Final Report Submitted to

The Cooperative Institute for Coastal and Estuarine Environmental Technology (CICEET)

Project Title: A Module for NOAA's GNOME Model to Provide Capability to Simulate Deepwater Oil and Gas Spills

Principal Investigator :
Dr. Poojitha D. Yapa
Professor of Civil and Environmental Engineering
Clarkson University
Potsdam, NY 13699-5710
Phone: 315-268-7980; *Fax:* 315-268-7985

September 30, 2003

I. Accomplishments

A, B, and C: Scheduled Tasks, their Progress, and Difficulties Encountered

The table below shows the Scheduled tasks and their progress. The text below the table comments on each item and explains any difficulties encountered.

	Item	Progress
1	Review literature on oil droplet size distribution	Done
2	Review GNOME structure	Done
3	Formulate methods to estimate the oil droplet distribution and verify with data	Done
4	Implement the above formulation to CDOG	Done
5	Make Changes to CDOG necessary to make linking to GNOME possible	Done
6	Work with NOAA Staff to make necessary changes in GNOME	Done

Review GNOME structure

GNOME executables, user manuals were downloaded. The source code was obtained from NOAA Staff. GNOME structure was thoroughly reviewed.

Review literature on oil droplet size distribution and implement methods to estimate the oil droplet distribution within CDOG

We reviewed a large number of papers on this subject. Data available is not as good as one wants on this aspect. Using the limited data available an algorithm was formulated to calculate the drop sizes. It is now implemented in CDOG. We discuss the available literature and the algorithmic details in a paper that is under preparation to be submitted to a major journal. Excerpts of this paper are included in this report in Appendix II. During the DSTF workshop (see below) the lack of knowledge and data in this particular subject was also identified as an area further research.,

Make Changes to CDOG necessary to make linking to GNOME possible

Many telephone conferences were held between the NOAA staff and Clarkson staff to decide on the data that will be integrated between CDOG and GNOME. In addition to the telephone conferences, many e-mails were exchanged to decide on these issues. The process took more effort than originally anticipated due to following reasons: a) inherent differences

between the way CDOG outputs data and what is expected by GNOME (CDOG mainly deals with the water column and is three-dimensional where as GNOME deals with the water surface and is two-dimensional); b) the differences in the grid system between the models for input data; and c) to make sure that allowances are made for future expansions. There were several rounds of data file exchanges to help make these decisions. Every round involved modifications to CDOG code.

Work with NOAA staff to make necessary changes to GNOME

We worked with several NOAA/HAZMAT staff members. Many telephone meetings were held and many e-mails were exchanged to decide on the necessary changes. The issues included data formats and ranges. The personnel at NOAA included C.J. Beegle-Krause, Caitlin O'Connor, Debbie Payton, Glen Watabayashi, and Bill Lehr. There was very close communication with C.J. Beegle-Krause, Caitlin O'Connor. GNOME now has a version where CDOG is integrated into it.

Summary comments

A workshop was sponsored by Deep Spill Task Force (DSTF) and was on April 3 and 4, 2003 at NOAA/HAZMAT facilities in Seattle, WA. Using the Integration of CDOG and GNOME, 13 scenarios were simulated and presented. The audience included the administrators and the end users (e.g. MMS, NOAA, oil companies) of the product.

The results of the work from this project was published as a paper in the proceedings of the Arctic Marine Oil Spill Program Technical Seminar (AMOP Conference) held in Victoria, Canada during June 10-12, 2003. The paper was also orally presented at the same conference.

CDOG is now available as a module from GNOME. Appendix I gives details of a paper published at AMOP conference on integration of CDOG and GNOME.

I. E and F : Data, Manuscripts, Reports, and Presentations

These items are attached as Appendices I and II.

II. Tasks and Activities for Next Reporting Period

All tasks have been accomplished.

III. Expenditures

The expenditures are in the same range as the anticipated costs for the tasks accomplished.

APPENDIX I

Integration of the CDOG Deep Water Oil and Gas Blowout Model with the NOAA GNOME Trajectory Model

Poojitha D. Yapa, Fanghui Chen
Department of Civil and Envir. Engineering
Clarkson University, Potsdam, NY 13699, USA
pdy@clarkson.edu

C. J. Beegle-Krause
NOAA/ORR/HAZMAT
7600 Sand Point Way NE, Seattle, WA 98115, USA

Abstract

CDOG (Clarkson Deepwater Oil and Gas) model simulates the behavior of oil and gas released from deepwater. The model integrates the hydrodynamics of the jet/plume, the thermodynamics, and the hydrate formation and decomposition. CDOG simulates both the dynamics of the jet/plume phase and the far-field conditions during the rise of oil/gas/hydrates through the water column. CDOG allows gas/hydrates to separate from the main plume if there are strong cross currents. Despite the comprehensive nature of CDOG, it is designed to simulate the oil and gas behavior only until they reach the water surface. CDOG at present does not include weathering processes once oil starts floating on the water surface. No Graphic User Interface (GUI) or data visualization were included in the original CDOG design. GNOME (General NOAA Oil Modeling Environment) is the trajectory model developed and used by NOAA for spill response. GNOME has several user "modes" to allow users of different levels of sophistication to setup and use the model. GNOME is designed to be flexible; allowing the model to use circulation fields from a number of different models in different formats, and includes a number of diagnostic tools. This paper reports the integration of CDOG with GNOME so that users can run CDOG using a GUI and then easily transition to running GNOME for the surface trajectories. This integration allows the users to run complete scenarios – the types that are required in comprehensive contingency planning.

1 Introduction

With the increasing oil exploration and production from deepwater locations, models are needed for emergencies and contingency planning. About 5 years ago, the Minerals Management Services of the U.S. Department of the Interior in co-operation with a consortium of 22 oil companies embarked on a plan to develop a computer model that can simulate oil and gas behavior from a deepwater blow out. This model, the Clarkson Deepwater Oil and Gas (CDOG) model, is now complete. The model simulations have been compared with consortium sponsored field experiments codenamed "Deepspill" off the coast in Norway.

CDOG simulates the behavior of oil and gas released from deepwater. The model integrates the hydrodynamics of the jet/plume, the thermodynamics, and the hydrate formation and decomposition. CDOG simulates both the dynamics of the jet/plume phase and

the far-field conditions during the rise of oil/gas/hydrates through the water column. CDOG allows gas/hydrates to separate from the main plume if there are strong cross currents. Gas dissolution during travel is also simulated. Despite the comprehensive nature of CDOG, it is designed to simulate the oil and gas behavior only until they reach the water surface. CDOG at present does not include weathering processes once oil starts floating on the water surface. No Graphic User Interface (GUI) or data visualization were included in the original CDOG design.

GNOME (General NOAA Oil Modeling Environment, Beegle-Krause 2001) is the trajectory model developed and used by NOAA for spill response. It is available as a public domain model for analyzing the behavior of oil during a real or potential oil spill, and as a drill scenario tool. GNOME was designed with "usability" in mind for both responders and the public. The model has several user "modes" to allow users of different levels of sophistication to setup and use the model. GNOME is designed to be flexible; allowing the model to use circulation fields from a number of different models in different formats, and includes a number of diagnostic tools.

A project is now in progress to integrate CDOG with GNOME so that users can run CDOG using a GUI and then easily transition to running GNOME for the surface trajectories. This integration allows the users to run complete scenarios – the types that are required in comprehensive contingency planning. This paper reports on the integration process.

2 Characteristics of a Deepwater Blowout Model

A model for simulating the behavior of oil and gas from a deepwater blowout differs from those developed for simulating oil or gas from relatively shallow water depth in several aspects. Some of the key features are listed below.

i. Under high pressure and low temperature, gas may be converted to a solid like phase called gas hydrate. Gas hydrates consist of gas and water, and are a slush like (similar to frazil ice) compound.

ii. Free gas may dissolve into water during its long journey to the surface and thus change the buoyancy of the jet/plume as well.

iii. Under high pressure, gas behaviour is better described by a non-ideal gas state equation instead of an ideal one.

iv. Size of gas bubbles and their buoyant velocities (or slip velocities) can no longer be approximated as constant, considering the processes of gas hydrate formation, decomposition, gas dissolution, and gas expansion.

v. Gas may escape from a jet/plume due to the gas bubble slip velocity if the jet/plume is significantly bent in a cross flow.

3 CDOG Model description

CDOG model formulation was presented in Yapa et al. (2001). An updated and more detailed formulation can be found in Zheng et al. (2003). Complete model formulation details are beyond the scope of this paper. A brief description of the model is provided in this section. CDOG model integrates hydrodynamics and thermodynamics of the jet/plume, the thermodynamics and kinetics of hydrate formation and decomposition, and gas dissolution. The model uses an improved integrated formulation for computing buoyant velocity of gases and hydrates. The model also uses an improved integrated formulation to compute dissolution that gives good results even in deepwater conditions, where the behavior of gas

may not be ideal. The model can simulate the behaviour of oil and gas in strong cross flow conditions where gases may separate from the main plume. Each CDOG module was tested by comparing the computational results with available data.

3.1 Simulation of Hydrate Formation and Decomposition

To model the hydrate formation rate in the gas phase of the jet/plume, three processes need to be considered: hydrate kinetics, mass transfer, and heat transfer. Mass transfer transports gas for hydrate formation to the point of reaction. Heat transfer re-distributes the heat released from hydrate formation around the solid hydrate to change the water temperature. No previous research has taken into account all of the above three processes when simulating the hydrate formation in a deepwater gas plume.

The model assumes that the gas molecules diffuse through the porous hydrate shell due to the concentration gradient of gas, and react with water to form hydrates at the hydrate-water interface. The latent heat of hydrate formation is included in the heat transfer computations. CDOG uses the method of Englezos et al. (1987a, 1987b) for hydrate kinetics and then integrates it with the mass and heat transfer phenomena to model the hydrate formation. In this method the fugacity difference is the main driving force for hydrate formation. CDOG models decomposition of gas hydrates by the method described by Kim et al. (1987). The main differences are the fugacity terms are now reversed. The hydrates module has been verified by using the limited available data (Zheng, et al. , 2003)

3.2 Computation of Dissolution Gases

In models that considered the gas dissolution, the dissolution mass transfer rate was computed using the simple Henry's law. In deepwater, the solubility is affected by high pressures. The dissolution computed by using simple Henry's law deviates significantly from the actual values in ultra-high pressure conditions.

In deepwater, the behavior of gas is non-ideal due to high ambient pressure. The solubility of gas in water is strongly dependent on the ambient pressure, temperature, and salinity. Gas bubbles may experience significant variations in sizes and shapes because of gas expansion and dissolution. Therefore, ways of estimating solubility and mass transfer coefficient need to be improved. The deviation of a real gas from the ideal gas can be described by the compressibility factor Z. Complete details of dissolution computations are given in Zheng and Yapa (2002).

3.2 Buoyant velocity of oil, gas, and hydrate particles/bubbles

The buoyant velocities of oil, gas, and hydrate droplets/particles are needed to simulate their movement under field conditions beyond the jet/plume phase. The buoyant velocities are also needed to simulate the slip between these droplets/bubbles and the main plume fluid during the plume phase.

There are a large number of models that use a two equation approach to calculate the buoyant velocity of a particle (solid, liquid, or gas). In this approach the buoyant velocity of a particle is estimated by assuming the particle to be spherical and rigid and applying the force balance between buoyancy and drag forces. The drag coefficient C_D is approximated as an inverse function of Reynolds number R for small R values and C_D as a constant for larger R values. The resulting commonly used 2-equation approach: Stokes and Reynolds equations, has been used in many oil and gas spill models (e.g. Yapa et al., 1999; Johansen, 2000).

Based on the work of Clift et al., (1978), Zheng and Yapa (2000) described an integrated 8 equation approach to calculate the velocity of bubbles/droplets in a liquid. This approach accounts for the increase in drag at larger diameters. They also showed through comparison with experimental data that this integrated approach gives excellent comparison and better results than the two-equation approach for bubble/droplet velocity and covers wider range of sizes/shapes (spherical, ellipsoidal, and spherical-cap). Details of the compuational method and comparisons can be found in Zheng and Yapa (2000).

3.3 Jet/plume model with integrated thermodynamics and gas hydrate kinetics

The hydrate thermodynamics, formation and decomposition kinetics, heat and mass transfer and the algorithms for rise velocity and free gas dissolution are integrated with the jet/plume hydrodynamics to form the model.

The model uses the Lagrangian integral control volume (CV) approach (Yapa and Zheng, 1997). The Lagrangian integral control volume method simulates the gross behavior of the plume at a cross section. Gas bubble size is part of the input data. Gas bubble size distribution inside a given CV at a given time is assumed to be uniform, but the gas bubble size can vary with time due to dissolution and hydrate formation. The larger particles influence the plume behavior more. For gas bubbles in 5~10 mm diameter range in seawater, Zheng and Yapa (2000) showed that the slip velocity is approximately constant. Therefore, the assumptions above are reasonable.

3.4 Governing Equations for Main Jet/Plume

Without a strong crossflow, the gas phase is expected to occupy the inner core of the plume (Yapa and Zheng, 1997). The number of bubbles in a control volume (CV), N, is given by $\frac{J_N h}{w + w_b}$, where, J_N= bubble number flux; h = height of a CV; w = vertical velocity of plume liquid; and w_b = gas slip velocity.

Under strong crossflow conditions, the jet/plume will bend significantly soon after release. The slip velocity between the gas phase and liquid phase causes the gases to move to the upper boundary side in a bent jet/plume. This can cause separation of gas from the main jet/plume. The model is based on a Lagrangian frame moving with the CV and keeps updating the mass of each phase in CV. In a bent jet/plume with gas separation taken into account, the number of bubbles in a CV is $N = \frac{f \cdot J \cdot h}{v_j + w_b \sin j} = f \cdot J \cdot t$, where τ is the time taken for one bubble to travel through the length of the CV; v_j=the jet/plume velocity; f = the angle of the jet/plume axis from the horizontal; f = a fraction that represent the gas portion left in CV, J = the number flux of bubbles. If there is no gas separation $f = 1$, $J = J_N = constant$. f, J, and t determine the change of gas mass in a CV. The following equations are applied to a CV.

3.4.1 Conservation of Liquid Mass

$$\frac{dm_l}{dt} = r_a Q_e - f \cdot J \cdot t \cdot n_h \frac{dn}{dt} M_w \qquad (1)$$

where m_l = the liquid mass in CV [kg] = $r_l p b^2 (1-be) h$; b = the ratio between the cross-sectional area occupied by gas (inter-dispersed with liquid) and the cross-section area of the

CV (Yapa and Zheng, 1997*);e* = volume fraction of gas bubbles with hydrate shell, where $e = \frac{r_l - r}{r_l - r_{com}}$, $r_{com} = \frac{r_b r_b^3 + r_h (r_h^3 - r_b^3)}{r_h^3}$, and r_l, r, r_{com}, r_b, r_h, and r_a = densities respectively of the liquid part of CV, gas-liquid mixture in plume, combined gas and hydrate shells, gas, hydrate, and ambient fluid [kg/m^3]; r_b, r_h = the inner and outer radii of a gas bubble with a hydrate shell, if no hydrate then $r_h = r_b$; Q_e = entrainment rate for ambient water [m^3/s]; n_h = hydrate number; M_w = molecular weight of water [kg/mol]; dn/dt = hydrate formation rate for one bubble [mol/s]; $dn/dt > 0$ for hydrate formation and < 0 for hydrate decomposition. The last term of the right-hand side of Eq. 1 represents the rate of loss/gain of water mass due to hydrate formation/decomposition

3.4.2 Loss of Gas Mass Due to Hydrate Formation and Free Gas Dissolution

$$\Delta m_b = -f \cdot J \cdot t \cdot (\frac{dn}{dt} + \frac{dn_s}{dt}) M_g \Delta t \qquad (2)$$

in which Dm_b = loss of gas mass due to hydrate formation and dissolution in CV [kg] and m_b = the mass of gas in CV [kg]; M_g = molecular weight of gas [kg/mol]; $\frac{dn_s}{dt}$ = rate of gas dissolution for one gas bubble[mol/s]; Dt = time step [s].

3.4.3 Conservation of Momentum

The momentum equations are applied to the average conditions within a CV, i.e. the distribution within the CV is ignored. However, the slip velocity between gas/hydrate and liquid is taken into account. An assumption made here is that the drag force due to the change of the flow field is not significant.

$$\frac{d}{dt}[(m_l + m_b + m_h) \cdot u] = u_a r_a Q_e - u r_{com} Q_g \qquad (3)$$

$$\frac{d}{dt}[(m_l + m_b + m_h) \cdot v] = v_a r_a Q_e - v r_{com} Q_g \qquad (4)$$

$$\frac{d}{dt}[m_l w + (m_b + m_h)(w + w_b)] = w_a r_a Q_e - w r_{com} Q_g + (r_a - r_l) g p b^2 (1 - b e) h + (r_a - r_{com}) g p b^2 b e h \qquad (5)$$

in which m_h = hydrate mass in a CV [kg]; u, v, w = the cross sectional averaged velocity of the CV in three orthogonal directions, and r_{com} = the composite density of gas bubble with hydrate shell, and Q_g = the volume flux of gas (with possible hydrate shell) going out of the CV. The slip velocity, w_b, is calculated using the composite density of the gas and hydrate. The first terms on the right-hand side of Eqs. 3 to 5 represent the momentum from the entrained liquid mass. The second terms in Eqs. 3 to 5 represent the loss of momentum due to gas (in gas or hydrate form) that moves outside the jet/plume boundaries. The third term in Eq. 5 is related to the vertical force acting on the liquid part. The last term in Eq. 5 is related to the vertical force acting on the gas bubbles including hydrate shells. complete details on computing b and Q_g were give in Chen and Yapa (2001).

3.4.4 Conservation of Heat

$$\frac{d}{dt}[(C_{pl}m_l + C_{ph}m_h)T] = C_{pl}T_a r_a Q_e + f \cdot J \cdot t \cdot \frac{dn}{dt} l \quad (6)$$

in which C_{pl} , C_{ph} = specific heat of liquid and hydrate at constant pressure [J/kg·K]; T = temperature of plume [K]; T_a = temperature of ambient fluid [K]; λ = latent heat of hydrate formation or decomposition [J/mol].

The heat content of gas is ignored in Eq. 6. This term is very small compared to the others. The first term of the right-hand side of Eq. 6 represents the heat input from entrained water. The second term represents the change in heat energy due to gain or loss in the form of latent heat during hydrate formation or decomposition contributed by the gas portion of CV.

3.4.4 Conservation of Salinity, and Oil Mass

$$\frac{d(m_l I)}{dt} = I_a \frac{dm_l}{dt} \quad (7)$$

in which I = a symbol representing salinity, S, or oil concentration by mass C (depending on the property the equation describes). Eq. 7 states that the change of salinity or oil mass in the CV is due to the input contributed by the entrained mass.

3.4.5 Entrainment

The entrainment of ambient water into the plume (Qe in Eq. 1) impacts the fate of the jet/plume. Therefore, computing the entrainment amount Qe is important. Many models used constant coefficient methods to compute entrainment. However, with the constant coefficient method case by case evaluation of the coefficient is needed. Lee and Cheung (1990) computed the entrainment as the sum of shear-induced entrainment and forced entrainment. Forced entrainment was computed based on the ambient flow interception in the "windward" side of the buoyant jet. Lee and Cheung's (1990) algorithm was adopted and extended to full 3-D by Yapa and Zheng (1997). The comparisons between laboratory and field experiments with computer simulations using this entrainment yielded very good results (Zheng and Yapa, 1998; Yapa et. al., 1999). This algorithm does not need a change of coefficients from case to case. The same formulation is used in this paper to compute entrainment.

3.5 Modeling the Gas Separation

Gas separates because it gradually moves to one side of the jet. CDOG allows gases and hydrates to separate from the main plume if the release is in a strong cross flow condition. The model assumes that the space occupied by gas (this is an area of gas/fluid mix at a cross section in CV) expands at the same expansion rate as the overall jet/plume of CV, preserving the shape (part of a circle); hence the gas volume fraction (ε) in gas portion may decrease (dilute) due to turbulent entrainment. Details of the algorithm, especially the numerical implementation, are given in Chen and Yapa (2001).

4 Comparison of CDOG Simulations with "Deepspill" Field Experiments

CDOG was used to simulate and compare with large scale field experiments code named "Deepspill". Detailed simulations of "Deepspill" experiments using CDOG Model and the comparisons can be found in Chen and Yapa (2003). In this paper an excerpt of these

simulations are shown to demonstrate the level of comparison that can be expected. A description of the "Deepspill" is given in the next section based on the SINTEF report (Johansen et al. 2001).

4.1 A description of "deepspill"

The "Deepspill" project was organized as a Joint Industry Project (JIP) involving 22 oil companies and the Minerals Management Service (MMS) of the US Department of the Interior. The 4-day field experiment from June 26 to June 29, 2000 cost US $ 2.5 million. The "Deepspill" was conducted in the Norwegian Sea at the Helland Hansen site (65º00'N, 04º50'E) and included four controlled discharges of oil and gas from a water depth of 844 meters. The main objective of the experiments was to obtain data for verification and testing of numerical models for simulating accidental releases in deep waters. In addition, the experiments were aimed at testing equipment for monitoring and surveillance of accidental releases in deep waters, and evaluation of the safety aspect of accidental releases of gas and oil in deep waters. Details of the "Deepspill" field experiments can be found in Johansen et al., (2001)

4.2 Summary of input conditions for cdog model simulations

This section describes the basic information and key input data needed to run the CDOG model for "Deepspill" experiments. Deepspill consisted of one pilot experiment (Nitrogeon gas), two oil and gas mixture releases, and one gas only release. Since, the major expected use after integration is the behaviour of oil when an oil/gas mixture is released, we will only discuss the two experiments that consist of oil and gas release.

Table 1 summarizes some of the input conditions used in simulations. Most of these are based on experimental conditions. Detailed descriptions follow. In Table 1, items 1 to 3 are for identification purposes. Items 9, 10, and 14 are not independent parameters but were computed based on other parameters from the experiments, but presented here because they enhance the understanding of the experimental conditions. The value in item 15 was decided based on qualitative experimental conditions and an analysis involving different bubble diameters. In the range of 2 to 7 mm diameter the results were not impacted by the choice of gas bubble size. It was the implicit understandings of the experimenters that the bubble size was in this range. The values in items 16 and 17 were selected based on the previous experience of the authors and others. Although these can be viewed as tuning coefficients, they were kept constant for all simulations. These values do not affect the results of the plume phase of the model; they affect only the far-field conditions. The temperature of the released mixtures is equal to the ambient temperature.

4.3 Simulations of Deepspill

The two field experiments were labeled A and B in Table 1. Figures 1 and 2 present the comparison between field observations and the numerical simulations for the two experiments. The gray bars represent the echo sounder data. The solid line shows the profile of the jet/plume simulated by CDOG. Data available are time averaged. The simulated profiles shown are also time averaged for the first hour.

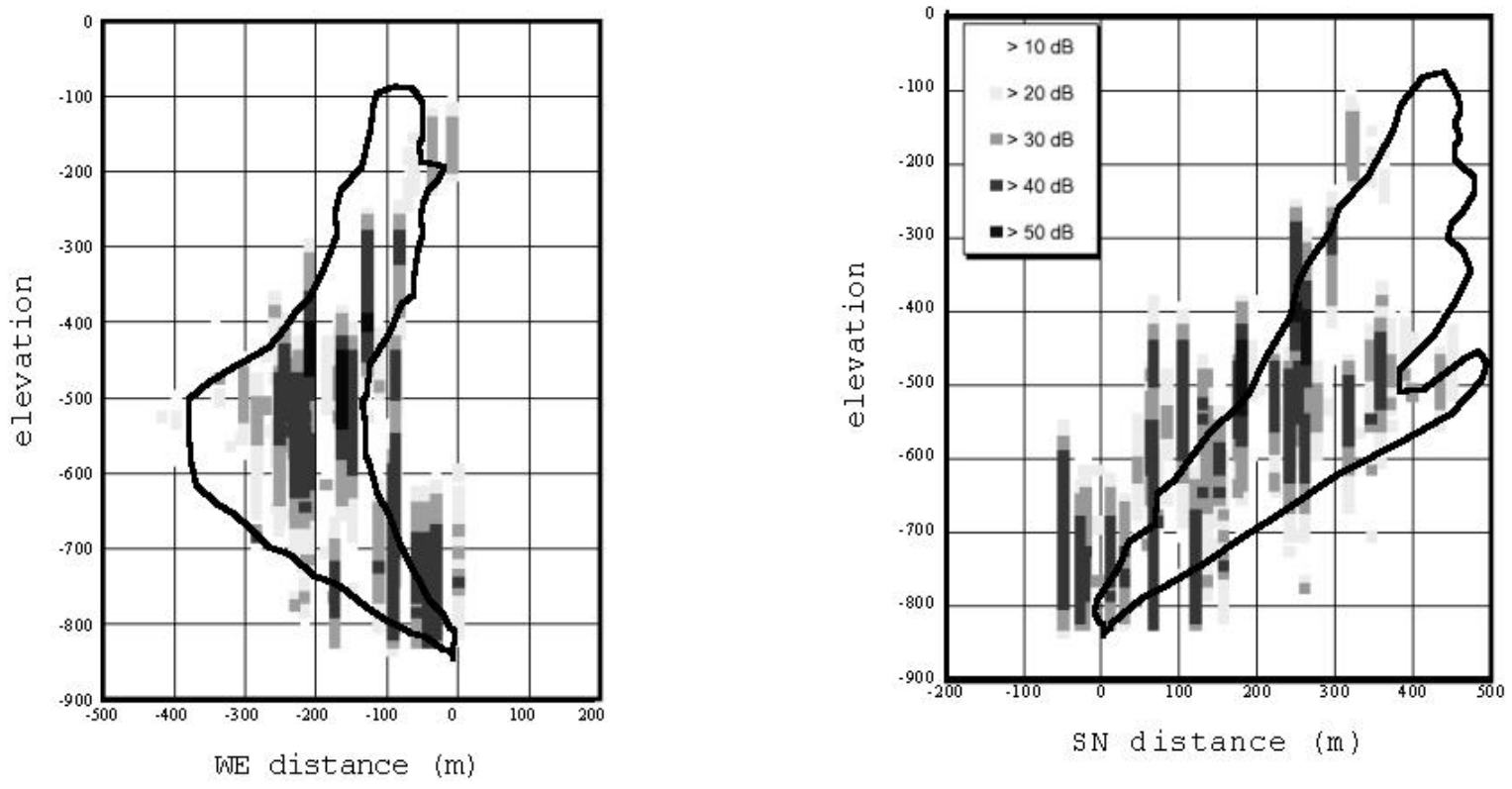
What is shown here is a brief summary of the comparison between the CDOG simulations and the "Deepspill" experimental data. For complete details of simulations the reader is referred to Chen and Yapa (2003). The complete comparisons included not only the

side view but also the bird's eye view for several water layers (average layer thickness is 300 m). Furthermore, Chen and Yapa (2003) also discuss the difference between simulations with hydrate formation suppressed and without suppression. For the two cases here with oil and gas mixture the hydrate suppression did not cause significant changes to the result. During the "Deepspill" experiments the hydrates were not observed.

Table 1 "Deepspill" Experiments and Simulation Conditions

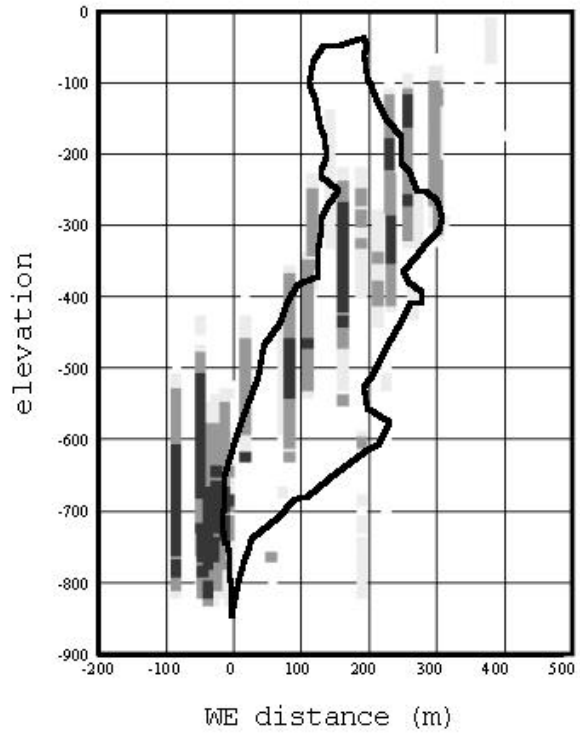
Item No.	Parameters or description	Field Experiments	
		Diesel/Gas Expt.	Crude/Gas Expt.
1	Simulation label	A	B
2	Date	June 27	June 29
3	Starting time (UTC)	6:35 am	5:14 am
4	Release period (min)	50	60
5	Oil or Water discharge rate (m^3/min)	1.0	1.0
6	Gas discharge rate (Sm^3/s)	0.6	0.7
7	Diameter of the orifice (m)	0.12	0.12
8	Density of discharged oil (kg/m^3)	854.8	842.5
9	Standard GOR (SI unit / Industry)	36 / 202	42 / 236
10	GOR at nozzle (SI unit / Industry)	0.35 / 1.97	0.40 / 2.25
11	Temperature of released mixtures (°C)	-0.77	-0.77
12	Oil droplet size distribution	Field data	Field data
13	Velocity profiles	Field data	Field data
14	Initial composite release velocity (m/s)	1.99	2.08
15	Initial gas bubble diameter (mm)	5	5
16	Horizontal diffusion coefficient (m^2/s)	0.01	0.01
17	Vertical diffusion coefficient (m^2/s)	0.001	0.001

Notes:
Standard GOR SI unit = $Nm^3s^{-1}/m^3/s^{-1}$. Standard GOR industry unit = Scfd / bopd
GOR at nozzle SI unit = $m^3s^{-1}/m^3/s^{-1}$ .GOR at nozzle industry unit = Scfd / bopd
Scfd = under standard condition, cubic feet per day
bopd = barrels of oil per day

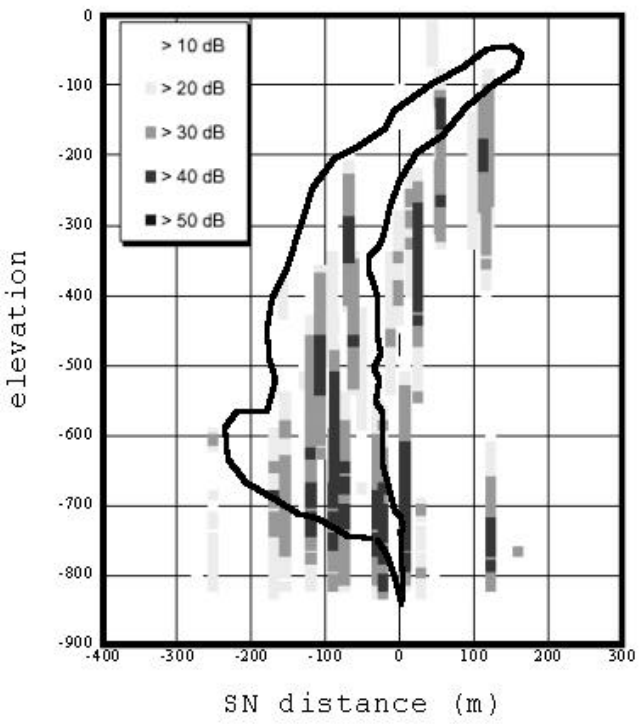


(a) (b)

Fig. 1 Side view of the Echo Sounder data and the CDOG simulated profile (thick line)– Diesel/Gas Experiment (Case A)



(a)



(b)

Fig. 2 Side views of the Echo Sounder data and the CDOG simulated profile (thick line) – Crude/Gas Experiment (Case B)

4.4 Brief discussion of CDOG model simulation comparisons

Figures 1 and 2 show the comparisons of simulation results with experimental data for cases A and B (oil and gas releases). Views (a) and (b) in each figure show the profiles in the West-East plane and South-North plane respectively. Overall comparison is not affected significantly by the presence or absence of hydrates. The plume phase however is affected slightly by the hydrates. The plume height for cases A and B are 31m and 135m respectively. The plume diameters at the end of the plume phase are 19m and 47m respectively. The observed data did not make a distinction as to a point where the plume dynamics would cease. These cases represent a small blowout in a real scenario. The gas to oil ratio (GOR) was relatively low. Because of the relatively larger proportions of oil present, the impact of the hydrate formation was not felt by the overall plume. Considering the complexities involved in such large scale field experiments, the comparison between the observed data and the simulations are reasonably good except near the point of release.

There is a large 'offset' value of the plume width close to the bottom. It is unlikely that the plume will increase to this size within just a few meters above the bottom. This discrepancy is caused by possible experimental difficulties under field conditions. Further discussion, of these difficulties is beyond the scope of the present paper.

5 GNOME Model

The NOAA GNOME (General NOAA Oil Modeling Environment) model is the spill trajectory model used by NOAA/HAZMAT for spill response. The model is designed to be a very flexible tool. GNOME is a standard Eulerian-Lagrangian model where the oil or other tracer is simulated with Lagrangian particles that move within the continuous Eulerian environmental fields. The user can choose to input a weather forecast or a sophisticated numerical wind model to drive the oil on the water's surface, and a variety of current models can be used from current patterns to baroclinic 3-dimensional time dependent circulation models.

Special attention has been paid to developing the GNOME interface to be easy to use for both responders and more general users. GNOME has four user modes, depending on the users skill level with trajectory modeling. More novice users can use Standard or GIS model to run spill scenarios with the assistance of a Location File to set up the hydrodynamics (http://response.restoration.noaa.gov/software/gnome/gnome.html). More sophisticated users would use Diagnostic Model to set up their own circulation spills and run spill simulations. Diagnostic mode is what NOAA/HAZMAT responders use for spill response. Another Batch processing mode is available to run thousands of trajectories for statistical planning.

The GNOME model can simulate a number of different spill types including drawing the oil distribution from an overflight map to initialize the model. Coupling the model to CDOG will allow GNOME users to leverage the CDOG model to simulate deep-well spills. The CDOG model will only be available to Diagnostic mode users. GNOME allows the user to set up the CDOG model from the GNOME graphical user interface (GUI), run CDOG and then GNOME simulates the spill forward from the time the first Lagrangian particle reaches the surface. At this time only the surface oil is visualized in GNOME, though the fields for creating graphics of the subsurface oil are available for later development.

The GNOME interface is designed to streamline setting up the CDOG model. For example, by selecting methane or natural gas in the deep plume, GNOME sets up more than one initialization field in the CDOG model. In the case of natural gas, the next questions give

the user default values with ranges for the user to customize the natural gas being simulated. Wherever possible, error checks are included in the interface to assist users in setting up the models correctly under the time pressure of response. Preset climatological temperature and salinity (T,S) profiles are included for the user to select by region if more detailed data is unavailable. The user may enter their own T,S profiles, either from measurements or climatologic data (e.g. the 1994 World Ocean Atlas annual, seasonal or monthly climatologic data available for download from the National Virtual Ocean Data System through the Live Access Server (http://www.ferret.noaa.gov/nopp).

6 Examples of use after the integration of CDOG and GNOME

The GNOME interface to CDOG is under development and the first version is ready. Over time the interface and methodology is likely to evolve as it is used and improvements are made to make it more "usable". The process now involves the seven steps listed below. GNOME defines CDOG as a new type of "spill" within the program architecture, so the process of setting up a CDOG spill simply adds a few new steps to setting up a surface spill in GNOME.

1. Setting the overall model run parameters relating to start and end dates of the simulation.
2. Loading or creating a "map" which may or may not contain shoreline. The CDOG model is not concerned with oil beaching as the oil rises. However, GNOME requires that the user input a map for trajectory purposes, since the oil on the surface could travel long distances and, eventually, contact land. We have improved GNOME for use with CDOG so that the map may have no defined shoreline (i.e. only open water) for short term open ocean simulations.
3. Set up the CDOG parameters. This section if given more detail below.
4. Export the CDOG settings to CDOG once the user is ready.
5. Run CDOG.
6. Import CDOG results.
7. Run GNOME

We would expect at some later time that steps 4-7 will be combined into a single step.

NOAA/HAZMAT has a philosophy of making the interfaces to response software as user friendly as possible, so that hurried responders during the middle of the night can focus on the task at hand. This requires setting up dialogs to reflect how people think about a problem, rather than just what the computer needs. For example, in GNOME Location Files that involve river scenarios, the user in asked to river gauge height or flow rate rather than values related to the parameters within the GNOME model. Sophisticated calculations are done in the background to translate what a human understands and can measure into setting up the hydrodynamic model. With CDOG we are trying to anticipate what information a responder might have during a deep well spill, since we have not had to respond to such a spill as yet.

Setting up a spill in CDOG begins with setting the location (*x*,*y*,*x)* and length of time for the oil/gas release (Figure 3). From this dialog, the user may branch out to set further characteristics of the release, and circulation related parameters.

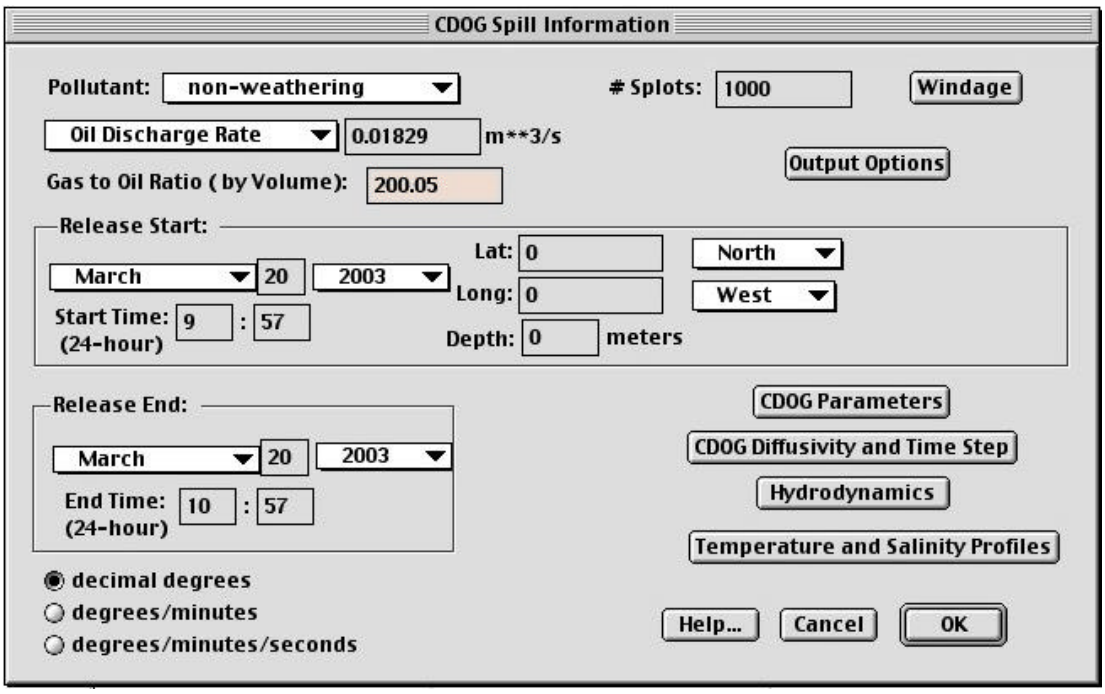


Figure 3. Spill information in for GNOME to transfer to CDOG.

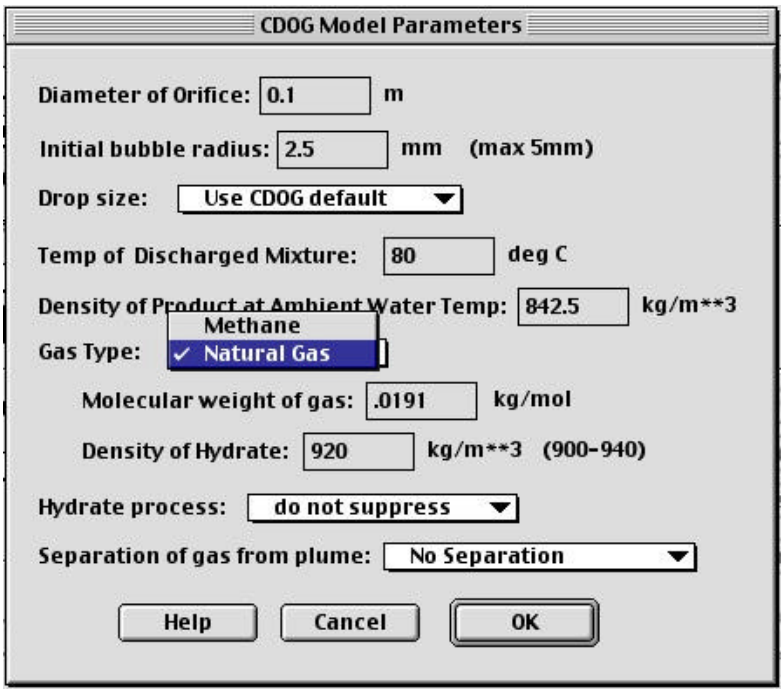


Figure 4. More detail of CDOG inputs. For example, Gas Type gives the user a pulldown to choose between methane and Natural Gas.

CDOG requires detailed information on the release and it's chemical properties (Figure 4). Wherever possible, NOAA/HAZMAT is exchanging flags in the program data files for pull downs in the user interface to make it easier to use.

7 Summary

The paper provides a brief description of CDOG, and simulation to compare the results with "Deepspill" field experiments. The CDOG model can take into account the complete three-dimensional variation in ambient velocity, temperature, and salinity. Ambient velocity variations are used for entrainment calculations, jet/plume hydrodynamical equations, and computing the far-field movement of oil and gas. Ambient temperature and salinity variations are taken into account in computing the ambient density stratification. Furthermore, the ambient temperature also affects the jet/plume thermodynamics and gas phase changes and dissolution. The model considers possible gas separation when a jet/plume is in the presence of a relatively strong ambient current.

This paper reports the integration of CDOG (Clarkson Deepwater Oil and Gas) model with GNOME General NOAA Oil Modeling Environment). GNOME provides the graphical user interface for data input to CDOG. The integration allows the user the access to many diagnostic tools available with GNOME. The purpose of the integration is to provide the users the ability to make complete spill scenarios from deepwater blowouts. This includes the subsurface spill as well as the oil slick transport and fate once the oil reaches the water surface. These complete scenario simulations are very useful for emergency response planning.

Acknowledgements

The development of CDOG was supported by the Minerals Management Service (MMS) of the U.S. Department of the Interior. The authors acknowledge the support of Robert LaBelle and Walter Johnson of the MMS, Dan Allen, Cortis Cooper, and Timothy Finnigan of Chevron, USA and the members of the Deep Spill Task Force (DSTF) for this work. The integration of CDOG with GNOME was made possible through the support of Corporate Institute for Coastal and Estuarine Environment Technology (CICEET) at the University of New Hampshire. The authors acknowledge the support of Debbie Payton, Glen Watabayashi and Caitlin O'Connor during this phase during this phase. The paper represents the views of the authors and not the MMS or DSTF.

References

Beegle-Krause, CJ (2001). "General NOAA Oil Modeling Environment (GNOME): A New Spill Trajectory Model," *Proceedings of the International Oil Spill Conference,* Tampa, Florida, March 26-29, 2001. Mira Digital Publishing Inc. pp 856-871.

Clift, R., Grace, J.R., and Weber, M.E. (1978). *Bubbles, Drops, and Particles*. Academic Press, New York.

Chen, F.H. and Yapa, P.D. (2001). "Modeling gas separation from a bent oil and gas jet/plume," *Proceedings, International Marine Environmental Modelling Seminar (IMEMS) conference*, New Orleans, LA, October.

Chen, F.H. and Yapa, P.D. (2003). "A Model for Simulating Deepwater Oil and Gas Blowouts - Part II : Comparison of Numerical Simulations with "Deepspill" Field

Experiments", *Journal of Hydraulic Research*, IAHR, (scheduled for publication in June)

Englezos, P., Kalogerakis, N., Dholabhai, P. D., Bishnoi, P. R. (1987a). "Kinetics of formation of methane and ethane gas hydrates," *Chemical Engineering Science*, Vol. 42, No. 11, 2647-2658.

Englezos, P., Kalogerakis, N., Dholabhai, P. D., Bishnoi, P. R. (1987b). "Kinetics of gas hydrates formation from mixtures of methane and ethane," *Chemical Engineering Science*, Vol. 42, No. 11, 2659-2666.

Johansen, O. (2000). "DeepBlow – a Lagrangian plume model for deep water blowouts," *Spill Science & Technology Bulletin*, Vol. 6, 103-111

Johansen, O., Rye, H., Melbye, A., Jensen, H., Serigstad, B., Knutsen, T. (2001) "Deep spill JIP experimental discharges of gas and oil at Helland Hansen, *Parts I, II, and III - Technical report*, SINTEF Applied Chemistry, Norway.

Kim, H. C., Bishnoi, P. R., Heidemann, R. A., and Rizvi, S. S. H. (1987). "Kinetics of methane hydrate decomposition," *Chemical Engineering Science*, Vol. 42, No. 7, 1645-1653.

Lee, J.H.W. and Cheung, V., (1990), "Generalized Lagragian model for buoyant jets in current." *Journal of Environmental Engineering, ASCE*., 116(6), 1085-1106.

Yapa, P.D., and Zheng, L. (1997). "Simulation of Oil Spills from Underwater Accidents I: Model Development," *Journal of Hydraulic Research*, IAHR, Vol. 35, No. 5, 673-687.

Yapa P. D., Zheng, L., and Nakata, K. (1999). "Modeling underwater oil/gas jets and plumes," *Journal of Hydraulic Engineering*, ASCE, Vol. 125, No. 5, 481-491.

Yapa, P. D., Zheng, L, and Chen, F.H. (2001). "A Model for Deepwater Oil/Gas Blowouts," *Marine Pollution Bulletin, (The International Journal for Marine Environmental Scientists/Engineers)*, Elsevier Science Publications, UK, Vol. 43, Nov., 234-241

Zheng, L., and Yapa, P.D., (1998). "Simulation of Oil Spills from Underwater Accidents II: Model Verification," *Journal of Hydraulic Research, IAHR*, February, 117-134.

Zheng, L. and Yapa, P. D. (2000). "Buoyant velocity of spherical and non-spherical bubbles/droplets," *Journal of Hydraulic Engineering*, ASCE, November, 852-854.

Zheng, L. and Yapa, P.D. (2002). "Modeling Gas Dissolution in Deepwater Oil/Gas Spills," *Journal of Marine Systems*, Elsevier, the Netherlands, March, 31/4, 299-309.

Zheng, L., Yapa, P. D., and Chen, F.H. (2003). "A Model for Simulating Deepwater Oil and Gas Blowouts - Part I: Theory and Model Formulation" *Journal of Hydraulic Research, IAHR, (scheduled for publication in June)*