report

ERMS

Environmental Risk Management System

Program participants: - Agip - Conoco Phillips - Exxon Mobil - Hydro - Petrobras - Shell - Statoil - Total

STF80MK A06204 Open
Date: 11.07.2006

ERMS Report no. 23

Implementation of the near-field module in the ERMS model

Øistein Johansen, Ismail Durgut

SINTEF

SINTEF Materials and Chemistry

Address: NO-7465 Trondheim, NORWAY
Location: Brattørkaia 17B, 4. etg.
Telephone: +47 4000 3730
Fax: +47 930 70730

Enterprise No.: NO 948 007 029 MVA

SINTEF REPORT

TITLE

ERMS REPORT NO. 23

IMPLEMENTATION OF THE NEAR-FIELD MODULE IN THE ERMS MODEL

AUTHOR(S)

Øistein Johansen and Ismail Durgut

CLIENT(S)

ConocoPhillips, Eni, ExxonMobil, Hydro, Shell, Statoil, Total, Petrobras (The ERMS project participants)

REPORT NO.	CLASSIFICATION	CLIENTS REF.	
STF80MK A06204	Unrestricted	ERMS Steering Committee members	
CLASS. THIS PAGE	ISBN	PROJECT NO.	NO. OF PAGES/APPENDICES
Unrestricted	82-14-03768-9	661365	17
ELECTRONIC FILE CODE		PROJECT MANAGER (NAME, SIGN.)	CHECKED BY (NAME, SIGN.)
ERMS report no 23_Near-field module.doc		Ivar Singsaas	Kjell Skogness
FILE CODE	DATE	APPROVED BY (NAME, POSITION, SIGN.)	
	2006-07-11	For Tore Aunaas, Research Director	

ABSTRACT

This report describes the basic concepts used in the simulation of the near-field spreading processes in the ERMS model. SINTEF's DeepBlow model has been the starting point for the development of a stand-alone general multiphase plume model (Plume3D). The stand-alone Plume3D model also includes near-field far-field coupling, and far-field tracking for dispersed bubbles, droplets or mineral particles. This model was finally implemented as a near-field subroutine in the ERMS model. However, as far-field tracking is already taken care of in the ERMS model, the implementation mainly involved the plume model as such, and the near-field far-field coupling processes.

The first chapter in this report deals with the concepts used for representing the separation process and the method used for handing over particles to the far-field model. The second chapter deals with specific solutions related to implementation in the ERMS model. Example applications of the near field module are presented in the final chapter.

KEYWORDS	ENGLISH	NORWEGIAN
GROUP 1	Environment	Miljø
GROUP 2	Numerical modelling	Numerisk modellering
SELECTED BY AUTHOR	Near field	Nærsone
	Drift and spreading	Drift og spredning

TABLE OF CONTENTS

INTRODUCTION

This report describes the basic concepts used in the simulation of the near-field spreading processes in the ERMS model. SINTEF's DeepBlow model has been the starting point for the development of a stand-alone general multiphase plume model (Plume3D). Subsequently, the Plume3D model was implemented as a near-field module in the ERMS model.

The major novel processes that were included in the stand alone Plume3D model are as follows:

- Mineral particles are accounted for in addition to the present constituents of the discharge (oil, water and gas).
- The mineral particles are defined in terms of particle size classes and density categories (i.e. a number of particle size classes must be defined for both drilling mud and cuttings).
- Separation of oil droplets and mineral particles from the plume is accounted for, and separated particles are handed over to the far-field model for subsequent tracking.

Separation of gas bubbles from the plume was already accounted for in the stand-alone Plume3D model, but an extension of the model was made to account for the drift and fate of separated gas bubbles in the far-field. However, this particular extension (tracking of gas bubbles) is not included in the ERMS model, where far-field tracking is limited to dispersed oil droplets, mineral particles and dissolved chemicals.

The first chapter in the following deals with concepts used for representing the separation process and the method used for handing over particles to the far-field model. The second chapter deals with specific solutions related to implementation in the ERMS model. Example applications of the near field module are presented in the final chapter.

1 NEAR-FIELD FAR FIELD COUPLING

In this chapter, we describe the concepts used in the near-field far-field coupling in general terms, mostly based on the solutions chosen in the stand-alone Plume 3D model. Specific details concerning the practical implementation of the near-field model in the ERMS model are described in the next chapter.

1.1 Brief description of the Plume3D model

The plume model discussed in the present report was originally developed with support from the Norwegian Deep-sea Program (NDP), which was organised by Norwegian oil companies engaged in deep-sea exploration drillings in Norwegian waters. Prior to the model development, SINTEF had conducted a literature study on deep water processes with support from NDP (Johansen 1997). The major findings from this study were as follows:

- For blowouts at shallow to moderate depths the gas may be considered an ideal gas with a specific volume decreasing linearly with pressure. However, when the blowout takes place at greater depths, the gas can no longer be assumed to behave as an ideal gas, and the pressure and temperature dependent compressibility factor (z-factor) must be introduced in the pressure-volume relationship.
- Secondly, the fraction of gas dissolved in the oil will increase with pressure. This implies that the gas mass fraction of the well flow at the outlet will be reduced compared to the gas mass fraction predicted by the GOR.
- Dissolution of gas from rising bubbles into ambient water may be negligible for blowouts at shallow to moderate depths, since the residence time of the gas bubbles is expected to be short. In deep waters, the rise time of the gas bubbles will be significantly longer. Since the solubility of the gas also will be high due to high ambient pressures, dissolution of gas in seawater may be expected to cause a significant reduction in the buoyancy flux.
- In addition, natural gas tends to form gas hydrates at elevated pressures and low temperatures. Thus, when a blowout takes place at larger depths, the gas may be converted to hydrate in contact with cold bottom water. If that happens, the contribution of the gas to the buoyancy flux will vanish, and the considerably lower buoyancy caused by gas hydrates and oil will instead drive the rise of the plume. In such cases, even small, stable density gradients in the ambient water may be expected to cause trapping of the plume.
- After trapping of the plume, oil may finally arrive at the sea surface due to the buoyancy of individual oil droplets. The resulting surface spreading of the oil will then depend on the size distribution of the oil droplets and the strength and variability of the ambient current. This situation differs significantly when blowouts occur at moderate depths. In such cases, surface spreading of the oil will be governed by the radial outflow of water entrained by the rising gas bubble plume.

Thus, in order to cope with the particulars of deep water processes, the following processes had to be included in a deep water plume model.

- Effects of cross-currents
- Non-ideal gas behaviour
- Dissolution of gas from bubbles to ambient water.
- Formation and subsequent disintegration of gas hydrates

The first modification (effects of cross-currents) implies the introduction of the mechanism of forced entrainment in the first place. However, when this is included, the plume may be found to bend over due to the entrainment of momentum from the ambient water. This implies a potential for vertical leakage of gas bubbles from the plume.

The second modification (non-ideal gas behaviour) implies the introduction of a pressure and temperature dependent compressibility factor (z-factor) in the pressure-volume-temperature (PVT) relationship of the gas. This z-factor also depends on the composition of the gas phase, and is a well-known subject in petroleum physics.

The third modification implies that the process of dissolution of gases from bubbles into ambient water must be included. This process is governed by the mass transfer coefficient between the rising gas bubbles and ambient water, the specific surface area between gas bubbles and water, and the solubility of the gas in seawater.

The last modification implies that the potential conversion of gas into hydrate in contact with seawater must be introduced. At the same time, the buoyancy of hydrates formed from the gas must substitute the buoyancy of the gas bubbles. In order to facilitate the introduction of these modifications in total, it was found convenient to base the model on the Lagrangian concept used in more recent developments. In the Lagrangian models, the plume is represented by a series of non-interfering elements. Each element, which can be thought of as a cylinder or section of a bent cone, is characterised by its mass, location, width (radius), length (thickness), average velocity, pollutant concentration, temperature and salinity. These parameters will change as the element moves along the trajectory, i.e., the element may increase in mass due to shear-induced and forced entrainment, while rising by buoyancy and bent over by the cross-flow.

General aspects of the application of the Lagrangian model concept for multiphase plumes, and the special solutions required for deep water, are described in more detail in the DeepBlow model paper (Johansen 2000). Comparisons of model predictions with field experiments are presented in the DeepSpill paper (Johansen et al., 2003), and a review of more recent developments on deep water plume modelling is given in Johansen, 2003. An update of the DeepBlow model to cope with these findings was made as a part of another NDP-project, DeepRisk (Johansen 2004, 2005).

Finally, as a part of the ERMS project, the DeepBlow model was developed into a general purpose multiphase plume model, namely Plume3D. This implied inclusion of new constituents, mineral particles and dissolved chemicals, as an addition to the constituents already accounted for in the Deepblow model, i.e. formation water, oil droplets and gas. As described in the following, the Plume3D model was first developed as a stand-alone model, and later implemented as a near-field module of the ERMS-model.

1.2 Separation of dispersed constituents from the plume

In the DeepBlow model, the separation process was included for one constituent only, e.g. for gas bubbles. The calculated leak rate of gas bubbles from the plume was based on the deviation of the plume trajectory from the vertical, and the rise velocity of the bubbles – "checked" by the entrainment velocity related to turbulent entrainment of ambient water. However, this concept can be applied to any other dispersed constituents as well (oil droplets, mineral particles).

By using the well-mixed hypothesis, the leak rate of a dispersed constituent "i" can be derived from its concentration C_i [kg/m^3], separation velocity $W_S^{(i)}$ [m/s], and the projected area A [m^2] of the plume element (see *Figure 1.1*). With $A = 2\,b\,h$, where b is the plume radius and h is the length of the plume element, the equation for the leak rate of gas bubbles can be written as:

$$\dot{Q}_i = C_i W_S^{(i)} A \text{ [kg/s]}$$

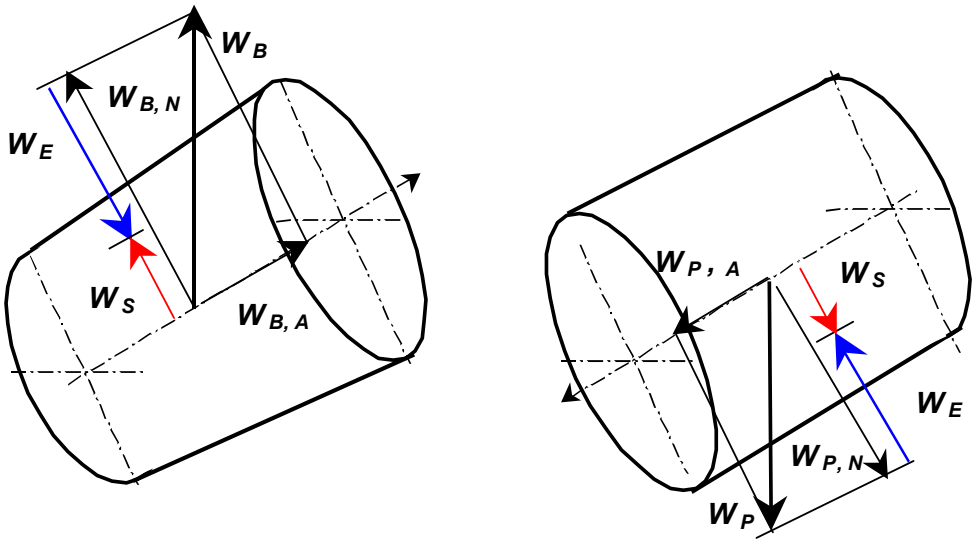

Figure 1.1 Calculation of net separation velocity for buoyant bubbles or droplets (left) and mineral particles (right). The normal component of the rise velocity of the bubbles $W_{B,N}$ or the sinking velocity of the particles $W_{P,N}$ is "checked" by the entrainment velocity W_E, forming a net separation velocity W_S normal to the plume trajectory. The axial component of the rise velocity adds to the axial plume velocity and reduces the retention time of the bubble or particle in the plume element.

When applied in the plume model, this concept will cause a gradual reduction in time of the mass of a given dispersed constituent in the plume element.

Since constituents that have left the plume will have no influence on the dynamic behaviour of the plume (e.g. in terms of momentum, buoyancy etc.), further handling of the separated dispersed constituents can be left to the far-field module. The concept for handing over the separated dispersed constituents to the far-field is discussed in the next section.

1.3 Simulating plume behaviour in the far-field module

In the far-field module, each dispersed constituent "i" in the discharge – such as gas, oil and mineral particles – is subdivided into a number of size classes "j". Parameters associated with a given size class of a given constituent in the discharge might be represented formally by $\mathbf{V}(i, j)$, where the label "i" represents the type of constituent (e.g. gas), while the label "j" represents its size class. Time series of the masses of these classes are generated in the near-field simulation, expressed in terms of $G(i, j, k)$, where k is the time step number – corresponding to the retention time $\tau = k\,\Delta\tau$ when referred to the plume element, where $\Delta\tau$ is the timestep in the plume simulation.

The near-field module is run each time the ambient current is changed in the far-field model. Arrays of constituent and size class masses $G(i, j, k)$ and retention times $\tau(k)$ are stored in the computer memory together with time series of the coordinates of the plume element $X, Y, Z(k)$. The size classes of each constituent in the discharge are tracked as particles in the far-field model.

As long as the particles stay inside the plume, the following *advection rule* applies:

- The position x, y, z of the particle at a given time t are determined by time interpolation of the plume coordinate $X, Y, Z(k)$.

A spherical random displacement of length $R = b(t)$ is added to account for the radial expansion of the plume, where b is the plume radius interpolated to the time t since the model particle was released.

When the particles have separated from the plume, the general far-field advection rules apply:

- The advection of a particle is determined by superimposing the particle's inherent rising or settling velocity $W(i, j)$ on the local ambient current velocities u, v, w.

The separation of the particles from the plume is determined with a probabilistic approach. A separation probability $P_S(i, j)$ is computed at each time step for each size class of each constituent. The state transition from *inside plume* to *out of plume* is checked by drawing a random number, `Rand` (0,1), and comparing it with the separation probability:

With `Inside_Plume` defined as a logical variable, we may write:

```
Inside_Plume =(Rand > Ps).
```

As soon as the particle has been found to separate from the plume (`Inside_Plume = false`), the particle will enter into the far field and no more calculation of the separation probability or checking is required. Thus, a logical variable `Inside_Plume(i,j)` is associated with each particle in the far-field module, defined as `Inside_Plume` = `true` when the particle is released.

The separation probability can be defined as

$$P_S(i, j, t) = \frac{\Delta G}{G(i, j)_t}, \text{ where } \Delta G = G(i, j)_t - G(i, j)_{t+\Delta t}$$

The time t in this equation corresponds to the time since release of the model particle, and $G(i,j)_t$ and $G(i,j)_{t+\Delta t}$ must be determined by interpolation in the time-series $G_{i,j,k}$ of the constituent masses from the preceding near-field simulation on the basis of the corresponding retention times (see Figure 1.2).

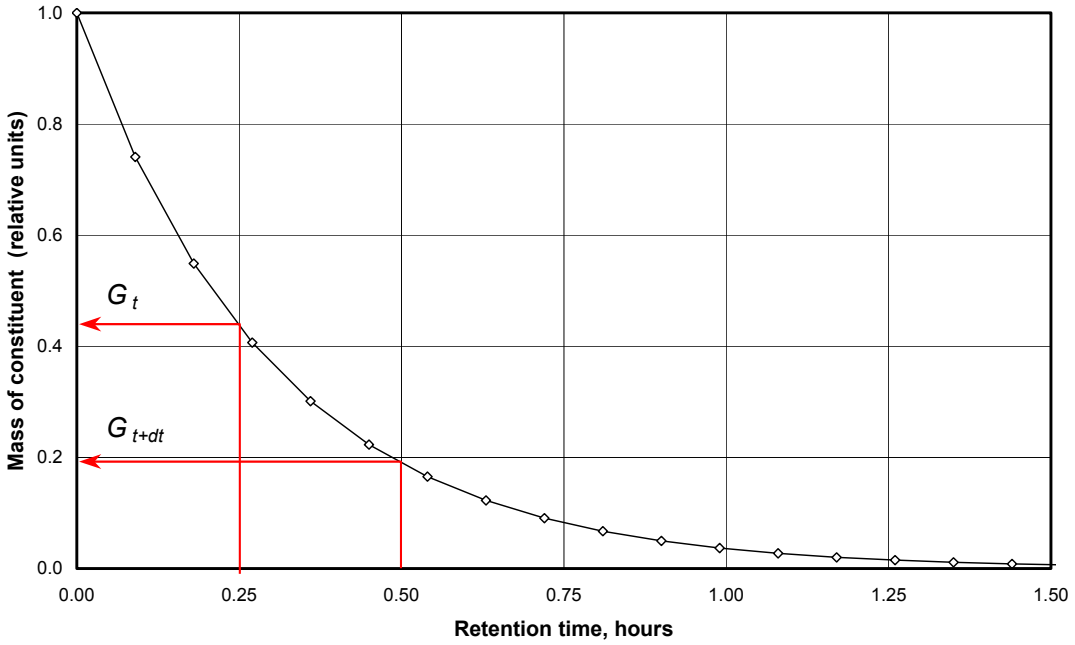

Figure 1.2 Remaining mass of a constituent must be determined by interpolation in the mass vs. retention time arrays computed in the near-field model. In this illustration, the remaining masses G_t and G_{t+dt} are supposed to be determined at the interval of the far-field model timestep. The separation probability for this constituent from one timestep t to the next t + dt will then be $P_S = (G_t - G_{t+dt})/G_t$.

1.4 Handling the post-plume phase

The plume phase might terminate as the plume rises to the sea surface or sinks to the seabed. In other cases, the plume phase can be assumed to terminate when the plume is trapped at intermediate depths (see *Figure 1.3*).

In general, after plume termination, special advection rules will have to be applied for the far-field particles depending on how the plume terminated, as indicated in *Table 1.1*:

Table 1.1 Advection rules for various plume termination scenarios.

Mode of termination	Advection rule
Trapped plumes	The common far-field rules apply after the plume is trapped.
Surfacing plumes	A radial spreading velocity is superimposed on the common far-field advection rule after the plume phase has reached the surface (source flow).
Bottom touching plumes	A radial spreading velocity must be superimposed after the plume phase has reached the seabed (modified source flow including a bottom friction term).

The solution for the surfacing plumes is based on the work of Fanneløp and Sjøen (1980). A solution for bottom touching plumes is developed on the same principles, with the spreading rate "checked" by a bottom friction term. More details on the application of the application of these principles are given in the DeepRisk report (Johansen 2005).

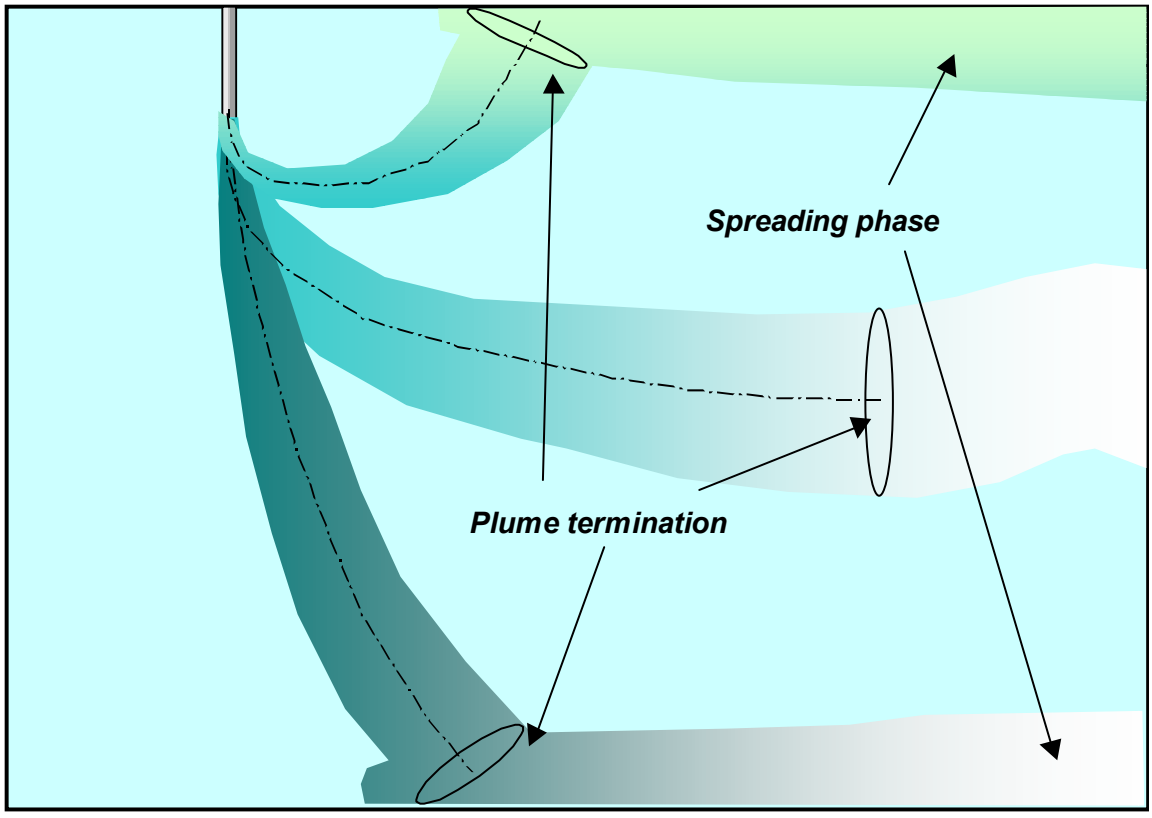

Figure 1.3 Possible plume termination scenarios: surfacing plumes, trapped plumes and bottom-touching plumes. In the case of surfacing and bottom touching plumes, the plume phase will be followed by a spreading phase.

Figure 1.4 shows results from a Plume3D simulation of a bottom touching plume, represented by a possible near sea bed discharge of produced water. The produced water discharge has a significantly higher salinity than ambient sea water (75 ppt compared to 35 ppt for sea water). While the outlet is oriented upwards at an angle of 45 degrees 10 m above the sea bed, the plume will eventually sink to the bottom due to the high density of the release. In both cases, the ambient current is in the order of 10 to 15 cm/s in the north-east direction. The top view shows the interaction of the plume with the sea bed in terms of a lateral dispersion of the model particles, while the side view shows the effect in terms of a mixed layer with increasing thickness in the downstream direction.

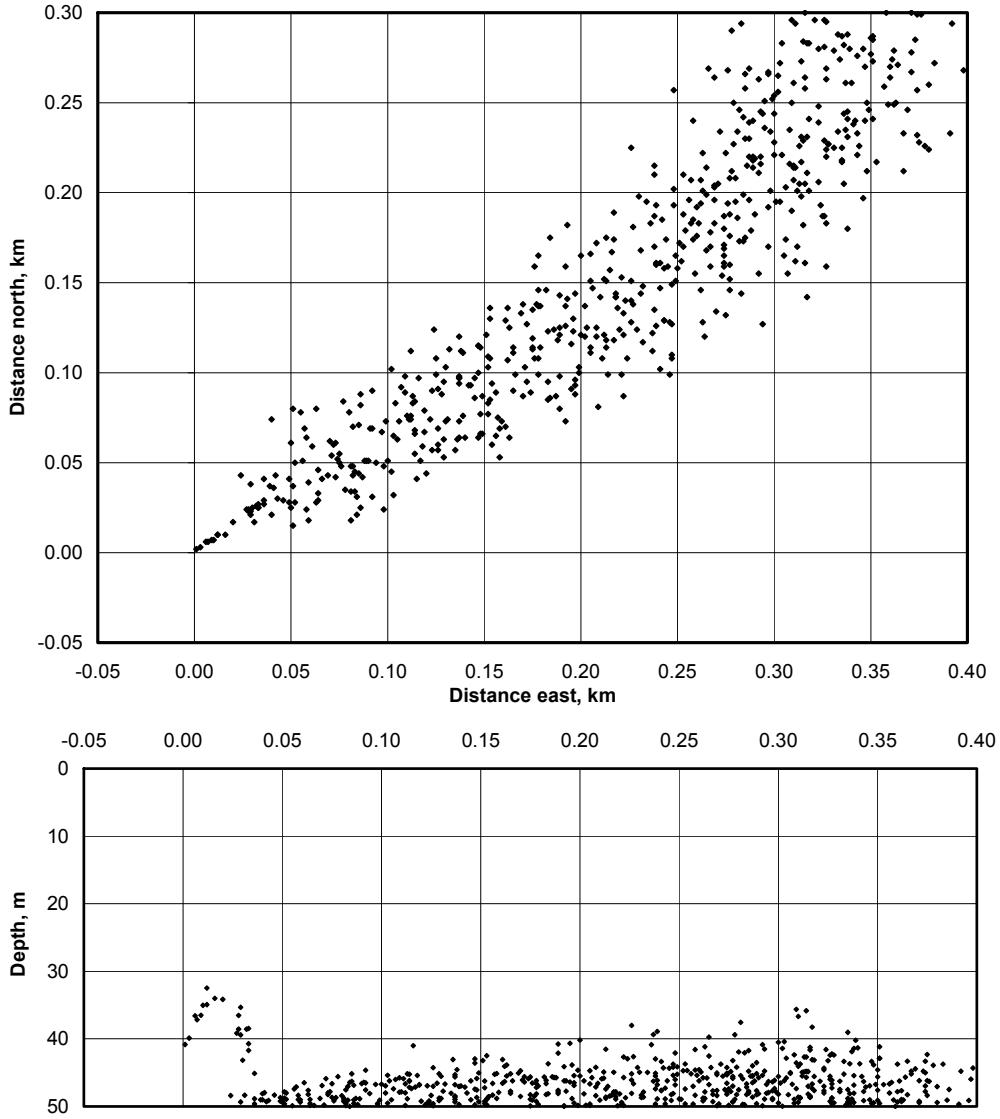

Figure 1.4. Example calculation with Plume3D for a bottom touching plume. Simulation of produced water discharges released 45 degrees upwards 10 m above sea bed in 50 m water depth. Distribution of dispersed oil droplets shown in top view (top) and side view (bottom) 1 hour after start of discharge.

2 IMPLEMENTATION OF PLUME3D IN THE ERMS MODEL

This chapter includes a discussion of the approach that was used in the implementation the Plume3D model in the ERMS model.

2.1 Supplementary input requirements

As a starting point, the main parameters to be provided to run the near field model were considered:

- Water depth at the discharge point
- Orientation of exit jet (horizontal, vertical), exit diameter, and exit temperature
- Composition of the discharge, defined in terms of flow rates of gas, oil, water, dissolved chemicals, and mineral particles, together with the density of each constituent.
- Vertical profiles of ocean currents, temperature and salinity

Some of these parameters are already defined in a general ERMS model simulation, while others have to be provided specifically to run the near field model. For instance, when an oil spill is concerned, both the release rate of oil and the oil type are specified by the user in the general *Release Site* user form, but additional data will be required if the release is taking place subsea (pipeline leak or subsea blowout). To account for this, if the release depth specified by the user is below the sea surface (greater then 0), the user will be presented with an additional *Near Field Info* form (*Figure 2.1*), where data required defining the exit conditions are requested (exit jet orientation, exit diameter, density of gas at standard conditions, and gas to liquid ratio).

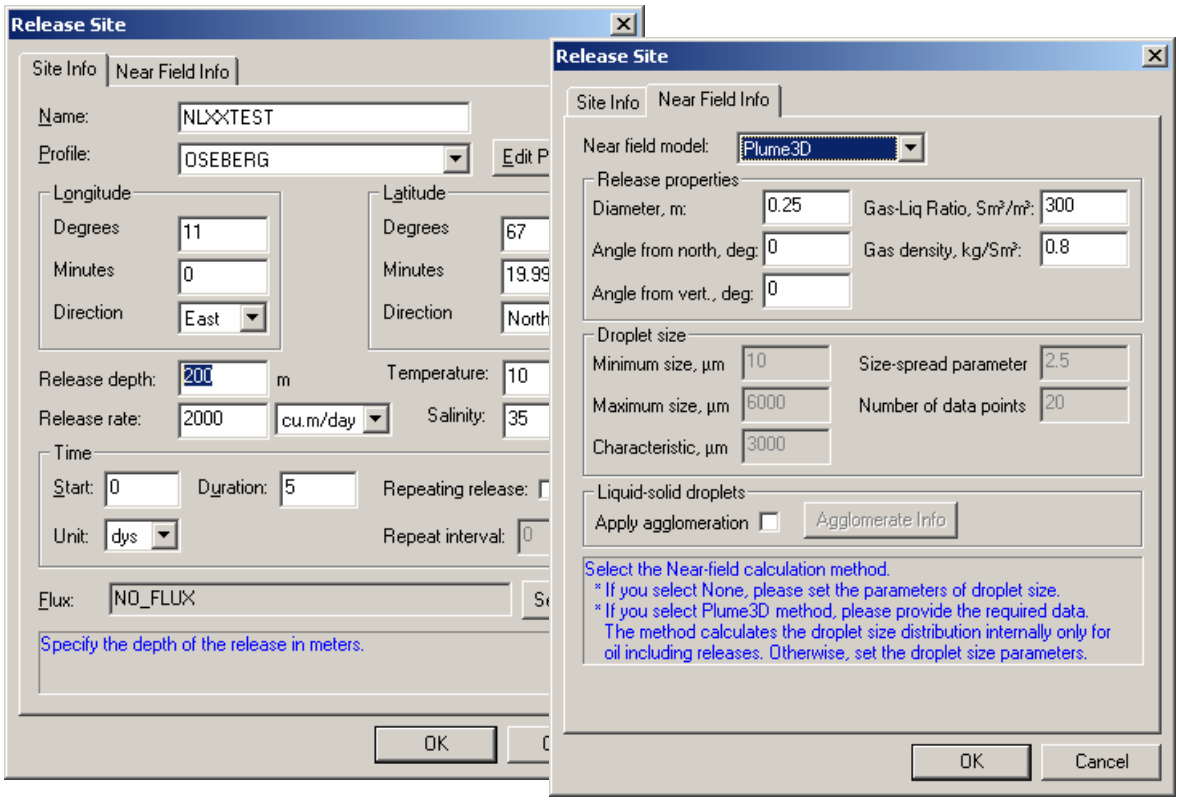

Figure 2.1. The ERMS model Release site form *(left) is extended with a* Near Field Info *form (right) when subsea discharges are involved (Release depth > 0 m).*

To avoid a mismatch with the local depth in the ERMS model, the release depth specified by the user is checked against the water depth at the release location. If the specified depth of release is greater than the local water depth, the release depth is adjusted to the local water depth.

The ERMS model is normally run with time variable 3D ocean currents data, and to account for changes in the currents, a new near field model run is made each time the current is changing (i.e. at the time steps of the current data). The vertical current profile required to run the near field model is based on current data extracted for the spill location. Hydrographic profiles are not generally required in the ERMS model, and a specific user form has been included where the user specifies the profile in terms of sea temperature and salinity at a corresponding set of water depths. This profile is used for all releases in a given ERMS model run (the option of simultaneous simulations of more than one release is enabled in the ERMS model).

It should also be noted that a special *Near Field* user form is added in the *Model Parameter Setup*, but changes in this user form will normally not be required for the common ERMS user. One exception could be in case of particularly shallow discharges with high buoyancy, where shorter time steps than the default setting (0.5 seconds) might be required, or deep water discharges with low buoyancy, where longer time steps might be appropriate. To give the user some control of the performance of the near field model, some of the major plume variables are recorded in a log-file (*Figure 2.2*). The various columns are defined as follows:

Variable name	Explanation
Time (s)	Time from start of the plume simulation
X, Y, H (m)	Plume coordinates in directions east, north and distance from surface.
R (m)	Plume radius
U, W (cm/s)	Horizontal and vertical velocity of plume element
T (^{o}C) and S (ppt)	Temperature and salinity of plume water
Conc (ppm)	Oil concentration in the plume, mg/L
Gas (%)	Mass fraction of gas remaining in the plume
Diss (%)	Mass fraction of gas dissolved in the plume water
Hydr (%)	Mass fraction of gas converted to gas hydrates

NLTEST00.p3dlog - Notepad

File Edit Format Help

Site-1 --- Plume3D profiles at 0.0000 days

Time(s)	X(m)	Y(m)	H(m)	R(m)	U(cm/s)	W(cm/s)	T(degC)	S(ppt)	Conc.ppm	Gas(%)	Diss	Hydr
0.5	0.0	0.0	160.9	0.7	4.3	167.4	7.0	35.0	0.99E+04	100.00	0.00	0.00
3.5	-0.1	0.1	155.6	1.3	5.0	169.8	7.0	35.0	0.23E+04	99.40	0.60	0.00
6.5	-0.2	0.3	150.8	1.9	5.1	153.4	7.0	35.0	0.12E+04	98.81	1.19	0.00
9.5	-0.3	0.4	146.4	2.4	5.1	143.0	7.0	35.0	0.76E+03	98.24	1.76	0.00
12.5	-0.4	0.5	142.2	2.9	5.1	135.6	7.0	35.0	0.54E+03	97.67	2.33	0.00
15.5	-0.4	0.6	138.2	3.4	5.1	130.0	7.0	35.0	0.42E+03	97.12	2.88	0.00
18.5	-0.5	0.8	134.4	3.8	5.2	125.6	7.0	35.0	0.34E+03	96.56	3.44	0.00
21.5	-0.6	0.9	130.7	4.2	5.2	122.0	7.0	35.0	0.28E+03	96.02	3.98	0.00
24.5	-0.7	1.0	127.1	4.7	5.2	119.0	7.0	35.0	0.24E+03	95.48	4.52	0.00
27.5	-0.8	1.2	123.6	5.1	5.2	116.4	7.0	35.0	0.20E+03	94.95	5.05	0.00
30.5	-0.9	1.3	120.1	5.5	5.2	114.2	7.0	35.0	0.18E+03	94.42	5.58	0.00
33.5	-1.0	1.4	116.7	5.9	5.2	112.3	7.0	35.0	0.16E+03	93.90	6.10	0.00
36.5	-1.0	1.5	113.4	6.3	5.2	110.7	7.0	35.0	0.14E+03	93.39	6.61	0.00
39.5	-1.1	1.7	110.1	6.6	5.2	109.2	7.0	35.0	0.13E+03	92.88	7.12	0.00
42.5	-1.2	1.8	106.9	7.0	5.2	107.9	7.0	35.0	0.12E+03	92.37	7.63	0.00
45.5	-1.3	1.9	103.6	7.4	5.2	106.8	7.0	35.0	0.11E+03	91.87	8.13	0.00
48.5	-1.4	2.1	100.4	7.7	5.2	105.7	7.0	35.0	0.97E+02	91.37	8.63	0.00
51.5	-1.5	2.2	97.3	8.1	5.2	104.8	7.0	35.0	0.89E+02	90.88	9.12	0.00
54.5	-1.6	2.3	94.2	8.4	5.2	104.0	7.0	35.0	0.83E+02	90.39	9.61	0.00
57.5	-1.7	2.4	91.1	8.8	5.2	103.3	7.0	35.0	0.77E+02	89.90	10.10	0.00
60.5	-1.7	2.6	88.0	9.1	5.2	102.6	7.0	35.0	0.72E+02	89.42	10.58	0.00
63.5	-1.8	2.7	84.9	9.4	5.2	102.0	7.0	35.0	0.67E+02	88.95	11.05	0.00
66.5	-1.9	2.8	81.9	9.8	5.2	101.4	7.1	35.0	0.63E+02	88.47	11.53	0.00
69.5	-2.0	3.0	78.8	10.1	5.2	101.0	7.1	35.0	0.59E+02	88.01	11.99	0.00
72.5	-2.1	3.1	75.8	10.4	5.2	100.6	7.1	35.0	0.56E+02	87.54	12.46	0.00
75.5	-2.2	3.2	72.8	10.7	5.2	100.2	7.1	35.0	0.53E+02	87.08	12.92	0.00
78.5	-2.3	3.3	69.8	11.1	5.2	99.9	7.2	35.0	0.50E+02	86.63	13.37	0.00

Figure 2.2. Example of log-file produced when the Plume3D model is activated. The log-file is named according to the ERMS scenario and stored in the Modelout *directory.*

Things to look for in the log-file are:

- The plume terminates at intermediate depth with finite rise velocity ($W \neq 0$):
  - Normally, when the plume terminates at intermediate depths, the rise velocity will be zero (trapped plume). Termination at an intermediate depth with a non-zero rise velocity may indicate that the chosen Plume3D time step is too short: With a too short time step, the tracking time may not be sufficient to get the plume to the surface with the maximum permitted number of time steps. Make a new try with a longer timestep (see *Figure 2.3*).
- Each plume is represented with only a few records in the log-file:
  - To obtain a better representation of the plume rise, the plume model *Time step* should be reduced in the Near Field Model Parameter form. The same effect might be obtained with a reduced *Output interval*. Note that the output interval is specified as number of time steps (see *Figure 2.3*).

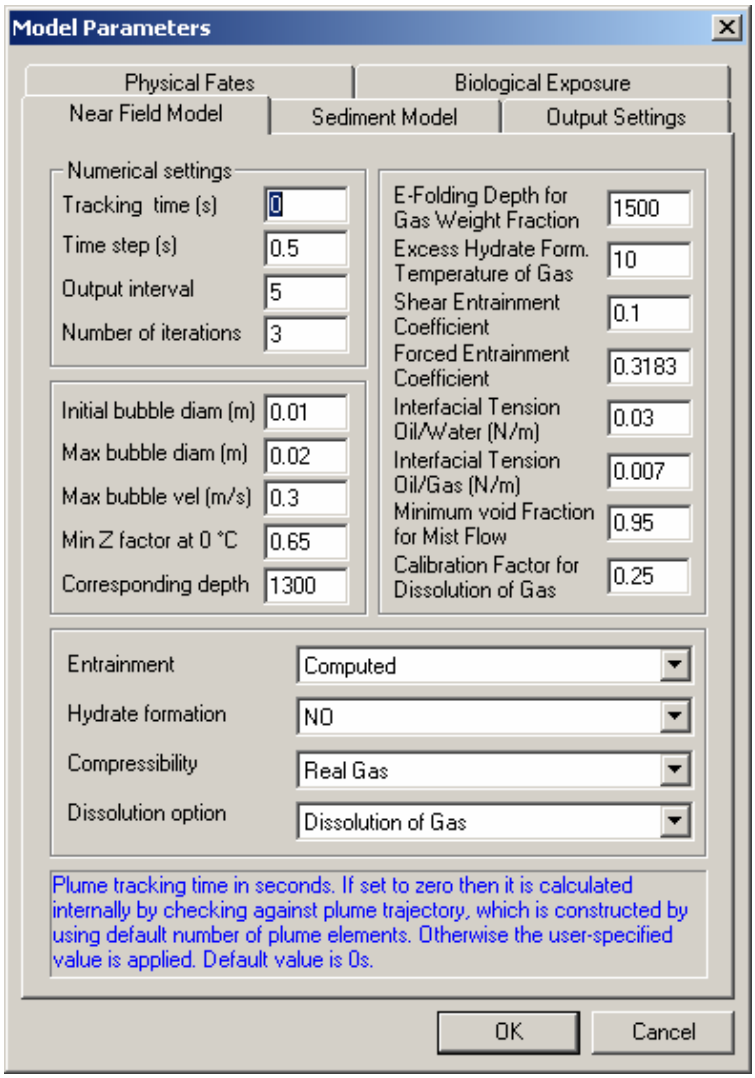

Figure 2.3. Near Field Model Parameter form. The tracking time *should in general be chosen as zero, implying that the plume model will be terminated automatically when the plume is trapped at intermediate depth, or comes to the surface or to the sea bed. The plume model time step may be decreased or increased when required, but the default value of 0.5 seconds is usually a good choice. Note that for the common user, the* time step *and the* output interval *will normally be the only model parameters to be considered changed in the Near Field Model Parameter Setup.*

2.2 Near field – far field coupling

As indicated in the previous chapter, the Plume3D model is implemented as a near-field module, implying that the module is installed as a subroutine in the ERMS model, sharing data internally with the rest of the model. A new plume simulation is made each time the current is changing, providing fresh data on the plume geometry (i.e. the time development of the plume trajectory and plume radius, see *Figure 2.2*), in addition to an inventory of the remaining mass fractions of the different constituents in the plume (gas, oil, mineral particles). In the ERMS model, new particles are released at arbitrary times t_{rel} within each model timestep Δt. If the drift time $t = \Delta t - t_{rel}$ of the particle at the end of the timestep is shorter than the plume rise time t_p, the position allocated to the particles is determined by interpolation in the stored plume geometry. If t is larger than t_p, the particle position will be determined by one of the far-field advection rules defined in Table 1.1 (see *Figure 2.4*).

For trapped plumes, the point of trapping is used as a starting point for the particle. For surfacing or bottom touching plumes, the particle starting position is defined by the surfacing position or the bottom touching position of the plume. The position of the particle at the end of the timestep is then determined by adding the subsequent displacement of the particle in the remaining time t' of the timestep, $t' = \Delta t - t_{rel} - t_p$ due to the ambient current, the rise or sink velocity of the particles and the eventual radial current induced by the surfacing (or bottoming) of the plume.

In the following time steps, particles with lifetime t less than the plume rise time t_p continues to be positioned by interpolation in the stored plume geometry data. However, as explained in the previous chapter, buoyant particles may sink or rise out of the plume at an earlier stage. This is accounted for by the probabilistic approach described previously, utilizing stored information from the plume model on the mass fraction remaining in the plume of the various constituents as a function of time.

Thus, in short, when inside the plume, particles are positioned by interpolation in the stored plume geometry data. Particles may leave the plume either due to "time out" (drift time exceeds the plume rise time), or due to "leakage" (buoyant particles escape from the plume). Once the particles are outside the plume, the particle motion is determined by the far-field "advection rules" (ambient current, particle sinking/rising velocity, radial displacement due to surfacing/bottoming of the plume).

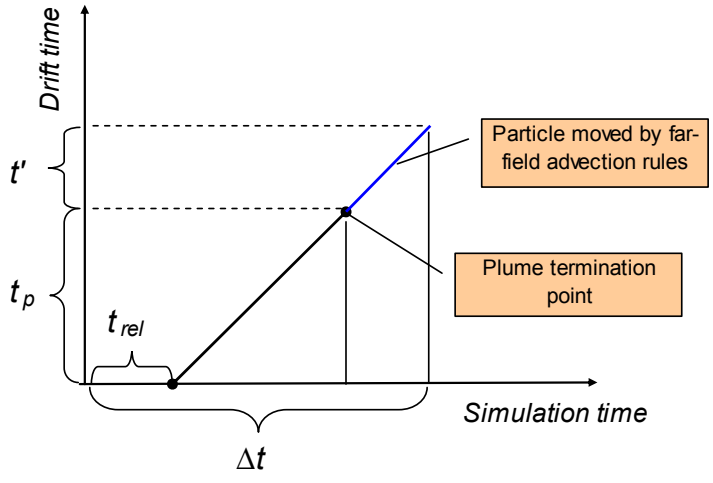

Figure 2.4. Advection of particles after termination of plume

3 APPLICATIONS OF THE NEAR FIELD MODULE

As mentioned earlier, the Plume3D was initially developed for simulating sub sea blowouts of oil and gas in deep water (i.e. the DeepBlow model), but has later been extended into a general multiphase plume model to be utilized also for produced water discharges and drilling discharges.

In absence of a near field model, produced water discharges would be simulated as diffuse discharges transported by the ambient current and dispersed by oceanic turbulence. For drilling discharges, a vertical settling velocity would be added to represent the settling velocity of the individual grains. The inclusion of a near field model makes it possible to account for drift and spreading that are induced by the initial momentum and buoyancy of the discharge. Some examples are shown in the following to demonstrate the importance of the near field processes.

3.1 Produced water discharges

A produced water discharge of 18 000 tonnes per day has been chosen as the first example of the application of the near field model. The produced water is discharged vertically downwards at 40 m depth with a temperature of 75°C and a salinity of 48 ppt. The discharge contains 10 % by volume of air (gas to liquid ratio 0.1 Sm^3/m^3). A summer-stratified temperature and salinity profile is chosen in the ambient water. *Figure 3.1* shows the situation about half an hour from the start of the release. The solid dots on the map represent dissolved components in the discharge; while the concentration of the dissolved components is shown by coloured cells (see legend). A vertical cross section is inserted (indicated by arrow on the map), to show the side view of the plume of produced water. Due to the stable stratification, the plume is trapped at a depth of about 20 meters.

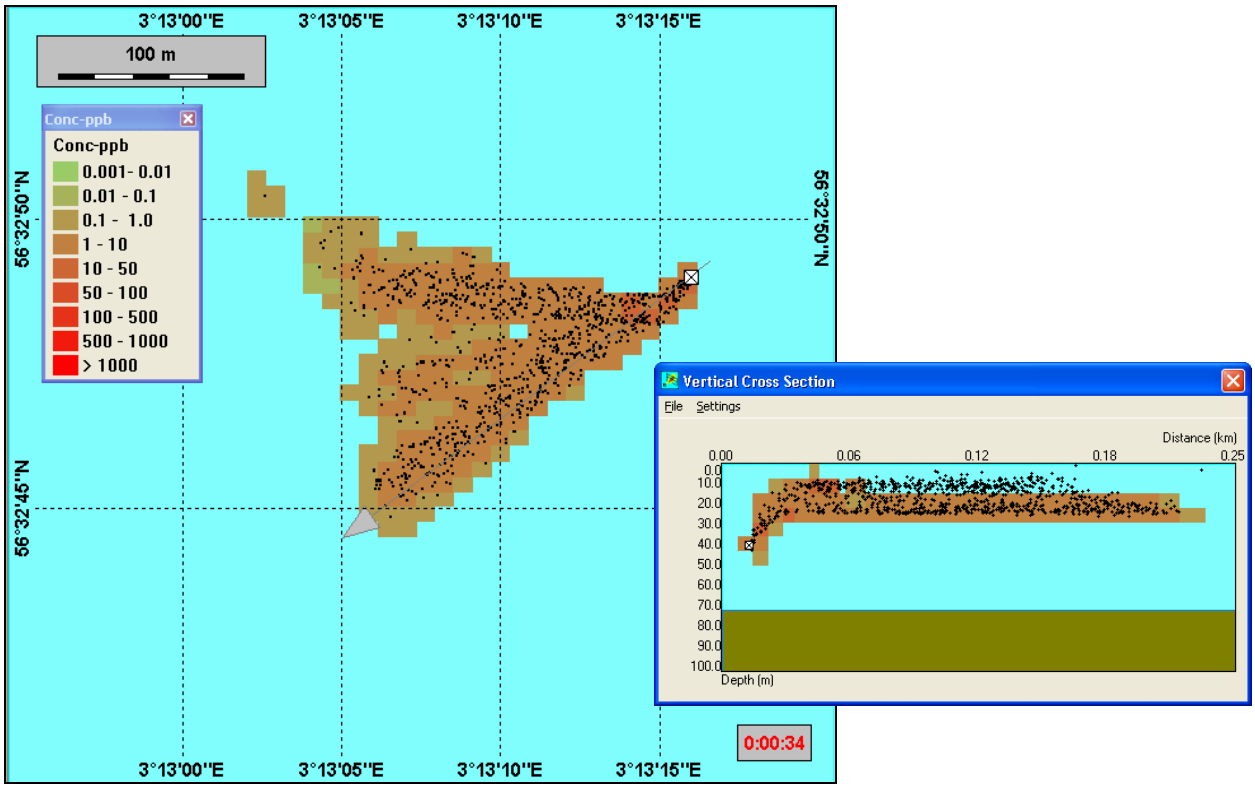

Figure 3.1. Simulation of a produced water discharge. The discharge conditions are given in the text.

3.2 Drilling discharges

A drilling discharge of 1600 tonnes during four days has been chosen as the second example of the application of the near field model. The drilling material is discharged horizontally at 30 m depth. A summer-stratified temperature and salinity profile is chosen in the ambient water.

Figure 3.2 shows the results about half an hour from the start of the release. The black dots on the map represent dissolved components in the discharge (drilling chemicals); while the brownish dots represent mineral grains (drilling mud and cuttings). A vertical cross section is inserted (indicated by the arrow on the map) to show the side view of the plume. The picture shown in the vertical cross section shows the particles in the discharge, and indicates that the coarse, grained material sinks out of the plume, while the finest grains (mud) stays in the plume, which is trapped within the water column at a depth somewhat below the release depth (30 m).

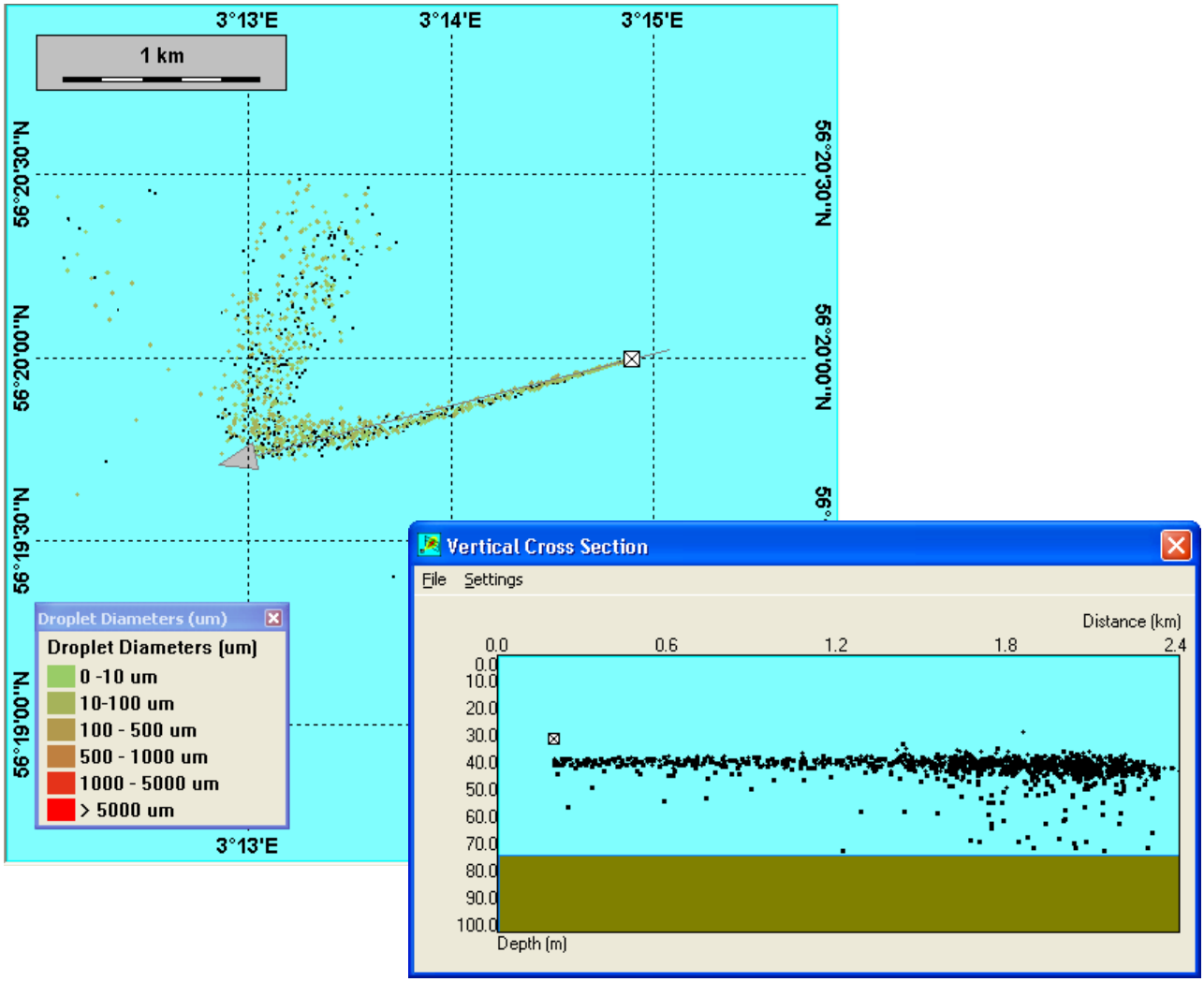

Figure 3.2. Simulation of a drilling discharge about one day after start of release. The discharge conditions are given in the text. Discharge point is shown with a cross within a square.

3.3 Subsea blowout

To complete the picture, we have included simulations of a subsea oil and gas blowout as a third example of possible applications of the ERMS near-field module. The blowout takes place at the sea bed in a water depth of 430 m. The oil discharge rate is chosen as 2600 m^3/day. The gas-to-oil ratio is 300 Sm^3/m^3.

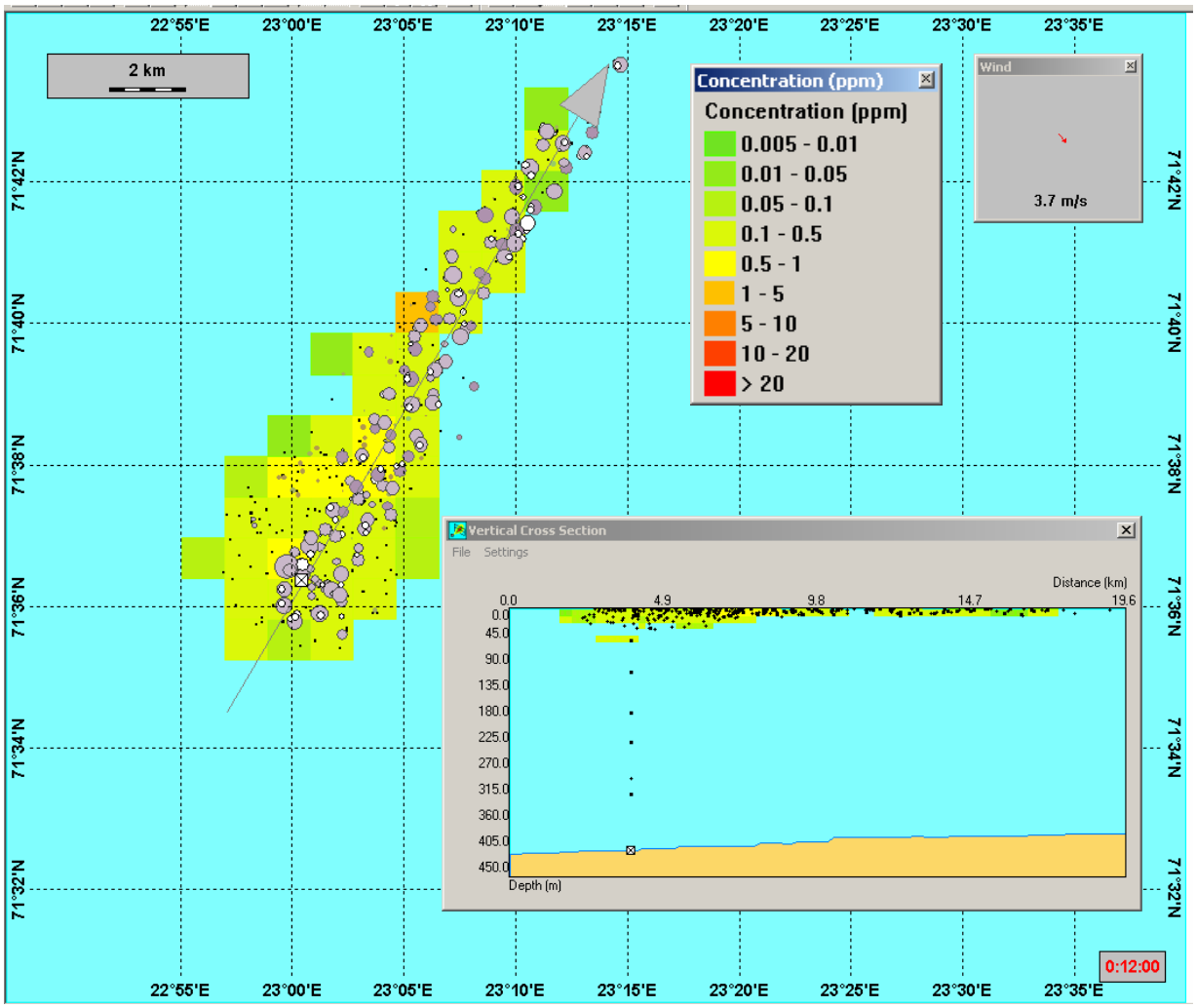

Figure 3.3. Simulation of a subsea oil and gas blowout from 430 m depth. Results are shown 12 hours after the start of the release. The map shows the distribution of particles representing surface oil (circles with shades of grey), dispersed oil droplets (circles in shades of brown), and dissolved oil (black dots). Maximum total hydrocarbon concentrations are shown by colour (see concentration legend). A vertical transect is inserted, showing a side view of the rising plume and the cloud of dispersed droplets and dissolved oil components.

4 REFERENCES

Fanneløp, T.K. and K. Sjøen, 1980: Hydrodynamics of underwater blowouts. *Norwegian Maritime Research*, No. 4, pp. 17-33.

Johansen, Ø., 1997: Hydratdannelse og dråpestørrelse ved dypvannsutblåsninger. SINTEF Report STF66 F97082 (in Norwegian), SINTEF Material and Chemistry, Trondheim, 55 pp.

Johansen, Ø. 2000: DeepBlow, a Lagrangian Plume Model for Deep Water Blowouts Spills, Science & Technology Bulletin, Vol. 6, No. 2, pp.103-111.

Johansen, Ø., H. Rye, C.Cooper, 2003: DeepSpill—Field Study of a Simulated Oil and Gas Blowout in Deep Water, Spill Science & Technology Bulletin, Vol. 8, No. 5–6, pp. 433–443.

Johansen, Ø., 2003: Development and verification of deep-water blowout models , Marine Pollution Bulletin vol. 47, pp. 360–368.

Johansen, Ø., 2004: Physical/Chemical Fate of Gas from Deep Water Blowouts. SINTEF Report STF66 F04050, SINTEF Material and Chemistry, Trondheim, 34 pp.

Johansen, Ø., 2005: Recent Advances In Deep Water Plume Modelling, Deep Risk – Activity 2b. SINTEF Report STF80MK F05353, SINTEF Material and Chemistry, Trondheim, 27 pp.