

# 1 Sub Sea Pipeline Gas Release Model

## 1.1 General Model Characteristics

The MMS Pipeline Gas Release Computer Model (WCDgas.exe) delivered with this report provides a methodology to predict the behavior of gas discharges from seafloor pipelines. The model can be used for worst-case guillotine break scenarios as well as smaller diameter punctures. The primary focus in this study has been on worst-case release scenarios as this was the main interest of MMS in this project. Inputs to WCDgas are parameters describing the configuration and characteristics of a pipeline system, the fluid it contains, and the leak or break from which the discharge occurs. Key outputs are the evolution of the release rate over time, the total mass of gas released, and an estimate of its surfacing rate and area of the boiling zone. The system is composed of a Release Module (that determines the gas release rate at the seafloor from the pipeline rupture) and a Near Field Module (that models the movement of the gas from the seafloor to the water surface), linked together with necessary databases through a Graphical User Interface (GUI).

Limitations of application are:

- Single “tree” pipeline networks with all branches (pipelines) converging toward a single outlet point at its root;
- No closed re-circulating loops;
- One and only one leakage point;
- Maximum of 100 pipeline segments per branch (i.e. between junctions) and 5 junction points;
- Maximum of 5 pipeline segments attached to a single junction;
- Only pipeline objects may connect directly to junctions or connectors;
- Pipeline object connected to non-pipeline objects at both ends;
- Maximum of 50 branches (series of Pipeline objects between junctions);
- Connection object connects exactly two pipeline objects;
- Junction object connects at least 3, and not more than 10 pipeline objects;
- An Inlet must be at the start of an incoming branch;
- Outlet object must connect to only one incoming pipeline object;
- Leakage point must be attached to a Pipeline object
- Diameter of leak cannot exceed pipe diameter. (This is checked and corrected automatically in the Release Module.)

Necessary inputs for simulation of a given scenario are:

- Gas composition:
  - Fraction of each component in the gas (mol %)
  - Mol weight for each component (can use defaults provided)
  - Liquid densities (g/cm<sup>3</sup>)
- Flow inlet properties:
  - Depth (positive down; negative above mean sea level)

- Total gas flow rate
- Fluid temperature
- Closing (or shut-in) time
- Pipeline or riser segment
  - Length
  - Inside diameter
  - Roughness coefficient (can default to  $5.0 \times 10^{-5}$ )
  - Heat transfer coefficient (can default to 1 J/s)
  - Ambient temperature
- Pipe connector or junction
  - Depth
- Outlet (to remainder of pipeline system or storage)
  - Depth
  - Fluid pressure
  - Closing (or shut-in) time
- Leak properties
  - Distance from upstream endpoint
  - Nominal diameter (not larger than pipeline diameter)
  - Water depth.

## **1.2 Model Installation Instructions**

Run the file WCDgas\_2-0\_Setup.exe to install the program. Follow the instructions provided in the installation package.

## **1.3 Basic Model Use Instructions**

Double-click the WCDgas.exe file or the WCDgas icon on the desktop and the main program window appears with an empty work desk, [Figure 1.1](#). The first row of menu items are referred to as the “Main Menu” items in the remainder of this report. All functions of the model can be accessed from the main menu. The second row of menu items are basic file handling options. The third row houses a number of icons that provide quick access to pipeline objects, pipeline integrity checking and scenario calculation initiation.

Click objects on the toolbar (either using the third row icons or the drop-down submenu items in the Object menu) and click again on the work-desk to construct a diagram of the pipeline system of interest. Alternatively open a pre-defined scenario from the ‘File’ menu option. When the diagram is printed or saved, the contents of the work-desk will be saved. All of the pipeline network information and leak characteristic for a scenario are saved in a file with an extension designation of “wcd”. The saved data is retrieved from this file when the scenario is re-opened using WCDgas.

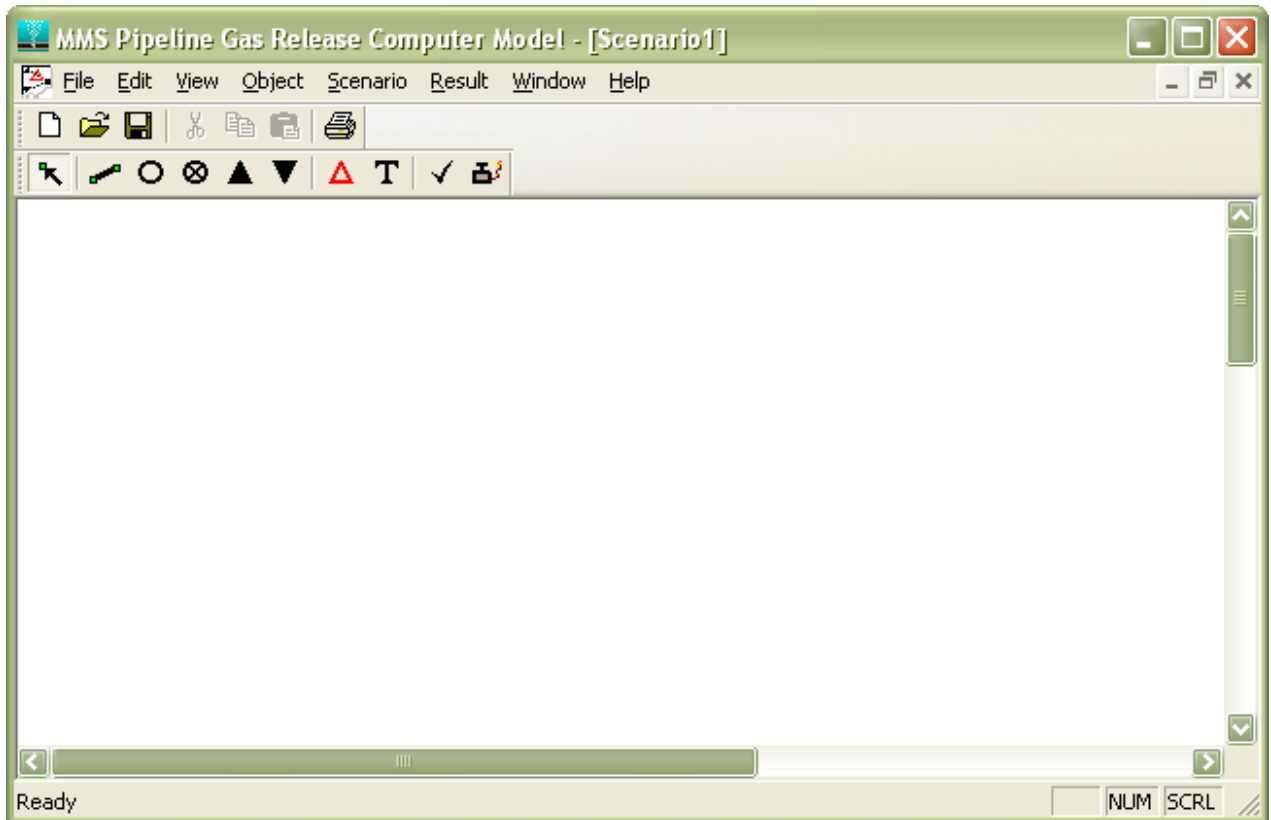


Figure 1.1 Main window for WCDgas.

### 1.3.1 Create a New Scenario

A scenario consists of a constellation of connected objects, each assigned a set of parameters. Under the Object menu or on the object toolbar there are six options:

- Pipeline
- Connection
- Junction
- Inlet
- Outlet
- Leakage

The parameters defining each object are given in [Table 1.1](#).

Choose the component you want to place on your work desk, and click it into the work area. An example pipeline network is shown in [Figure 1.2](#).

When a pipeline segment or leak point is inserted into the work area, it will appear with small green boxes defining the connection points. Pipeline segments can be resized by dragging one of these green boxes. Objects can be moved on the work-desk by click-and-drag.

Table 1.1 Parameters defining objects in a discharge scenario

Object	Parameters				
Pipeline Segment	Length	Diameter	Roughness	Heat Transfer Coefficient	Ambient Temperature
Connector	Depth				
Junction	Depth				
Inlet (Flow Source)	Depth	Flow Rate	Fluid Temperature	Closing (or Shut-in) Time	
Outlet (Flow Sink)	Depth	Pressure			
Leak Point	Distance from upstream endpoint	Nominal Diameter	Depth at leak location	Back Pressure	

To delete an object, click on the object and press the Delete key (or use the Delete command under the Edit menu).

All objects must be connected together before a scenario will run. Pipelines and Leak Points can connect to Connectors, Junctions, Inlets, and Outlets at any of their blue connection markers. The green box in the center of the Leak Point, or at the end of the Pipeline segment, will turn red when the connection has been properly made.

To verify that a connection has been properly made, click and drag the Connector, Junction, Inlet, or Outlet (not the Pipeline Segment or Leak Point), and see that the attached object follows after. (Clicking on the pipeline element will detach it from its connectors and junctions.)

**Notes:**

1. When opening an existing scenario, some pipelines may appear to be disconnected from their junctions and connectors. This is a visual effect resulting from the use of long text strings in names of elements, and does not affect the integrity of the scenario. These text strings mask the placement of objects. Simply click on the junctions and connectors, and the pipelines will return to their correct positions.
2. The layout on the desktop is generally not to scale. Only the parameters such as length and depth) allocated to each element in the diagram are used in the actual calculations. Moving an object manually on the desktop will not alter the basis for the computations in the WCDgas.

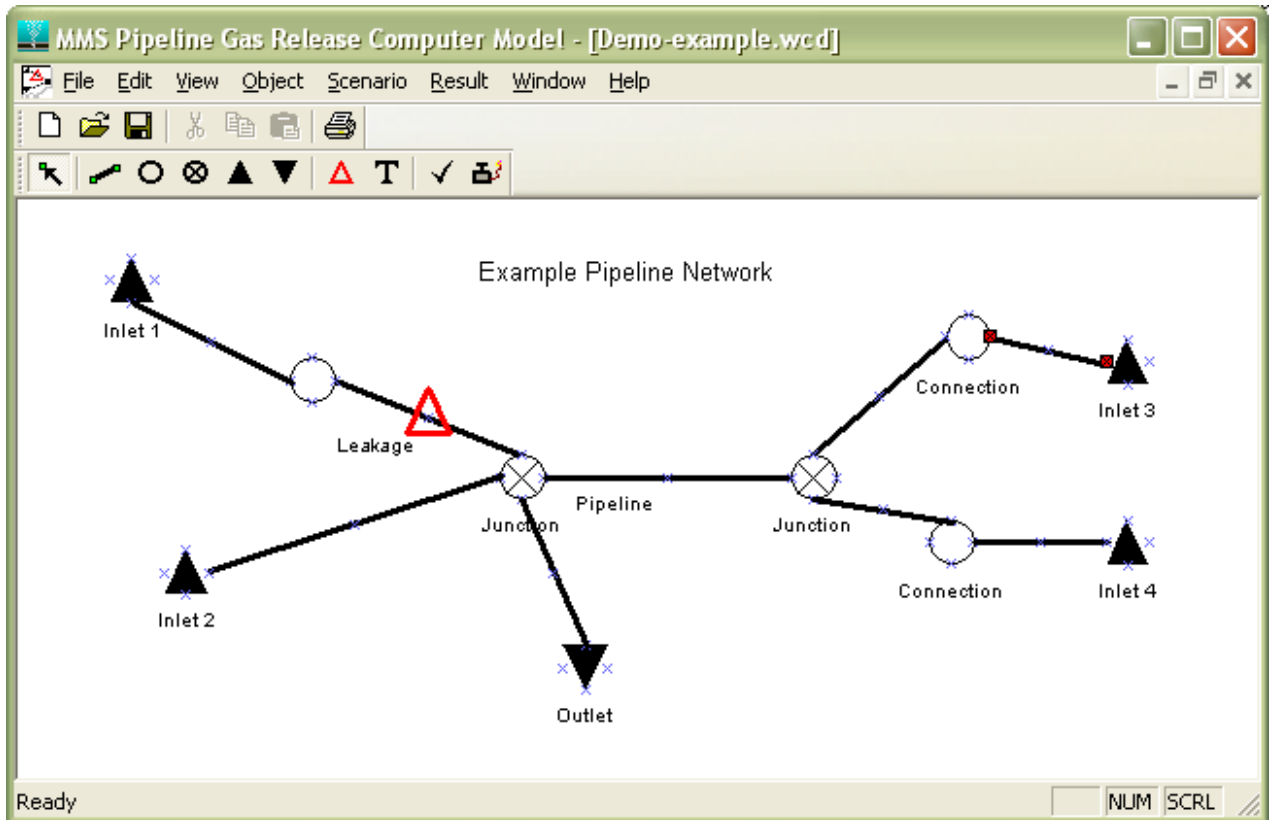


Figure 1.2 Example Pipeline Network

### 1.3.2 Object Properties

After placing a selected object one has to supply required parameters. Right-click the object and choose Properties, or double-click and the Properties box appears. Fill in specifications for each object. [Figure 1.3](#) shows the Object Properties dialog boxes. The following sections provide additional details regarding the data entered in these dialogs.

#### 1.3.2.1 Flow Inlet Specifications

For every flow inlet in the network the depth, total gas flow rate, fluid temperature and the closing time have to be specified. The specified flow rate is fixed until the inlet choke closes. The closing time is the duration from the time the leak occurred to the time production is shut down.

#### 1.3.2.2 Flow Outlet Specification

In contrast to the inlet specifications where several inlets are possible, only one pipeline outlet can be specified. At the pipeline (or network) outlet, the receiving pressure is required. The outlet pressure is the fixed pressure at the outlet of the pipeline, typically upstream of a choke at the receiving facility. This receiving pressure is usually known.

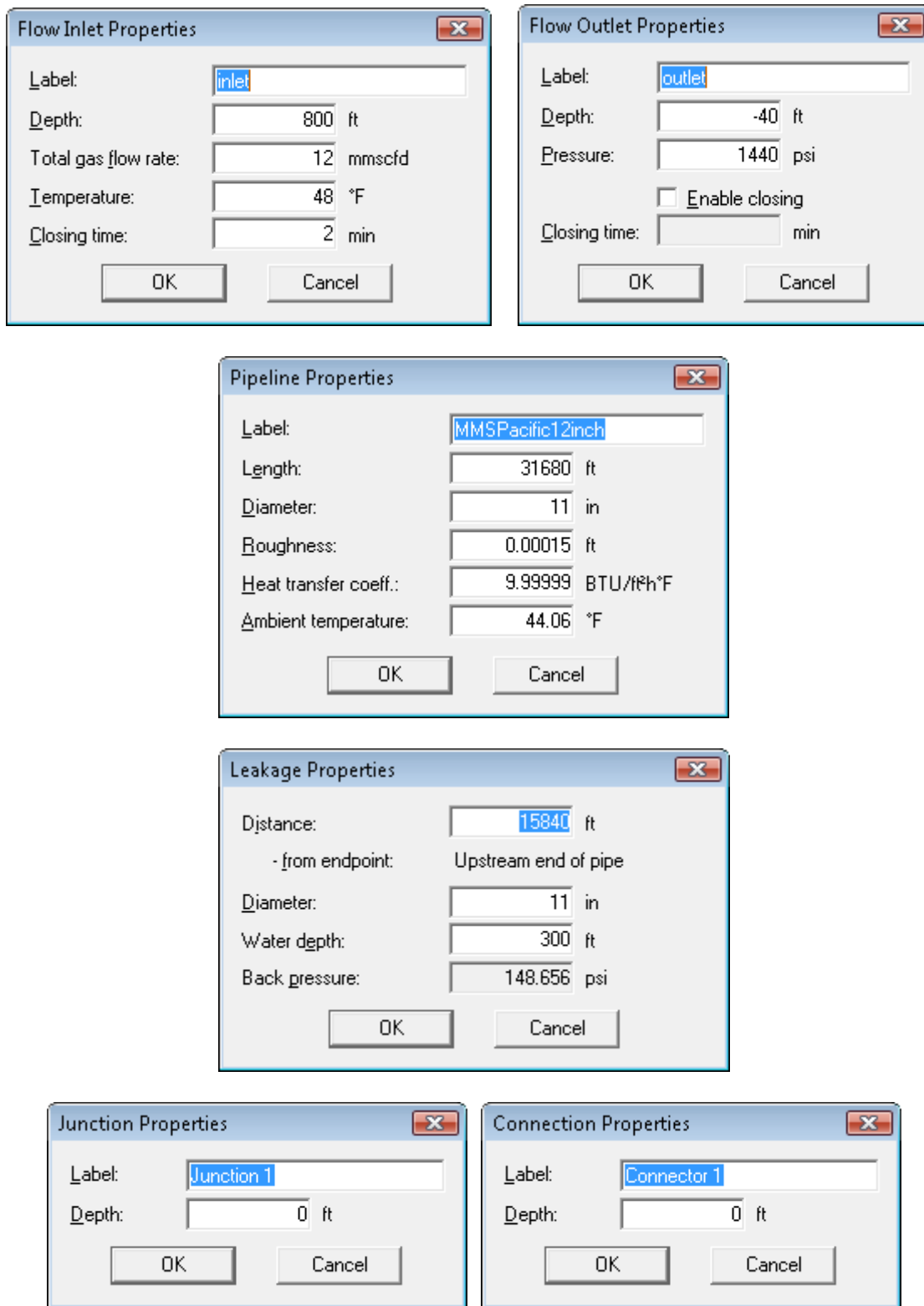


Figure 1.3 Object properties input dialogs. Labeling objects is not required, but is recommended as an aid in locating problems with the scenario setup.

Note: The outlet pressure can be the same as operating pressure, but that depends on the definition of "operating pressure". Usually, the "operating pressure" is used in connection with the "maximum operating pressure" and is the design limit of the pipeline or equipment, i.e. the pressure should not exceed the maximum anywhere in the system. Operating pressure can be this pressure, it can be the pipeline input pressure, or it can be an average pressure in the pipeline. Based on the user specified flow rate and the outlet pressure, the model calculates the pressure drop and hence, the pipeline inlet pressure as well as the entire pressure profile in the pipeline network.

The software handles networks with several inlets, but only one outlet and one leak or rupture as seen example in [Figure 1.4](#).

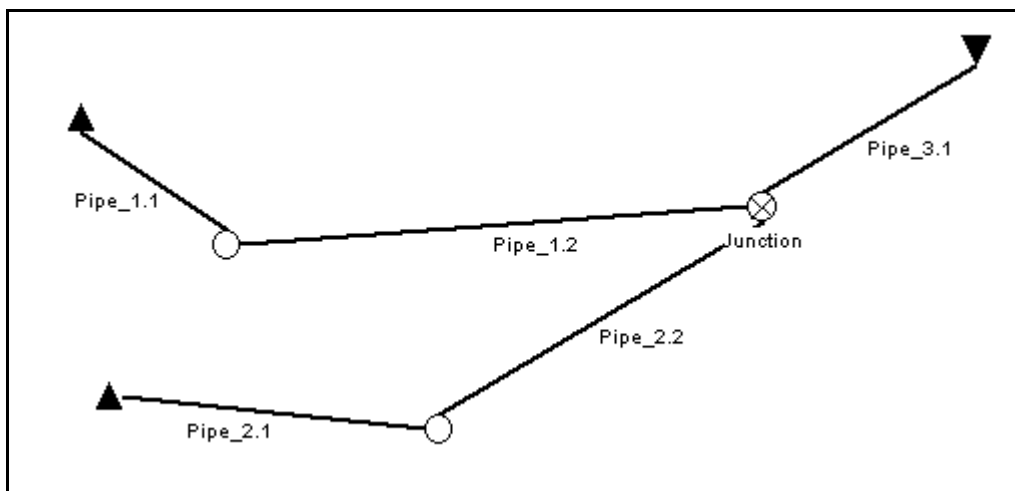


Figure 1.4 Example of a network with two inlets and one outlet

### 1.3.2.3 Pipeline Properties

Pipelines should be modeled with several pipe segments to account for the seabed topography and variation in inclination. Every pipeline segment is labelled with a description, given a length and an internal diameter. A pipeline has to be connected to an inlet, connection, junction or outlet.

The pipe roughness is used when calculating the frictional pressure drop in the pipeline. The internal pipe roughness for the gas pipelines will usually be low, typically  $10E-5$  ft, and smaller changes are not believed to have a significant effect on the pressure drop.

The overall heat transfer coefficient "U" is used to calculate the heat transfer from the fluid and radially through the pipe wall layers to the surroundings at ambient temperature.

A typical U value for an unburied, un-insulated pipeline can be 10 btu/(hr ft<sup>2</sup> degF). A typical U value for an insulated (and buried) pipeline can be 0.5 - 2 btu/(hr ft<sup>2</sup> degF).

#### **1.3.2.4 Pipeline Connection**

A pipeline is usually modeled with several pipeline segments with different angles. Between two segments, a connection is included with a depth specification.

#### **1.3.2.5 Pipeline Junction**

A pipeline junction defines a point with three or more pipelines are connected and is used when modeling networks. The required input data is depth.

#### **1.3.2.6 Leakage Properties**

The leak is modeled by a critical choke with a diameter equal to the leak size. The leak is snapped to a pipeline and the distance from the upstream end of the pipe is specified. The water depth is used to calculate the ambient back-pressure at the leakage.

For large ruptures, the simulations can become unstable because of rapid pressure transients. A workaround is to run the simulation with a smaller leak diameter.

### **1.3.3 Verify Pipeline Layout**

After creating your scenario, select Scenario menu\Verify Layout (or the Network Check button on the toolbar). This checks a number of potential problems in the network layout, such as:

- Missing or invalid object parameters,
- Outlet point connected to more than one in-coming branch,
- Pipeline segment shorter than depth difference between endpoints,
- Maximum number of objects exceeded (100 pipeline segments, 5 junction, 10 segments per junction), and
- More than one leakage point found.

In general, these messages are self explanatory, and lead the user quickly to the problem area. If the “valid network” message appears one can continue with the analysis. Otherwise the message-box identifies the problem.

Shortcut: Select Calculate Discharge from the Scenario menu (or use the Worst Case discharge button on the toolbar). This automatically runs the Verify Layout test prior to estimating the discharge for the given scenario.

### **1.3.4 Gas Composition**

The property of the gas in the pipeline is specified on a compositional basis using WCDgas’s “Scenario->Gas Composition” menu item. The available components include nitrogen, carbon dioxide, hydrogen sulfide and hydrocarbon components from C1 to C10. [Figure 1.5](#) provides an example gas composition. The user enters the Mol% value for each component present in the gas. The total mole fractions must sum to 100%. The gas composition is entered and stored for each individual scenario.





The Release module must be run before the Near Field module, since the latter uses results from the former to compute the timing, rates, and boiling zone of gas at the sea surface. To set up the Near Field module, select the menu item Scenario, Near Field Setup ([Figure 1.6](#)).

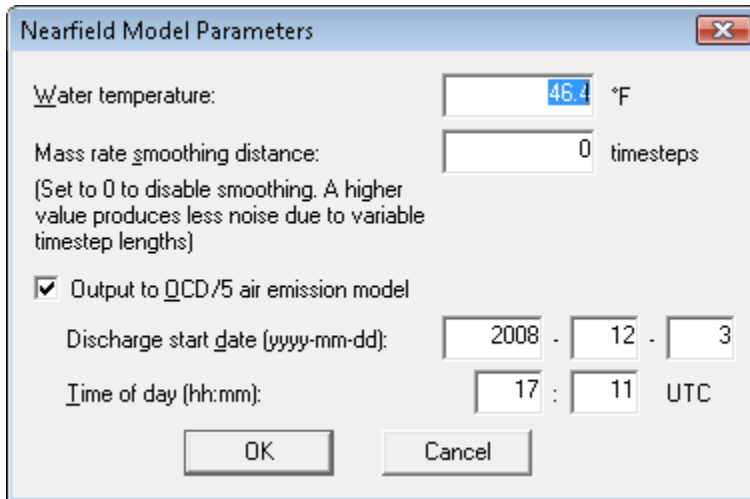


Figure 1.6 Near Field setup dialog box.

Description of entries in Near Field Setup Dialog shown in [Figure 1.6](#):

- **Water temperature:** This is the temperature of the sea water at the location of the leakage point.
- **Mass rate smoothing distance:** The output can be averaged over several samples (or successive time steps) to dampen some of the artifacts that can occur when the input data to the Nearfield Module becomes very noisy (“ripples” in the curves, for instance). Note that this “input” is equivalent to the “output” or result produced by the preceding Release Module. Using a value of 3, for example, means that the averaging at each point in time will consider 3 points before and 3 points after the current sample, a total of 7 point to average each point in the output time series.
- **Output to OCD/5 air emission model:** This is optional, but if used it will produce an extra text output file (.DAT) that can be used by the OCD/5 atmospheric dispersion model. A starting date and time (using UTC time zone) is specified to correspond to the first time that gas was observed to emerge at the surface. This is used to produce the corresponding timestamps in the .DAT file.

## 1.4 Discharge Calculation

The gas release predictions are initiated using the Scenario->Calculate options after the pipeline network has been established and the gas composition entered. The discharge of gas from the pipeline rupture and the movement of the gas from the rupture to the surface both can be modeled independently using the “Calculate Discharge” and Calculate Nearfield” options in sequence. Since both models complete their calculations quickly it

is more efficient in most cases to simply use the “Calculate All” option that automatically runs the two processes in sequence. The algorithms used to predict the behavior of the gas as it exits the pipeline puncture are described in detail in [Appendix A](#). The algorithms used to predict the behavior of the gas as it rises to the surface are described in detail in [Appendix B](#).

## 1.5 View Simulation Results

Once the scenario has been completed the results can be viewed using the main menu “Result” dialogs. A Discharge Summary similar to [Figure 1.7](#) automatically appears at completion of the discharge calculation (if this option is selected in the main menu Options dialog), and is also accessible via the Result menu. This summarizes the gas flow characteristics at the rupture location.

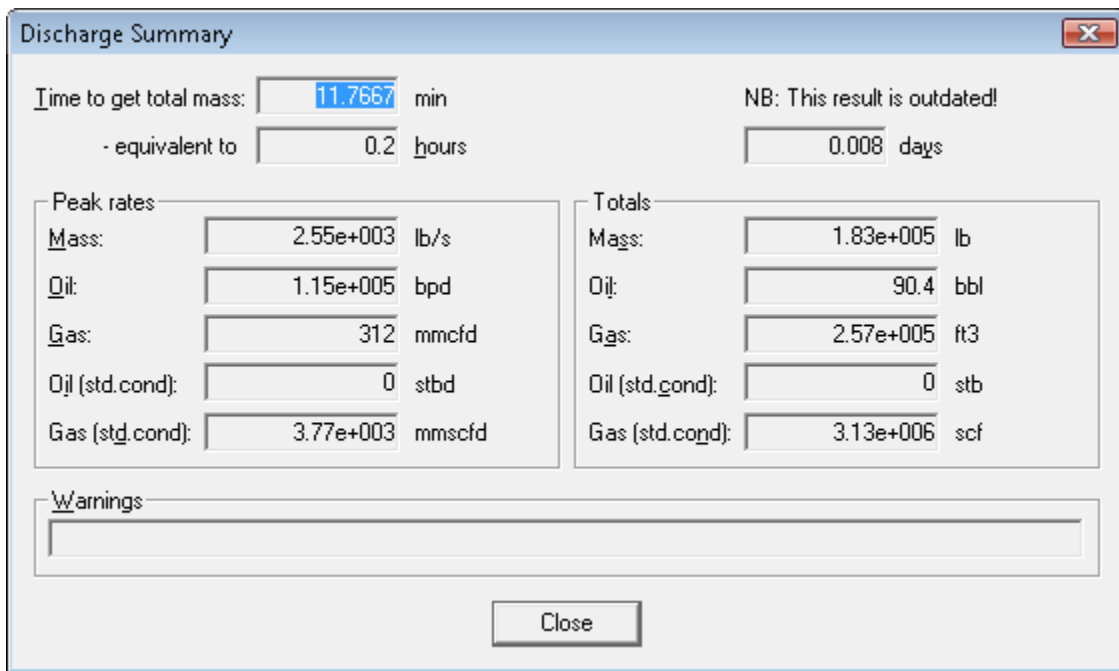


Figure 1.7 Sample Discharge Summary Report

Time series plots of the gas discharge characteristics at the rupture location and at the water surface can be generated using the “Results->Release Plots” and “Results->Nearfield Plots” options. In each of these options several variables can be selected for plotting using the selection dialog at the top of the plot. [Figure 1.8](#) is an example plot of gas release at the leak or rupture location.

The variables that can be plotted at the discharge point include:

- Accumulated Mass
- Total Mass Flow Rate
- Gas Mass Flow Rate
- Oil Mass Flow Rate
- Gas Flow Rate at Standard and Outlet Conditions
- Oil Flow Rate at Standard and Outlet Conditions
- Pressure and Temperature at the Rupture
- Pressure at the Pipeline Inlet and Outlet
- Total Mass Flow Rate at the Pipeline Outlet

Time series results that can be plotted from the Nearfield module are:

- Accumulated mass surfaced (kg)
- Vertical velocity of gas at the surface (m/s)
- Radius of gas bubble plume at the surface (m)
- Gas rise time rupture to surface(s)
- Gas mass flow rate at surface (kg/s)

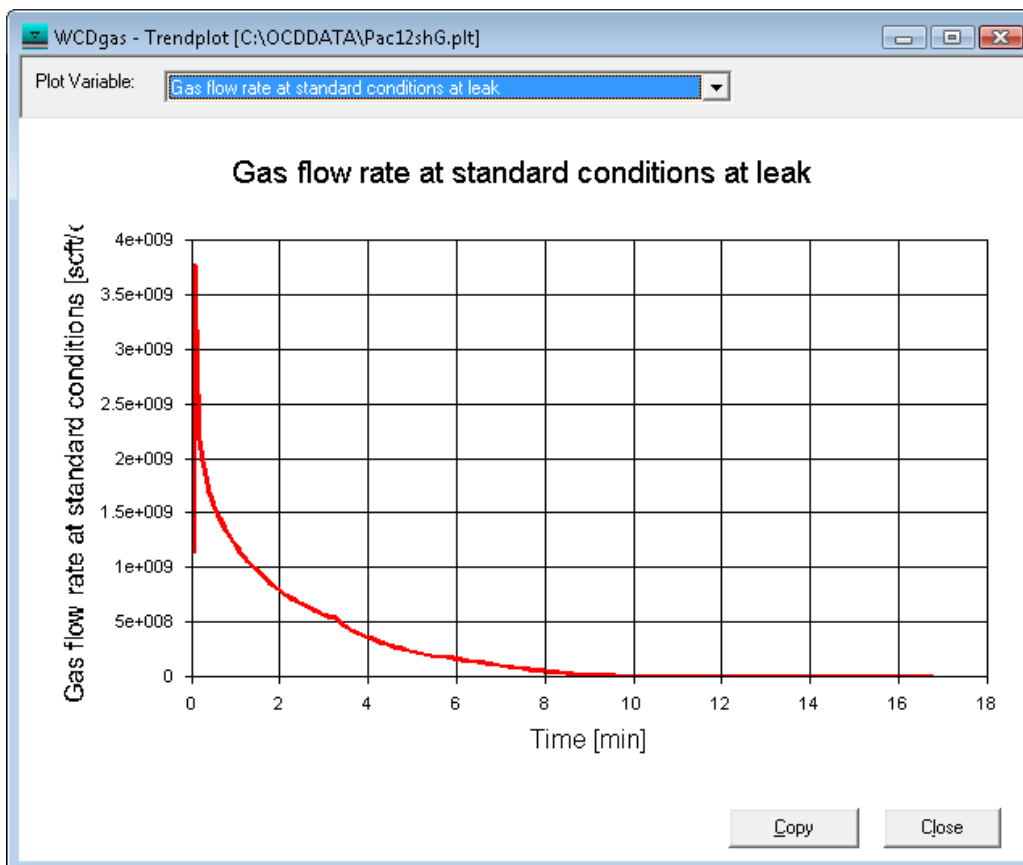


Figure 1.8 Example Plot of Gas Release at the Rupture Location

# Appendix A: Pipeline Rupture Release Algorithm Details

## 1 The Peng Robinson Equation of State

### 1.1 General

A compositional model is used to predict the hydrocarbon phase behavior and thermodynamical properties. The calculations are based on the concept of an equilibrium constant, K value, defined as the ratio of the mole fraction of the component in the gas phase,  $y_i$  to the mole fraction of the same component in the liquid phase,  $x_i$ .

$$K_i = \frac{y_i}{x_i}$$

Unlike a single component fluid, a multi component mixture exhibits a phase envelope rather than a single equilibrium curve. This implies that pressures and temperatures inside the phase envelope, both liquid and gas phases exist in equilibrium.

The software requires a compositional input describing the hydrocarbon fluid and uses the Peng-Robinson Equation of State (EOS) to calculate the required fluid properties as functions of pressure and temperature. The equation of state is a thermodynamic equation describing the state of matter under a given set of physical conditions. The compressibility factors and phase distributions are determined from the EOS and the fluid properties are calculated. These will act as input to the two- phase flow model.

The Peng-Robinson equation is expressible in terms of the critical properties and the acentric factor. The equation is applicable to calculations of fluid properties in natural gas processes and is expected to provide good accuracy for the scenarios intended for the release model. The following chapters give a brief overview of the equation. For more details see [Peng 1976](#).

## 1.2 Equations

The following equations describe the Peng-Robinson Equation of State:

$$p = \frac{RT}{V_m - b} - \frac{a\alpha}{V_m^2 + 2bV_m - b^2}$$

$$a = \frac{0.45724R^2T_c^2}{P_c}$$

$$b = \frac{0.07780RT_c}{P_c}$$

$$\alpha = \left(1 + (0.37464 + 1.54226\omega - 0.26992\omega^2)(1 - T_r^{0.5})\right)^2$$

$$T_r = \frac{T}{T_c}$$

where,

$V_m$  - molar volume,  $V/n$

$\omega$  - acentric factor

$R$  - universal gas constant,  $8.314472 \text{ J/(K mol)}$

$T_c$  - critical temperature

$P_c$  - critical pressure

An alternative form in terms of the compressibility factor  $Z$  replacing the molar volume from the real gas law is:

$$Z^3 - (1 + B - 2B) \cdot Z^2 + (A - B^2 - 2B - 2B^2) \cdot Z - A \cdot B + B^2 + B^3 = 0$$

where,

$$A = \frac{a \cdot p}{R^2 \cdot T^2}$$

$$B = \frac{b \cdot p}{R \cdot T}$$

This equation is used both for the gas phase and for the liquid phase.

$$Z_L^3 - (1 + B_L - 2B_L) \cdot Z_L^2 + (A_L - B_L^2 - 2B_L - 2B_L^2) \cdot Z_L - A_L \cdot B_L + B_L^2 + B_L^3 = 0$$

$$Z_G^3 - (1 + B_G - 2B_G) \cdot Z_G^2 + (A_G - B_G^2 - 2B_G - 2B_G^2) \cdot Z_G - A_G \cdot B_G + B_G^2 + B_G^3 = 0$$

### 1.3 Critical values and acentric factors

Table A.1 shows the critical values and factors used by the equation of state.

**Table A.1: Critical values, acentric factors and compressibility factor**

		Critical temp	Critical pressure	Accen fac	Compr fac	Critical vol	Mol weight
Symbol	Name	K	bar	-	-	cm/mol	g/mol
H	hydrogen	33.2	12.8	-0.220	0.276	129.0	2.016
CO2	carbon diox.	304.2	72.8	0.225	0.274	94.0	44.010
N2	nitrogen	126.2	33.5	0.035	0.290	89.5	28.073
C1	methane	190.6	45.2	0.013	0.288	99.0	16.043
H2S	hydr. sulfide	373.2	88.2	0.100	0.284	98.5	34.080
C2	ethane	305.4	48.2	0.098	0.285	148.0	30.070
C3	propane	369.8	41.9	0.152	0.281	203.0	44.097
n-C4	n-buthane	425.2	37.5	0.193	0.274	255.0	58.124
i-C4	i-buthane	408.1	36.0	0.176	0.283	263.0	58.124
n-C5	n-penthane	469.6	33.3	0.251	0.262	304.0	72.151
C6	hexane	507.4	29.3	0.296	0.260	370.0	86.178
C7	heptane	540.2	27.0	0.351	0.263	432.0	100.205
C8	octane	568.8	24.5	0.394	0.259	492.0	114.232
C9	nonane	594.6	22.8	0.444	0.260	548.0	128.259
C10	decane	619.2	20.8	0.490	0.247	603.0	142.286

### 1.4 Solution algorithm

The compressibility factors for liquid and gas phase, the equations are solved iteratively. The following is a step by step algorithm to calculate the equilibrium constants.

1. The input data for the calculation are the pressure, temperature and fluid composition.
2.  $K_i$  values for each component are guessed using the Wilson correlation (see below)
3. On basis of the assumed  $K_i$  values, perform the flash calculations (see below)
4. Compositions of liquid and gas phases obtained from flash calculations can be used to determine the fugacity  $y$  coefficients from each component
5. Use the fugacity coefficient ratios to calculate the equilibrium constants  $K_i$  for each component
6. Compare the guessed constants calculated in step 2 with the calculated values in step 5.
7. If the convergence tolerance is satisfied for all components, the values of equilibrium constants are used to calculate the phase compositions required in

determining phase physical properties. If not, the calculated values are used as the new guesses and steps 3 to 6 are repeated until convergence is achieved.



The Wilson correlation is used to estimate the K values initially as in step 2 above.

$$K_i = \frac{P_{ci}}{P} \exp \left[ 5.37 (1 + \omega_i) \left( 1 - \frac{T_{ci}}{T} \right) \right]$$

The flash calculations in step 3 above are performed using the following equation:

$$\sum_{i=1}^n (y_i - x_i) = \sum_{i=1}^n \frac{z_i (K_i - 1)}{(K_i - 1) \frac{G}{F} + 1} = f \left( \frac{G}{F} \right) = 0$$

Where,

- F - number of moles of composition
- L - number of moles of liquid
- G - number of moles of gas
- $z_i$  - mole fraction of component i in composition
- $x_i$  - mole fraction of component i in liquid phase
- $y_i$  - mole fraction of component i in gas phase
- n - total number of components in composition

Once the compressibility factors of each phase are determined from the iterative procedure, all the required vapor and liquid properties can be determined. These include densities, viscosities, enthalpies, conductivities, heat capacities and surface tension.

## 2 Dynamic Flow Simulation Model

### 2.1 General

This model is a transient two-phase flow model based on conservation equations. Two separate mass and momentum equations for gas and liquid and one energy equation. Estimation of gas release rates is based on flashing, integration, choking effects and fluid flow behavior in the system. Total volume released is calculated from:

- Rate variation and release time,
- Leak detection time and production rates,
- Shutdown time for each component in the system,
- Location of rupture,
- Property changes with pressure and temperature,
- Frictional and hydrostatic pressure drop,

## **2.2 Data requirements**

To provide the release results, the software needs the following information:

- Geometrical description of the flow lines
- Compositional input of the hydrocarbon fluid
- Receiving pressure at the outlet of the system
- Leak position and size

## **2.3 Geometrical discretization**

Pipeline length and diameter must be specified. It might be of importance to specify dips and peaks along the pipeline where condensate could accumulate. Generally, finer grid results in more accurate calculations. Each user specified pipeline is discretized into a number of sections in the model and calculations are done for each of the section elements in the system. The computational time increases with the number of sections, and a short single pipeline is much faster to simulate than a complex network with many internal sections.

## **2.4 Leak modeling**

The leak/rupture in a pipeline is modeled by implementation of a critical choke model with a diameter equal the equivalent diameter of the leak. The model handles both sub critical and critical flow. If the gas velocity in a choke exceeds the critical velocity, critical flow conditions are used.

## 2.5 Nomenclature for a pipeline layout

The physical elements used to define a pipeline layout in the model are as follows:

- Pipe - an element with a given diameter, length, height, roughness and u value
- Branch - one or more connecting pipes in series (Figure A.1)
- Connection - connects two pipes
- Junction - connects two or more branches.
- Network - two or more connected branches (Figure A.2)

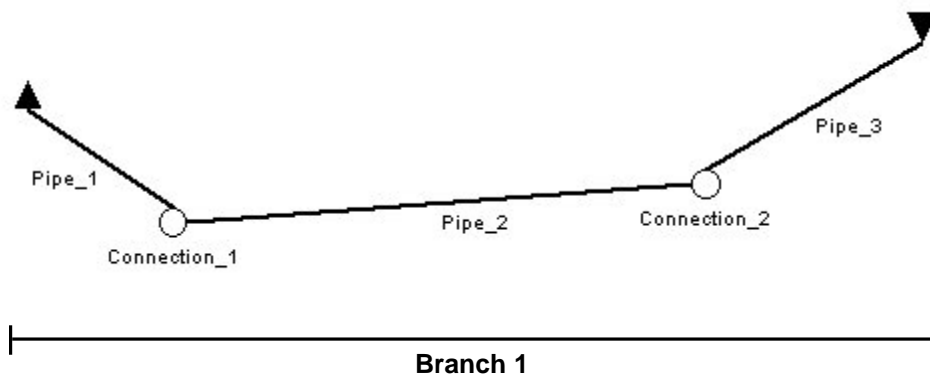


Figure A.1 Example sketch of a branch

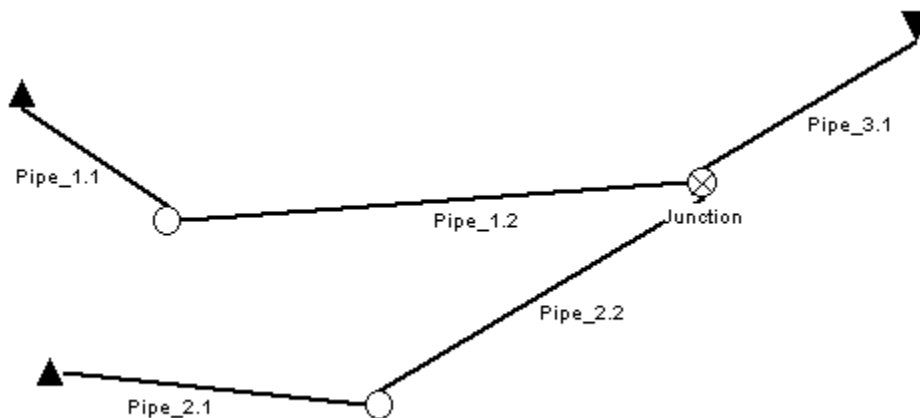


Figure A.2 Example sketch of a network

A network can only have one outlet and one leak/rupture, but may have several inlets.

### **3 References**

Peng, D. and Robinson, D.B. 1976. "A New Two Constant Equation of State", Ind. Eng. Chem. Fund.

## Appendix B: Gas Bubble Plume Algorithm Details

### Plume Modelling

#### 3.1 Sub sea gas bubble plume calculations

The gas bubble plume calculations are based on the following input data:

Discharge depth  $H_0$ , m

Gas mass flux  $q$ , kg/s

Gas density  $\rho$ , kg/Sm<sup>3</sup> (@ 1 atm and 15 °C)

Sea temperature  $\theta_s$ , °C

Here, the gas mass flux is presumed to be delivered by the sub sea gas leak module in terms of a table of leak rates and corresponding times from the start of the leak. The gas mass flux is used together with discharge depth, sea temperature, and gas density to determine the volume flow rate  $V_0$  (m<sup>3</sup>/s) at the discharge depth:

$$V_0 = q / \rho_0 \quad (1)$$

where  $\rho_0 = \rho \frac{H_0 + 10}{10} \frac{273 + 15}{273 + \theta_s}$ , assuming ideal gas

In the expression above, the number 10 corresponds to 10 m water column, which equals a hydrostatic pressure of one atmosphere. The volume flux at the discharge depth is used to define the buoyancy flux parameter  $\phi_0 = gV_0 / \pi$

The bubble plume calculations are based on Fanneløp's general non-dimensional solution for underwater gas releases, shown in graphical form at Figure B.1 (Fanneløp and Sjøen 1980, Fanneløp 1994). The critical assumption in the development of the solution is that the mass flux of gas is conserved, while the gas volume varies with hydrostatic pressure according to the ideal gas law. The expansion of the gas is assumed to be isothermal. Moreover, the initial momentum of the discharged gas is neglected, as well as possible effects of crossflow and stratification (due to vertical temperature and salinity gradients). This implies that the solution is valid for large gas leaks at moderate depths, but may be less reliable for small leak rates and large water depths due to enhanced influence of factors such as cross flow, stratification and dissolution of gas in the water masses (Johansen 2000).

The plume is defined by three variables – plume radius  $b_p$ , centerline velocity  $w_p$ , and plume rise time  $t_p$  – all functions of the vertical distance  $z$  from the discharge point. These variables may be expressed in terms of non-dimensional variables,  $X$ ,  $B$ ,  $W$  and  $T$ :

$$X = z/H, B = b_p/2\alpha H, W = w_p/M \text{ and } T = t_p M/H \quad (2)$$

where  $H = H_0 + 10$  and  $M = \left[ \phi_0 \frac{\lambda^2 + 1}{2\alpha^2 H} \right]^{1/3}$

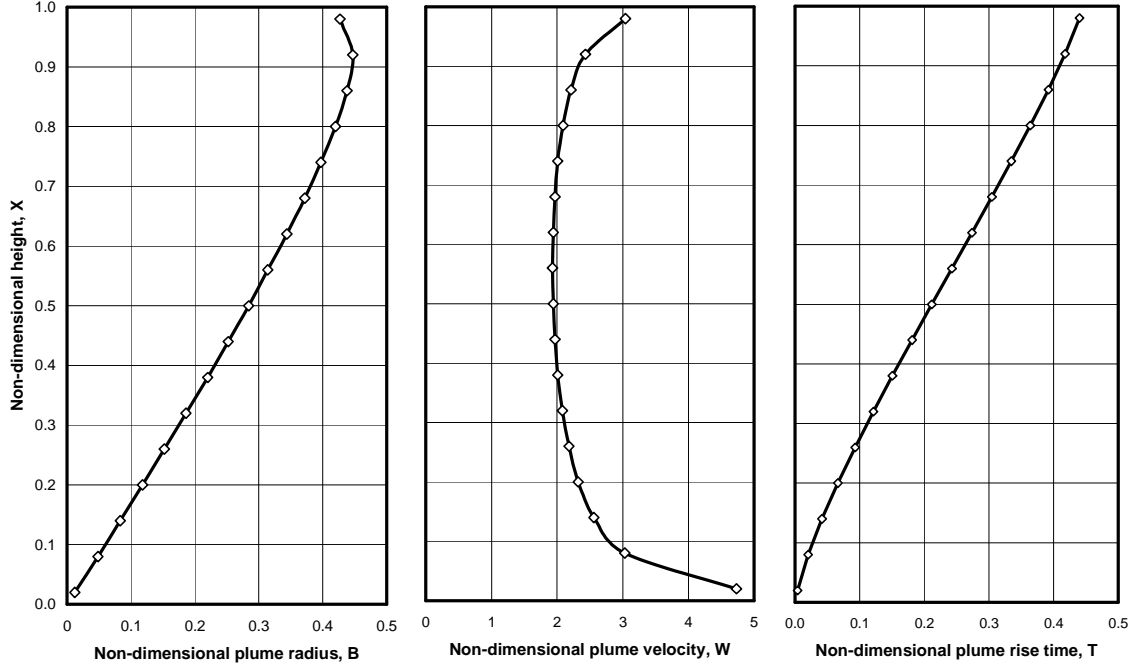


Figure B.1 Fanneløp's general gas bubble plume solution in non-dimensional form. Note that the non-dimensional rise time  $T$  is derived from the non-dimensional plume velocity  $W$ .

The non-dimensional plume rise time  $T$  is derived from the non-dimensional plume velocity by the integral

$$T = \int_0^X dX / W \quad (3)$$

The parameter  $\alpha$  is the entrainment coefficient ( $\alpha = 0.1$ ), and  $\lambda$  is a shape factor representing the ratio between the buoyancy and velocity profiles ( $\lambda = 0.65$ ), both assumed constant with depth.

## 4 Surfacing of gas

For the present purpose, the general solution presented in Chapter 1 may be curve fitted or interpolated from tabulated values and used to determine the plume variable  $b_p$ ,  $w_p$  and  $t_p$  on the basis of the input variables  $q$ ,  $H_0$ ,  $\rho$  and  $\theta_s$ . The plume variables are used to estimate the time dependent gas flow rates  $q_a$  (kg/s) to the atmosphere and the corresponding radius  $R_B$  (m) of the boiling zone.

### 4.1 Surfacing rate

The time of surfacing is determined from the time of discharge and the plume rise time corresponding to a given gas discharge rate. Thus, with gas flow rates  $q(i)$  tabulated at consecutive time steps ( $i = 1, 2, \dots$ ), the corresponding list of times of surfacing  $t_s$  will represent the sum of the release time  $t_r$  and the rise time  $t_p$ :

$$t_s(i) = t_r(i) + (1 + \beta)t_p(i) \quad (4)$$

where the factor  $\beta$  is introduced to account for the fact that the computed rise time  $t_p$  is derived from the centerline plume velocity, while the rise time of a certain fraction of the gas flow will be longer due to the presumed Gaussian velocity distribution in the plume. Calculations based on a plume shape factor  $\lambda = 0.65$  show that the Gaussian velocity profile will cause a time lag in the surfacing gas flow rate of about 1/3 of the center line plume rise time, i.e.  $\beta = 0.333$  (Figure B.2).

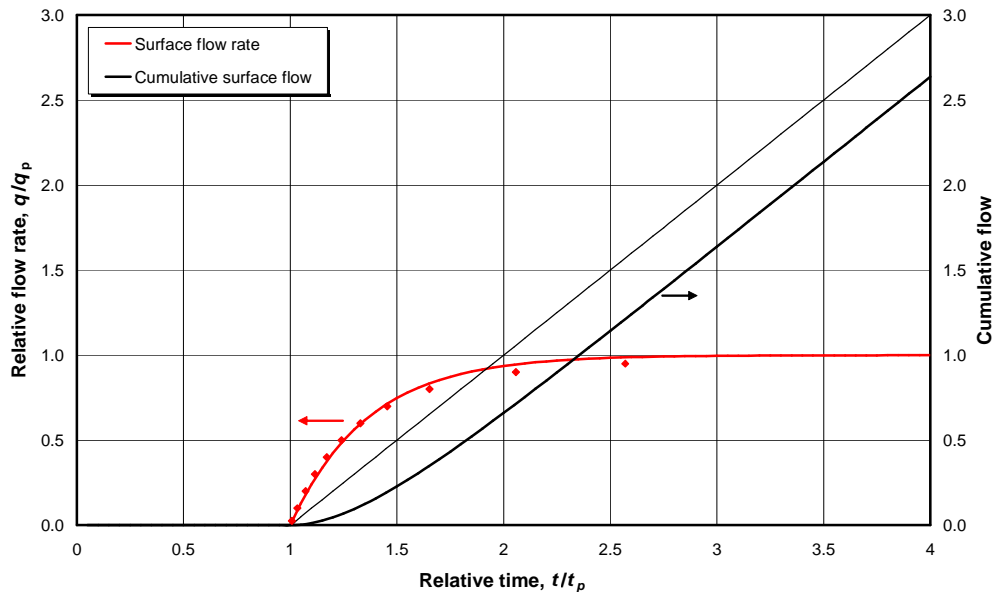


Figure B.2 Surfacing gas flow computed for a Gaussian plume velocity profile with shape factor  $\lambda = 0.65$ . The red line shows the surface flow rate  $q$  (kg/s) relative to the gas flow rate  $q_p$  in the plume, approximated with an exponential formula. The red markers show

computed values for a plume with Gaussian velocity and density profiles and shape factor  $\lambda = 0.65$ . The thick black line shows the cumulative gas flow  $Q$ , found by time integration of the red line. The thin black line is drawn for comparison and shows the cumulative gas flow without reduction, i.e.  $Q_p = q_p t$  (kg).

Figure B.2 is based on the assumption that the time development of the surface flow rate can be approximated by an exponential function of the form

$$q(t) = q_p (1 - \exp(-t/\tau)), \quad (5)$$

where  $\tau$  is the time constant. Computations made for a plume with Gaussian velocity and density profiles support this assumption, and indicates a time constant of about of 1/3 of the centerline plume rise time (see red markers on Figure B.2). Time integration of this exponential function gives the following expression for the cumulative surfacing gas flow:

$$Q(t) = q_p [t - \tau(1 - \exp(-t/\tau))] \quad (6)$$

For large times,  $t \gg \tau$ , this equation can be approximated by  $Q \approx q_p (t - \tau)$ , which can be seen to imply a time lag  $\tau$  in the cumulative surface gas flow.

The time dependent gas flow rate to the atmosphere is determined from the discharged mass of gas  $\Delta Q = q \Delta t_r$  (kg) in the time interval  $\Delta t_r = t_r(i) - t_r(i-1)$ , divided by the corresponding surfacing time period  $\Delta t_s = t_s(i) - t_s(i-1)$ :

$$q_a = \Delta Q / \Delta t_s = q \Delta t_r / \Delta t_s \quad (7)$$

A gas leak from a pipeline rupture will in general imply a sharp decrease in leak rate with time. Since the plume rise time  $t_p$  will increase with decreasing gas discharge rates, this equation implies that the gas flow rate (kg/s) to the atmosphere will tend to be reduced relative to the gas discharge rate at the leak point. This also implies that the gas release to the atmosphere will last longer than the release period at the discharge point.

## 4.2 Boiling zone

The surface flow generated by a surfacing gas bubble plume has been investigated by Fanneløp and Sjøen (1980) and Milgram and Burgess (1984). Fanneløp and Sjøen derived a model for the zone where the flow is predominantly horizontal, while Milgram and Burgess focused on the turning region. In the present context, however, we need a continuous representation of the flow pattern. For this purpose, an algebraic solution has been derived that fulfils the continuity equation for volume flow, based on an assumed exponential reduction of the vertical velocity as the plume approaches the surface where the vertical velocity will be zero.

The centerline velocity in the turning zone is thus specified as  $w(h) = w_p [1 - \exp(-h/h_0)]$ , where  $h$  is the depth and  $h_0$  is a characteristic depth of the radial flow of entrained water.



A Gaussian velocity profile is assumed in the undisturbed cross section of the plume with a centerline velocity  $w_p$  and a characteristic radius  $b_p$ .

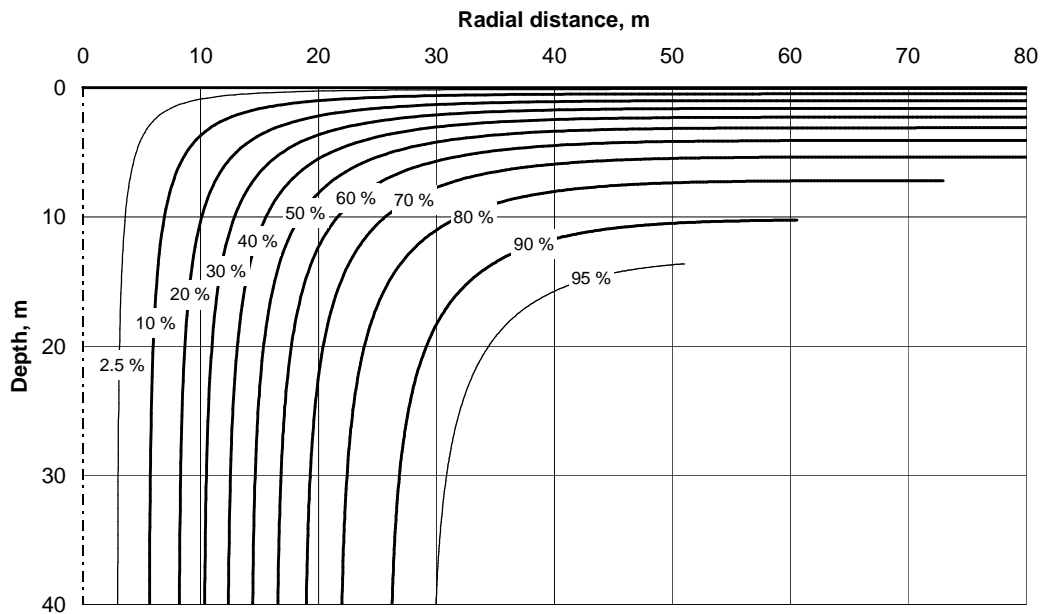
The algebraic model gives radial and vertical velocities  $u, w(r, h)$  at a certain radius  $r$  and depth  $h$ :

$$u(r, h) = 0.5 \frac{w_p b_p^2}{r h_0} \left[ 1 - \exp\left(-\frac{r^2}{b_p^2}\right) \right] \exp\left(-\frac{h}{h_0}\right) \quad (8 a)$$

$$w(r, h) = w_p \exp\left(-\frac{r^2}{b_p^2}\right) \left[ 1 - \exp\left(-\frac{h}{h_0}\right) \right] \quad (8 b)$$

The characteristic depth  $h_0 = a b_p$  is related to the plume radius by a parameter  $a = 0.37$  which has been tuned to match the initial conditions in the zone of radial flow derived by Fanneløp and Sjøen (1980). The corresponding flow field is visualized in terms of flow lines in Figure B.3, top frame.

The radial distribution of the surfacing gas flux may be determined by computing trajectories of gas bubbles rising with a slip velocity  $w_b$  in the flow field generated by the surfacing plume (see Figure B.3, bottom frame).



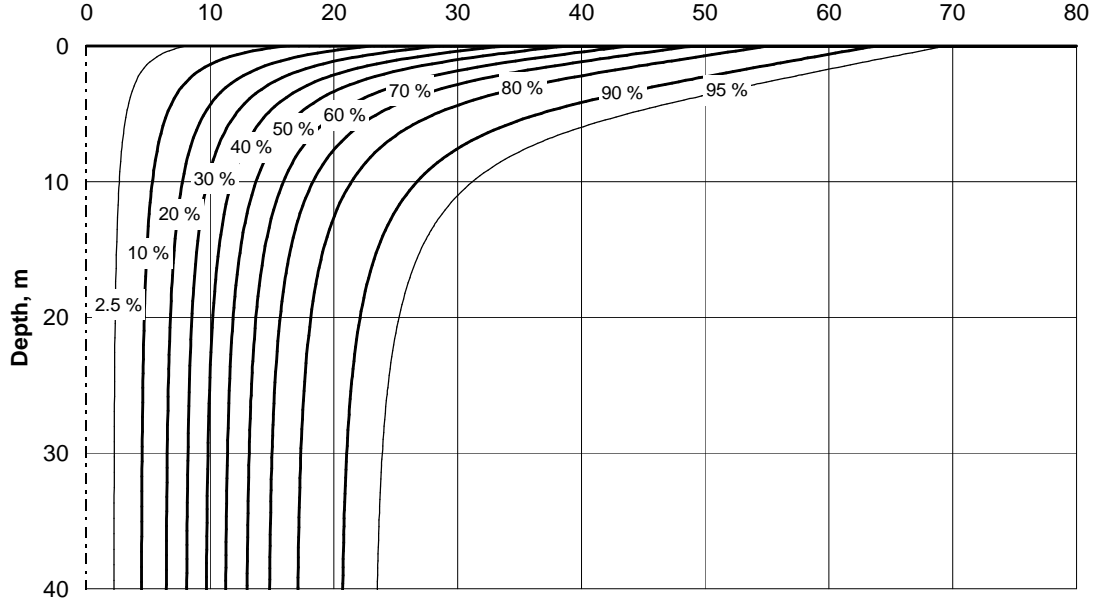


Figure B.3 Flow field generated by a surfacing gas bubble plume. The top frame shows flow lines in the water, while the bottom frame shows flow lines for gas, presuming a bubble rise velocity of 0.3 m/s.

Here, the slip velocity for gas bubbles is assumed to be 0.3 m/s in correspondence with Fanneløp and Sjøen's assumptions. Each flow line shown in the graph is enclosing a certain fraction of the gas flow in the undisturbed plume as indicated by the legend on the graph. For the present purpose, we have chosen to define the radius enclosing 90 % of the gas flow as the radius of the boiling zone, i.e.  $R_B = R_{90}$ .

In order to facilitate the computation of the boiling zone, we have calculated the radius of the boiling zone for an arbitrary set of the plume parameters  $b_p$  and  $w_p$ . A curve fit of the results in non-dimensional form is shown at Figure B.4. The thin line represents a best fit power law function based on the data points:

$$R_{90} / b_p - 1 = 0.29 (w_p / w_b)^{0.68} \quad (9)$$

where  $w_p$  and  $w_b$  (m/s) are the plume velocity computed at the surface and the bubble slip velocity.

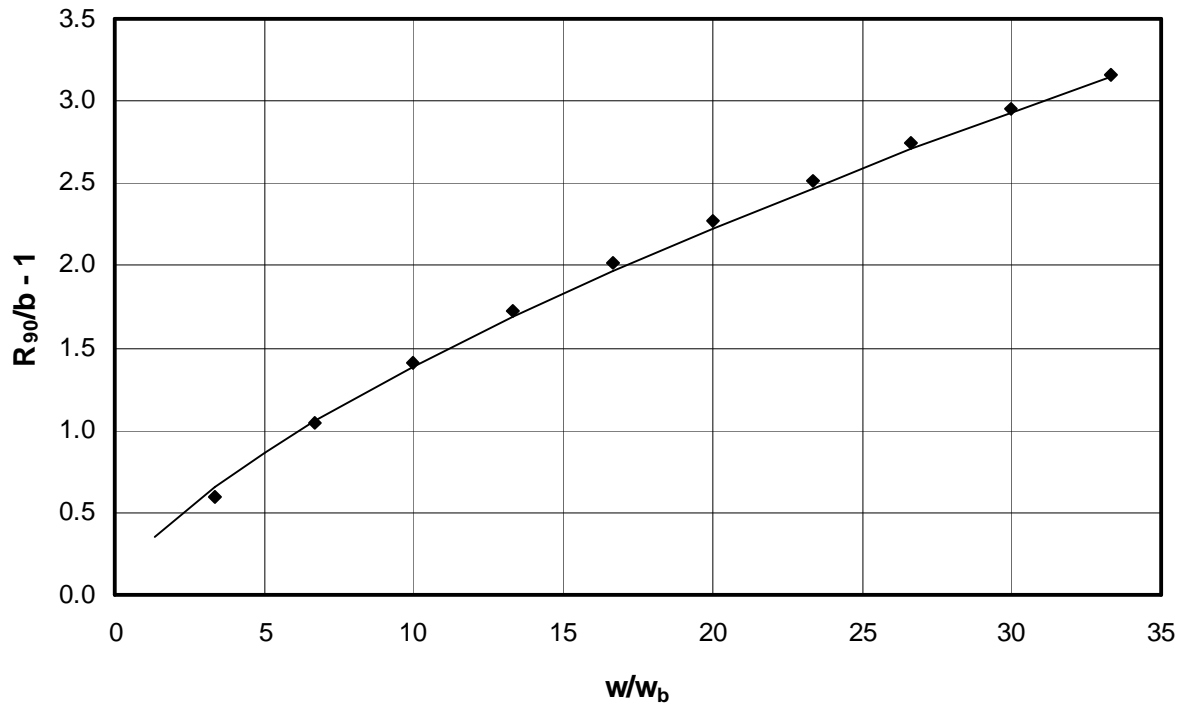


Figure B.4 Normalized plot of the radius of the boiling zone  $R_{90}$  computed with the flow line approach described above. The plume parameters  $w$  and  $b$  are varied, while the slip velocity  $w_b$  is kept constant ( $w_b = 0.3$  m/s). The thin line represents the best fit power law function (Eq. 9)

## 5 Transient leaks

As all integral plume models, Fanneløp's general solution for sub sea gas bubble plumes is based on the assumption of a stationary source. In the present context, this stationary solution is applied to instantaneous gas leak rates which vary with time. In general, such a quasi-stationary approach is presumed to be valid for slowly varying sources as long as the rise time is short relative to the time scale for the change in flow rate. However, in case of pipeline ruptures, strong transients may be expected at the start of the leak, with high initial leak rates tailing off with time as the internal pressure in the pipeline is reduced to the ambient hydrostatic pressure.

One important issue in conjunction with transient leaks is the starting plume phenomenon – i.e. the gradual build up of a plume from a source which is turned on suddenly and then maintained at a constant rate. A theory for starting plumes which has been proposed by Bettelini and Fanneløp (1993) indicates that in the initial phase, the front of the plume will develop as a nearly spherical cap attached to a “normal” coned plume below. The cap will rise slower than the plume, and thus accumulate gas as it rises towards the surface. Thus, gas may be expected to be released into the atmosphere in a strong burst as the cap reaches the sea surface. However, with the rapidly diminishing leak rates which can be expected in case of a pipeline rupture, it is not obvious that a cap can be maintained: the diminishing leak rate at the source will cause the plume to slow down, and possibly lose speed relative to the cap.

Neglecting the starting plume issue, we expect that the main effects of a diminishing leak rate will be (1) a reduction in the mass flow rate of gas through the sea surface relative to the mass flow rate at the source, and (2) an extension of the time period when gas is leaking into the atmosphere relative to the leak period at the source. These effects are demonstrated at Figure B.5 which show surfacing gas flow computed from a gas leak from an assumed pipeline rupture at 400 m depth.

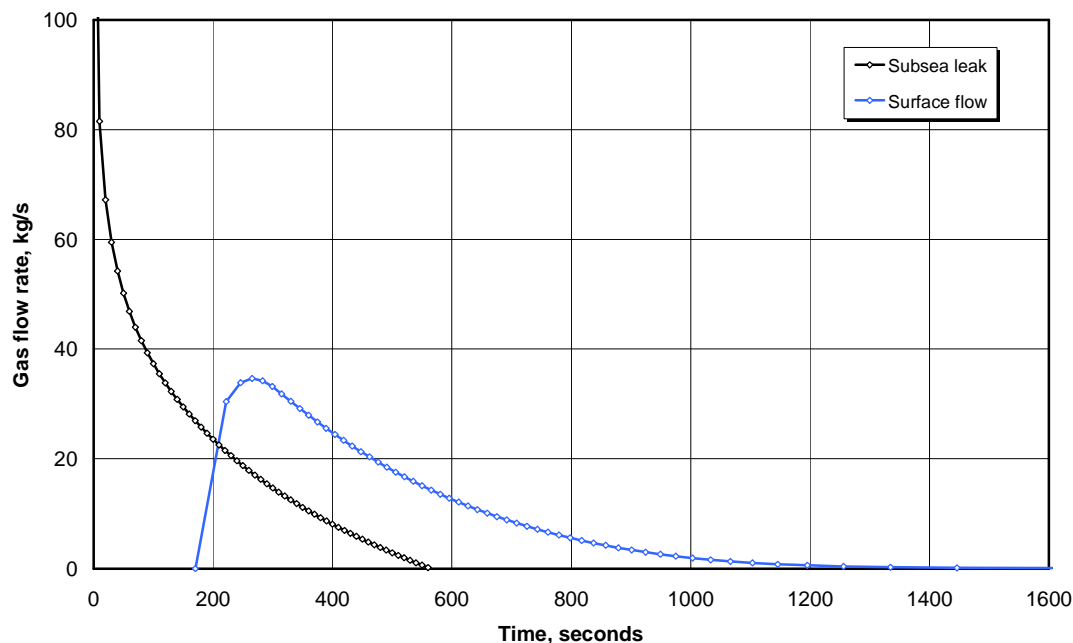


Figure B.5 Surface gas flow computed for a major sub sea gas pipe rupture at 400 m depth. The curves show the sub sea leak rate (black) and the corresponding surface flow rate (blue). The sub sea leak rate is sampled at 10 seconds intervals.

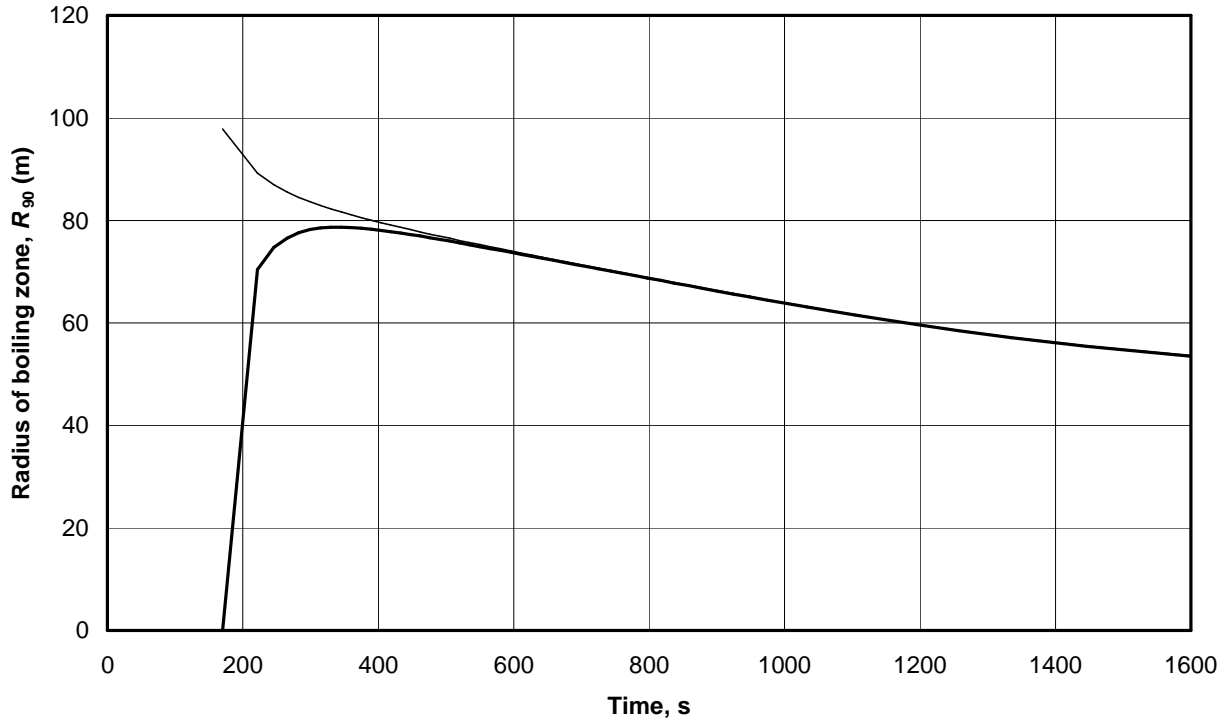


Figure B.6 Radius of boiling zone  $R_{90}$  computed for the same case as shown in Figure B.5. Thin line shows  $R_{90}$  computed by equation 9, while the thick line accounts for a gradual build up of the boiling zone (see text).

Figure B.5 shows that in this case, the surfacing of gas will be delayed by about 3 minutes (170 seconds) due to the rise time of the initial plume, and also reduced considerably in strength due to increasing rise times caused by a continuous reduction in the sub sea leak rates. For the same reason, the surface gas flow will also last for a considerably longer period than the sub sea leak.

Figure B.6 shows the radius of the boiling zone for the same case. The thin line shows the boiling zone computed directly from the plume radius and rise velocity by equation 9, while the thick line includes a gradual growth of the area of the boiling zone with time corresponding to the build up of the gas flow rate, i.e.  $\pi R^2(t)/\pi R_{90}^2 = q(t)/q_p$ , which implies

$$R(t) = R_{90} \left(1 - \exp(-t/\tau)\right)^{1/2} \quad (10)$$

where  $\tau$  is the time constant introduced in section 4.

## 6 Summary of model concept and formulas

This chapter summarizes the model concepts and formulas to be used in the programming of the model of gas behaviour in the water column as a result of the previous discussion. Note that the equations listed in this chapter are numbered in brackets, to be distinguished from equation numbers used in the previous chapters.

### 6.1 Sub sea gas bubble plume

*Input variables*

- Discharge depth  $H_0$ , m
- Gas mass flux  $q$ , kg/s
- Gas density  $\rho$ , kg/Sm<sup>3</sup> (@ 1 atmosphere and 15 °C)
- Sea temperature  $\theta_s$ , °C

*Volume flow rate  $V_0$  (m<sup>3</sup>/s) at the discharge depth*

$$V_0 = q / \rho_0 \quad [1]$$

where  $\rho_0 = \rho \frac{H_0 + 10}{10} \frac{273 + 15}{273 + \theta_s}$ , assuming ideal gas

The number 10 corresponds to 10 m water column, which equals a hydrostatic pressure of one atmosphere. The volume flux at the discharge depth is used to define the buoyancy flux parameter:

$$\phi_0 = gV_0 / \pi \quad [2]$$

*Fanneløp's general bubble plume model*

Plume variables, all given as a function of the distance  $z$  (m) above the leak point:

$b_p$ : plume radius, m

$w_p$ : plume velocity, m/s

$t_p$ : rise time, s

Non-dimensional variables:

$$X = z / H, B = b_p / 2\alpha H, W = w_p / M \text{ and } T = t_p M / H \quad [3]$$

where  $H = H_0 + 10$  and  $M = \left[ \phi_0 \frac{\lambda^2 + 1}{2\alpha^2 H} \right]^{1/3}$

The parameter  $\alpha$  is the entrainment coefficient ( $\alpha = 0.1$ ), and  $\lambda$  is a shape factor representing the ratio between the buoyancy and velocity profiles ( $\lambda = 0.65$ ), both assumed constant with depth.

Values of  $B$ ,  $W$  and  $T$  are given in Table B.2 as a function of the non-dimensional depth  $X$ .

Table B.2 Fanneløp's general bubble plume solution for isothermal expansion. Dimensionless plume variable  $B$ ,  $W$  and  $T$  as a function of dimensionless height  $X$ .

<b>X</b>	<b>B</b>	<b>W</b>	<b>T</b>
0.02	0.012	4.73	0.004
0.08	0.048	3.03	0.020
0.14	0.083	2.56	0.042
0.20	0.118	2.32	0.067
0.26	0.152	2.18	0.093
0.32	0.186	2.08	0.122
0.38	0.22	2.01	0.151
0.44	0.252	1.97	0.181
0.50	0.284	1.94	0.212
0.56	0.314	1.93	0.243
0.62	0.344	1.94	0.274
0.68	0.372	1.97	0.304
0.74	0.397	2.01	0.335
0.80	0.42	2.09	0.364
0.86	0.438	2.21	0.392
0.92	0.447	2.43	0.418
0.98	0.427	3.04	0.440

## 6.2 Surfacing of gas

### Input variables

Time series of leak rates  $q = f(t_r)$ , where  $t_r$  is release time. The leak rate values  $q(i)$  should preferably be tabulated at fixed time increments  $t(i) = i \times \Delta t$ .

### Surfacing time

Compute surfacing times from release time  $t_r$  and plume rise time  $t_p$ :

$$t_s(i) = t_r(i) + (1 + \beta)t_p(i) \quad [4]$$

where the shape factor  $\beta = 0.333$

### Surfacing rate

Compute surfacing rates from

$$q_a = \Delta Q / \Delta t_s = q \Delta t_r / \Delta t_s \quad [5]$$

where  $\Delta t_r = t_r(i) - t_r(i-1)$  and  $\Delta t_s = t_s(i) - t_s(i-1)$

### Boiling zone

Compute radius of boiling zone ( $R_{90}$ ) from:



$$R_{90}/b_p = 1 + 0.29(w_p/w_b)^{0.68} \quad [6]$$

where  $w_p$  and  $w_b$  (m/s) are the plume velocity computed at the surface and the bubble slip velocity (0.3 m/s).

For transient leaks, the gradual increase in the radius if the boiling zone may accounted for by use of the following equation:

$$R(t) = R_{90}(1 - \exp(-t/\tau))^{1/2} \quad [7]$$

where the time constant  $\tau = t_p/3$  and  $t$  is the cumulative surfacing time (time of surfacing counted from start of leak).

### 6.3 Computational procedures

Procedure to be applied for each subsequent leak rate in the time series	Eq.
1 Compute volume flow rate $V_0$ at discharge depth from gas mass flux $q$ , gas density $\rho$ and water depth $H_0$ , corrected for sea temperature $\theta_s$ .	[1]
2 Compute buoyancy flux parameter $\phi_0$ from $V_0$ .	[2]
3 Determine non-dimensional plume variables $B$ , $W$ and $T$ at sea surface, $X=H_0/H$ by interpolation in the general solution.	Table B.2
4 Compute plume variables $b_p$ , $w_p$ and $t_p$ at sea surface from $B$ , $W$ and $T$ via the scaling variables $H$ and $M$ .	[3]
5 Compute surfacing time $t_s$ from release time $t_r$ and plume rise time $t_p$	[4]
6 Compute surfacing rate $q_a$ from leak rate $q$ and incremental release and surfacing times	[5]
7 Compute radius of boiling zone $R_{90}$ from plume radius $b_p$ and plume rise velocity $w_p$ , corrected for transients with time constant $\tau$ derived from plume rise time $t_p$	[6, 7]

## 7 Testing and validation of the gas bubble plume model

The bubble plume model chosen for this project is based on the work of Fanneløp and Sjøen, published in 1980 (Fanneløp and Sjøen, 1980). The authors developed the governing equations for a bubble plume in stagnant and homogeneous water, and solved them in a non dimensional form. The results were presented as tables of the non dimensional plume rise velocity and diameter,  $W$  and  $B$  as a function of the non dimensional plume height  $X$ . The non dimensional plume variables were defined as

$$X = z/H, B = b/2\alpha H, W = w/M,$$

where  $M = \left[ \frac{\phi_0(1 + \lambda^2)}{2\alpha^2 H} \right]^{1/3}$ ,  $H = D + H_0$  and  $\phi_0 = g V_0 / \pi$ .

Here,  $D$  is the discharge depth and  $V_0$  is the volume flow of gas at the discharge point.

The solution contains two empirical constant, the entrainment coefficient  $\alpha$  and the profile parameter  $\lambda$ , the latter defining the ratio between the length scale of the buoyancy profile and the velocity profile. These parameters were determined from gas plume experiments conducted by Fanneløp and Sjøen in a 5 meter deep basin. The experiments indicated that the entrainment coefficient was depending on the gas flow rate. By including experimental results from other sources, they concluded that for small gas flow rates, the values were in the range  $\alpha \approx 0.07 - 0.08$ , and for larger flow rates the value approaches  $\alpha \approx 0.1$  or larger ( $> 10$  L/s at standard conditions). The profile parameter was estimated to  $\lambda = 0.65$ . The values for large gas flow rates have been confirmed by recent research, but smaller values have been found for the entrainment coefficient in small scale tests (Seol et al. 2007).

Fanneløp's general solution for subsea bubble plumes was developed for homogeneous water with no cross flow. For such conditions, the model has been verified against experiments. However, Socolofsky and Adams (2002) identified two factors that could change the plume behaviour in stratified waters with crossflow; the first is the possible separation of bubbles from an inclined plume, and the second is the possible trapping of the plume due to a stable stratification (the plume reaches a level of neutral buoyancy). They proposed expressions for the separation height  $h_S$  and the trapping height  $h_T$  based on experimental studies:

$$h_S = 5.1 F_B / (u u_S^{2.4})^{0.88} \quad \text{and} \quad h_T = 2.8 (F_B / N^3)^{1/4}$$

Here,  $u$  is the strength of the cross flow and  $u_S$  is the rise velocity of gas bubbles,

$F_B = V_0 g(\rho - \rho_G) / \rho$  is the buoyancy flux, and  $N = \left[ - (g / \rho) (\partial \rho / \partial z) \right]^{1/2}$  is the Brunt-Vaisälä buoyancy frequency. The density of sea water,  $\rho$  is in general much larger than the density of the gas  $\rho_G$ .

These equations may be used as a check of the possible influence of cross flow and stratification: If the smallest of the resulting heights are larger than the release depth, the plume is likely to be under no influence of the crossflow or stratification. However, since both the strength of the cross flow and the stratification will vary with depth in real cases, practical application of this criterion is not obvious. As a demonstration, we have used Socolofsky's equation for the separation height to estimate the lower limit of the gas flow rate as a function of discharge depth (Figure B.7). The curves for 25 and 50 cm/s current speed show the gas flow rate that will give a separation height equal to the discharge depth. Smaller rates may cause separation of gas bubbles during plume rise, causing a change in the plume behaviour (eventually complete loss of buoyancy).

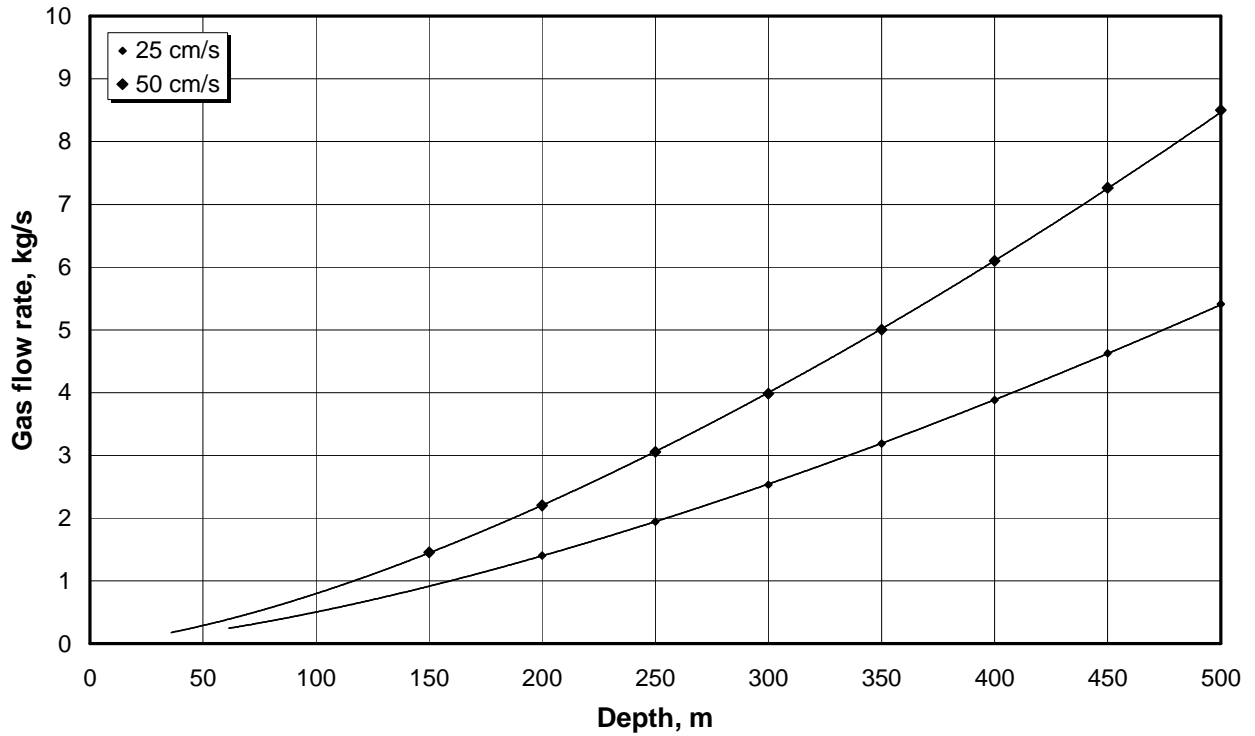


Figure B.7 Limiting gas flow rate as a function of discharge depth. The values are computed by use of Socolofsky's separation height formula.

The results in Figure B.7 show that with gas leak rates larger than 10 kg/s, the bubble plume can be expected to rise to the surface even from 500 m depth without separation of gas bubbles in cross flows up to 50 cm/s. It should be noted that in pipeline ruptures, gas leak rates can be one or more magnitudes above this limit. Moreover, due to gas expansion, the buoyancy flux generated by a pipeline rupture at the sea bed will increase as the plume approaches the surface. The present calculations, based on the buoyancy flux at the discharge level are thus on the conservative side.

A similar presentation might be made for the trapping height, but the equation for the trapping height presumes a constant density gradient. In reality, mainly the surface layers in the ocean tend to show any clear stratification (50 meter and shallower), while sea temperature and salinity gradients normally will be small (and negligible) near the sea bed. This, and the strong gain in buoyancy flux due to gas expansion as the plume rises to the surface, makes it very unlikely that a gas bubble plume from a major sub sea pipeline rupture will be trapped before it reaches the sea surface.

## 8 References

Bettelini, M.S.G. and T.K. Fanneløp, 1993: Underwater plume from an instantaneously started source. *Applied Ocean Research*, Vol. 15, pp. 195-206.

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

Fanneløp, T.K., 1994: *Fluid Mechanics for Industrial Safety and Environmental Protection*, Industrial Safety Series, Vol. 3, Elsevier Amsterdam, pp. 296-305.

Johansen, Ø, 2000: DeepBlow – a Lagrangian Plume Model for Deep Water Blowouts. *Spill Science & Technology Bulletin*, Vol. 6, pp. 103-111.

Milgram, J.H. and J.J. Burgess, 1984: Measurements of the surface flow above round bubble plumes. *Applied Ocean Research*, Vol. 6, pp. 40-44.

Seol, D-G., T. Bhaumik, C. Bergman and S.A. Socolofsky, 2007: Particle Image Velocimetry measurements of the Mean Flow Characteristics in a Bubble Plume. *Journal of Engineering Mechanics*, Vol. 133, pp. 665-676.