waterloopkundig laboratorium
delft hydraulics laboratory

combined use of hydraulic and mathematical models in the design of a once-through cooling circuit along an estuary

H. Ligteringen

publication no. 179

May 1977

combined use of hydraulic and mathematical models in the design of a once-through cooling circuit along an estuary

paper presented at the Waste Heat
Management and Utilization Conference,
Miami Beach, Florida, May 9-11, 1977

H. Ligteringen

publication no. 179

May 1977

CONTENTS

COMBINED USE OF HYDRAULIC AND MATHEMATICAL MODELS IN THE DESIGN OF A ONCE-THROUGH COOLING CIRCUIT ALONG AN ESTUARY

by H. Ligteringen, Delft Hydraulics Laboratory, Delft, The Netherlands.

Abstract

For powerplants to be located along tidal estuaries, the temperatures in the intake and outlet of the cooling-water circuit and the temperature field at the outfall site can be influenced by:

(i) the outlet design, affecting the near-field temperature distribution, and
(ii) the intake design, provided that temperature varies sufficiently with depth at the point of intake

Further it is necessary to consider

(iii) the return of heat, a far-field effect, caused by the reversal of tidal currents

The geometry being a complicated one, item (i) and (ii) require a hydraulic near-field model. Often a mathematical model is the most economical tool to study item (iii). This paper illustrates how a study, involving both prediction methods was performed for the Borssele Powerplant along the Western-Scheldt estuary in the Netherlands.

1 Introduction

The Borssele Powerplant is located along the Western Scheldt in the Netherlands (see Figure 1). Its present capacity is 860 MWe. The possibilities of a plant expansion to a capacity of 5000 MWe have been investigated. The corresponding rise in cooling water flowrate for once-through cooling would be from 30 m^3/s to approximately 250 m^3/s, with a temperature increase over the condensors of 10^oC.

As the Western Scheldt is a tidal estuary, during a certain part of each tidal period the cooling water intake will be at the downstream side of the outlet. Consequently recirculation of heated effluent may occur, especially because of the relative short distance between the existing and the projected outlet and intake locations (Figure 2).

To limit this recirculation was one of the most important aspects of the study concerning the temperature effects of this vast heat disposal, executed in 1975 and 1976 by the Delft Hydraulics Laboratory.

With a maximum allowable temperature $T_{max} = 30^{o}C$ in the outlet and natural summer temperatures exceeding $20^{o}C$, every degree temperature increase in the intake would mean a reduction in plant capacity, apart from the loss of efficiency. In winter too high surface temperatures, causing an appreciable increase in fog formation, would be unfavourable with regard to the traffic of ships, since the navigation channel runs close to the plant-site.
In view of these considerations the study concentrated upon the following questions:

- which design to select for the new intake and the new outlet to minimize recirculation, taking the existing cooling circuit into account
- predicting the temperatures in the existing and the new intake
- predicting the increase in background temperatures of the ambient water

2 Methods of analysis

Tentative design considerations showed that the effect of return heat upon the intake temperature could be significant. This far-field effect has to be accepted as it is little affected by the design of intake and outlet. Since the same analysis indicated that the direct recirculation could be high, when the intake is at the downstram side of the outlet, all efforts had to be concentrated on the design of the intake and the outlet in order to reduce this part of the recirculation.

The effects of design and the irregularity of the near-field geometry upon recirculation can not be sufficiently incorporated in a mathematical model nor in any of the existing mathematical prediction techniques [1]. The study was therefore divided into the following parts:

- a hydraulic model study to evaluate two basically different designs of both the new outlet and intake and to optimize the selected basic design. Further this model provided proper boundary conditions for the far-field mathematical model with respect to initial dilution around the outlet
- a mathematical model study to determine the effect of return heat upon intake temperatures and the environmental effects
- a field study to obtain data both to verify the hydraulic model and to determine the dispersive characteristics of the Western Scheldt, as necessary input for the mathematical model

In the hydraulic-model study, which is described in further detail in section 3, a number of tidal stages with corresponding current velocities and patterns were investigated consecutively as quasi-steady situations (Figure 3). Only for the situation after LWS the transient behaviour of the tidal flow was reproduced in the hydraulic model to study the accumulation of heat during the period of approximately 1 hour before up to 1 hour after LWS.

The idealized mathematical model, which is elaborated upon in section 4 is based on the superposition principle. It computes the time-dependent spatial distribution of temperature resulting from an instantaneous heat discharge at $t = 0$ under influence of

- dispersion
- advection
- heat exchange to the atmosphere

The dispersive characteristics occuring in the estuary were obtained from instantaneous dye-release experiments at full scale. In accordance with the superposition method the variation of concentration with time as observed in these experiments is assumed to be representative for all subsequent releases into which the continuous discharge of heated effluent can be divided. The total temperature elevation at time t at a location consists of the contribution of all releases between $t = 0$ and t. The introduction of a sinusoidal component in the advection equation makes it possible to simulate schematically the tidal movement of the ambient water (see Figure 4).

3 Hydraulic-model investigation

The hydraulic model, scale 1 in 75 both horizontally and vertically, represented a prototype area of 4 km along the bank on both sides of the powerstation by 2 km perpendicular to the shore (see Figure 5). The model boundaries were designed to facilitate an easy change of current direction or waterlevel.
The heat production in the model was achieved by means of three large boilers, from which a servo-regulated system injected the required amount of hot water into the model internal cooling-water circuits, to maintain a constant temperature difference between intake and outlet of $\Delta T_o = 10^oC$.
The circuits of the existing and projected cooling-water flows were separated in order to detect possible differences in recirculation. The effect of heat-exchange to the atmosphere could be neglected, since within the model area temperature reduction due to this phenomenon was less than 2%.

The temperature distribution in the model and the temperatures in intake and outlet were measured by means of 75 thermistors (YS-423). The data collection was achieved with a Philips PM 2400 data-logging system. The model was calibrated in two steps: firstly the current-patterns and local velocities during the tidal phases to be investigated were reproduced according to field measurements. Thereafter the temperature distribution around the existing outlet was compared with the results of a prototype field-survey for two tidal stages: LWS and maximum ebb flow.

Two basically different design approaches were tested in the model:

- a high initial mixing design, effectuated with a mulitport-diffuser system, in combination with open-intake channels
- a low initial mixing design, achieved by means of a wide shallow outlet channel, in combination with a skimmer-wall intake

The results of a preliminary evaluation of both designs are given in table 1 (section 3.3) and did not show a distinctive preference for one system or the other.

3.1 Multiport-diffuser design

The considerations leading to this design were:

- reduction of surface temperature by high initial dilution (fog prevention)
- reduction of recirculation, also in the existing intake
- relative simple design of the intake, advantageous because of the importance of an undisturbed foreland to the stability of the embankment
- the existance of a wide and relatively level clay formation at some depth at the outfall-site offering an ideal opportunity to apply multiport diffusers, preventing scouring or sedimentation around the pipes.

The width of this natural plateau was approximately 500 m normal to the shoreline, its depth averaged 20 m. Making use of the available space diffuser pipes of 500 m length having a discharge of 55 m^3/s each would discharge at a flowrate per unit width $q = 0.11\ m^2/s$. The initial design was developed on the basis of experimental data concerning dilution of two-dimensional diffusers in stagnant water [2]. For this situation, occuring around slack water, an initial dilution $S = 12$ could be achieved at reasonable construction and operation costs.

Under assumption that the effluent will mix fully with the receiving water for cross currents exceeding U_a = 0.10 m/s [2], the ratio of total cooling-water discharge and ambient flowrate indicate a dilution S = 12 for a cross-current velocity U_a = 0.25 m/s. Maximum dilution S ≃ 50 occurs for maximum ebb and flood flow.

The spacing of the diffuser pipes (see Figure 6) was based on the demand for sufficient cold water to maintain a dilution S = 12 during the period with velocities ranging from - 0.25 m/s to + 0.25 m/s around LWS and HWS. Strictly two-dimensional computations led to a distance of 700 m between consecutive diffuser lines. In the first design tested in the model three-dimensional aspects were accounted for to some extent: a distance of 500 m was chosen initially.

The model tests showed good agreement with the expectations as far as initial dilution in stagnant water and the depth of the heated upper layer were concerned. However the computations with respect to the accumulation of heat during the periods of slack water proved to be too conservative. Where accumulation is defined as the effect of re-entrainment of discharged water, causing a decrease in the required dilution accompanied by higher surface temperatures, this phenomenon hardly occured in the results of the tests at the end of the LWS period for the initial design with 500 m spacing (a) and for respectively 350 m (b), 200 m (c) and 50 m (d) as is shown in Figure 7.

This fact must be atributed to three-dimensional and cross-current effects. The 3-D effect becomes more important when the distance between the diffusers is relatively large. The cross-current effect becomes more important when the distance between the diffusers is relatively small, as by then the length (measured in the direction of flow) of the area occupied by the diffusers gets small in comparison with the tidal path during the above mentioned period with velocities below 0.25 m/s. This insight could not be obtained without a hydraulic model.

Both effects led to an appreciable reduction in the total length of pipeline, required to transport the effluent to the diffusers.

3.2 Selective withdrawal at the intakes

As table 1 indicates, a potential disadvantage of the diffuser design is the interaction between existing low-mixing outlet and the open channels of the projected intakes. This could be avoided by providing both existing and new intakes with one skimmer wall (see Figure 8).

Other considerations with respect to this alternative were that in principle direct recirculation could be eliminated fully. Further it was expected that, due to the increase heat exchange to the atmosphere during the first hours after discharge, the increase in background temperatures might be smaller than in case of the diffuser outlet. A specific local aspect was the fact that a skimmer wall would provide a better flowcondition at the sharp contraction by the remnant of an old dike protruding into the Western Scheldt between existing intake and outlet.

A definite disadvantage of this design would be the occurence of high temperatures at the surface around the outfall site, both with respect to ecological conditions and the fog problem. Finally the unstable foreland at the projected intake location made the construction of a skimmer wall very complicated and probably costly.

The design of the skimmer wall was based on theoretical and experimental data on initial depth at the outlet and on selective withdrawal in stagnant ambient water [3, 4]. The initial depth of the heated upper layer near the outlet could be approximated by

$$\frac{h_1}{h_o} = F_o^{2/3} \qquad (F_o < 0,75)$$

in which h_o = depth outlet channel
h_1 = depth upper layer
F_o = densimetric Froude number of the outlet

Determining the dimensions of the skimmer wall the assumption was made that during the transport of the effluent from outlet to intake turbulence would cause a deepening of the heated layer, not affecting the temperature of the lower layer (see Figure 9^a). Taking this into account the skimmer wall design was made using the stagnant water formula for complete selective withdrawal [4]:

$$\left(\frac{Q}{B}\right)^2 = 0,2 \frac{\Delta\rho}{\rho} g h_r^3 \quad \left(\frac{h_r}{D} > 1,5\right)$$

in which Q = intake flowrate
B = width of the skimmer wall
h_r = thickness cold lower layer
$\Delta\rho$ = density difference between the two layers

The experiments, however, showed turbulence to cause a considerable mixing of the heated effluent over the whole depth (see Figure 9^b), invalidating the assumption of no effect upon the lower-layer temperatures.
The skimmer wall functioned still with respect to the remaining density stratification, but the temperature increase in the lower layer caused unavoidably some recirculation. The high mixing-rate must be due to local effects, possibly a secondary flow in the contraction around the above mentioned dike. Again a hydraulic model was necessary to show these effects.

3.3 Optimization of the multiport-diffusers

The final evaluation of high- and low mixing design, presented in table 1, turned the balance in favor of the multiport-diffuser system.
Since the 3-D and cross-current effects between outlet and intake were so clearly demonstrated, an attempt was now made for further economization of this system by staggering the diffuser part of the consecutive pipes and reducing the maximum distance the longest diffusers reached into the estuary. Figure 10 shows the final result of this process.
The spacing between the diffuser-pipes was slightly increased as compared with the minimum distance found in this first tests: 100 m proved to be necessary for reasons of construction. Since the cost aspect was involved in this part of the investigation some allowance was made for higher temperatures. The temperature elevations were still acceptable in the final design, with diffuser lengths of 150 m, staggered over a total width of 250 m normal to the flowdirection. However further reduction showed a rapid increase, indicating that the limits of the dilution capacity were reached.
The total length of diffuser pipe was in this way reduced from 2000 m to 800 m, in addition to the reduction of transport line achieved in the first part of the investigation.

4 Mathematical computations

The effect of return heat was computed with the following purpose:

- to get an estimate of the overall increase of intake temperatures, determining the total recirculation
- to estimate the maximum absolute temperatures in the outlet in the summer, with regard to the criteria posed

- to provide insight into the dependancy between increase of background temperature-distribution and initial mixing. This question was related to the choice between a high- and low-mixing design.

In the model the Western Scheldt is schematized into an infinitely long, rectangular channel, its depth corresponding with the average prototype depth.
As already stated in par. 2 the actual computation of the excess temperature at a given location is a simple integration of the contributions of all previous heat discharges towards that location. The assumption is made that subsequent releases into which the continuous discharge of heat is divided do not influence each other, for instance due to density effects. This assumption was acceptable in view of the fact that in the actual computations the discharge was stopped during the last half tidal cycle of the integration process.
A brief outline of the mathematics is given below. For a more detailed analysis one is referred to existing literature [5, 6].
The phenomena to be taken into account are:

- diffusion, the concentration at time t in location (x,y) due to a release of a mass M at t = 0 in (0.0) (see Figure 4) is in first approximation described by the equation:

$$c(x,y,t) = c(0,0,t)e^{-\pi \frac{b}{a} \{ x^2 + (\frac{a}{b})^2 y^2 \} \frac{c(0,0,t)}{M}} \qquad (1)$$

in which:

$$\frac{c(0,0,t)}{M} = \frac{1}{\pi\sigma^2} \cdot \frac{a^2 + b^2}{2ab} \qquad (2)$$

From the instantaneous release experiments, executed in the estuary, the value of a/b and the variation of σ^2 (σ = standard deviation of the concentration) with time were determined. Again the assumption is made that this information holds for all subsequent releases from which the actual discharge consists.

- advection, the centroid of the dispersing plume (x_c,y_c) follows a prescribed path, as in the present case the schematized tidal movement:

$$x_c(t) = x_c(0) \cos \frac{2\pi t}{T}$$

$$y_c(t) = y_c(0) \qquad \text{(drift currents in y-direction neglected)}$$

- heat exchange to the atmosphere, introduced into the computation by means of a decay factor k:

$$c(t) = c(0)\ e^{-kt} \quad \text{and} \quad k = \frac{K_E}{\rho c_w h}$$

in which : K_E = heat-exchange coefficient, based on the concept of equilibrium temperature

h = characteristic depth

ρ = density sea water

c_w = specific heat

The actual heat dissipation to the atmosphere is fluctuating daily. The coefficient used in the computation is an average value, but still a function of the meteorological condition in the region.

Summarizing, the concentration at time t in (x,y) of a fixed coordinate system is given:

$$c(x,y,t) = \frac{1}{h} \int_0^t \frac{Q(t-\tau)}{\pi\sigma^2} \frac{a^2+b^2}{2ab}\ e^{-(\xi^2/a^2 + \eta^2/b^2)\frac{a^2+b^2}{\sigma^2} - k\tau}\ d\tau$$

in which: $\frac{Q\ d\tau}{h} = M$

and $\left.\begin{array}{l} \xi = x - x_c \\ \eta = y - y_c \end{array}\right\}$ see Figure 4

In the actual computations a continuous heat discharge Q = 2500 Mcal/s was maintained from t = 0 until $t - \frac{1}{2}T$, if t is the time at which the temperature elevations are computed and T = tidal period. The discharge conditions were simulated according to the hydraulic-model results after initial dilution. The discharge was stopped during the last half tidal cycle in order to seperate the return heat completely from the direct-recirculation effects, determined in the hydraulic model. Computations were thus executed for a number of tidal stages.

An example of the computed isotherm patterns at various tidal stages is presented in Figure 11. Each picture gives the temperature elevations due to the continuous heat discharge up to half a tidal period from the time of the picture. Comparison of the temperature elevation at LWS for a multiport-diffuser outlet and an open outlet did not show appreciable differences, which fact contributed to the previously mentioned choice of the diffuser-system.

Further computations dealt with the sensitivity of the intake temperatures for changes in heat-exchange coefficient and for additional advective components in x or y direction.
For the execution of the dye-tracer field survey and the computations the assistance of the Dutch Public Works department is gratefully acknowledged.

5 Combining results of hydraulic and mathematical models. Conclusions

From the computations, presented in the previous section, the contribution of return heat, discharged more than 1/2 T before, to the time-varying intake temperature can be derived. As is indicated in Figure 12 other components may be, depending on the tidal stage:

- the direct recirculation
- the return heat, discharged less than 1/2 T before, or a combination of both.

The direct recirculation follows from the hydraulic model tests. Determining the effect of return heat, discharged during the last half tidal period, requires the rate of temperature reduction by ambient turbulence to be known. This was derived from the hydraulic-model test results. Heat exchange to the atmosphere was neglected in this computation.
From Figure 12 it can be concluded that the contribution of return heat to the total recirculation is large as compared to the direct recirculation. The advantage of limiting the direct recirculation is the peak temperatures, which govern the instantaneous level of heat disposal and electricity production, are avoided. The fluctuation of the ambient temperature elevation is limited to 1.0 ^{o}C. It is also clear from this Figure that further optimization of the design with regard to temperature elevation is unrealistic.
The Borssele study may provide a good example of a combination of hydraulic- and mathematical models in the solution of a design problem.
In designing the outfall and intake the hydraulic model turned out to be an indispensable tool, because of the insight it provided into:

- bottom effects
- shoreline effects
- turbulence effects
- 3-D effects and
- cross-current effects

Due to this insight a considerable reduction of construction costs could be achieved without impairing the quality of the design with respect to recirculation.

The mathematical-model computations showed the considerable effect of return heat upon intake temperatures. By doing so they provided an additional criterium for the point where further improvement of the outfall design stopped to make sense.

HL

References

[1] Dunn, W.E., Policastro, A.J. and Paddock, R.A. Water Resources Research Programme. Surface Thermal Plumes: Evaluation of mathematical models for the near- and complete field. Argonne National Laboratory, Report ANL/WR-75-3, May 1975.

[2] Jirka, G.H. and Harleman, D.R.F. The Mechanics of submerged diffusers for buoyant discharges in shallow water. M.I.T., Dep. of Civ. Eng., Ralph M. Parsons Laboratory, Report 169, 1973.

[3] Craya, A. Recherches theoriques sur l'ecoulement de couches superposées de fluides de densités différentes. La Houille Blanche, No. 4, 1949.

[4] Selective withdrawal, two- and three-dimensional investigations, Delft Hydraulics Laboratory, reports M 1204 and M 1209, 1973 (in Dutch).

[5] Schönfeld, J.C. Turbulent diffusion, disposal from a source at sea (in Dutch). Public Works Department, Report MFA 6411, 1964.

[6] Abraham, G. and van Dam, G.C. On the predictability of waste concentrations. Paper E1, F.A.O. Techn. Conf. on Marine Pollution and its effects on living resources and fishing, Rome 1970.

HL

Design criteria	preliminary evaluation		final evaluation	
	multiport diffuser	selective withdrawal	multiport diffuser	selective withdrawal
direct recirculation	-	+	∿	∿
interaction circuits	-	+	∿	∿
construction (costs)	+	-	+	-
fog prevention	+	-	+	-
total evaluation	?	?	+	-

+	advantaged	∿	no preference
-	disadvantaged		

Table 1. Preliminary and final evaluation high- and low-mixing design.

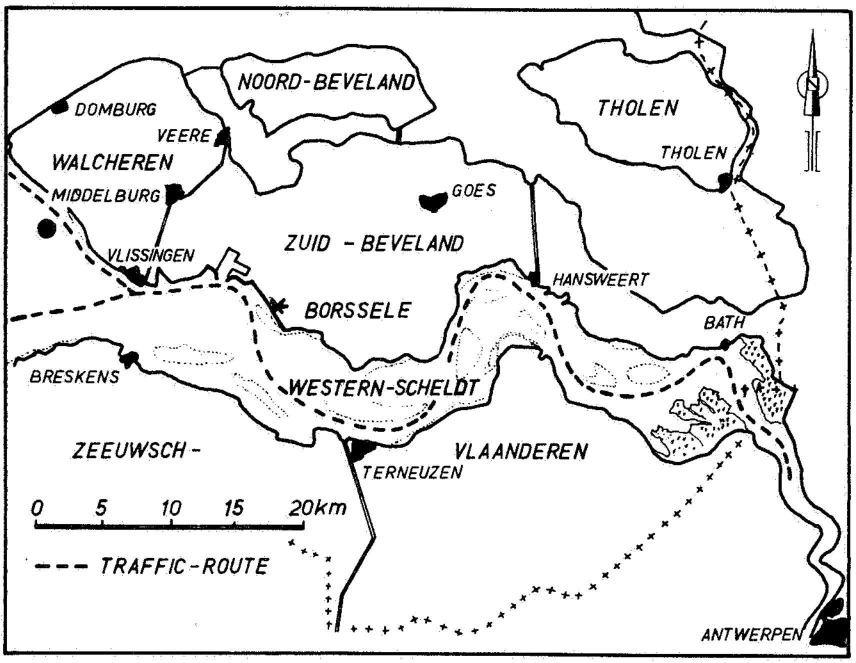


FIG. 1 THE WESTERN-SCHELDT ESTUARY

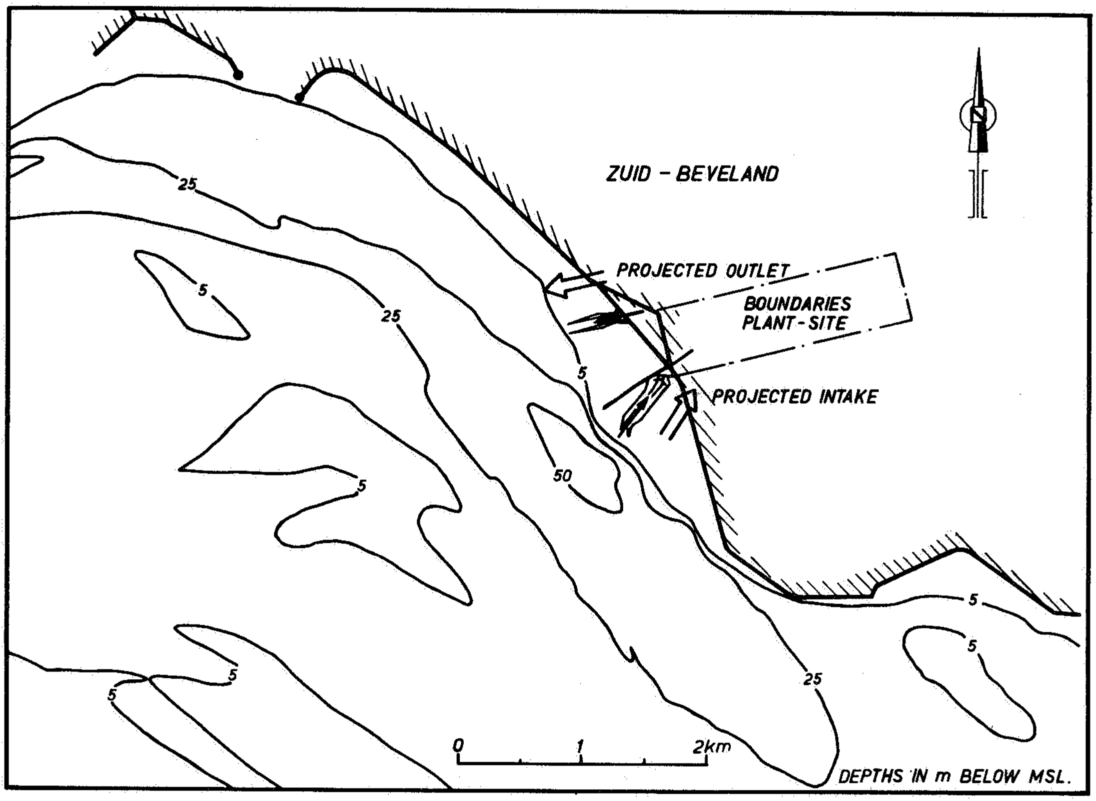


FIG.2 SITE SITUATION

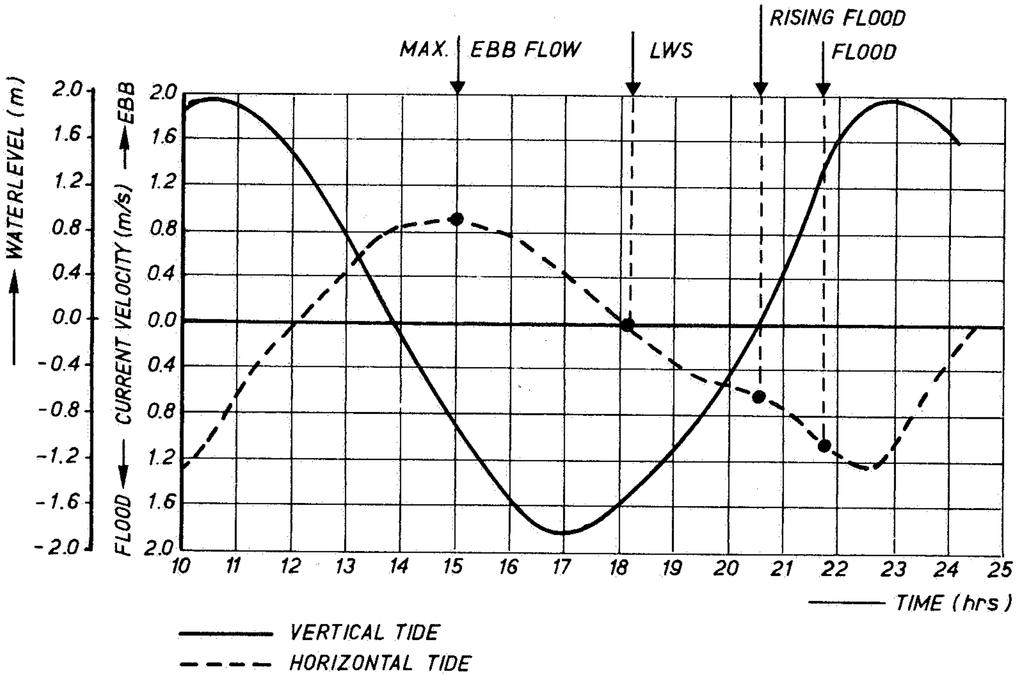


FIG. 3 HORIZONTAL AND VERTICAL TIDAL MOVEMENT IN THE WESTERN-SCHELDT

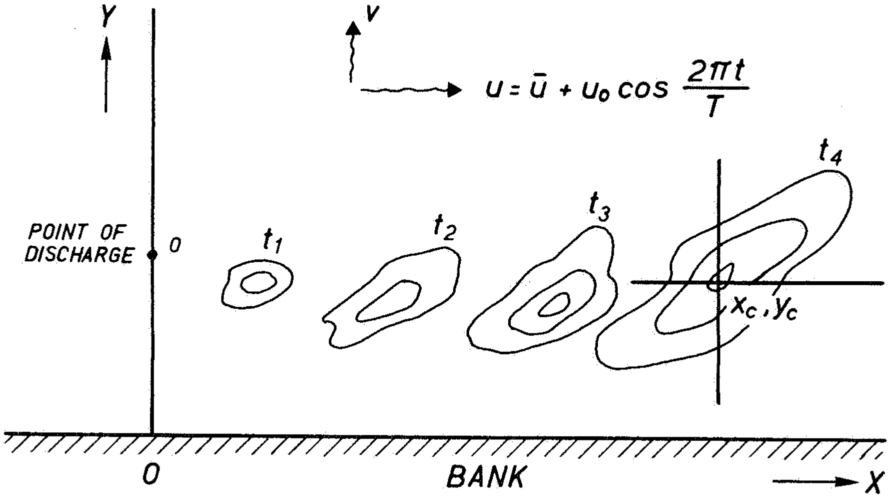


FIG. 4 DISPERSIVE-AND ADVECTIVE EFFECTS ON AN INSTANTANEOUS RELEASE

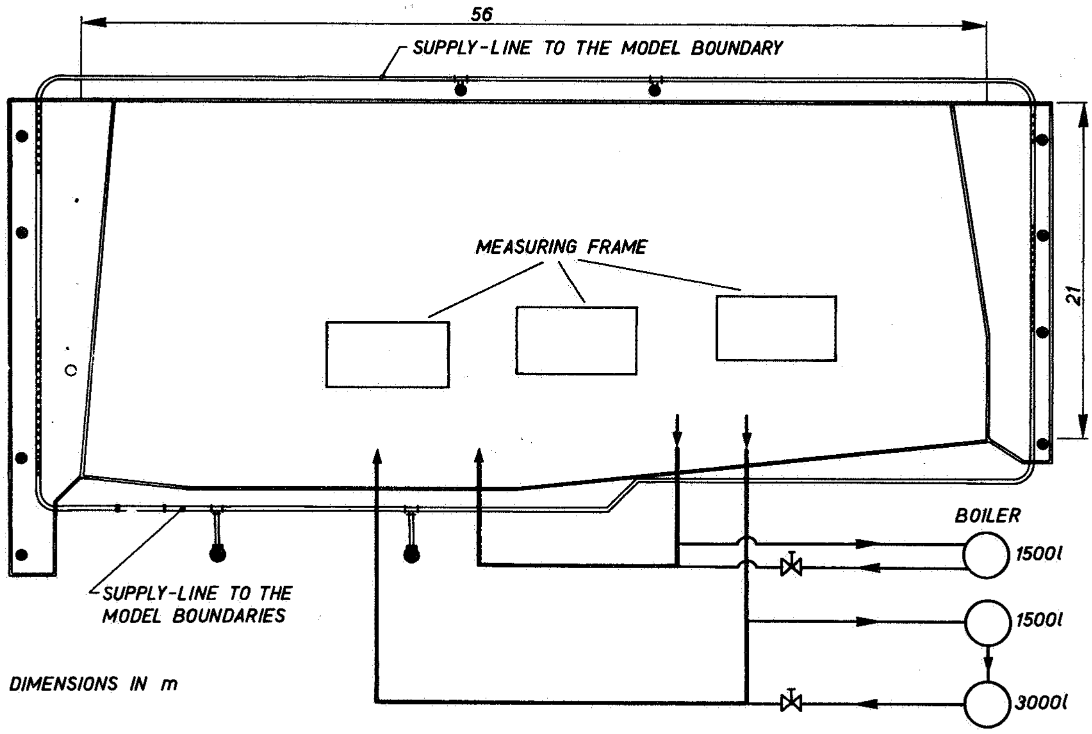


FIG. 5 LAY OUT OF THE HYDRAULIC MODEL

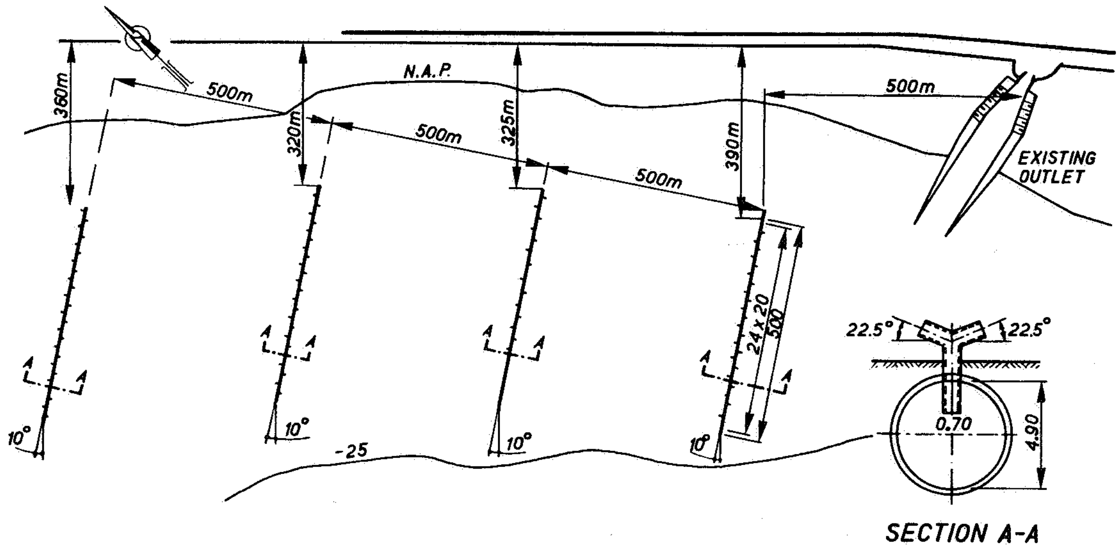


FIG. 6 FIRST DESIGN OF MULTIPORT-DIFFUSER SYSTEM

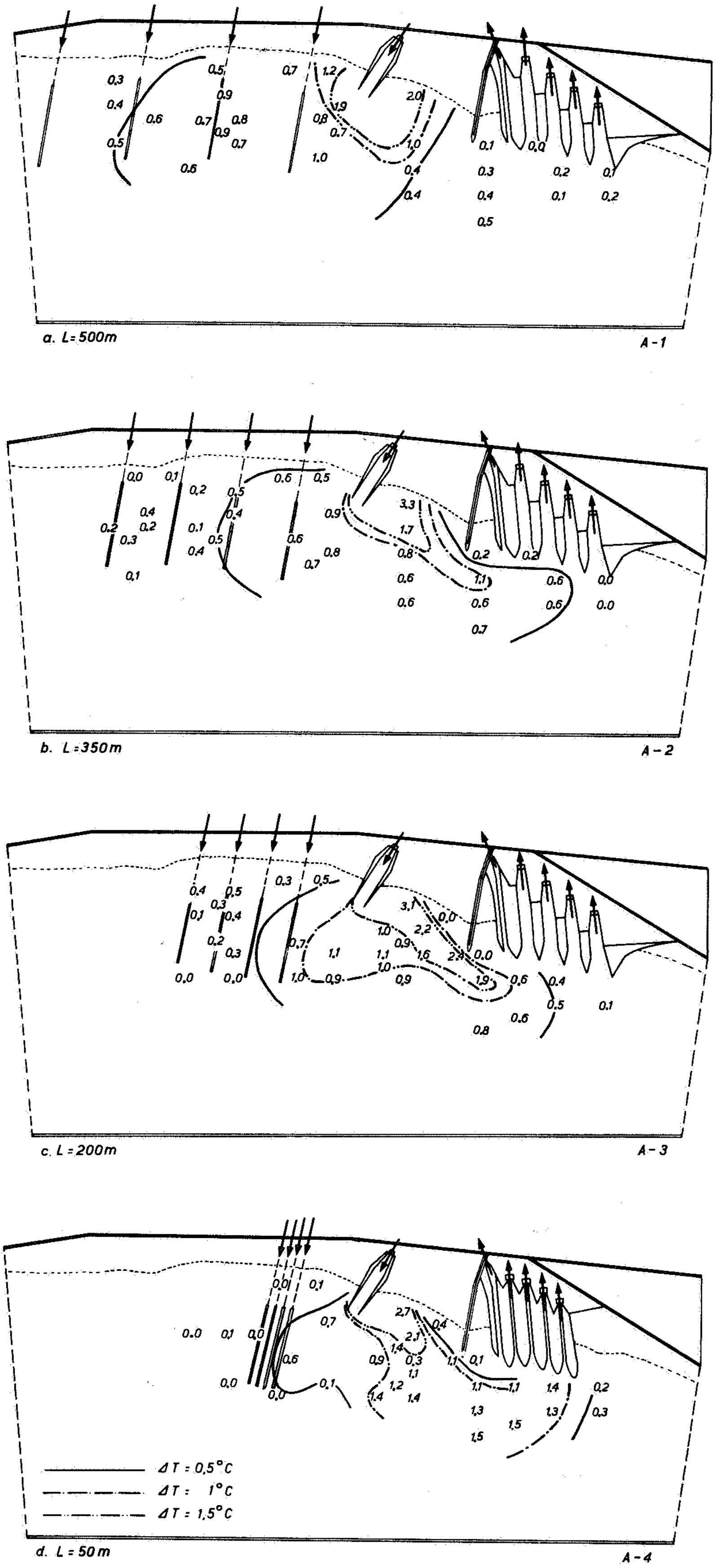


FIG. 7 RESULTS DIFFUSER TEST L.W.S.

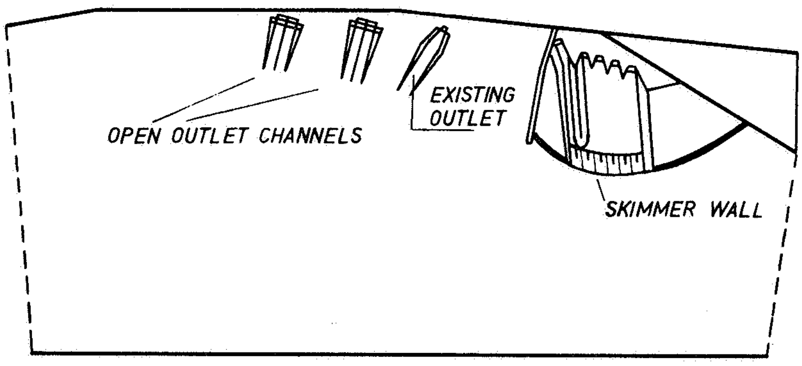
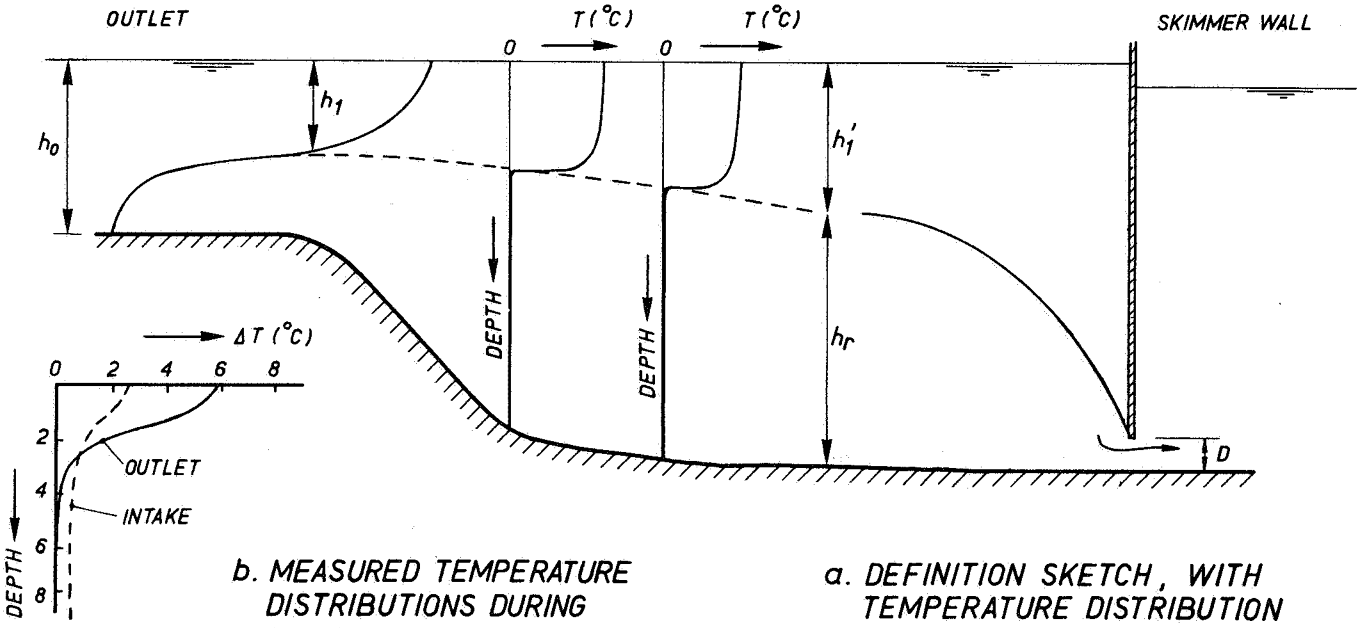


FIG. 8 LOW-MIXING DESIGN WITH SKIMMER WALL



b. MEASURED TEMPERATURE DISTRIBUTIONS DURING FLOOD FLOW

a. DEFINITION SKETCH, WITH TEMPERATURE DISTRIBUTION ASSUMED IN DESIGN

FIG. 9 VERTICAL DIFFUSION BETWEEN OUTLET AND INTAKE

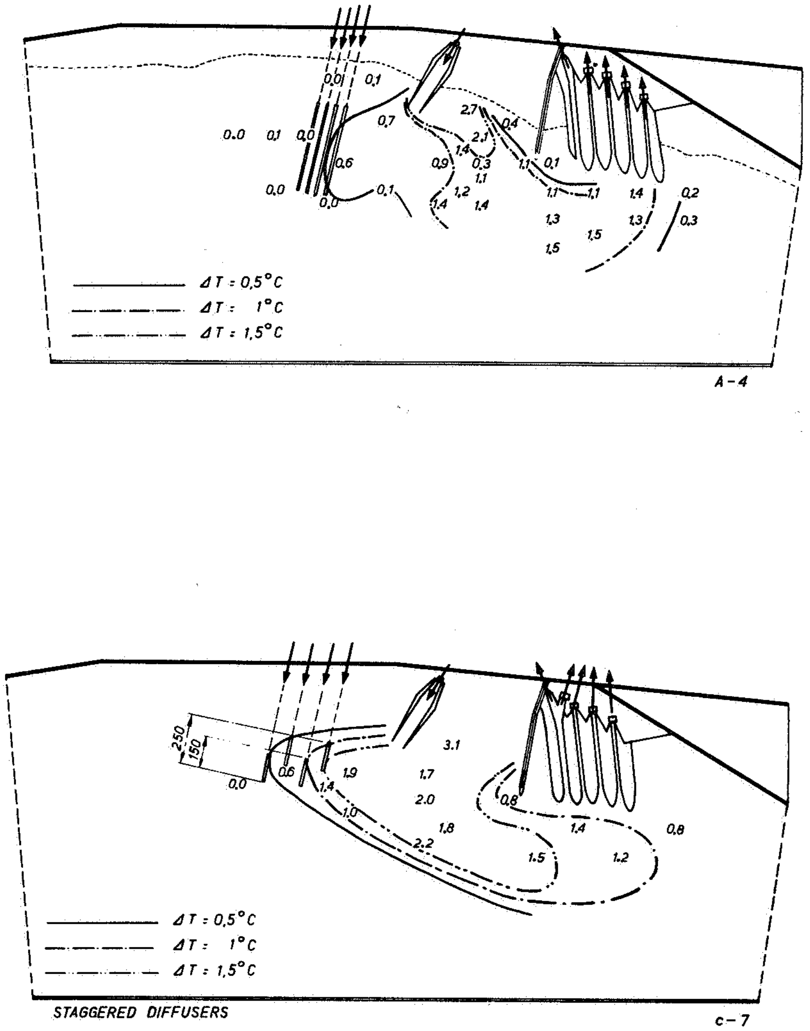


FIG. 10 FINAL OPTIMIZATION OF DIFFUSER LAY-OUT

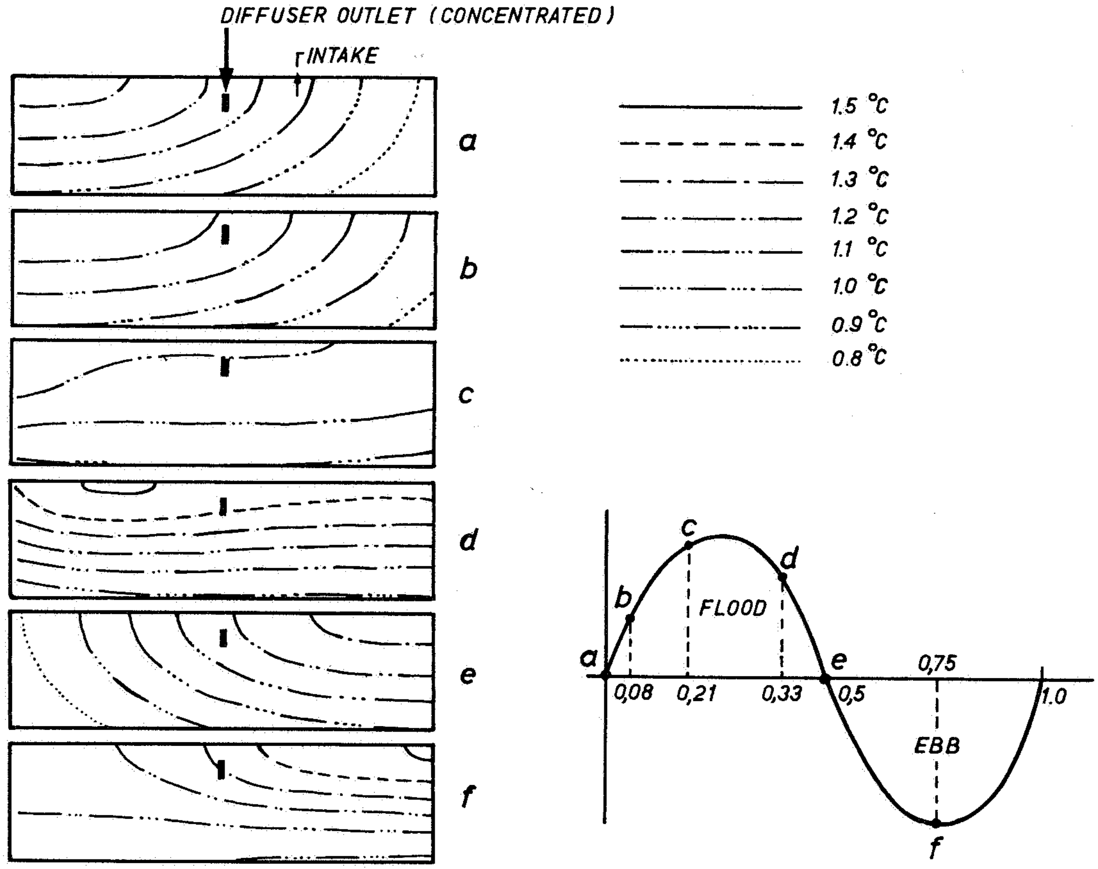


FIG. 11 RESULTS OF THE IDEALIZED MATHEMATICAL COMPUTATIONS

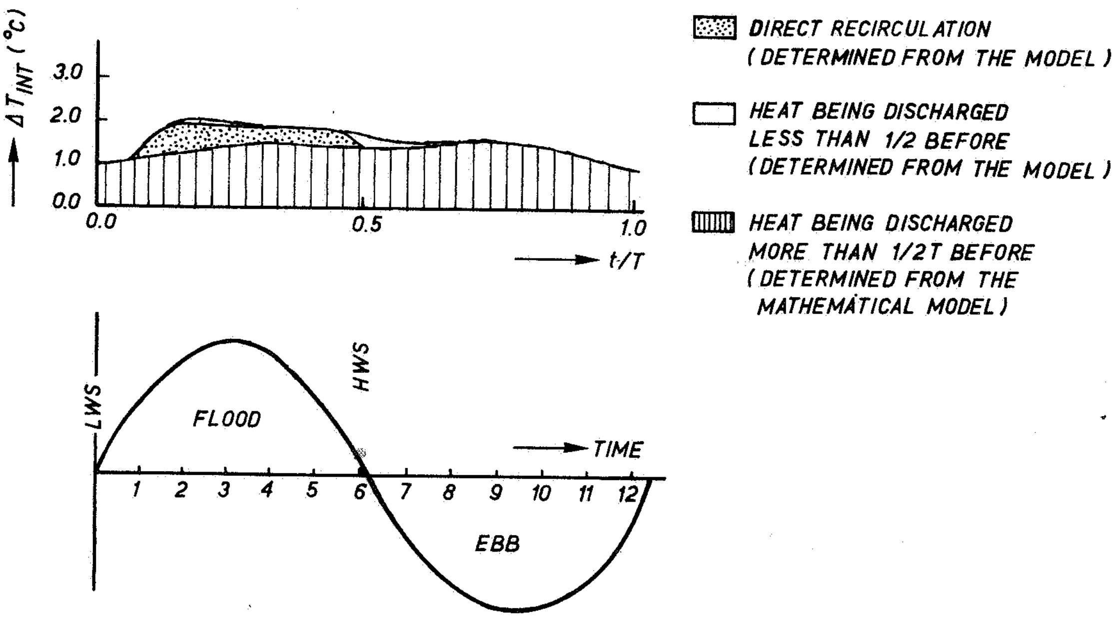


FIG. 12 TOTAL TEMPERATURE INCREASE IN THE INTAKE