of the Delft3D model are still very preliminary and much can be improved, but these initial results are very promising.

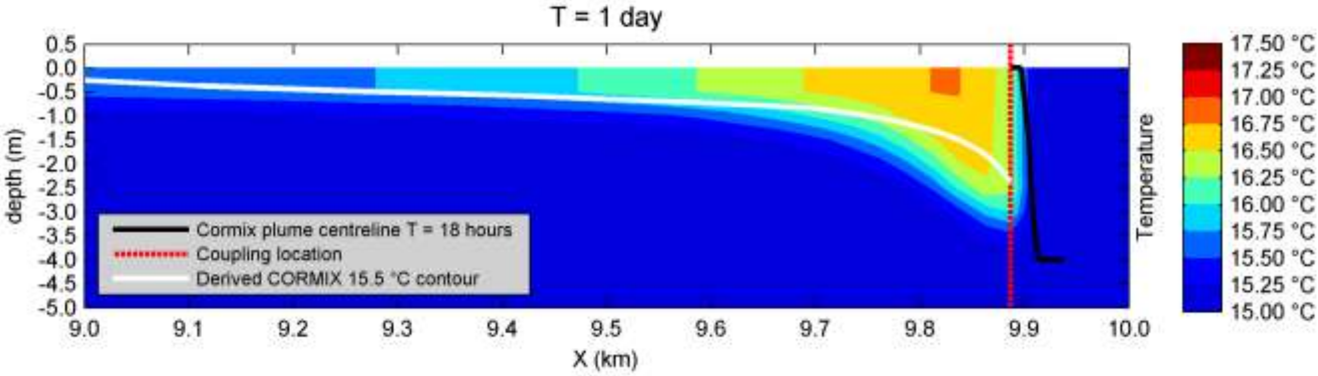

Figure 3 Preliminary results of a coupled CORMIX – Delft3D-FLOW computation. The contours in this vertical cross section along the plume show the temperatures derived by Delft3D-FLOW after coupling to CORMIX beyond the near field region (coupling location at red dashed line). The near field plume trajectory is presented in black. The Delft3D-FLOW results are compared to the CORMIX results in the derived 15.5°C contour line (white line).

Future developments

The first developments in the coupling of CORMIX and Delft3D-FLOW have been successfully completed and show promising results for future developments. These future developments include the autonomous running of the coupled modelling system, also under tidal conditions, the implementation of the other flow classes of CORMIX and the validation of the coupled modelling system against measured data. Furthermore, more fundamental research questions that are planned to be addressed are the (physically) optimal location and time step for coupling CORMIX and Delft3D-FLOW, taking into account the travelling time of the plume under different conditions.

CONCLUSIONS AND DISCUSSION

In order to accurately determine the dispersion, recirculation and environmental impacts of waste water outfall plumes, it is important to be able to model the different characteristics of the outfall plume in detail; from the near field (small scales, metres around the outfall) to the far field (large scales, up to several kilometres away). To do this in an efficient and accurate way, near and far field models need to be coupled in a dynamic (online) and comprehensive way.

In this study, the first developments were made to couple the CORMIX expert system to Delft3D-FLOW, in order to include the plume behaviour in the near and intermediate zone (between the near and far field) in the coupled modelling system. For this coupling, the DESA method by Choi and Lee (2007) was used that includes sinks along the jet trajectory in the far field model to also represent entrainment effects in the far field model.

The coupled modelling system was developed and tested from which the following conclusions can be drawn:

- The online, dynamically coupled near and far field models produce a more realistic flow field around the jet than the traditional method of coupling with an undiluted source at the end of the near field.
- A more comprehensive modelling approach to model the full plume trajectory is obtained by coupling Delft3D-FLOW with CORMIX, in which also the plume behaviour in the intermediate zone is also included in contrast to coupling only a jet-integral model that does not include plume boundary interaction. This new coupling allows for modelling the near and far field in a

physically more accurate way and allows for coupling at a more suitable location along the plume trajectory.

- CORMIX is able to cover many different flow classes and is therefore applicable in many cases. The present development of the coupling to Delft3D-FLOW will result in a very comprehensive and practical modelling system for application in many different outfall situations.

Based on the present study, a number of points for discussion and further research were identified:

- The coupling between CORMIX and Delft3D-FLOW is presently still under development. The first preliminary results, based on a limited number of implemented flow classes in this coupling, are presented in this paper. However, other flow classes need to be implemented in this coupling to extend its applicability in different practical cases.
- The results of both the coupling of jet-integral models and CORMIX to Delft3D-FLOW need to be further validated against laboratory and field data. This is especially useful for stagnant or weak-dynamic situations.
- The (physically) optimal location and time step for coupling CORMIX and Delft3D, taking into account the travelling time of the plume under different conditions, is subject to further research.

When the development of the coupling between CORMIX and Delft3D is fully completed, this will result in an accurate and comprehensive method for modelling the complete trajectory of a waste water outfall plume, allowing an accurate assessment of the environmental effects and the design of possible mitigating measures.

REFERENCES

Bleninger, Tobias, Jirka, Gerhard H., 2008, Modelling and environmentally sound management of brine discharges from desalination plants, Desalination 221 (2008) 585–597

Bleninger, Tobias, 2006, Coupled 3D Hydrodynamic Models for Submarine Outfalls: Environmental Hydraulic Design and Control of Multiport Diffusers. Thesis, Institute for Hydromechanics, University of Karlsruhe, Karlsruhe.

Blumberg, A.F., and Mellor, G.L., 1987, “A description of a three-dimensional coastal ocean circulation model” Three-Dimensional Coastal Ocean Models, Vol. 4, edited by N.Heaps, pp. 208, American Geophysical Union, Washington, D.C.

Choi, K.W. and Lee, H. W., 2007,” Distributed Entrainment Sink Approach for Modeling Mixing and Transport in the Intermediate Field”, Journal of Hydraulic Engineering Vol. 133, No. 7

Delvigne, G., 1979, “Round buoyant jet with three-dimensional trajectory in ambient flow”, Paper presented at the 18th congress of the IAHR, Cagliary, Italy, September 1979

Doneker, Robert L., Jirka, Gerhard H, 2001, CORMIX-GI systems for mixing zone analysis of brine wastewater disposal, Desalination 139 (2001) 263-274

Jirka, G.H., Doneker, R.L. and Hinton, S.W., 1996, "User’s Manual for CORMIX: A Hydrodynamic Mixing Zone Model and Decision Support System for Pollutant Discharges into Surface Waters", U.S. Environmental Protection Agency, Tech. Rep., Environmental Research Lab, Athens, Georgia, USA

Lee, J. H. W., Cheung, V., Wang, W. P. and Cheung, S. K. B., 2000, “Lagrangian modeling and visualization of rosette outfall plumes”, *Proc. Hydrodynamics*, Lowa

Lesser, G.R., Roelvink, J.A., Kester van, J.A.Th.M., and Stelling, G. S., 2004, “Development and Validation of a three-dimensional morphological model”, Journal of Coastal Engineering 51

Morelissen, R., Kaaij, T. van der , Bleninger, T., 2011, Dynamic Coupling Of Near field And Far field Models For Simulating Outfalls, IDA World Congress 2011, Perth, Western Australia IDAWC/PER11-014

Rodi, W., “Turbulence models and their application in hydraulics – A state of the art review”, Delft, The Netherlands: International association of Hydraulic research