REPORT

A dispersal model for wastewater

Development and evaluation of the MALMAK model

Mikael Malmaeus Jakob Malm Magnus Karlsson

B2080
Dec 2012

The report approved:
2013-12-17

John Munthe
Vice President, Research

IVL Swedish Environmental Research Institute

IVL Swedish Environmental Research Institute

Organization

IVL Swedish Environmental Research Institute Ltd.

Address

P.O. Box 21060
SE-100 31 Stockholm

Telephone

+46 (0)8-598 563 00

Report Summary

Project title

A dispersal model for wastewater - Development and evaluation of the MALMAK model

Project sponsor

Stiftelsen Institutet för Vatten- och Luftvårdsforskning

Author

Mikael Malmaeus Jakob Malm Magnus Karlsson

Title and subtitle of the report

Dispersal model for wastewater

Development and evaluation of the MALMAK model

Summary

In this report the development of a model for simulation of waste water or cooling water dispersal is presented. The model, called MALMAK, is designed to provide information suitable for environmental impact assessments of the initial spreading of plumes around point sources. It works well with the limited information typically available for assessments of discharges to water recipients and does not require detailed monitoring of ambient conditions.

The model development described in this report includes an algorithm for predicting the plume width from the momentum of ambient and effluent water, a new model interface and an inbuilt graphic module enabling simulation results to be directly represented in a digital map. The model has also been validated against empirical data and against a more complex hydrodynamic model and was found to give reasonable estimates of effluent concentrations in aquatic environments.

Keyword

Waste water, Thermal discharge, Plume dispersal, Simulation model, MALMAK

Bibliographic data

IVL Report BXXXX

The report can be ordered via

Homepage: www.ivl.se, e-mail: publicationservice@ivl.se, fax+46 (0)8-598 563 90, or via IVL, P.O. Box 21060, SE-100 31 Stockholm Sweden

Contents

Introduction

Background

Dispersion in aquatic environments is a complex process dependent on the sum of different movements created by, e.g. turbulent eddies, waves and currents. When developing models it is always necessary to simplify processes, i.e. to conceptualize nature. The appropriate level of model complexity depends on various factors including data availability and the required level of detail. A dilemma is that complex models generally need many model coefficients, and each of these coefficients is uncertain, introducing errors that perturb model calculations. The level of detail acquired in advanced modeling is often of little use since small variations in boundary conditions and initial values may radically reduce model accuracy.

In the field of dispersion modeling there is a tradition of using relatively complex models requiring extensive data on water currents, bottom topography and initial dispersion coefficients. Such models are sometimes useful to study local transport processes, effluent characteristics and suitable diffusor design. They are at the same time costly to operate in terms of data requirements and expert knowledge. A common situation near modern industries is that effluents are relatively harmless and subject to extensive water treatment, and that order-of-magnitude estimates of dilution and dispersion are sufficient for most purposes.

In connection with a number of permit applications for process industries in recent years a basic model for predicting dispersion and dilution of effluent and cooling water plumes was demanded. To this end, during a number of years, a dispersion model has successively been developed at IVL. The model has been applied in a dozen case studies of mill and power plant effluent dispersion, mainly in Swedish coastal and inland waters. The results indicate that the model has the necessary qualifications to be an effective tool for environmental management.

In order to increase the availability of the model and to further improve the model performance a project has been carried out at IVL. A reference group consisting of IVL colleagues Erik Lindblom, Jonas Fejes, Karin Eliaeson, Anna Palm Cousins and Tony Persson is acknowledged for providing valuable input. In this report the proceedings from the project are documented.

Aim

The aim of the project described in this report has been a) to further develop the dispersion model, b) to further test the model against other models and empirical data and c) to document and communicate elements of the model structure and functions. The latter aim is partly accomplished by this report.

The model - MALMAK

In the following sections the version of the MALMAK model developed in this project is presented. In the following section the new developments are briefly described, and in the subsequent section the model function is explained.

Model development

The fundamental function of the MALMAK model is to calculate the dilution of a waste water plume that is transported away from the point of discharge by advection. The main numerical method applied by the model can be characterized as a cell-centered finite volume method for the conservation of mass and momentum. In the basic configuration of the model, the waste field is analyzed using a polar discretization, and a spatial mean-value for dilution is computed in each segment in a steady-state solution. It is possible to use crude spatial boundary conditions for cases such as discharge into rivers. Analysis of temperature variation in waste fields due to thermal discharge can also be carried out by the model. A detailed description of basic model equations is given in Malm (2010) and in **Appendix I** of this report.

In the context of the reported project several model developments were carried out, some of which are described in the following three subsections.

Determination of plume width and direction from momentum

A mathematical advancement of the new model version is the calculation of the plume width expansion with distance from the discharge point. Conservation of momentum controls the direction of the plume and the spread angle is determined by excess momentum within the plume relative the surroundings.

It is not unusual that effluent water is released directly from a pipe without a diffusor. In that case there will be a main direction in the momentum of the water near the release point. The direction of the effluent plume will hence be influenced by the ambient current. In order to calculate the further flow direction of the plume, the relation between plume momentum and the momentum of the water infiltrating the plume needs to be known.

In the updated model version, the momentum through each plume segment is calculated as well as the momentum of the infiltrating water. The plume direction is determined by vector addition. The relative surplus momentum also affects the spreading angle.

Graphic representation

The spatial dilution pattern resulting from the model simulations may be represented directly on 2D maps supplied to the user. Images in bitmap format can be imported by MALMAK and the discharge point, plume direction and spatial scale is defined by the user. The extension of the waste field exceeding a given concentration is then shown superimposed on the image as in the example below (**Fig. 1**).

Figure 1 *Example of a plume simulation represented by the model on a bitmap image.*

User interface

The user interface has also been elaborated in the new model version (as later shown in Fig. 6). With the new interface the user can easily navigate through the simulation process by specifying the type of discharge (thermal, other), discharge point (single point, diffusor), physical properties (discharge, ambient current, water depth) and specifications for the simulation (temporal and spatial resolution).

Model description

In the MALMAK model, the simulated plume is diluted within a surface area that is increasing proportionally to time squared. The simulation area, *i.e.* the plume extension, is divided into discrete segments as illustrated in **Figure 2**.

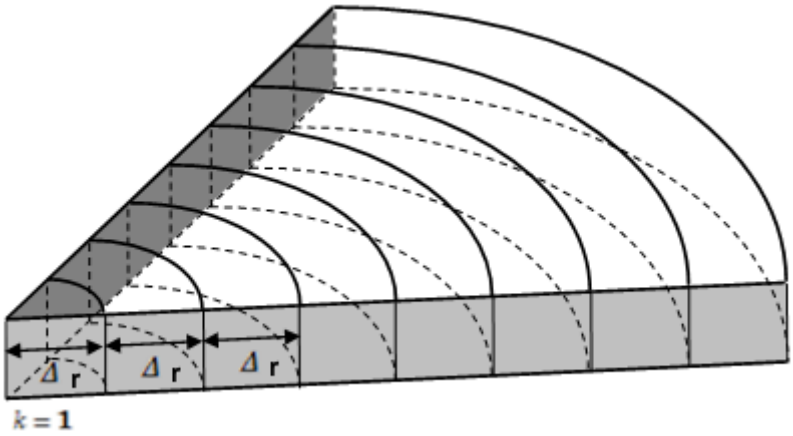

Figure 2 *Model representation of plume discretization, where* k *represents the segment number and* Δr *the segment thickness.*

Instant total mixing within each segment is assumed. Dilution of the initial plume concentration occurs as the plume expands with distance from the point of discharge and ambient water is assimilated. Incorporation of ambient water inside the plume is illustrated in **Figure 3**.

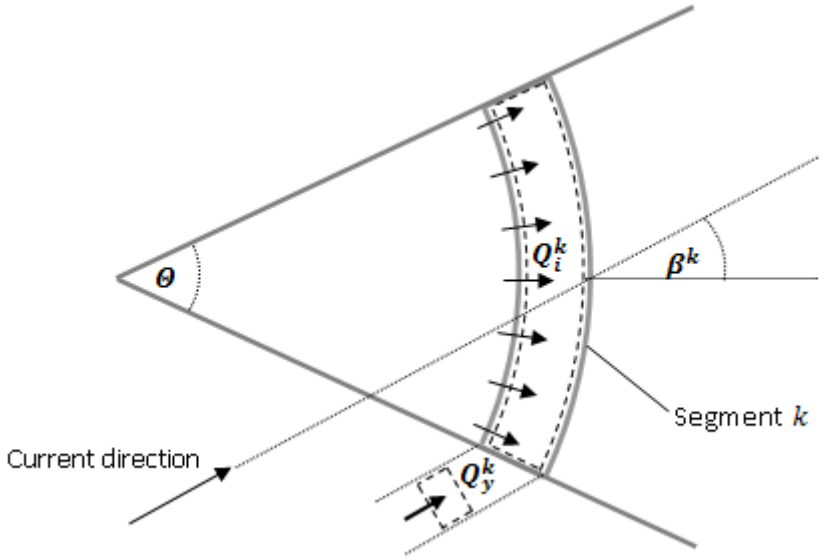

Figure 3 *Mixing of plume advection* Q_i^k *with ambient current* Q_y^k *in segment* k.

The spreading angle of the plume (Θ) and the plume direction relative to the ambient current (β^k) can either be specified by the user or estimated by the model based on the momentum relation between the plume and the ambient water.

If canal mode is chosen by the user, simulation of plume dispersal in a laterally confined environment can be performed, as illustrated in **Figure 4**.

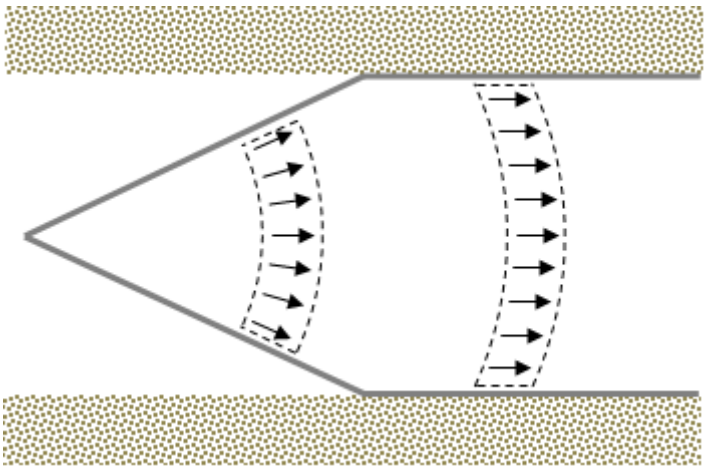

Figure 4 *Plume dispersion in a canal.*

The model can also be used to calculate water temperature in recipients of thermal discharges including equations for energy exchange with the atmosphere. An example of model output from a simulation of thermal discharge is shown in **Figure 5**.

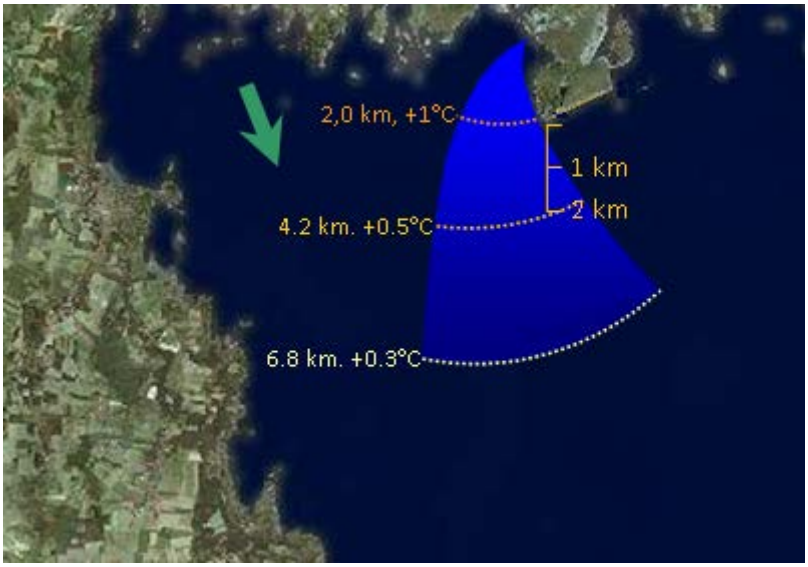

Figure 5 *Example of model output from simulation of thermal discharge. Model results agree with observed over-temperatures of 0.5-1 °C between 1 and 2 km south of the cape indicated in the figure.*

As illustrated in Figure 5 the model results may be presented on a map which may be easily uploaded by the user in the model interface. MALMAK 3.0 has a Swedish interface which is shown in **Figure 6**.

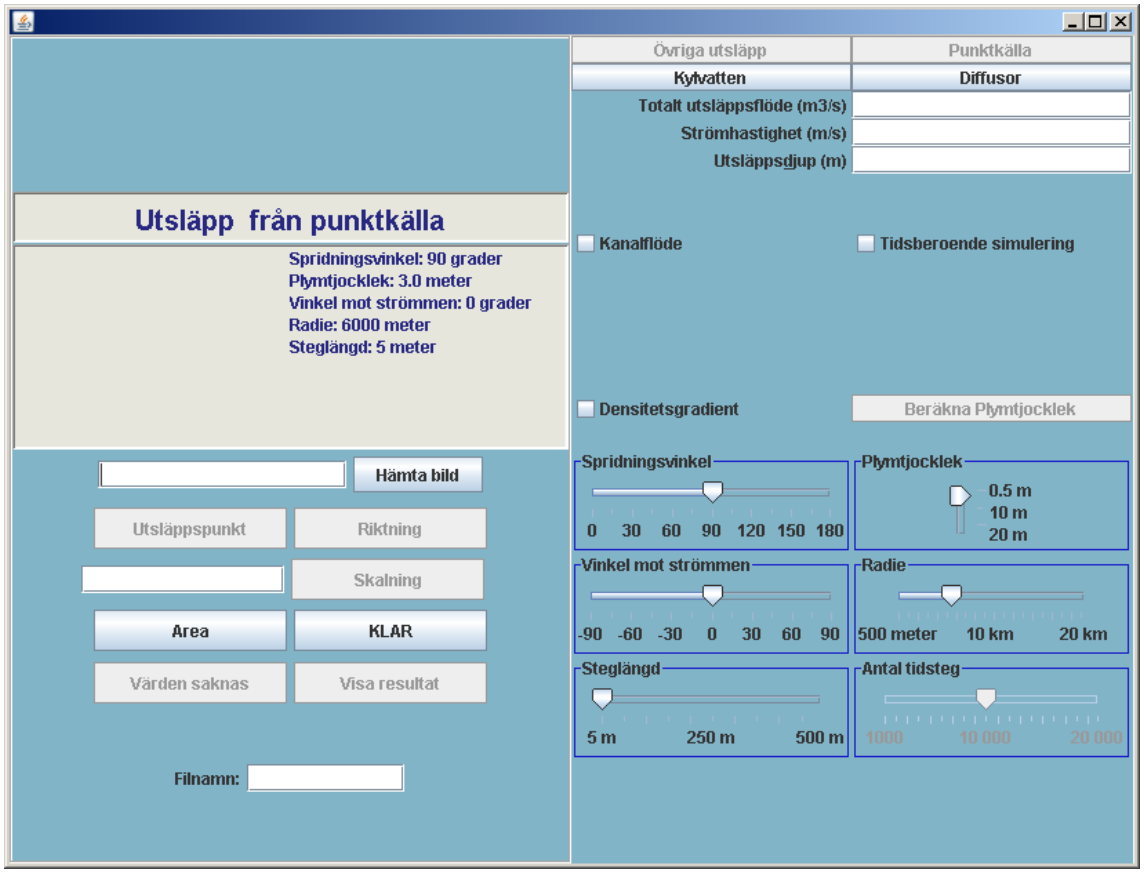

Figure 6 *Model interface of MALMAK 3.0. Fields in the lower left quadrant allow the user to load a geographic map into the model and visually scale to the simulation parameters.*

The model may be used to simulate dynamic events with variation in effluent intensity or ambient conditions, as well as static conditions with permanent discharge. The user may specify spatial and temporal resolution, but the level of detail is limited by the model simplicity. In the end mass and momentum are always conserved. A more detailed account of model equations is given by Malm (2010).

Model testing

The model has been employed in a number of practical applications during recent years. In several cases there have been some opportunities to compare model outcome with empirical observations of effluent dispersal. To evaluate model accuracy we will in the following sections describe a number of case studies where model outcome has been compared either with empirical observations or compared with results from other simulation models.

Validation of simulated temperature regime in River Visman

MALMAK was applied in a simulation of the dispersal of a 0.2 m^3/s cooling water plume from a pulp mill in River Visman (average discharge 1.2 m^3/s). Results were evaluated in three stations located 2.4 km, 3.2 km and 13 km downstream of the mill were the water

temperature was also measured. The results shown in **Table 1** suggest that the simulated temperatures were close to the measured values.

Table 1 *Validation of simulated temperature regime in River Visman.*

Month	2.4 km		3.2 km		13 km	
	Measured	Modeled	Measured	Modeled	Measured	Modeled
January	7 °C	7,0 °C	6 °C	5,6 °C	4 °C	3.4 °C
April	12 °C	11,7 °C	11 °C	10,4 C	8 °C	7,5 °C
July	29 °C	28,6 °C	27 °C	26,5 °C	22 °C	22,1 °C
October	16 °C	15,5 °C	14 °C	13,9 °C	10 °C	10 °C

Model comparison outside Swedish pulp mill Mönsterås

Outside the Swedish pulp mill Mönsterås, the Swedish Meteorological and Hydrological Institute (SMHI) applied a three-dimensional numerical computer model (PHOENICS) to simulate the dispersion of effluent discharge in order to design a new diffusor (Ivarsson, 1999). We have carried out simulations with MALMAK applying the same ambient conditions to compare the two models. The SMHI-model gives a much more detailed pattern of plume dispersal around the mill, but the average concentration at various distances from the source in the plume direction is similar between the two models. In **Figure 7** a comparison is shown at four distances from the pipe. The error bars show the variation between simulations depending on current speed (0-10 cm/s), spreading angle (45-60°) and spreading direction. These simulations demonstrate that if order-of-magnitude estimates of effluent dispersal at different distances from a point source are sufficient, the two models give similar results.

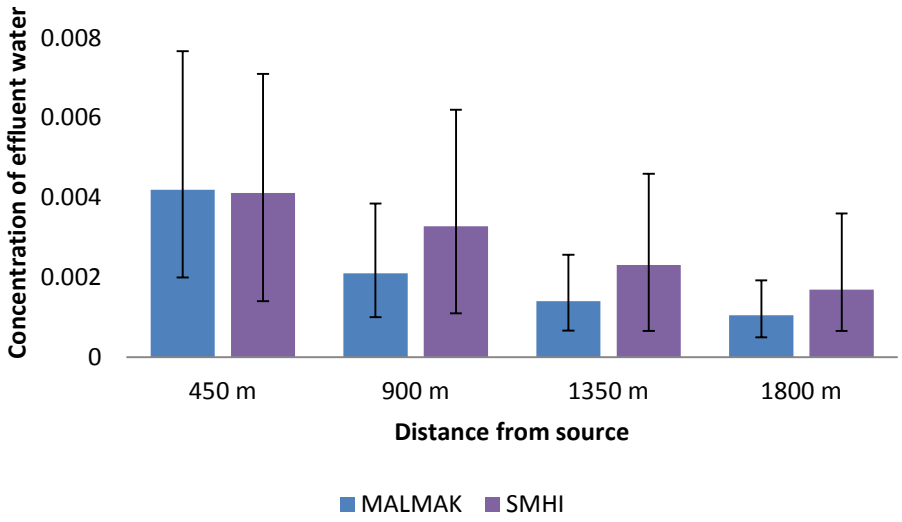

Figure 7 *Average concentration of effluent water at different distances from the source in six simulations with MALMAK and five simulations with PHOENICS.*

Concluding remarks

The model development described in this report has improved the practical usefulness of the MALMAK model. The model interface and the inbuilt graphic module enabling simulation results to be directly represented in a digital map provide a functional and flexible tool for evaluation of plume dispersal in a realistic setting. The inclusion of momentum as a determinant of the plume width has improved model accuracy. The model has been validated against empirical data and against a more complex hydrodynamic model and was found to give reasonable estimates of effluent concentrations in aquatic environments.

MALMAK does not assume detailed knowledge about water currents or bottom topography to be applicable. Detailed information provided by advanced models is sometimes necessary, but those models require large resources to be used. And even if predictions are completely accurate under certain assumptions they are often impossible to generalize in a complex reality. Average concentrations at various distances from the discharge point, as provided by the MALMAK model, is often the level of information required by operators, authorities and stakeholders to assess the long-term impact and appropriateness of industrial effluents.

Regardless of the model choice and circumstances there are always uncertainties involved in environmental assessments and thus a need for safety margins when implications from such assessments are derived. For reliable results, models must always be combined with sound judgment. Keeping these things in mind, MALMAK may often be a useful tool for making adequate environmental impact assessments of wastewater dispersal.

References

Ivarsson M., 1999. Spridningsberäkningar i Kalmarsund för Mönsterås Bruk (In Swedish). SMHI Report Nr. 1, 1999.

Malm J., 2010. Numerisk modell för spridning av avloppsvatten (in Swedish). Master thesis, Uppsala University. UPTEC W10 015.

Appendix I

NUMERIC MODEL

The model calculates the dilution or the temperature variation in an effluent plume that is transported away from the discharge point by water currents.

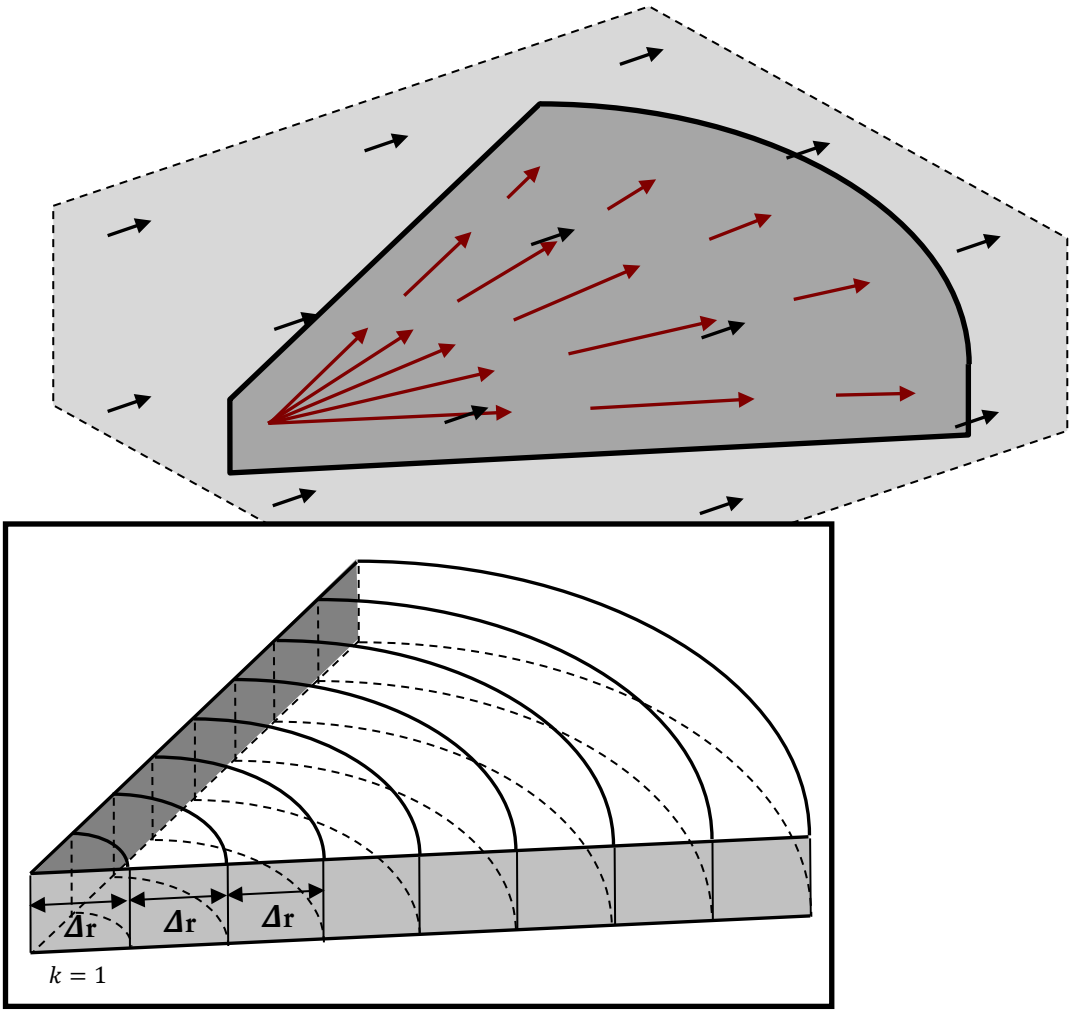

Figure 1 *Model representation of an extending waste water plume in a surrounding water current.*

1 DISCRETISATION OF THE SIMULERATION AREA

The simulation area consists of the effluent plume zone divided into circle segments with width Δr (**fig. 1**). In this report k is used as an index of the segments starting in the effluent point, $k = 1, 2, \ldots N$, where N is the number of segments. (Sometimes $k = 0$ also

exists. In that case it indicates input to the first segment or boundary conditions in a calculation.)

The volume V_{seg}^{k} of a segment is given by:

$$V_{seg}^{k} = \Delta r^2(k - 0{,}5)h\Theta, \quad (1)$$

where Θ is the spreading angle in radians and *h* is the thickness of the segment (**fig. 2**).

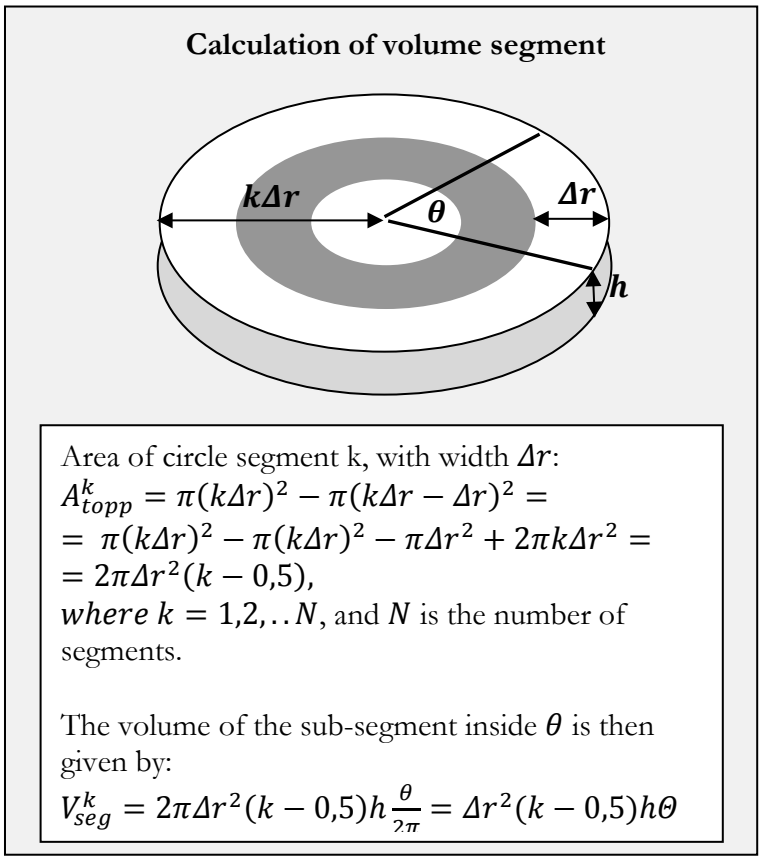

Figure 2 *Calculation of volume segment*

2 FLUX CALCULATIONS

The calculations in the model are based on mass balance, *i.e.*, the inflow to each segment always equals the outflow. The outflow from one segment is part of the inflow to the next segment. For a given segment in a polar coordinate system (Θ, r) the flux is assumed to be constant in the Θ direction.

The total flux Q^k through a segment consists of an inner flux Q_i^k, and of an outer flux Q_y^k:

$$Q^k = Q_i^k + Q_y^k \,, \tag{2}$$

where $Q_i^k = Q^{k-1}$

"Outer flux" implies the flow infiltrating the simulation area in the given segment.

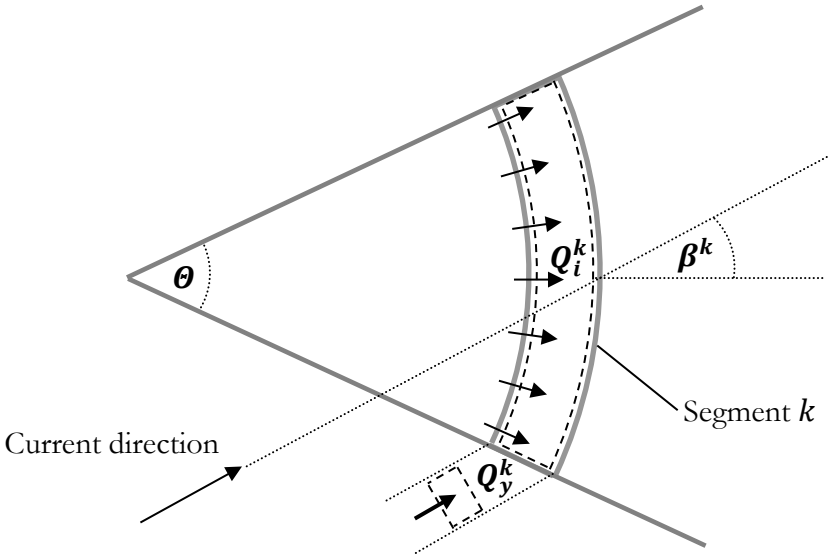

Figure 3 *The inner flux Q_i^k and the outer flow Q_y^k in segment k*

The flux Q_i^k hence equals the total outflow from the previous segment. In the first segment we have $Q_i^1 = Q_a$, where Q_a is the waste water discharge flow (input to the model).

The surrounding current infiltrates each segment with a flux Q_y^k, which size is given by the current velocity and the projected area of the segment against the current direction. The area of the side of a segment is Δrh and the area projected on the segment s^k is given by:

$$s^k = cos\left(\frac{\pi}{2} - \left(\frac{\Theta}{2} + |\beta^k|\right)\right)\Delta rh \tag{3}$$

where β^k is the angle between the center line of the waste water plume and the current direction (**fig. 4**) and h is the plume thickness. When $|\beta^k| < \frac{\Theta}{2}$ the plume is infiltrated on both sides and then the following expression for s^k is defined:

$$s^k = \left(cos\left(\frac{\pi}{2} - \left(\frac{\Theta}{2} + |\beta^k|\right)\right) + sin\left(\frac{\Theta}{2} - |\beta^k|\right)\right)\Delta rh \tag{4}$$

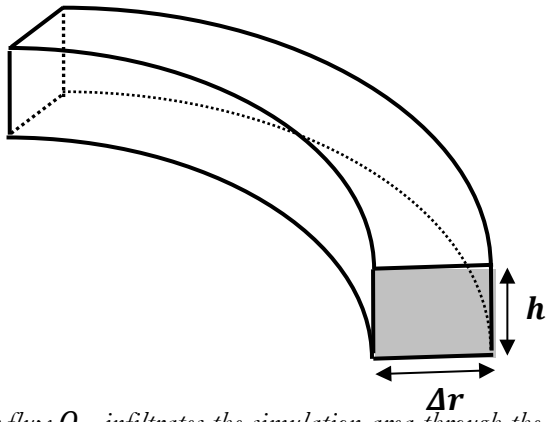

Figure 4 *The outer flux Q_y infiltrates the simulation area through the side of the segments.*

The infiltrating flux Q_y^k at current velocity *u* is then:

$$Q_y^k = s^k u \quad (5)$$

The largest possible spreading angle Θ that the model can handle is π radians and the angle against the current β must be within the interval: $-\left(\frac{\pi}{2}-\frac{\Theta}{2}\right) < \beta < \left(\frac{\pi}{2}-\frac{\Theta}{2}\right)$. The angle β^1 for the first segment is input to the model.

2.1 Channel flow

In the model it is possible to select so called channel flow which makes it possible to perform simulations of discharges in streams. This means that a limit is set for the transversal plume extension and when the waste water plume takes this value the two-dimensional flow is changed into a one-dimensional flow. This implies a different discretization with constant size of the segments. It also implies that the waste water plume is no longer infiltrated by surrounding water currents. The infiltration flux may instead be replaced by a constant groundwater infiltration per length unit of the stream.

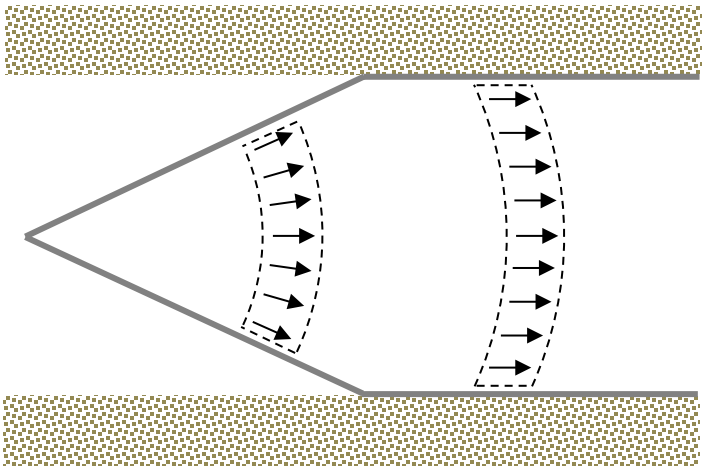

Figure 5 *Transformation into channel flow.*

3 CALCULATION OF CONCENTRATION

The model can either simulate time dependent processes, where the concentration inside the plume varies with time, or directly calculate the final concentration using Euler's method. In the latter case it is assumed that equilibrium is reached inside the plume between inflow and outflow of waste water and that the concentration in each segment no longer varies with time. This assumes a permanent discharge flux and constant current velocity.

3.1 Time dependent concentration

In the following, the model calculations of the concentration for time-dependent simulations are described.

In the calculations instant mixing in each segment is assumed and hence the same concentration applies to the whole segment. For each time step and segment the model records the volume V^k that the waste water would occupy in the segment if it was of initial concentration.

The concentration C^k is calculated in each segment by dividing the waste water volume V^k by the volume of the segment V_{seg}^k, that is:

$$C^k(t) = \frac{V^k(t)}{V_{seg}^k} \quad (6)$$

Note that C^k is a dimensionless factor indicating how the concentration in the segment is related to the initial waste water concentration. $C^k = 0{,}5$ means, for example, that the concentration in the segment is half of the concentration of the waste water leaving the diffusor.

The volume V^k is calculated at each time step as:

$$V^k(t+1) = V^k(t) + V_{in}^k(t) - V_{ut}^k(t) \quad (7)$$

where V_{in}^k and V_{ut}^k are the volumes of waste water flowing in and out from the segment at the actual time step, respectively.

If dt is the time step, then

$$V_{ut}^k(t) = Q^k C^k(t) dt \quad (8)$$

V_{in}^k always equals the outflowing volume in the adjacent segment at a given time step, that is:

$$V_{in}^k(t) = V_{ut}^{k-1}(t) \quad (9)$$

In the first segment $V_{in}^1 = Q_a(t)dt$, where Q_a is the waste water flux (input). At the first time step and the first segment $V^1(1) = 0$ and $C^1(1) = 0$. At each time step the segments are updated in sequence. Below follows the sequence of calculations for each segment and time step:

1. $Q_i^k(t) = Q^{k-1}(t)$, $Q_a(t)$ in the first segment.
2. $s^k(t)$ according to equation 3 or 4.
3. $Q_y^k(t) = s^k u(\mathrm{t})$
4. $\beta^{k+1}(t) = \beta^k(t) - \Delta\beta^k(t)$ (see section 4, equation 15.)
5. $Q^k(t) = Q_i^k(t) + Q_y^k(t)$
6. $V_{in}^k(t) = V_{ut}^{k-1}(t)$, $Q_a(t)dt$ in the first segment.
7. $V_{ut}^k(t) = Q^k C^k(t)dt$
8. $V^k(t+1) = V^k(t) + V_{in}^k(t) - V_{ut}^k(t)$
9. $C^k(t+1) = \frac{V^k(t+1)}{V_{seg}^k}$

$Q_a(t)$ and u(t) are inputs to the model and thus functions defining how the waste water flow and/or the water current vary with time.

3.2 Equilibrium concentration

If the waste water flux and current velocity are constant the equilibrium concentration is calculated using Euler's method. Equilibrium inside the plume implies that the concentration in each segment does not vary with time, that is:

$C^k(t) = C^k(t+1)$, which plugged into (6) gives that: $V^k(t) = V^k(t+1)$.

Since (21): $V^k(t+1) = V^k(t) + V_{in}^k(t) - V_{ut}^k(t)$ we know that:

$$V_{in}^k(t) = V_{ut}^k(t) \qquad (10)$$

In the following, the time vector is dropped since it is no longer needed.

From (9) and (10) we get that $V_{ut}^{k-1} = V_{ut}^k$, which together with (8) gives that:

$$Q^{k-1}C^{k-1} = Q^k C^k$$

Once the fluxes in the segments are calculated the equilibrium concentration can thus be calculated as:

$$C^k = C^{k-1}\frac{Q^{k-1}}{Q^k}, \qquad (C^0 = 1 \text{ and } Q^0 = Q_a) \qquad (11)$$

This is Euler's method applied for the differential equation:

$$\frac{dC}{dx} = \frac{-dQ}{dx}\frac{C}{Q} \qquad (12)$$

With constant conditions the following calculations are performed for each segment starting with the first segment:

1. $Q_i^k = Q^{k-1}$
2. s^k according to equation 3 or 4.
3. $Q_y^k = s^k u$
4. $\beta^{k+1} = \beta^k - \Delta\beta^k$ (see section 4, equation 25.)
5. $Q^k = Q_i^k + Q_y^k$
6. $C^k = C^{k-1}\frac{Q^{k-1}}{Q^k}$, $\quad (C^0 = 1 \text{ och } Q^0 = Q_a)$

4 TEMPERATURE CALCULATIONS

These calculations are based on an expression from Wörman (2000), which is an analytical determination of temperature variations in a stream downstream from a thermal discharge.

Since the flux in the model is two-dimensional the arc length is used here instead of the width. Ground water in Wörman (2000) is replaced by the infiltration of surrounding water in the waste water plume due to water currents. Variables affecting the energy exchange with the atmosphere are hence temperature, flux and the arc length of the plume. The retention time V_{seg}^k/Q^k is inversely proportional to the flux which implies that the water passing a certain distance loses less energy with increasing water flux. The energy exchange with the atmosphere is also proportional to the plume area facing the atmosphere. The upper area A_{topp}^k of segment *k* is given by:

$A_{topp}^k = \Delta r b^k$, where $b^k = \Delta r(k - 0{,}5)\Theta$ is the arc length in the middle of the segment (see also figure 3 and 7).

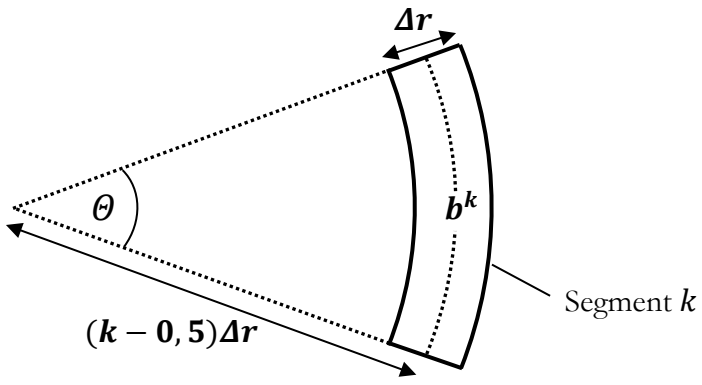

Figure 6 *Upper area of the segments, $A_{topp}^k = b^k \Delta r$, where b^k is the arc length in the middle of the segment.*

The term $\frac{1}{Q}\frac{dQ}{dx}$ is the dilution. In this numerical model dQ is approximated by:

$dQ = \Delta Q^k = Q^k - Q^{k-1}$ and $dx = \Delta r$ (width of the segments)

The following terms are introduced:

$T_{\ddot{o}v}^k = -(T_u - T^k)$ and $T_{kons} = (T_a - T_u)$,

$T_{\ddot{o}v}^0$ is hence the over-temperature of the thermal discharge compared to the diluting water when it exits the effluent point.

With terms as above the expression for dT in the numerical model is given by:

$$dT = \Delta T^k = \frac{c_a b^k}{\rho c_p Q^k}\Delta r(-T_{\ddot{o}v}^{k-1} + T_{kons}) + \frac{\Delta Q^k}{Q^k}(-T_{\ddot{o}v}^{k-1}) \quad (13)$$

where $k = 1, 2, \ldots N$, and N is the number of segments.
The term $\frac{c_a b^k}{\rho c_p Q^k}\Delta r(-T_{\ddot{o}v}^{k-1} + T_{kons})$ is the part of the temperature change explained by energy exchange with the atmosphere and the term $\frac{\Delta Q^k}{Q^k}(-T_{\ddot{o}v}^{k-1})$ describes the influence from dilution.

First dilution is considered: $\frac{\Delta Q^k}{Q^k}\left(-T_{\ddot{o}v}^{k-1}\right) = \frac{Q^k - Q^{k-1}}{Q^k}\left(-T_{\ddot{o}v}^{k-1}\right)$.

Since $Q^{k-1} = Q^k \frac{C^k}{C^{k-1}}$ we have that: $\frac{Q^k - Q^{k-1}}{Q^k}\left(-T_{\ddot{o}v}^{k-1}\right) = \frac{Q^k - Q^k\frac{C^k}{C^{k-1}}}{Q^k}\left(-T_{\ddot{o}v}^{k-1}\right) = \left(1 - \frac{C^k}{C^{k-1}}\right)\left(-T_{\ddot{o}v}^{k-1}\right)$

Using this expression Euler's method is then used to calculate the temperature $T_{\ddot{o}v}^k$ in the segments:

$$T_{\ddot{o}v}^k = T_{\ddot{o}v}^{k-1} + \Delta T^k = T_{\ddot{o}v}^{k-1} + \frac{c_a b^k}{\rho c_p Q^k}\Delta r\left(-T_{\ddot{o}v}^{k-1} + T_{kons}\right) + \left(1 - \frac{C^k}{C^{k-1}}\right)\left(-T_{\ddot{o}v}^{k-1}\right)$$

$$= \frac{c_a b^k}{\rho c_p Q^k}\Delta r\left(-T_{\ddot{o}v}^{k-1} + T_{kons}\right) + \frac{C^k}{C^{k-1}}T_{\ddot{o}v}^{k-1},\ C^0 = 1 \quad (14)$$

Hence first the temperature in the first segment is calculated and the result is then used to calculate the temperature in the next segment, and so forth. In simulations with constant conditions equation (13) is the seventh calculation in each segment, that is the update in each segment is performed as follows:

1. $Q_i^k = Q^{k-1}$

2. s^k according to equation 3 or 4.
3. $Q_y^k = s^k u$
4. $\beta^{k+1} = \beta^k - \Delta\beta^k$ (see section 4, equation 25.)
5. $Q^k = Q_i^k + Q_y^k$
6. $C^k = C^{k-1}\frac{Q^{k-1}}{Q^k}$, $(C^0 = 1 \text{ and } Q^0 = Q_a)$
7. $T_{öv}^k = \frac{c_a b^k}{\rho c_p Q^k}\Delta r\left(-T_{öv}^{k-1} + T_{kons}\right) + \frac{C^k}{C^{k-1}}T_{öv}^{k-1}$

4.1 Effect of dilution on temperature

Ignoring the influence of the atmosphere on water temperature the following is true for the first segment:

$$T_{öv}^1 = T_{öv}^0 + \Delta T_u^1 = T_{öv}^0 + \left(1 - \frac{C^1}{C^0}\right)(-T_{öv}^0)$$

Since $C^0 = 1$, we have that: $T_{öv}^1 = C^1 T_{öv}^0$

In the same way for the second segment:

$$T_{öv}^2 = T_{öv}^1 + \Delta T_u^2 = T_{öv}^1 + \left(1 - \frac{C^2}{C^1}\right)(-T_{öv}^1) = \frac{C^2}{C^1}T_{öv}^1 = C^2 T_{öv}^0$$

By induction follows that:

$$T_{öv}^k = C^k T_{öv}^0 \qquad (15)$$

As expected the effect of dilution on temperature is such that the temperature relation between two segments is directly proportional to the relation of the concentrations in each segment. Once equilibrium concentrations are determined only the temperature in the first segment is needed to calculate the effect of dilution on temperature in the other segments.

5 MOMENTUM CALCULATIONS

It is not unusual that waste water is released directly from a pipe without a diffusor. In that case there is a principal direction in the waste water momentum at the discharge point. The direction of the waste water plume will subsequently be influenced by the surrounding current. To calculate the resulting direction of movement of the waste water plume, the relation between the plume momentum P_i^k and the momentum of the infiltrating water in each segment P_y^k needs to be known.

To calculate the plume momentum the momentum of the volume $Q^k dt$ that during a small time step dt is transported through a section of a segment k is first considered (a section along the arc in the middle of the segment).

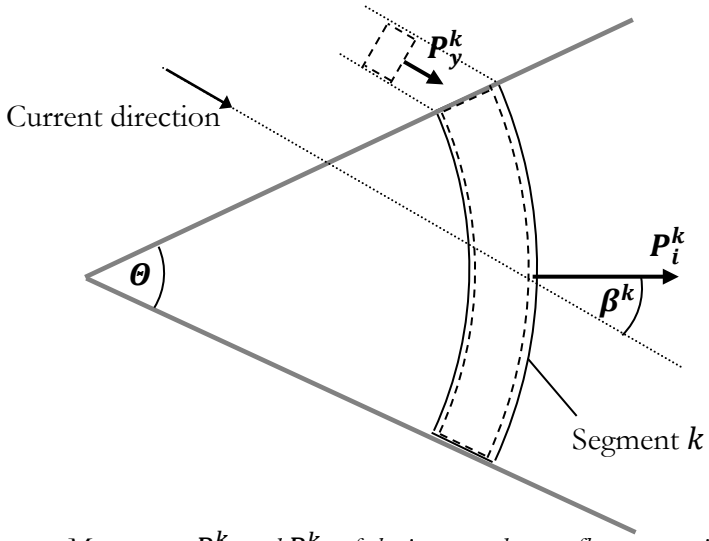

Figure 7 *Momentum P_i^k and P_y^k of the inner and outer flux, respectively.*

This volume is regarded as a homogenous "system" and P_{tot}^k will denote the absolute value of the total inner momentum of this system ignoring the direction, that is $P_{tot}^k = \rho_o Q_i^k u^k dt$, where ρ_o is the densitety and $u^k = \frac{Q_i^k}{b^k h}$ is the current velocity through the section (not to be confused with u which is the surrounding current velocity).
A function $g(y)$ is defined for the interval $\frac{-w^k}{2} \leq y \leq \frac{w^k}{2}$ such that $g_m = p^k = \frac{P_{tot}^k}{w^k}$,

g_m is hence the mean of the function $g(y)$ in the interval and w^k is the plume width in the middle of segment k. Hence:

$$P_{tot}^k = \int_{\frac{-w^k}{2}}^{\frac{w^k}{2}} g(y)dy \qquad (16)$$

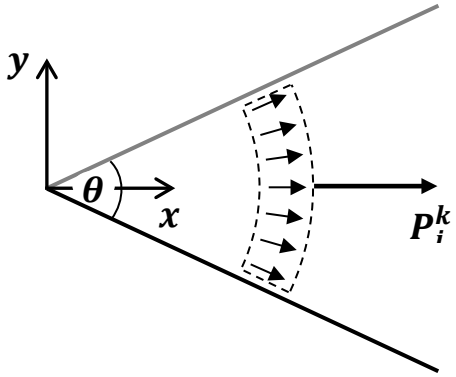

Figure 8 *The inner momentum elements with absolute values p^k together make up the total inner momentum P_i^k.*

Due to symmetry only the x-components of the inner momentum elements contribute to P_i^k which thus represents the total momentum of the system including direction (**fig. 8**).

$$P_i^k = \int_{\frac{-w^k}{2}}^{\frac{w^k}{2}} f(y)dy \tag{17}$$

$$f(y) = p^k \left(1 - \left(\frac{y}{L}\right)^2\right)^{0,5}, \; L = (k - 0{,}5)\Delta r \tag{18}$$

To facilitate the integration the inverse substitution is introduced:
$y = h(\alpha) = Lsin(\alpha), \frac{-\Theta}{2} \leq \alpha \leq \frac{\Theta}{2}$ (Θ = spreading angle).

$dy = Lcos(\alpha)d\alpha$

$f(h(\alpha)) = p^k cos(\alpha)$

The integral can then be solved:

$$P_i^k = \int_{\frac{-\Theta}{2}}^{\frac{\Theta}{2}} p^k Lcos^2(\alpha)d\alpha = p^k \frac{L}{2}\left(\Theta + sin(\Theta)\right) \tag{19}$$

$P_{tot}^k = g_m w^k$

$P_i^k = f_m w^k$, where f_m is the mean of $f(y)$ in the interval: $\frac{-w^k}{2} \leq y \leq \frac{w^k}{2}$

This gives that:

$$\frac{P_i^k}{f_m} = \frac{P_{tot}^k}{g_m} \rightarrow \; P_i^k = P_{tot}^k \frac{f_m}{g_m} \tag{20}$$

$$f_m = \frac{1}{w^k}\int_{\frac{-w^k}{2}}^{\frac{w^k}{2}} f(y)dy = \frac{p^k L(\Theta + sin(\Theta))}{2w^k} \tag{21}$$

$sin\left(\frac{\Theta}{2}\right) = \frac{w^k}{2L} \rightarrow w^k = 2Lsin\left(\frac{\Theta}{2}\right)$. Plugged into (20):

$$f_m = \frac{p^k(\Theta + sin(\Theta))}{4sin\left(\frac{\Theta}{2}\right)} \tag{22}$$

$$P_i^k = P_{tot}^k \frac{f_m}{g_m} = \rho_o Q_i^k u^k dt \left(\frac{\Theta + sin(\Theta)}{4sin\left(\frac{\Theta}{2}\right)} \right) \quad (23)$$

The momentum P_y^k of the infiltrating water is given by:

$$P_y^k = \rho_a Q_y^k u dt \quad (24)$$

The angle β^1 which at the point of discharge is the difference between the direction of the waste water plume and the direction of the surrounding current is needed as input to the model. The more water that is infiltrated in the plume the greater impact on the plume direction by the added momentum. The plume direction will thus successively adapt to the direction of the surrounding current.

The angle β^2 which is the angle between the directions of the plume and the surrounding current in segment 2 is thus determined by the relation between P_i^1 and P_y^1. In the same way β^{k+1} is then determined by the relation between P_i^k and P_y^k.

$$\beta^{k+1} = \beta^k - \Delta\beta^k \quad (25)$$

The angle $\Delta\beta^k$ is calculated using vector addition:

$$\Delta\beta^k = atan\left(\frac{P_y^k \sin(\beta^k)}{P_i^k + P_y^k \cos(\beta^k)} \right)$$

$$= atan\left(\frac{\rho_a Q_y^k u \sin(\beta^k)}{(Q_i^k)^2 \frac{\rho_o}{\Delta r(k-0{,}5)\Theta h}\left(\frac{\Theta + sin(\Theta)}{4sin\left(\frac{\Theta}{2}\right)} \right) + \rho_a Q_y^k u \cos(\beta^k)} \right) \quad (26)$$

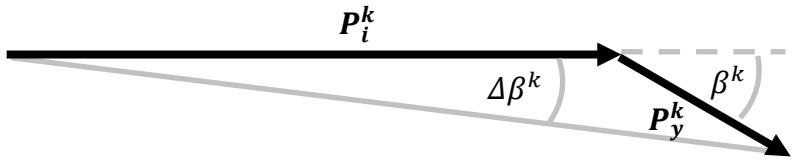

Figure 9 *The plume direction is calculated using vector addition.*

6 REFERENCE:

Wörman A., 1998. Beräkningen av temperaturförändringen i vattendrag nedströms om ett varmvattenutsläpp (In Swedish). Institutionen för geovetenskaper, Uppsala Universitet, stencil, 5p.