Journal of Hydroscience and Hydraulic Engineering
Vol. 12, No. 2, November, 1994, 51-67

REFINED PREDICTIVE MODEL FOR THERMAL DIFFUSION EXTENT NEAR AN OUTLET

BY

Akira Wada

Tokai University, School of Marine Science and Technology, Orido, Shimizu, Shizuoka, Japan

SYNOPSIS

The object of this report is to evolve an analysis by means of a three-dimensional mathematical model so as to reproduce with a high accuracy the mixing phenomenon of a surface densimetric jet in the area near the outlet, and to promote the practical use of a technique for predicting the thermal diffusion in the high-temperature area near the outlet through a comparative study between the result of the analysis and the result of a hydraulic test.

It can be presumed that the present model is effective for numerical calculation of the surface densimetric jet because it shows a consistent agreement with experimental values near the outlet. By this, a more accurate prediction of the high temperature rise zone (3-5℃) with the discharge of warm water can be realized.

INTRODUCTION

It seems that the prediction of thermal diffusion in the area far away from the outlet by a surface discharge method has now reached an almost practical stage[1),2),3)] . This prediction technique uses a two-dimensional model which expresses exponentially the vertical distribution of flow velocities and water temperatures; and consists of equations considering a flow owing to the discharge of cooling water, a current and turbulence in the sea area, a heat exchange between the atmosphere and the water surface, etc. The model is intended to predict the diffusion in the low-temperature rise area (1~ 3℃). With an increase in the discharge rate of warm water due to an increase in the capacity of power plants in recent years, the high water-temperature rise region near the outlet is expanding more than ever, making it necessary to carry out more highly accurate diffusion prediction with a high temperature-rise region as a subject area.

The concept of entrainment coefficients is introduced in the development of the conventional densimetric jet theory. In addition, an analytical method based on the assumption that the velocity distribution and density distribution in the cross section of the jet axis are similar is generally adopted. In this theory;, however, the boundary between the water surface and the water bottom cannot be taken into consideration, and an analytical solution can only be applied up to a process in which the densimetric jet reaches the water surface or bottom.

The object of this report is to evolve an analysis by means of a three-dimensional mathematical model so as to reproduce with a high accuracy the mixing phenomenon of a surface densimetric jet in the area near the outlet; and to promote the practical use of a technique for predicting the thermal diffusion in the high-temperature area near the outlet through a comparative study between the result of analysis and the result of a hydraulic test.

PREDICTIVE METHOD FOR THERMAL DIFFUSION

Thermal Diffusion Process

The principal factors that govern the diffusion and cooling process of discharged warm water include mixed dilution owing to the entrainment of surrounding cooler water in the area near the outlet, advection and dispersion of the coastal current in the far field, eddy diffusion owing to the turbulence of surrounding water, heat exchange between the sea surface and the atmosphere, etc. (See Fig.1) These dominant factors are complicated as they vary with the quantity of discharged warm water, the characteristics of flow in the front sea area and the method of discharging the warm water. In particular, the discharge method is normally classified broadly into a surface discharge method and a submerged discharge method. In the surface discharge method, which releases the warm water into the vicinity of the water surface at a relatively low speed, the eddy diffusion, the advection and dispersion of the coastal current and the heat exchange with the atmosphere are the principal factors; in the discharge method that releases the warm water at a relatively high speed by submerging it in the water, the entrainment from surrounding water and the advective action of the coastal current are the principal factors.

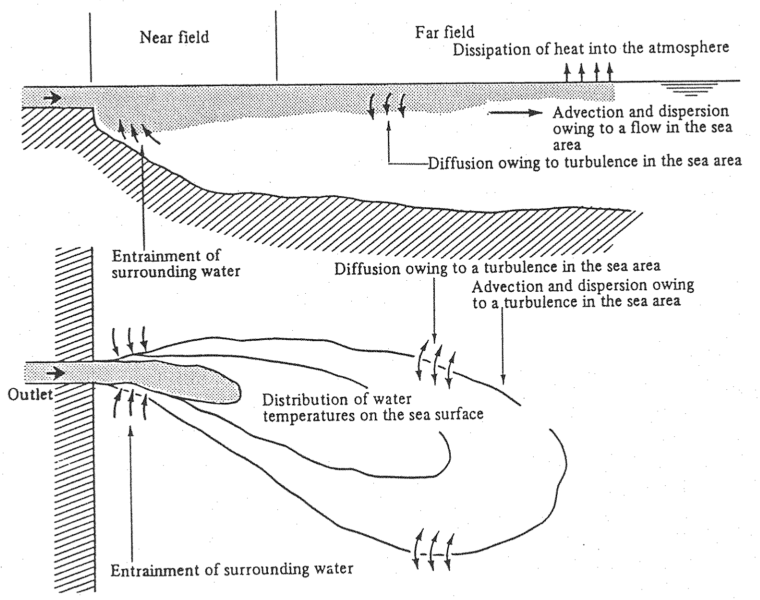

Fig.1 Process of decrease in the temperature of surface-discharged warm water.

Diffusion in the Far Field

As a method for the surface discharge of warm water in Japan, the low-velocity discharge whose densimetric Froude number at the outlet is about 2 is generally used. Because of this, the decrease of water temperature in the area near the outlet is small and the discharged warm water spreads over a wide area and a water temperature distribution is governed by the phenomena in the far field, that is, by the current or turbulence in the sea.

At present, as a technique for predicting the diffusion of discharged warm water in the far field, the simulation model based on the mathematical model is widely used. From a comparative study with the results of survey at many points, the appropriateness of this prediction technique has been proved and it is considered to have reached an almost practical stage. This technique aims to discern the horizontal spread of discharged warm water, and therefore the equation of fluid motion, the equation of continuity and the equation of diffusion which have been integrated vertically up to the interface containing the discharged warm water from the sea surface are used. However, the amount of the water temperature rise and the discharge flow velocity are not uniform in the vertical direction and are distributed exponentially.

DIFFUSION IN THE AREA NEAR THE OUTLET [4]

With an increase in the discharge rate of cooling water due to an increase in the capacity of power plants, the high-temperature region near the outlet is expanding more than ever, making it necessary to carry out a more highly accurate diffusion prediction.

The three-dimensional mathematical model developed in this report relates to the following two methods.

Mean Flow Model (3D Model)

It is known that when the warm water is released into the sea by the surface discharge method its behavior near the outlet belongs to the category of a surface densimetric jet.

The basic equations that govern the diffusion of a surface densimetric jet in the sea consists of the equations of conservation of momentum, mass, and thermal energy which spatially occupy three dimensions. In deducing the basic equation, the following hypotheses are established:

1) The densimetric jet is in a completely turbulent state.
2) Densimetric variation is included in the gravity term only. (Boussinesque's approximation holds.)
3) Density stratification does not exist. The equations that govern the motion of a densimetric jet may be expressed as follows:

A) Equation of the conservation of momentum

$$\frac{\partial u}{\partial t} + \frac{\partial u^2}{\partial x} + \frac{\partial uv}{\partial y} + \frac{\partial uw}{\partial z} = -\frac{1}{\rho_0}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}(Ax\frac{\partial u}{\partial x}) + \frac{\partial}{\partial y}(Ay\frac{\partial u}{\partial y}) + \frac{\partial}{\partial z}(Az\frac{\partial u}{\partial z}) \qquad (1)$$

$$\frac{\partial v}{\partial t} + \frac{\partial uv}{\partial x} + \frac{\partial v^2}{\partial y} + \frac{\partial vw}{\partial z} = -\frac{1}{\rho_0}\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}(Ax\frac{\partial v}{\partial x}) + \frac{\partial}{\partial y}(Ay\frac{\partial v}{\partial y}) + \frac{\partial}{\partial z}(Az\frac{\partial v}{\partial z}) \qquad (2)$$

$$\frac{\partial w}{\partial t} + \frac{\partial uw}{\partial x} + \frac{\partial vw}{\partial y} + \frac{\partial w^2}{\partial z} = -\frac{1}{\rho_0}\frac{\partial p}{\partial z} - g\frac{\Delta\rho}{\rho_0} + \frac{\partial}{\partial x}(Ax\frac{\partial w}{\partial x}) + \frac{\partial}{\partial y}(Ay\frac{\partial w}{\partial y}) + \frac{\partial}{\partial z}(Az\frac{\partial w}{\partial z}) \qquad (3)$$

B) Equation of continuity

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \qquad (4)$$

where,

x, y, z: Rectangular coordinate axes
u, v, w: Velocity components in the x, y and z directions
p : Pressure
ρ_0 : Reference density of discharged water
$\Delta\rho$: $\rho_\infty - \rho$,
ρ_∞: Density of surrounding water
ρ : Density at an arbitrary point within the plume
Ax, Av, Az: Coefficients of eddy viscosity in the x, y and z directions

C) Equation of the conservation of heat energy

At this time, the buoyancy effect of fluid can be simulated by a simultaneous solution to the heat conservation equation concerning water temperature and the dynamic equation of a fluid. The heat conservation equation can be expressed as follows:

$$\frac{\partial T}{\partial t} + u\frac{\partial T}{\partial x} + v\frac{\partial T}{\partial y} + w\frac{\partial T}{\partial z} = \frac{\partial}{\partial x}(Kx\frac{\partial T}{\partial x}) + \frac{\partial}{\partial y}(Ky\frac{\partial T}{\partial y}) + \frac{\partial}{\partial z}(Kz\frac{\partial T}{\partial z}) \quad (5)$$

where,

T : Water temperature,
Kx, Ky, Kz : Coefficients of diffusion in the x, y and z axis directions.

The density of sea water ρ is related to water temperature and chlorinity.

The eddy viscosity coefficient A is expressed by the equation shown below, using the mixed length model in which the half breadth $b_{1/2}$ of the plume is fundamentally assumed to be a mixed length.

$$A = C \cdot b_{1/2} \cdot W_{max} \quad (6)$$

where, C: constant; W_{max} : velocity in the center of the plume.

Another method is to determine the coefficient as the function of the Richardson number. With

$$A_z = A_0 + A_1 \cdot \exp(-mR_i) \quad (7)$$

the vertical viscosity coefficient inside the plume is determined for each cell, where A_0, A_1, m: constants; R_i : Richardson number for each cell.

The eddy diffusion coefficient K is expressed as K = A/Sc according to the turbulent Schmidt number Sc. In this calculation, however, it was assumed as Sc = 0.7 on the basis of existing research results.

As a solution to the basic equation, MAC's algorithm was used. Equations (1)-(4) are expressed by the following system:

$$\partial u/\partial t = -\frac{1}{\rho}\nabla p + F(u), \quad \nabla u = 0$$

F(u): term including a non-linear term and an eddy viscosity term.

With Δt as time interval, $\tilde{u}$ is obtained from

$$\tilde{u} = u^n - \frac{\Delta t}{\rho}\nabla p^n + \Delta t \cdot F(u^n),$$

and with

$$u^{n+1} = u^{n} - \frac{\Delta t}{\rho}\nabla p^{n+1} + \Delta t \cdot F(u^{n})$$

a difference between both equations is taken. Furthermore, considering divergence,

$$\left(\rho u^{n+1}\right) = 0$$

then

$$\nabla\left(\rho \tilde{u}^{n+1}\right) = \Delta t \cdot \nabla^{2} p', \qquad \left(p' = p^{n+1} - p^{n}\right)$$

is obtained. If p' is obtained from Poisson's equation, then

$$p^{n+1} = p^{n} + p'$$

The velocity components u, v and w in this analytical technique are defined on the surface of the cell, while the pressure and density are defined in the center position of the cell.

Quasi Three-Dimensional Model

This model is a method for calculating the vertical velocity from the equation (4) related to the preservation of flow without solving the equation of motion (3) in the vertical direction. Since this method does not require the repetitive calculation of pressure, reduction of calculation time can be expected. The pressure can be calculated by assuming hydrostatic pressure considering a rise in water level.

The pressure can be calculated by the following equation:

$$p = -\int_{\zeta}^{z} g\rho dz \tag{8}$$

The following equation is used as sea surface conditions:

$$\frac{\partial\zeta}{\partial t} + U_s\frac{\partial\zeta}{\partial x} + V_s\frac{\partial\zeta}{\partial y} - W_s = 0 \tag{9}$$

where,

ζ : Water level,

U_s, V_s, W_s: Flow velocities on the water surface.

Finally, the quasi three-dimensional model is given by the simultaneous equations (1), (2), (4), (8) and (9).

Turbulence Model

In the average flow model, it is expected that the values of turbulent viscosity and the turbulent diffusion coefficient will have a large effect on the whole distribution of flow velocities and water temperatures. Here, using the turbulence model whose concept of turbulent viscosity is made more general for the surface densimetric jet, its applicability is discussed.

In this model, to express the turbulent viscosity, k(turbulence energy) and ε (energy dissipation rate) are used as turbulence characteristic quantities, and these two transport equations are added to the equations of the mean flow model. Accordingly, advective and diffusive actions in the turbulence viscosity are expressed, and the model becomes higher in order than the mean flow model. The k equation is given as follows:

$$\frac{\partial k}{\partial t} + \frac{\partial uk}{\partial x} + \frac{\partial vk}{\partial y} + \frac{\partial wk}{\partial z}$$

$$= \frac{\partial}{\partial t}(\nu_x \frac{\partial k}{\partial x}) + \frac{\partial}{\partial y}(\nu_y \frac{\partial k}{\partial y}) + \frac{\partial}{\partial z}[(\frac{\nu_t}{\sigma_k} + \nu)\frac{\partial k}{\partial z}] + P\nu_0 + Gw - \epsilon \quad (10)$$

The ε equation model varies with the method of expressing the buoyancy effect. According to Esposito and Hauguel[5], the term that includes temperature variation is given by the following equations.

$$\frac{\partial \epsilon}{\partial t} + \frac{\partial u\epsilon}{\partial x} + \frac{\partial v\epsilon}{\partial y} + \frac{\partial w\epsilon}{\partial z} = \frac{\partial}{\partial x}(\nu_x \frac{\partial k}{\partial x}) + \frac{\partial}{\partial y}(\nu_y \frac{\partial k}{\partial y})$$

$$+ \frac{\partial}{\partial z}[(\frac{\nu_t}{\sigma_\epsilon} + \nu)\frac{\partial \epsilon}{\partial z}] + \frac{\epsilon}{k}[C_{1\epsilon} P_{\nu 0} - C_{2\epsilon}\epsilon + C_{3\epsilon} G_w] \quad (11)$$

$$P_{\nu o} = \nu_t [(\frac{\partial u}{\partial z})^2 + (\frac{\partial v}{\partial z})^2], \qquad Gw = \frac{g}{\rho} k_t \frac{\partial \rho}{\partial z}$$

The eddy viscosity coefficient and eddy diffusion coefficient are given by the following Prandtl-Kolmogorov equation, using dimensional analysis:

$$\nu_t = C_\mu \cdot \frac{k^2}{\varepsilon} \quad (12)$$

Accordingly, the following relationship holds:

$$\nu_x = \nu_y = Kx = Ky = \text{Const.}, \quad \nu_z = K_z = \nu_t + \nu \quad (13)$$

Constants are used by the combination of values shown in Table 1.

Table 1 Constants

C_μ	σ_k	σ_ε	$C_{1\varepsilon}$	$C_{2\varepsilon}$	$C_{3\varepsilon}$
0.09	1.0	1.3	1.44	1.92	1 when $\frac{\partial \rho}{\partial z} > 0$, 0 when $\frac{\partial \rho}{\partial z} < 0$

1) Setting k and ε at the outlet

Since k and ε are calculated by solving the transport equation, the setting value in the upstream section, particularly in the outlet, has an effect on the turbulent viscosity coefficient. In the present analysis, these values were defined like equation (14), and coefficient α and ℓ were set empirically.

$$\left.\begin{array}{l} k_{in} = \alpha U^2_{in} \\ \varepsilon_{in} = k_{in}^{3/2}/l \end{array}\right\} \quad (14)$$

In this calculation, α =0.01; ℓ =1.5cm (outlet height).

When determining these values, preliminary study was conducted in several cases. As a result, from the set values of α and ℓ , it was recognized that some difference took place in the size of turbulent viscosity and in its maximum position as well. Since the basis of making correct selection of α and ℓ was not clear, it was decided to set α as an experimental value as seen in the existing research result and to set ℓ with outlet height as a typical length.

2) Condition at wall

In case of wall turbulence, treatment near the wall is very important. In the surface densimetric jet, it can be considered that the wall friction is essentially an important. In the present analysis, free-slip is assumed, and boundary conditions for k and ε were established as follows:

$$\frac{\partial k}{\partial n}=\frac{\partial \varepsilon}{\partial n}=0 \qquad (15)$$

3) Turbulent viscosity coefficient

The $k-\varepsilon$ model is based on isotropic turbulence. There is , however, a question in applying the model to a non-isotropic current like the surface densimetric jet as it is. Also, much of energy production is caused by the velocity shear in the vertical direction. What is important seems to be a turbulent viscosity coefficient in the vertical direction.

Furthermore, the current comes to spread in horizontal layer owing to the effect of buoyancy. It cannot, therefore, be concluded that the local difference of horizontal viscosity has much effect on the current. Then, in this analysis, the turbulent viscosity was obtained from equations (12) and (13).

4) Boundary conditions on the sea surface

Boundary conditions at velocity components k and ε and temperature T are as shown below in the case that the sea surface is fixed horizontally.

$$\frac{\partial u}{\partial z}=\frac{\partial v}{\partial z}=0, \quad \frac{\partial k}{\partial z}=\frac{\partial \varepsilon}{\partial z}=0, \quad w=0, \quad -c\rho K_z\frac{\partial T}{\partial z}=0$$

Results and Evaluations

In a stationary sea area with a uniform density field, analysis was conducted in two cases ($F_{io}=4.0$, 2.56) where the buoyancy of a densimetric jet was different in each case. The topography was assumed to be a linear coastal line. The following information was obtained through the comparison between analytical results and hydraulic test results (see Figs. 2, 3 and 4).

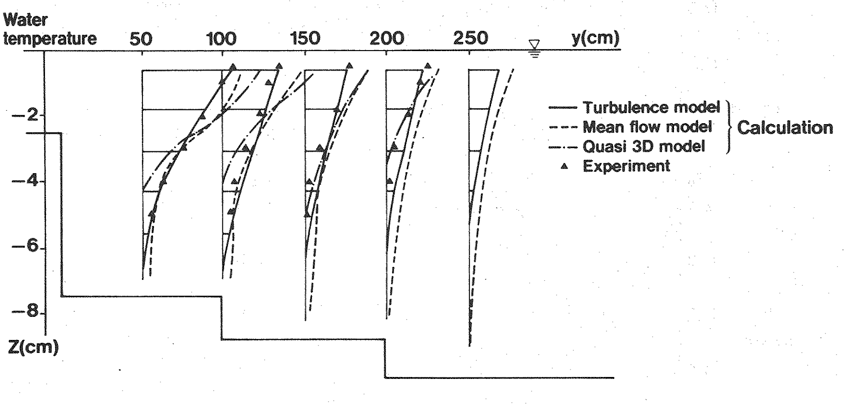

Fig.2 Vertical distribution of water temperatures in the center-axis cross section ($F_{io}=4.0$)

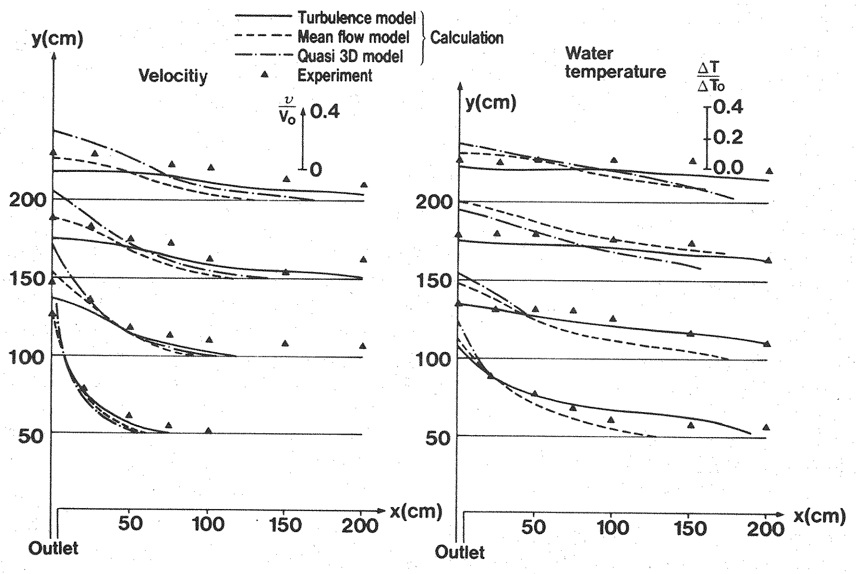

Fig.3 Horizontal distribution of velocities and water temperatures (F_{io}=4.0)

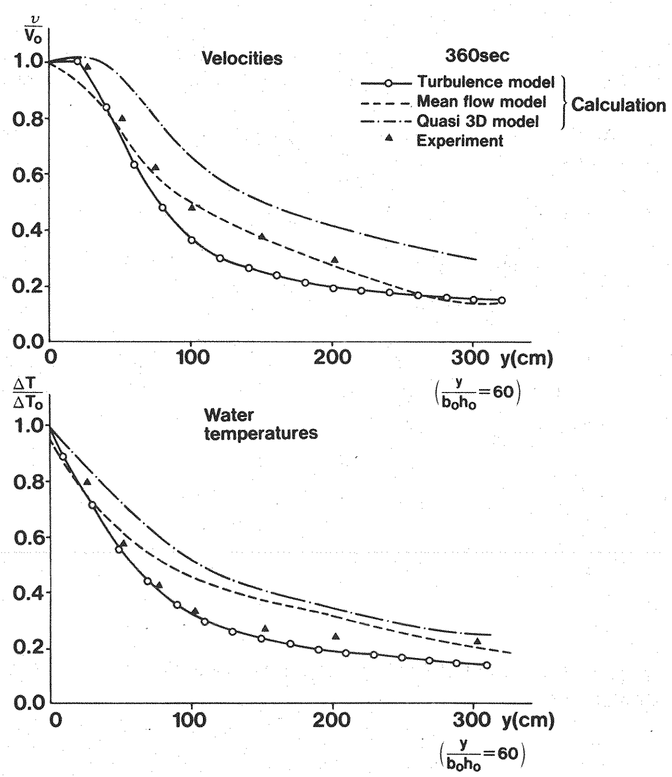

Fig.4 Decrease ratio of velocities and water temperatures in the center axis (F_{io}=4.0)

1) In mean flow model (3D model), the distribution of viscosity coefficients, particularly the values near the outlet have a large effect on the whole distribution of velocities and water temperatures. In this model, if an adequate distribution of viscosity coefficient is determined, the phenomena can be reproduced with a high accuracy. Methods based on equations (6) and (7) used here as a way of determining the coefficient of viscosity may be considered appropriate. As to judgment on the inside and outside of the plume and setting of field values, trial and error study will be needed.
2) On the other hand, in the quasi three-dimensional model, calculation errors are introduced in the buoyancy term, hence there is a doubt in its accuracy. As regards to economy of calculation cost, there are also such restrictions as calculation time interval, and it is therefore hard to judge that the model is a particularly credible.
3) It can be considered that the k-ε model is effective for numerical calculation of the surface densimetric jet because it shows a good agreement with experimental values near the outlet as compared with the mean flow model. Also, in making calculations, it is necessary to pay sufficient attention to the numerical difference method when estimating energy production term, etc. in the energy equation.
The reason is that the square of velocity gradient is included in the energy production term; depending on the way of taking grid intervals, therefore, there are some cases where correct evaluation cannot be accomplished.

ADDITIONAL REVIEW OF DIFFUSION PREDICTION MODEL

In calculations of the thermal discharges in three-dimensional space, study was conducted in accordance with the test scale using mean flow model (3D model), quasi three-dimensional model and k-ε model. As a result, it was found that the k-ε model exhibited a relatively high simulation property. In this chapter, therefore, fundamental checks are made to provide the actual-length scale, and the accuracy of discretization is examined using the k-ε model.

Effect of grid division

To examine the effect of the difference in grid division on flow and water-temperature distributions, tests were conducted in two cases as shown in Table 2.

Table 2

Calculating conditions (Test scale)	Grid division	
	Case 1	Case 2
Outlet breadth B = 10.0 cm Outlet height H = 2.5 cm Outlet flow rate Q = 250 cm^3/s Outlet velocity V = 10.0 cm/s	40× 32× 10 (Equal interval)	25× 33× 18 (Unequal interval)

In Case 1, the vertical division was coarse, so the turbulence energy distribution at the corner of the outlet was not sharp as compared with that in Case 2. Consequently, the turbulent viscosity in that part was large and the temperature distribution was hanging down. It is, therefore, evident that the division employed in Case 2 is needed in the vertical direction. (See Fig.5)

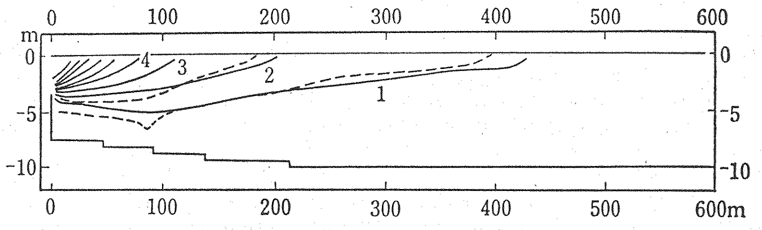

Fig.5 Vertical distribution of water temperature (— :Case 2, ····:Case 1).

Effect of the discretization of transport equations

The discretization of water-temperature transport equations was analyzed by changing it from the upstream difference of 1st-order accuracy to that of 3rd-order accuracy. As a result, the use of the 3rd-order accuracy upstream difference scheme showed a smaller horizontal spread of water temperatures near the outlet and could obtain results close to the experiment.

Study on the handling of water surface

1) In numerical calculation of surface jets, comparison was made between the case where a rise in water level was considered and the case where the water surface was fixed, and the handling of the water surface in this model was studied.

Using the model considering the rise in water level and the existing model fixing the water surface, the thickness of the outlet was assumed to be the same as the depth of water so that a remarkable ascending current can be produced due to a seabed friction.

Water-level rise at the front of the outlet derived from the model considering rise was ζ =1.72 cm. It was also learned that pressure increase was in correlation with $\Delta P=\rho\zeta$. Comparison of the pressure increase between the model considering rise in water level and the one with fixed water surface at the same depth (-0.5m) indicated that the fixed water surface model showed more increased pressure near the outlet. This was the result of excessive estimates of the interior fluid pressure by fixing the water surface. In the case of flow velocity comparison based on the sectional view of vertical direction, it should be noted that the flow can be transported further with the model considering rise in water level than with fixed water surface. However, when it came to the change of pressure increase by depth, both models, the one considering rise in water level and the one with fixed water surface, showed similar results, which proves that these two models provide consistency, and that the one with fixed surface water would present no problem except for near the outlet. Fig.6 shows decrease of pressure and flow velocity in the center axis, which illustrates the difference of these two models very clearly.

Also, considering that a rise in the water level is attributable to a friction on the seabed (no-slip condition), calculations were conducted under condition without friction on the seabed (free-slip condition). As a result, the downstream flow velocity showed little attenuation, and there was little rise in the water level ($\zeta \leq$ 0.5mm) as well.

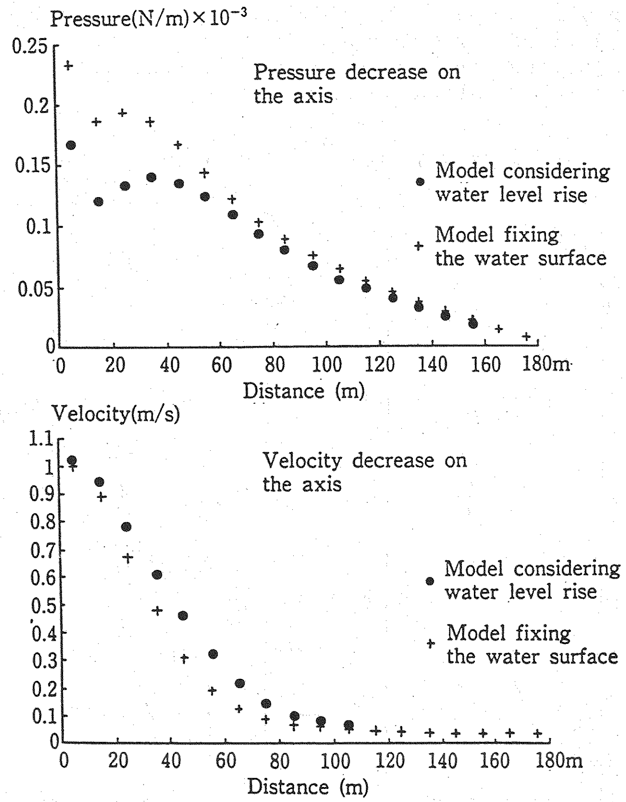

Fig.6 Variation of pressure and velocity in the plume axis.

2) To look into the cause of water level rise, the rise was computed using a model considering buoyancy and the results were compared to those of a homogeneous case but without buoyancy. Conditions of outfall were the same except for that of outfall temperature where T_{in}=30.0℃ and T_e=20.0℃.

Comparison of the pressure and velocity curves revealed that buoyancy did not prove to have much influence. Therefore, it was concluded that presence of buoyancy does not largely affect rise in water level and that buoyancy itself can be omitted in considering water level rise.

3) Results of a fundamental check as stated in 1) and 2) disclosed that, in case of the horizontal jet, the rise in water level is mainly caused by ascending current due to a seabed friction. Based on this finding, an analytical check corresponding to a model experiment with ordinary discharge of warm water was conducted.

Due to the difference between the height of outlet and the depth of the outlet, discharged flow is under the topographical condition that is not largely affected by seabed friction. As a result, the rise in water level is caused by entrainment of surrounding water and buoyancy. The rise obtained from the model considering rise in water level was ζ =0.5 cm. Fig.7 shows comparison of pressure increase and water temperature between both models. As seen from the figure, final results are not influenced by the fact whether the rise in water level is considered or not.

The difference between the model considering rise in water level and the one with fixed water surface was thus studied by the foregoing test calculations. In regards to the ordinary surface discharge of warm water, however, it was found that the two models did not present substantial difference since the rise in water level due to the seabed friction effect as well as buoyancy was small. Under the circumstances, it was decided to select a model with a fixed water surface to conduct calculations in our study because calculations could be done much faster, i.e. the application of algorithm for pressure calculations.

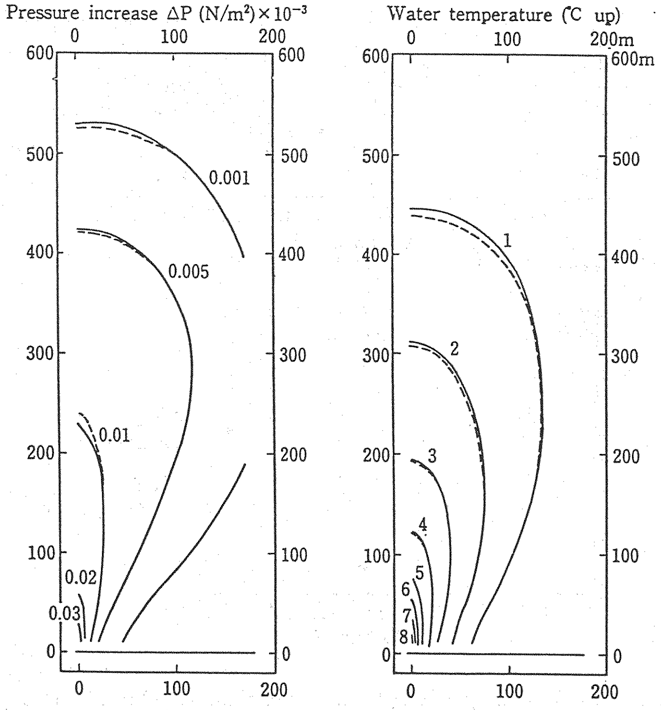

Fig.7 Horizontal distributions of pressure increase and water temperatures (Model considering rise in water level:— , Model fixing water surface: ····)

APPLICABILITY OF THE $\kappa-\varepsilon$ 3D MODEL [6)]

Calculating Conditions

To examine the applicability of this model, a comparative study was conducted with the result of a hydraulic model test (scale: 1/30) on the surface densimetric jet, which was carried out under an airconditioned circumstance. The size of the water tank was 20m× 35m and the velocity of flow was measured with an electromagnetic flowmeter.

Table 3

Dimension of outlet	Discharge flow velocity	Discharge flow rate	Water temp. difference	Densimetric Froude number
width B=11.25m height H=3.5m	V_{in}=0.7 m/s	Q=27.57 m³/s	T=7℃	F_{ro}=2.98

Analytical conditions were established to correspond to the experimental scale (scale=1/30) as in the following:

The topographical condition was straight along the coastal line and the outlet was set to step downward with water depth fixed at 10m.

Since k and ε are calculated by solving the transport equation, the setting value in the upstream section, particularly in the outlet, has an effect on the turbulent viscosity coefficient. In the present analysis, these values were defined like equation (16), and coefficients α and ℓ were set empirically.

$$k_{in} = \alpha U_{in}^2, \quad \varepsilon_{in} = k_{in}^{3/2}/l \tag{16}$$

In this calculation, α =0.01; ℓ =3.5m (outlet height).

Calculation Domain and Mesh Divisions

The calculation range was established as symmetrical with the central axis of jet flow, and one side of the symmetry was selected as the calculation domain. Calculation ranges in directions of off-shore and along the coastal line were decided based on test calculation results with mesh divisions and intervals as follows:

Table 4

direction	divisions	mesh interval	domain
coastal	36	x_{min} : 2.8125m ~ x_{max} : 20.0m	500m
off-shore	41	y_{min} : 2.8125m ~ y_{max} : 20.0m	600m
vertical	22	z_{min} : 0.35m ~ z_{max} : 0.95m	10m

These results account for a total of 32,472 mesh points.

Result of Analysis

A 3-hour test run was examined in our simulation with a time interval of 0.4 seconds and repetition of 27,000 times.

The vertical distribution of velocities and water temperatures is shown in Fig.8. A sharp variation of velocity and water temperature in the surface layer just after the outlet were reproduced in the experiments as well. The horizontal distributions of velocities and water temperatures in the surface layer are shown in Fig.9(1) and Fig.9(2).

Considering the accuracy of experiments and so on, the results obtained can be fully evaluated. The decrease ratio of velocities and temperatures is shown in Fig.10. Correspondence to experimental values is the same as when F_{io}=3.0 calculation values agree well with experimental values.

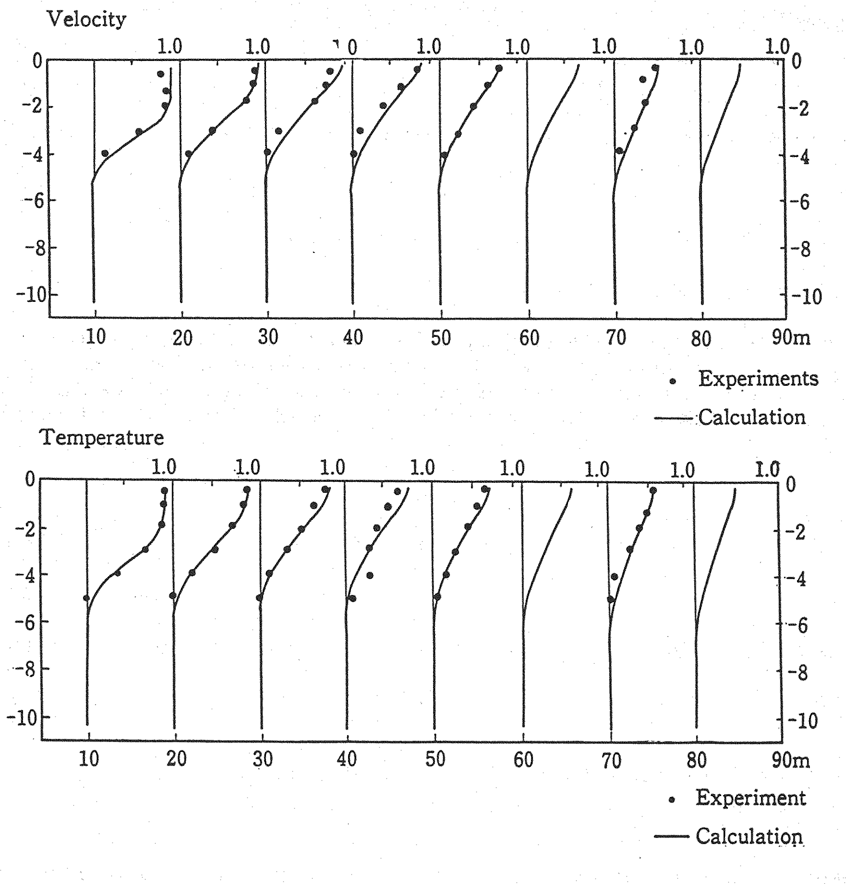

Fig.8 Vertical distributions of velocities and water temperatures in the plume axis section.

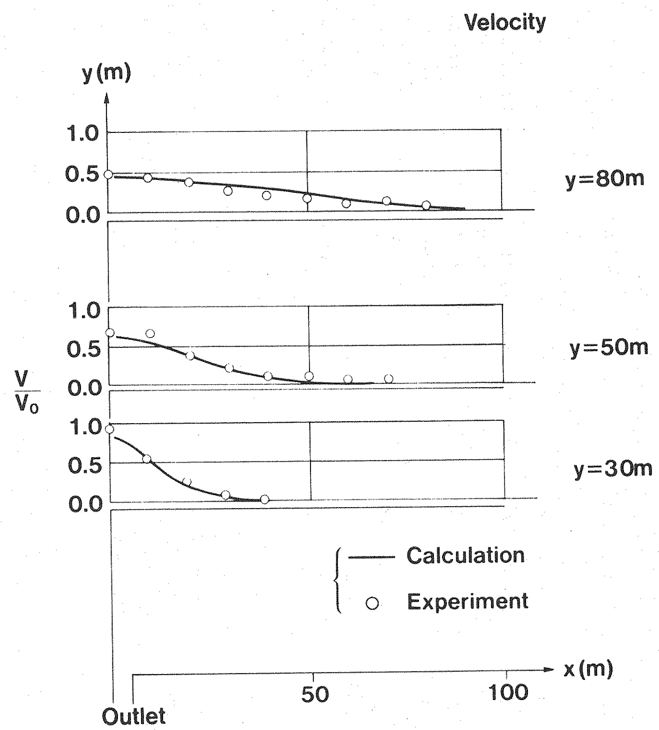

Fig.9(1) Horizontal distribution of velocities
(F_{io}=3.0)

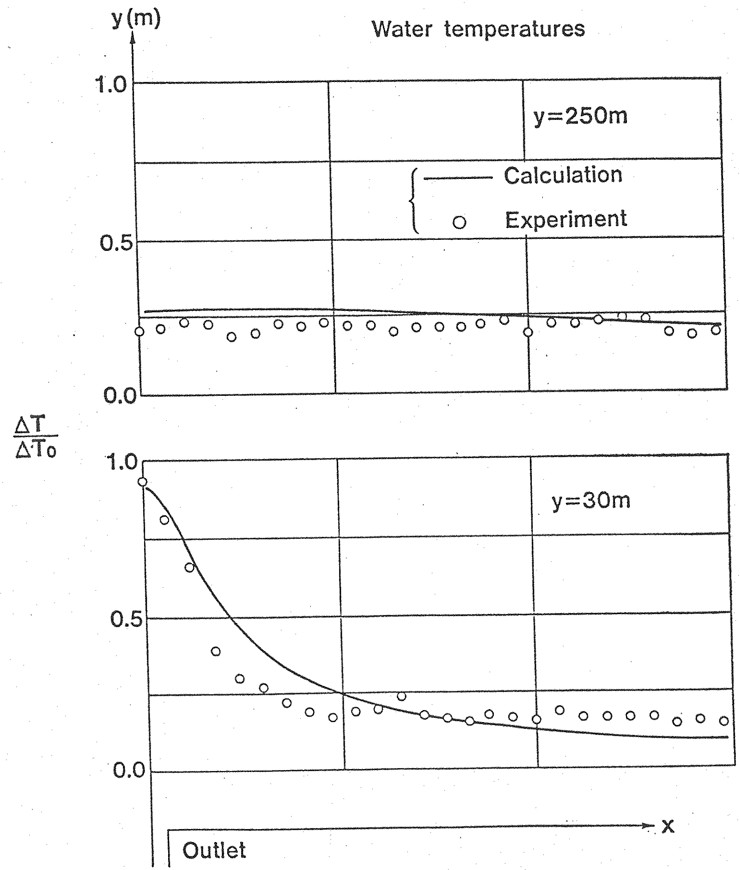

Fig.9(2) Horizontal distribution of water temperatures
(F_{io}=3.0)

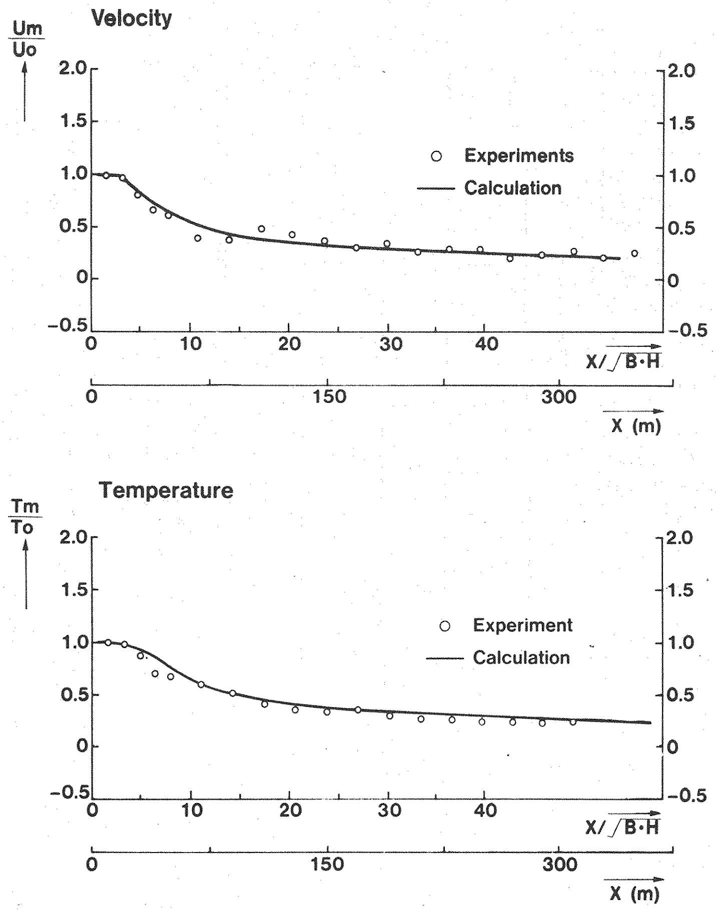

Fig.10 Decrease ratio of U_m and T_m in the center axis of plume.

CONCLUSION

It can be considered that the present model is effective for the numerical calculation of the surface densimetric jet because it shows a good agreement with experimental values near the outlet. By this, more accurate prediction of the high temperature rise zone (3-5℃) with the discharge of warm water can be realized.

For the future, it will be necessary to conduct studies on the combined method of three-dimensional models and conventional two-dimensional models, as well as on the applicability of the refined predictive model through comparison between the results of such modeling and the results of actual measurements at powerplants.

Acknowledgement

The author would like to acknowledge the cooperation of the Central Research Institute of Electric Power Industry, the Electric Power Technical Research Institute of Chubu Electric Power Co., Inc., Japanese Union Institute of Scientists and Engineers and Arc Information System Inc.

REFERENCES

1. Wada,A. and M.Kadoyu : Study on adaptability of prediction method of simulation analysis for diffusion of discharged warm water in the sea, Proceedings of the 17th Congress of International Association for Hydraulic Research, Vol.3, 1977.
2. Wada,A. and K.Miyaike : Change of water environment in the inner bay in consideration of heat balance, Proceedings of the 20th Congress of International Association for Hydraulic Research, Vol.4, pp.272-279, 1983.
3. Wada,A. : Prediction of the diffusion of discharged water for submerged outfall system, Proceedings of the 18th Congress of International Association for Hydraulic Research, Vol.4, pp.403-410, 1979.
4. Wada,A. and H.Araki : Development of predictive method for thermal diffusion in the sea near the cooling water outlet, Proceedings of the 3rd International Symposium on Refined Flow Modelling and Turbulence Measurements, pp.567-576, 1988.
5. Esposito,P. et A.Hauguel : Calcul des transitoires thermohydrauliques dans la cuue chaude de reacteurs a neutrons rapides, Proceedings of the 20the Congress of International Association for Hydraulic Research, Vol.4, pp.570-579, 1983.
6. Wada,A., Miura,M. and H.Suzuyama : Refined predictive model for thermal diffusion extent, Proceedings of the 4th International Symposium on Refined Flow Modelling and Turbulence Measurements, KA-27~ KA-34, 1990.

APPENDIX - NOTATION

The following symbols are used in this paper :

A_x, A_y, A_z	= Coefficients of eddy viscosity in the x, y and z axis directions
$b_{1/2}$	= Half breadth of plume
F_i	= Densimetric Froude number
K_x, K_y, K_z	= Coefficients of eddy diffusion in the x, y and z axis directions
k	= Turbulent energy
p	= Pressure
R_i	= Richardson number
T	= Water temperature
u, v, w	= Velocity components in the x, y and z directions
w_{max}	= Velocity in the center of plume
ε	= Energy dissipation rate
ρ	= Fluid density
ζ	= Water level