1

ANALYSIS OF THE MIXING BEHAVIOR OF THE HEATED WATER FROM THERMAL DIFFUSER

Il Won Seo[1], Tae Myoung Jeon[2], Eun Woo Son[3], and Seok Jae Kwon[4]

[1] Prof., Dept. of Civil Engineering, Seoul National Univ., Seoul, Korea
[2] Researcher, Daelim Technology Research Institute, Seoul, Korea
[3] Grad. Student, Dept. of Civil Engineering, Seoul National Univ., Seoul, Korea
[4] Post-Doctoral Researcher, Dept. of Civil Engineering, Seoul National Univ., Seoul, Korea

Abstract: The numerical model, FLUENT, was employed to investigate the effect of the heated water discharged from the diffuser of Boryung Power Plant. Temperature patterns of the thermal effluent discharged from two proposed types of the diffusers was evaluated for maximum flood and maximum ebb tide. The hydraulic model experiments were also performed in the reduced scale of 1/150 to verify the numerical simulation results. The buoyant jets discharged from the diffusers were found to be significantly affected by the ambient flows beyond the region where the effluent momentum was dissipated. Both the numerical and experimental results showed that the area of the excess isotherm for Type 1 diffuser was larger than that for Type 2 diffuser. Type 2 diffuser system was observed to be a more effective diffuser design than Type 1 diffuser system based on the temperature reduction and excess isotherm obtained from the numerical simulation in the ambient flows.

1. INTRODUCTION

Rapid dilution is required to minimize the harmful effects of discharge in both cases of treated sewage and thermal effluent discharged from wastewater plant and power plant, respectively. The most effective and economical way of reducing the harmful impacts of discharge is to analyze the characteristics of the discharge and to use a well-designed diffuser structure. The mixing caused by the discharge inflow can be divided into the jet governed by effluent momentum and the plume driven by buoyancy of the discharged water. The flow rate and discharge velocity of the thermal effluent are generally much larger than those of the treated sewage whereas the difference of relative density between the discharge and ambient water for the thermal effluent is smaller than that for the treated sewage. The thermal effluent with much higher momentum flux in shallow water results in unstable flow while the treated sewage with relatively lower momentum flux in sufficient depth leads to stable plumes (Jirka and Harleman, 1979). Therefore, the thermal effluent can be classified as a buoyant jet driven by large effluent momentum and the buoyancy force from the difference of temperature

between discharge and ambient water. However, the thermal effluent can be analyzed as a pure jet in the near field due to very slight effect of the buoyancy and can be treated as the buoyant flow after the effluent momentum dissipates. The huge thermal effluent with the temperature difference may have significant impacts on the coastal environments and the ocean ecosystem and also results in the reduction of power and operation efficiency after recirculation into the intake ports when the effluent does not mix with ambient water efficiently. Thus, well-designed diffuser to maximize the initial dilution and to minimize the impacts on the coastal region can be meaningful.

Three dimensional hydrodynamic model needs to be applied to simulate the three dimensional behavior of the thermal effluent flows. The three dimensional hydrodynamic model includes the FLUENT, FIDAP, FLOW-3D and STAR-CD. The FLUENT using FVM(Finite Volume Method) is the three dimensional CFD(Computational Fluid Dynamics) program based on the unstructured mesh and the calculation of free surface by VOF(Volume of Fluid) and can be applied to the various areas such as the analyses of steady or unsteady flows, laminar or turbulent flows, Newton or non-Newtonian fluid, compressible or incompressible fluid, heat transfer, flows in porous medium, combustion and etc. The GUI (Graphic User Interface) environment of the FLUENT is excellent, and its pre- and post-processing system is well-developed. Therefore, in this study, the FLUENT was used to investigate the hydrodynamic behavior of the thermal effluent and the characteristics of the temperature and flows in the interested region. The FLUENT was applied precedently to analyze hydraulic characteristics in upstream

dam and in spillway (Kim et al., 2003) and the mixing behavior of the heated water discharged from the submerged outlet of the Wolsung Nuclear Power Plant (Kim and Seo, 1997), etc. In addition, the numerical results were compared with the hydraulic experimental results. Capabilities of single jet numerical model with the k - ε model to simulate the behavior of the buoyant discharge was verified with the experimental results from available literature. The simulation results of a vertical section of diffuser ports were also compared with the experimental results (Harleman et al., 1968). The three-dimensional results of thermal discharge from Browns Ferry Nuclear Plant were compared with hydraulic test data (Lin and Hecker, 2002).

In this study, a numerical model is used to investigate the effect of the heated water discharged from the Boryung Power Plant. The hydraulic experiments were conducted to investigate the general behavior of the thermal effluent discharge in ambient flows and to compare the results with the numerical simulation results. In this study, two different types of the discharge system are evaluated. More efficient way is determined based on the temperature reduction and excess isotherm obtained from experiments and numerical simulation under various ambient flow conditions.

2. NUMERICAL ANALYSIS

2.1 Governing Equations of the FLUENT

The FLUENT developed by Fluent Inc. is the three dimensional program to analyze the fluid mechanics, heat transfer, chemical reaction and etc. The FLUENT is a CFD code to solve equations of the mass, momentum, energy, and chemical factors using FVM in unstructured

mesh, which can conduct the analysis with various fluids. This model also uses k - ε model and LES(Large Eddy Simulation) among the turbulence models. The prediction or calculation from this model can be used in the analysis of steady or unsteady flows, various laminar or turbulent flows, Newton or non-Newtonian fluid, compressible or incompressible fluid, heat transfer, flows in porous medium, combustion, multi-phase fluid behavior, and fluid noise. The governing equations used in the FLUENT consists of the continuity equation and Reynolds equation derived by integrating three dimensional Navier-Stokes equation with respect to time in the following forms:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad (1)$$

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\nu\frac{\partial u_i}{\partial x_j}-\overline{u'_i u'_j}\right)+b_i$$

$$(2)$$

where p is pressure, ν is kinematic viscosity, , u'_i , u'_j are the fluctuating velocity components in the i , j directions, respectively, and b_i is the body force in the i direction.

2.2 Procedure of Numerical Analysis

The numerical analysis using the FLUENT requires the generation of the mesh with the use of the mesh generation program, GAMBIT, and the input of boundary conditions. The methods to generate the mesh can be divided into Top-Down method and Bottom-Up method. The former is the method to divide the geometry into the three dimensional volume meshes after forming total frame of geometry while the latter

is the method to construct mesh from points to the lines, lines to planes, and then volumes. The FLUENT then forms the numerical model with suitable conditions and performs the analysis. The users can analyze the numerical results through monitoring the convergence of certain factor at a certain point and can terminate the program during running the program. The users can also use the diverse post-processing methods after numerical calculation.

2.3 Application of the FLUENT

This study selected the investigation region, that has the dimension of 1,500 m in the shoreline direction and 750 m from the shoreline to the offshore, with the diffusers at the center of the shoreline edge to investigate the characteristics of the thermal effluent diffusion as a result of the additional construction of diffusers of Boryung Power Plant with the ambient current. Topographic data was analyzed to choose the investigation region and to generate the computational grids. Fig. 1 shows the schematic sketch of the computation domain in the investigation region schematically. The fine grids were generated in the vicinity of the diffuser. The size of the grids with the shape of tetrahedron near the diffuser is about 1 m while the maximum size of the grids with the shape of hexahedron far from the diffuser is 3 m.

Fig. 2 provides the simplified shapes of the simulation region and the grids. The same area and interval of the selected region were applied to the different cases with varying water depth. Two different series were simulated: Series 1 for Type 1 diffuser system and Series 2 for Type 2 diffuser system. The result of grid generation for Series 1 shows 52,746 cells, 132,396 planes, and 31,026 points while the grids generated for
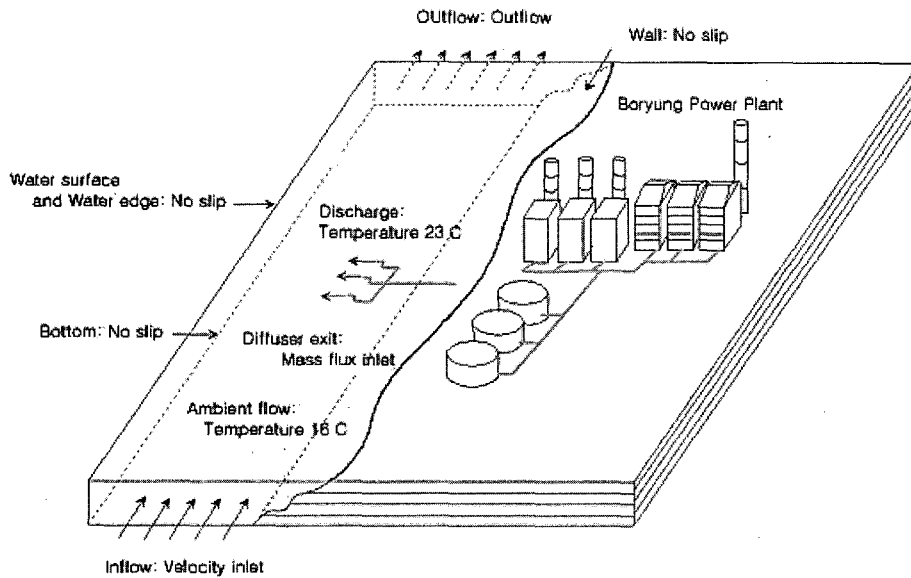
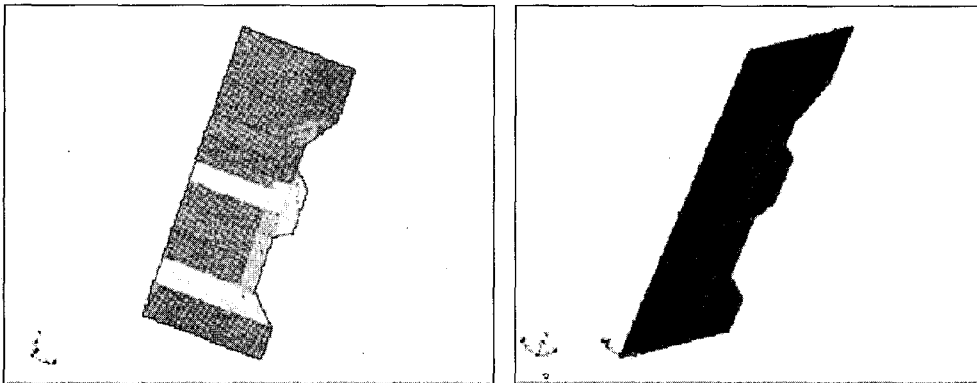**Figure 1. Schematic sketch of computational domain**

Figure 2. Mesh of computational domain

Series 2 consist of 57,652 cells, 143,526 planes, and 39,410 points.

Type 1 diffuser consists of three drainage pipes from the sump and a port at the end of each pipe, which is represented as the shape in Fig. 3. The exit velocity was estimated to be 1.9 m/s based on the calculation using the discharge and the area of the ports. The thermal effluent with the temperature higher by 7 °C than that of ambient water is simulated. Fig. 3 shows the grids of the outfalls generated by the GAMBIT for Series 1. It is shown that the outlet of the outfall is bifurcated with the rectangular shape. Type 2 diffuser consists of two drainage pipes from the sump and two ports on each drainage pipe varying water depth (two ports in the water depth of 12.0 m (MSL) and the other two ports in the water depth of 17.5 m (MSL)). However,
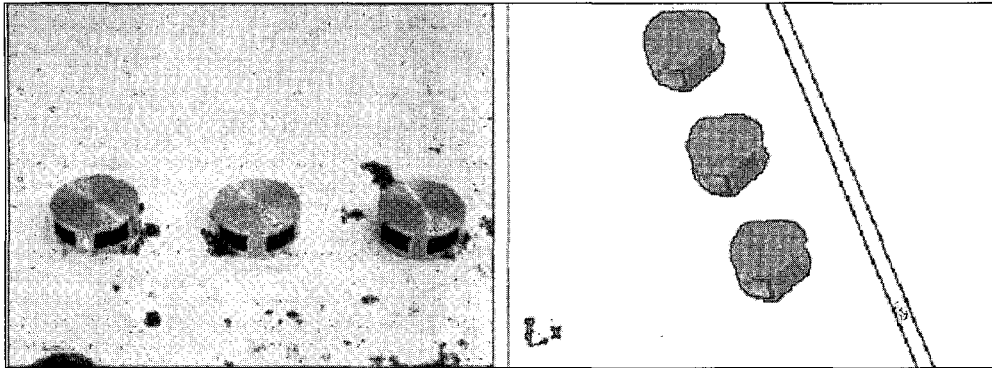
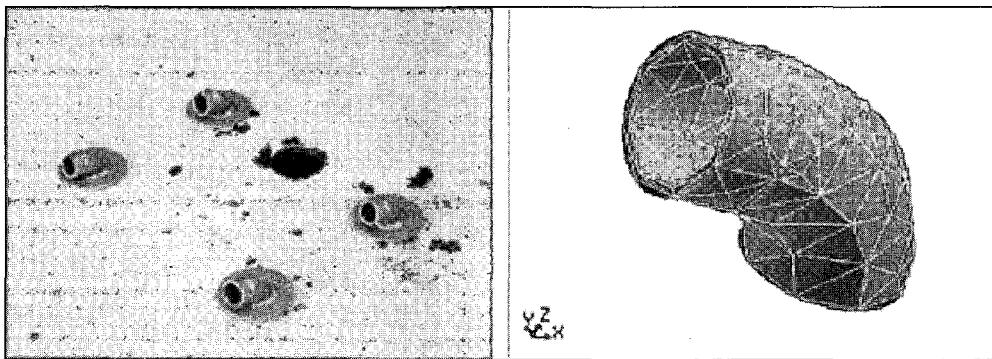**Figure 3. Picture and computational mesh of Type 1 diffuser**

Figure 4. Picture and computational mesh of Type 2 diffuser

the depth of the discharge was simplified to the average depth of the four ports. The exit velocity for Series 2 was estimated to be 2.7 m/s, and the thermal effluent was also simulated. Fig. 4 shows the shapes and grids of the outfalls for Type 2 diffuser.

2.4 Simulation Conditions

This numerical study assumed the hydrostatic pressure distribution along the vertical direction. The incompressible fluid with the density of 1024.8 kg/m^3 was used for the sea water. The k - ε model was used as the turbulence model, and the energy equation was included. The segregated solver was used among the segregated and coupled solvers provided by the FLUENT. The implicit method was chosen as

the difference method of the governing equations.

The boundary conditions applied in this study are summarized in Table 1. The inflow and outflow account for the inflow and outflow induced by tidal variation, respectively, according to the flood and ebb tides. The velocity and temperature of the inflows were properly inputted. The outflows were inputted as the condition for 'outflow', which means the fluids flow out with consistent flow characteristics. The jet inflow indicates the condition of the thermal effluent discharged from the ports. The temperature of the jet inflow was inputted as 296 K (23°C) which is higher by 7 °C than that for the ambient water. The exit velocities were inputted as 1.9 m/sec and 2.7 m/sec for

Table 1. Boundary Conditions

	Meaning	Boundary Conditions
inflow	ambient inflow	seawater: density = 1024.8 kg/m^3 temperature = 289 K (16 ℃) flow velocity = 0.85 m/sec
outflow	outflow of ambient flow and discharge	'Outflow' condition
jet inflow	thermal discharge diffuser exit	density = 1024.8 kg/m^3 temperature = 296 K (23 ℃) Case 101 discharge velocity = 1.9 m/sec Case 201 discharge velocity = 2.7 m/sec
bottom	bottom of the domain	Manning's Roughness Coeff. n=0.025 (Earth, some stones and weeds)
surface	watersurface of the domain	friction between seawater and air
wall	shoreline	Manning's Roughness Coeff. n=0.020 (Earth, smooth, no weeds)
waterwall	offshore edge of the domain	friction negligible

Type 1 and 2 diffusers, respectively. The boundary condition for the bottom was inputted considering the friction between the ambient water and the bottom, and the roughness height of the sea bed was taken into account based on sand. The lower friction for the water surface was considered based on the friction between air and sea water. The proper roughness height on the wall indicating the coastal region was also inputted considering the sandy plain and gravel and rocks on the coast. The friction for 'waterwall', the plane between the domain and the seawater out of the domain, was neglected.

3. EXPERIMENTS

Experiments have been performed in the rectangular tank that is 20 m long, 4.9 m wide, and 0.6 m high as shown Fig. 5. The model of the outfall was manufactured to reflect the representative characteristics for the geometry of the diffuser and the conditions of the thermal effluent discharged from the outfall at the Boryung Power Plant. The details of the model diffusers are described in the technical report, 'Experimental study on the influence of the thermal diffuser of Boryung Power Plant (Seo et al., 2004)'. The effluent was supplied from a specially manufactured hot water bath. The discharge from the hot water bath to the diffuser pipe was measured using an electro-magnetic flow meter. The temperature was measured using K-Type thermocouple sensors installed on an instrument carriage. Thermocouple sensors were connected to a 40-channel data logger. The velocity of ambient flow for the maximum flood was the same with that for the maximum ebb because the velocities of tidal flows measured in the field tests were similar in both tides.

In this study, the experimental conditions were selected to approximately reflect the operating conditions of the diffusers which would be considered for the additional construction in Boryung Power Plant for the maximum flood and maximum ebb tides. The
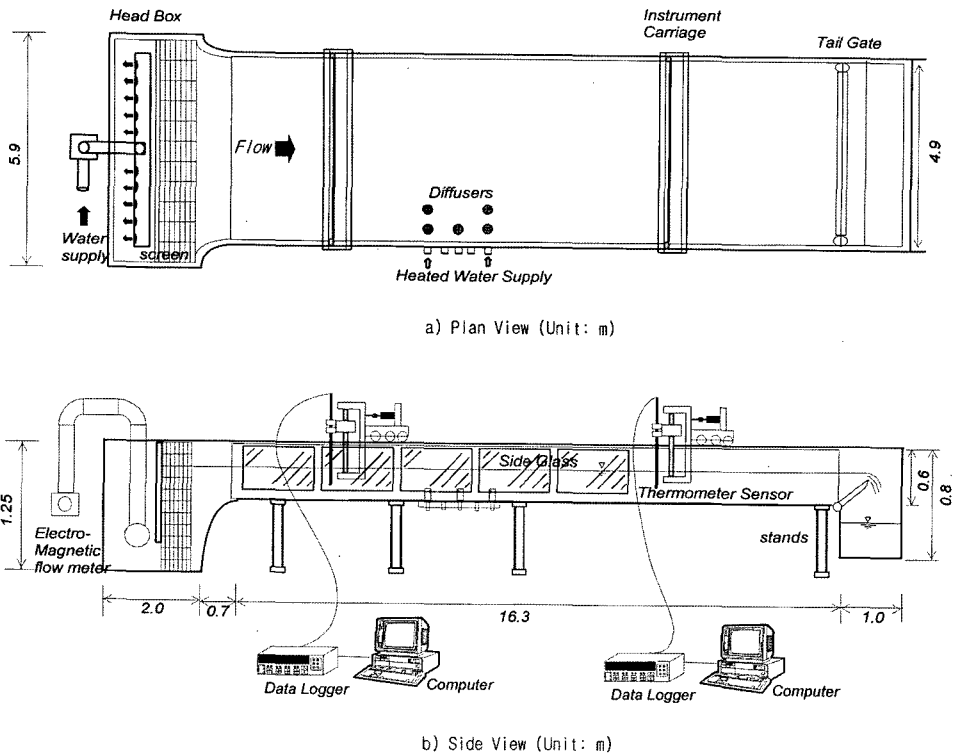
a) Plan View (Unit: m)

b) Side View (Unit: m)

Figure 5. Experimental Flume

relationship between the model and the prototype can be determined by densimetric Froude similarity rule, as follows

$$F_{jr} = \frac{U_{0r}}{\left(g'_r d_r\right)^{1/2}} \approx 1 \qquad (3)$$

where $U_{0r}\left(=U_{0m}/U_{0p}\right)$ is the ratio of the effluent velocity of the model to the prototype, $g'_r\left(=g'_m/g'_p\right)$ is the ratio of the effective gravitational acceleration of the model to the prototype, and $d_r(=d_m/d_p)$ is the ratio of the port diameter of the model to the prototype. In this study, the reduced scale of the model to the prototype was selected to be 1/150. The investigation area for the experiment was chosen as the rectangular region which has the

dimension of 1.5 km along the shoreline and 700 m to the offshore with the consideration of the maximum flood and maximum ebb tides. Table 2 shows the conditions of the prototype for the thermal discharges of Type 1 and 2 diffusers, which are reflected in the hydraulic model experiment.

4. RESULTS

4.1 Simulation Results

Fig. 6 shows the three dimensional velocity vector field for Type 1 diffuser simulated after 660 seconds from the effluent discharge. It is clearly observed that the thermal discharge is bent over by the effects of the ambient flow. The distance between the outfall and the region where the buoyant jet is bent over is approximately 30 m. The distance described in

Table 2. Conditions of Prototype

Series	Diffuser Type	Case	Tide	Ambient Flow Velocity (m/s)	Depth of Ambient Water (m)	Effluent Velocity (m/s)	Densi-metric Froude Number
1	Type 1	101	max. flood and ebb	0.84	13.50	1.89	11.34
		102	high	0.00	16.72	1.89	11.68
		103	low	0.00	10.28	1.89	12.13
2	Type 2	201	max. flood and ebb	0.84	17.75	2.70	14.58
		202	High	0.00	20.97	2.70	14.90
		203	low	0.00	14.53	2.70	15.47
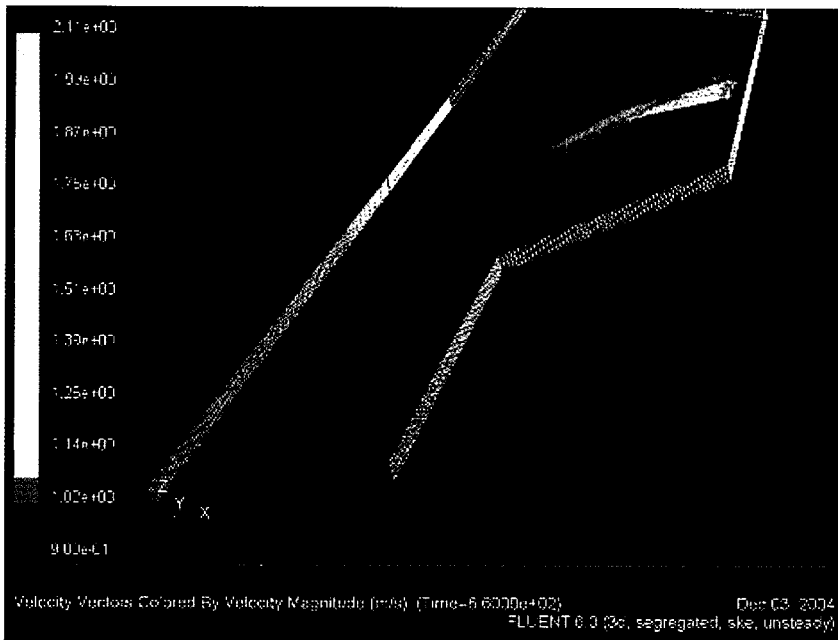
**Figure 6. 3D velocity vector field without ambient flow
(Case 101; Maximum ebb tide, T = 660 sec)**

this study indicates the distance in the direction perpendicular to the shoreline. It is noted that the flow velocity begins to increase behind the outfall. The effect of the edge is also observed around the region where the outflow area is reduced. Fig. 7 shows the velocity contour on the plane 1.2 m above the bottom. At this height, the effluent flow velocity of 1.5 m/s tends to persist up to 75 m from the outfall and then to

drop to the flow velocity of 1.0 m/s, which is slightly faster than the ambient velocity. The effluent flow velocity then decreases to 0.8 m/s, which is nearly the same as the ambient flow velocity. Fig. 8 presents the temperature contour on the plane 1.2 m above the bottom. After the temperature of the thermal effluent (23 ℃) drops rapidly to the temperature of 18 ℃, the temperature of the effluent tends to drop to 17 ℃
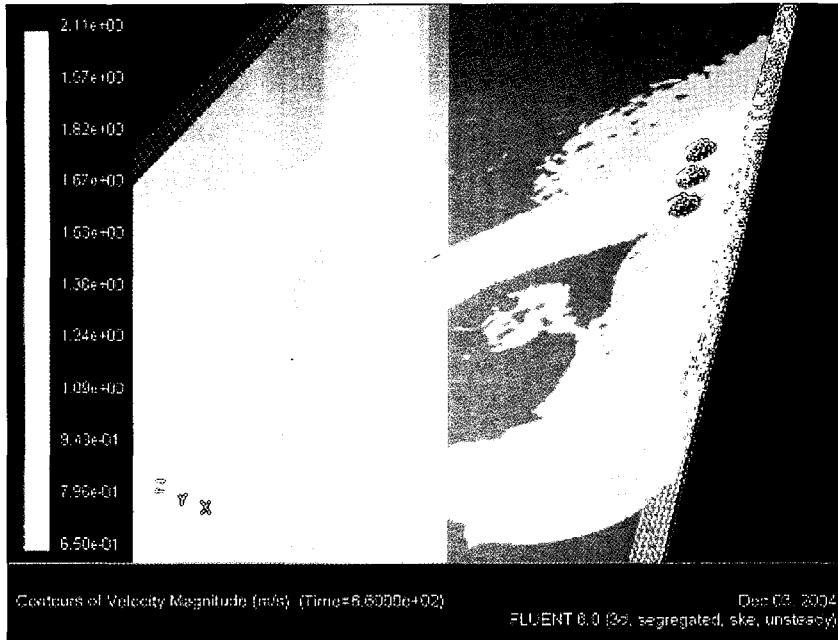
**Figure 7. Velocity contour at the height of 1.2 m above the bottom
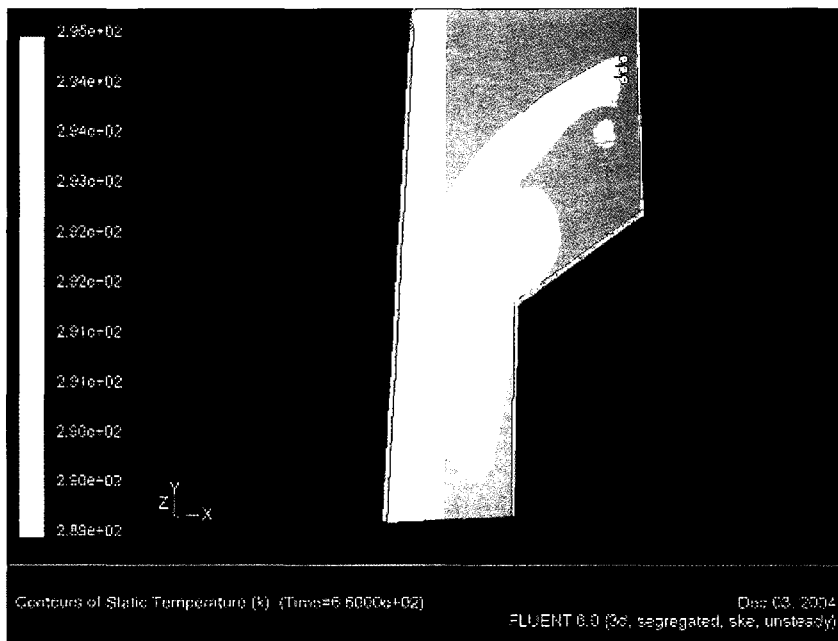(Case 101; Maximum ebb tide, T = 660 sec)**

**Figure 8. Temperature contour at the height of 1.2 m above the bottom
(Case 101; Maximum ebb tide. T = 660 sec)**

about 150 m from the outfall and then to maintain the temperature similar to the ambient temperature from the region 300 m from the outfall. Since the eddy in the velocity contour is observed around the region where the outflow area is reduced, the temperature distribution is also found to spread along the eddy in the direction perpendicular to the ambient flow. The tendency of the temperature distribution is generally similar to the velocity distribution. However, the temperature behind the outfall is not observed to vary.

Type 2 diffuser consists of two drainage pipes from the sump. One port is located at the end of each pipe, and the other port at 20 m behind to the onshore from the end of each pipe. The three dimensional velocity vector field simulated after 720 seconds from the effluent discharge shows that the buoyant jets discharged from the ports are also bent over by the effects of the ambient flow. The distance between the outfall and the

region where the buoyant jet is bent over is about 30 m. Fig. 9 shows the temperature contour on the plane 3 m above the bottom. This figure shows that the merging between the adjacent buoyant jets occurs where the buoyant jets are bent over. After the temperature of the thermal effluent (23℃) diminishes to the temperature of 18 ℃ rapidly, the temperature of the effluent tends to drop to 16.5 ~ 17 ℃, which is close to the ambient temperature about 100 m away from the outfall. The width of the excess isotherm distribution is observed to be 20 ~ 30 m, which is very narrow.

The excess isotherm (1~2℃) contours in the x-y plane at the center of the ports for Cases of 101 and 201 after 600 seconds from the effluent discharge are compared in Fig. 10. It is noted that Case 201 shows more effective dilution than that for Case 101 based on the temperature reduction and excess isotherm in the vicinity of the out-falls. Furthermore, the area of the excess isotherm
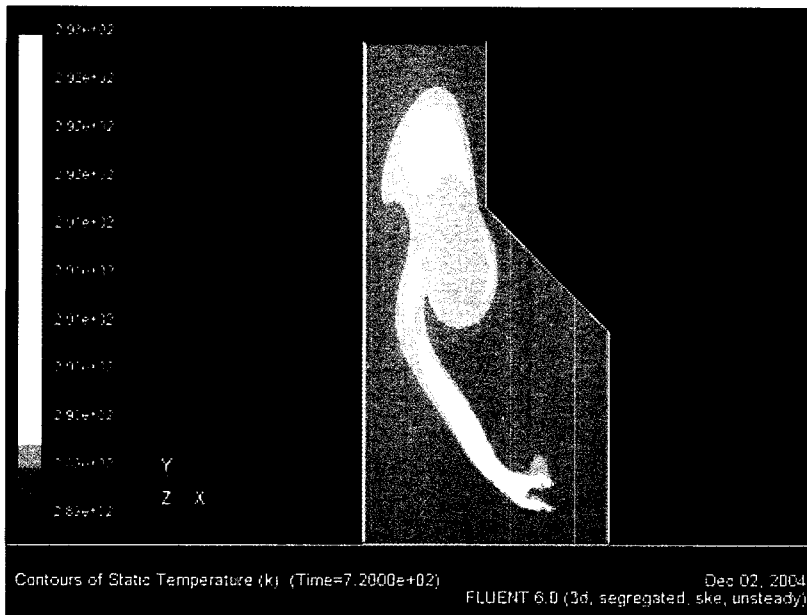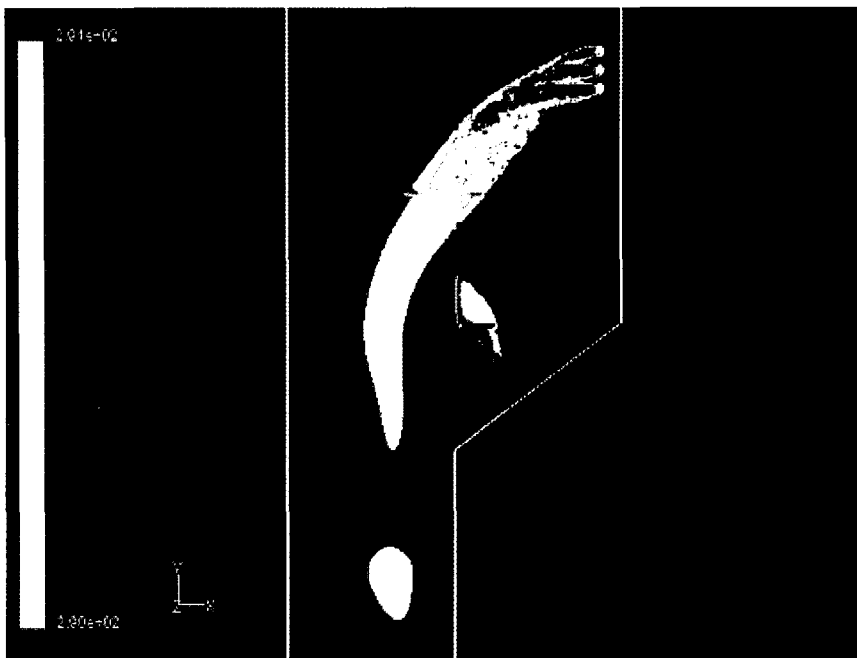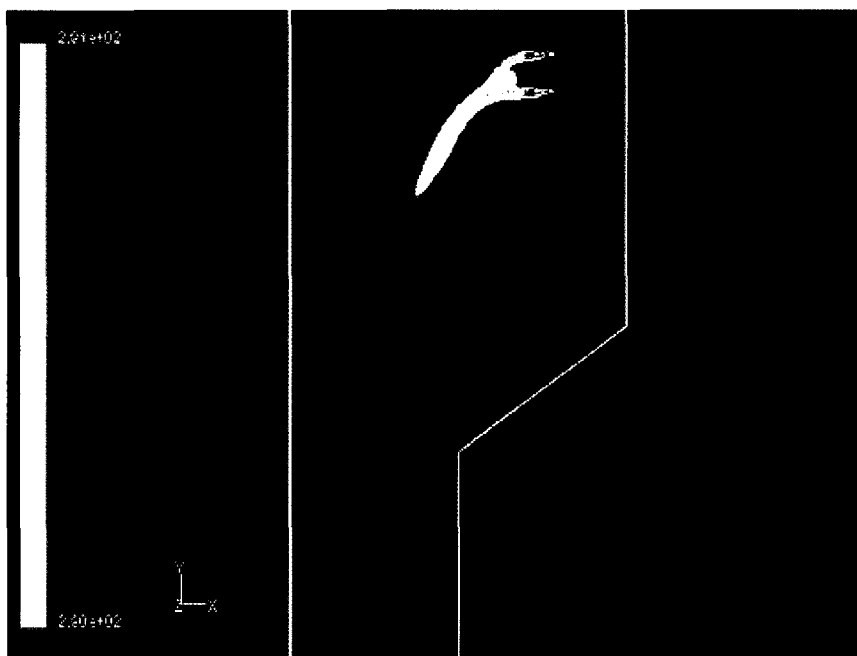

**Figure 9. Temperature contour at the height of 3 m above the bottom
(Case 201; Maximum flood tide, T = 720 sec)**

a) Case 101

b) Case 201

**Figure 10. 1~2 excess temperature contours at mid-height of the diffuser
(Maximum ebb tide, T = 600 sec)**

(1~2℃) for Case 201 is smaller than 1/4 of that for Case 101. This implies that the harmful impact of discharge on the ocean environment for Case 201 is much less than that for Case 101.

4.2 Comparison between Simulation Results and Experimental Results

Fig. 11 shows the temperature contours (a) and (b) on the water surface obtained from the hydraulic experiment and the numerical simulation, respectively, for Case 101. The experimental result for Case 101 shows that the excess isotherm higher than 0.2 ℃ is limited within 200 m to the offshore. The region of the excess isotherm higher than 0.2 ℃ was attached

to the shoreline beyond 600 m from the outfall along the shoreline. It is also observed that the region of the excess isotherm higher than 1.0 ℃ extends beyond 200 m from the outfall along the shoreline. The numerical simulation result shows that the buoyant jet extends more to the offshore than that obtained from the experiment, whereas the distribution in the x-direction is similar with that obtained in the experiment. Therefore, the buoyant jet simulated by the FLUENT does not reach the shoreline and shows larger area of the excess isotherm higher than 1.0℃ than that in the experimental result.
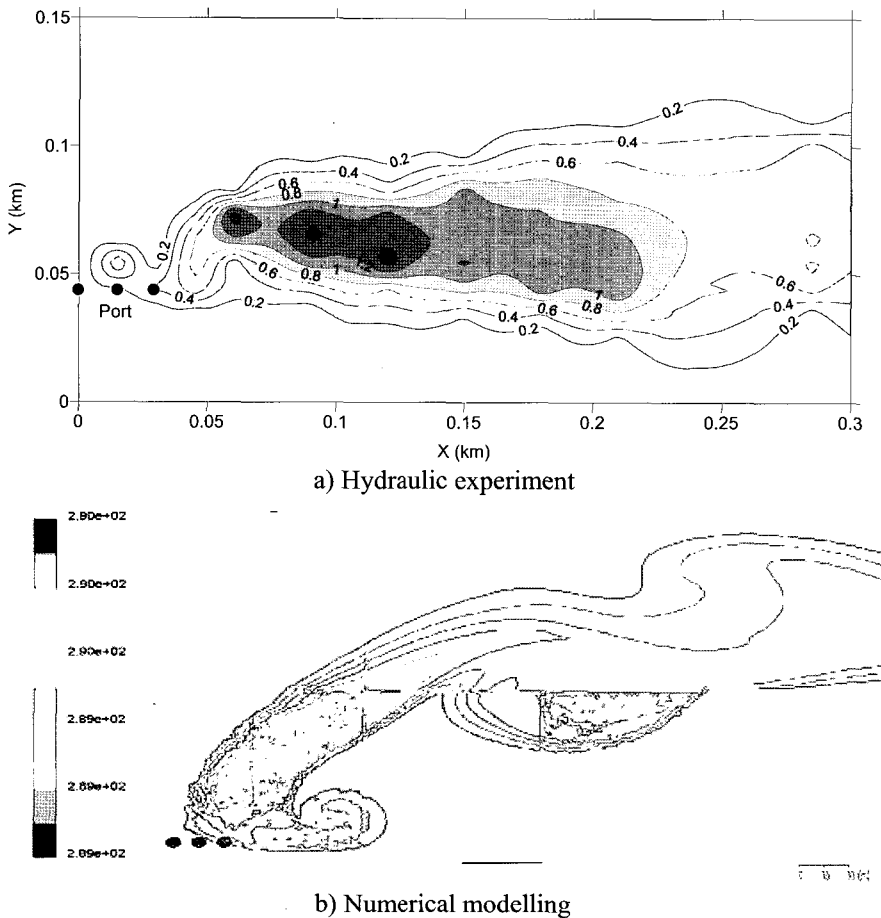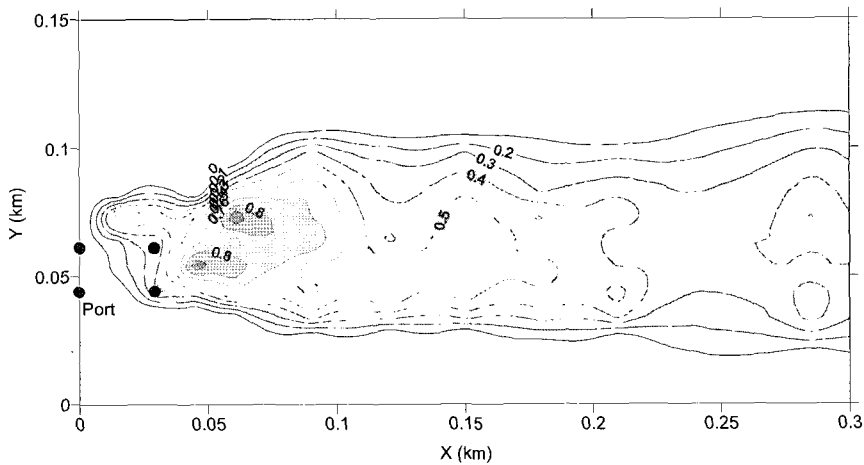
a) Hydraulic experiment

b) Numerical modelling

Figure 11. Water surface temperature contours of Case 101

Fig. 12 provides the temperature contours (a) and (b) on the water surface obtained from the experiment and the simulation, respectively, for Case 201. The experimental result for Case 201 shows that the excess isotherm higher than 0.2 ℃ is also limited within 200 m to the offshore. The region of the excess isotherm higher than 0.2 ℃ also reaches the shoreline beyond 600 m from the outfall along the shoreline. However, the region of the excess isotherm higher than 1.0 ℃ was not observed in Case 201. In the numerical simulation result, the width and area of the region of the excess

isotherm are quite larger than those in the experimental result. The direction of the trajectory is diagonal in the near field due to the ambient flow, and the variation of the temperature distribution whose gradient is steeper than that in the experimental result is somewhat regular.

5. CONCLUSION

The numerical model, the FLUENT, was used to investigate the buoyant mixing behavior of the thermal effluent and the effect of additional construction of diffusers of Boryung Power Plant

a) Hydraulic experiment
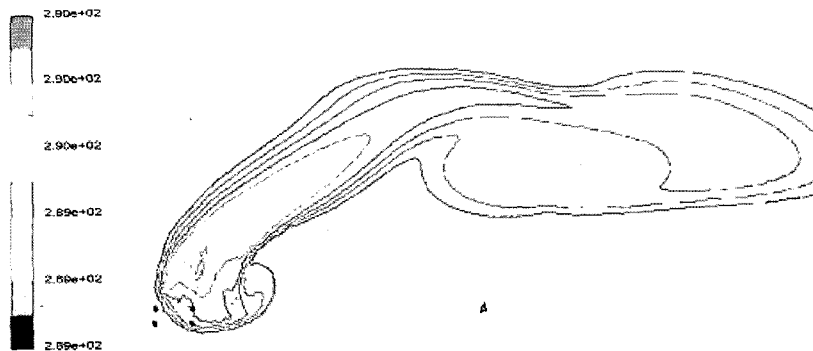
b) Numerical modelling

Figure 12. Water surface temperature contours of Case 201

in the investigation region. The hydraulic experiments were additionally conducted to compare the numerical simulation results with the experimental results. This study is focused on two different cases (Cases 101 and 201) varying the ways of the discharge with different types of diffusers.

In the selected region that has the dimension of 1500 m along shoreline and 750 m perpendicular to the shoreline, the grids are generated by the GAMBIT with Top-Down method. The k-ε model was used for the turbulence model, and the segregated solver was chosen. The boundary conditions are applied with the considerations of the velocity and temperature of the inflows and the jet inflow, the exit velocities, and the friction and roughness height on the bottom and walls. The hydraulic experiments with the scale ratio 1/150 were carried out in a rectangular tank. The experimental conditions were selected to approximately reflect the operating conditions of the proposed diffusers in Boryung Power Plant for the maximum flood and maximum ebb tides.

The thermal effluent discharge is observed to be bent over by the effects of the ambient flows in both Cases 101 and 201. In the velocity contours on the planes above the bottom, the effluent flow velocity tends to persist in the near field and then decreases to the velocity similar to the ambient flow velocity in far field. The initial temperature of the thermal effluent drops rapidly and tends to persist similar to the ambient temperature in the far field. The merging between the adjacent buoyant jets in Case 201 occurs around the region where the buoyant jets are bent over. Case 201 was found to show more effective dilution than Case 101

since the area of the excess isotherm (1~2 ℃) for Case 201 is less than 1/4 of that for Case 101.

The buoyant jets from numerical simulations for Cases 101 and 201 are more extended to the offshore than those in the experimental results whereas the distribution in the x-direction is similar with those obtained in the experiment. The numerical result also showed that the area of the excess isotherm higher than 1.0 ℃ for Case 101 is somewhat larger than that for Case 201. In the experimental result, the region of the excess isotherm higher than 1.0 ℃ was not observed in Case 201. Case 201 was observed to provide more efficient discharge based on the temperature reduction and excess isotherm obtained from both the experiments and numerical simulation with given conditions.

ACKNOWLEDGEMENTS

REFERENCES

Jirka, G.H., and Harleman, D.R.F. (1973). *The Mechanics of Submerged Multiport Diffusers for Buoyant Discharge in Shallow Water*. Technical Report No. 169, Ralph M. Parson Laboratory for Water Resources and Hydrodynamics, MIT Cambridge.

Kim, D.G., and Seo, I.W. (1997). Analysis of velocity structure of round wall. *Journal of Korea Water Resources Association*, Vol. 30, No. 5, pp. 467-475. (In Korean)

Kim, Y.H., Oh, J.S., and Seo, I.W. (2003). Analysis of hydraulic characteristics upstream of dam and in spillway using numerical models. *Journal of Korea Water Resources Association*, Vol. 36, No. 5, pp. 761-776. (In Korean)

Harleman, D.R.F., Hall, L.C., and Curtis, T.G. (1968). *Thermal diffusion of condenser water in a river during steady and unsteady flows with application to the T.V.A. Browns Ferry Nuclear Power Plant*. Hydro-dynamics Laboratory Report No. 111, MIT, Cambridge.

Lin, F., and Hecker, G.E. (2002). *3-D numerical modeling of thermal discharge from Browns Ferry Nuclear Plant*. Technical Report 176-2/M443F, Alden Research Laboratory, Inc., M.A.

Seo. I.W., Baek, K.O., Kwon, S.J., Lee K.W., and Kim, H.J. (2004). *Experimental study on the influence of Boryung Power Plant additional construction on environment and transport*. Technical Report, Korea Midland Power Co., Ltd. (In Korean)

Prof., Dept. of Civil Engineering, Seoul National Univ., Seoul, Korea

(e-mail: seoilwon@snu.ac.kr)

Researcher, Daelim Technology Research Institute, Seoul, Korea

Grad. Student, Dept. of Civil Engineering, Seoul National Univ., Seoul, Korea

Post-Doctoral Researcher, Dept. of Civil Engineering, Seoul National Univ., Seoul, Korea